STAAD.Pro

CONNECT Edition V22 Update 10

User Manual

Last Updated: April 05, 2022

Bentley®

Advancing Infrastructure

Table of Contents

1
Getting Started

This help file is intended to guide users who are new to STAAD.Pro as well as experienced users who want specific information on the basics of using the program.

Start by reading through this section which contains basic information on getting STAAD.Pro running on your computer.

It is then strongly recommended that you work through all three of the tutorials. These contain step-by-step procedures of modeling, analysis, design, and post processing different structures.

Next, read through the frequently performed tasks to learn how to perform common operations in the program.

GS. Welcome to STAAD.Pro

The ultimate power tool for computerized structural engineering.

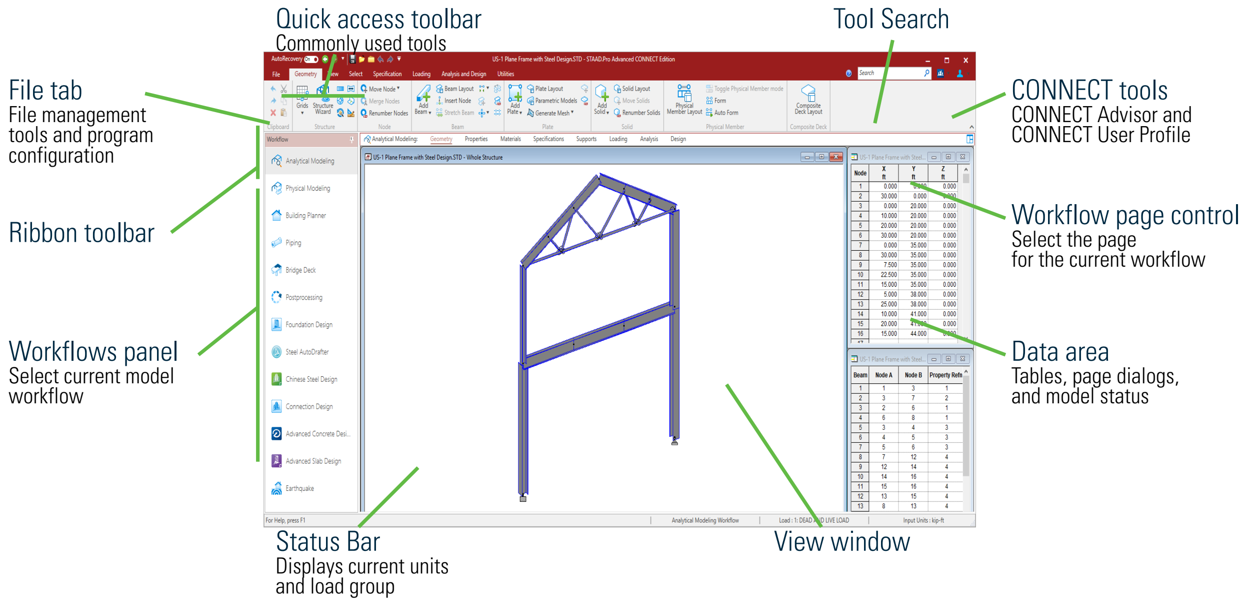
STAAD.Pro CONNECT Edition is the most popular structural engineering software product for 3D model generation, analysis and multi-material design. It has an intuitive, user-friendly graphical user interface, visualization tools, powerful analysis and design facilities and seamless integration to several other modeling and design software products. The software is fully compatible with supported Windows operating systems.

For static or dynamic analysis of bridges, containment structures, embedded structures (tunnels and culverts), pipe racks, steel, concrete, aluminum or timber buildings, transmission towers, stadiums or any other simple or complex structure, STAAD.Pro has been the choice of design professionals around the world for their specific analysis needs.

To learn about the latest features in STAAD.Pro, please refer the What's New? (on page 90).

GS. Overview of the STAAD.Pro Environment

Tip: Click on any label in the figure to jump to that section of the help.



1. File tab (on page 2694)
2. Ribbon Control Reference (on page 2694)
3. GS. Workflows in STAAD.Pro (on page 36)
4. GS. Quick Access Toolbar (on page 62)
5. GS. Tool Search (on page 64)
6. Bentley CONNECT Features (on page 2068)
7. GS. Page Control (on page 64)
8. GS.Data Area (on page 66)
9. GS. Status Bar (on page 84)
10. GS. View Window (on page 67)

GS. About STAAD.Pro Documentation

The documentation for STAAD.Pro consists of the following sections that make up the online help.

All the manuals can be accessed from the Help facilities of STAAD.Pro.

Getting Started (GS.)

This section contains information on the contents of the STAAD.Pro package, computer system requirements, installation process, copy protection issues and a description on how to run the programs in the package.

What's New?

This section contains the software release report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 06.

Tutorials (T.)

Tutorials that provide detailed and step-by-step explanation on using the programs are also provided.

Modeling (M.)

This section describes how to model structural elements and loads in STAAD.Pro using the Analytical Modeling workflow.

Analysis (A.)

This section of the help describes the analysis methods, data print commands, and how to perform analysis in STAAD.Pro.

Design (D.)

This section describes how to perform the design of structural elements using STAAD.Pro.

This section contains information on the various concrete, steel, timber, and aluminum design codes that are implemented in STAAD.Pro.

Postprocessing and Reports (P.)

This section describes how to review output, perform post-processing tasks, generate reports, and plot from STAAD.Pro.

Data Files and Interoperability (I.)

This section describes how to manually edit STAAD.Pro input files, import and export data, and interact with STAAD.Pro using external applications.

General Engineering Theory (G.)

This section deals with the theory behind the engineering calculations made by the STAAD engine.

Technical Reference of STAAD Commands (TR.)

This section includes an explanation of the commands available in the STAAD command file.

Ribbon Control Reference

This section describes the tools found in the ribbon toolbars.

Verification Examples (V.)

This section includes examples of STAAD.Pro example models along side reference (“textbook”) examples or a set of “hand” calculations. This provides you with validation of standard program calculations.

Application Examples (EX.)

This section includes examples of various problems that can be solved using the STAAD engine. The examples represent various structural analysis and design problems commonly encountered by structural engineers. Additional example models installed with the product are described here in brief.

OpenSTAAD (OS.)

This section describes the application programming interface for STAAD.Pro.

GS. Using Online Help

The Web help opens in your default web browser.

The Help window consists of two panes — the navigation pane on the left and the topic pane on the right.

The navigation pane has the following tabs:

- Contents — used for browsing topics.
- Index — index of help content.
- Glossary — glossary of the help content.
- Search — displays the search results.
- Tool Index — index of the tools.

Hypertext links, which appear in color and are underlined when you hover the pointer over them, let you move easily between related topics.

Note: In case you are not able to view the Web help properly in Internet Explorer, make sure you turn off the **Enable Protected Mode** check box in the Security tab of the Internet Options dialog.

Note: The text size in the topic pane on the right is controlled by your web browser. To learn how to change the web page’s text size, consult the web browser's Help file.

To open the Help window

1. Either:

   Click the **Help** tool in the top, right corner of the application window 

   or

   Press **<F1>**

   The Help window opens and the **Contents** tab is displayed.

The Help window consists of two panes — the navigation pane on the left and the topic pane on the right. At the top of the topic pane is a topic banner containing additional navigation tools.

To browse topics using the Contents tab

1. On the Contents tab, click the symbol preceding any link to expand its contents.
2. Continue expanding folders until you reach the desired topic.
3. Select a topic to display its content in the topic pane.

To display the next or previous topic according to the topic order shown in the Contents tab

1. Either:

View...	Select...
the next topic	click the right arrow in the help window's control bar 
the previous topic	click the left arrow in the help window's control bar 

To navigate in this manner, it is not necessary to first select the **Contents** tab.

To use the index to find help content

1. Select the **Index** tab.
2. Scroll through the index using the scroll bar to find a specific entry.

> **Tip:** Press **<Ctrl+F>** to use your web browser's find feature to locate instances of a word or phrase in the index.

3. Select the desired entry.
The content that the selected index entry is referencing displays in the topic pane.

To search for text in the help content

1. In the **Enter search terms** field, type the word or phrase for which you are searching.
2. Either:

   Click **Search**

   or

   Press **<Enter>**
   Results of the search display on the Search tab.
3. Click the title of any search result.
4. (Optional) Search for a term within a topic:
   a. Press **<Ctrl+F>**
   Your browser's search dialog opens.
   b. Type the word or phrase for which you are searching.
   c. Click **Next** to find each occurrence of the text in this topic.

Search results vary based on the quality of the search criteria entered in the Search field. The more specific the search criteria, the more narrow the search results. You can improve your search results by improving the search criteria. For example, a word is considered to be a group of contiguous alphanumeric characters. A phrase is a group of words and their punctuation. A search string is a word or phrase on which you search.

GS. Documentation Conventions

A number of typographical conventions are maintained throughout Bentley documentation, which makes it easier to identify and understand the information presented.

Notes, Hints, and Warnings	Items of special note are indicated as follows: **Note:** This is an item of general importance. **Tip:** This is optional time-saving information. **Warning:** This is information about actions that should not be performed under normal operating conditions.
File Path/File Name.extension	A fixed width typeface is used to indicate file names, file paths, and file extensions (e.g., .../`STAAD/Staadpro.exe`)
Interface Control	A bold typeface is used to indicate user controls, such as ribbon tabs, tool names, and dialog controls. (e.g., **File** > **Save As**).
User Input	A bold, fixed width typeface is used to indicate information which must be manually entered. (e.g., Type `DEAD LOAD` as the title for Load Case 1).

Terminology

- Click - This refers to the action of pressing a mouse button. When not specified, click means to press the left mouse button.
- Select - Synonymous with Click. Used when referring to an action in a menu, drop-down list, list box, or other control where multiple options are available to you.
- pop-up menu - A pop-up menu is displayed typically with a right-click of the mouse on an item in the interface.
- Window - Describes an on screen element which may be manipulated independently. Multiple windows may be open and interacted with simultaneously.
- Dialog - This is an on screen element which (typically) must be interacted with before returning to the main window.
- Cursor - Various selection tools are referred to as “cursors” in STAAD.Pro. Selecting one of these tools will change the mouse pointer icon to reflect the current selection mode.

Mathematical Notation

Similar to spelling conventions, American mathematical notation is used throughout the documentation. A serif typeface is typically used to clarify numbers or letters which might otherwise appear similar.

- Numbers greater than 999 are written using a comma (,) to separate every three digits.

  For example, the U.S. value of Young's Modulus is taken as 29,000,000 psi.

Warning: Do *not* use commas or spaces to separate digits within a number in a STAAD input file.

- Numbers with decimal fractions are written with a period to separate whole and fraction parts. For example, a beam with a length of 21.75 feet.
- Multiplication is represented with a raised –or middle– dot (·) or a multiplication symbol (×). For example, $P = F \cdot A$ or $P = F \times A$.
- Operation separators are used in the following order:

  1. parenthesis ( )
  2. square brackets [ ]
  3. curly brackets (i.e., braces) { }

  For example,

  $$F_a = [1 - (Kl/r)^2/(2 \cdot C_c^2)]F_y / \{5/3 + [3(Kl/r)/(8 \cdot C_c)] - [(Kl/r)^3/(8 \cdot C_c^3)]\}$$

  Which may also be represented as:

  $$F_a = \frac{\left[1 - \frac{\left(\frac{Kl}{r}\right)^2}{2C_c^{\,2}}\right]F_y}{\left\{\frac{5}{3} + \left[\frac{3\left(\frac{Kl}{r}\right)}{8C_c}\right] - \left[\frac{\left(\frac{Kl}{r}\right)^3}{8C_c^{\,3}}\right]\right\}}$$

GS. Where are the old manuals?

For many versions of STAAD.Pro, the help was primarily a collection of the user manuals in an online format. Though the organization has changed somewhat, all of the existing content is still available to you.

Technical Reference Manual

This chapters of this document have been separated into the logical portions of this help file. You can find each of the sections as follows:

1. "General Description" - General Engineering Theory (on page 2085)
2. "American Steel Design" - D1. American Codes (on page 1086)
3. "American Concrete Design" - D1.F. American Codes - Concrete Design per ACI 318 (on page 1198)
4. "American Timber Design" - D1.G. American Codes - Timber Design per AITC Code (on page 1248)
5. "Commands and Input Instructions" - Technical Reference of STAAD Commands (on page 2196)

International Design Codes Manual

This content is included in the D. Design Codes (on page 1086) section of the online help.

The American codes are now included here for consistency. The sections including American aluminium design, American transmission tower design, API steel design code, ANSI/AISC N690 design, and ASME NF codes have all been moved to the chapter for American design codes as well.

Release Report

This document is included as the What's New? (on page 90) section of the online help.

Getting Started and Tutorials

This document has been reorganized and is included primarily within the Getting Started section of the online help.

Graphical User Interface

This content is no longer collected into a sub-section, but has been expanded to cover modeling, analysis, design, post-processing, and interopability as described elsewhere in the online help.

Examples Manual

Most complete input examples, verification examples, and extended modeling examples have been collected into the Application Examples section of the help.

GS. About STAAD.Pro

STAAD.Pro is a general purpose structural analysis and design program with applications primarily in the building industry — commercial buildings, bridges and highway structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, culverts and other embedded structures, etc. The program hence consists of the following facilities to enable this task.

1. Graphical model generation utilities as well as text editor based commands for creating the mathematical model. Beam and column members are represented using lines. Walls, slabs and panel type entities are represented using triangular and quadrilateral finite elements. Solid blocks are represented using brick elements. These utilities allow you to create the geometry, assign properties, orient cross sections as desired, assign materials like steel, concrete, timber, aluminum, specify supports, apply loads explicitly as well as have the program generate loads, design parameters etc.
2. Analysis engines for performing linear elastic and p-delta analysis, finite element analysis, frequency extraction, and dynamic response (spectrum, time history, steady state, etc.).
3. Design engines for code checking and optimization of steel, aluminum and timber members. Reinforcement calculations for concrete beams, columns, slabs and shear walls. Design of shear and moment connections for steel members.
4. Result viewing, result verification and report generation tools for examining displacement diagrams, bending moment and shear force diagrams, beam, plate and solid stress contours, etc.
5. Peripheral tools for activities like import and export of data from and to other widely accepted formats, links with other popular software programs for niche areas like reinforced and prestressed concrete slab design, footing design, steel connection design, etc.
6. A library of exposed functions called OpenSTAAD which allows you to access the internal functions and routines in STAAD.Pro as well as its graphical commands to tap into STAAD.Pro's database and link input and output data to third-party software written using languages like C, C++, VB, VBA, FORTRAN, Java, Delphi, etc. Thus, OpenSTAAD can be used to link in-house or third-party applications with STAAD.Pro.

GS. System Requirements

The following hardware requirements are suggested minimums. Systems with increased capacity provide enhanced performance.

Processor	Intel® Pentium or AMD® processor 3.0 GHz or greater.
Memory	1 GB minimum, 2 GB recommended (4GB for STAAD.Pro Advanced). More memory almost always improves performance, particularly when working with larger models. 4 GB (8 GB for STAAD.Pro Advanced) or more can help speed up solutions for very large complex models with large numbers of load cases.
Video	Graphics card supported by OpenGL. See the graphics card manufacturer for latest information on graphics drivers. 256 MB of video RAM or higher is recommended. If insufficient video RAM or no graphics card supported by OpenGL can be found, the application will attempt to use software emulation. For optimal performance, graphics display color depth should be set to 24-bit or higher. When using a color depth setting of 16-bit, some inconsistencies will be noted.
Screen Resolution	A minimum screen resolution of 1280x1024 is required, but higher is recommended. **Note:** 4k displays are currently *not* supported.
Hard Disk	Requirements will vary depending on the modules you are installing. A typical minimum is 500MB free space.

Supported Operating Systems	• Windows 11 (64-bit) - Enterprise • Windows 10 (64-bit) - Home, Pro, Enterprise, and Education • Windows 8.1 (64-bit) - Standard, Pro, and Enterprise **Note:** You must have the following Microsoft updates installed on your machine: • KB4340917 - For Windows 10 Version 1803 Builds prior to 10.0.17134.191. The changes in KB4340917 are built into the August 2018 monthly update of Windows 10. • KB2999226 - For Windows 8.1, Windows Server 2008 R2 SP1, and Windows Server 2012 • KB2999226 - For Windows 8.1 (64bit) • KB2919355 - For Windows 8.1 and Windows Server 2012 Bentley does not support its software running on Microsoft operating systems versions that Microsoft has "retired". For more information on Microsoft's application retirement policy, click here. For similar information on Bentley products, refer to the Bentley Product Support article. **Note:** Testing is performed on the latest operating system updates from Microsoft at the time of release.

Additional RAM, disk space, and video memory will enhance the performance of STAAD.Pro.

The minimum amount of physical + virtual memory required by the program is over 600MB. You may need to ensure that adequate amounts of virtual memory are available and that parameters such as paging file sizes should be large enough or span over multiple drives if the free space on any one drive runs low.

Another issue to keep in mind is the location of the `TEMP` parameter as in the `SET TEMP` environment variable in Windows. While performing calculations, depending on the structure size, the program may create very large scratch files which are placed in the folder location associated with the `TEMP` parameter. You may want to point the `SET TEMP` variable to a folder on a drive that has disk space sufficiently large to accommodate the requirements for large size structures.

You should have a basic familiarity with Microsoft® Windows® systems in order to use the software.

GS. Installation and Licensing

For details on installation and licensing of this product, please refer to the product Readme file, which is installed in the same location as the product and this help file.

GS. STAAD.Pro License Options

	STAAD.Pro	STAAD.Pro Advanced	Structural Enterprise License
Structural Analysis and Design	✓	✓	✓
Advanced Analysis	-	✓	✓
Building Planner[1]	Included with SELECT	Included with SELECT	✓
AutoDrafter	Included with SELECT	Included with SELECT	✓
Separate license are required for the following:			
Foundation Design	Isolated, combined, and pile cap foundations only		✓
Connection Design	A subset of basic BCF and BCW connections only		✓
STAAD Advanced Concrete Design	Beam, column, and wall design only	✓	✓
Nuclear Design Codes	Separate license required		

[1]Building Planner was formerly known as STAAD PlanWin.

Related Links

- *GS. STAAD.Pro License Configuration dialog* (on page 57)
- *GS. Starting STAAD.Pro* (on page 33)

GS. Limits on Models

The following limits to model size are effective for STAAD.Pro CONNECT Edition V22 Update 2 (release 22.02.00) and later.

- Number of joints: 400,000*
- Joint number 1 to 999,999
- Number of Members, Physical Members, Plates, and Solids: 500,000*
- Member/Element numbers: 1 to 999,999
- Number of primary and combination cases: 10,101
- Load Case numbers: 1 to 99,999
- Number of modes and frequencies: 2,700
- Number of load cases that may be combined by a Repeat Load or Load Combination command: 550

* Some STAAD.Pro copies are available with much smaller limits, please check what limits you have purchased.

Notes:

The numerical limits should be considered as upper limits built into the software for those quantities on an individual basis. In practice, the actual maximums the program can handle are determined by the hardware resources as well as the limits imposed by the operating system. For example, it is highly improbable that a single model with 500,000 members and 10,101 load cases can be solved.

The memory demand of the program is determined by the combined effect of two or more of these terms. For example, when a steel design is performed, the memory required depends on the product of the members being designed (NMD) as well as the number of load cases being designed for (NL). That is, NMD ×NL. So the smaller the NMD, the larger the NL capacity and vice versa.

GS. Fundamentals

This section introduces you to some of the fundamental concepts necessary for using STAAD.Pro. You are strongly encouraged to review this section to gain a firm understanding of the core concepts of the program.

GS. Starting STAAD.Pro

1. Start STAAD.Pro by one of the following methods:

   double-click the STAAD.Pro CONNECT Edition icon on your desktop

   or

   click the Windows Start button and then select **Bentley Engineering** > **STAAD.Pro CONNECT Edition V22**

   or

   in the program group, select the STAAD.Pro icon.

   or

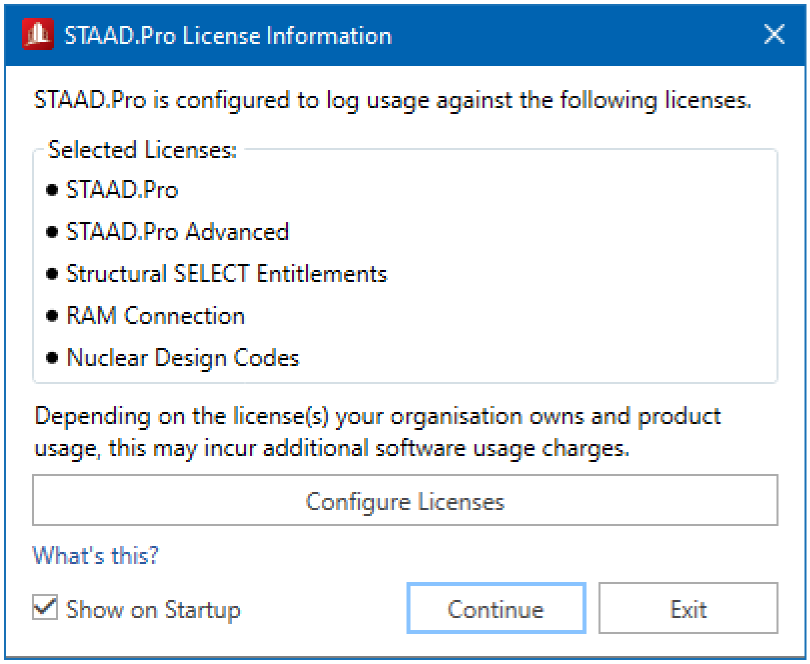
   in Windows Explorer, double-click a STAAD.Pro file (with the file extension `.std`)

   or

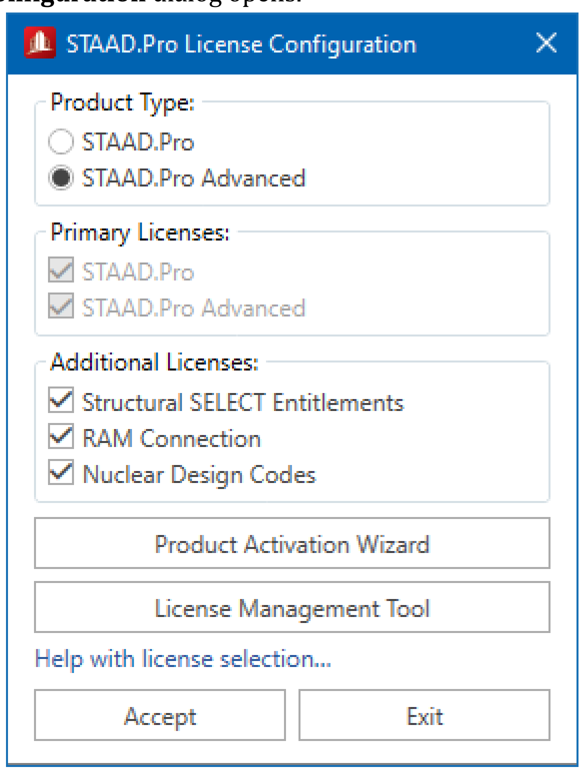
   in Windows Explorer, double-click the icon for `STAADPro.exe`.

   Note: Until you have activated a license for the program, it will remain in Limited mode and only function for 15 minutes before shutting down the application.

   The **STAAD.Pro License Information** dialog opens.



2. (Optional) To configure the license for this session:
   a. Click **Configure Licenses**.
   The **STAAD.Pro License Configuration** dialog opens.



   b. Select the **Product Type** and any **Additional Licenses** you need to use for this session.
   Please refer to the product ReadMe or click the **Help with license selection** link for additional details.
   c. Click **Accept**.

The STAAD.Pro window opens to the Start page.

3. Click **Continue**.
   The STAAD.Pro window opens to the Start page.

Related Links

- *GS. STAAD.Pro License Options* (on page 32)
- *GS. STAAD.Pro License Configuration dialog* (on page 57)
- *GS. STAAD.Pro License Configuration dialog* (on page 57)

GS. To create a new STAAD.Pro model

To create a new STAAD.Pro model, use the following procedure.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.
2. Enter a **File Name**.
3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.
4. Select the **Type** of model.
   This option selects the modeling method you want to use:

Analytical	For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
Physical	For creating a model using the STAAD.Pro Physical Modeler interface.
Building	For creating a building model structure using the Building workflow.

Note: When the **Z up** option is selected in the **Application Configuration** dialog, then the **Physical** model option is disabled for creating new models.

5. Select the system of **Units**.

English	English Imperial units: feet, pounds, seconds
Metric	System International units: meters, kilograms, seconds

Tip: The units can be changed later if necessary, at any stage of the model creation.

6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.
7. (Optional) Type a **Title** for the model.
8. Select a **Model Seed File** file by either:

   clicking **Browse** to locate a seed file

   or

   selecting a previously used seed file from the drop-down list
   A seed file is a STAAD.Pro input file that has been pre-populated with model data for use as a template.
9. Click **Create**.

The new STAAD.Pro model is opened in either the Analytical Modeling workflow, STAAD.Pro Physical Modeler, or the Building workflow based on your selection.

Related Links

- *GS. Start Page* (on page 54)
- *ProjectWise Project Association* (on page 2068)

GS. To open a STAAD.Pro model

To open an existing STAAD.Pro model, use the following procedure.

1. On the Start page, select **Open**.
   The New page opens to the **Recent** tab.

   Recently opened models are organized by the following categories:

   Analytical
   Physical
   ProjectWise
   Archive
   All
2. Either:

   select a recently used model from one of these tabs

   or

   select **Open Other Models** to search for a STAAD input file

   or

   select either **Sample Models** or **Verification Models** to open the example models which are installed with the program

   If you select a recent model, it will open. If you select to browse for a model, an **Open** dialog opens.
3. In the Open dialog, navigate to and select a STAAD.Pro file (file extension `*.std`) and then click **Open**.
   The selected file opens.

If the STAAD.Pro file does not have a ProjectWise Project associated with it, then you will be prompted to select one.

Related Links

- *GS. Start Page* (on page 54)
- *ProjectWise Project Association* (on page 2068)

GS. Workflows in STAAD.Pro

The program is organized to reflect the typical process of modeling, analyzing, and post-processing for a structure.

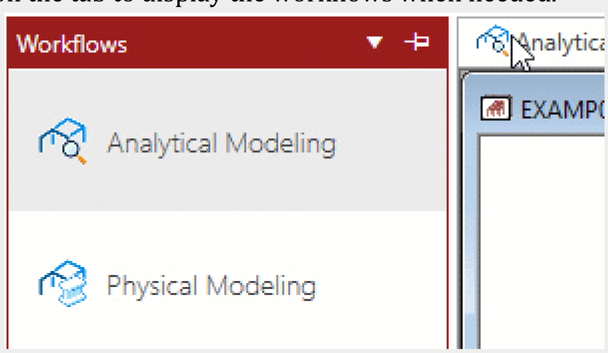
A workflow in STAAD.Pro groups all of the common tasks associated with a major stage of your structural project.

Overview of Workflows

Modeling (on page 629)	used to model your structure using analytical objects
M. Physical Modeling workflow (on page 919)	opens the STAAD.Pro Physical Modeler application, which is used to model using physical objects which will be subsequently decomposed into analytical objects when exported back into the STAAD.Pro interface
M. Building Planner workflow (on page 920)	used to generate concrete building models with the integrated STAAD Building Planner interface
M. Piping workflow (on page 902)	used to import piping support geometry and reactions from Bentley AutoPIPE
P. Postprocessing Workflow (on page 1979)	used to review the results of analysis and design as well as to build reports
D. Foundation Design (on page 1078)	used to select load and support data to send to STAAD Foundation Advanced for design
P. Steel AutoDrafter Workflow (on page 2018)	used to generate construction drawings for steel-framed structures
D. To open a model in the Chinese Steel Design workflow (on page 979)	used to design the analyzed steel structure per the Chinese design code in SSDD

D. Connection Design workflow (on page 1014)	used to specify steel connection types and design them in RAM Connection
D. Advanced Concrete Design (on page 1071)	opens the Advanced Concrete Design (RCDC) application, which is used for the design of concrete building structures
D. Advanced Slab Design (on page 1073)	used to define concrete slabs and transfer data to RAM Concept
P. Earthquake workflow (on page 2032)	this postprocessing workflow is used to check if the structure conforms to the basic geometric recommendations made in Eurocode 8 (EC8).

Tip: Once you are familiar with the Workflows, you can free up some screen area by unpinning the **Workflows** panel from the program window. It will then collapse to a tab on the left side of the application window. You can expand the panel by clicking on the tab to display the workflows when needed.



Where did the Modes and Page Controls go?

In previous generations of STAAD.Pro, a series of page controls were listed vertically along the left side of the view window. A series of operation modes were listed along the top of the view window (and also accessed via the **Modes** menu.

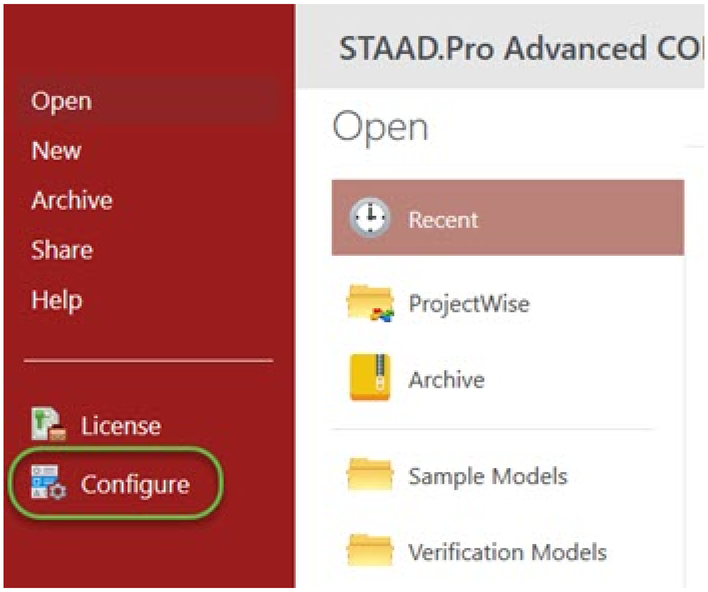
These modes have now been replaced by the **Workflows** panel. Similarly, the page controls are available for the current workflow just below the ribbon bar. These are now organized in a single row, rather than the former groupings.

GS. To customize the workflows panel

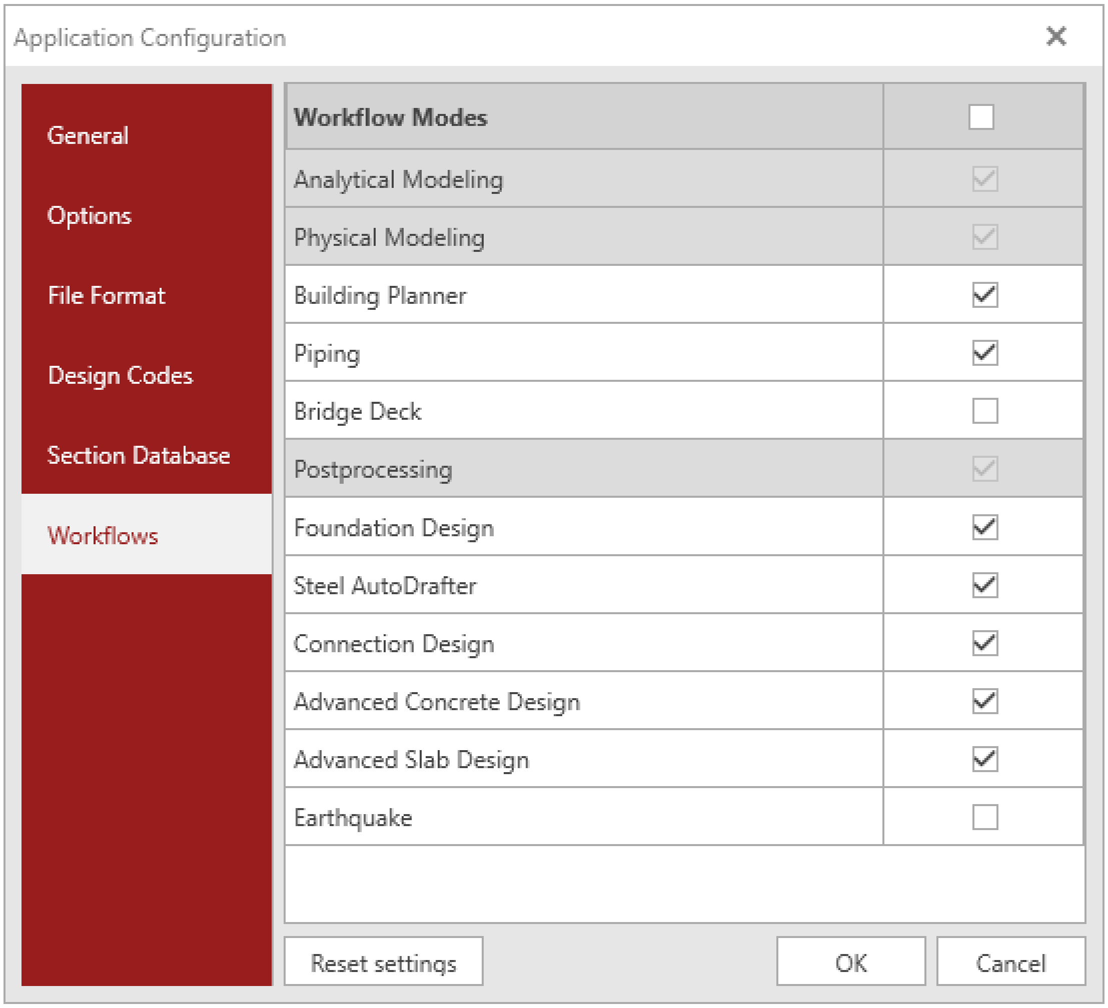
To hide unused workflows in the STAAD.Pro interface, use the following procedure.

Close any open STAAD.Pro models to display the Start page.

1. On the **Start** page, click **Configure**.



   The **Configure Program** dialog opens.
2. Select the **Workflows** tab.



3. Uncheck any workflows you want hidden in the application interface.

   Tip: To reset the Workflows display to the default, check the **Workflow Modes** option.

4. Click **OK**.

The next time you open a model in STAAD.Pro these workflows will be hidden. You can un-hide them at any time by checking the options in the **Application Configuration** dialog.

Related Links

- *GS. Application Configuration dialog* (on page 59)

GS. Selecting Objects in STAAD.Pro

With even relatively small structural models containing many nodes, beams, plates, loads, and other model objects, it is important that you be able to select the correct objects for assigning parameters. STAAD.Pro has a variety of tools to allow you to select objects by type, filter, boolean logic, and more.

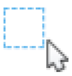
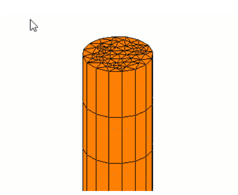
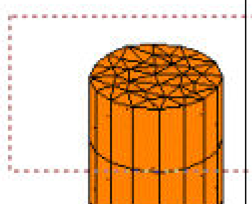
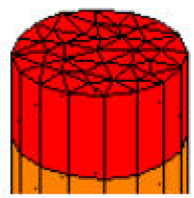
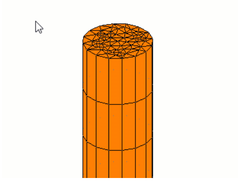
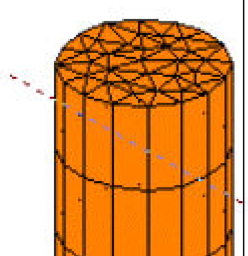
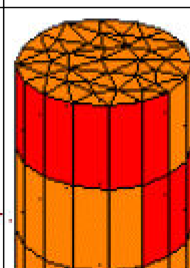
In certain modes of selection, objects are chosen by clicking on their entry in a list. For selecting more than one object, press and hold **<Ctrl>** while clicking.

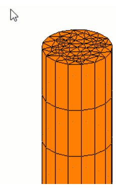
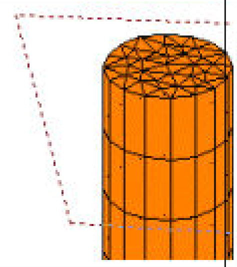
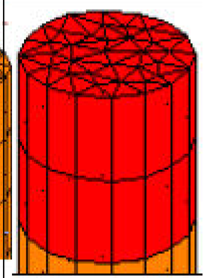
Selection Method

STAAD.Pro allows you to select one of several methods for graphical selection. These are described in the table below.

To choose a different the selection method, either:

- Select it from the **Select** ribbon tab in the **Modes** group, or
- right-click in the view window and select **Selection Mode** from the pop-up menu.

Selection Type	Description	Mouse Action	Resulting Selection Set	Shortcut
 **Drag box**	A rectangular selection box activated by holding down the left mouse button and dragging the pointer to create a windowed area in the View window. **Tip:** This is sometimes referred to as a “rubber band” window. 			
 **Drag line**	Click and hold down the left mouse button to draw a line. All entities which the line passes through will be selected. 			**<Crtl+Shift+F3>**

Selection Type	Description	Mouse Action	Resulting Selection Set	Shortcut
 **Region**	Click the outer points of any polygonal region. Close the polygon by double-clicking. All entities fully enclosed in the polygon are selected. 			**<Crtl+Shift+F2>**

GS. Units in STAAD.Pro

STAAD.Pro uses a set of base units, called current units, for all input and output. The Current Units can be changed at any time to simplify entering input or interpreting results. The current input units are set using the **Set Current Input Units** dialog.

Tip: There are several some instances in the program where the Current Units cannot be changed, such as when an input dialog is open. In this case, you can use the **<F2>** key to input different units.

STAAD.Pro also includes an extensive unit conversion utility.

Display Units

STAAD.Pro can use a different set of units for displaying values in the active view window. These are controlled through several tabs on the **Options** dialog.

Base Unit System

There are two base unit systems in the program which control the units (i.e., length, force, temperature, etc.) in which, values, specifically results and other information presented in the tables and reports, are displayed in. The base unit system also dictates what type of default values the program will use when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types - Steel, Concrete, Aluminum - selected from the program's library (Please refer to Technical Reference of STAAD Commands (on page 2196) for details). These two unit systems are:

English (Foot, Pound, etc.) and
Metric (KN, Meter, etc.)

Tip: If you recall, one of the choices made at the time of installing STAAD.Pro is this base unit system setting. That choice will serve as the default until you specifically change it.

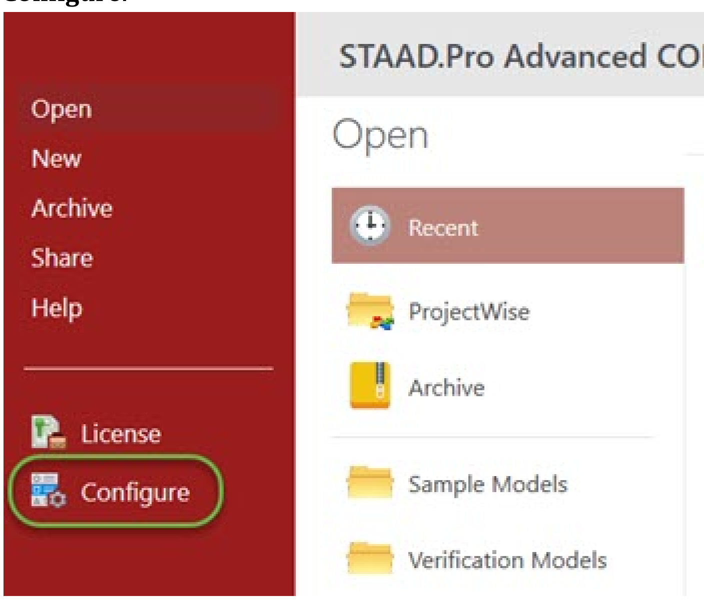
Selecting Different Input Units in Dialogs

While you may typically wish to work in one unit system - or even with a specific unit of force or length, it is not uncommon to use a different unit for input. STAAD.Pro allows for the input of a value in any unit in most dialog boxes. This avoids the need to manually change the unit before and after adding a command or to convert a unit to something less commonly used. A unit is entered in the most convenient format and then converted to the current units for you. This is especially useful when adding or editing loads, where the current input units cannot be changed.

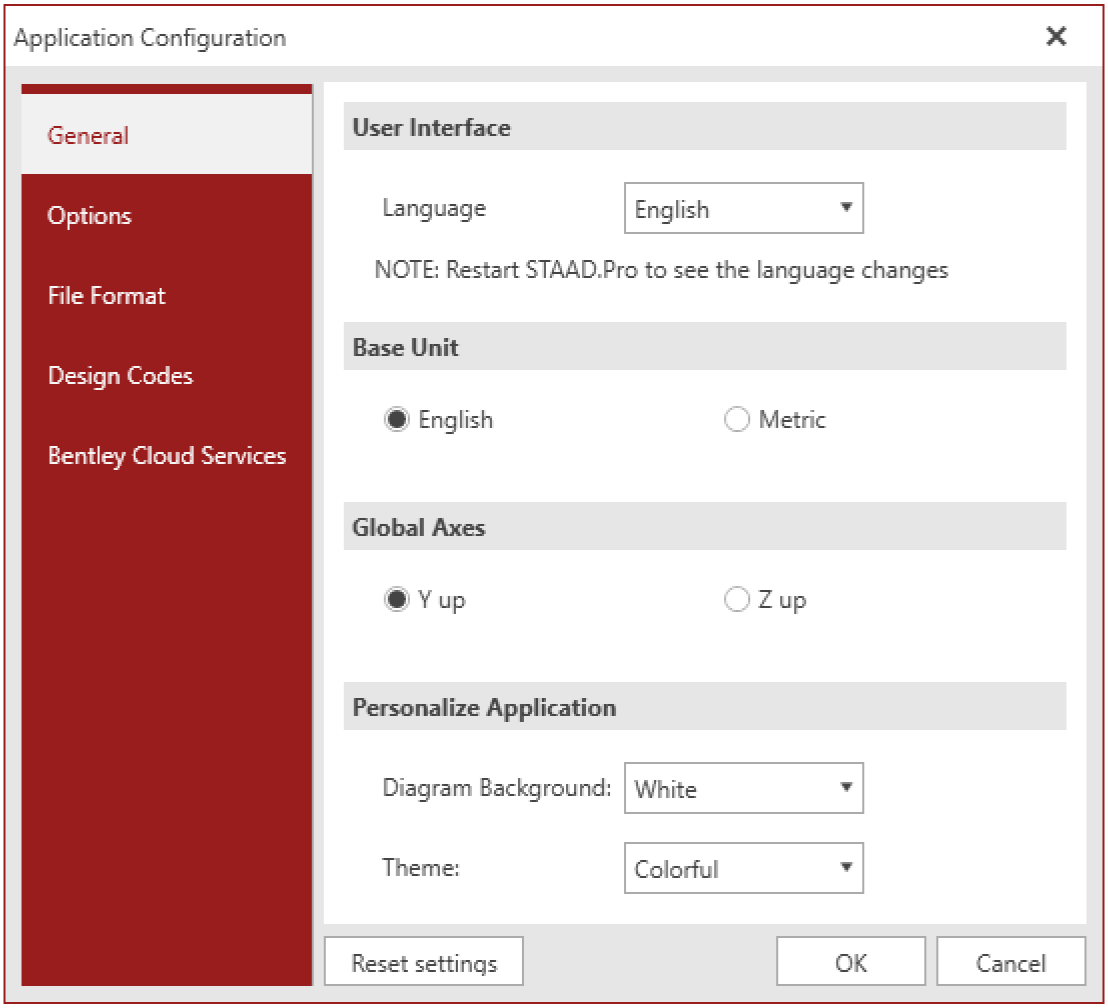
GS.To change the system units

There are two base unit systems in the program which control the units (length, force, temperature, etc.) in which values — specifically results and other information presented in the tables and report— are displayed. The base unit system also dictates what type of default values the program uses when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types (i.e.., steel, Concrete, Aluminum) selected from the program's library (Please refer to Built-In Material Constants (on page 2308)). These two unit systems are English (Foot, Pound, etc.) and Metric (KN, Meter, etc.).

1. On the **Start** page, click **Configure**.



   The **Configure Program** dialog opens.
2. Select appropriate system of units in the **Base Unit** group on the **General**.



3. Click **OK**.

Related Links

- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *GS. Application Configuration dialog* (on page 59)

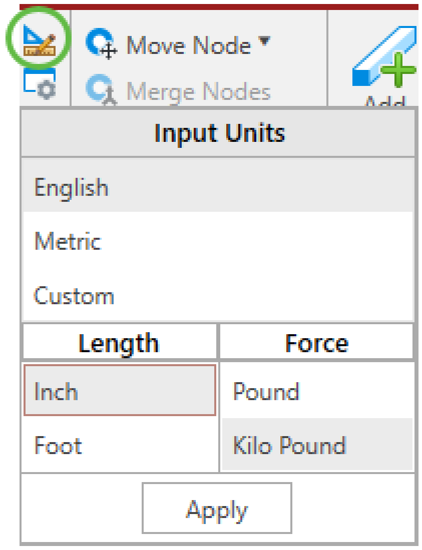
GS. To set the current input units

To change the current length and force units for inputs, use the following procedure.

The current input units used for creating the model and assigning attributes such as properties, offsets, loads, etc. can be changed to use convenient units.

1. On the **Geometry** ribbon tab, select the **Input Units** tool in the **Structure** group.

The **Set Input Units** pop-up opens.



2. Select the base system of **Input Units**:

   English

   or

   Metric

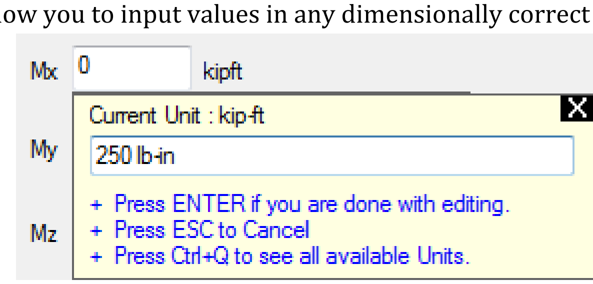
   or

   Custom - to mix different systems of units
3. Select a unit for **Length** on the left and unit for **Force** on the right.
4. Click **Apply**.

GS. To convert units within a dialog field

To input a unit different than the current unit within a dialog field, use the following procedure.

1. Click within a field that takes a dimensional value as input.
2. Press **<F2>**.
   A pop-up dialog opens to allow you to input values in any dimensionally correct unit.



3. (Optional) Press **<Ctrl+Q>** to open a list of dimensionally correct units and select one to populate this pop-up dialog.
4. Type the value in the known units.

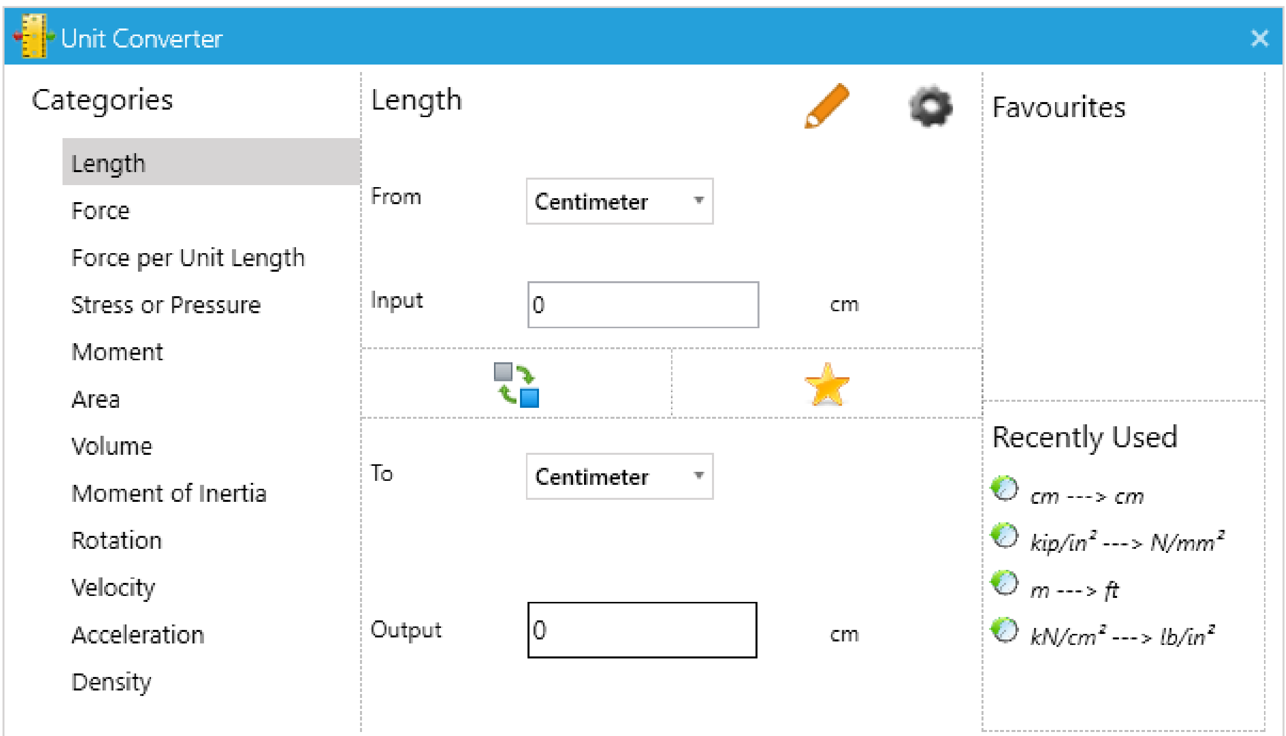
   Tip: Values can also be input using a fractional system (e.g., 5 7/8").

5. Once you are finished entering the units, press **<Return>**.
   The unit is converted into the Current Unit system.

GS. **STAAD.Pro Converter** utility

An extensive unit converter is included with the program which can convert values from one unit system to another.

Opens when the **Unit Converter** tool is selected from the **Tools** group on the **Utilities** ribbon tab.



Tools

<table>
<tr><th>Tool icon</th><th>Description</th></tr>
<tr><td>
Custom</td><td>Click to add a custom unit conversion to the current unit category. In the Custom Units table, type a From unit label, the conversion Factor, and a To unit label and then click Ok.

For example, you can type the following to add yards to the Length conversion:
<table><tr><th>From</th><th>Factor</th><th>To</th></tr><tr><td>Feet</td><td>3</td><td>Yards</td></tr></table></td></tr>
</table>

Tool icon	Description
 **Settings**	Click to open the **Settings** dialog.
 **Swap units**	Click to switch the From and To units for the current category.
 **Add to favourites**	Click to add the current conversion settings to the **Favourites** list on the right.

Settings dialog

Click the **Settings** tool to display the **Unit Converter** settings.

Setting	Description
Theme	Select a display theme from the drop-down list.
Output decimal places	Set the decimal places to display in the conversion output.
Max no of favourite items	Set the number of items to display in the favourites list.
Max no of recently used items	Set the number of items to display in the recently used items list.
Top Most	Check this settings to keep the **Unit Converter** dialog on top of all other windows on your screen, even when other windows are active.

GS. Coordinates in STAAD.Pro

STAAD.Pro Physical Modeler uses a conventional Cartesian coordinate system, with the global Y axis assumed as vertical (i.e., “Y up”, or the height of the structure is parallel to the global Y axis).

Global Coordinate System

This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal “right hand” rule. This coordinate system may be used to define the joint locations and loading directions. The translational degrees of freedom are denoted by u1, u2, & u3 and the rotational degrees of freedom are denoted by u4, u5, & u6.

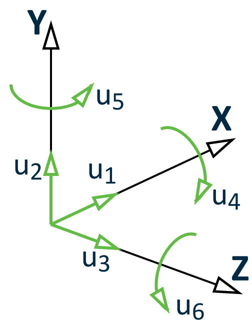
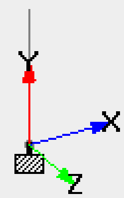


Figure 1: Degrees of freedom as used in STAAD.Pro

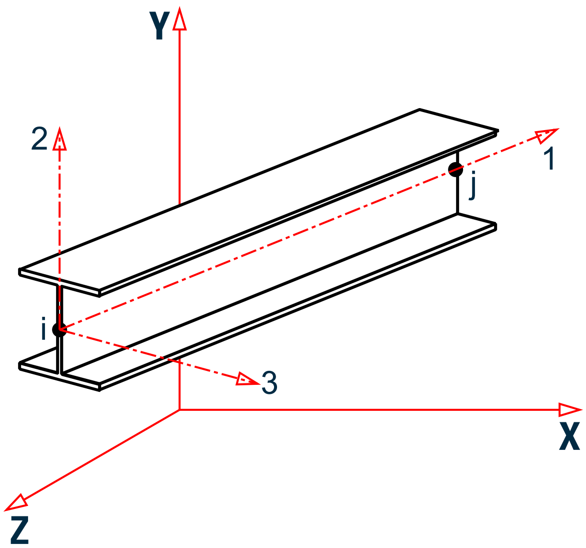
Note: You can toggle on a color-coded axes label placed at the origin (**<Shift+I>**) in which: X is blue, Y is red, and Z is green.



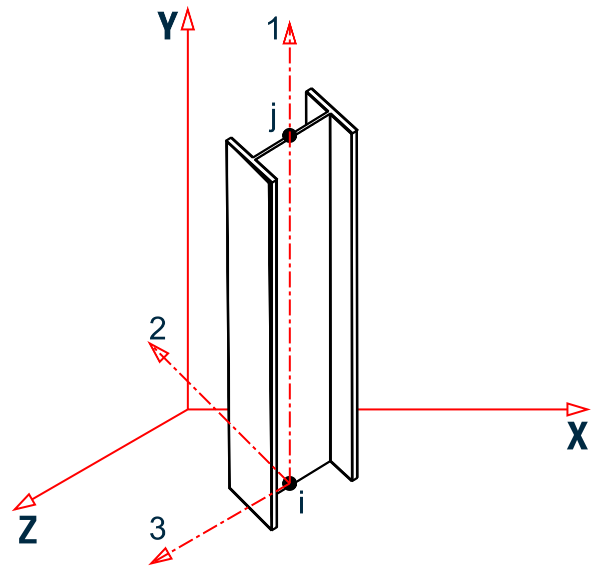
Local Coordinates

Each member or surface has its own local Cartesian coordinate system which is also oriented using the “right hand” rule.

The longitudinal axis of a member is the first axis, with the positive axis taken from the i to the j ends (i.e, Ni and Nj). The second axis is the oriented such that the 1-2 plane is then parallel to the global Y axis. The third axis is then normal to the first and second local axes as defined by the right hand rule. The local 2 and 3 axes coincide with the two principle moments of inertia of the cross-section. In the special case of a vertical member (where the local 1 axis is parallel to the global Y axis; i.e., a column), the local 3 axis is then made parallel to the global Z axis and the 2 axis is oriented respectively.



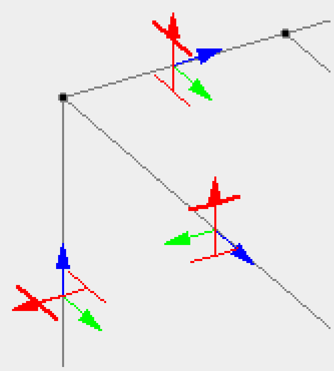
(A)



(B)

Figure 2: Local coordinates of (A) a member with arbitrary orientation and (B) oriented vertically

Note: Local axes (**<Shift+O>**) are color-coded as follows in the graphical view window: x is blue, y is red, and z is green.



For surfaces, the local x axis is aligned with the edge defined by the first two nodes of the surface (i.e., N1 and N2). The program then calculates the area of each triangle formed by any other nodes to determine the largest area. This triangle determines the plane of the surface and the local y axis lies perpendicular to the x axis within this plane. The third axis is then orthogonal to the surface as defined by the “right hand” rule.

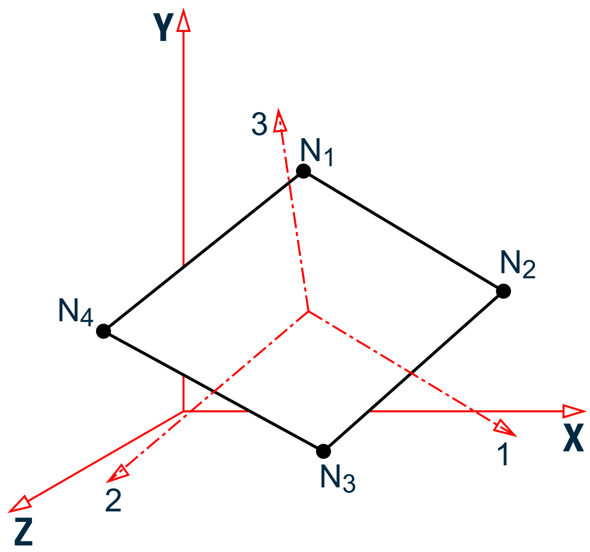
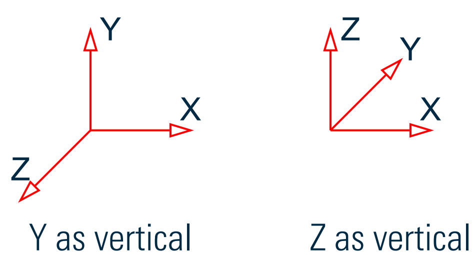


Figure 3: Local coordinates of a surface

GS. To use Z as the vertical axis

To re-orient the global axis so that Z is vertical, use the following procedure.

You must close any open STAAD.Pro models in order change program configuration.



Note: Both options follow the "right hand rule."

Figure 4: Global axis orientation options in STAAD.Pro

Important: Not all STAAD.Pro features are compatible with Z UP orientation. Refer to SET { Y | Z } UP (on page 2210) for additional details.

1. On the Start page, select **Configure**.
   The **Application Configuration** dialog opens.
2. On the **General** tab, select **Z up** in the **Global Axis** group.
3. Click **OK**.

New models will use the Z up orientation of the global axis.

Tip: You can repeat this procedure except to select Y up to return to set Y as vertical.

Any input files which have the `SET Z UP` command will automatically use this orientation.

GS. Load Types in STAAD.Pro

Several load cases may be created for a structure and each load case may contain several individual load specifications. Load cases may also be created by combining several existing load cases. A load case consisting of explicitly defined loads is called a Primary Load Case. A load case which combines the results of previously defined cases is called a Combination Load Case.

Tip: Here, we wish to introduce you to some of the load terminology and types used in STAAD.Pro. These concepts are critical in understanding how to correctly model loads on a structure.

Primary Load Cases

A primary load case is a set of explicitly defined loads, presumably from the same physical source, which will be passed to the analysis engine during the analysis of the model. Each of these explicitly defined loads is referred to as a Load Item.

In the STAAD input file, a primary load case is indicated by the `LOAD n` command.

Refer to TR.32 Loading Specifications (on page 2461) for details.

Some examples of where a primary load case would be used are:

- All dead load on a structure.
- The reducible live load on an office building.
- The easterly wind load.
- The dead and live composite load on a pedestrian bridge.
- The superimposed dead load on a post-tensioned floor.

Load Combinations

A load combination is a set of load results which are combined algebraically to produce a superimposed set of results for post-processing. Therefore, a load combination instructs the program to take the results of previously solved primary load cases, factor them appropriately, and combine the values using Algebraic, SRSS or Absolute methods.

In the STAAD input file, a load combination is indicated by the `LOAD COMBINATION n` command.

Refer to TR.35 Load Combination Specification (on page 2616) for details.

Reference Loads

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. Reference Loads may be defined for this purpose. This is similar to a Repeat Load command, but has the added benefit of not being solved in its own right.

This converts a real load case to something similar to a load case definition. A reference load case is solved only when it is later called in a load case. The benefit is that it enables you to define as many load cases as you wish, but instruct the program to actually solve only a limited number of "real" load cases, thus limiting the amount of results to be examined.

Refer to TR.33 Reference Load Cases - Application (on page 2613) for details.

Notional Loads

A number of design codes require that a notional load be considered. Typically, this is defined lateral load equal to a percentage of the gravity loads. STAAD.Pro uses a feature similar to a the Repeat Load, but one that Primary and /or Reference load cases can be selected and a percentage of them can be applied in the appropriate global direction at each framing level.

These lateral loads are a requirement for some design codes (e.g., AISC 360-10).

Refer to TR.32.13 Notional Loads (on page 2610) for details.

Load Lists

A load list is primarily used to specify a list of existing load cases and load combinations to be used for subsequent processes, such as design, printing, etc.

Refer to TR.39 Load List Specification (on page 2662) for details.

Load Envelopes

Load Envelopes are a means for clustering a set of load cases under a single moniker (number). If one or more tasks have to be performed for a set of load cases (such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc.) this feature is convenient.

This is an alternative to Load Lists, and is primarily used in post-processing and design. Load Envelopes also have keyword types which identify their intended use in design.

Refer to TR.40 Load Envelope (on page 2663) for details.

Load Definitions

Definitions contain the options you use to define data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases, snow load cases, and time history load cases.

Refer to TR.31 Definition of Load Systems (on page 2345) for details.

GS. STAAD Input Files

A STAAD model consists of a data file made up of simple, English-language like commands, using a format native to STAAD.Pro. This file specifies all input, analysis, and output commands used to inform the STAAD engine as to the geometry of the structural model, loads acting on this model, and instructs the engine on the analysis and design operations to be performed.

There are two methods for building a model and assigning the structure data using STAAD.Pro:

a. using the graphical model generation, or
b. directly editing the command input file using the STAAD.Pro Editor

The graphical method of creation (a) involves using either the Analytical Modeling or Physical Modeling workflow to draw the model using the graphical tools, and assigning data such as properties, material constants, loads, etc., using the various tools and dialog boxes of that mode. In this approach, the command file is automatically created by the program while you work. The input command file is updated whenever your structure is saved.

The direct editing method (b) involves creating or editing the command input file (file extension `.std`) directly using the STAAD Editor utility. These commands are explained in detail in Technical Reference of STAAD Commands (on page 2196) and several examples which illustrate this method are provided in the Application Examples (on page 6251).

Tip: The command input file is a plain text file so you can use any plain text editing software you prefer. However, it is recommended you use the STAAD.Pro Editor as this program contains tools such as Intellisense to complete commands and syntax checking.

Tip: For a hands on illustration of both these methods, it is recommend that you take a look at the Tutorials (on page 387).

Regardless of the means used to *create* the STAAD input file, once the model is ready for analysis, you will use the graphical environment to initiate analysis, review the analysis and/or design output data, and perform advanced design operations.

GS. Object Properties Inspection

You can inspect the properties and results of most analytical and physical model objects by double-clicking them in the view window.

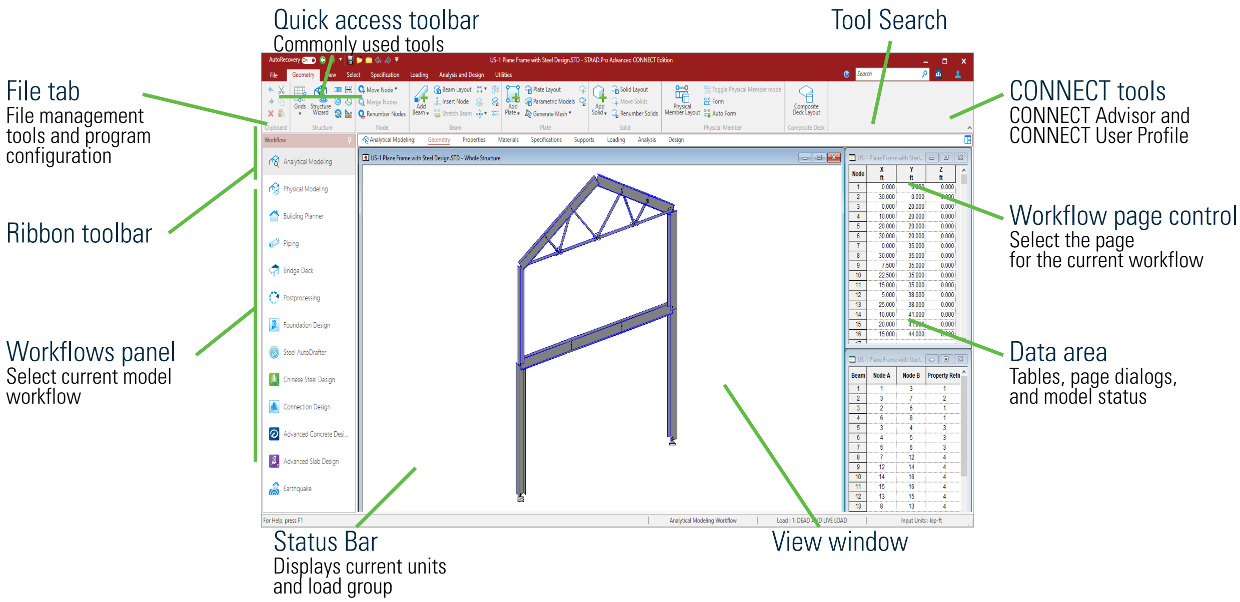
The dialog that opens reflects the type of object selected as well as the assigned specifications, properties, available analysis and design results, etc.

Related Links

- *Node dialog* (on page 3046)
- *Beam dialog* (on page 3050)
- *Physical Member dialog* (on page 3053)
- *Plate dialog* (on page 3059)
- *Surface Query dialog* (on page 3060)
- *Solid dialog* (on page 3062)

GS. Application Window Layout

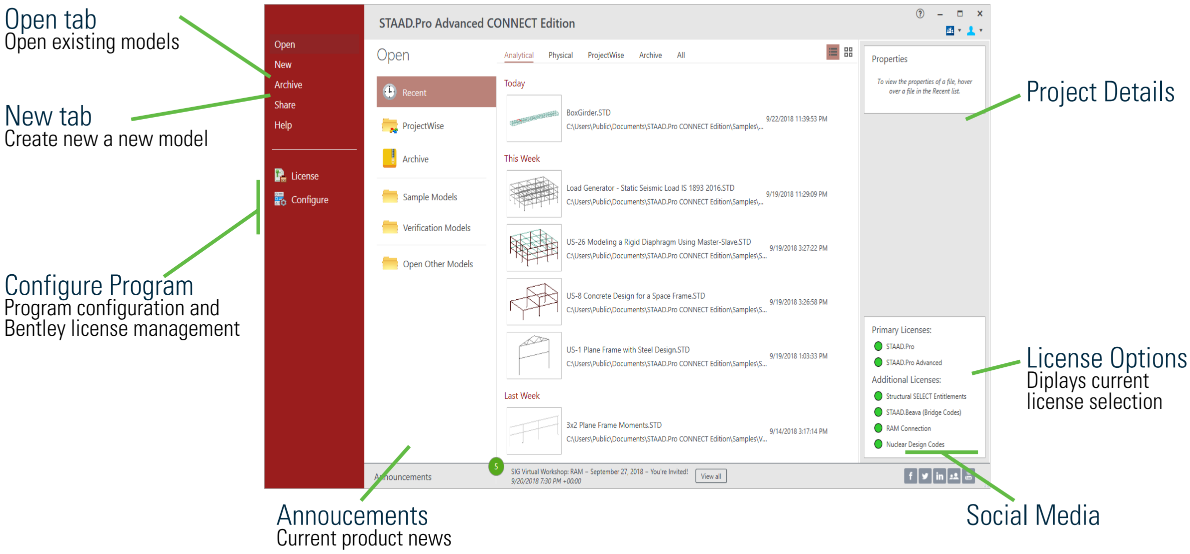
It is helpful to take some time to familiarize yourself with the components of the window.



1. File tab (on page 2694)
2. Ribbon Control Reference (on page 2694)
3. GS. Workflows in STAAD.Pro (on page 36)
4. GS. Quick Access Toolbar (on page 62)
5. GS. Tool Search (on page 64)
6. Bentley CONNECT Features (on page 2068)
7. GS. Page Control (on page 64)
8. GS.Data Area (on page 66)
9. GS. Status Bar (on page 84)
10. GS. View Window (on page 67)

GS. Start Page

The start page is displayed when STAAD.Pro opens or there is no model currently open.



Open tab

Used to open an existing STAAD input file or Archive. You can also open STAAD projects from a ProjectWise data source.

Select the **Recent** tab to view recent files by type. Hover your mouse pointer over any file to display the file and CONNECT Project properties.

The **Additional License** options are available here to allow you to select which license options you want to use before opening a file.

Tip: If you hover your mouse pointer over any item in the recent files list, you will see several tool icons on the right-hand side:

- **Pint to list** – pin this project to the top of the list
- **Remove from list** – remove this project from the list
- **Open Containing Folder** – open the folder where the project is saved in Windows Explorer

New tab

Used to create a new STAAD project. Options are available to start with an analytical model, a physical model, or in STAAD Building Planner.

The **Additional License** options are available here to allow you to select which license options you want to use before creating a file.

Archive tab

Used to create or extract STAAD.Pro archive files.

Share tab

Table 1: Share menu items

Tool name	Description
 **Send Mail**	Used to share STAAD.Pro project files as an e-mail attachment. The selected project files associated with a STAAD.Pro input file are saved as a STAAD.Pro ZIP file (file extension `.stz`).
 **ProjectWise**	Used to save STAAD.Pro project files to a ProjectWise data source. The selected project files associated with a STAAD.Pro input file are saved as a STAAD.Pro ZIP file (file extension `.stz`).

Help tab

Table 2: Help menu items

Tool name	Description
Contents	Opens the online help in your web browser.
OpenSTAAD Help	Opens the online help in your web browser to the OpenSTAAD reference section.
Technical Support	Opens the Worldwide Technical Support Resources dialog, which offers a map of worldwide technical support contacts for STAAD.Pro.
ReadMe	Opens the STAAD.Pro Read Me file in your web browser.
Knowledge Base and FAQs	Opens the STAAD.Pro Support Solutions page on Bentley Communities in your web browser.
Discussion Group	Opens the RAM | STAAD Forum on Bentley Communities in your web browser.
Tutorials	Opens the Bentley LEARNserver in your web browser. Here you can access training and learning paths for STAAD.Pro and other Bentley products.

Table 3: About menu items

Tool name	Description
About STAAD.Pro	Opens the About STAAD.Pro CONNECT Edtion dialog, which contains version, licensing, and legal information about the product. Any Technical Preview items contained in the current version will also be listed here.
Product News	Opens the STAAD.Pro product page at Bentley.com in your web browser.
Home Page	Opens the Bentley.com home page in your web browser.
Ideas	Provide feedback with any ideas you have to improve STAAD.Pro.

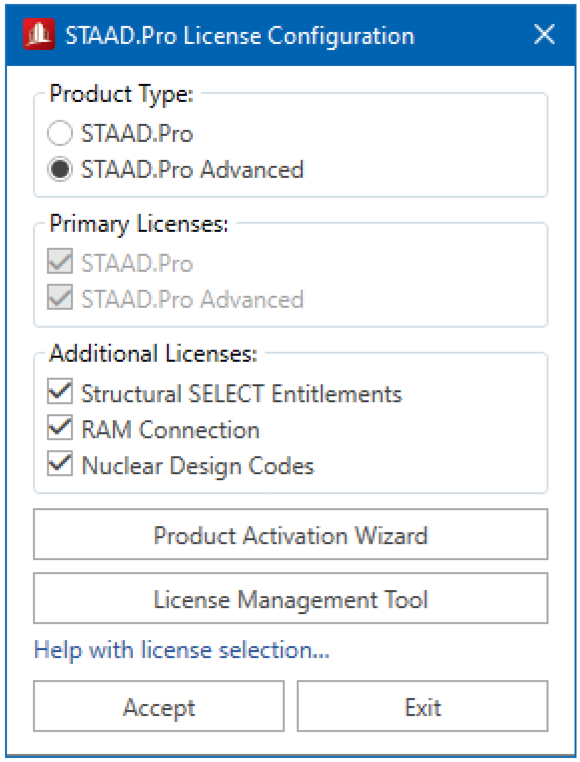
Related Links

- *GS. To create a new STAAD.Pro model* (on page 35)
- *GS. To open a STAAD.Pro model* (on page 36)
- *M. To start a STAAD Model in the Building Planner workflow* (on page 920)
- *I. To open a STAAD input file from a ProjectWise repository* (on page 2074)
- *I. To create an archive* (on page 2067)
- *I. To open an archive file* (on page 2067)
- *I. To extract an archive* (on page 2068)
- *I. To share a STAAD.Pro project in ProjectWise* (on page 2076)

GS. **STAAD.Pro License Configuration** dialog

Used to manage license options for STAAD.Pro.

Opens when you select **License** from the Start page menu.



Product Type

Select the primary license to use for the application here.

- **STAAD.Pro** – using this license you will have access to the primary features and capabilities available in STAAD.Pro, including all design codes
- **STAAD.Pro Advanced** – The extends the basic STAAD.Pro application with additional capabilities such as an eigen-based advanced buckling analysis, geometric non-linear analysis, and other advanced analysis methods, a faster analysis solver, and access to advanced concrete design using RCDC.

Primary Licenses

This displays the status of each license type associated with your account.

Additional License

Select additional licenses to use:

- Structural SELECT Entitlements
- RAM Connection
- Nuclear Design Codes

Product Activation Wizard

Opens the CONNECT License Client Activation window, which is used to activate or to reserve a license.

License Management Tool

Opens the Bentley Licensing Tool window, which is used to manage your current license entitlements, managed reserved licenses, and review the status of product licenses available to you.

Help with license selection

Click to open the latest details on license selection information on Bentley Communities.

Accept

Save the changes to the license configuration and close the dialog.

Exit

Close the dialog without saving any changes to the license configuration.

Related Links

- *GS. STAAD.Pro License Options* (on page 32)
- *GS. Starting STAAD.Pro* (on page 33)
- *GS. Starting STAAD.Pro* (on page 33)

GS. **Application Configuration** dialog

Used to set program-wide parameters for STAAD.Pro.

Opens when **Configure** is selected on the application Start page.

General tab

Language	Select the user interface language from the drop-down list. You must restart the program to see that language used in user interface.
Base Unit	There are two base unit systems in the program which control the units (length, force, temperature, etc.) in which, values – specifically results and other information presented in the tables and reports – are displayed. The base unit system also dictates what type of default values the program will use when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types – Steel, Concrete, Aluminum – selected from the program's library . These two unit systems are **English** (Foot, Pound, etc.) and **Metric** (KN, Meter, etc.)
Global Axis	Select which axis represents the gravity direction in your model by default. Subsequently all modeling, analysis, and post-processing items would be based on this coordinate system. Select either the **Y up** or **Z up** option. **Note:** Some features in STAAD.Pro require that Y up be used for the global axis orientation. **Note:** When the **Z up** option is selected in the **Application Configuration** dialog, then the **Physical** model option is disabled for creating new models.
Diagram Background	Select either a **White** or **Black** background for the view window. This selection will also set some default colors in which beams, plates and solids are drawn. For example, a white background is accompanied by black lines for drawing beams.
Theme	Select the basic color theme used in the application.

Options tab

Startup	**Display License Options**	Check this option during the application launch to be prompted with License Information dialog (see GS. Starting STAAD.Pro (on page 33)) listing enabled licensing options.
	Display STAAD News Feed	Check this option to open the news feed reader when the program starts.
	Display CONNECT Project Chooser when opening STAAD model	Check this option to display the CONNECT Project Chooser dialog when opening a STAAD model.

Analysis	**Warn if analysis performed across network**	Check this option to receive a warning if the analysis file is located on a network location. In some network instances, this can cause a drastric increase in analysis time.
	Use temporary location	Check this option to use a temporary location on your local computer for analysis. This can improve analysis speed for network files. The **Folder** used can be typed in or selected using by clicking **Browse**.
Additional Options	**Save Auxiliary Data on close**	During the course of performing analysis, design, and post-processing operations, multiple files associated with a STAAD project are generated. Set this option to direct the program to save all files in the STAAD project when the project is closed.
	Delete Results Data If Out of Sync	Set this option to delete results files when a project is opened if the input file has been modified more recently than those input files. This option is helpful in ensuring that results invalidated by recent changes to the input file will not be used in error.
	Remove Bentley (B) logo from Report	Set this option to remove the Bentley logo from reports generated in STAAD.Pro. **Tip:** You can always add a custom logo and branding to your reports.
	Save AutoRecovery information every X minutes	Check this option and specify the AutoRecovery time interval in order to have the program automatically save a backup of your program status. In the event of a program crash, the program will detect the autorecovery file the next time you open the STAAD project and prompt you to open the recovered state.

File Format tab

Input format	**Single Line Format Commands**	Select whether **Joint** coordinates as well as **Member**, **Plate**, and **Solid** incidences will be written one per input line or multiple per input line by setting the associated options here.
	Write expanded list	Set this option to instruct the program to write out joint, member or element numbers individually, for example: `1 2 3 4 5` instead of `1 TO 5` and consequently, creates voluminous input.
	Joint Coordinate Significant Figures	Specify the number of significant digits used for nodal coordinates. This is used for models which require a high degree of precision.
Output format	**Add key file information**	Set this option to Instruct the program to list all the key files that were used with the analysis including the date of the principal files. **Note:** Even if the general case is that the Key Information is displayed, it can be tuned off in a particular analysis if the data file contains the command `SET NOFILE`.

Error format	**Create Error File**	You may choose not to have the error file created at all, though this isn't advisable. Un-check the box provided for that purpose if you choose to do so.
	Log maximum errors per single input	A single line of input may contain more than one error. We can set how many such errors we want shown for each input line.
	Log maximum errors	You can also set the maximum number of errors you want reported for the entire file. That number is 100 by default.

Design Codes tab

STAAD.Pro supports several major international design codes. Select the default design code to use when selecting the material-based design in the program.

Note: You may not use more than one design code in one single run, even if the Input Command File has more than one `CODE` command. Also note the design code specified by the Design Code tab must match that specified in the Input Command File.

Section Database tab

Select this tab to open the Section Database dialog, which is used to select the default catalog section database file to use for each country. You can also specify user-defined tables for use in the program.

Workflows tab

Select this top to choose which workflows are displayed in the **Workflows** panel of the application.

Related Links

- *GS.To change the system units* (on page 43)
- *GS. To customize the workflows panel* (on page 39)

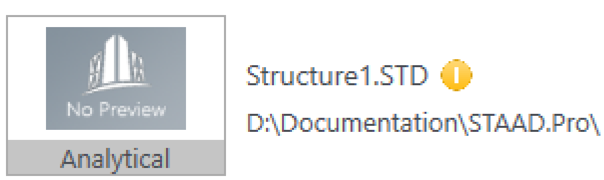
GS. To remove missing files from the recent files list

If you have some files that have moved or cannot be accessed currently, you can remove them from the recent files list on the Start page by using the following procedure.

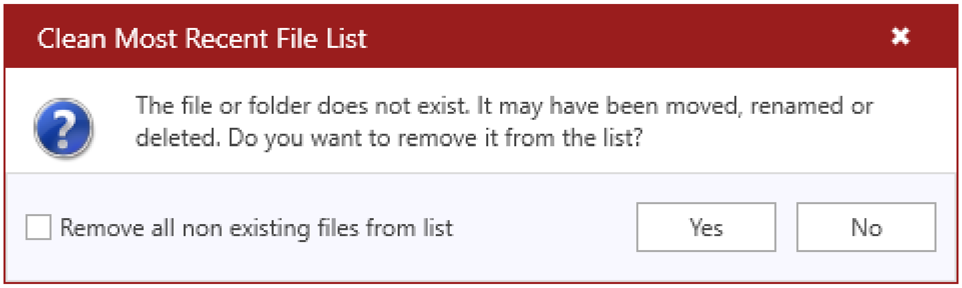
You can access the **Recent** files list from the program Start page when no models are open.

Though this procedure is entirely optional, having many recent files can slow the performance of STAAD.Pro when opening the program.

1. Click on the yellow information icon that indicates a missing file.



   A message dialog opens with the option to remove this file entry from the list.



2. (Optional) You can check the **Remove all non existing files from list** option to remove all the files that are missing.
3. Click **Yes**.

GS. Quick Access Toolbar

The quick access toolbar is located just above the ribbon controls. It contains some of the most frequently used tools in a convenient location.

To customize the Quick Access Toolbar by adding other tools, click the menu arrow and then select **More Commands**. The **Customize Quick Access Toolbar** dialog opens, which allows you to select which tools are displayed in the Quick Access Toolbar.

Note: The **Command File** and **Analysis Output** are hidden by default. Click the menu arrow in the to toggle the display of these or other tools.

Table 4: Default Quick Access Toolbar tools

Tool	Description	Shortcut
 **AutoRecovery**	Used to toggle the AutoRecovery feature on and off. This can be helpful if AutoRecovery causes performance issues with larger model files.	
 **Back**	Navigates to the previous workflow page.	
 **Forward**	Navigates to the next workflow page in the page history. **Note:** Click the down arrow next to this tool to jump to a page anywhere in the page history. A checkmark is placed by the current page.	

Tool	Description	Shortcut
 **Save**	Saves any changes made to the current model. Saves any changes made to the current model.	**<Ctrl+S>**
 **Open**	Opens the Start page open tab, which is used to select a model to open in the program.	**<Ctrl+O>**
 **Close**	Closes the current model and returns to the Start page.	
 **Undo**	Undoes the previous operation.	**<Ctrl+Z>**
 **Redo**	Undoes the previous undo operation.	**<Ctrl+Y>**
 **Command File**	Opens the current input command file (file extension `.std`.) in the STAAD.Pro Editor. If any change has been made in the structure that has not been saved, you are prompted to save the structure first. **Note:** Refer to I. STAAD.Pro Editor (on page 2043) for additional assistance.	
 **Analysis Output**	Opens the results of a successful Analysis and Design run in the STAAD.Pro Editor window.	

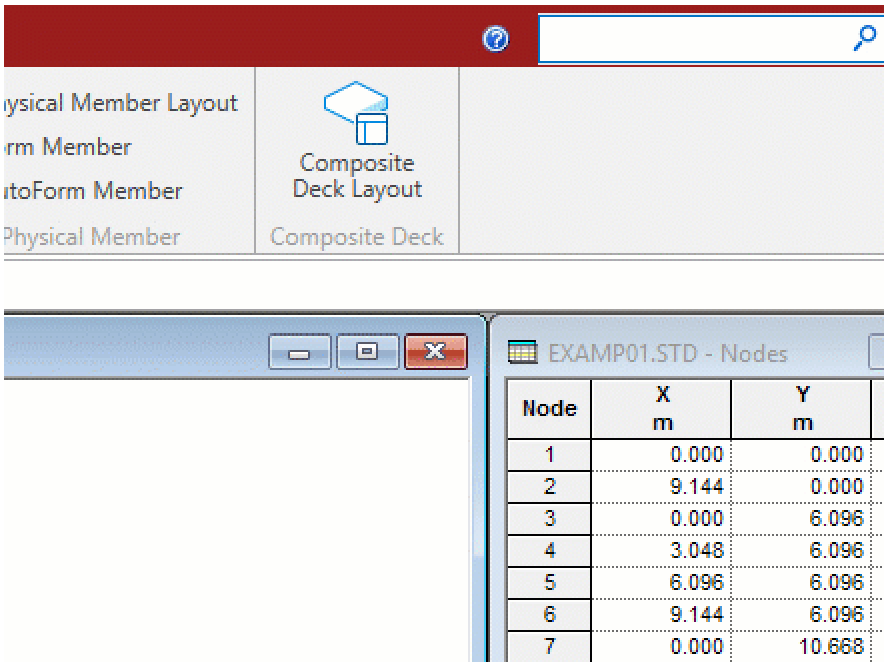
Related Links

- *I. To enable auto-recovery* (on page 2065)

GS. Tool Search

You can search for any tool by typing part of the name in the Search field.

Tip: Hover your mouse pointer over any search result to see the location of the tool in each workflow, ribbon tab, and group.



In the search results drop-down, click **Show Details** to display the tool tip description for each tool in the results list.

GS. Page Control

Within each workflow, there are a series of Pages available which layout the order of that workflow.

Though not required, it is recommended to work from left to right along the pages within any workflow.

Tip: If you close some of the dialogs for a page and need to display them again, simply click the **Restore View** tool () on the right-hand side of the page control bar.

Tip: You can use the **Navigate Backward** and **Navigate Forward** buttons found in the Quick Access Toolbar to move along your page navigation history.

Analytical Modeling

Pages in the Modeling (on page 629):

Geometry
Properties

Materials
Specifications
Supports
Loading
Analysis
Design

Building Planner

Pages in the M. Building Planner workflow (on page 920):

Slab
Column
Beam
Geometry
Shearwall
Support
Release
Load
Design

Piping

Pages in the M. Piping workflow (on page 902):

Pipe Runs
Supports

Bridge Deck

Pages in the M. Bridge Deck workflow (on page 908):

Deck

Postprocessing

Pages in the P. Postprocessing Workflow (on page 1979):

Displacements
Reactions
Beam Results
Plate Results
Solid Results
Dynamics
Reports

Foundation

Pages in the P. Postprocessing Workflow (on page 1979):

Foundation

Steel AutoDrafter

Pages in the P. Steel AutoDrafter Workflow (on page 2018):

Layout
Drawing
Material Take Off

Chinese Steel Design

Pages in the D. Chinese Steel Design (on page 978):

Parameters
Results

Connection Design

Pages in the D. Connection Design workflow (on page 1014):

Connections
Results
Seismic Frame

Advanced Slab Design

Pages in the D. Advanced Slab Design (on page 1073):

Envelopes
Slab Design

Earthquake

Pages in the P. Earthquake workflow (on page 2032):

EC8 Stiffness
EC8 Plans
EC8 Elevations

Note: The **Physical Modeling** workflow and **Concrete Design** workflow do not have pages, but instead launch separate application windows.

GS.Data Area

The dialogs and tables to enter and review data for your model open along the right-hand side of the View window.

As you select different workflows and pages in each workflow, the data area updates with the corresponding dialogs and tables for that page.

GS. View Window

The graphical view window allows you to view the model or portions of the model using selected views.

This is where the model drawings and results are displayed in graphical form. You can use tools to draw your mode model as well as graphically assign loads or specifications.

GS. Right-Click Pop-up Menu

Provides convenient access to frequently used tools and utilities. The menu pops up when the right mouse button is clicked.

Note: Some of the items in the menu are context-sensitive - that is, they vary depending on previous actions or selected elements in the window.

No Selection

Menu item		Description
Cut		Used to cut selected object(s) (delete and copy to clipboard). The deleted objects may then be pasted.
Copy		Used to copy selected object(s) to clipboard for subsequent pasting.
Paste		Opens the **Paste With Move** dialog, which is used to specify the insertion point for pasting clipboard elements into the model.
Select Cursor >	Nodes	Used to graphically select nodes.
	Beams	Used to graphically select beams.
	Plates	Used to graphically select plates.
	Surface	Used to graphically select surface elements.
	Solids	Used to graphically select solids.
	Plates & Solids	Used to graphically select plates and solids with one cursor (ignores other object types).
	Geometry	Used to graphically select nodes, members and elements of the structure simultaneously. To select nodes, members or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold **<Ctrl>** while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components.

Menu item		Description
	Physical Member	Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously. Members may be user defined or may be generated automatically using the Auto Form Member tool. To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down **<Ctrl>** while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor.
Selection Mode >	Drag Box	Click to activate the drag box selection mode.
	Drag Line	Click to activate the drag line selection mode.
	Region	Click to activate the region selection mode.
Take Picture		Used to take a snapshot image of current view. The picture is automatically added to a picture album.
Add Beam		Used to add beam by clicking start and end nodes.
Tables		Opens the **Tables** dialog, which is used to display and close different tables, such as Node coordinates, Beam incidences, Node displacements, etc. irrespective of the current page.
Connection Tags >	View Connection Tags	Opens the **Assign Connection Tags** dialog.
	Check Connection Tags	(Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.
Open View		Opens the **Open View** dialog, which is used to open a previously saved structural view window.
Labels		Opens the **Diagrams** dialog to the **Labels** tab, which is used to select various display labels for different components of the structure.
Orientation		Opens the **Orientation** dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.

Menu item	Description
Structure Diagrams	Opens the **Diagrams** dialog, which is used to customize the view of the structure by setting different view-related parameters.
Model View Details	Opens the **Structural Diagram Info** dialog, which contains counts of the objects in your model.
3D Rendering	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Beam Selection

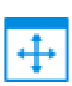
Menu item		Description
Move		Opens the Move Entities dialog, which is used to specify the translational offset for moving a selection of beams.
Insert Node		Opens the **Insert Node into Beam #** dialog, which is used to insert one or more nodes at specified distances along selected members.
Form Member		Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.
Assign Loads		
Properties		Opens the **Beam** dialog which displays properties for the selected member. Additional tabs become available after analysis and design is performed.
Define Section Profile		(disabled)
Connection Tags >	Assign	Opens the **Assign Connection Tags** dialog and **New Connection tag** dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.

Menu item		Description
	Remove	Opens the **Assign Connection Tags** dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.
New View		Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.
Create Seismic Frame		Opens the **Seismic Frames** dialog, which is used to specify the type of seismic frame used for the selected members creating a seismic frame definition during connection design.
Add Member Attribute		Opens the Member Attribute which is used to select and apply a member attribute to selected members. Refer to TR.29.1 Struclink Member Attribute (on page 2340) for additional information on the Struclink attribute which is assigned using this dialog.
List/Delete Member Attribute		Opens the Member Attribute dialog to display the currently used Member Attribute for a single selected member.

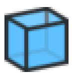
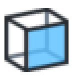
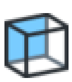
GS. Right-Click View Tools menu

Hold **<Shift>** when right-clicking in the View window to display the **View** ribbon tab **Tools** group at the mouse pointer.

Table 5: Tools group

Tool name	Description	Shortcut
 **Zoom Window**	Used to define the boundaries of a rectangular area of the active view to be displayed within the current view.	
 **Whole Structure**	Fits the current view to display the entire structure limits. Resets view rotation to orthogonal.	

Tool name	Description	Shortcut
 **Zoom In**	Zoom in the current view by a preset percentage magnification.	scroll wheel up
 **Zoom Out**	Zoom out the current view by a preset percentage magnification.	scroll wheel down
 **Pan**	Used to view a different part of the design without changing the view magnification. **Tip:** Press and hold the middle mouse button (typically the scroll wheel button) to quickly pan in the view window.	
 **Zoom Extents**	Adjusts the view magnification so that the entire model is visible in the view.	
 **Zoom Factor**	Opens the **Zoom Factor** dialog, which is used to zoom in or out of the structure by specifying a magnification factor. Factors less than one Zoom out.	
 **Zoom Previous**	Undoes the last **Zoom Factor** or **Zoom Window** operation.	
 **Dynamic Zoom**	Used to define the boundaries of a rectangular area of the active view to be displayed within a new view. The previous view window highlights the location of the zoomed portion.	

Tool name	Description	Shortcut
 **Magnifying Glass**	Provides a magnified portion of the current view window when you click and drag the pointer.	
 **Isometric View**	(Default view) View the model as an isometric projection.	
 **Front View**	View the structure model from the positive Z axis.	
 **Left View**	View the structure model from the negative X axis.	
 **Top View**	View the foundation mode in plan; from the positive Y axis.	
 **Back View**	View the structure model from the negative Z axis.	
 **Right View**	View the structure model from the positive X axis.	

Tool name	Description	Shortcut
 **Bottom View**	View the structure model from the negative Y axis.	
 **Rotate Up**	Rotate the structure model forward about the X axis.	**<↑>** (Up arrow key)
 **Rotate Left**	Rotate the structure model forward about the Y axis.	**<←>** (Left arrow key)
 **Spin Left**	Rotate the structure model forward about the Z axis.	**<Ctrl+←>** (Ctrl +Left arrow keys)
 **Rotate Down**	Rotate the foundation mode backward about the X axis.	**<↓>** (Down arrow key)
 **Rotate Right**	Rotate the structure model backward about the Y axis.	**<→>** (Right arrow key)
 **Spin Right**	Rotate the structure model backward about the Z axis.	**<Ctrl+→>** (Ctrl +Right arrow keys)

Tool name	Description	Shortcut
 **Toggle View Rotation Mode**	Used to select an node as the center of rotation. When toggled on, pressing **<Ctrl+Shift>** and clicking a node will set that node as the center of rotation.	
 **Orientation**	Opens the **Orientation** dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.	**<F4>**
 **Always Fit in Current Window**	Instructs the program on what guidelines to use when drawing a selected set of objects on the screen. It displays the selected portion of the model to a size governed by optimum usage of the dimensions of the current window. When you select various tools the program or switch workflows the size of the drawing window frequently changes. With this tool turned on, the model or selected portions of it, will be drawn in such a manner that all the entities will be drawn within the bounds of the drawing area. This means, the size to which the entities are drawn will correspondingly increase or decrease. With this tool turned off, the size of the entities will remain constant, but that means it may or may not fit within the bounds of the drawing window.	

GS. Quick Commands Pop-up menu

This menu Opens when you press the space bar when the View window has the application focus.

You can customize the Quick Commands with any tool by clicking the icon and then adding tools from the **Customize Quick Commands Popup** dialog.

Table 6: Selection group

Tool name		Description
 **Geometry >**	**Geometry Cursor**	Used to graphically select nodes, members and elements of the structure simultaneously. To select nodes, members or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold **<Ctrl>** while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components.
	All Geometry	Selects all nodes, members, elements, and solids in the model.
	Inverse Geometry Selection	Used to select all objects but the ones which are currently selected.
	Geometry List	Opens the **Select Geometry** dialog, which is used to select one or more model entities (other than nodes) from a list of all entities in the model.
	Entities Parallel To > **XY** | **YZ** | **XZ**	Used to select all beams, elements, and surfaces which are parallel to a specified global axis. Select the desired global axis from the sub-menu.
	Geometry Connecting To > **Node** | **Beam** | **Plate** | **Solid**	Used to find out all the entities that are connected to (i.e., have a common node with) any particular node, beam, plate, or solid. A dialog opens prompting you to select the node, beam, plate, or solid number. You can select to apply immediately or click OK to see all the model entities that are connected to the selection.
	Highlight Entities Sequentially	Opens the **Visual Check** dialog, which is used to sequentially highlight a specific group of entities (beams, plates, or solids) in numerical order. Controls on the speed of the selection are also included.
 **Solids >**	**Solids Cursor**	Used to graphically select solids.
	All Solids	Selects all solids in the model.
	Inverse Solids Selection	All solids in the selection set are deselected and any previously unselected solids are added to the selection set.
	Solids List	Opens the **Select Solids** dialog, which is used to select one or more solids from a list of all solids in the model.

Tool name		Description
	Solids Connecting To > **Node | Beam | Plate | Solid**	Select solids that connect to the selected object type.
 **Members**		Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously. Members may be user defined or may be generated automatically using the Auto Form Member tool. To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down **<Ctrl>** while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor.
 **Nodes >**	**Nodes Cursor**	Used to graphically select nodes.
	All Nodes	Selects all nodes in the model.
	All Supports	Selects all supported nodes in the model.
	Inverse Node Selection	All nodes in the selection set are deselected and any previously unselected nodes are added to the selection set.
	Nodes List	Opens the **Select Nodes** dialog, which is used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.
 **Beams >**	**Beams Cursor**	Used to graphically select beams.
	All Beams	Used to select all beams.
	Inverse Beam Selection	All beams in the selection set are deselected and any previously unselected beams are added to the selection set.
	Beam List	Opens the **Select Beams** dialog, which is used to select one or more beams from a list of all beams in the model.
	Beams Parallel To > **X | Y | Z**	Select beams that are parallel to the selected global axis.

Tool name		Description
	Beams Connecting To > **Node** | **Beam** | **Plate** | **Solid**	Select beams that connect to the selected object type.
 **Plates >**	**Plates Cursor**	Used to graphically select plates.
	All Plates	Selects all plates in the model.
	Inverse Plate Selection	All plates in the selection set are deselected and any previously unselected plates are added to the selection set.
	Plate List	Opens the **Select Plates** dialog, which is used to select one or more plates from a list of all plates in the model.
	Plates Parallel To > **XY** | **YZ** | **XZ**	Select plates that are parallel to the selected global axes plane.
	Plates Connecting To > **Node** | **Beam** | **Plate** | **Solid**	Select plates that connect to the selected object type.
 **Text Cursor**		Used to add comments and titles to pictures and result diagrams. The added text can be plotted, too. The inserted text can be deleted, moved and modified using the text cursor. Refer to the **Insert Text** on the **Utilities** ribbon for a detailed description on inserting text and modifying it using the text cursor.
 **Filtered Selection Cursor**		Used to select multiple types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in one pass. This will reduce the time required to create new views and help quickly identify the location of certain entities on your structure. **Note:** Before this cursor can be used, the actual filter parameters must be defined in advance in the **Selection Filters** dialog.
 **Group Selection**		Opens the **Select Groups** dialog, which is used to select objects in a named group.

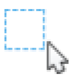
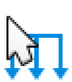
Tool name	Description
 **By Property Name**	Used to select nodes and members based on specifications associated with them. A number of specification types are included in the sub-menu list.
 **Missing Properties**	Used to select beams, plates, or solids that lack critical input data (Property, Density, Elasticity, Poisson's ratio, and Alpha) in the active view.
 **Drag Box**	Click to activate the drag box selection mode.
 **Drag Line**	Click to activate the drag line selection mode.
 **Region**	Click to activate the region selection mode.
 **Previous Selection Cursor**	Selects the last object(s) selected (if they have been deselected).
 **Loads Cursor**	Used to modify any load already applied on the model by double clicking it.. When selected, the mouse pointer changes to the Load Edit Cursor.

Table 7: Labels group

Tool name	Description	Shortcut
Labels Settings	Opens the **Diagrams** dialog to the **Labels** tab, which is used to customize the view of the structure by setting different view-related parameters.	
Node Labels	Select any of the label types from the drop-down list to turn the display of that label on or off. **Tip:** Additional labels are controlled on the **Diagrams** dialog **Labels** tab.	**<Shift+N>**
Beam Labels		**<Shift+B>**
Plate Labels		**<Shift+P>**
Solid Labels		**<Shift+C>**
Individual Node Labels	When the object label type is activated, use these tools to show or hide the individual labels of objects. This can be useful for large models or views where you wish to display multiple types of labels simultaneously. **Note:** These tools require you to select the **Always Use Current Label Settings** option and then check the **Use Partial Labeling Mode** option on the **Diagrams** dialog **Labels** tab.	
Individual Beam Labels		

Tool name	Description	Shortcut
Individual Plate Labels		
Individual Solid Labels		

Table 8: Display group

Tool name	Description
Load	Select the active load case, load combination, or load envelope from the pop-up dialog.
View Loading Diagram	Click to toggle the display of the current load case on the structure.

Table 9: Geometry Tools group

Tool Name		Description
Structure Tools >	**Multiple Structures**	Opens the **List of Structures** dialog, which is used to determine if the current model consists of more than one unconnected structure. Select a Structure in the list to see each structure highlighted.
	Beam Plate Connectivity	Used to check for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.

Tool Name		Description
	Merge Properties	Used to merge the properties of two or more similar objects. When a STAAD input file has a number of references of the same property, this tool can be used to consolidate all these properties into a single command. When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command. Clicking the Yes button, all instances of a given section property will be collated into a single property reference. **Note:** Properties references with differing additional parameters will not be collated. Properties references with differing assigned material properties will not be collated.
	Cut Section	Opens the **Section** dialog, which is used to cut a section through the structure along a specified global plane at a desired location of the 3rd axis.
 **Node Tools >**	**Duplicate Nodes**	Opens the **Remove Duplicate Nodes** dialog, which is used to set the tolerance distance between two nodes the program should consider as duplicate. Used for detecting the presence of two or more instances of the same node.
	Orphan Nodes	Highlights all nodes in the structure which are not connected to any member, element, or solid.
	Remove Orphan Nodes	Used to remove all detected orphan nodes.
	Node to Node Distance	Display the distance between nodes.
	Remove Node to Node Distance	Used to remove the display of all node to node dimensions from the current view.
 **Beam Tools >**	**Duplicate Beams**	Opens the **Remove Duplicate Beams** dialog, which is used to set the tolerance distance between two beams the program should consider as duplicate. Used for detecting the presence of two or more instances of the same beam.

Tool Name		Description
	Zero Length	Opens the **Zero Length Tolerance** dialog, which is used to set the tolerance for zero length members. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero. This tool detects such members.
	Overlapping Collinear Members	When two members are collinear, and further, at least one of the nodes of one of those members happens to lie within the span of the other, but the two members are not connected at that node, those two members are considered as overlapping collinear members. This tool detects such members. The usefulness of this tool comes from the fact that it enables you to detect modeling errors which are not easily visible. Two lines overlapping on the drawing area become indistinguishable on large models, and this is one of the tools that can spot such errors.
	Beam Incidence	Used to reverse the incidence of selected beams so that the global coordinates of the end node are farther from the origin than that of the start node.
	Dimension Beams	Display the dimension of the members in the structure.
 **Plate Tools >**	**Duplicate Plates**	Opens the **Remove Duplicate Plates** dialog, which is used to set the tolerance distance between two plates the program should consider as duplicate. Used for detecting the presence of two or more instances of the same plate.
	Warped Plates	A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.
	Plate Connectivity	Checks the model for plates that overlap or intersect each other at their boundaries and, if found, opens the Overlapping Collinear Plates dialog. Typically, these overlaps cause improper load transfers or instabilities in the model.
 **Solid Tools >**	**Negative Volume**	Used to verify that the solid elements in their model have the proper sequence (order) of node numbering to prevent warnings in the output file of solids containing negative volumes.
	Warped Solids	Used to check if a solid element is warped. Warped solids cannot be analyzed and will produce errors in the output file.

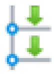
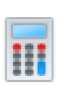
Tool Name	Description
 **Physical Member Restraints**	Used to automatically generate top and bottom flange restraint conditions for the selected physical members. **Note:** The PBRACE command generated by this menu item is valid only for D2.B.12 Physical Member Design (on page 1372) per AS 4100-1998 (Australian) steel design. When selected, the program will search through all the nodes along the Physical Member and determine if any other beam is connected besides the beams in the current physical member definition. If another beam is present, a full restraint is placed on both the flanges. Otherwise, an unrestrained condition is imposed on both the flanges.
 **Groups**	Opens the **Create Group** dialog, which is used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name. **Note:** If no groups exist, you will be prompted to create a group using the **Define Group Name** dialog.

Table 10: Tools group

Tool Name		Description
 **Connection Tags >**	**Load Connection Tag File**	
	Edit Connection Tag File	
	View Connection Tags	Opens the **Assign Connection Tags** dialog.
	Check Connection Tags	(Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.
 **Calculator**		Opens the **STAAD.Pro Calculator** window, which is capable of performing mathematical operations.

Tool Name	Description
 **Unit Converter**	Opens the **STAAD.Pro Converter** window, which is used to convert data from one unit system to another.
 **Take Picture**	Used to take a snapshot image of current view. The picture is automatically added to a picture album. **Tip:** The picture is stored in a STAAD-native format and can be subsequently included in custom reports during Report Setup. The Copy Picture and Export View tools may be used to copy a model image or to save it to a common file format.
Copy Picture	Used to copy the current picture to the clipboard for pasting in other Windows applications such as image editors, spreadsheet applications, or word processors,.
 **AVI File**	Opens the **Create AVI File** dialog, which is used to create a video file recording of for animated deflection, section displacement, mode shape, and plate stress contour diagrams.

GS. Status Bar

The program window status bar displays the current workflow along with tips on next actions based on the current tools. The active load case and current input units are also displayed.

GS. Keyboard Shortcuts

Shortcut keys for menu commands are indicated in the menus and/or related dialog boxes.

Note: Most ribbon tabs are accessible by pressing the **<Alt>** key and then the letter key corresponding in the tooltip over that ribbon tab. STAAD.Pro also supports shortcut keys for window and dialog box navigation and other functions.

Table 11: Working with files

Action	Shortcut
Create a new STAAD.Pro project file.	**<Ctrl+N>**
Open an existing STAAD.Pro project file.	**<Ctrl+O>**

Action	Shortcut
Save the current project file.	**<Ctrl+S>**
Print the output report.	**<Ctrl+P>**
Quit the program.	**<Alt+F4>**

Table 12: Editing models

Action	Shortcut
Undo an action.	**<Ctrl+Z>**
Redo an action	**<Ctrl+Y>**
Cancel an action.	**<Esc>**
Copy a selected element.	**<Ctrl+C>**
Paste an element from the clipboard.	**<Ctrl+V>**
Cut an element (copies the selected element to the clipboard and deletes the selected copy).	**<Ctrl+X>**
Delete a selected element.	**<Del>**
Move selected objects.	**<F2>**

Table 13: Navigating the view window

Action	Shortcut
Rotate model up or down in the view window.	Up or Down arrow keys (<↑>, <↓>)
Rotate model left or right in the view window.	Left or Right arrow keys (<←>, <→>)
Spin mode left or right in the view window.	**<Ctrl+Left>** or **<Ctrl+Right>** arrow keys
Close the current active view; except when the active view is the main structure view.	**<Ctrl+F4>**
Open the **Orientation** dialog, which is used to precisely control the zoom and rotation of the active view window.	**<F4>**
Refresh the active view window.	**<F5>**
Tile all view windows, tables, forms, etc. horizontally.	**<Shift+F4>**
Cascade all view windows, tables, forms, etc.	**<Shift+F5>**

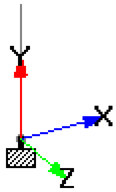
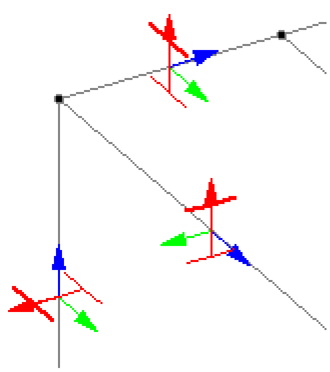
Action	Shortcut
Tile all view windows, tables, forms, etc. vertically.	**<Ctrl+Shift+F4>**
Animation in full screen (Postprocessing mode).	**<F12>**
Restore default window layout for the current page.	**<Ctrl+Tab>**

Table 14: Working with models

Action	Shortcut
Create a new group from selected entities.	**<Ctrl+G>**
Initiate the analysis and, if used, batch design for the current input file.	**<Ctrl+F5>**
Open the relevant help topic.	**<F1>**

Table 15: Toggling the display of model labels

Action	Shortcut
Display member specifications (i.e., truss, beta angle, etc.).	**<Shift+A>**
Show axes window.	**<Ctrl+Shift+A>**
Display beam numbers.	**<Shift+B>**
Display the design brief for physical members.	**<Ctrl+Shift+B>**
Display solid element numbers.	**<Shift+C>**
Display dimensions.	**<Shift+D>**
Display the beam ends, color coded for start and end.	**<Shift+E>**
Display the design envelope for physical members.	**<Ctrl+Shift+E>**
Display floor loading.	**<Shift+F>**
Show the diagram information.	**<Shift+G>**
Display the design group for physical members.	**<Ctrl+Shift+G>**
Display the wind load tributary area.	**<Shift+H>**

Action	Shortcut
Show axes at origin. Axes are color-coded: X is blue, Y is red, and Z is green. 	**<Shift+I>**
Display the entity reference numbers.	**<Shift+J>**
Display node point labels.	**<Shift+K>**
Display the control-dependent links.	**<Shift+L>**
Display the material label for each entity.	**<Shift+M>**
Display physical member numbers.	**<Ctrl+Shift+M>**
Display node numbers.	**<Shift+N>**
Display the local beam axis (beam orientation). Local axes are color-coded: x is blue, y is red, and z is green. 	**<Shift+O>**
Display plate element numbers.	**<Shift+P>**
Display surface element numbers.	**<Ctrl+Shift+P>**
Display member/element releases.	**<Shift+R>**
Display the support node labels.	**<Shift+S>**
Display the local plate element axis (plate orientation).	**<Shift+T>**
Display the local surface element axis (surface orientation).	**<Ctrl+Shift+T>**

Action	Shortcut
Display load values.	**<Shift+V>**
Display wind loads.	**<Shift+W>**
Display member sections.	**<Shift+X>**
Display the floor load distribution.	**<Shift+Y>**

Table 16: Model rendering

Action	Shortcut
View the analytical model (no outline)	**<Ctrl+0>**
View the model in wireframe (outline of members)	**<Ctrl+1>**
View the model filled (members drawn solid)	**<Ctrl+2>**
View the rendered model window	**<Ctrl+4>**

GS. Services and Support Information

These resources are provided to help you get answers for your technical questions on STAAD.Pro.

Service Request Manager

http://apps.bentley.com/srmanager/ProductSupport — Create and track a service request using Bentley Systems' online site for reporting problems or suggesting new features.

Communities Support

https://communities.bentley.com/support — Find solutions to issues, get answers to questions, or get priority assistance for critical issues.

FAQ and TechNotes

https://communities.bentley.com/products/structural/structural_analysis___design/w/structural_analysis_and_design__wiki/4057.staad-pro-support-solutions

Forums

http://communities.bentley.com/products/structural/structural_analysis___design/f/5932 — Post questions in the Bentley Communities forums to receive help and advice from fellow users and members of Bentley's product support groups.

GS.Nuclear Safety Related Features

An installation of STAAD.Pro provides users with an extensive set of features and capabilities. It is recognized that STAAD.Pro can be used on projects that require the highest level of safety such as nuclear power stations. Therefore, a set of key features as listed below have been defined as "Nuclear Safety Related Features." For any user who has signed up to the QA&R program, any serious defect discovered in any of these features in a build of STAAD.Pro supplied as part of the QA&R that is determined to be of Critical or High Severity, will be notified as per the reporting requirements.

The following input command driven features of STAAD Analysis and Design engine under default axis system (Y-up) are consider as "Nuclear Safety Related Features" of the software product.

1. Static Analysis
   a. first order elastic analysis
   b. P-Delta analysis
2. Dynamic Analysis
3. Features related to application of general loadings applied at/on nodes, members, plates and solids.
4. Features related to application of local boundary conditions (e.g., member releases, offsets etc.)
5. Features related to application of global boundary conditions (e.g., fixed, pinned, fixed but, spring supports)
6. Automated load generation features as per US codes (e.g., ASCE, IBC, UBC) such as:
   a. Wind load generation per ASCE 7
   b. Seismic load generation
   c. Response spectrum load generation
   d. Moving load generation

Design Codes:

1. The following STAAD command file driven Steel design are available:
   a. AISC 360-2016
   b. AISC 360-2010
   c. AISC 360-2005
   d. AISC ASD 9th Edition (1989)
   e. AISC LRFD 2nd Edition (1994), 3rd Edition (2001)
   f. AISC N690 1984 and 1994 with Supplements S1 and S2
   g. ASME NF3000 1974, 1977, 1989, 1998, and 2004
   h. CAN/CSA S16-14
   i. CAN/CSA S16-09
2. The following STAAD command file driven Concrete design are available:
   a. ACI 318 1999, 2002, 2005, 2008, 2011

The "Nuclear Safety Related Features" are limited to the STAAD Analysis & Design engine only. Note that any other features, such as the graphical user interface and interfaces to link to external applications such as AutoPipe, RAM Connection, RAM Concept etc. are *not* considered as "Nuclear Safety Related Features" of the product/program.

2 What's New?

The Software Release Report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 06. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro CONNECT Edition V22

CONNECT Edition V22 Update 10

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 10 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 9 (release 22.09). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.10.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.10.00-1.1 Auto-Recovery Enhancements

The auto-recovery process in STAAD.Pro has been further enhanced to protect your files from unexpected application failures.

These enhancements include the following:

STAAD Backup File

During opening and saving of the STAAD file, a file with model name and extension `.SBK` (i.e., `*<modelname>*.sbk`) will be created in the model folder. This file is a duplicate the STAAD input file.

To recover the STAAD input file, simply rename the file replacing the `.SBK` extension with `.STD`.

> **Note:** The `.SBK` file is not automatically modified or updated with the given time interval in auto-recovery process.

Auto-Recovery Backups Directory

In addition to the creation of auto-recovery file (i.e. *<modelname>_autorecovery.zip*) in the model folder, a copy of the same will be preserved under:

```
%localappdata%\Bentley\Engineering\STAAD.Pro CONNECT Edition\AutoRecovery Backups
\<modelname>-<unique id>
```

Since the auto-recovery file gets removed when saving or closing the model, this ensures the preservation of last changes made in the event of an unexpected application failure.

In case of any failures in which the STAAD input file and its ancillary files gets corrupted, you can retrieve the auto-recovery backups from the above-mentioned location.

Note: To ensure uniqueness of the folder name, the unique id suffixed is derived from the model path. Hence, two files with same name existing in two different folders, will have the different unique ids and thus prevented from overwritten.

Refactored Auto-Recovery Process

Also, the process has been restructured such that an auto recovery zip file is created when a model is opened (this is by design for unexpected crash), and the auto recovery process is paused until the next modification of the analytical model.

Note: For the physical model (e.g., when STAAD.Pro Physical Modeler is open), the auto-recovery process does not stop. Rather, the auto-recovery process is triggered at a given time interval.

RR 22.10.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.10.00-2.1 Individual Member Design using RAM SBeam

You can now perform member design on single beam members using RAM SBeam within STAAD.Pro Physical Modeler.

Steel beams can be designed as composite or non-composite members.

Note: You must have a separate license for RAM SBeam to use this feature. RAM SBeam must be installed on the computer along with STAAD.Pro.

RAM SBeam is used for designing single-span, steel beams (with or without cantilever on each end). The beams can be designed with or without composite decking. The following requirements must be applicable to the members for exporting:

- only wide flange, channel, and rectangular hollow tubes sections may be used
- multiple members must be connected and colinear

Refer to the STAAD.Pro Physical Modeler help for additional details on exporting members to RAM SBeam for design.

RR 22.10.00-2.2 Variable Surface Thickness

STAAD.Pro Physical Modeler is now capable of assigning thicknesses to physical surfaces.

Refer to the STAAD.Pro Physical Modeler help for additional details on specifying thickness to surfaces.

RR 22.10.00-2.3 Modular Tank with Tapered Walls

A new template has been added to the Structure Wizard in STAAD.Pro Physical Modeler for modelling tanks with tapered walls.

Note: This feature requires Structural SELECT Entitlements, which are provided with an active SELECT or ELS subscription.

This tank model includes discrete wall and slab sections for use with concrete design in the RCDC application. This model can have tapered wall thickness. The lateral load due to the sloped walls are calculated as both vertical and horizontal components.

Limitations:

- this tank can only have a single level
- this tank cannot contain interior columns or beams

Refer to the STAAD.Pro Physical Modeler help for additional details on creating a tank model with tapered walls.

RR 22.10.00-2.4 Automatic Load Combination Enhancements per 2021 Chinese Code

The load combinations per the Chinese GB 55002-2021 code have been added to the automatic load combination generator.

The ultimate load combination (ULC) and service load combination (SLC) categories are labeled as a prefix in the **Select Load Combination Category** drop-down list on the **Auto Load Combination** dialog.

Related Links

- *Auto Load Combination dialog* (on page 2872)

RR 22.10.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.10.00-3.1 Tapered Member Design per EC3

Design of tapered members per EN 1993-1-1:2005 is now available in STAAD.Pro.

STAAD.Pro can design the following section profiles per EN 1993-1-1:2005:

- tapered I-section (constant flanges, web tapered)
- tapered tube (tapered square hollow section)
- tapered pipe (tapered round hollow section)

A new D5.C.6 Design Parameters (on page 1502), HGT, has been added for use with the design of tapered I-sections. Other design parameters directly related to the tapered member checks are LY and EFT.

The following clauses of EN 1993-1-1:2005 are checked for tapered sections:

- 6.2.3 - tension
- 6.2.4 - compression
- 6.2.6 - shear
- 6.2.5 - moment
- 6.2.7 - torsion:

  (1) - pure torsion
  (5) - yield criteria for interaction of axial, bending and shear (including shear from torsion)
  (9) - reduced shear due to torsion
- 6.2.8 - reduced moment due to shear
- 6.2.9.1 - combined moment and axial force
- BB.3.2 - Stable length for tapered members (applies to tapered I-sections) - see Tapered Members (on page 1484)

Related Links
- *D5.C.6 Design Parameters* (on page 1502)

RR 22.10.00-3.2 Single Angle Brace Members in AISC 360-16

The engineer now has control over the use of how the provisions of Section E5 are applied to the design of single angle members in compression per AISC 360-16.

Previously, it was assumed that all of the conditions for single angle members were met such that the effects of eccentricity could be ignored and that the effective slenderness ratio may be used. Further, it was assumed that any single angle was an individual member and that the modified slenderness ratio of Cl. E5(a) always applied. These are conservative assumptions and may result in sections that are not the most economical. Therefore, two new parameters have been added which allow the engineer to control the specifics of how single angle members in compression are designed per E5.

For AISC 360-16, single angle members in compression can be designed using a modified slenderness method provided a set of conditions are met for that member. It is left to the engineer to confirm that conditions E5.(1) through E5.(3) are met and to specify this for single angle member using the `E5P` parameter. Those conditions are:

1) Members are loaded at the ends in compression through the same leg.
2) Members are attached by welding or by connections with a minimum of two bolts.
3) There are no intermediate transverse loads.

If one or more of these conditions are not met, then the `E5P 1` parameter value should be assigned to those members.

Note: Conditions E5.(4) and E5.(5) are checked by the program:

4) Slenderness ratio, $L_c / r \le 200$
5) For unequal leg angles, the ratio of leg widths is < 1.7.

The effective slenderness ratio method used for single angles also depends on if those angle members are:

1. individual members or are web members of planar trusses (in which case, Cl. E5(a) applies - `IMM 0`)
2. web members of box or space trusses (in which case, Cl. E5(b) applies - `IMM 1`)

A message is issued if the `E5P` or `IMM` parameter is used with any section type other than a single angle.

Note: The E5P and IMM parameters are only applicable to standard catalog or UPT angles which also are assigned as truss members.

Related Links

- *D1.A.6 Design Parameters* (on page 1100)

RR 22.10.00-3.3 Chinese Response Spectrum Enhancements

Reponse spectra per the GB50011 2010 code may now consider torsional effects of the load.

Inherent and Accidental Torsion

Note: STAAD.Pro does not support the coupled torsion methodology as per GB50011-2010. This implementation enables the general "inherent and accidental torsion" for GB50011 response spectrum analysis. Note that this implementation does *not* comply with Cl. 5.2.3 of GB50011-2010.

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either CQC or SRSS) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to the positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. For example, consider two joints J2 and J3 whose maximum joint displacement in the global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X-direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, the torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from the response spectrum along with torsion for all modes are combined using a specified modal combination method to get the final maximum possible joint displacements.

Related Links

- *TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010* (on page 2531)
- *Response Spectra tab* (on page 2854)
- *Generated Spectrum dialog* (on page 2862)
- *Spectrum Parameters dialog* (on page 2862)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)

RR 22.10.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.10.00-4.1 Design of General Sections in Chinese Steel Design Workflow

The Chinese Steel Design Workflow now supports the design of combined (i.e., built-up) sections which have been specified as "general" prismatic sections in analysis.

Related Links

- *General Section dialog* (on page 3014)

CONNECT Edition V22 Update 9

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 9 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 8 (release 22.08). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.09.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.09.00-1.1 Mass Model Generation

STAAD.Pro can now automatically generate mass model loads based on a selection of load cases for inclusion as a new reference load case or in a existing dynamic load case.

Both primary load cases and reference loads can be used for generating the mass model. A load factor is applied to masses which act in the X, Y, and Z directions.

Related Links

- *Create Mass Model dialog* (on page 2870)
- *M. To generate a mass model* (on page 899)

RR 22.09.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.09.00-2.1 Cable and Truss Specifications

STAAD.Pro Physical Modeler is now capable of assigning cable and truss specifications to physical members.

The cable specification details can also be assigned directly to the physical members.

Now, when any axial behavior (tension-only, compresison-only, cable, or truss) is assigned, the program will default to not segmenting those members when the analytical model is generated.

Refer to the STAAD.Pro Physical Modeler help for additional details on specifying axial behavior properties of members.

RR 22.09.00-2.2 Notional Load Enhancements

You may now directly specify the notional load factor and directions for notional loads in automatically generated load combinations.

There is also now an option to specify the notional load factor to be calculated based on number of floors as per the Chinese GB 50017-2017 code.

Related Links

- *Auto Load Combination dialog* (on page 2872)
- *M. To automatically generate load combinations* (on page 888)

RR 22.09.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.09.00-3.1 Tension Slenderness Checks in AISC 360-16

The slenderness checks for members in tension have been refactored to allow greater control over the limits in STAAD.Pro.

Two new parameters have been added for AISC 360-16 to control the checks made for slenderness:

- `SRT` specifies the method used for checking slenderness. The default is to now use the conventional method from in AISC 360-10 (using the length given by the `TSL` parameter). An option is also available to check per clause D1 of the specification
- `TSL` is available to specify the length considered for tension slenderness checks. This defaults to the analytical member length.

Related Links

- *D1.A.6 Design Parameters* (on page 1100)

RR 22.09.00-3.2 Projected Loading on Plates

Full pressure loads on plate elements can now be applied along global axes on the projected plate area.

The resulting pressure, $p' = p\frac{A_{xz}}{A_{xyz}}$

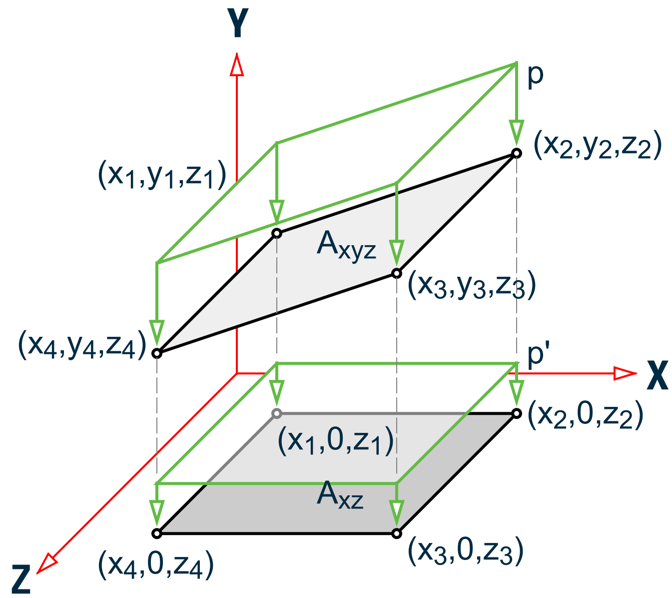


Figure 5: Projected pressure, p, in the PY direction

Related Links

- *Plate Loads tab* (on page 2847)
- *M. To add pressure load on a plate* (on page 837)

RR 22.09.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.09.00-4.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.6 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.6 (CONNECT Edition V13.6).
- New truss-gusset-branch connection for AS4100, NZS3404 and AISC360.
- New tubular connections for mitred knee and beam and column splices joints for AISC360, CSAS16 and EN1993.
- And other enhancements and additions.

See the full list of RAM Connection v13.6 Release Notes at Bentley Communities.

RR 22.09.00-4.2 P-Delta Analysis Support for Chinese Steel Design Workflow

Design of members using a P-Delta analysis is now supported in the Chinese Steel Design Workflow.
Related Links

- *D. To add a new solution set* (on page 979)

CONNECT Edition V22 Update 8

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 8 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 7 (release 22.07). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.08.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.08.00-1.1 ProjectWise 365 Compatibility

STAAD.Pro is compatible with Bentley's ProjectWise 365 project collaboration product.

RR 22.08.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.08.00-2.1 Indian Wind Load Calculator per IS-875 (Part 3): 2015

The program can now calculate wind load vs height intensity values per the Indian IS-875 (Part 3): 2015 code.
Related Links
- *IS-875 (Part 3): Wind Load dialog* (on page 2890)
- *Wind Load Generation - IS 875 (Part 3): 2015 dialog* (on page 2867)
- *M. To add an IS-875 (Part 3): 2015 wind load* (on page 850)

RR 22.08.00-2.2 Chinese Wind Load Calculator Enhancements

Several enhancements have been made to the wind load definition calculator used for the Chinese GB 50009 2012 code.

The program can now accommodate circular building cross sections (i.e., chimneys).

The method for identifying the structure faces has also been improved when defining the shape factors.

Related Links
- *Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog* (on page 2887)
- *M. To add a GB50009 wind load definition* (on page 848)

RR 22.08.00-2.3 Modular Tank with Stepped Walls

The Structure Wizard tank module in STAAD.Pro Physical Modeler is now capable of modeling stepped walls.

Note: This feature requires Structural SELECT Entitlements, which are provided with an active SELECT or ELS subscription.

Refer to the STAAD.Pro Physical Modeler help for additional details on creating a tank model with stepped walls.

RR 22.08.00-2.4 Group Management in SPPM

When creating groups in STAAD.Pro Physical Modeler from a selection, you now have the option to select which object types are included in the group.

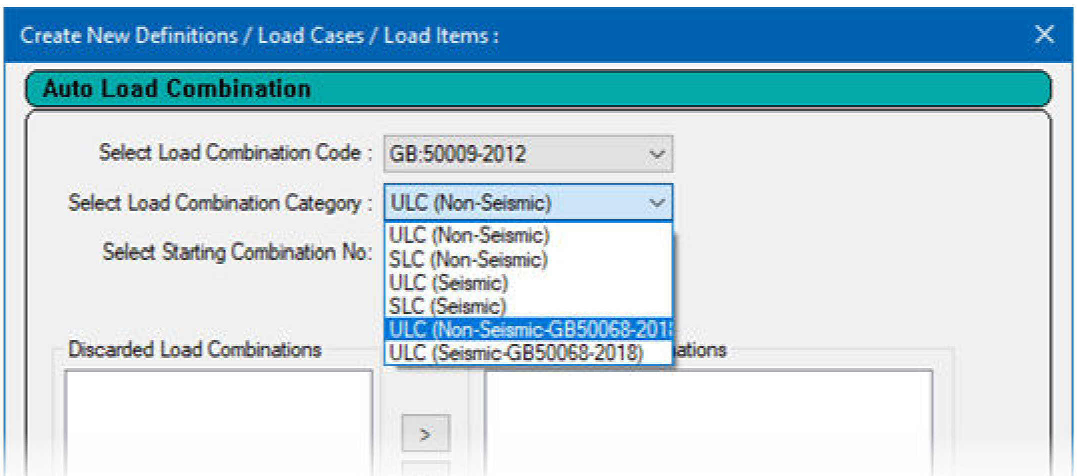
Groups created in STAAD.Pro Physical Modeler are now transferred to the STAAD input file.

Refer to the STAAD.Pro Physical Modeler help for additional details on creating groups.

RR 22.08.00-2.5 Chinese Automatic Load Combination Enhancements

The load combinations per the Chinese GB 50009-2012 code have been separated into ultimate load combination (ULC) and service load combination (SLC) categories.

These categories are labeled as a prefix in the **Select Load Combination Category** drop-down list on the **Auto Load Combination** dialog.



Related Links

- *Auto Load Combination dialog* (on page 2872)

RR 22.08.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.08.00-3.1 Steel Design per CSA S16 2019

Design of hot-rolled steel members per the Canadian CSA S16 2019 code are now available in STAAD.Pro.

Member sizes can be checked against the requirements of the design code as well as be selected for the current loading conditions.

RR 22.08.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.08.00-4.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.5 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.5 (CONNECT Edition V13.5).
- IS800 vertical bracing connections for column - beam - brace joints.
- Tubular end plates and mitred knee connections for AS and NZS codes.

RR 22.08.00-4.2 Plate Stress Envelopes

You can now view an envelope of plate stresses for either plate center stresses or plate corner stresses in the postprocessing workflow.

The Plate Center Stress and Plate Corner Stress tables now include additional tabs for displaying the envelopes.

You can now also select an envelope of all loads or a previously defined envelope from the Load Type drop-down in the Diagrams dialog Plate Stress Contour tab (on page 2758).

You can also include plate stress envelope tables in reports.

Related Links

- *P. Plate Corner Stress table* (on page 2009)
- *P. Plate Center Stress table* (on page 2009)
- *Diagrams dialog* (on page 2751)

RR 22.08.00-4.3 Custom Materials in Chinese Steel Design

You can now add custom steel material definitions for use in the Chinese Steel Design workflow.

A new **Materials** tool is used to open the dialog where user-defined materials can be added. The dialog also included the default material values (taken from Table 4.4.1 of GB50017-2017).

Related Links

- *Material Parameter dialog* (on page 3013)
- *D. To add a custom material definition* (on page 982)

RR 22.08.00-4.4 Multiple Parameters in Chinese Steel Design

You can now create “solution sets” with in the Chinese Steel Design workflow to quickly compare different sets of parameters within the same model.

Related Links

- *Configure Solution dialog* (on page 3019)
- *D. To add a new solution set* (on page 979)

CONNECT Edition V22 Update 7

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 7 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 6 (release 22.06). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.07.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.07.00-1.1 ASCE 7-2016 Load Combination Generation

STAAD.Pro can now automatically generate load combinations per the ASCE 7-16 specification.

Seismic Load Type Directions

To aid in the load combinations per ASCE 7-16, load cases marked with the “seismic” load type are now divided into “horizontal” and “vertical” directions.

This is used to accommodate load combinations which separate between vertical and horizontal seismic directions. With “Y up” models, the horizontal seismic loads act in the global X and Z directions. When the “Z up” option is used, then the horizontal seismic loads act in the global X and Y directions.

Note: Any existing STAAD file opened which contains load cases labeled as “Seismic” will be automatically converted to the “Seismic - Horizontal” type.

Related Links

- *TR.32 Loading Specifications* (on page 2461)
- *Create Primary Load Case dialog* (on page 2835)
- *M. To create a new primary load case* (on page 830)

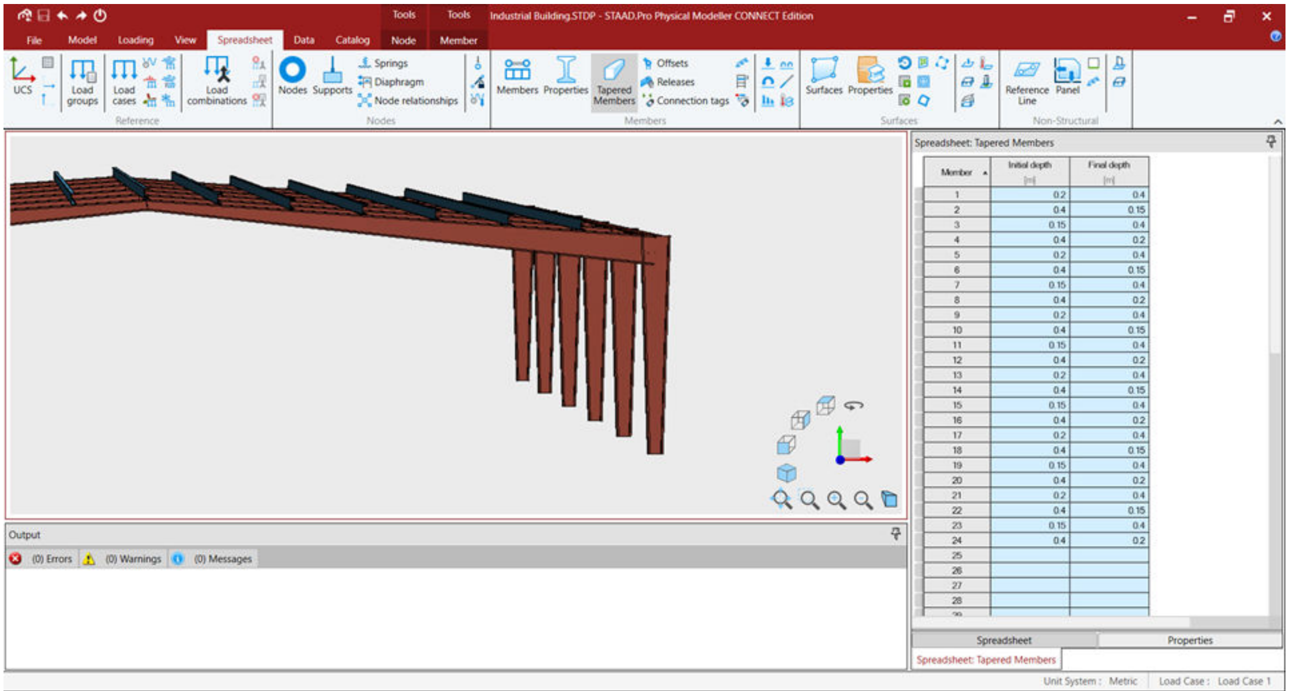
RR 22.07.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.07.00-2.1 Tapered Members in Physical Modeler

STAAD.Pro Physical Modeler can now be used to model web-tapered I shape, rectangular tube, and pipe members.

Refer to the STAAD.Pro Physical Modeler help for additional details on assigning web tapers in the physical modeler.



RR 22.07.00-2.2 Multiple Analyses in Physical Modeler

You can now specify multiple analysis commands within STAAD.Pro Physical Modeler. These may utilize different analysis methods, different load sets, or both.

You can also manage multiple analyses in **Analysis Sets** to quickly set each as active or inactive and edit them individually.

Refer to the STAAD.Pro Physical Modeler help for additional details on assigning web tapers in the physical modeler.

RR 22.07.00-2.3 Improved Structure Wizard in Physical Modeler

Additional templates and script-editing capabilities have been added to the STAAD.Pro Physical Modeler Structure Wizard.

Refer to the STAAD.Pro Physical Modeler help for additional details on using the Structure Wizard.

RR 22.07.00-2.4 Chinese Wind Load Calculator per GB 50009-2012

The program can now calculate wind load vs height intensity values per the Chinese GB 50009-2012 code.
Related Links

- *Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog* (on page 2887)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *M. To add a GB50009 wind load definition* (on page 848)

RR 22.07.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.07.00-3.1 Response Spectra per GB 50011-2010

Automatically generated response spectra loads per the Chinese GB 50011-2010 code are now available in STAAD.Pro.
Related Links
- *TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010* (on page 2531)
- *Response Spectra tab* (on page 2854)
- *Generated Spectrum dialog* (on page 2862)
- *Spectrum Parameters dialog* (on page 2862)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)

RR 22.07.00-3.2 Element Offsets Use with Additional Analysis Methods

The program can now analyze models with element offsets using dynamic analysis, direct analysis, and P-Delta analysis.
Related Links
- *TR.25.2 Element Offset Specification* (on page 2298)

RR 22.07.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.07.00-4.1 Chinese Steel Design Workflow

The design of steel sections to the Chinese code is now integrated into the STAAD.Pro interface as the Chinese Steel Design workflow

Note: Previously, the SSDD application was only available as a standalone application.

CONNECT Edition V22 Update 6

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 6 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 5 (release 22.05). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.06.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.06.00-1.1 Improved Surface Mesh Workflow

The workflow and dialogs to generate automatically meshed surfaces has been improved.

You can also specify the usage of a surface by wall or slab, as well as a sub-type definition for the use in tank structures.

Related Links

- *Add Parametric Model dialog* (on page 2737)
- *M. To create a parametric surface model* (on page 671)

RR 22.06.00-1.1 Ideas Submission Button

Do you have a suggestion for STAAD.Pro you would like to share with us?

On the **File** ribbon tab, select the **Help** tab and then click the **Ideas** button. You can then share your suggestion in our Bentley Communities forum specifically for submitting STAAD suggestions.

You can also vote for other suggestions that you would like to see implemented.

Related Links

- *Help tab* (on page 2704)

RR 22.06.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.06.00-2.1 Physical Modeler Analysis Commands

STAAD.Pro Physical Modeler can now be used to specify the analysis method and parameters used in STAAD.Pro.

Refer to the STAAD.Pro Physical Modeler help for additional details on analysis in the physical modeler.

RR 22.06.00-2.2 Surface Releases

You can now specify edge and corner releases for surfaces in STAAD.Pro Physical Modeler.

Refer to the STAAD.Pro Physical Modeler help for additional details on changing surface boundary or node releases.

RR 22.06.00-2.3 Modular Tank Structure Wizard

You can now rapidly model concrete tanks in STAAD.Pro Physical Modeler using the new Structure Wizard facility.

Note: This feature requires Structural SELECT Entitlements, which are provided with an active SELECT or ELS subscription.

Refer to the STAAD.Pro Physical Modeler help for additional details on creating a tank model.

RR 22.06.00-2.4 Direct Analysis Properties

You can now assign direct member properties for use with the AISC direct analysis directly in STAAD.Pro Physical Modeler.

Refer to the STAAD.Pro Physical Modeler help for additional details on assigning direct analysis properties.

RR 22.06.00-2.5 iTwin Synchronizer

You can now use Bentley's iTwin Synchronizer to push and pull updates to and from ISM repositories via either a local ISM repository or iModelHub.

Note: This functionality replaces the previous ISM Structural Synchronizer workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using iTwin Synchronizer and how to synchronize data with iModelHub.

Technical Preview: This feature is part of a Technical Preview. It has not been reviewed per Bentley's standard quality assurance program and should be considered for review only.

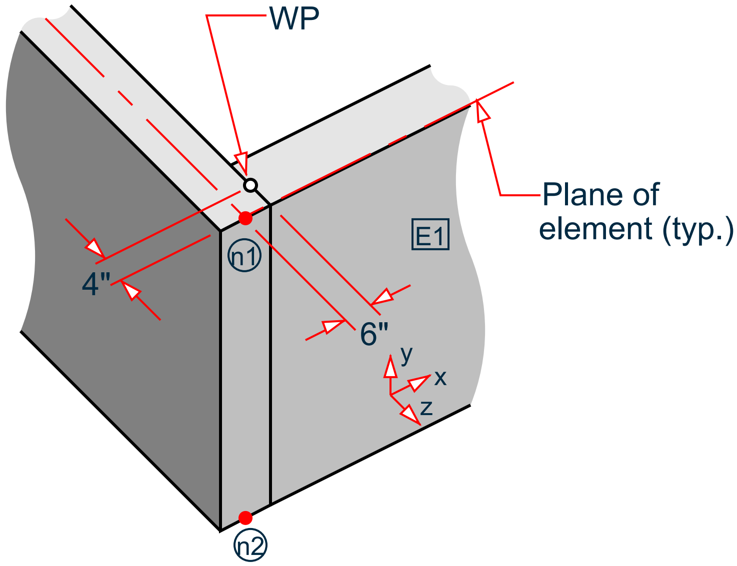
RR 22.06.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.06.00-3.1 Element Offsets

The program can now use rigid link offsets at the corners of plate elements.

This feature is limited to use with linear, static analysis for this release.



Related Links

- *TR.25.2 Element Offset Specification* (on page 2298)
- *Plate Specs dialog* (on page 2791)
- *To assign plate offsets* (on page 813)

RR 22.06.00-3.2 Design of Solid Rods per IS800

The program can now design solid rod sections per IS 800-2007 using both the LSD and WSD methods.

Note: Solid round sections can be found in the Dutch (Netherlands) profiles. Alternatively, solid round prismatic shapes with steel material can be used.

RR 22.06.00-3.3 Static Seismic Loading per GB50011-2010

The program can now apply static seismic loading per the Chinese GB50011-2010 (2016 edition) code.

Note: As GB50011-2010 static seismic was added in CONNECT Edition V22 Update 6, in order to use the 2001 edition you must directly specify the code year. Otherwise, the program will default to the latest code edition as is typical for STAAD.Pro.

Related Links

- *TR.31.2.6 Chinese Static Seismic per GB50011-2010* (on page 2375)
- *Add New Seismic Definitions dialog* (on page 2895)

RR 22.06.00-3.4 Design of Lipped Sections per AISI 2016

The program can now design lipped sections (C shapes, Z shapes, and angles) per the 2016 edition of the AISI design code.

RR 22.06.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.01.00-4.2 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.4 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.4 (CONNECT Edition V13 Update 4).
- Seismic design enhancements for AISC 360-16 / AISC 358-16
- New gusset to brace connections
- Enhancements to gusset CVR and CBB connections for AISC codes
- Enhancements to horizontal gussets - HCBB and HBBB connections
- Enhancements to base plate connections
- Update databases
- Drawing joints

CONNECT Edition V22 Update 5

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 5 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 4 (release 22.04). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.05.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.05.00-1.1 Model Seed File

You can now use an existing STAAD input file as a template, or “seed” file, to include initial data in a new model.

When creating a new model, you will have the option to locate a STAAD input file to use as a seed file or to select a previously used seed file from a drop-down list.

Related Links

- *GS. To create a new STAAD.Pro model* (on page 35)

RR 22.05.00-1.2 Auto-Recovery Enhancements

The auto-recovery feature in STAAD.Pro has been enhanced to improve performance with large models.

The improved auto-recovery facility will only run if the model has been changed since the last Save action.

Auto-recovery can be turned on or off by selecting the option in the Quick Access toolbar. This allows you to prevent any performance issues while working with larger files between your Save actions. This will not stop an active auto-recovery process, but will prevent it running subsequently.

Related Links

- *GS. Quick Access Toolbar* (on page 62)
- *I. To enable auto-recovery* (on page 2065)

RR 22.05.00-1.3 iTwin Design Review Ad Hoc Method

STAAD.Pro Physical Modeler now incorporates Bentley's Design Review Ad Hoc method for collaborating on design projects.

iTwin Design Review Ad Hoc Workflow enables quick peer reviews of design work-in-progress deliverables while eliminating the inefficiency of traditional design review workflows and provides a great opportunity for collaboration with team members. You can learn more by visiting https://www.bentley.com/en/products/product-line/digital-twins/itwin-design-review .

Design Review is found by selecting the **iTwin Services** tool on the **Model** ribbon tab in STAAD.Pro Physical Modeler. Refer to the STAAD.Pro Physical Modeler help section iTwin Services for additional details.

Technical Preview: This feature is part of a Technical Preview. It has not been reviewed per Bentley's standard quality assurance program and should be considered for review only.

RR 22.05.00-1.4 Cloud Analysis

STAAD.Pro now allows you to upload and analyze your STAAD models in Bentley's Cloud Services.

For large or complex models, this can save you significant computing time.

Technical Preview: This feature is part of a Technical Preview. It has not been reviewed per Bentley's standard quality assurance program and should be considered for review only.

Related Links

- *A. To run analysis on the cloud* (on page 967)

RR 22.05.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.05.00-2.1 Physical Modeler Automatic Load Combinations

STAAD.Pro Physical Modeler can now generate automatic load combinations based on load case type.

Refer to the STAAD.Pro Physical Modeler help for additional details on creating automatic load combinations.

RR 22.05.00-2.2 Notional Loads in the Physical Modeler

STAAD.Pro Physical Modeler can now generate load combinations with notional loads.

Refer to the STAAD.Pro Physical Modeler help for additional details on adding a notional load.

RR 22.05.00-2.3 Physical Modeler ASCE 7 Wind Loads

STAAD.Pro Physical Modeler can now generate wind loads per ASCE 7-10 or ASC 7-16.

Refer to the STAAD.Pro Physical Modeler help for additional details on adding an ASCE 7 wind load definition.

RR 22.05.00-2.4 Physical Modeler Node Relationships

STAAD.Pro Physical Modeler can now define rigid links and other relationships between control and dependent nodes.

Refer to the STAAD.Pro Physical Modeler help for additional details on defining node relationships.

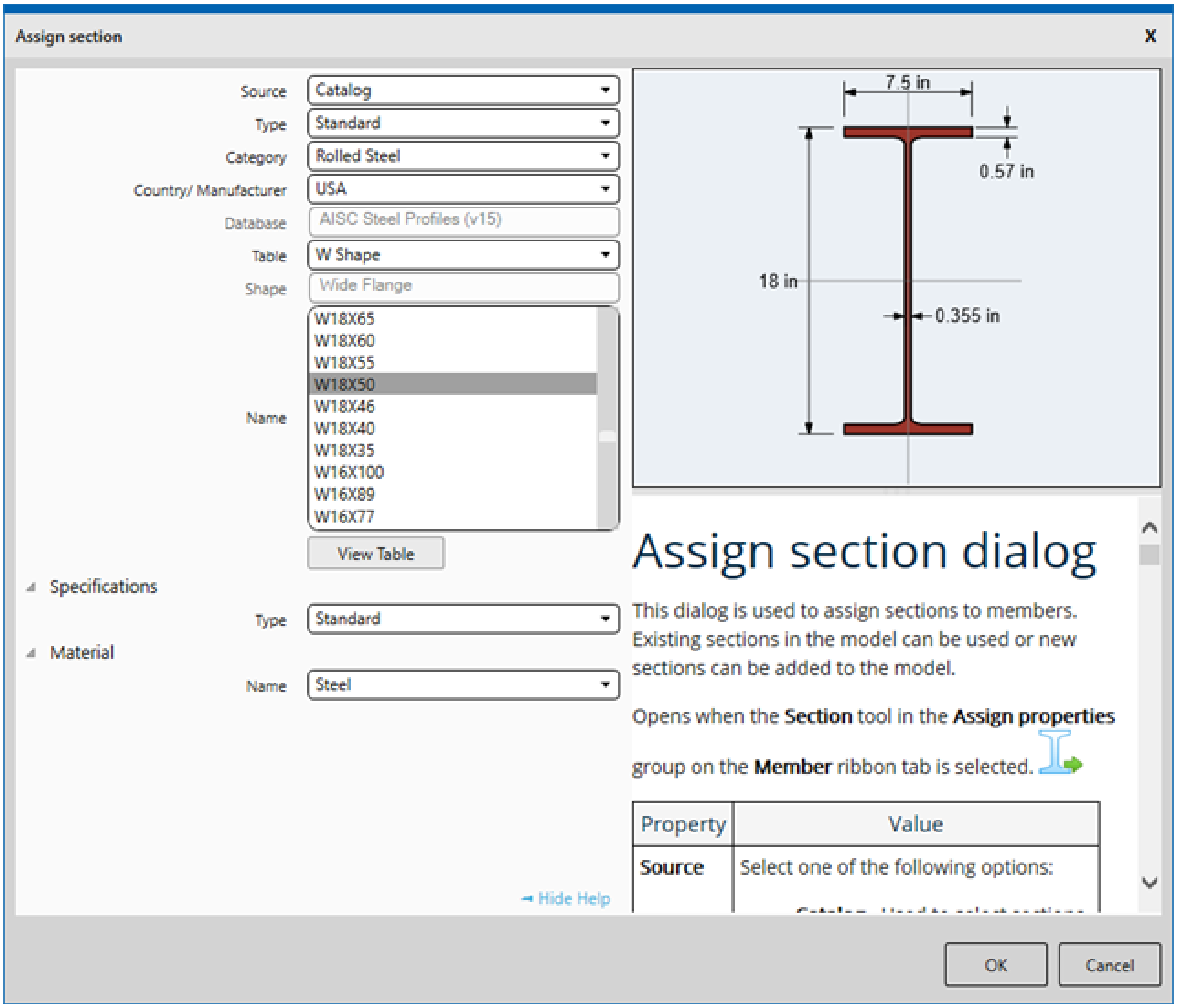
RR 22.05.00-2.5 Physical Modeler Section Database

STAAD.Pro Physical Modeler now shares the same section database as STAAD.Pro.

The Section Database Manager can now be accessed from within STAAD.Pro Physical Modeler. This means there is a 1:1 correlation between the available sections in both the physical and analytical modeling workflows. This ensures that any section assigned in STAAD.Pro Physical Modeler will exactly correspond to the same during analysis, design, and post-processing.

Note: Castellated rolled steel beams, R-Ceco rolled steel sections, European bulb flat sections, European flat bar sections, European solid square sections, steel joists, some cold-formed steel sections, AITC glulam sections, and aluminium profiles are not supported in STAAD.Pro Physical Modeler.

The **Assign section** dialog in STAAD.Pro Physical Modeler has further been enhanced to include a dimensioned preview of the currently selected cross section to assist in member section assignment. This is available in the Help panel.



RR 22.05.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.05.00-3.1 IS 800-2007 Seismic Detailing per Section 12

The program can now check seismic detailing per Section 12 of IS 800 2007 when a IS 1893 seismic definition is used for earthquake loading.

Note: This is only available for design using the LFD method. It is not applicable to design using the ASD method.

Additional load combinations for use with IS800-2007 section 12. These are contained in a new load combination table in the automatic load combinations generator.

New design parameters are available for this check. The `SEISMIC` is used to initiate a seismic check per Section 12. The `IMM` parameter is used to specify the member type. The values of the `SSY` and `SSZ` parameters are used to specify the frame type (as previous) or to indicate that the member is not part of a lateral earthquake load in that direction (the default for these parameters).

RR 22.05.00-3.2 SP 16.13330.2017 Steel Design

The program can now perform steel design based on the Russian code Сп 16.13330.2017 *стальные конструкции* (SP 16.13330.2017 *Steel Structures*).

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

RR 22.05.00-3.3 Wind Loads per ASCE 7-16

The program can now automatically generate wind load intensity values per the 2016 edition of ASCE 7 *Minimum Design Loads for Buildings and Other Structures*.

The 2016 edition ASCE 7 now accounts for the height above sea level of the structure through a new coefficient, K_e as well as changes to the values used for other coefficients. These include the horizontal cross-section types used in the calculation of K_d as well as a new enclosure classification, partially open, for GC^{pi}.

$$q_z = 0.00256 K_z K_{zt} K_d V^2 \quad \text{(eq. 26.10-1)}$$

Related Links

- *ASCE 7 Wind Load dialog box* (on page 2882)
- *M. To add an ASCE 7 wind load definition* (on page 845)

RR 22.05.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.05.00-4.1 General and UPT Shapes in Steel AutoDrafter

The Steel AutoDrafter workflow has been updated to support section profiles defined as general shapes or user-provided table shapes.

RR 22.05.00-4.2 SSDD Integration in STAAD.Pro

Design of steel members per the Chinese code GB50017-2017 can now be initiated directly from STAAD.Pro.

The SSDD application can be launched by selecting the **Chinese Steel Design** workflow.

Related Links

- *D. To open a model in the Chinese Steel Design workflow* (on page 979)

RR 22.05.00-4.2 RC Designer is Retired

The Interactive Concrete Design workflow, formerly named RC Designer, has been completely retired from STAAD.Pro.

In order to perform interactive concrete design, the Advanced Concrete Design workflow may be used.

RR 22.05.00-4.4 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.3 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.3 (CONNECT Edition V13.3).
- Support for channel sections for AISC, CSA, AS, NZS, EN, and IS code designs of single plate, through plate, standard tee, flexible end plate, double-angle web cleats, flange plate beam splices, and base plates.
- CSA S16-14 bracing connections for column - beam - brace joints.

CONNECT Edition V22 Update 4

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 4 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 3 (release 22.03). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

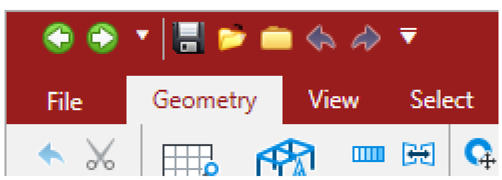
RR 22.04.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.04.00-1.1 Workflow Navigation Buttons

Forward and back buttons have been added to help navigate through the workflow pages.

When selecting workflow pages or ribbon tabs, you may want to move to a previously selected page. The **Back to** tool can be used for this. The **Forward to** tool can then be used to move to the next page in the history (i.e., the reverse order of the backwards navigation).



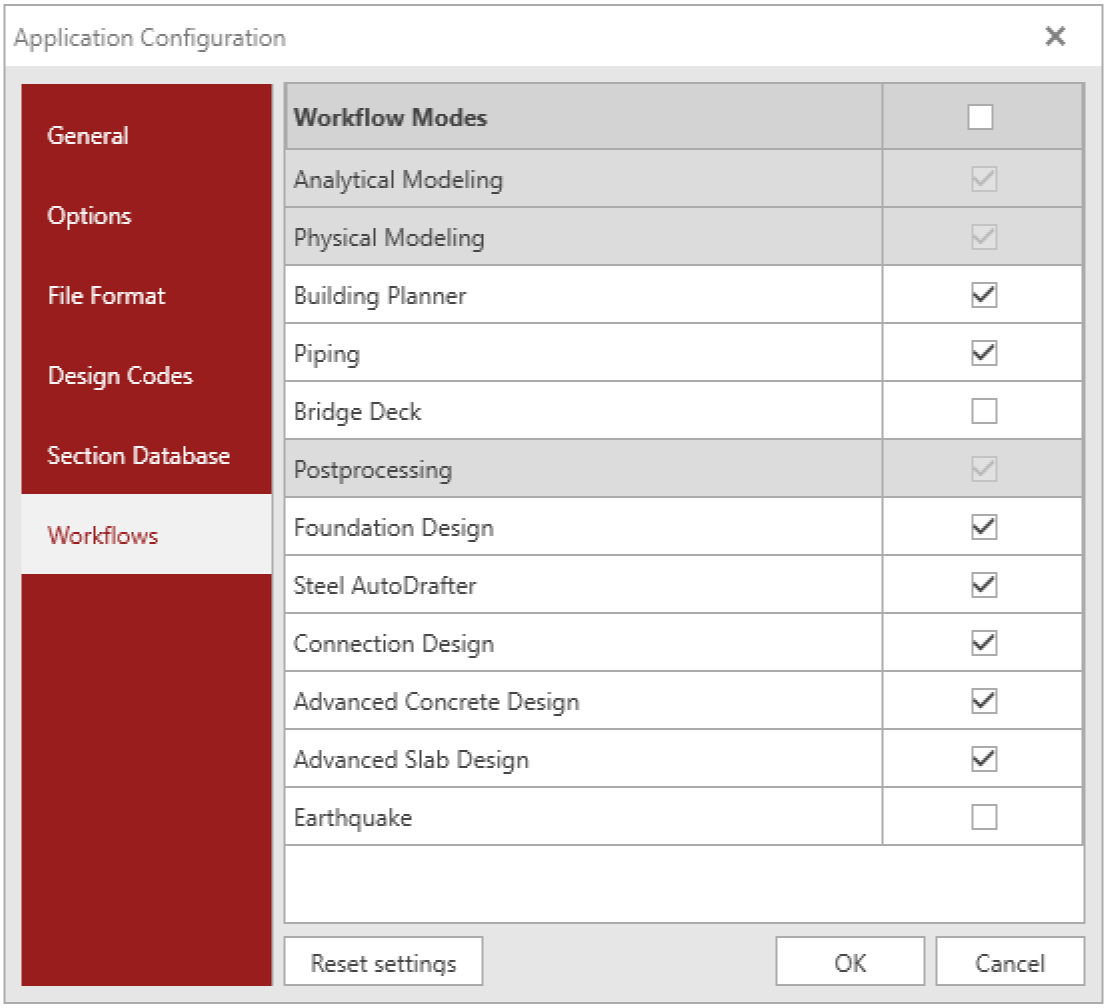
Related Links

- *GS. Page Control* (on page 64)
- *GS. Quick Access Toolbar* (on page 62)

RR 22.04.00-1.2 Workflow Display Customization

You can select which STAAD.Pro workflows are displayed.

The **Application Configuration** dialog contains a **Workflows** tab that allows you to select which workflows are displayed. Uncheck any workflows you want hidden to clean up your work space.



Tip: To reset the Workflows display to the default, check the **Workflow Modes** option.

Related Links

- *GS. Application Configuration dialog* (on page 59)
- *GS. To customize the workflows panel* (on page 39)

RR 22.04.00-1.3 Compare Restore Points

You can compare previously created restore points (backups) with one another or with the current model.

Related Links

- *I. To compare backups* (on page 2066)

RR 22.04.00-1.4 Remove Missing Files from Recent Files List

You can now remove one or all missing files from the recent files list.

Related Links

- *GS. To remove missing files from the recent files list* (on page 61)

RR 22.04.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.04.00-2.1 Physical Modeler Move Tool

STAAD.Pro Physical Modeler includes a Move tool which can be used to move selected portions of the model.

This tool also includes an option to retain connectivity, so the connections between the moved portions and unmoved portions remain intact.

Refer to the STAAD.Pro Physical Modeler help for additional details on moving selected portions of the model.

RR 22.04.00-2.2 Physical Modeler Reference Line Supports

You can now model linear supports along any reference line in STAAD.Pro Physical Modeler.

These supports can be restraints or springs and act similar to surface edge supports.

Refer to the STAAD.Pro Physical Modeler help for additional details on adding reference line supports.

RR 22.04.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.04.00-3.1 Consolidation to Advanced Solver

The advanced solver is now solely used in STAAD.Pro, regardless of license.

The Basic solver has been deprecated. No action is required when launching the program or when running an analysis.

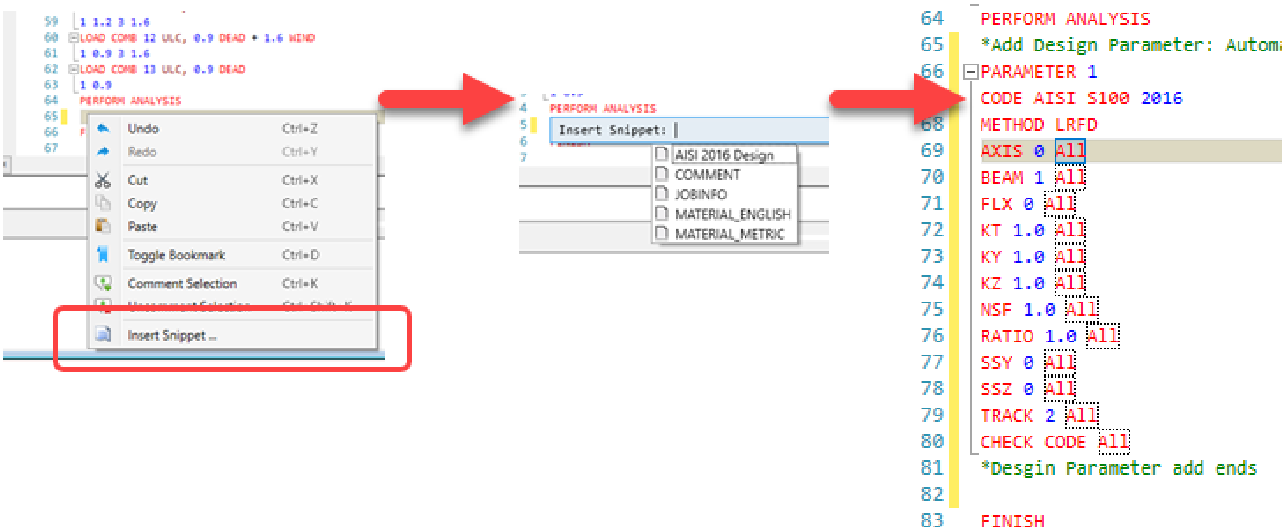
Note: The STAAD.Pro Advanced license is still required for advanced features, such as time history analysis.

RR 22.04.00-3.2 AISI S100-16 Cold Formed Steel Design

The program can now perform cold-formed steel design based on the 2016 edition of the AISI S100 code “North American Specification for the Design of Cold-Formed Steel Structural Members”.

Note: The 1996 edition of the AISI design code has been deprecated. If this edition of the code is specifying using `AISI 1996` in the input file, an error message will be presented in the output.

Tip: When using the STAAD.Pro Editor, you can right-click and select **Insert Snippet** from the pop-up menu and then select **AISI 2016 Design** from the snippets list. This will insert the Parameter block with the commonly used parameters set to their defaults for all members. Refer to I. To insert a code snippet (on page 2051) for details on using snippets.



RR 22.04.00-3.3 IS 1893 2015 Part 4 Static Seismic Loads

The program can now add static seismic loads per IS 1893 2015 (Part 4) *Industrial Structures Including Stack-Like Structures*.

Note: This implementation is only applicable to industrial structures. It is *not* valid for stack-like structures.

Related Links

- *TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load* (on page 2398)

RR 22.04.00-3.4 IS 1893 2015 Part 4 Response Spectra

The program can now add response spectra loads per IS 1893 2015 (Part 4) *Industrial Structures Including Stack-Like Structures*.

Related Links

- *TR.32.10.1.9 Response Spectrum Specification per IS: 1893 (Part 4)-2015* (on page 2555)

RR 22.04.00-3.5 Eurocode 3 Belgian NA Updated to 2018

The program can now perform steel design based on the 2018 edition of the Belgian national annex to EN 1993-1-1 ANB: 2018.

RR 22.04.00-3.6 SP 16.13330.2017 Steel Design

The program can now perform steel design based on the Russian code Сп 16.13330.2017 *стальные конструкции* (SP 16.13330.2017 *Steel Structures*).

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

RR 22.04.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.04.00-4.1 RAM Connection Templates in Connection Tags

An additional element has been added to the connection tag XML structure to allow associating connection tag types with RAM Connection templates.

Related Links

- *D. Connection Tags XML File Schema* (on page 1052)

RR 22.04.00-4.2 Prismatic Shapes in Steel AutoDrafter

The Steel AutoDrafter workflow has been updated to support section profiles defined as prismatic.

This includes prismatic shapes of the following type:

- Solid Circle (defined with YD),
- Solid Rectangle (defined with YD and ZD),
- Tee (defined with TD, ZD, YB and ZB),
- Trapezoidal (defined with YD, ZD and ZB),
- General (displayed as a rectangular solid using the values of YD and ZD),
- Tapered I (displayed using the average dimensions of web and flange),
- Tapered Tube (displayed using the average dimensions).

CONNECT Edition V22 Update 3

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 2 (release 22.02). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.03.00-1 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.03.00-1.1 Physical Modeler Surface Regions and Region Loads

You can now designate surface regions within STAAD.Pro Physical Modeler surfaces. These can be used to model drop panels, surface recesses, and patch loadings.

Surface regions are “children” of surfaces but can have different material properties and thicknesses. You can also align the thickness of a surface region to either the face or center of the “parent” surface.

Refer to the STAAD.Pro Physical Modeler help for additional details on:

- creating a surface region
- applying a pressure load over a surface region
- applying a hydrostatic load to a surface region

RR 22.03.00-1.2 Physical Modeler Reference Lines and Loads

You can now add reference lines to surfaces in STAAD.Pro Physical Modeler. A reference line can be used to control meshing as well as to add a line load anywhere within a surface.

Refer to the STAAD.Pro Physical Modeler help for additional details on:

- drawing a reference line
- adding a reference line load

RR 22.03.00-2 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.03.00-2.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.2 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.2 (CONNECT Edition V13.2).
- Connection Design per the CSA S16-14 (Canadian) code

RR 22.03.00-2.2 RCDC Features for STAAD.Pro License Users

Some features of the Advanced Concrete Design workflow using the RCDC application is now available to STAAD.Pro (basic) license users.

The full set of features in RCDC is available to STAAD.Pro Advance license users. For STAAD.Pro (basic), the following features are now available:

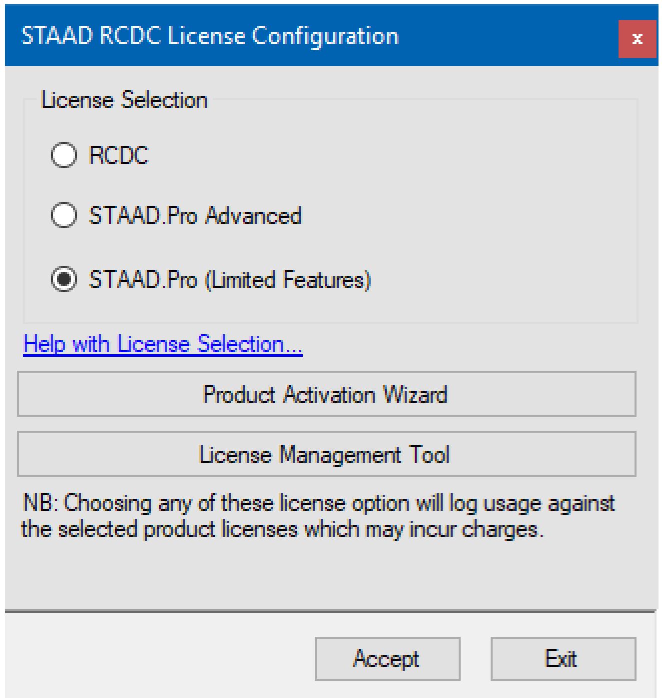
- beam design
- column design
- shear wall design

Other features available to STAAD.Pro (basic) license users include generation of the following, limited to on-screen only with no editing:

- Bill of Quantities summary
- Bar Bending Schedule generation
- Drawing generation

Other RCDC features are disabled under a STAAD.Pro (basic) license.

Note: Use caution when opening and saving an RCDC model created under a STAAD.Pro (basic) license. Opening and saving that file in RCDC under a RCDC or STAAD.Pro Advance license will result in that file no longer being able to be opened under a STAAD.Pro (basic) license.



Interactive Concrete Designer

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

The legacy Concrete Design workflow has been fully deprecated as of CONNECT Edition V22 Update 5, you will no longer be prompted with the option to switch to Concrete Design Workflow and launch RC Designer; even for the legacy projects which used the Concrete Design Workflow. Advanced Concrete Design Workflow only opens RCDC

CONNECT Edition V22 Update 2

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 2 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 Update 1 (release 22.01). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.02.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.02.00-1.1 Temporary Folder for Analysis Over Network

Analysis of input file located on a network drive can result in significantly increases analysis times. STAAD.Pro now gives you the option to specify a local temporary folder to use for when the input files are on a network.

There is also a setting to provide a warning message when the analysis is being performed over a network. Both warning message and temporary folder locations are configured in the **Application Configuration** dialog **Options** tab.

Related Links

- *GS. Application Configuration dialog* (on page 59)

RR 22.02.00-1.2 Updated Backup and Restore of Projects

A new system for creating backups and restoring from backups has been implemented. This now includes physical model files for projects.

A restore point is a backup of all files associated with a STAAD input model, including a physical model if one is present. The option to include analysis output files is also available when creating a restore point.

Note: This method replaces the previous method of creating backups, including automated backups, in STAAD.Pro.

A new auto-recovery feature has been added to replace the previous Autosave method. This feature will automatically create backups based on a specified time interval (default of 5 minutes) of your entire STAAD project, including physical models. In the event of a program crash or if the program is simply closed without saving changes, the program will detect the auto recovery files and prompt you to open from that recovery state when you re-open the associated STAAD model.

Related Links

- *I. To create a restore point* (on page 2065)
- *I. To restore a model from a backup* (on page 2066)
- *I. To enable auto-recovery* (on page 2065)

RR 22.02.00-1.3 Chinese Steel Sections Update

The database of Chinese steel profiles has be updated to GB/T 706 2016.

RR 22.02.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.02.00-2.1 Physical Modeler Snow Loads

STAAD.Pro Physical Modeler can now generate snow load cases for use directly in the Analytical Modeling workflow.

STAAD.Pro Physical Modeler also now allows you to “group” model objects (members, surfaces, and nodes) together for applying loads such as snow loads.

Refer to the STAAD.Pro Physical Modeler help for additional details on using snow loads.

RR 22.02.00-2.2 Physical Modeler Wind Loads

STAAD.Pro Physical Modeler can now generate wind load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using wind loads.

RR 22.02.00-2.3 Physical Modeler Temperature & Strain Loads

STAAD.Pro Physical Modeler can now apply temperature and strain loads for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using temperature or strain loads on members and applying temperature loads to surfaces.

RR 22.02.00-2.4 Physical Modeler Variable Pressure and Hydrostatic Loads

STAAD.Pro Physical Modeler can now apply variable pressure loads and hydrostatic to surfaces.

Refer to the STAAD.Pro Physical Modeler help for additional details on using variable pressure loads.

Hydrostatic loads are a means of generating a variable pressure loads using fluid density and depth parameters.

Refer to the STAAD.Pro Physical Modeler help for additional details on using hydrostatic loads.

RR 22.02.00-2.5 Physical Modeler Inclined Nodal Loads

STAAD.Pro Physical Modeler can now apply nodal loads along user-defined directions in addition to global axes.

Refer to the STAAD.Pro Physical Modeler help for additional details on using inclined nodal loads.

RR 22.02.00-2.6 Physical Modeler Edit Surface Mode

STAAD.Pro Physical Modeler has a new tool which allows you to edit surfaces (such as materials, thickness, openings, etc.) in an isolated, elevation view. You can then merge the surface edits back to your main model.

Refer to the STAAD.Pro Physical Modeler help for additional details on adding polygonal openings and adding circular openings to surfaces using this feature.

RR 22.02.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.01.00-3.3 IS 13920 2016 Seismic Joint Checks

STAAD.Pro can now check the relative strength of beams and columns at a joint per Clause 7.2 of IS 13920-2016.

Clause 7.2 of IS:13920-2016 checks the relative strength of beams and columns at a joint to evaluate if any column at a joint is acting as a gravity column or is a part of lateral load resisting system. After the concrete design is completed per IS 13920-2016, then this relative strength check is automatically performed at joints in which *all* connecting beams and columns were designed. Thus, no new commands are required for this check to be performed so long as all concrete members at a joint are designed. Per Cl 7.2, the sum of all column moment resistances at a joint shall be greater that 1.4× the sum of all beam moment resistances at the same point.

Note: STAAD.Pro does not have designate columns as gravity load structural elements. Rather, the program considers all columns as lateral load resisting. If this check fails at any joint, the column dimensions, beam dimension, or both should be changed to make the column suitable for the lateral load resisting system.

It is recommend that the design of all concrete beams and columns be performed in one concrete block (a “concrete block” is the information contained in the `START CONCRETE DESIGN … END CONCRETE DESIGN` input section) so this check can be performed. If multiple concrete blocks are defined, all previous relative strength check result will be deleted since multiple design of beams and column designs may change the relative strength of beams and columns at a joint. The strength check performed after the last concrete block will be considered valid and is available as result.

The output file will issue a note regarding the results of this check. A detailed output of the checks can be included in the output using the `TRACK 4` parameter in the concrete design. The detailed output of the joint relative strength checks are also included in the *`filename`*`_13920.txt` file (which is generated regardless of the `TRACK` parameter used or check status).

Note: The program assumes that all beams connected to a column are horiztonal and parallel to global X and Z axes. If beams do not meet this requirement a warning message is issued in the analysis output.

Related Links

- *D8.F.4 Design Parameters* (on page 1686)

RR 21.03.00-3.1 Response Spectra per IBC 2018 / ASCE 7-16

Automatically generated response spectra loads per IBC 2018 / ASCE 7-16 are now available in STAAD.Pro.

The spectral acceleration data is now retrieved using the USGS dynamic web service (where as in older editions of the code this was extracted from text files made available from the USGS).

Notes:

- Exceptions 1 in Clause 11.4.8 of ASCE 7-16 is implemented. In exception 1, for Site class E and S_s > 1.0, Fa value is taking the same for site class C. In this case, there is slight reduction in the seismic force generated.
- The exception of Clause 2 in 11.4.8 of ASCE 7-16 is *not* implemented in STAAD.Pro for response spectra.
- The vertical ground motion for seismic design as described in clause 11.9 is *not* implemented as the program cannot determine the vertical time period, T_v.
- ASCE 7-2016 Clause 12.9.1.4 stipulates that if the combined response for modal base shear, V, is less than that of the calculated base shear from the equivalent lateral force procedure, V_t, then the response spectra values should be scaled up by V/V_t. STAAD.Pro does not automatically check this nor increase the response quantities in this situation. You can check this by simply analyzing TR.31.2.17 IBC 2018 Seismic Load Definition (on page 2416) and comparing the base shear values. The X, Y, or Z factors for the response spectrum can be then increased accordingly if required.

Related Links

- *TR.32.10.1.13 Response Spectrum Specification per IBC 2018* (on page 2578)

RR 22.01.00-3.3 ASCE 7 Seismic Irregularities Checks

STAAD.Pro can now check for vertical and plan irregularities per the ASCE 7 05, 10, and 2016 codes for structures with defined rigid floor diaphragms.

Note: The program can additionally check for soft stories per the ASCE 7 code.

For ASCE 7-05/10/16, the program can check horizontal irregularities (torsional and reentrant corners) and vertical irregularies (mass). Irregular modes of oscillation are not considered for this code.

ASCE 7 Example Output

An example output section of an ASCE 7-2016 seismic irregularities check:

```
                    STAAD.PRO IRREGULARITIES CHECK - (  ASCE7-2016   )   v1.0
                    ****************************************************

--TORSION IRREGULARITY CHECKS

Torsion Irregularity Check
Ref: Fig. C12.3-1 T1- Ratio Limit(s): 1.20, 1.40
------------------------------------------------

Dia.   Extreme Points of Dia in X       Extreme Points of Dia in Z
      Node     Disp.     Node     Disp.     Node     Disp.     Node     Disp.
               (mm)               (mm)               (mm)                (mm)
------------------------------------------------------------------------
   1      3    0.09130      1    0.09993      4    0.09484      1    0.10559
   2     15    0.29636     13    0.30993     16    0.29819     13    0.34410

            Diaphragm ΔX-max/avg ΔZ-max/avg Status
            --------------------------------------
                  1      1.0451   1.0537        OK
                  2      1.0224   1.0715        OK
--GEOMETRY IRREGULARITY CHECKS

Re-Entrant Corner Check.
(Ref: Fig. C12.3-1 T2- Ratio Limit: 0.15 )
------------------------------------------

    Node      Re-Entrant X-Proj  X-Proj/Lx Z-Proj  Z-Proj/Lz  Status
 Connectivity     Node    ( m)             ( m)
----------------------------------------------------------------------
        6->         5      0.0000  0.0000    7.0000  0.7778 Re-Entrant
        4                  1.0000  0.2000    0.0000  0.0000
       18->        17      0.0000  0.0000    7.0000  0.7778 Re-Entrant
       16                  1.0000  0.2000    0.0000  0.0000

                  Diaphragm:    Lx:       Lz:
                              ( m)     ( m)
                  ----------------------------
                       1      5.0000    9.0000
                       2      5.0000    9.0000
--MASS IRREGULARITY CHECKS

Mass Irregularity Check
Ref: Fig. C12.3-2 T2- Ratio Limit: 1.50
---------------------------------------

Dia.   Level     Mass      Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)     ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     0.000    341.643    253.287    Base      1.349    N/A     OK
   2     5.000    253.287    Top        341.643    N/A   0.741     OK
```

Related Links

- *TR.28.2 Floor Diaphragm* (on page 2328)

- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.2 Check Irregularities* (on page 2333)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

RR 22.02.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.02.00-4.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.1 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.1 (CONNECT Edition V13.1).
- Connection Design per the NZS 3404 1997 (New Zealand) code

RR 22.02.00-4.2 Multiple Steel Design Results

The program can now display the results from different steel designs (parameter blocks).

If an input file contains multiple parameter blocks (steel design parameter sets) or multiple design commands (steel code check or member selections) within a single parameter block, then the results are available in the Postprocessing workflow. Parameter sets are numbered and multiple design commands in the same parameter block are given as decimals.

Note: This feature is compatible with AISC 360-16 only in this release.

Note: For AISC 360-2016, a maximum of nine (9) design commands may be specified in a parameter block.

The **Layouts** > **Utilization** table displays results from each parameter block or design command. Similarly, the STAAD Output Viewer displays the results for each in the Contents pane.

Multiple Parameter Blocks

```
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD ASD
CHECK CODE MEMBER 1 TO 10
*
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD LRFD
CHECK CODE MEMBER 1 TO 10
```

Multiple Design Commands

```
PARAMETER 1
CODE AISC UNIFIED 2016
TRACK 2 ALL
```

```
CHECK CODE MEMBER 1 TO 5
SELECT MEMBER 1 to 5
```

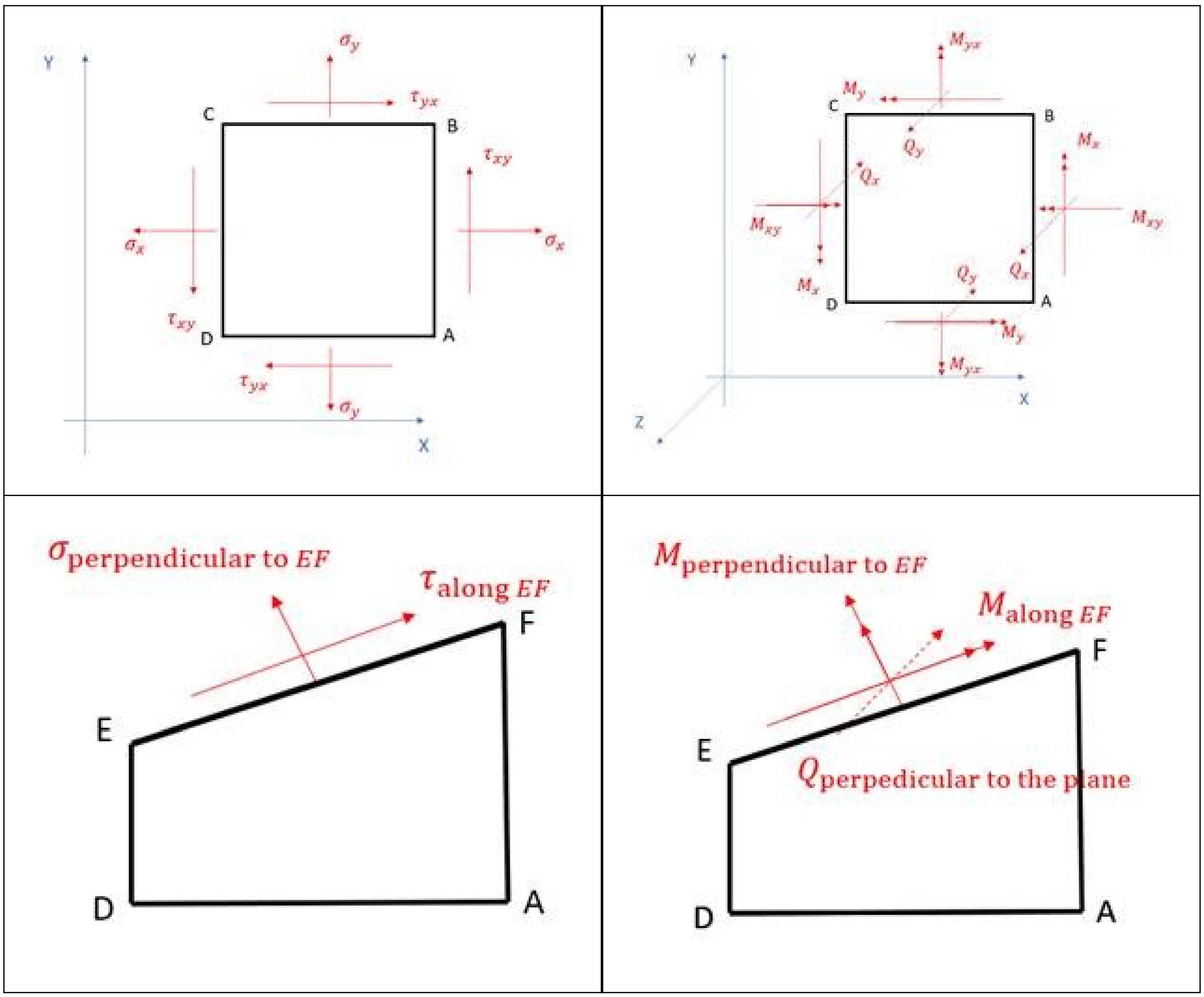
Related Links

- *P. To display steel design utilization ratios* (on page 1986)

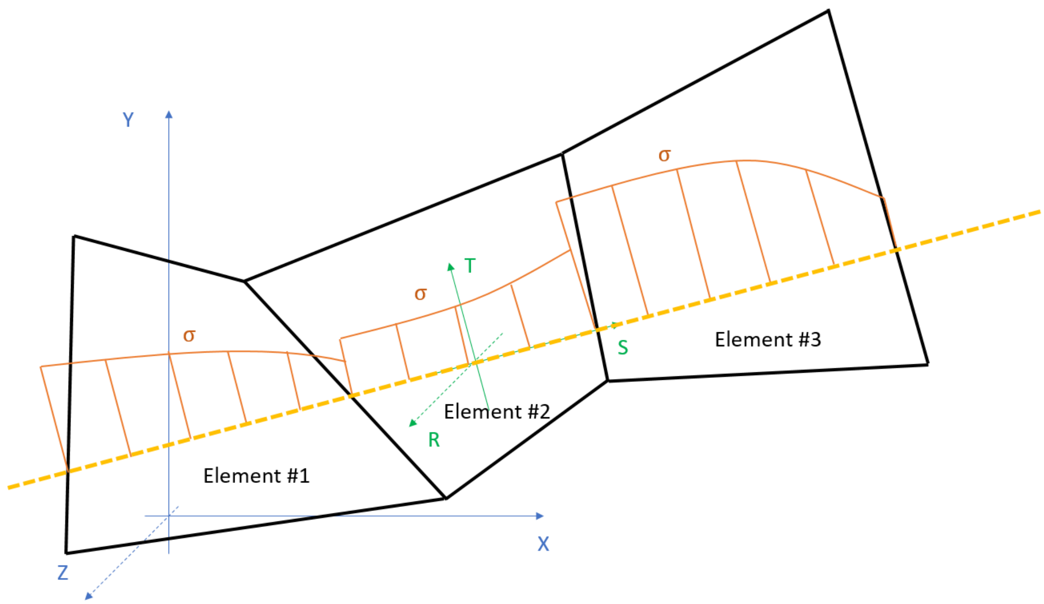
RR 22.02.00-4.3 Results Along a Cut Line

Stresses that are continuous across element boundaries and forces which are equilibrium can now be obtained any arbitrary cut line for elements in the Post Processing workflow.

Given a finite element, ABCD, which is subject to a set of stresses, it is possible using mechanics theory to cut that element along an arbitrary line, EF, and resolve the forces at the cut surface.



However, the forces of adjacent finite elements obtained this way are not balanced (i.e., not in equilibrium) and the stress values are discontinuous across element boundaries.



STAAD.Pro has stress result options for cut lines which employ a method of mathematically processing the FEA stress results which are then in equilibrium against the cross-section forces and are continuous across element boundaries. The cut line coordinate system, STR, is used (i.e., a local coordinate system with S as the axis along the cut line, T in the plane of the elements, and R normal to those elements).

The following nomenclature is used for the cut line-oriented results available in STAAD.Pro:

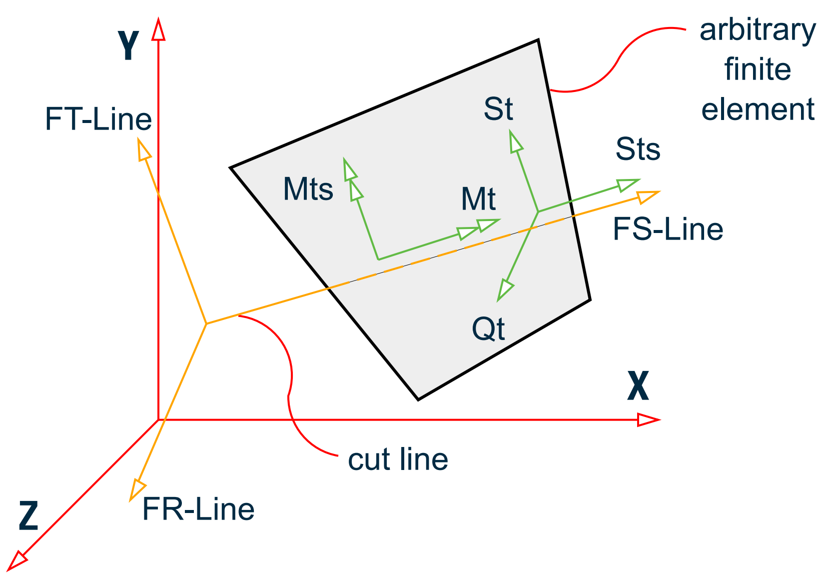


Figure 6: Stresses along a cut line for "-Line" stress types

St Normal stress perpendicular to the cut line, σ_t.

Sts Shear stress along the cut line, τ_t.

Mt Bending moment along the cut line, M_t.

Mts Bending Moment about the T axis. This is in the plane of the plate and perpendicular to the cut line (i.e. the S axis).

Qt Out-of-plane shear, Q_t

Forces listed in the table when these results are selected are given in the in a coordinate system relative to the cut line (i.e., “FS” is along the cut line, “FT” is in the plane of the element, and “FR” is normal to the element). This is indicated by the “-Line” suffix of the results.

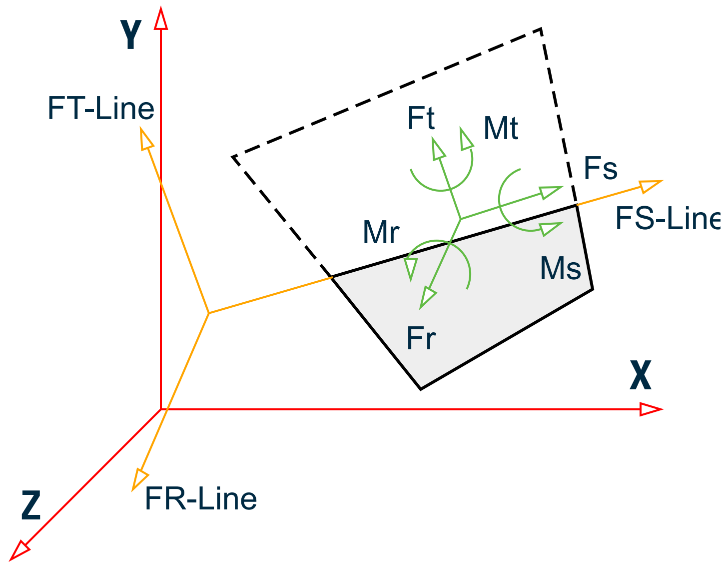


Figure 7: Forces along a cut line for “-Line” stress types

Fs Resultant force along the cutline

Ms Resultant moment about the cutline

Ft Resultant force perpindicular to the culine in the plane of the plate

Mt Resultant moment about the line perpindicular to the cutline in the plane of the plate

Fr Resultant force perpindicular to the cutline out of the plane of the plate

Mr Resultant moment about the line perpindicular to the cutline out of the plane of the plate

Related Links

- *P. To display plate results along a cut line* (on page 1988)

CONNECT Edition V22 Update 1

The Software Release Report for STAAD.Pro CONNECT Edition V22 Update 1 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition V22 (release 22.00). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.01.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 22.01.00-1.1 Reduced Section Properties per IS1893 2016

IS1893 2016 Clause 6.4.3 calls for a reduction of moment of inertia values by 0.35 for beams and 0.7 for columns of concrete members only while analyzing the structure for static seismic and response spectrum/linear dynamic analysis. STAAD.Pro now includes a code-specific option for reduced section properties of concrete members which will automatically generate a separate stiffness matrix for these IS1893 2016 load cases. For all other load cases, the analysis will be performed using the unreduced stiffness matrix.

You may use this feature by either selecting the new **Code Specific** option and then selecting **IS1893 2016** as the code in the **Member Specification** dialog Property Reduction Factors tab. You can manually enter the command using the `MEMBER CRACKED CODE IS1893 2016` command.

Note: Automated stiffness reduction requires a STAAD.Pro Advanced license.

Using the code-specific member reduction properties helps by automating the analysis for when reduced section properties are required by the IS1893 2016 code. A separate stiffness matrix is produced using the reduced stiffness values for use with load cases containing IS1893 2016 static seismic and response spectrum loads. The unreduced stiffness matrix is then used for all other load cases.

The prime advantage of this feature is that this analysis is completely automated once the reduction factors are defined. Previously, in order to comply with the IS1893 2016 Clause 6.4.3 provision, you would have to specify reduced section properties for members, analyze the structure for only the IS1893 2016 seismic load cases, then use a `CHANGE` command to reset the stiffness reduction factors back to unity, and then re-analyze the structure for the remaining load cases. Care had to be taken that only concrete members had the reduction factor applied, as the global `MEMBER CRACKED` command can be applied to *any* member. For large models with hundreds and thousands of members and hundreds of load cases, the implementation of clause 6.4.3 using the previous reduction command would be very cumbersome.

Use of the code-specific reduction factors automates the analysis procedure. These reduction factors are only applied to concrete members with the member list or member groups (non-concrete members are ignored for this command). The separate stiffness matrix is automatically created using these reduction factors for use with IS1893 2016 seismic loads only while the unreduced stiffness matrix is used for all other load cases. No re-analysis or change commands are required.

Related Links

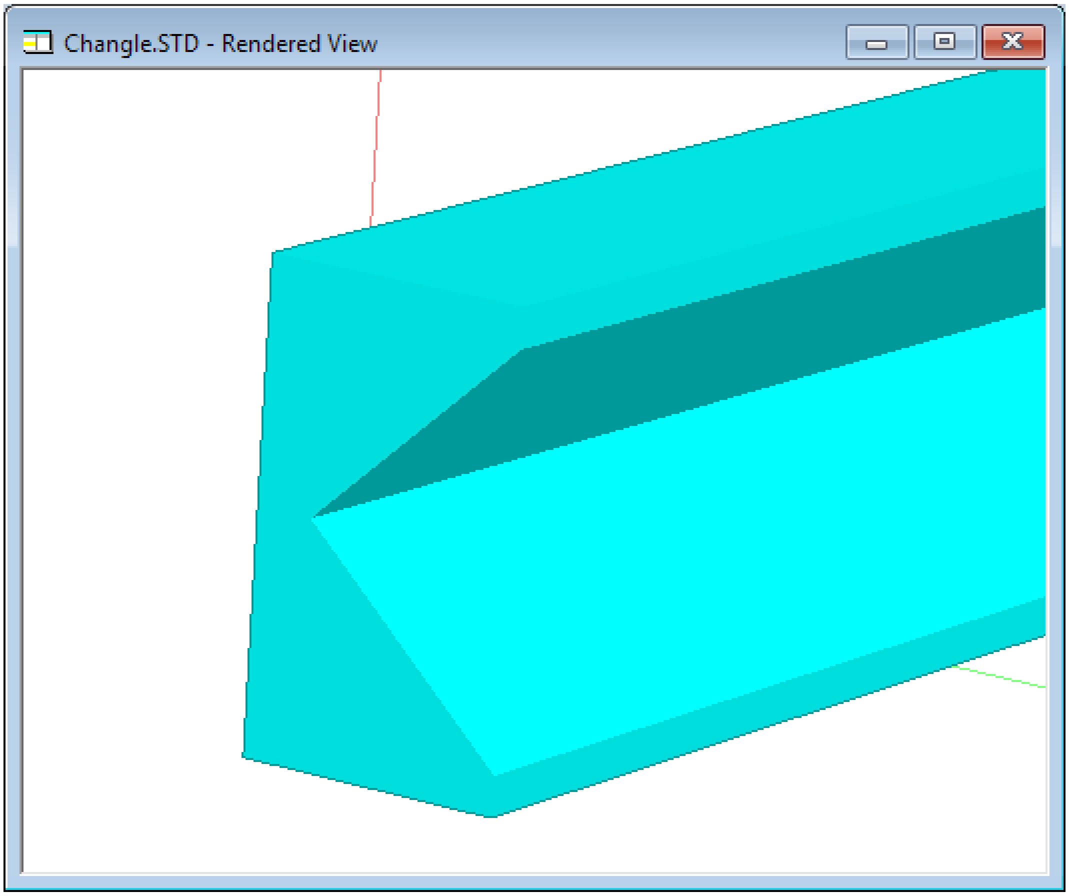
- *Member Specification dialog* (on page 2786)
- *TR.20.10 Member Property Reduction Factors* (on page 2282)
- *M. To assign cracked section properties to a member* (on page 807)

RR 22.01.00-1.2 Import General Sections from Section Wizard

General shapes can now be imported from Section Wizard for use in the program.

Note: This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.

Tip: Changing the structure diagram to display 3D rendering types for Full Sections or Sections Outline will display the shape profile. Similarly, 3D renderings of the structure will also display the shape profile.



Related Links

- *To export shape for use in STAAD.Pro* (on page 775)
- *M. To use a general shape created in Section Wizard* (on page 746)

RR 22.01.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 22.01.00-2.1 Physical Modeler Static Seismic Loads

STAAD.Pro Physical Modeler can now generate static (equivalent lateral force) seismic load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using static seismic loads.

RR 22.01.00-2.2 Physical Modeler Response Spectra Loads

STAAD.Pro Physical Modeler can now generate response spectra load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using response spectrum loads.

RR 22.01.00-2.3 Physical Modeler Time History Loads

STAAD.Pro Physical Modeler can now generate time history load cases for use directly in the Analytical Modeling workflow.

Refer to the STAAD.Pro Physical Modeler help for additional details on using time history loads.

RR 22.01.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 22.01.00-3.1 Static Seismic Loads per IBC 2018 / ASCE 7-16

Automatically generated equivalent lateral seismic loads per IBC 2018 / ASCE 7-16 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor.

Automatic load combinations using the tables from IBC 2018 can be added using the Auto Load Combination dialog (on page 2872), as well.

Changes from IBC 2015/ASCE 7-10 Code

The changes in seismic provisions of IBC 2018/ASCE 7-16 over the previous version IBC 2015/ASCE 7-10 are mainly observed in Mapped Spectral Acceleration parameters outlined in chapter 22. STAAD.Pro uses the spectral acceleration data in text format available in USGS site https://earthquake.usgs.gov/hazards/designmaps/datasets/.

Note: The mapped spectral response acceleration values for IBC 2018 are retrieved from dynamic data provided by the USGS when `ZIP` code or `LAT/LONG` methods are used. An internet connection is required for this. If an internet connection is not available, then the values should be input using the `SS` and `S1` parameters.

The other significant change in the seismic provisions are in long-period site coefficient, F_a, and short-period site coefficient, F_v. Table 11.4-1 and 11.4-2 of the code provides the value of F_a and F_v respectively. These values are used for the IBC 2018 implementation in STAAD.Pro.

Note: Other changes to this portion of the code from the previous edition are not relevant to the calculations performed by STAAD.Pro.

Related Links

- *TR.31.2.17 IBC 2018 Seismic Load Definition* (on page 2416)

RR 22.01.00-3.2 IS 801 Cold-Formed Steel Design

The design of cold-formed steel members per the IS 801 code is again available in STAAD.Pro.

RR 22.01.00-3.3 IS 1893 2016 Seismic Irregularities Checks

STAAD.Pro can now check for vertical and plan irregularities per the IS 1893 2016 code for structures with defined rigid floor diaphragms.

Note: The program can additionally check for soft stories per the IS 1893 2016 code.

For IS 1893 2016, the program can check horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6.

IS 1893 2016 Example Output

An example output section of an IS 1893 2016 seismic irregularities check:

```
-IRREGULARITY CHECKS

                    STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                    *********************************************************

                         Including Amendment no. 2 November 2020
                    *********************************************************

--TORSION IRREGULARITY CHECKS

 Torsion Irregularity Check
 Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
 ---------------------------------------------------------------------
 edi : Design Eccentricity
 esi : Static Eccentricity
 bi  : Floor/Diaphragm plan dimension perpendicular to force direction
 For Details Refer Clause 7.8 IS1893:2016-Part-1
 ---------------------------------------------------------------------


 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
    1      7    0.4984      2    0.4662    0.4823    1.0333     PASS
    2     12    2.4871      9    2.3990    2.4430    1.0180     PASS
    3     16    6.1637     13    6.0157    6.0897    1.0122     PASS

 Using edi = esi - 0.05bi
 ------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
    1      2    0.4984      7    0.4662    0.4823    1.0333     PASS
    2      9    2.4871     12    2.3990    2.4430    1.0180     PASS
    3     13    6.1637     16    6.0157    6.0897    1.0122     PASS

 Using edi = 1.5esi + 0.05bi
 ---------------------------
```

```
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.   Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)       (mm)       Disp.
------------------------------------------------------------------------
   1      3    0.2699      2    0.2383    0.2541    1.0622     PASS
   2     10    1.0087      9    0.9347    0.9717    1.0381     PASS
   3     14    2.0463     13    1.9160    1.9812    1.0329     PASS

Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.   Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)       (mm)       Disp.
------------------------------------------------------------------------
   1      3    0.3916      2    0.2319    0.3118    1.2562   WARNING*
   2     10    1.3253      9    0.9131    1.1192    1.1841     PASS
   3     14    2.5782     13    1.8732    2.2257    1.1584     PASS

  *** WARNING: The floor is irregular.  Please ensure conformance
   with Cl. 7.1, Table 5, Sl No. (i) sec-i.a or sec-i.b.
```

Related Links

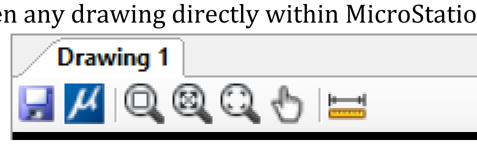
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.2 Check Irregularities* (on page 2333)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

RR 22.01.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 22.01.00-4.1 Steel AutoDrafter Workflow

The Steel AutoDrafter can now open any drawing directly within MicroStation.



Note: If you have MicroStation PowerDraft installed instead of MicroStation, you will have that product's icon instead of the MicroStation icon in the drawing toolbar.

Related Links

- *P.To open a drawing in MicroStation* (on page 2027)

RR 22.01.00-4.2 RAM Connection Workflow Update

RAM Connection CONNECT Edition V13.0 is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 13.0 (CONNECT Edition V13).
- Connection Design per the AS4100 1998 (Australian) code

CONNECT Edition V22

The Software Release Report for STAAD.Pro CONNECT Edition V22 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 3 (release 21.03). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 22.00.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

Related Links

- *GS. STAAD.Pro License Configuration dialog* (on page 57)
- *GS. Starting STAAD.Pro* (on page 33)

RR 22.00.00-1.1 CONNECT Licensing

STAAD.Pro now uses Bentley's CONNECT License system, Bentley's new process for product activation and usage tracking, improving our licensing capabilities with features such as:

- License alert notifications when you are approaching a custom usage threshold
- Replacing site activation keys with user validation, enhancing security around your Bentley licenses and subscriptions

With traditional SELECT Licensing, product activation has been through an activation key that an Organization distributed to all users. With CONNECT Licensing, product activation is managed by user sign in through the CONNECTION Client, which is installed on each machine that uses Bentley applications. This offers a more secure and manageable system as it offers usage alerts, notifying your users when they are about to reach a certain usage limit set by the Administrator.

Detailed information is available at Bentley Communities Licensing and Activation Wiki: https://communities.bentley.com/products/licensing/w/licensing__wiki/37813/connect-licensing for detailed information.

Related Links

- *GS. STAAD.Pro License Configuration dialog* (on page 57)
- *GS. Starting STAAD.Pro* (on page 33)

RR 22.00.00-1.2 Structural Entitlements

The Steel AutoDrafter and Building Planner Workflows added in STAAD.Pro CONNECT Edition v21.03.00.146 as Technical Preview features are now fully commercial and are licensed with a new 'Structural SELECT Entitlement' license.

This license is provided free with a current STAAD.Pro (including STAAD.Pro Advanced) or Structural Enterprise License.

Related Links

- *GS. STAAD.Pro License Configuration dialog* (on page 57)

STAAD.Pro CONNECT Edition

CONNECT Edition Update 3

The Software Release Report for STAAD.Pro CONNECT Edition Update 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 2 (release 21.00.02). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.03.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.03.00-1.1 APL Apollo Structural Tube Sections

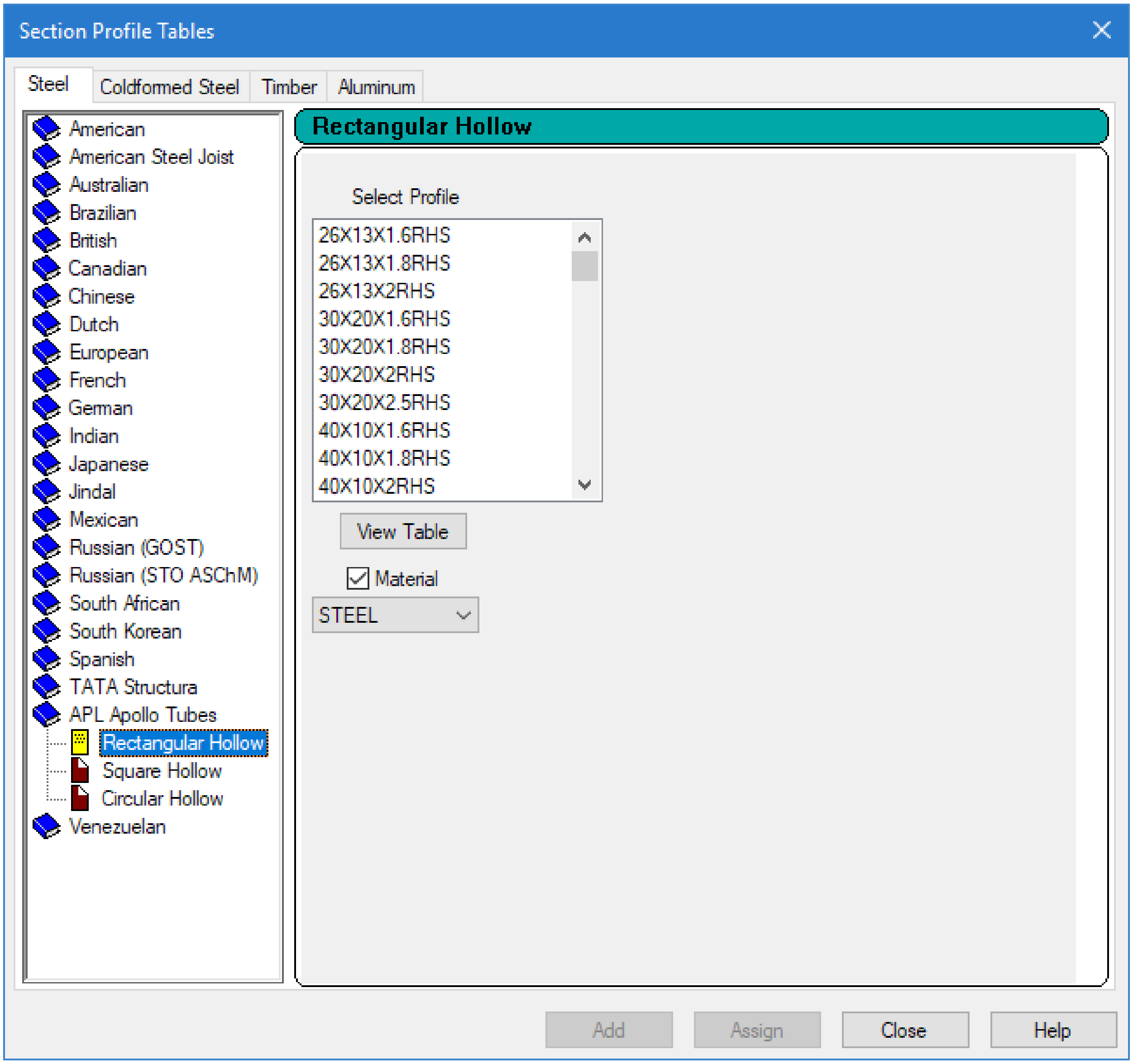
The catalog of available steel sections has been expanded to include the hollow sections from APL Apollo (India).

To specify an APL Apollo tube section

1. Select the **Specifications** ribbon tab.
2. On the **Properties - Whole Structure** dialog, click **Section Databse**.

   The **Section Profiles Table** dialog opens.
3. Select **APL Apollo Tubes** in the list of tables on the **Steel** tab.
4. Select the profile type to use from the list of tables:

   Rectangular Hollow
   Square Hollow
   Circular Hollow



5. Select the profile from the **Select Profile** list.
6. (Optional) Check the **Material** option to specify the selected material with the profile.
7. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members.

Related Links

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *TR.20 Member Property Specification* (on page 2255)
- *Section Profile Tables dialog* (on page 2793)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-1.2 Steel Grade A1085 Profiles in AISC 360-16 and 360-10

Steel pipes and tubes using A1085 grade material can now be designed using AISC 360-16 and AISC 360-10 design codes.

A new set of section tables are available in the American Steel Tables for HSS Rectangle and HSS Round shapes to make use of the A1085 steel grade. Sections using this steel grade have properties which use the *full* nominal wall thickness.

Note: Use of profiles from the _A1085 steel tables will result in that material be used, regardless of the SGR parameter for AISC 360-16 or WTYP parameter for AISC 360-10.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *D1.A.4.1 AISC Steel Table* (on page 1088)
- *D1.A.6 Design Parameters* (on page 1100)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-1.3 Inclined Loads Input via Create New Load Items Dialog

Inclined loads can now be easily added via the graphical user interface in the **Create New Load Items** dialog.

Previously, an inclined nodal load –that is a nodal load applied in coordinate system other than the global coordinate system– required editing the STAAD input file. These loads can now be applied by defining a reference node, relative reference point, or absolute reference point in the graphical user interface's Analytical Modeling workflow.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *Nodal Load tab* (on page 2841)
- *M. To add a nodal load* (on page 832)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-2 Features Affecting the Preprocessor

The following section describes the new features that have been added that affect the preprocessor section of the program, also known as the Analytical Modeling workflow and Physical Modeling workflow (or STAAD.Pro Physical Modeler).

RR 21.03.00-2.1 Physical Modeling Load Cases and Combinations

STAAD.Pro Physical Modeler can now generate load cases and combinations for use directly in the Analytical Modeling workflow.

RR 21.03.00-2.1 Physical Modeling Circular Openings

STAAD.Pro Physical Modeler can now model circular openings in surfaces with a defined radius.

RR 21.03.00-2.3 Physical Modeling Connection Tags

You can now assign connection tags from with STAAD.Pro Physical Modeler. These will be exported to the analytical model in STAAD.Pro as well as to a corresponding ISM repository.

RR 21.03.00-2.4 Miscellaneous Enhancements to the Physical Modeler

The following additional enhancements have been made to the STAAD.Pro Physical Modeler.

- A new file format is used. When opening a model created in a previous version of STAAD.Pro Physical Modeler, you will be prompted to save to the new format.
- New **Spreadsheet** and **Data** ribbon tabs have been added to make the spreadsheet controls consistent with other tools in the application. The **Spreadsheet** ribbon tab is used to select the current spreadsheet. The **Data** ribbon tab is used to perform actions on the current spreadsheet, view window, or output panel (similar to right-click pop-up menus in each of those respective application areas).
- The ISM workflow has been enhanced to place common settings in the program configuration. This helps to streamline ISM repository actions.

RR 21.03.00-2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

Automatically generated equivalent lateral seismic loads per IBC 2015 / ASCE 7-10 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor.

Automatic load combinations using the tables from IBC 2015 can be added using the Auto Load Combination dialog (on page 2872), as well.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *D1.A.10.5 Combined Forces and Torsion* (on page 1119)
- *P. Steel AutoDrafter Workflow* (on page 2018)
- *D8.F. Indian Codes - Concrete Design per IS 13920-2016* (on page 1683)

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.1 Response Spectra per IBC 2015 / ASCE 7-10

Automatically generated response spectra loads per IBC 2015 / ASCE 7-10 are now available in STAAD.Pro.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *TR.32.10.1.12 Response Spectrum Specification per IBC 2015* (on page 2573)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.3 Static Seismic Loads per IS 1893 2016

Automatically generated equivalent lateral seismic loads per IS 1893 2016 are now available in STAAD.Pro.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load* (on page 2393)

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.3 Response Spectra Loads per IS 1893 2016

Automatically generated response spectra loads per IS 1893 2016 are now available in STAAD.Pro.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10
- *TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016* (on page 2544)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.5 Soft Story Checks per IS 1893 2016

The check soft story option may be added to a floor diaphragm command set to check for soft stories per IS1893:2016. The previous (approximate) method for checking soft stories implemented for IS 1893 2002 has been deprecated in favor of a displacement-based approach using rigid diaphragms.

Soft story checking a process by which designers check stiffness of a story with that of the story above. If the stiffness of a story is lower than that of the story above, the story is considered as a Soft story. Having a soft story in the building makes that story venerable to Earthquake. As per the revised IS 1893 Part-1:2016, if a soft story is detected, structural configuration needs to be change with the introduction of additional lateral load resisting elements in the story detected as soft story. The previous version of IS 1893 Part-1, had provisions to increase design story shear in the soft story and increase percent of steel in the lateral load resisting elements in that soft story. This provision has been withheld in the present version of the code. This makes the situation a stringent one for the designer if a story is detected as a soft story.

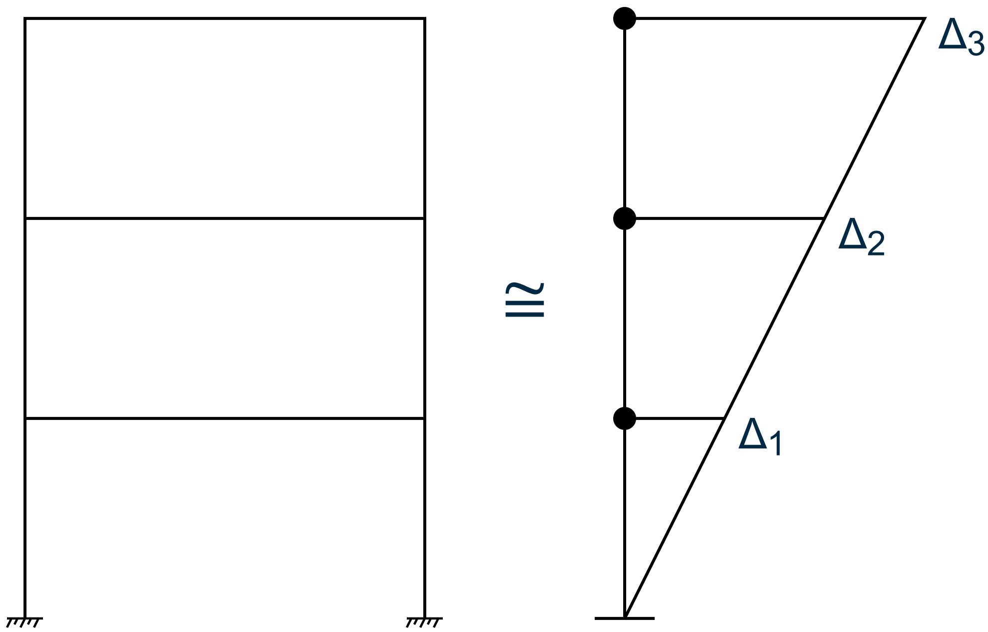
Soft Story Checking

There was an approximate methodology available in STAAD.Pro. The program used to compute story stiffness of a story by summing up the lateral stiffness of columns and shear walls (modelled using surface elements). The stiffness of a column is calculated as $12EI / L^3$ where E is the Young's modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as $Ph^3/3EI + 1.2Ph/AG$ (i.e. summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area and G is the shear modulus of the wall (E and I carry usual meaning).

This method is approximate one and works in the presence of FLOOR HEIGHT commands. On studying the literatures available on this subject, this method proved to be approximate as it fails to capture actual story stiffness including the effect of bracings. Story stiffness should be computed using a displacement based approach. STAAD.Pro is currently equipped with the facility to consider the in-plane stiffness of slabs as Rigid Diaphragm. In STAAD.Pro lateral stiffness is calculated only when the floor is modeled as rigid floor diaphragm since it functions as transferring story shears and torsional moments to lateral force-resisting members during earthquake.

Story Stiffness Computation Example

Consider a multi-story building:



The points are control nodes. The story stiffness is it defined as the inverse of inter-story drift when a *unit load* is applied at that story only.

The unit load applied is along X, Z and θY.

Consider the displacement of 2nd story. The displacements are:

Story 1: ΔX1, ΔZ1, θY1;
Story 2: ΔX2, ΔZ2, θY2

Relative Displacement for story 2 is:

RX2 = ΔX2 - ΔX1; RZ2 = ΔZ2 – ΔZ1; RθY2 = θY2 - θY1

Story Stiffness X =1/((ΔX2 – ΔX1))

Story Stiffness Z =1/((ΔZ2 – ΔZ1));

Story Stiffness θY = 1/((θY2 - θY1))

The rest is the same for all the other stories.

Note: This story stiffness is not the same as column story stiffness which is calculated as *(nCol × 12EI/L3)*, n= number of columns.

Command Input

The story stiffness can be printed using the `PRINT STORY STIFNESS` command.

If soft story check is required to be performed following commands are required to be defined immediately after rigid floor diaphragm is specified in the `FLOOR DIAPHRAGM` command.

Related Links

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.6 Design of I-Sections with Cover Plates for Torsion per AISC 360-16

STAAD.Pro can now design I section shapes with top, bottom, or both cover plates for torsion per the AISC 360-16 code.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.7 Concrete Design in Metric per ACI 318-14

STAAD.Pro will now report the concrete design results per ACI 318-14 in metric values when the input is given in metric.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *D1.F.3 Design Parameters* (on page 1201)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-3.8 Concrete Design per IS13920-2016

STAAD.Pro is now capable of performing design of concrete beams and columns per IS13920-2016.
Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 21.03.00-4.1 Steel AutoDrafter Workflow

The Steel AutoDrafter workflow is used to automatically generate construction drawings and material take-off tables for your steel framed structure.

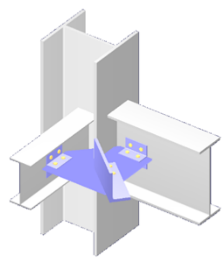
Technical Preview: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

RR 21.03.00-4.2 Horizontal Brace Connection Design in Connection Design

Connection Design is now capable of designing horizontal bracing connections to beam-beam-brace connections and column-beam-brace connections.



Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *D. To design an HBBB connection* (on page 1035)
- *D. To design an HCBB connection* (on page 1036)
- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

CONNECT Edition Update 2

The Software Release Report for STAAD.Pro CONNECT Edition Update 2 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition Update 1 (release 21.00.01). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.00.02-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.00.02-1.1Select ribbon tab

A new ribbon tab containing tools for selecting model objects has been added to several of the workflows.

Having quick access to the selection tools on a single ribbon tab makes selecting objects and then further refining the selection set easier.

The Select ribbon tab is available in multiple workflows, so you no longer have to switch to the Analytical modeling workflow to change the object selection as required in some instances in previous releases.

Related Links

- *Select tab* (on page 2769)

RR 21.00.02-1.2 New Output Viewer

The STAAD.Pro Editor is now used to view the analysis results file in the Analytical modeling workflow.

The updated editor is now used to navigate the results file (file extension `.anl`).

Related Links

- *P. To view analysis results* (on page 1978)

RR 21.00.02-2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.00.02-2.1 AISC 360-16

The program can now perform steel design of members per ANSI/AISC 360-16 *Specifications for Structural Steel Buildings*.

The seismic provisions of AISC 341-16 are also implemented.

Performance Enhancements - Design for `CODE CHECK` / `MEMBER SELECT` up to 20 times faster than AISC 360-10.

Detailed Design output - `TRACK` 0,1, & 2 reports improved to display more intermediate results.

AISC 360-16 Design

To use the 2016 edition, specify the command:

```
CODE AISC UNIFIED 2016
```

The checking or selection of members is performed per the following sections of AISC 360-16 and AISC 341-16:

- Section Classification: Chapter B
- Tension Chapter D
- Compression: Chapter E
- Flexure: Chapter F
- Shear: Chapter G
- Torsion (Including checks as per DG-9 for certain profile shapes – see below): Chapter H & Design Guide 9 for Torsion design
- Combined actions – Chapter H for:
  - Axial and/or

- Flexure and/or
- Shear and/or
- Torsion
- Checks for Seismic suitability – AISC 341-16

The following section profile shapes are allowed for design:

- Rolled Shapes
  - I-Shape
  - Channel
  - Single angle
  - Solid Circular bar
  - Solid Rectangular bar
  - Tee
  - HSS rectangular
  - HSS round
- Built Up Shapes
  - Built Up Channel
  - Built Up I
  - Double Channels – back to back
  - Double Channels – face to face
  - Double Angles
  - Built Up Box
  - I With Cover Plates (Top and/or bottom)
- Tapered Shapes
  - I with web tapered
  - Square hollow shape with wall tapered
  - Circular hollow shape diameter tapered

Note: Certain checks (e.g., DG-9 torsion checks) will only be performed for certain profile shapes (non-HSS shapes).

RR 21.00.02-2.2 ACI 318-14

The program can now design concrete columns and beams per ACI 318-14 *Building Code Requirements for Structural Concrete*

The following describes the code clauses from ACI 318-14 implemented for beams and columns.

Note: Plate elements and deep beams cannot be designed per ACI 318-14. Use an older edition of the ACI code to design these elements if needed.

Materials

- Beams

  19.2.2
  19.2.3.1

19.2.4.2

- Columns

  19.2.2
  19.2.4.2

Minimum Reinforcement

- Beams

  9.6.1.1
  9.6.1.2
  9.6.1.3
  9.7.2.1
  9.7.6.4.2
  9.7.6.4.3

- Columns

  10.6.1
  10.7.2.1
  10.7.3.1
  10.7.6.1
  10.7.6.2
  25.2.3

Clauses 9.7.6.4.2 and 9.7.6.4.3 are not included in the standard design criteria of the minimum reinforcement since the design of these criteria should consider the rebar including the bar detailing and thus they should be applied in a second design after performing the strength design and the bar detailing.

Clause 9.6.1.3 is also not included in the standard criteria since the reduction of the minimum reinforcement is optional.

Service

- Beams

  9.7.2.2
  9.7.2.3

Strength

- Beams

  9.5
  9.5.1
  9.5.2
  9.5.3
  9.5.4
  9.6.3.1
  9.6.3.3
  9.6.4.2

9.7.5.2
9.7.6.2.2
9.7.6.3.1
9.7.6.3.3
9.7.3.3
9.6.4.3
20.2.2.4a
20.2.2.1
20.2.2.2
21.2.1
22.2.2.2
22.5.5
22.2.2.3
22.7.4
22.5.1.2
22.7.6.1
22.7.7.1

- Columns

10.5
10.5.1
10.5.2
10.5.3
10.6.2
10.7.6.5.2
20.2.2.4a
20.2.2.1
20.2.2.2
22.5
22.2.2.2
22.5.5
22.2.2.3
22.7.4
22.5.1.2
22.7.6.1

Ductility

- Beams

9.3.3.1

Development Length

- Beams

25.4.1.4
25.4.2

25.4.3
- Columns

  25.4.1.4
  25.4.2
  25.4.3

Span Detailing

- Beams

  9.7.3.3
  9.7.3.8
  9.7.7

RR 21.00.02-2.3 Russian Wind Load per SP 20.13330.2016

Static and dynamic wind loads per the SP 20.13330.2016 SNiP 2.01.07-85 code can now be applied in STAAD.Pro.

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2157) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the `WIND LOAD` command.

Related Links

- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Wind Load tab* (on page 2850)
- *M. To add a SNiP wind load definition* (on page 847)
- *M. To apply a dynamic wind load per SP 20.13330.2016* (on page 856)

CONNECT Edition Update 1

The Software Release Report for STAAD.Pro CONNECT Edition Update 1 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro CONNECT Edition (release 21.00.00). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.00.01-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.00.01-1.1 Share to ProjectWise

You can now share a STAAD.Pro project directly to a ProjectWise data source. STAAD.Pro lets you select which files associated with the input (`.std`) file you want to include.

Related Links

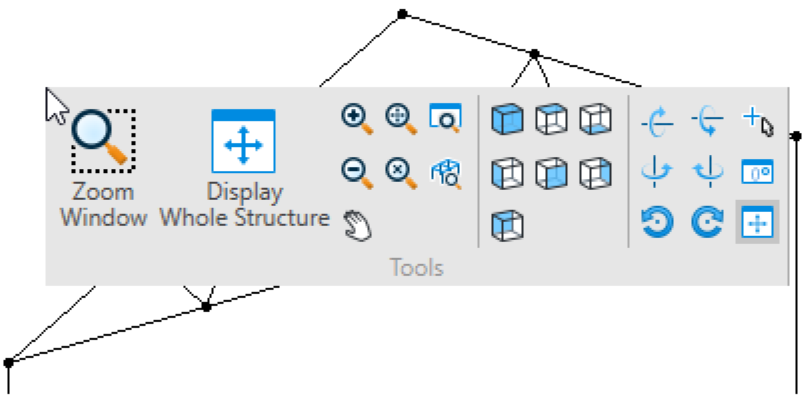
- *I. To share a STAAD.Pro project in ProjectWise* (on page 2076)

RR 21.00.01-1.2 Quick Commands

Two new sets of quick commands have been added to the graphical view window to allow for quicker model navigation and modeling.

Quick Navigation

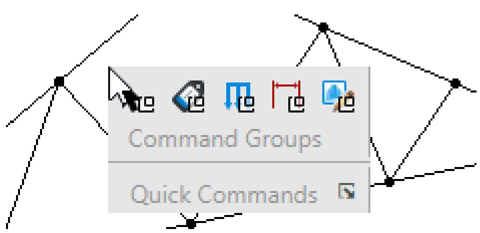
Press **<Shift>** when right-clicking in the view window to open the **View** ribbon tab **Tools** group and your mouse pointer location.



Quick Commands

Press the space bar when the view window has the application focus to open the Quick Commands menu. This customizable pop-up menu contains the following groups of tools:

- **Geometry** ribbon tab **Selection** group
- **View** ribbon tab **Labels** group
- **Loading** ribbon tab **Display** group
- **Utilities** ribbon tab **Geometry Tools** group
- **Utilities** ribbon tab **Utilities** group



You can customize the Quick Commands with any tool by clicking the icon and then adding tools from the **Customize Quick Commands Popup** dialog.

Related Links

- *GS. Right-Click View Tools menu* (on page 70)
- *GS. Quick Commands Pop-up menu* (on page 74)

RR 21.00.01-2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

RR 21.00.01-2.1 Ritz Vector Analysis with User-Defined Starting Load Vectors

You can now directly specify the starting load vectors for use with load dependent Ritz vector method for the eigen solution.

Related Links

- *TR.31.9 Defining Starting Load* (on page 2459)
- *M. To use starting vectors with load-dependent Ritz vectors* (on page 885)

RR 21.00.01-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.00.01-3.1 AISC 360-16 Technical Preview

The program can now perform steel design of members per ANSI/AISC 360-16 *Specifications for Structural Steel Buildings*.

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

Related Links

- *D1.A. American Codes - Steel Design per AISC 360 Unified Specification* (on page 1086)

CONNECT Edition

The Software Release Report for STAAD.Pro CONNECT Edition contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8*i* (SELECTseries 6) (release 20.07.11). This document should be read in conjunction with all other STAAD.Pro help files, including the Revision History document.

RR 21.00.00-1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

RR 21.00.00-1.1 New User Interface

The STAAD.Pro user interface has been updated and refined.

Ribbon Controls

The menus and toolbars have been replaced with a series of tabbed toolbars along the top of the application window. These are context sensitive to the current “workflow” as well as to the current model selection. There are fewer tools displayed at one time, so they can be larger and thus easier to identify. However, the tools you need are always available by selecting the tab (similar to menus in a tradition Windows program).

The icons for the tools are updated to reflect a consistent theme, further making the correct tool easy to identify.

Workflows Panel

The system of application modes has been replaced with the Workflows panel.

Tip: Once you become familiar with the workflows, you can hide the panel to display a larger viewing area. Simply click the Workflow tab again to select a different workflow.

Page Control Bar

The pages within each workflow have been streamlined from those in the previous modes. A single row of pages is displayed for the current workflow, just above the view window. Select the appropriate page for your current task to display the corresponding dialogs and tables.

Tip: If you close some of the dialogs for a page and need to display them again, simply click the **Restore View** tool () on the right-hand side of the page control bar.

RR 21.00.00-1.2 CONNECT Advisor

CONNECT Advisor has since been deprecated and is no longer installed with STAAD.Pro. Please refer to the CONNECTION Client application.

RR 21.00.00-1.3 64-bit Analysis Engine

The STAAD analysis and design engine now utilizes a 64-bit solver.

This allows you to analyze larger, more complex models with more objects and load cases. The analysis engine is faster for models of all sizes.

RR 21.00.00-1.4 Web Help

The product documentation is now delivered as a browser-based “Web Help.” The Web help opens in your default browser.

Refer to GS. Using Online Help (on page 25) for additional details on using Web Help.

RR 21.00.00-1.5 STAAD.Pro Script Editor

A new macro editor is available to create and edit custom scripts to extend STAAD.Pro.

You can launch the new STAAD.Pro Script Editor application by selecting the **Macro Editor** tool in the **Developer** group on the **Utilities** ribbon tab.

RR 21.00.00-2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

RR 21.00.00-2.1 STAAD.Pro Physical Modeler

You can use physical modeling concepts to generate structure geometry and reference load cases through the STAAD.Pro Physical Modeler interface.

You can launch STAAD.Pro Physical Modeler by selecting this as the modeling method when creating a new model on the Start page or by selecting the **Physical Modeling** workflow for an empty model.

Notes:

- Syncing physical model data will overwrite any model geometry created in the analytical modeling interfaces (i.e, traditional STAAD.Pro user interface or STAAD.Pro Editor).
- You cannot edit geometry, materials, reference loads, etc. created in the physical modeling interface from either the STAAD.Pro user interface or the STAAD.Pro Editor. Physical models must be edited in STAAD.Pro Physical Modeler.
- It is recommend that you begin a physical model by selecting the Physical Modeler from the New Model Wizard.

The Tutorials (on page 387) have been re-written to incorporate the physical modeling workflow. It is recommended that even experienced STAAD users work through these tutorials to familiarize themselves with STAAD.Pro Physical Modeler.

Similarly, take some time to review the **Quick Overview** and **Getting Started** sections of the STAAD.Pro Physical Modeler help as they describe many of the differences in working in STAAD.Pro Physical Modeler versus the traditional STAAD.Pro interface and STAAD input files.

RR 21.00.00-2.2 NBCC 2010 Seismic Load

The program can now generate seismic loads per the National Building Code of Canada, 2010 edition.

Both the equivalent lateral force method and seismic response spectra are now supported per the NRC 2010.

Related Links

- *TR.31.2.4 Canadian Seismic Code (NRC) - 2010* (on page 2362)
- *TR.32.10.1.3 Response Spectrum Specification per NRC 2010* (on page 2512)

RR 21.00.00-3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

RR 21.00.00-3.1 NZS3404:1997 Steel Design

The program can now perform steel design per NZS3404: 1997 *New Zealand Standard for Steel Structures*, Parts 1 & 2, including Amendments 1 & 2.

Refer to D11.A. New Zealand Codes - Steel Design per NZS 3404-1997 (on page 1765) for details on this code implementation.

RR 21.00.00-3.2 ACI 318-14 Technical Preview

The program can now design concrete columns and beams per ACI 318-14 *Building Code Requirements for Structural Concrete*.

Note: This feature was released as Technical Preview. It has been released as a commercial feature in a more recent release.

Related Links

- *D1.F.1 Design Operations* (on page 1198)

RR 21.00.00-4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the post processing and interactive design workflows. These are explained in the following pages.

RR 21.00.00-4.1 RAM Connection Workflow Update

RAM Connection CONNECT Edition is now supported in STAAD.Pro CONNECT Edition.

- Now compatible with versions of RAM Connection CONNECT Edition up through release 11.1 (CONNECT Edition Update 1).
- Connection Design per the IS800-2007(Indian) code

STAAD.Pro V8*i*

The Software Release Reports for STAAD.Pro V8*i* contain detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 03. These documents should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

V8*i* (SELECTseries 6)

The Software Release Report for STAAD.Pro V8*i* (SELECTseries 6) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 5) (release 20.07.10) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-11.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

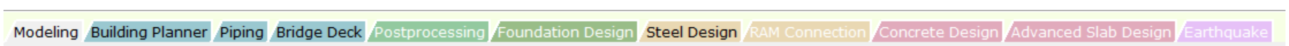
Note: Items labeled with an asterisk (*) were added or updated in the Service Pack 1 release of V8i (SELECTseries 6).

Note: Items labeled with an obelisk (i.e., dagger) (†) were added or updated in the Service Pack 2 release of V8i (SELECTseries 6).

AD.2007-11.1.1 Mode Bar Reorganization

The modes in STAAD.Pro have been reorganized to reflect a linear workflow through modeling, postprocessing, and design stages of a project.

Note: This feature has been replaced by the Workflows feature with the CONNECT Edition Edition of STAAD.Pro.



Additionally, the tabs are color coded by group:

- Blue: preprocessing and modeling
- Light green: postprocessing
- Dark green: foundation design
- Yellow: steel design
- Red: concrete design
- Magenta: earthquake checks

AD.2007-11.1.2 Advanced Analysis Engine Enhancements

The advanced analysis engine has been completely reformatted with new routines to provide even faster methods to build and solve the stiffness matrix.

Arnoldi/Lanczos Eigen Method

In addition to the standard supbspace iteration method for eigen solution, the Advanced Math Solver can use the Arnoldi/Lanczos method. For large scale eigen value problems, the Arnoldi method is very efficient.

Load Dependent Ritz Vectors

Ritz vector analysis can be used for dynamically loaded structures to more efficiently evaluate the relevant modes. It can require significantly less computational effort to evaluate a large structure when compared to natural free-vibration methods.

Autoshfiting of Eigen Vectors

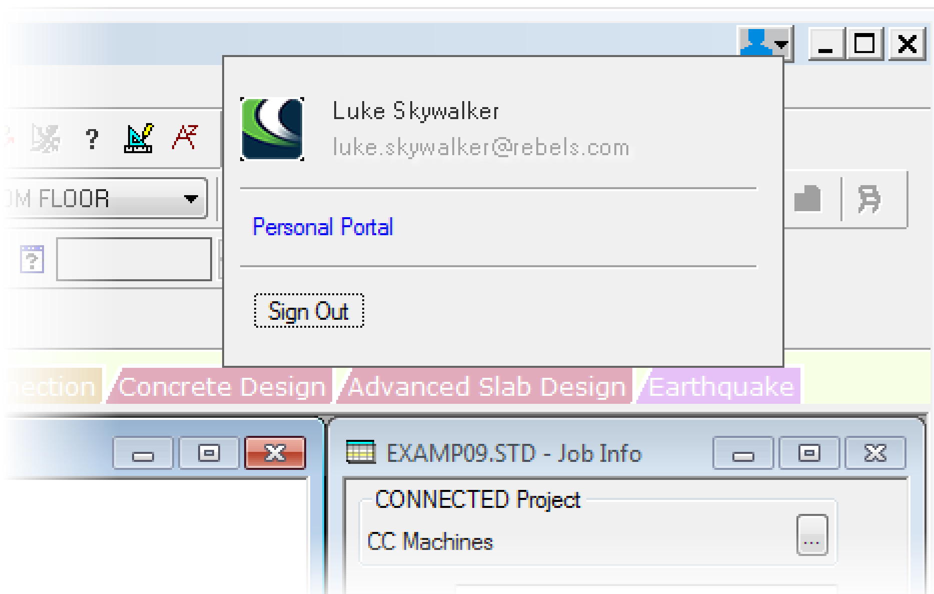
For subspace iteration and the Arnoldi/Lanczos methods, an autoshift option can be used to reduce memory demand for large structures.

Refer to G.17.3.1 Solution of the Eigenproblem (on page 2155) for details.

AD.2007-11.1.3* CONNECT Enabled

STAAD.Pro is now CONNECT Enabled. This means that you can associated your STAAD.Pro models with ProjectWise Projects from within the application to track application usage per project.

You must sign in with your Bentley CONNECT account. The Bentley CONNECTION Client is used to sign in and the status is displayed in the user icon in the top-right corner of the STAAD.Pro application window.



A new menu, **Cloud Services**, has been added which contains ProjectWise Project features, links to cloud portals, and a link to AD.2007-10.1.3* Scenario Services (on page 161) .

Once you start a new model or open an existing model that is not yet associated with a ProjectWise Project, you will be prompted to assign a ProjectWise Project to your model. A list of all registered projects in your organization is displayed. You can also register new projects (Only users with Admin/Co-admin roles can register a project). STAAD.Pro will display the ProjectWise Project name in the **Job Info** dialog as well as in the window title bar.

Tip: As of CONNECT Edition, the **Cloud Services** tools can be found on the **File** ribbon tab in the Backstage.

For additional information on CONNECTED Projects, please see http://www.bentley.com/en-US/Promo/CONNECT/why+connect.htm.

Refer to Cloud Services tab (on page 2703) for details.

AD.2007-11.1.4† RAM Connection CONNECT Edition v11 Support

STAAD.Pro now supports RAM Connection CONNECT Edition (v11.0) and in the Connection Design workflow.

- STAAD.Pro is now compatible with versions of RAM Connection CONNECT Edition up through release 11.0. The following design codes are supported:
  - AISC 360-10 (LRFD and ASD)
  - GB 50017-2003 (China)
  - EN 1993-1-8 (Eurocode 3)
- You can D. To Export Connection Designs to a Report (on page 1037) directly to a Microsoft® Office Word® document from the STAAD.Pro interface.

AD.2007-11.1.5† Connection Tags Enhancements

You can now assign user-defined equations for checking connection tags in STAAD.Pro. The XML Schema for the connection tag files has been updated. See " D. Connection Tags XML File Schema (on page 1052) " in the User Interface help for details on the schema, as well as on using wildcards and created user-defined equations.

AD.2007-11.1.6† Add Member Enhancements

The Add Member functionality in the graphical user interface has been enhanced to facility the saving, loading, and managing of user-defined attribute sets.

Select **Geometry** > **Add Beam** > **Set New Member Attributes** to open the Define Member Attributes dialog (on page 2729). Here you can define the default attributes for various predefined attribute sets, as well as create user-defined attribute sets.

AD.2007-11.1.7† ISM Integration

STAAD.Pro is now capable of transferring data to and from v5 of Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

ISM v5 includes support information, ProjectWise integration, and substructures. Refer to the ISM documentation for additional details.

AD.2007-11.2 Features Affecting the Preprocessor

This section describes features that have been added that affect the preprocessor section of the program, also known as the Modeling Mode.

AD.2007-11.2.1 Building Planner

This modeling tool allows you to rapidly generate concrete building structure models for analysis and design in STAAD.Pro.

PlanWin and FrameWin have now been directly integrated into the STAAD.Pro user interface as the **Building Planner** mode.

The Building Planner workflow is included in SELECT or in a Structural Enterprise License.

Refer to M. Building Planner workflow (on page 920) for details.

AD.2007-11.2.2 STAAD Editor

A new STAAD editor has been added which has advanced features for manually editing your STAAD input files, including IntelliSense.

Model Navigation

Associated commands are "blocked" together in the editor such that can be managed easier. Larger blocks can be collapsed to show only the first line/command, making it easier to navigate through large models.

A document outline of the model is also in a separate pane. You can double-click any entry in the outline to jump to that point in the input file.

IntelliSense

Intelligent code completion is now integrated into the STAAD editor. This allows context-aware code completion suggestions when you type in the editor.

Hover your mouse pointer over the suggested keywords to view tool tips for each.

Section profiles in standard tables are also linked to the editor. When entering a table shape, simply select it from the list of shapes in the drop-down list.

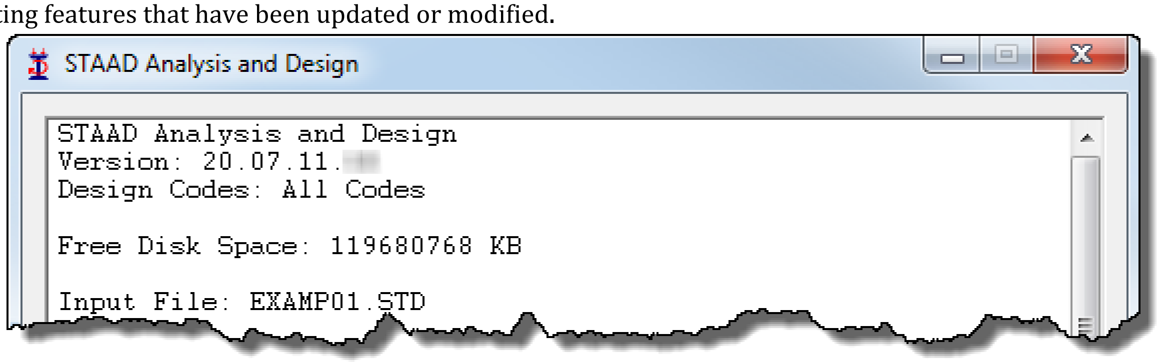
Enhanced Help

Press **<F1>** to view the help topic related to the command keyword from the Technical Reference help.

Refer to the I. STAAD.Pro Editor (on page 2043) help for details.

AD.2007-11.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.



Note: Items labeled with an obelisk (i.e., dagger) (†) were added or updated in the Service Pack 2 release of V8i (SELECTseries 6).

Note: Items labeled with a double obelisk (i.e., dagger) (††) were added or updated in the Service Pack 3 release of V8i (SELECTseries 6).

AD.2007-11.3.1 ACI 318-11 Concrete Design

Concrete design per the 2011 edition of the ACI 318 specification (ACI 318-11) is now available in STAAD.Pro.

Several new parameters have been introduced to accommodate design of slender columns using the moment magnification procedure for sway and non-sway frames.

Refer to D1.F.5.6.1 Slenderness Effects and Analysis Consideration (on page 1243) for details.

To design concrete columns per ACI 318-11

The general outline of this procedure applies to designing all concrete members per ACI 318-11, but the specific parameters mentioned pertain only to the design of columns using the moment magnification method.

1. In the Modeling mode, select the **Design | Concrete** tab.

   The **Concrete Design - Whole Structure** dialog opens.
2. In the **Current Code** drop-down list, select **ACI 318-11**.

   This will insert the following commands into the STAAD.Pro input file:
   ```
   CODE ACI
   ```
3. Click **Define Parameters**.

   The **Design Parameters** dialog opens.
4. Select the **MMAG** parameter and type **0** for the option and then click **Add**.

   This instructs the program to use the detailed calculations considering sway or non-sway frames. .
5. Select the SWY parameter and select the appropriate bracing condition for the frame containing the columns to be designed and then click **Add**.
6. (Optional) Set values for BDY, BDZ, SKY, SKZ, SLY, SLZ, SQY, SQZ, SWY, and TRN parameters as necessary (i.e., when the defaults to not match the column conditions).
7. Close the Design Parameters dialog and assign the parameters to the appropriate members.

> **Tip:** The parameters mentioned here are only applicable to columns. They will be ignored for beam members.

AD.2007-11.3.2 CAN/CSA S16-14 Steel Design

Steel design per the 2014 edition of the Canadian S16 code, *Limit States Design of Steel Structures*, is now available.

The performance of designs per CAN/CSA S16-09 has also been improved. The changes in code checks from the 09 and 14 edition of the codes are noted in D4.E.6.1 Members Subject to Axial Forces (on page 1450).

To specify a design using CAN/CSA-S16-14

1. Select the **Design | Steel** page.

   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** list, select the **CAN/CSA-S16-14**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.

   > **Note:** Refer to the descriptions in the dialog or to D4.E.7 Design Parameters (on page 1458) for definitions of the design parameters and default values used.
5. Click **Add**.

   This will insert the following commands into the STAAD input file:
   ```
   CODE CANADIAN
   ```

   > **Tip:** To use the older edition of the Canadian steel design code, the command `CODE CANADIAN 2009` is required.

AD.2007-11.3.3 SANS 10162-1:2011 Steel Design

Steel design per the South African SANS 10162-1:2011 code is now available.

Specifying a design using SANS 10162-1:2011

1. Select the **Design | Steel** page.

   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** list, select the **SANS10162-1:2011**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.

   Note: Refer to the descriptions in the dialog or to D14.C.7 Design Parameters (on page 1974) for definitions of the design parameters and default values used.
5. Click **Add**.

   This will insert the following commands into the STAAD input file:

   ```
   CODE SANS10162-1: 2011
   ```

   Tip: Alternately, `SANS10162` will also use the latest edition. To use the older edition of the South African steel design code, `SANS10162-1: 1993` is required.

Refer to D14.C. South African Codes - Steel Design Per SANS 10162-1:2011 (on page 1966) for details.

AD.2007-11.3.4 SP 63.13330-2012 Concrete Design

Concrete design per the Russian code SP 63.13330-2012 is now supported.

Refer to D13.D. Russian Codes - Concrete Design Per SP 63.1330.2012 (on page 1918) help for details.

AD.2007-11.3.5 AISC 360-05/10 Metric Steel Design

Steel design in metric per the AISC 360-05/10 code is now supported.

When you select metric units in STAAD.Pro, the full user interface (including section properties, results, etc.) displays the correct metric units for AISC 360-05/10 steel design. Additionally, the results are also fully in metric units.

AD.2007-11.3.6 AISC 360-05/10 Tapered Member Design

The design of tapered square and round tube shapes as well as tapered I-shaped members is now available per the AISC 360-10 code.

The design of tapered members was previously available under the AISC 360-05 code, but the performance of design per this code has also been enhanced.

Refer to D1.A.5.7 Design of Web-Tapered Members (on page 1098) for details.

AD.2007-11.3.7 AISC 341-05/10 Seismic Provision Checks

Seismic provisions of AISC 341-05/10 can now be checked as part of an AISC 360-05/10 member design.

Several additional parameters have been added to the AISC 360 unified codes (both 2005 and 2010 editions) to check the additional seismic provisions of AISC 341.

Refer to D1.A.9 Seismic Provision Checking per AISC 341 (on page 1112) for details.

AD.2007-11.3.8 IBC 2012 / ASCE 7-10 Seismic Loads

Automatically generated equivalent lateral seismic loads and response spectra per IBC 2012 / ASCE 7-10 are now available in STAAD.Pro.

The load parameters are assigned using the User Interface or can be manually entered in the STAAD editor.

Automatic load combinations using the tables from IBC 2012 can be added using the Auto Load Combination dialog (on page 2872), as well.

To specify an IBC 2012/ASCE 7-10 static seismic load

1. Either:

   Select **Commands** > **Loading** > **Definitions** > **Seismic Load** > **IBC 2012 Load**

   or

   Select the **Definitions** section in the **Load & Definitions** dialog and then click **Add.**

   The **Create New Definitions / Load Cases / Load Items** dialog opens with the Add New Seismic Definitions dialog (on page 2895) tab selected.

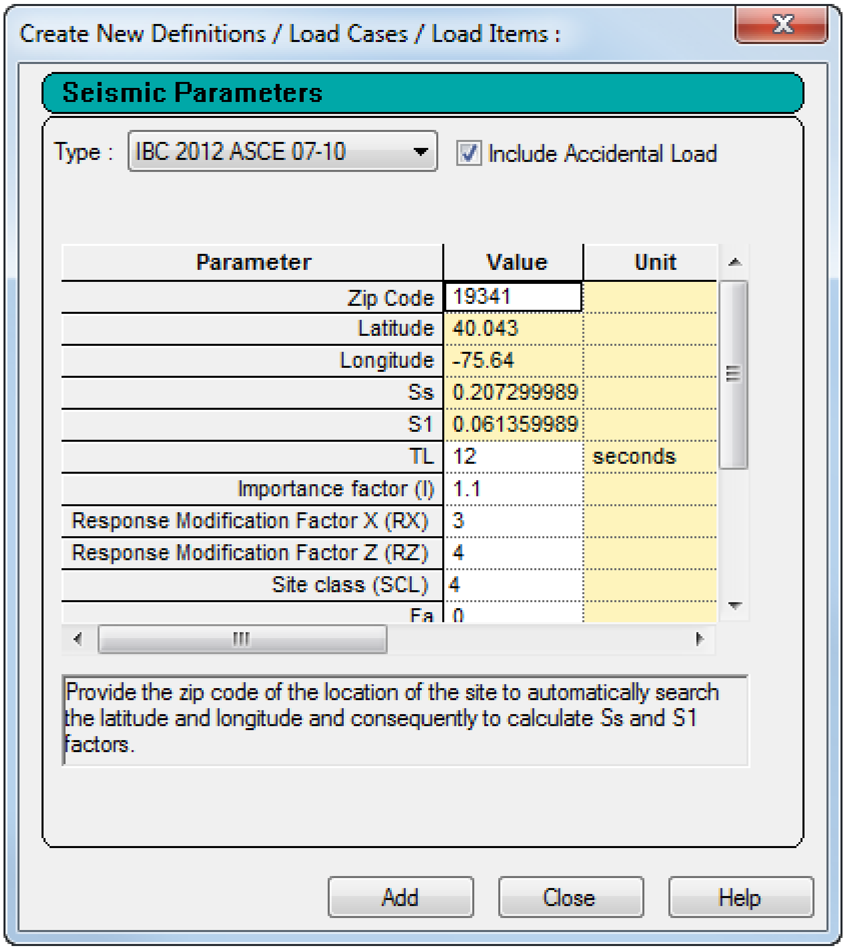


Figure 8: The Seismic Parameters as a stand-alone dialog

2. Select **IBC 2012 ASCE 7-10** in the **Type** drop-down list.
3. (Optional) Select the **Include Accidental Load** checkbox to consider accidental torsion.
4. Either:

   Type a **ZIP code**

   or

   Type **Latitude** and **Longitude** values

   or

   Type **Ss** and **S1** mapped values.
5. Specify the required seismic load parameters:
   a. Type long-period transition time, **TL**, value in the indicated units.
   b. Type an **Importance factor (I)** value.
   c. Type **Response Modification Factor X (RX)** and **Response Modification Factor Z (RZ)** values, respectively.
   d. Select the **Site class (SCL)** from the drop-down list.
6. (Optional) Specify the following optional seismic parameters as necessary:

a. Type short-period site coefficient, **Fa**, and long-period site coefficient, **Fv**, values, respectively.

> **Note:** These values are required if **Site class (SCL)** of **6** is selected (corresponds to IBC 2012 site class of F).

b. Type a **CT** value used to calculate the time period.
c. Type the **Period in X Direction (PX)** and **Period in Z Direction (PZ)** values, respectively. Otherwise these values are calculated from the code.
d. Type an exponent value, **x**, for use in ASCE 7-10 equation 12.8-7.

7. Click **Add**.

   The IBC 2012 parameters are added to the **Seismic Definitions** section of the **Load & Definition** dialog.

Refer to TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409) for details.

AD.2007-11.3.9 Eurocode 3 Steel Grades

SGR values now are aligned with Table 3.1: EN 1993-1-1: 2005.

`SGR` 3, 4 have been changed to represent S450 and S275 N/NL grades respectively.

Related Links

- *D5.C.6 Design Parameters* (on page 1502)

AD.2007-11.3.10 AIJ 2002 and 2005 New Design Parameters

The following design parameters have been added or updated in the AIJ 2002 and 2005 design codes in STAAD.Pro.

- The `MISES` parameter now has an option to ignore torsion in the calculation of the Von Mises stresses.
- A new parameter, `MBG`, has been added to ignore the web of H-shape, I-shape, and channel sections in calculating the section modulus about the local Z axis.
- A new parameter, `YNG`, has been added to the AIJ 2002 implementation to instruct the design engine how to evaluate Young's Modulus, E.

Refer to the codes in D9. Japanese Codes (on page 1713) help for details.

AD.2007-11.3.11 Steel Design Code Performance Improvements

Performance of steel design per CSA S16-S09 and IS800:2007 codes has been improved.

AD.2007-11.3.12† NRC 2005 Seismic Loads Updates and Additions

Seismic loads per the National Building Code of Canada's NRC 2005 have been updated and expanded to include torsion in the static load equivalent method as well as response spectrum loading.

AD.2007-11.3.13† IS 800 Design of Additional Steel Shapes

The design of both double-channel face-to-face and I-sections with cover plates is now available per the IS 800 2007 specification.

Refer to “ TR.20.1 Assigning Properties from Steel Tables (on page 2257) ” for details on assigning double-channel front-to-front sections (FR) and I-sections with top and bottom plates (TB).

AD.2007-11.3.14† AISC 360 Design of I-Sections with Cover Plates

The design of I-sections with cover plates is now available per the AISC 360 2005 and 2010 specifications.

Refer to " TR.20.1 Assigning Properties from Steel Tables (on page 2257) " for details on the assumptions and limit states used in the design of these sections.

Refer to " TR.20.1 Assigning Properties from Steel Tables (on page 2257) " for details on assigning I-sections with top and bottom plates (TB).

AD.2007-11.3.15†† IS 800 Design Updates

The following additions and enhancements were added to steel design per the IS 800 2007 specification:

- Flexural-torsional buckling of single angles per Cl. 7.5.1.2.
- Minimum web thickness is checked against the serviceability requirements in Cl. 8.6.1.1 and the compression flange buckling requirement of Cl. 8.6.1.2.
- The various sections of Amendment-1 to the IS:800-2007 Code, dated January 2012, are implemented in STAAD.Pro as applicable.

AD.2007-11.3.16†† Floor and One-Way Load Panel Information Printing

The program can now include floor load and one-way load panel information in the output (.ANL) file or in a external text file.

Refer to the Floor Load tab (on page 2845) in the User Interface help for information on using this feature through the graphical user interface.

Refer to TR.32.4 Area, One-way, and Floor Load Specifications (on page 2475) for information on the command input for this feature.

V8*i* (SELECTseries 5)

The Software Release Report for STAAD.Pro V8*i* (SELECTseries 5) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 4) (release 20.07.09) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-10.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

Note: Items labeled with an asterisk (*) were added or updated in the QA&R release of V8i (SELECTseries 5) (Build 20.07.10.XX).

AD.2007-10.1.1 Bentley Trust Licensing

STAAD.Pro now utilizes Benltey's Trust Licensing program.

This allows you or your organization the flexibility to utilize the program as needed without interruption, regardless of the availability of licenses. The usage data is collected periodically for billing at a later time for your convenience.

Refer to the *Getting Started & Troubleshooting Guide* (installed with STAAD.Pro) for additional information on license setup for STAAD.Pro.

Visit Bentley.com for more information on Bentley's licensing solutions.

AD.2007-10.1.2 Brazilian Steel Databases

Brazilian steel section databases have been added to the section profile tables included in STAAD.Pro.

To select a Brazilian steel profile shape

1. Select the **General | Property** page.

   The **Properties - Whole Structure** dialog opens.
2. Click **Section Database** on the **Properties - Whole Structure** dialog.

   The **Section Profile Table** dialog opens.
3. On the Steel tab, expand the **Brazilian** entry to select a shape class.
4. Select a shape and specify any shape parameters in the dialog.
5. Click **Add**.

   The section is added to the list on the **Properties - Whole Structure** dialog Section tab.

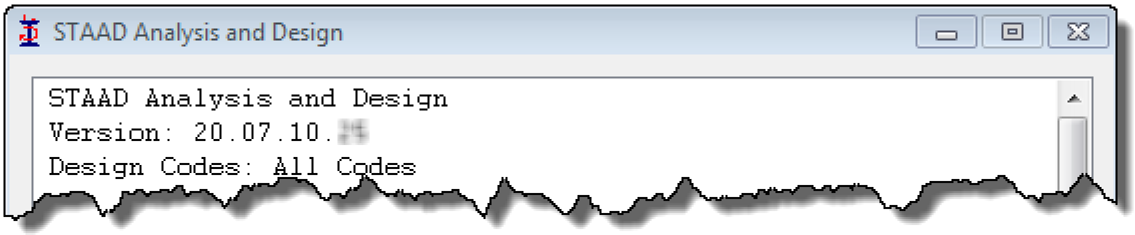
AD.2007-10.1.3* Scenario Services

STAAD.Pro projects can be run in Bentley's Scenario Services (formerly ProjectWise Scenario Services or Bentley CONNECT Scenario Services) server with a SELECT account. This allows you to run large models or complex, nonlinear analyses in the “cloud,” leveraging extensive computing power. Additionally, STAAD.Pro models can be added Multiple Discipline Optimization (MDO) projects.

Refer to Cloud Services tab (on page 2703) for additional information on using this feature.

AD.2007-10.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.



Note: Items labeled with an asterisk (*) were added or updated in the QA&R release of V8i (SELECTseries 5) (Build 20.07.10.XX).

AD.2007-10.2.1 Advanced Cable Analysis

An advanced, nonlinear cable analysis has been added to STAAD.Pro. The nonlinear nature of the static solver can utilize either the Newton-Rapshon method or a modified Newton-Raphson method for reduced computational effort. The cables are considered as geometrically nonlinear.

Refer to the following procedures:

- M. To add a member specification (on page 802)
- A. To specify a nonlinear cable analysis (on page 955)
- A. To specify post-analysis print commands (on page 963)

Refer to G.17.2.8 Advanced Nonlinear Cable Analysis (on page 2150) for details on the analysis methodology used by STAAD.Pro.

AD.2007-10.2.2 Colombian Seismic Code

Static lateral forces for seismic loads based on *Reglamento Colombiano Sismo Resistente* (NSR-10) (2010 edition of Colombian seismic code)

Note: Refer to TR.31.2.8 Colombian NSR-10 Seismic Load (on page 2384) for details on the command parameters.

AD.2007-10.2.3 Canadian Steel Code Update

Steel design CAN/CSA-S16-09, *Design of Steel Structures*, is now available in STAAD.Pro.

Refer to D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19 (on page 1447) for additional information.

To specify a design using CAN/CSA-S16-09

1. Select the **Design | Steel** page.

   The **Steel Design - Whole Structure** dialog opens.

2. In the **Current Code** list, select the **CAN/CSA-S16-09**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.

   Note: Refer to the descriptions in the dialog or to D4.E.7 Design Parameters (on page 1458) for definitions of the design parameters and default values used.
5. Click **Add**.

   This will insert the following commands into the STAAD input file:

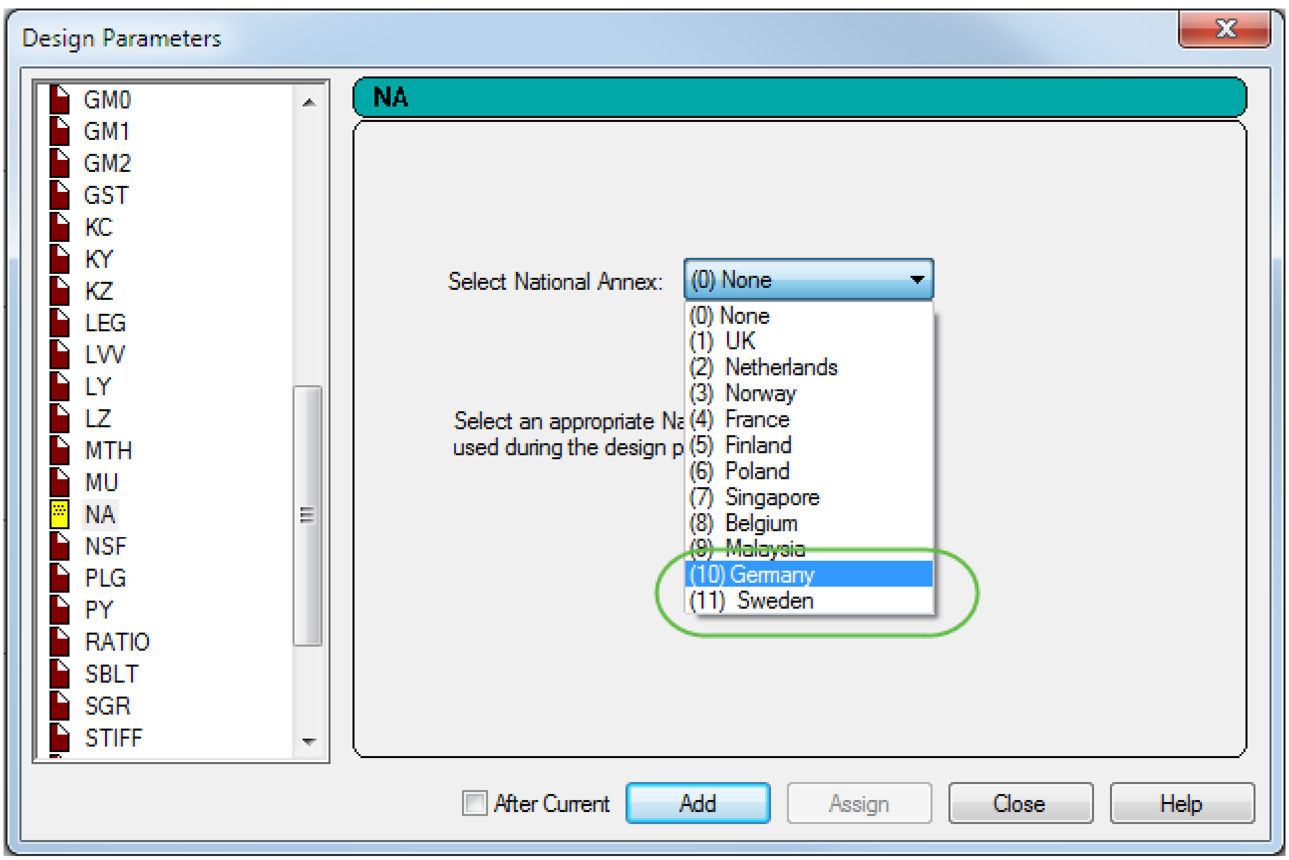
```
CODE CANADIAN
```

   Tip: To use the older edition of the Canadian steel design code, the command `CODE CANADIAN 2001` is required.

AD.2007-10.2.4 Eurocode 3 National Annexes

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Germany and Sweden.

As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.



AD.2007-10.2.4.1 German National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the German NA to EC3, use the following procedure.

The German National Annex document referred to is “DIN EN 1993-1-1:2005.”

When the German National Annex to EC3 is used for design, the output section title is revised to include the German National Annex (National Annex to DIN EN 1993-1-1:2005). Additionally, the partial safety factors used are included in the output and are specified in the German NA.

For additional information, please refer to D5.D.12 German National Annex to EC3 (on page 1575) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) .

1. In the Modeling mode, select the **Design | Steel** tab.
   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** drop-down list, select **EN 1993-1-1:2005**.
3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the **(10) Germany** in the **Select National Annex** drop-down list.
6. Click **Add**.

   This will insert the following commands into the STAAD.Pro input file:

```
CODE EN 1993-1-1:2005
NA 10
```

AD.2007-10.2.4.2 Swedish National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Swedish NA to EC3, use the following procedure.

The Swedish National Annex document referred to is “BFS EN 1993-1-1:2005.”

When the Swedish National Annex to EC3 is used for design, the output section title is revised to include the Swedish National Annex (National Annex to EN 1993-1-1:2005). Additionally, the partial safety factors used are included in the output and are specified in the Swedish NA.

For additional information, please refer to D5.D.13 Swedish National Annex to EC3 (on page 1579) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) .

1. In the Modeling mode, select the **Design | Steel** tab.
   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** drop-down list, select **EN 1993-1-1:2005**.
3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the **(11) Sweden** in the **Select National Annex** drop-down list.
6. Click **Add**.

   This will insert the following commands into the STAAD.Pro input file:

```
CODE EN 1993-1-1:2005
NA 11
```

AD.2007-10.2.5 AISC 360-10 Torsion Design

STAAD.Pro can now design open shapes for stresses due to torsion per AISC Design Guide #9, *Torsional Analysis of Structural Steel Members*.

A new `TORSION` parameter for the AISC 360-10 steel design code which can be set to instruct the program to perform torsion checks per this guide.

Refer to D1.A.5.6 Design for Torsion (on page 1096) for details on the checks performed.

To include torsion checks per Design Guide #9

1. In the Modeling mode, select the **Design | Steel** tab.

   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** drop-down list, select **AISC 360-10**.
3. Click **Define Parameters**.

   The **Design Parameters** dialog opens.
4. Select the **TORSION** parameter and select **1** for the option.
5. (Optional) Select the **TRACK** parameter and select **3** for the option.

   This will include detailed output for the Design Guide #9 checks in the output.
6. Click **Add**.

   This will insert the following commands into the STAAD.Pro input file:

   ```
   CODE AISC UNIFIED 2010
   ```

   ```
   TORSION 1
   ```

   ```
   TRACK 3
   ```
7. For the `TORSION` parameter (and for the optional `TRACK` parameter), you must specify a member list using the assignment tools in the **Steel Design - Whole Structure** dialog.

AD.2007-10.2.6 Missing Mass

Missing mass can now be specified for time history load definitions.

Missing mass and rigid body modes are reported in the output and can be selected in the post-processing mode drop-down (* for mm & + for rb)

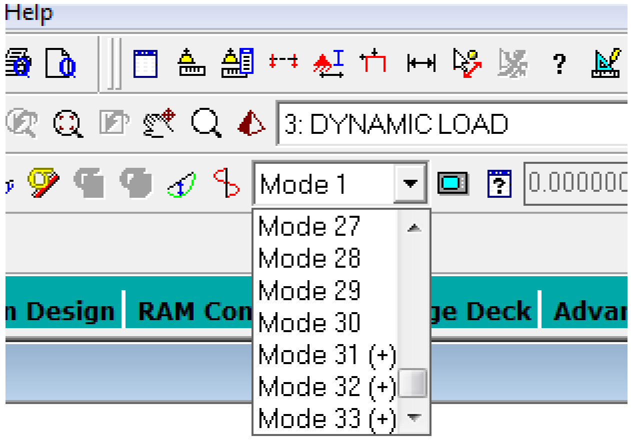


Figure 9: Missing mass modes are marked with an (*) in the Modes drop-down list

Rigid body modes are used internally to establish relative and absolute structure displacement, particularly for structures supported on relatively weak springs.

Refer to TR.31.4 Definition of Time History Load (on page 2441) and TR.34.2 Modal Calculation Command (on page 2615) for additional information.

AD.2007-10.2.7* Wind Loads per Russian Design Code SP 20.13330.2011

Refer to TR.31.3 Definition of Wind Load (on page 2435) for additional information.

AD.2007-10.2.8* Response Spectrum Specification per SP 14.13330.2011

Refer to TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011 (on page 2589) for additional information.

AD.2007-10.2.9* Steel Design per Russian Design Code SP 16.13330.2011

Steel member design per SP 16.13330.2011, *Steel Structures*, is now available in STAAD.Pro.

Refer to D13.C. Russian Codes - Steel Design Per SP 16.13330.2011 (on page 1905) for additional information.

Specifying a design using SP 16.13330.2011

1. Select the **Design | Steel** page.

   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** list, select the **SP 16.13330.2011**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.

> **Note:** Refer to the descriptions in the dialog or to D13.C.4 Design Parameters (on page 1910) for definitions of the design parameters and default values used.

5. Click **Add**.

   This will insert the following commands into the STAAD input file:

   ```
   CODE RUSSIAN
   ```

   > **Tip:** To use the older edition of the Russian steel design code, the command `CODE RUSSIAN 1990` is required.

6. Assign the code parameters to members as necessary.

AD.2007-10.3 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

> **Note:** Items labeled with an asterisk (*) were added or updated in the QA&R release of V8i (SELECTseries 5) (Build 20.07.10.XX).

AD.2007-10.3.1 RAM Connection Mode Update

RAM Connection V8i features are now supported in STAAD.Pro V8i (SELECTseries 5) RAM Connection mode.

- Now compatible with versions of RAM Connection V8i up through release 8.0 (SELECTseries 4).
- New templates:
  - US: MEP Knee BCF, Moment End Plate BCF, Moment End Plate BCW, Moment End Plate BS Apex, and Moment End Plate BS
  - UK: Bolted End Plate BS Apex
- New sections added to the RAM Connection database for connection design:
  - Canadian WS, MS, S, HP, Angel, Channel, MC Channel, and HSS Round shapes
  - Brazilian I, Angle, and Channel shapes

AD.2007-10.3.2* CAN/CSA-A23.3-10 in RC Designer

> **Note:** The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

V8*i* (SELECTseries 4)

The Software Release Report for STAAD.Pro V8*i* (SELECTseries 4) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-09.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

Note: Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 4) (Build 20.07.09.22).

AD.2007-09.1.1 I-Section with Different Flange Shapes

User-provided steel table Wide Flange sections can now have different flange sizes on top and bottom.

Previously, the sections were required to be doubly symmetric. Now, you may specify a different bottom flange size.

To add a UPT I-Section with different flange sizes

1. (Optional) Select **Tools** > **Select Input Units**. Select appropriate units for length and click **OK**.
2. Select **Tools** > **Create User Table**.

   If no User Defined Table exists, you will be prompted to create one.

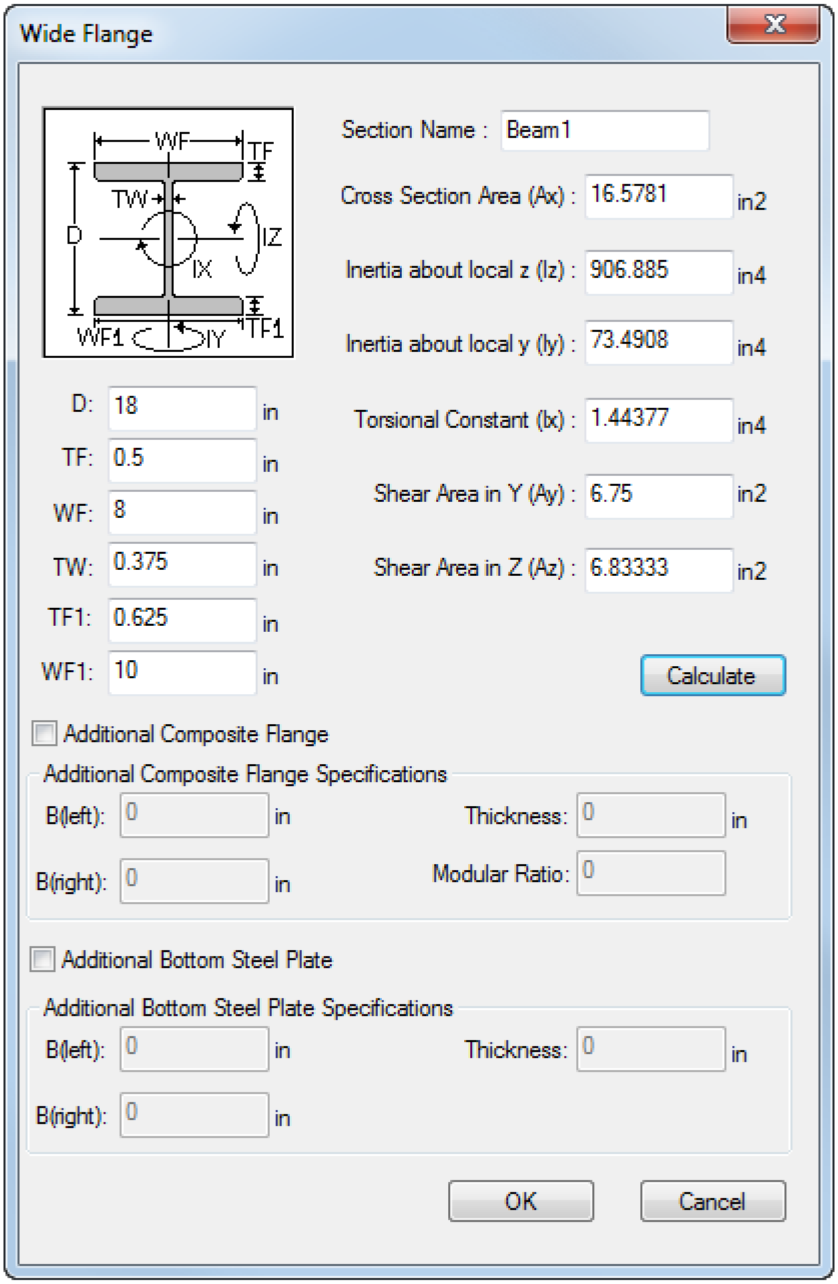
   The **Create User Provided Table** dialog opens.
3. Click **New Table**.

   The **Select Section Type** dialog opens.
4. In the **Select Section Type** list, select **WIDE FLANGE** and then click **OK**.

   Tip: You can also use this dialog to save the User Provided Table as a separate file.

   The dialog closes and the **Select Existing Table** list in the **Create User Provided Table** dialog now has at least one entry.
5. Click **Add New Property**.

   The **Wide Flange** dialog opens.



6. Type dimensions in the **D**, **TF**, **WF**, **TW**, **TF1**, and, **WF1** fields to define the section.

   Two new fields— **TF1** and **WF1** —are used to specify different dimensions for the bottom flange. If these fields are left empty, the values for the top flange are used (doubly symmetric section).

   Note: Refer to TR.19.1 Wide Flange (on page 2244) for detailed descriptions on properties of Wide Flange sections in User Steel tables.

7. Click **Calculate**.

   The derivative cross section properties are calculated.

8. Click **OK**.

The dialog closes and the section is added to the **Table Data** list in the **Create User Provided Table** dialog.

AD.2007-09.1.2 I-Section with Flange Plates or Composite Slab

User-provided steel table Wide Flange sections can now have an additional bottom flange plate or composite concrete on the top flange.

These options are available in the Wide Flange dialog in the user interface. The additional flange plates or composite slab data is then used along with the base section properties.

To add a UPT I-Section with flange plates and composite slab

1. (Optional) Select **Tools** > **Select Input Units.** Select appropriate units for length and click **OK**.
2. Select **Tools** > **Create User Table**.

   If no User Defined Table exists, you will be prompted to create one.

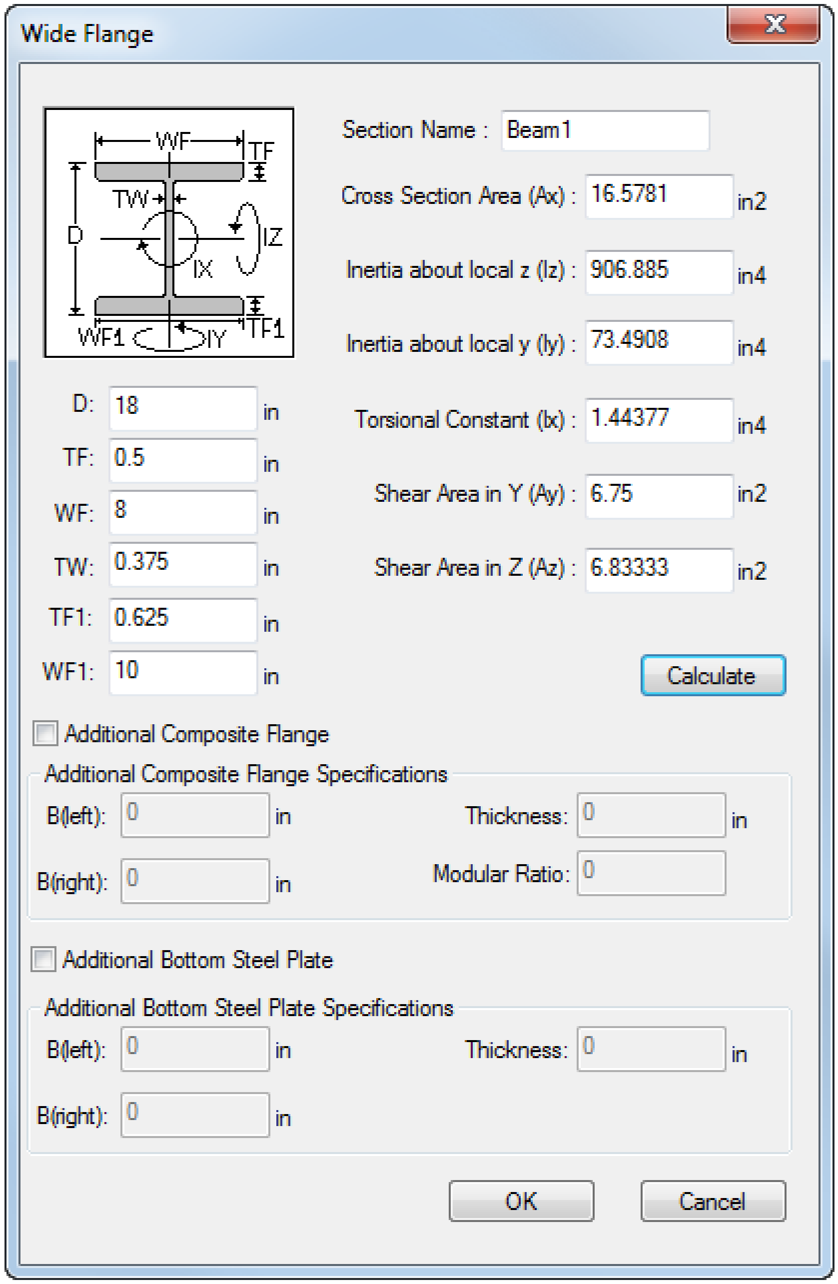
   The **Create User Provided Table** dialog opens.
3. Click **New Table**.

   The **Select Section Type** dialog opens.
4. In the **Select Section Type** list, select **WIDE FLANGE** and then click **OK**.

   Tip: You can also use this dialog to save the User Provided Table as a separate file.

   The dialog closes and the **Select Existing Table** list in the **Create User Provided Table** dialog now has at least one entry.
5. Click **Add New Property**.

   The **Wide Flange** dialog opens.



6. Add a composite flange:
   a. Check the option for **Additional Composite Flange**.
   b. Type dimensions in the **B(letf)**, **B(right)**, **Thickness**, and, **Modular Ratio** fields to define the composite slab.

Note: Refer to TR.19.1 Wide Flange (on page 2244) for detailed descriptions on properties of Wide Flange sections in User Steel tables.

7. To add a bottom cover plate, select the **Additional Bottom Steel Plate** option and type values for **B(left)**, **B(right)**, and **Thickness**.

Tip: Top cover plates cannot be added in association with a composite slab, only bottom cover plates.

8. Click **Calculate**.

   The derivative cross section properties are calculated.
9. Click **OK**.

   The dialog closes and the section is added to the **Table Data** list in the **Create User Provided Table** dialog.

AD.2007-09.1.3 ISM Integration

STAAD.Pro is now capable of transferring data to and from v3 of Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

ISM v3 includes support for foundation information. STAAD Foundation Advanced is now ISM Enabled, so this allows for an improved method for transferring and updating support information for STAAD Foundation Advanced, as well as other ISM Enabled products using v3 of ISM, such as detailing products like ProConcrete.

AD.2007-09.1.4* Connection Tags

This feature is used to both create connection data within STAAD.Pro and to transfer it to other programs via the ISM link. This data can then be used in third-party programs, such as Tekla Structures®. The connections can then be checked against a defined capacity.

Connection tags are assigned and checked from within the STAAD.Pro User Interface by use of the right-click pop-up menu. When a beam member is selected (using the beam cursor), a sub-menu for Connection Tags is provided with tools for using connection tags.

Note: Refer to D. Connection Tags sub menu (on page 1066) for additional information on using this feature.

Connection tags consist of two pieces of data:

i. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to D. Connection Tags XML File Schema (on page 1052) for additional information on the required structure of this XML file.
ii. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the `DEFINE MEMBER ATTRIBUTE` command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names in the associated XML file. Refer to TR.29.2 Connection Tag Member Attribute (on page 2341) for additional information on this command.

AD.2007-09.2 Features Affecting the Pre-Processor

This section describes features that have been added that affect the pre-processor section of the program, also known as the Modeling Mode.

AD.2007-09.2.1 Print Center of Rigidity

The `PRINT DIAPHRAGM CR` command may be used to obtain a print-out of the center of rigidity and center of mass at each rigid diaphragm in the model.

The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

Related Links

- *TR.42 Print Specifications* (on page 2666)
- *A. To output the center of rigidity* (on page 964)

AD.2007-09.2.2 Load & Definition

A new feature has been added to include horizontal torsion for rigid floor diaphragms in the equivalent static seismic analysis. This torsion —referred to as the natural torsion— accounts for the static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm, to be used to multiply the UBC, IBC, 1893, etc.

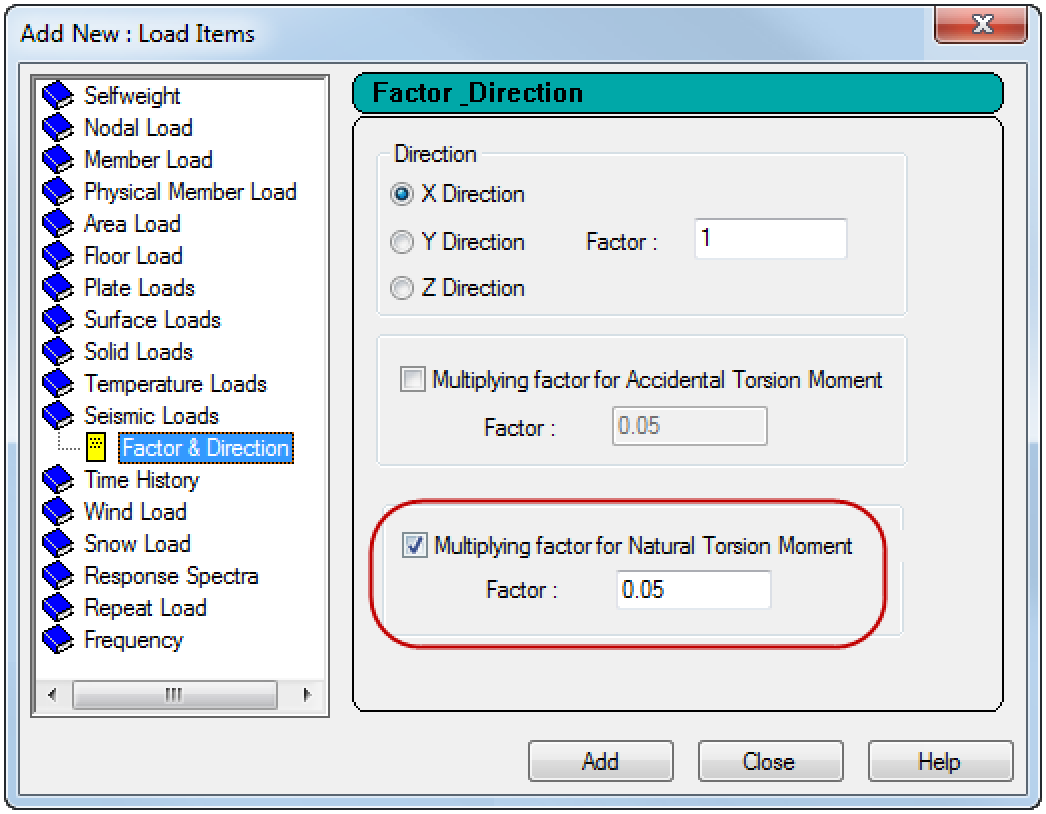
Refer to TR.32.12.2 Generation of Seismic Loads (on page 2598) for additional details.

Adding Natural Torsion factor to a static seismic load

The natural torsion factor is specified along with the accidental torsion factor.

1. Add a Seismic Load definition to your model.
2. Add a Load Case with the Loading Type of **Seismic** to your model.
3. Select the seismic load case in the **Load & Definition** dialog and click **Add**.

   The **Add New: Load Items** dialog opens.
4. Select the Seismic Loads tab.
5. Set the option to use the **Multiplying factor for Natural Torsion Moment** and type a value (less than or equal to one).



6. Select the direction and specify the accidental torsion option as necessary.
7. Click **Add**.

Related Links

- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *M. To add a seismic load* (on page 860)

AD.2007-09.2.3 Check for Soft Story

The `CHECK SOFT STORY` option may added to a `FLOOR DIAPHRAGM` command set to check for soft stories per either IS1893:2002 or ASCE7-95 codes.

Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command `PRINT STORY STIFFNESS` in the post analysis print commands section.

Related Links

- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.42 Print Specifications* (on page 2666)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

AD.2007-09.2.4 Check Story Drift

The `PRINT STORY DRIFT` command can now be used to check the story drift against a code-specified maximum drift ratio.

Story drift is calculated as the relative horizontal displacement of two adjacent floors in a building. Inter-story drift can also be expressed as some factor times the story height. The allowable factor generally varies from one country code to another and may also vary depending on the type of loading. For example, in IS 1893: 2002 for seismic loading the allowable limit for inter-story drift is 0.004 times the story height whereas in IS 875 for wind loading it is 0.002 times the story height.

The drift at a particular story level in either lateral direction is calculated as the average of all the joint displacements present in that floor level. However if floor diaphragm is present, the drift is calculated at the center of mass (i.e,. control joint) of the floor.

Note: Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command `PRINT STORY STIFFNESS` in the post analysis print commands section.

Refer to TR.42 Print Specifications (on page 2666) for additional information.

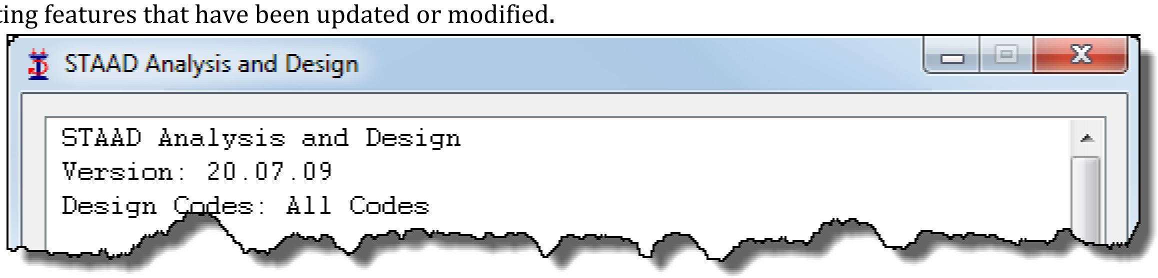
Note: For dynamic IS 1893: 2002 response spectrum, the story drift check is performed by adding a command line within the load case, rather than in the post-analysis print commands. Refer to TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534) for additional information.

Related Links

- *TR.42 Print Specifications* (on page 2666)
- *A. To check for inter-story drift* (on page 965)

AD.2007-09.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.



Note: Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 4) (Build 20.07.09.22).

AD.2007-09.3.1 Steel Design per AISC 360-10

The design of steel sections per ANSI/AISC 360-10 *Specification for Structural Steel Buildings* (AISC 360-10) is now available in STAAD.Pro.

Refer to D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1086) for additional information.

To specify a design using AISC 360-10

1. Select the **Design | Steel** page.

   The **Steel Design - Whole Structure** dialog opens.
2. In the **Current Code** list, select the **AISC 360-10**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.
5. Click **Add**.

   This will insert the following commands into the STAAD input file:

```
CODE AISC UNIFIED 2010
```

AD.2007-09.3.2 Concrete Design per ACI 318-08

The design of concrete members per the ACI code in STAAD.Pro has been updated to the 2008 edition of the code.

A new parameter, LWF, has been added to account for the lightweight concrete reduction factor, λ, in equations 11-3, 11-4, and 11-5 of the code.

Refer to D1.F. American Codes - Concrete Design per ACI 318 (on page 1198) for additional information.

To specify a design using ACI 318-08 in batch mode

Note: Design per the 2008 edition of the ACI code is currently only available in the batch mode.

1. Select the **Design | Concrete** page.

   The **Concrete Design - Whole Structure** dialog opens.
2. In the **Current Code** list, select the **ACI 318 2008**.
3. Click **Define Parameters** list.

   The **Define Parameters** dialog opens.
4. Specify one or more design parameters as necessary.
5. Click **Add**.

   This will insert the following commands into the STAAD input file:

```
CODE ACI
```

AD.2007-09.3.3 Malaysian National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Malaysian NA to EC3, use the following procedure.

The Malaysian National Annex document referred to is “MS EN 1993-1-1:2005.”

When the Malaysian National Annex to EC3 is used for design, the output section title is revised to include the Malaysian National Annex (National Annex to MS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Malaysian NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

Note: For additional information, please refer to D5.D.11 Malaysian National Annex to EC3 (on page 1570) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) .

1. In the Modeling mode, select the **Design | Steel** tab.
   The **Steel Design - Whole Structure** dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters…**.
   The **Design Parameters** dialog box opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(9) Malaysia**.
6. Click **Add**.

   This will insert the following commands into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 9
```

AD.2007-09.3.4 Star Angle Design per IS-800

The design of “star” angle arrangements per IS 800:2007 have been implemented in STAAD.Pro to for members subjected to axial force only. Such star angles are often used for the legs of transmission or communication towers and as bracing members in industrial buildings.

Refer to D8.A. Indian Codes - Steel Design per IS 800 - 2007 (on page 1607) for details on Limit States and Member Property Specifications in IS 800:2007 design.

Notes

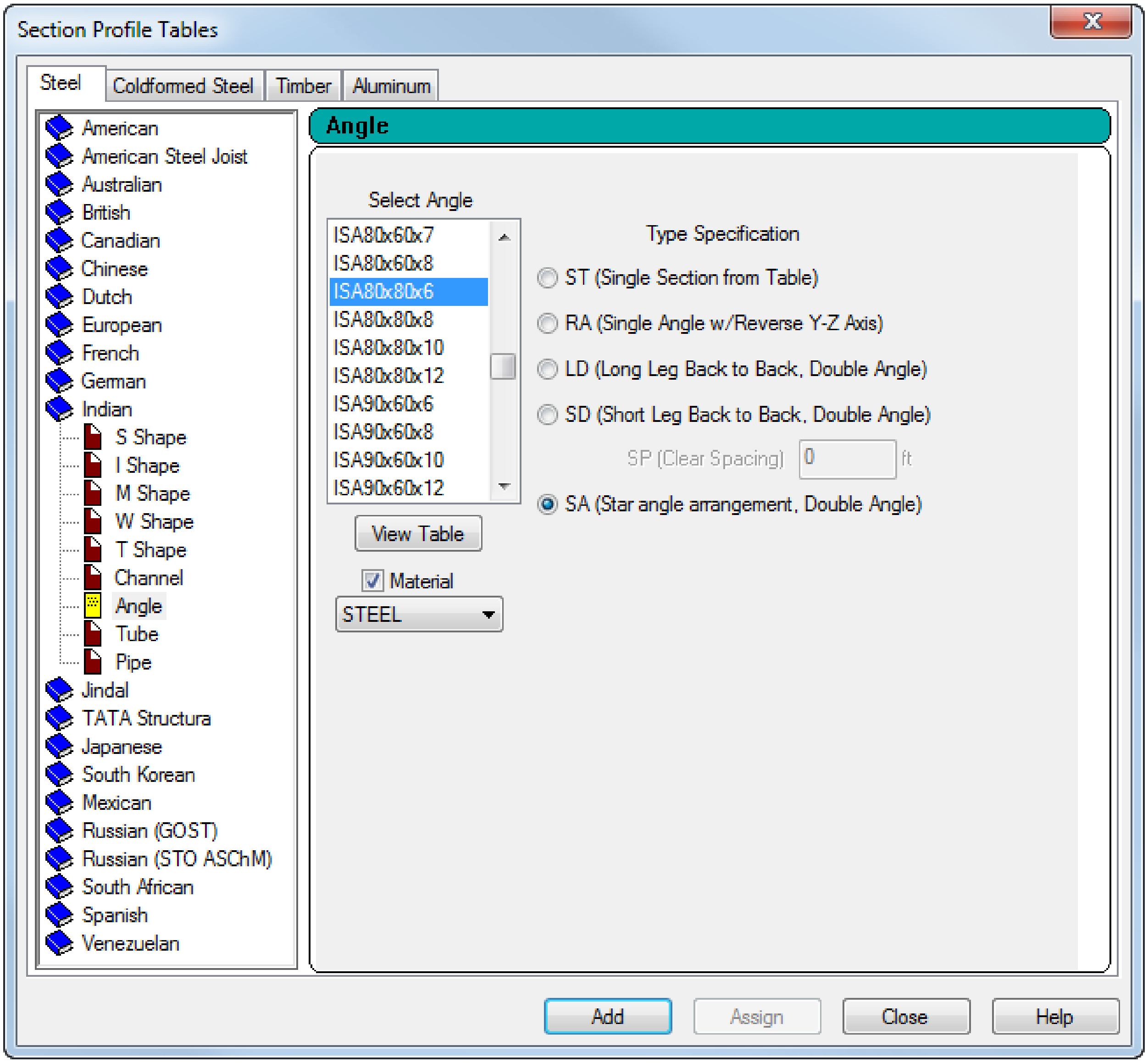
- The design of star angles is only supported for IS 800:2007.
- Members must be declared as a TRUSS member (axial only). An error will be reported in the output if any other member specification is used.
- It is assumed that the star angle arrangement is a welded shape. Plated shapes are not accounted for in the program.

To specify a star angle arrangement for IS 800:2007 design

Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2257) for additional information on assigning properties from steel tables.

1. Select **Commands** > **Member Property** > **Steel Table** > **Indian**.

   The **Section Profile Tables** dialog opens to the Indian steel table.



2. Select the **Angle** tab.
3. Chose an angle section from the **Select Angle** list.
4. Select the **SA (Star angle arrangement, Double Angle)** Type Specification.
5. Click **Add**.

   The section is added to the model and is available for member assignment.

AD.2007-09.3.5 IS 1893 (Part 4) 2005 Seismic Loads

Seismic loads per IS 1893 (Part 4) 2005 for industrial and stack-like structures can be generated in STAAD.Pro. These loads have been incorporated into the existing implementation of IS 1893 (Part 1) 2002 loads.

To add a seismic load per IS 1893 (Part 4) 2005

1. Select **Commands** > **Loading** > **Definitions** > **Seismic Load** > **IS 1893 - 2002**.

The Create New Definitions / Load Cases / Load Items dialog opens to the Seismic Parameters tab (only tab visible in this interface) and the **IS 1893 - 2002/2005** code is selected.

2. Set the option for **Include 1893 Part 4**.
3. Do either of the following:
   - Click Generate to open the IS:1893 Seismic Parameters dialog, or
   - Type values for the command parameters directly in the current dialog table.
4. Click **Add**.

Note: Refer to TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387) for additional information.

AD.2007-09.3.6* ABS/SRSS Combination

A modification to the ABS (Absolute value) and SRSS (square root sum of the squares) results combinations methods.

Additional processing is now used so the resulting combination of results contain appropriate sign/direction as is required when used for design or code checking when used with the ASME NF or AISC 360 05/10 codes.

To instruct the analysis and design engine to automatically generate these load combinations for ABS or SRSS combinations, either:

- use the new, optional `GENERATE` parameter in the `LOAD COMBINATION` command, or

  Note: Refer to TR.35 Load Combination Specification (on page 2616) for additional information.
- select the **Generate Combination** option in the Define Load Combinations dialog (on page 2836).

All of the permutations of positive or negative sign effects in each DOF (degree of freedom) are now considered by the analysis engine. That is, in each of the 6 DOF, both positive and negative sign are considered which results in 64 possible cases (i.e, 2^6). As a result of this, the SRSS and ABS combinations are now presented as load envelopes (i.e., maximum and minimum values) at each member end. This is reflected in both reports and post-processing mode tables. Similarly, in the structure view in the post-processing mode, the force diagrams are now displayed graphically as an envelope rather than a single curve.

Output from the following commands displays pairs of results rows for a load at a joint (marked by an asterisk in the output):

```
PRINT SUPPORT REACTION
```

```
PRINT MEMBER FORCE
```

```
PRINT SECTION FORCE
```

```
PRINT PMEMB FORCE
```

```
PRINT MEMBER STRESS
```

V8*i* (SELECTseries 3)

The Software Release Report for SELECTseries 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i SELECTseries 2 (release 20.07.07) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

The Software Release Report for STAAD.Pro V8*i* SELECTseries 3 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i SELECTseries 2 (release 20.07.07) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-08.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-08.1.1 ISM Integration

STAAD.Pro is now capable of transferring data to and from Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

STAAD.Pro is now capable of transferring data to and from Bentley's Integrated Structural Modeling technology by means of the StructLink utility.

AD.2007-08.1.2 Export to SACS

A new macro is included with STAAD.Pro which is used to export the current STAAD.Pro model to a SACS model, which can be opened in the SACS system Interactive Modeling program.

SACS (Structural Analysis Computer System) is a finite element structural analysis suite of programs for the offshore and civil engineering industries.

Related Links

- *I. To export to a SACS input file* (on page 2081)

AD.2007-08.1.3 European Cold Formed Sections per EN10219-2

Sections defined in the publication, EN10219-2:*1997 Cold formed welded structural hollow sections of non-alloy and fine grained structural steel. Part 2: Tolerances, dimensions and sectional properties* have been added to library of cross sections available in the program.

The following section profiles are included as separate tables in the program:

- Table 6: Circular Hollow Sections (CHS)
- Table 7: Square Hollow Sections (SHS)
- Table 8: Rectangular Hollow Sections (RHS)

To Specify a European cold formed section

1. Select the **General | Property** page.

   The **Properties - Whole Structure** dialog box opens.
2. In the **Properties - Whole Structure** dialog box, click **Section Database**.

   The **Section Profile Tables** dialog box opens.
3. Select the **Coldformed Steel** tab.
4. Select one of the tables available in the **European Cold Formed** section.
5. Select an entry from the **Select Profile** list.
6. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members..

AD.2007-08.1.4 Japanese JIS Hollow Sections

The catalog of available Japanese hot rolled steel sections has been expanded to include the hollow sections from JIS publications.

- JIS G3444:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as "General pipe sections". The existing `PIPE` table has been updated to ensure that it is consistent with this table.
- JIS G3475:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for circular hollow sections and are classed as "Architectural pipe sections". A new table for these Circular Hollow (CHS) will be added.
- JIS G3466:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* defines properties for square and rectangular hollow sections. These will be added to the existing Japanese Sections database as two new tables: Rectangular Hollow (RHS) and Square Hollow (SHS)

Refer to D9.C.4 Built-in Japanese Steel Section Library (on page 1729) for additional details on using the Built-in Japanese Steel Section Library.

To specify a Japanese hollow section

1. Select the **General | Property** page.

   The **Properties - Whole Structure** dialog box opens.
2. In the **Properties - Whole Structure** dialog box, click **Section Database**.

   The **Section Profile Tables** dialog box opens.
3. Select the **Steel** tab.
4. Select one of the tables available in the **Japanese** section: Pipe, Rectangular Hollow, Square Hollow, or Circular Hollow.
5. Select an entry from the **Select Profile** list.
6. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members..

AD.2007-08.2 Features Affecting the Pre-Processor

This section describes features that have been added that affect the pre-processor section of the program, also known as the Modeling Mode.

AD.2007-08.2.1 Wind Load Generation per ASCE 7-10

Wind loads intensity values may now be automatically generated per the 2010 edition of SEI/ASCE 7 using the ASCE 7 Wind Load dialog box (on page 2882) in the user interface. This dialog allows you to generate a wind loading pattern based on the parameters used in this specification.

The primary change in wind load evaluation between the 2002 and 2010 editions of the ASCE 7 specification involve the method of calculating the force coefficient, C_f, for solid freestanding walls and solid signs. The formula provided in the commentary related to Figure 6-20 in the specification is used:

$$C_f = \{1.563 + 0.008542 ln(x) - 0.06148 \cdot y + 0.009011[ln(x)]^2 - 0.2603 \cdot y^2 - 0.08393 \cdot y ln(x)\}/0.85$$

Where:

$x = B/s$

$y = s/h$

To generate a wind load intensity per ASCE 7-10

1. Add a wind load definition to the model.
2. Select this definition in the **Load & Definition** dialog box and click **Add**.

   The **Add New: Wind Definition** dialog box opens.
3. On the Intensity tab, select the as **Custom** in the drop-down list and click **Calculate as per ASCE-7**.

   The **ASCE-7: Wind Load** dialog box opens.
4. On the Common Data tab, select **2010** as the ASCE 7 - edition.
5. Specify or set the parameters needed to define the load.

   Click **Apply** prior to selecting a different dialog tab to update the dialog for the specified parameters.
6. Click **OK**.

AD.2007-08.2.2 Single Mass Model

A new load type, mass, is available for reference load cases. This is used to create a single mass model for all dynamic loads (i.e., seismic, response spectrum, time history, etc.). This load case can be used for seismic loads in lieu of a weight table, reducing repetitive data entry for analysis methods which would require the same data.

A new load type, mass, is available for reference load cases. This is used to create a single mass model for all dynamic loads (i.e., seismic, response spectrum, time history, etc.). This load case can be used for seismic loads in lieu of a weight table, reducing repetitive data entry for analysis methods which would require the same data.

Related Links

- *TR.31.6 Defining Reference Load Types* (on page 2453)
- *M. To add a mass model reference load* (on page 900)
- *M. To add mass loads to the mass model reference load* (on page 900)

- *M. To add weight by a reference load to a seismic load definition* (on page 859)

AD.2007-08.2.3 Eurocode Load Combination Generator

A new macro has been included with the program to generate load combinations for the Strength limit state per *Eurocode – Basis of structural design, BS EN 1990:2002+A1:2005*.

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

These equations specify the following combinations of loads:

$$\sum_{j\geq 1} \gamma_{G,j}G_{k,j} + \gamma_p P + \gamma_{Q,1}Q_{k,1} + \sum_{i>1} \gamma_{Q,1}\psi_{0,1}Q_{k,1}$$

Alternatively, for the strength limit state, the less favorable of equations 6.10a and 6.10b may be used:

$$\sum_{j\geq 1} \gamma_{G,j}G_{k,j} + \gamma_p P + \gamma_{Q,1}\psi_{0,1}Q_{k,1} + \sum_{i>1} \gamma_{Q,1}\psi_{0,1}Q_{k,1} \tag{6.10a}$$

$$\sum_{j\geq 1} \xi_j\gamma_{G,j}G_{k,j} + \gamma_p P + \gamma_{Q,1}Q_{k,1} + \sum_{i>1} \gamma_{Q,1}\psi_{0,1}Q_{k,1} \tag{6.10b}$$

where

Gk	=	Permanent actions
P	=	Prestress actions
Qk	=	Variable actions

Note: The effects in each of the above equations are always additive. If any effect is negative (that is, would reduce the final sum), its effect is taken as zero.

AD.2007-08.2.4 Rigid Floor Diaphragms

A new feature has been added to easily model a rigid floor diaphragm without the need to specify a control joint at each. When specified, this command directs the engine to perform the following:

a. Calculate the center of mass for each rigid diaphragm (where control joint is to be located) considering the mass model of the structure. The mass must be modeled using mass reference load. See TR.31.6 Defining Reference Load Types (on page 2453)

b. Create, internally, an analytical node at the center of mass location to be included during analysis (unless a control node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.

Tip: The center of mass of each diaphragm is included in the preprocessing output. To include the center of rigidity in the post-processing output, you must include the `PRINT DIAPHRAGM CR` command. See TR.42 Print Specifications (on page 2666)

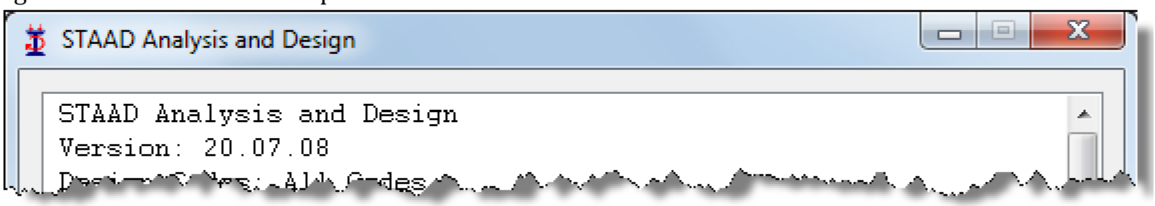
c. Search all nodes available within a diaphragm and add them as dependent nodes; with the control node located at the center of mass for the diaphragm (or at the specified control node)

Related Links

- *TR.28.2 Floor Diaphragm* (on page 2328)
- *Node Specification dialog* (on page 2784)
- *M. To assign nodes to a floor diaphragm* (on page 818)

AD.2007-08.3 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.



AD.2007-08.3.1 API 2A WSD 21st Ed. Update

Joint checking for tubular members per the American Petroleum Institute 2A-WSD code has been updated to the 21st Edition (December 2000) of that code, including errata and supplements 1 to 3 (latest: Supplement 3 - March 2008). Additionally, the process of joint design has been simplified.

Note: Only simple joints and overlapping joints will be considered by the program. Other type such as grouted joints, joints with ring stiffeners etc are not be considered.

The clauses/sections in the API code that have been dealt with are:

- 4.2.1 Material strength
- 4.2.3 Minimum Capacity
- 4.2.4 Joint Classification
- 4.3 Simple joints
- 4.4 Overlapping joints

Refer to D1.J. American Codes - Steel Design per API 2A-WSD 2000 (on page 1283) for additional information on the methodology used.

External Joint Data File

In previous versions, the `LEG` design parameter was used to direct the program to generate a separate joint data file or to check a user-specified file. This file is now automatically generated or checked as needed.

As the API code allows for mixed joint types, the PUNCH column in the input file has been replaced with K, X, and Y columns which are used to designate fractional contributions of each joint class. Similarly, overlapping joints are indicated by using a negative value for the `GAP` between braces and the member number of the overlapping member in the `OB` column (replaces THETAT used in previous versions).

If the `filename.PUN` file is not detected in the same folder as the current STAAD input file (where "`Filename`" is the same as the `.STD` file), then the program assumes this is the initial joint design and this file is created. If, however, this file is detected, the program assumes that the that the joint design has been performed at least once and will use this file to perform the joint checks.

To check tubular member joints per API

The checking of joints is an iterative process done by means of an automatically generated text file.

1. Model a structure as you normally would.

> **Note:** Only circular pipe members are considered.

> **Tip:** Using `TRUSS` specifications helps to reduce analysis time.

2. In the **Steel Design - Whole Structure** dialog box, select `API` as the design code.
3. (Optional) Specify the factor of safety used for joint checks using the new `FSJ` design parameter.
4. (Optional) Specify all other necessary design parameters.

   The `LEG` parameter is no longer used to generate or check joint data files.
5. Specify `CODE CHECK` or `SELECT MEMBER` commands as needed and perform the analysis.

   If this the `filename.PUN` file is not detected in the same folder as the input file, the program assumes this is the first time the structure is being analyzed and generates this input file with default data for the detected joints. Each joint is assumed to be a Y joint by default.
6. Modify the default joint data in the `filename.PUN` as needed to describe the actual joint conditions.

   A text editor can be used to make changes to this file. Be sure to save changes once complete.
7. Re-analyze the STAAD input file.

   Joint check results follow the steel design output.
8. Repeat steps 3 through 7 as needed to make changes in the structure.

AD.2007-08.3.2 Shear Buckling per EC3

The design code checks performed per Eurocode 3 have been updated to include checks for shear buckling in I Sections and PFC Sections. Eurocode 3 – Part 1 (EN 1993-1-1:2005) states in Cl.6.2.6 that the checks for Shear buckling are to be based on the procedure in EN 1993-1-5. STAAD.Pro performs checks based on the methods Section 5 of EN 1993-1-5:2006.

In the case of an unstiffened web, the program will check the unstiffened web capacity. If the demand due to applied loads is greater than this capacity, the program will calculate a suitable spacing for transverse stiffener plates in order to meet the demand.

In the case of a web with transverse stiffeners, the program will check the capacity of the web considering the provided stiffener spacing. The program will consider both the buckling capacity of the web as well as the flange. If the demand due to applied loads is greater than this capacity, the program will calculate a reduced stiffener spacing for transverse stiffener plates in order to meet the demand.

> **Note:** Only transverse stiffeners are taken into account. The effect of any longitudinal stiffeners is ignored.

The distance between transverse stiffeners is specified by the `STIFF` parameter, in the current units of length. If no value is specified, the program assumes a spacing equal to either the member length or depth of beam, whichever is greater.

The output file provides recommendations on the evaluation of stiffeners (e.g., adding stiffeners at a spacing or increasing the web thickness).

Refer to D5.C.5.3 Members Subject to Shear (on page 1486) for additional information.

AD.2007-08.3.3 IS800:2007 Working Stress Method

The IS:800-2007 Steel code was deviated in concept from its -1984 version (based on Working Stress Method) and introduced the Limit State Method of Design. The entire 2007 version of the code is devoted to the Limit State Method of Design, except Chapter 11. This Chapter comprises of a couple of pages and has the guideline for the Design of Steel sections as per working stress method (WSM). The approach of this new working stress method is different from its earlier version and utilizes the concept of Section Slenderness and Section Classification.

The program now includes design per the Working Stress Method (WSM) methodology in addition to Limit State method for design of steel structures per IS800:2007.

Some minor corrections to the Limit State Design option have also been made.

To specify a design using IS 800:2007 Working Stress Method

1. In the Modeling mode, select the **Design | Steel** tab.

   The **Steel Design - Whole Structure** dialog box opens.
2. In the **Current Code** drop-down menu, select **IS800 2007 WSD**.
3. Click **Define Parameters...**.

   The **Design Parameters** dialog box opens.
4. Click **Add**.

   This will insert the following commands into the STAAD input file:
   ```
   CODE IS800 WSD
   ```

AD.2007-08.3.4 Surface Element Selfweight

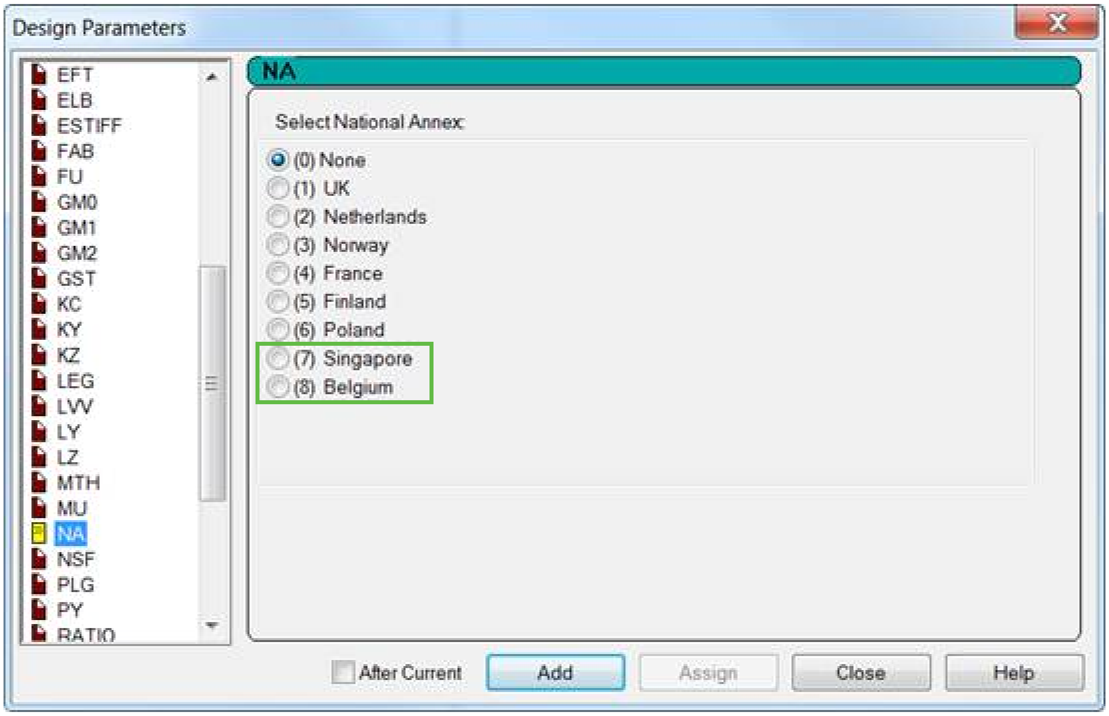
A new surface load has been added to include the self weight of surface elements. This command can be used to calculate and include the weight of surface elements in the analysis of a structure.

Related Links

- *TR.32.9.2 Surface Selfweight Load* (on page 2497)
- *M. To add a surface selfweight load* (on page 841)

AD.2007-08.3.5 Eurocode 3 National Annexes

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Singapore and Beglium. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.



AD.2007-08.3.5.1 Belgian National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Belgian NA to EC3, use the following procedure.

The Belgian National Annex document referred to is "NBN EN 1993-1-1:2005."

When the Belgian National Annex to EC3 is used for design, the output section title is revised to include the Belgian National Annex (National Annex to NBN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Belgian NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

Note: For additional information, please refer to D5.D.10 Belgian National Annex to EC3 (on page 1563) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) .

1. In the Modeling mode, select the **Design | Steel** tab.
   The **Steel Design - Whole Structure** dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters...**.
   The **Design Parameters** dialog box opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(8) Belgium**.
6. Click **Add**.

   This will insert the following commands into the STAAD input file:

   ```
   CODE EN 1993-1-1:2005
   NA 8
   ```

AD.2007-08.3.5.2 Singaporean National Annex to Eurocode 3 (EN 1993-1-1:2005)

To specify a design using the Belgium NA to EC3, use the following procedure.

The Singaporean National Annex document referred to is "SS EN 1993-1-1:2005."

When the Singaporean National Annex to EC3 is used for design, the output section title is revised to include the Singaporean National Annex (National Annex to SS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Singaporean NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

Note: For additional information, please refer to D5.D.9 Singaporean National Annex to EC3 (on page 1556) and D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) .

1. In the Modeling mode, select the **Design | Steel** tab.
   The **Steel Design - Whole Structure** dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters...**.
   The **Design Parameters** dialog box opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(7) Singapore**.
6. Click **Add**.

   This will insert the following commands into the STAAD input file:

   ```
   CODE EN 1993-1-1:2005
   NA 7
   ```

AD.2007-08.3.6 EC3 Slender Circular Hollow Sections

Slender circular hollow (pipe) sections may now be designed per Eurocode 3 (EN 1993-1-6:2007).

Eurocode 3 – Part 1 (EN 1993-1-1:2005) –hereafter EC3-6– states in Cl. 6.2.2.5 (5) that the design of slender circular hollow sections is to be based on the procedure in EN 1993-1-6. EC3 6 deals with the design of shell structures. EC3-6 does not, however, specify additional or modified safety factors. Therefore, the default safety factors from EN 1993-1-1 are used.

Note: You can change these values through the `GM0`, `GM1`, and `GM2` design parameters.

The program checks the plastic and buckling limit states for primary stresses based on the stress design method described in EC3-6.

Refer to D5.C.5.6 Design of Slender pipe sections to EN 1993-1-6 (on page 1497) for details on the methodology and calculations used in this design.

AD.2007-08.3.7 User Defined Section for EC3

The feature to design user-provided table (UPT) general sections has now been introduced steel members designed per Eurocode 3 (EN 1993-1-1:2005). However, rather than assuming that the section will behave like an I section, you are given the option of choosing the 'section-type' he would like to design the member for.

This is achieved through the introduction of a new design parameter, `GST`, that has the following values:

1. I-Section (Default)
2. Single Channel

3. Rectangular Hollow Section
4. Circular Hollow Section
5. Angle Section
6. Tee Section

Unless specified using the GST parameter, a general section will be assumed to be an I- Section.

Note: This parameter will be ignored if assigned to any section other than a General Section.

The design procedure will then account for the section type and proceed with the design as necessary. The output report will also indicate the section type considered for the design of the UPT section. The design output will indicate the section as follows:

```
            *     1 ST   IPE100       (UPT: DESIGNED AS I-SECTION)

                       FAIL     EC-6.3.2 LTB       8.591        1
                       0.00          0.00        -31.25       2.50
 |---------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     1 UNIT - kN,m  SECTION CLASS  1  |
 |MCZ=    9.2 MCY=    2.1 PC=    12.3 PT=   242.1 MB=    3.6 PV=   68.7|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
 | PZ=   242.05    FX/PZ =   0.00       MRZ=    9.2      MRY=    2.1   |
 |---------------------------------------------------------------------|
```

Refer to D5.C.6 Design Parameters (on page 1502) for additional information on all EC3 design parameters.

AD.2007-08.4 Features Affecting Post Processing

The following new feature has been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

AD.2007-08.4.1 Eurocode 2:2004 Slab Design

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007.08.4.2 Changes to STAAD.foundation license

The license for STAAD.foundation V8i (release 5.3) is now included with STAAD.Pro contains the capability for design the following rigid foundation types:

- Isolated
- Combined
- Pile Cap

STAAD.foundation functions as a design module similar to others included with STAAD.Pro.

Contact Bentley Systems for a full license to use all of the design capabilities for STAAD.foundation or to upgrade to STAAD Foundation Advanced V8i, which includes an improved user interface, foundation toolkit wizards, plant foundation types, and FEM analysis for mat foundations.

AD.2007-08.5 Nuclear Related Features

The following new features have been added for the NRC release.

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) NRC (build 20.07.08.22) or higher.

AD.2007-08.5.1 Design per ASME NF 3000-2001

The design per ASME NF 3000-2001 is now available as a new design code option.

Design of members per ASME NF 3000 - 2001 requires the *STAAD Nuclear Design Codes* SELECT Code Pack.

Refer to D1.L.4. ASME NF 3000 - 2001 & 2004 Codes (on page 1338) for additional information.

AD.2007-08.5.2 ASME NF 3000-2001 Service Level Conditions

All editions of the ASME NF 3000 code now can be designed for different service level conditions as defined by that code.

Service Level Conditions are basically the loading conditions for which the plant structure and its components are to be designed. The same primary load can be multiplied by different factors to signify the different service levels. Also the load combinations for various service levels are different and pre-defined by the code.

Each ASME NF 3000 code edition now contains the a new design parameter, `SLR`, which is used to specify the service condition level as defined in the codes. For the case of service level D (failure), three additional new parameters –`KS`, `KV`, and `KBK`– are used to directly specify the service level factors.

Design of members per ASME NF 3000 - 2001 requires the *STAAD Nuclear Design Codes* SELECT Code Pack.

Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1348) for additional information.

AD.2007-08.5.3 TATA Structura Sections

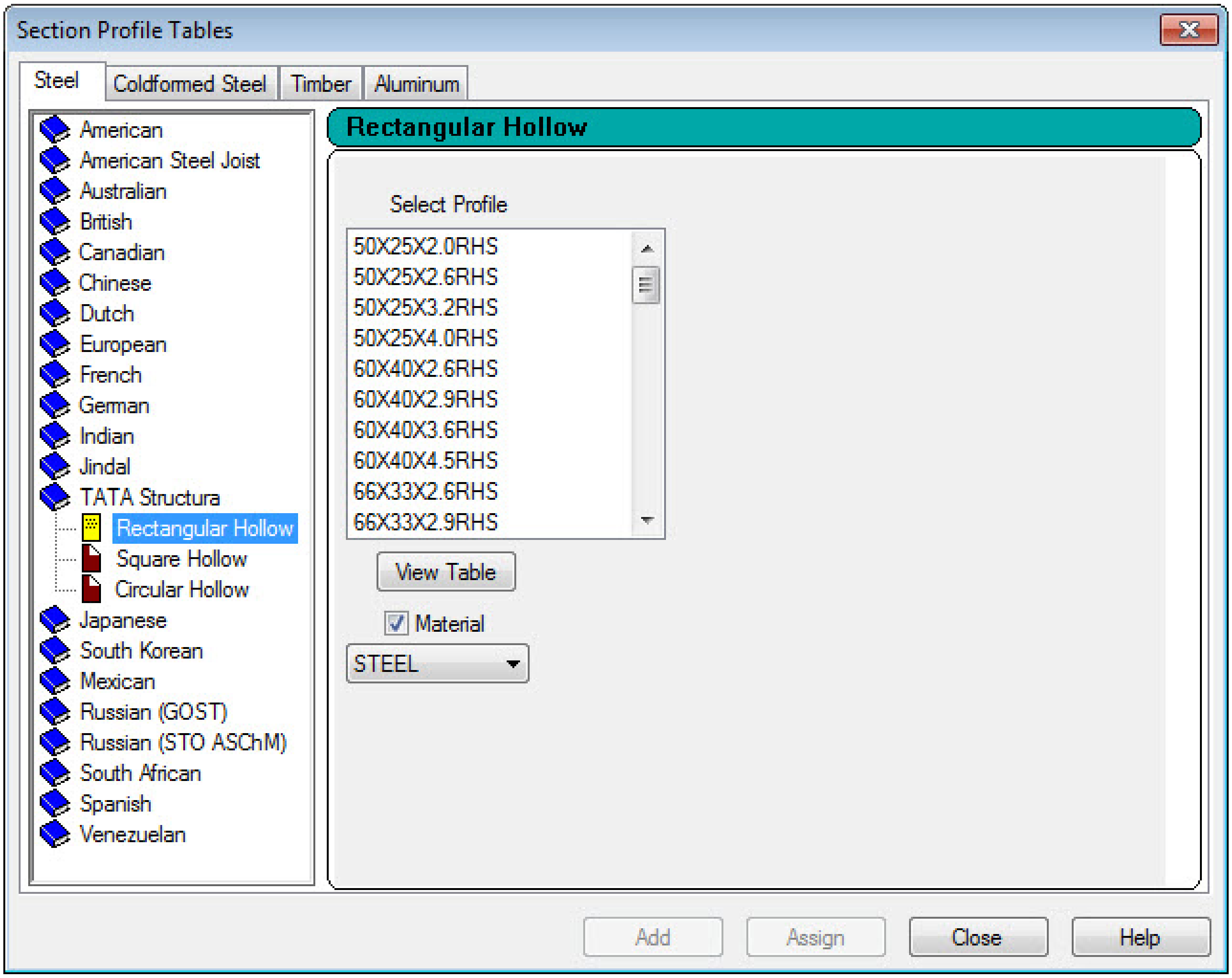
The catalog of available TATA Structura (Indian) hot rolled steel sections has been expanded to include the hollow sections from TATA publications.

Specifying a TATA Structura hollow section

1. Select the **General | Property** page.

   The **Properties - Whole Structure** dialog box opens.
2. In the **Properties - Whole Structure** dialog box, click **Section Database**.

   The **Section Profile Tables** dialog box opens.
3. Select the **Steel** tab.
4. Select one of the tables available in the **TATA Structura** section: Rectangular Hollow, Square Hollow, or Circular Hollow.



5. Select an entry from the **Select Profile** list.
6. Click **Add**.

   The section is now added to the Section tab in the **Properties - Whole Structure** dialog box and can be assigned to members..

V8*i* (SELECTseries 2)

The Software Release Report for STAAD.Pro V8*i* (SELECTseries 2) contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8*i* (SELECTseries 1) (release 20.07.06) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-07.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-07.1.1 Academic Licensing

In order to ensure that the next generation of engineer that emerges from the higher education system is up to speed using our applications, Bentley has a policy of providing software to Universities and Colleges at a favorable rate.

Students can now use STAAD.Pro under an Academic License, which is obtained through a SELECT account. Contact your regional engineer or visit Bentley.com to obtain a license.

Note: When using an Academic License, the program window title bar and About window indicate this. Similarly, all output (Analysis files and Reports generated from STAAD.Pro) are marked as "Academic License User."

Caution: The Advanced Analysis Engine is not available when using the program under an Academic Licence.

AD.2007-07.1.2 StructLink and PipeLink Plug-ins

Two plug-ins are available to you when installing STAAD.Pro: StructLink and PipeLink.

- StructLink is a utility used for the bi-direction exchange of data between STAAD.Pro and ProSteel V8i.
- PipeLink is an all new utility used for the exchange of pipe stress model data between STAAD.Pro and AutoPIPE V8i. Refer to AD.2007-07.3.4 AutoPIPE V8i (SELECTseries 2) and PipeLink support (on page 209) for additional the bi-directional data exchange capabilities made available through this utility.

Note: During the installation of STAAD.Pro, select the option to install additional programs and utilities in order to have these two utilities installed.

Refer to the documentation included with these plug-ins for additional information on their use.

AD.2007-07.1.3 Structural Dashboard Integration

Bentley's Structural Dashboard is now integrated into STAAD.Pro V8i.

Bentley's Structural Dashboard V8i is a free utility application which allows you to manage workflows and project files as well as keep up to date with latest products, news, and Be Communities happenings. This program can now be accessed from within STAAD.Pro and will launch whenever STAAD.Pro is started to assist you in managing your entire project workflow.

When STAAD.Pro is first launched after installing Structural Dashboard, a welcome dialog opens to allow you to set the automatic launch option. To launch the program and continue allowing it to launch whenever a Bentley Structural program starts, leave the option selected and click the **OK** button. Otherwise, you can de-select this option before proceeding.

Launch Structural Dashboard on program startup	Set this option to open the Structural Dashboard application whenever STAAD.Pro is opened. **Tip:** This setting can be changed at any time from within the Structural Dashboard program.
Show me this dialog on startup	Set this option to display this dialog whenever STAAD.Pro is opened. **Tip:** This setting can be changed at any time from the Configure Program dialog File Options tab.

To launch Structural Dashboard from STAAD.Pro

1. Select **File** > **Structural Dashboard...**.

   The Bentley Structural Dashboard V8i program opens.

> **Note:** If Structural Dashboard has not been installed, this menu item is inactive. You can download the program from http://www.bentley.com/en-US/Promo/ISM/downloads/.

> **Note:** Refer to Section 2.3.1 of the User Interface Manual for additional help in using the Structural Dashboard with STAAD.Pro.

AD.2007-07.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

> **Note:** Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.31).

AD.2007-07.2.1 Time History Spectrum Enhancements

New options have been added to the spectrum input for a Time History definition which allow you to output time history input data, a Response Spectrum for a Time History load, or to use frequency-spectrum pairs. These options can be added by modifying the input command file.

Two new output options are available for reporting time history input and synthetic time history ground acceleration data used by the program for a time history load with the spectrum generation option. You can control the amount of output generated (as this can be quite large) as well.

A new option has also been added to allow you to instruct the program to use frequency-spectra pairs in lieu of period-spectra pairs for the time history spectrum input.

Related Links

- *TR.31.4 Definition of Time History Load* (on page 2441)
- *M. To generate output for time history spectrum* (on page 878)
- *M. To use frequency-spectra pairs in a time history load* (on page 878)

AD.2007-07.2.2 Response Spectrum Signed Results and IMR Load Cases

Two methods to produce signed response spectrum results have been added to the STAAD.Pro analysis engine. The Dominant and Sign commands may be used in the input file to produce signed output. Additionally, STAAD.Pro now includes an option to automatically generate new load cases based on a specified number of modes from the response spectrum.

Signed Results

STAAD.Pro can now assign a mathematical sign (positive or negative) to the modal results by one of two means. The first method allows you to select a DOMINANT mode, the sign of which will then be applied to all other modes. The second method will produce signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign.

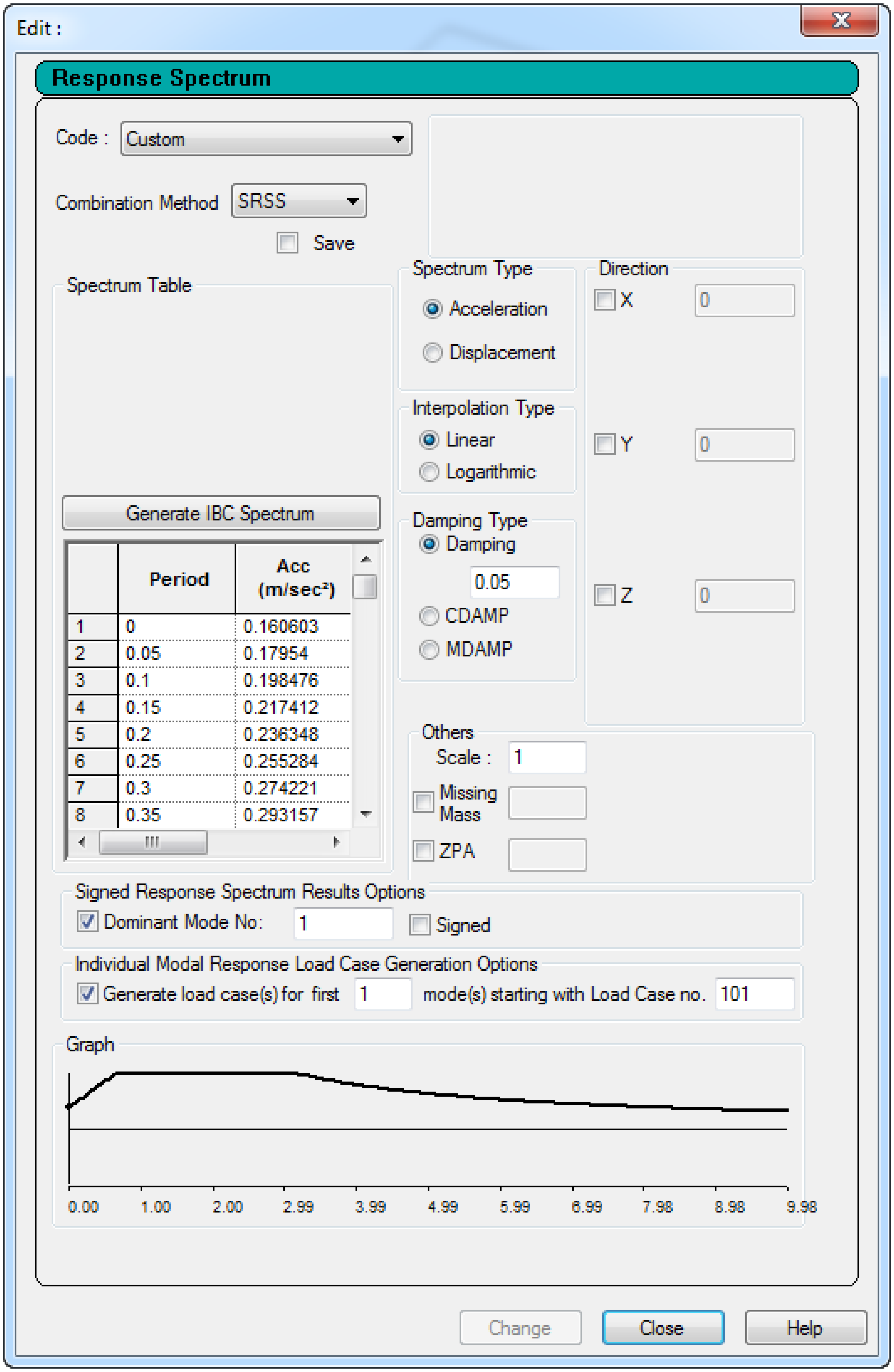
Individual Modal Response Case Generation

The Individual Modal Response (IMR) load cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere. You can use subsequent load cases with Repeat Load combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). You can also use the Repeat Load capability to combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

The modal accelerations are multiplied by the nodal masses to produce equivalent static lateral forces for each modal load case.

Note: When the IMR option is entered for a Spectrum case, then a Perform Analysis & Change must be entered after each such Spectrum case.


Edit :
Response Spectrum
Code : Custom
Combination Method SRSS
Save
Spectrum Table
Generate IBC Spectrum
Period
Acc (m/sec²)
1 0 0.160603
2 0.05 0.17954
3 0.1 0.198476
4 0.15 0.217412
5 0.2 0.236348
6 0.25 0.255284
7 0.3 0.274221
8 0.35 0.293157
Spectrum Type
Acceleration
Displacement
Interpolation Type
Linear
Logarithmic
Damping Type
Damping
0.05
CDAMP
MDAMP
Direction
X 0
Y 0
Z 0
Others
Scale : 1
Missing Mass
ZPA
Signed Response Spectrum Results Options
Dominant Mode No: 1
Signed
Individual Modal Response Load Case Generation Options
Generate load case(s) for first 1 mode(s) starting with Load Case no. 101
Graph
0.00 1.00 2.00 2.99 3.99 4.99 5.99 6.99 7.98 8.98 9.98
Change
Close
Help

To add Signed results to a Response Spectrum

1. Select **Commands** > **Loading** > **Load Commands**.

   or

   Select the General **|** Load & Definition page and then click the **New...** button.

   The Create New Load Items dialog opens.
2. Select the Response Spectra tab.
3. Select the **Code** you wish to use.

   Note: See below for using IMR generation options. All other parameters are same as previous versions.
4. Select the option to use the **Dominant Mode No.** to assign the same sign as the selected mode to all modes.
5. (Optional) Select to provide **Signed** results to

   Note: Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.
6. Click the **Add** button to add this response spectrum load.

To add Individual Modal Response results to a Response Spectrum

1. Select **Commands** > **Loading** > **Load Commands**.

   or

   Select the General **|** Load & Definition page and then click the **New...** button.

   The Create New Load Items dialog opens.
2. Select the Response Spectra tab.
3. Select the **Code** you wish to use.

   Note: See above for options to add signed results. All other parameters are same as previous versions.

   Note: The Individual Modal Response case generation is not available for SNiP II code response spectra.
4. Select the option to **Generate load cases for ...** to individual modal response load cases.
5. (Optional) Specify the number of modes for which load cases will be generated.

   Note: Selecting this option will not use the Dominant Mode No., but rather will create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.
6. (Optional) Specify a beginning load case number for the first primary load case generated from the IMR.
7. Click the **Add** button to add this response spectrum load.

Related Links

- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *Response Spectra tab* (on page 2854)
- *M. To add a generic response spectrum* (on page 861)

AD.2007-07.2.3 Design of Class 4 (Slender) Steel Sections per S16-01

An update to the Canadian Steel Design code has been added for the design of Class 4 (slender) steel sections per CAN/CSA-S16-01. Previous versions of STAAD.Pro were capable of designing Section Classes 1, 2, or 3.

The design of slender Class 4 steel sections does not require any different actions or input. The analysis engine will determine if a section meets the criteria for a Class 4 section and then perform the necessary checks, if design checks have been requested for that member.

Methodology

Refer to D4.B.6 Member Resistances (on page 1424) for a detailed description of the methodology used in STAAD.Pro for performing the design of Class 4 sections per S16-01. A verification problem using this feature has also been added to Section 3B.10.

AD.2007-07.2.4 Von Mises Stresses per AIJ 2002 and 2005

Design per AIJ (Japanese) steel design codes has been updated to include checking members in accordance with Von Mises stress criteria in AIJ 2005. This check is a requirement for the design of steel structures in nuclear power plants in Japan.

The von Mises stress equation is calculated when the new `MISES` parameter has been set to a value of 1 (the default value of 0 does *not* check this condition). The calculated forces and moments are combined per the von Mises stress criteria.

To specify a von Mises stress check in an AIJ 2002 or AIJ 2005 design

1. Create a model with steel members.
2. Select either **AIJ 2002** or **AIJ 2005** for the **Current Code** on the **Design | Steel** page.
3. Click the **Define Parameters...** button.

   The Design Parameters dialog opens.
4. Select the **MISES** tab in the parameters list.
5. Select option 1 to instruct STAAD.Pro to perform the von Mises stress check as part of the steel design.
6. Click the **Add** button to add this parameter.
7. Close the Design Parameters dialog.
8. Assign the `MISES` parameter to members as needed, just as you would any other design parameter.

How the von Mises check results are included in the output depends on the level of detail (`TRACK` parameter) selected:

Track 0 or 1	The von Mises stress is reported if this ratio is the critical condition.
Track 2	The value for $f_m = \sqrt{(\sigma_x{}^2 + 3\times\tau_{xy}{}^2)}$ (numerator in the von Mises stress ratio equation) is displayed in the Stresses output category. When the von Mises check ratio is the critical condition, the value of the ratio is reported.
Track 4	Used for deflection checks only. Von Mises checks are not reported.

Methodology

Refer to D9.B.4 Von Mises Stresses Check (on page 1727) or D9.C.10 Von Mises Stresses Check (on page 1742) for a detailed description of the methodology used in STAAD.Pro for performing von Mises stress checks per AIJ 2002 or 2005.

AD.2007-07.2.5 Norsok N-004

The design of tubular steel (European round pipe sections) members per NORSOK N-004 Rev 2, October 2004 has been included in STAAD.Pro. The program will perform the member design for ultimate limit states (and optional deflection checks for serviceability). The tubular joints can also be automatically generated and checked per the code.

The NORSOK code has been added to the steel design code list available in STAAD.Pro. Selecting this code allows you to assign parameters, including defining the water level above the origin (for calculating hydrostatic pressure) or the

Note: N-004 refers to the superseded version of Eurocode 3 (DD ENV 1993-1-1) in several places. In such cases, the corresponding clause from the latest version of EC-3 (EN 1993-1-1:2005) has been used in the STAAD.Pro implementation.

To perform a member design per the NORSOK N-004 code

1. Create a model with steel tubular members.

   Caution: The Norsok code only supports pipe sections. Errors will be presented in sections other than pipe members are used.

2. Select **NORSOK** for the **Current Code** on the **Design | Steel** page.
3. Click the **Define Parameters** button.

   The Design Parameters dialog opens.

4. Specify parameters as required.

   Note: The height of water level above the origin is specified using the HYD parameter. Alternatively, the PSD parameter may be used to define the water pressure.

5. Close the Design Parameters dialog.
6. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

To perform a joint check per the NORSOK N-004 code

1. Add the `CHECK JOINT` command to a new `PARAMETER` manually in the STAAD.Pro input file using the Editor.
2. Perform an preliminary design by selecting **Analyze** > **Run Analysis...**.

   The program creates an external text file titled `filename_JOINTS.NGo` which contains the automatically generated chord and brace definitions associated with the nodes included in the `CHECK JOINT` command. All joints are classified as Y by default.

3. Open the text file using a text editor program (i.e., Notepad or STAAD Editor).
4. Manually edit the joint classifications as needed.
5. (Optional) Edit the Brace and Chord definitions as needed.

> **Note:** The Brace and Chord members at each joint are assumed based on the relative cross section dimensions. Lengths of Chord and Brace members are taken as the analytical beam member length.

6. Save the text file and the re-analyze the structure

Methodology

Refer to D12.B. Norwegian Codes - Steel Design per NORSOK N-004 (on page 1837) for a detailed description of the methodology used in STAAD.Pro for performing steel tube member design per NORSOK N-004.

AD.2007-07.2.6 EC3 Torsion Design

Design per EC3 [EN 1993-1-1:2005] has been enhanced to include the design of members subject to torsion. You may select to have the program execute basic or detailed torsion stress checks. Torsion design checks can be performed on I-sections, H-Sections, Channel sections, and structural hollow sections (RHS, SHS, CHS).

> **Note:** The default behavior is to neglect torsion. The new `TORSION` parameter must be set to either 1 (basic) or 2 (detailed) to perform torsion design.

To include torsion design for EC3 steel design members

1. Create a model with steel members.
2. Select **EN 1993-1-1:2005** for the **Current Code** on the **Design | Steel** page.
3. Click the **Define Parameters...** button.

   The Design Parameters dialog opens.
4. Select the **TOR**(sion) tab in the parameters list.
5. Select either option 1 (von Mises check excluding warping effects) or option 2 (detailed checks including warping effects) to include design for torsion and click the **Add** button to add this parameter.
6. Specify the loading and support conditions of members subject to torsion using the **CMT** tab in the parameters list and click the **Add** button to add this parameter.
7. (Optional) For the cases of a concentrated torque (CMT = 2,3, or 6) somewhere along the member length (other the default of mid-span), specify the location of the torque using the **ALH** tab in the parameters list and click the **Add** button to add this parameter.
8. (Optional) Specify the effective length of members for torsion using the **EFT** tab in the parameters list and click the **Add** button to add this parameter.
9. Close the Design Parameters dialog.
10. Assign the torsion-related parameters to members as needed, just as you would any other design parameter.

Torsion design in EC3 is given in Cl. 6.2.7 of EN 1993-1-1:2005. Therefore, this clause is used primarily for this implementation.

EN 1993-1-1:2005 does not deal with members subject to the combined effects of torsion and lateral torsional buckling. However, EN 1993-1-6 considers such a condition in Appendix A. Therefore, STAAD.Pro uses Appendix A of EN 1993-1-6 to check for members subject to combined torsion and LTB.

The following clauses from EC3 are then considered:

- Cl. 6.2.7(1)
- Cl. 6.2.7(9)
- Cl. 6.2.7(5)
- EC-3 -6 App A

When torsion design is included (TOR = 1 or 2), then the EC3 design output includes the following sections:

- Basic (TORSION = 1) - The ratio calculated for stress interaction per EC-6.2.7(5) is displayed for each load case, along with the calculated values of axial force, shear in Y and Z, Bending about Y and Z, and torsion.
- Detailed (TORSION = 2) - The additional clauses viz. 6.2.7(1), 6.2.7(9) and EC3-6 A-1 will be included in the output. The stress interaction ratio per each is displayed for each load case, along with the calculated force and moment values used. Additional torsion calculation details are provided as well.

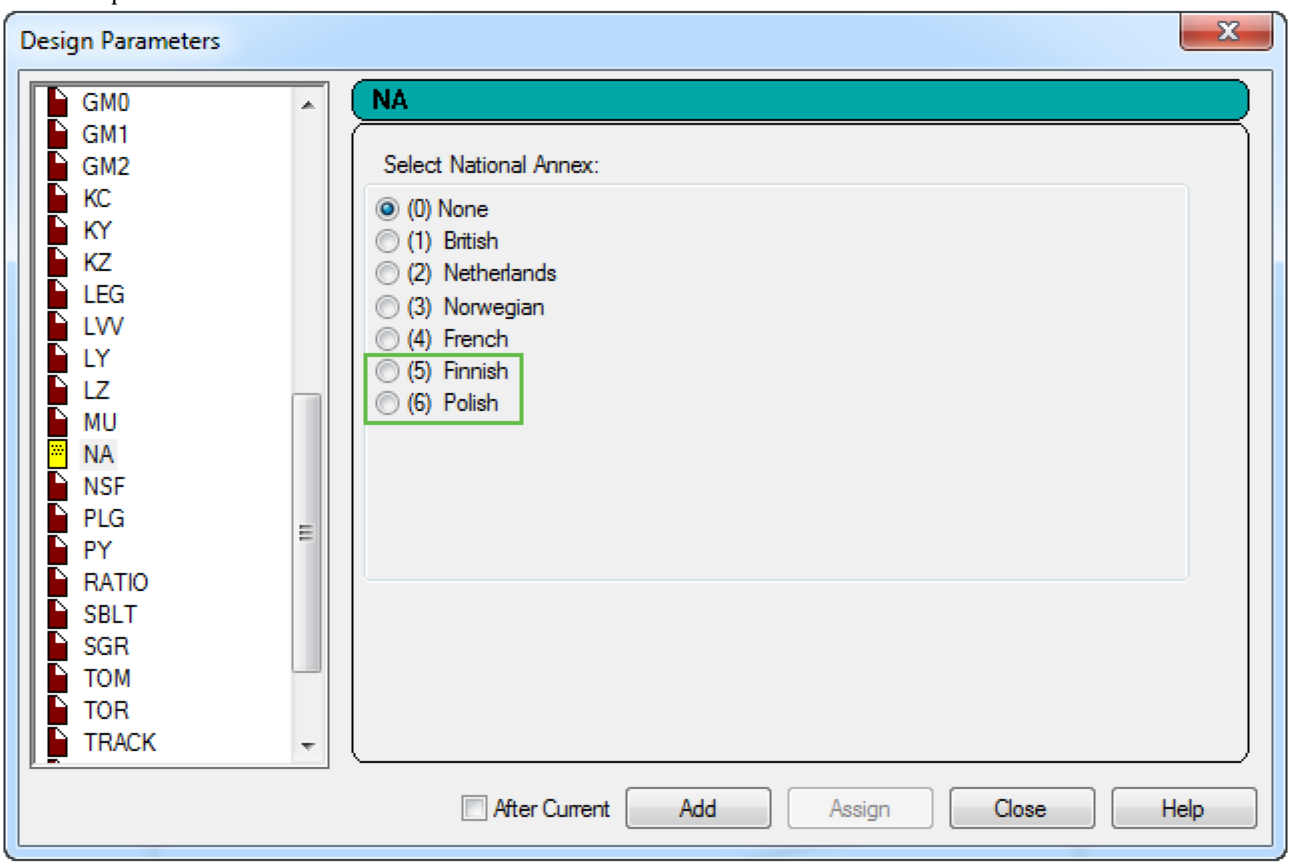
Note: If Torsion design is selected for a member which does not have any torsional moment, a warning is displayed in the output.

Methodology

Refer to D5.C.5.4 Members Subject to Torsion (on page 1488) for a detailed description of the methodology used in STAAD.Pro for performing torsion stress checks per EC3.

AD.2007-07.2.7 Eurocode 3 National Annex

Two additional country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: Finland and Poland. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.



AD.2007-07.2.7.1 Finnish National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-Finnish NA Steel Design, use the following procedure.

The Finnish National Annex document referred to is "National Annex to Standard SFS-EN 1993-1-1."

1. In the Modeling mode, click the **Design** > **Steel tab**.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(5) Finland**.
6. Click the **Add** button.

   This will insert the following commands into the STAAD input file:

   ```
   CODE EN 1993-1-1:2005
   NA 5
   ```

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516)

When the Finnish National Annex to EC3 is used for design, the output section title is revised to include the Finnish National Annex (National Annex to SFS-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Finnish NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

AD.2007-07.2.7.2 Polish National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-Polish NA Steel Design, use the following procedure.

The Polish National Annex document referred to is "National Annex to Standard PN-EN 1993-1-1".

1. In the Modeling mode, click the **Design** > **Steel tab**.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters...** button.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(6) Poland**.
6. Select the new **PLG** parameter in the list box.

   Note: This parameter is used to select if additional checks per clause 6.3.3 will be performed for designs using the Polish National Annex.
7. Click the **Add** button.

   This will insert the following commands into the STAAD input file:

   ```
   CODE EN 1993-1-1:2005
   NA 6
   ```

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516)

When the Polish National Annex to EC3 is used for design, the output section title is revised to include the Polish National Annex (National Annex to PN-EN 1993-1-1). Additionally, the partial safety factors used are included in the output and are as specified in the Polish NA. The value for C1 and k factors used in the calculation of the elastic critical moment are also included in the report.

AD.2007-07.2.8 AS4100 Physical Member Design

The workflow for design steel members per AS 4100:1998 has been updated to incorporate the use of physical members. Physical members are groups composed of a series of analytical beam elements of the same section and which are colinear (analytical beams are the beams used in modeling in STAAD.Pro). Using physical beams allows you to design for the actual conditions of the structure and assign specifications based on the true conditions of a steel member.

Some of the physical member design updates apply to design codes other than AS 4100, such as checks in the STAAD.Pro analysis engine for physical member overlapping and colinearity. These checks were previously made in the graphical interface but now they will checked again in the engine in the event you have manually generated the STAAD.Pro input file.

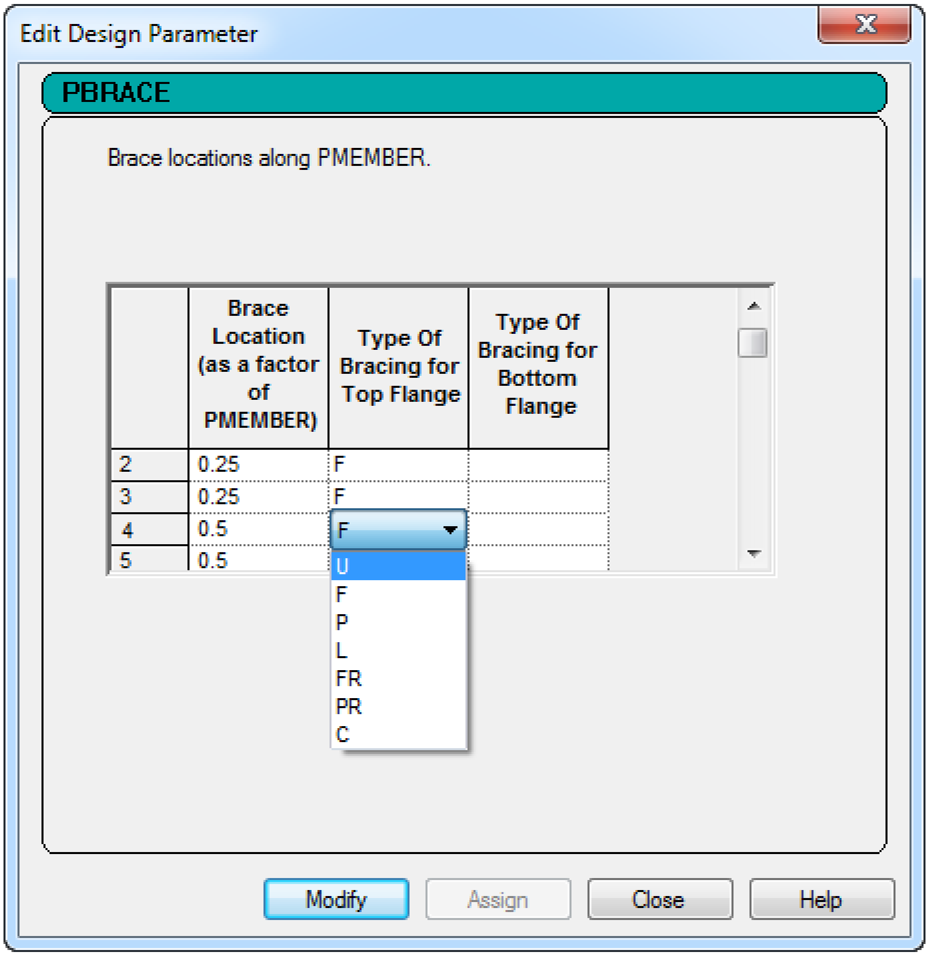
The physical member mode is initiated through the Toggle Physical Member mode tool found in the Steel Design toolbar (which docked on the left hand side of the screen by default). This “mode” is used when modeling the structure and any parameter or specification added while this tool is toggled on will then only be available for physical member groups.

Tip: Command entries in the tree, material properties, and specifications will be designated with “(Physical)” when added in this mode.

Physical members are then formed or selected using the tools in the Steel Design toolbar. Refer to M. Physical Members (on page 663) for additional information.

Physical Member Restraints

A new parameter has been added for AS 4100 physical members to describe the bracing conditions/locations on a physical member. This parameter describes where the restraint is located along the length of the physical member and the type of restraint on the top or bottom flange.



Engine Physical Member Validation

When creating physical members, the STAAD.Pro graphical interface will check to ensure that the analytical members included in a physical member definition are both interconnected and colinear. However, it is not uncommon for input files to be generated outside of the STAAD.Pro graphical environment. Thus, these checks are now performed by the STAAD.Pro Analysis & Design engine again when an analysis is performed. You will be alerted if either condition is not met.

Tip: The analytical members contained in a physical member definition must be colinear (or, all lying in a straight line). Each adjacent analytical member must be within 5° of one another to meet this condition.

Refer to D2.B.8 Design Parameters (on page 1364) for additional information using the new SGR (steel grade) and LHT (load height position) design parameters for steel design per AS4100. Refer to D2.B.12 Physical Member Design (on page 1372) for additional information on using the PBRACE parameter and performing the design of physical members per AS4100.

AD.2007-07.2.9 SNiP 2.23-81 Steel Design

Several minor enhancements have been made to STAAD.Pro regarding steel design per the SNiP 2.23-81 code.

The following corrections and enhancements were made to the SNiP 2.23-81 steel design code implementation in STAAD.Pro:

- PHI and NIU factors messages extensions explaining different design results cases
- Additional bending check for non axial compression/ tension with small eccentricity
- Compressed steel member design by weakened section
- Full Check by combinations envelope for different sections of each steel member. Print of Analysis results for each member section
- Additional parameters for Steel grade by EC3 in EN 10025-2 steel tables
- Correction of other minor bugs and errors

AD.2007-07.2.10 Geometric Nonlinear Analysis Cycle Control

You are now able to limit the analysis cycle using a displacement limit control. These controls can be found on the Analysis/Print Commands dialog Nonlinear Analysis tab or input manually in the input command file.

The displacement limit control allows you to select a nodal displacement degree of freedom be monitored during a geometric nonlinear analysis. A target displacement is set and, if the number of load steps set is two or greater, the analysis will proceed step-by-step until the target displacement is met or exceeded. This provides you with an additional, practical means of limiting the number of steps used in a geometric nonlinear (GNL) analysis.

Related Links

- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *Nonlinear Analysis tab* (on page 2922)
- *A. To specify a nonlinear analysis* (on page 954)

AD.2007-07.2.11 Jindal Steel Section Database

To add a section from the available JPSL catalog, use the following procedure.

A number of Jindal Power & Steel Limited (JPSL) catalog sections have been added to the section database.

1. Select the **General | Property** page.
2. In the **Properties - Whole Structure** dialog, click **Section Database**.
   The **Section Profile Tables** dialog opens.
3. On the Steel tab, select the **Jindal** entry in the table families list.
4. Select the table, section, and type specification.
5. Click **Add**.

AD.2007-07.2.12* Design per ASME NF 3000 2004 Code

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented per the 2004 edition of this code and the **Design | Steel** page has been updated to allow the design parameters to be defined and assigned.

To perform a steel design per ASME NF 3000 2004

Use the following procedure to specify post analysis steel design code checking requirements for the ASME NF code.

1. Create a model with steel members.

2. Select **ASME NF3000 2004** for the **Current Code** on the **Design | Steel** page.
3. Click **Define Parameters...**.

   The **Design Parameters** dialog opens.
4. Specify parameters as required.
5. Close the **Design Parameters** dialog.
6. Assign parameters to members as needed.
7. Select **Analysis** > **Run Analyze** (or press CTRL+F5).

For more information on the technical requirements of this design code, including the full set of parameters and default values, refer to D1.L.4. ASME NF 3000 - 2001 & 2004 Codes (on page 1338) .

Note: The STAAD Nuclear Code pack is required to perform designs per an ASME NF 3000 code.

AD.2007-07.2.13* Update to ANSI AISC N690 1984 & 1994 Codes

Four new Stress Limit Coefficients (SLC) parameters have been added for designs per ANSI/AISC N690 1984/1994 codes.

These parameters, `SFC`, `SFT`, `SMZ`, and `SMY`, all default to 1.0 and are used to control the interaction equations in Section Q1.6 of the ANSI/AISC N690 1984/1994 codes.

Equations Q1.6-1a, Q1.6-1b, Q1.6-2 and Q1.6-3 of code ANSI/AISC N690 1994 will be rewritten as follows:

- Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

  $$f_a/F_a + C_{my}f_{by}/[(1 - f_a/F'_{ey})F_{by}] + C_{mz}f_{bz}/[(1 - f_a/F'_{ez})F_{bz}] \le 1.0$$

  and Q1.6-1b

  $$f_a/(0.6 F_y) + f_{by}/F_{by} + f_{bz}/F_{bz} \le 1.0$$

  when, $f_a / F_a > 0.15$, as per section Q1.6.1 of the code.
- Otherwise, equation Q1.6-2 must be satisfied:

  $$f_a/F_a + f_{by}/F_{by} + f_{bz}/F_{bz} \le 1.0$$
- Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q 1.6-1b:

  $$f_a/(0.6 F_y) + f_{by}/F_{by} + f_{bz}/F_{bz} \le 1.0$$

Refer to D1.K.2.1 Design Process (on page 1303) for additional information on using the ANSI N690 1984 and 1994 codes.

AD.2007-07.2.14* Load Combination Enhancements

It is now possible to refer to a previously defined load combination within a new load combination. For example, a SRSS combination of individual response spectrum cases can now be referenced in a load combination along with dead load, live load, etc.

There are no changes to the input file syntax. Load combination definitions may now refer to existing load combination numbers along with load case numbers.

Note: There is no limit to the amount of load combination “nesting” which can be done in STAAD.Pro, other than the total limit of load cases and load combinations allowed by the program.

Related Links

- *TR.35 Load Combination Specification* (on page 2616)
- *Define Load Combinations dialog* (on page 2836)
- *M. To define a new load combination* (on page 886)

AD.2007-07.2.15* Enhancement to Maximum Number of Response Spectrum Load Cases

STAAD.Pro now supports up to 50 response spectrum load cases, instead of the previous limit of four.

Refer to TR.32.10.1 Response Spectrum Analysis (on page 2498) for additional information on using Response Spectra.

AD.2007-07.3 Features Affecting Post Processing

Several new features have been added and existing features have been modified in the Post Processing modes. These are explained in the following pages.

AD.2007-07.3.1 RAM Connection V8i (SELECTseries 1) Support

The enhancements included in Bentley's RAM Connection V8i (releases 6 and 7) are now available in STAAD.Pro. This includes new connection types, new codes, and design for seismic loads.

Some of the new features and enhancements include:

- Now compatible with versions of RAM Connection V8i up through release 7.0 (SELECTseries 3).
- British Design Code - Connection design per BS5950-1:2000 (British standard) has been added. This code can now be selected along side AISC codes.
- AISC Seismic Provisions - The seismic provisions of AISC 341-05 have now been added for connections per AISC codes.
- Seismic Frame Management - A new Seismic Frames page has been added to assign the lateral seismic resisting system classification to frames for connection design. This page is also used to add plastic hinge locations to beam members.
- Base Plate Design is now available for AISC (ASD & LRFD) connections. Column base plates are available in the Smart Connections dialog and Column-Brace gusset base plates are available in the Gusset Connections dialog.
- The RAM Material dialog has been expanded to accommodate for various materials from different countries. UK steel, bolt, and weld types have been added, as well as concrete and anchor bolt materials for US base plate design.

  Note: A notification message may be displayed when selecting the RAM Connection mode that you need to provide some additional material properties.
- Several new selection methods have been added to the **Select** > **By Joints** > sub-menu.
- Reports have been enhanced with code references and display of formulas used.

AD.2007-07.3.2 RC Designer

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-07.3.2.1 ACI 318 Metric

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-07.3.2.2 GB50010

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-07.3.2.3 IS456 with Seismic Design per IS13920

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

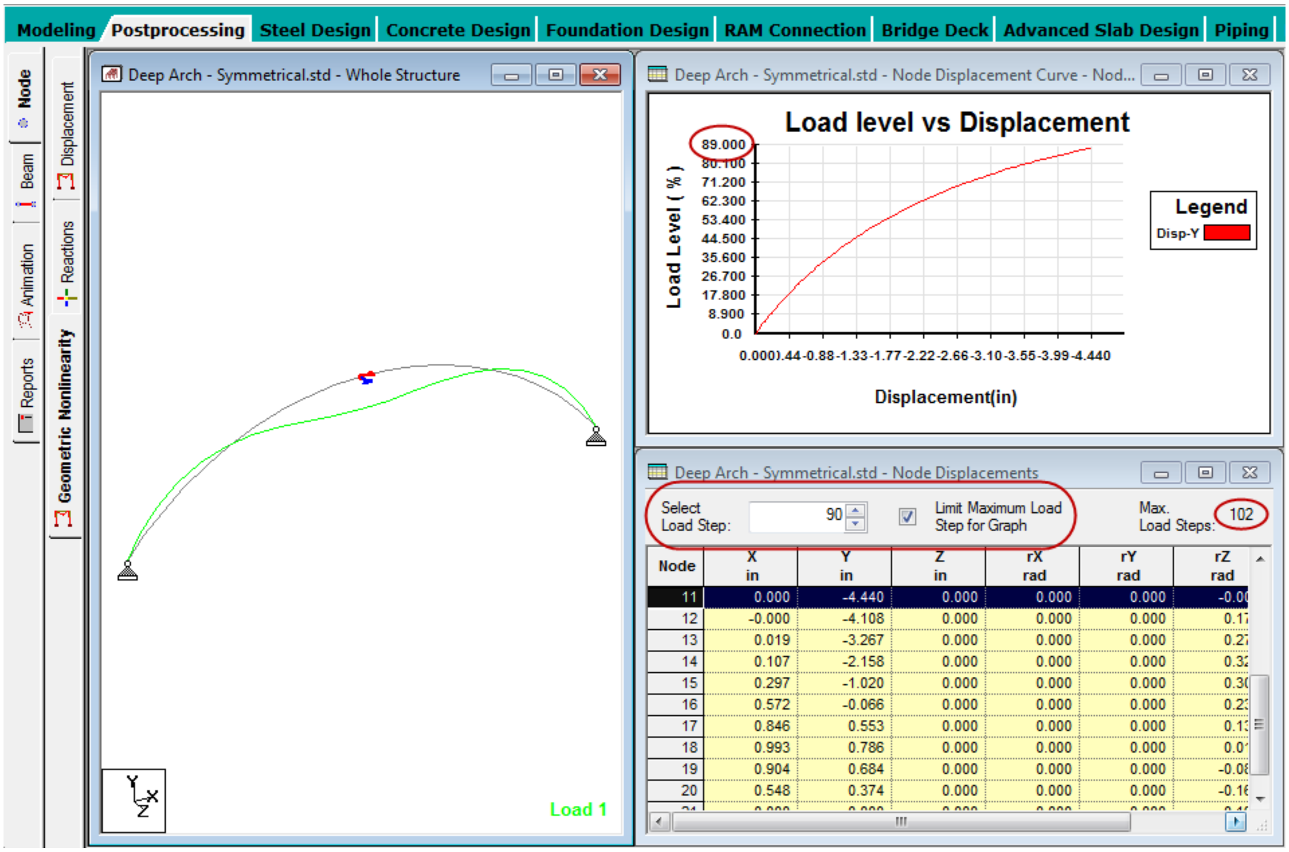
AD.2007-07.3.3 Enhanced Geometric Nonlinear Post Processing

The graphical display of the results of a geometric nonlinear analysis have been improved. The Graphs are now easier to read and a new control has been added to limit the maximum Load Step displayed.

When a structure is analyzed with a geometric nonlinear analysis, the nodal displacements can be viewed by selecting the Node **|** Nonlinearity page in the Post-Processing mode. The Node Displacements Curve has been enhanced and a new control has been added to the Node Displacements table to limit the maximum Load Step plotted.

In the event that a load level is specified which exceeds the non-linear buckling capacity of a structure, the analysis performed by STAAD.Pro will produce exceedingly large post-buckling displacements. This signifies that the load steps in this post-buckling level are beyond the scope of designed performance of the STAAD engine (and likely beyond the load level and effects intended by the engineer). In these cases, the scale of the non-linear load displacement curve was did not adequately display the pre-buckling characteristics due to the very large scale required to display the post-buckling displacements.

Now, in the event of a large post-buckling load, the maximum load level scale of the Load Level vs. Displacement graph can be limited to the currently selected load step by selecting this option on the Node Displacements table, thus allowing you to see the pre-buckling behavior.



AD.2007-07.3.4 AutoPIPE V8*i* (SELECTseries 2) and PipeLink support

STAAD.Pro is now capable of two-way data exchange with AutoPIPE via a new PipeLink utility. Additionally, the tools for transferring loads between the pipe model and the structural model have been enhanced such that you can now select, update, and remove loads to be applied to the structural model.

Note: For details on the updates to Bentley AutoPIPE V8i, refer to the documentation included with that product.

Some of the new features and enhancements include:

- The integration between STAAD.Pro and AutoPIPE has been enhanced to allow bi-directional transfer of data between STAAD.Pro and AutoPIPE. This is accomplished through a new plug-in program called PipeLink, which is used to generate common database files for use in both AutoPIPE and STAAD.Pro. Export Revised Model dialog (on page 2961) back to AutoPIPE is now possible.
- The system now allows pipe models to be imported from a number of databases and these models may all be stored locally. The active model can be selected from the Pipe Model dialog (on page 2962).
- The loading transfer process has been significantly revised. The mappings between the pipe loadings and the Transfer Pipe Reactions to Structure Model dialog (on page 2970) are recorded so that these structure loadings may be revised or removed rather than the previous once-only method. Further, loads can now be modified individually.
- Connections can now be individually selected to transfer moment to the supporting STAAD model structure by selecting the Pipe Supports table (on page 2968) option at each connected Pipe Node.

- The program now connects pipe supports – rather than pipe nodes – to the structure. This allows both node and support labels to be used. This allows additional filters in the Support Connection Wizard (on page 2964) and also allows STAAD.Pro to import multiple supports at a single support node.
- Pipe supports are now graphically displayed with icons which reflect the nature of the pipe support. The display of these icons, along with other pipe model elements, can be controlled through a new tab in the Diagrams dialog Pipe tab (on page 2972).

AD.2007-07.3.5 Transverse IRC Loading in STAAD.Beava

For the specific bridge width, IRC (Indian Road Congress) chapter 6-2000, table 2, clause 207.4 defines the rules to combine the live loads. This new feature allows you to either use these IRC live load rules or use an iterative, custom method. If the IRC rule option is selected, this function uses the appropriate live loads and number of design lanes and the generates all the possible load combinations as stated for this particular bridge width. Otherwise, you can select a specific live load and design lane to generate combinations.

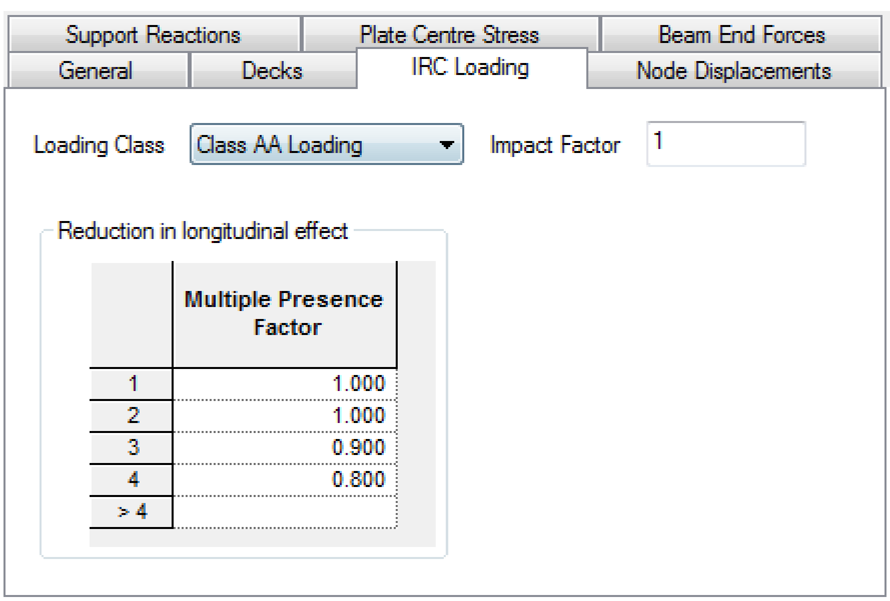
Loading rules per IRC Chapter 6 are applied in much the same way as previous codes. The defined roadway for the selected deck(s) is divided into design lanes and the selected load class is applied to the structure achieve the specified actions.

To specify loading per IRC Chapter 6

1. Open an analyzed bridge model in the Bridge Deck mode.
2. Create Deck and Roadway definitions.
3. Generate influence surfaces for the structure.
4. Select **Loading** > **Run Load Generator...**.

   The Load Generation Parameters dialog General tab opens.
5. Select IRC Chapter 3 for the Design Code and select the appropriate Limit State.

   The <code> tab updates to display IRC Loading.
6. Select the **IRC Loading** tab.



7. Select the appropriate **Loading Class**.

> **Note:** Combinations of the AA, B, and 70R vehicles have been added to the included vehicle definitions. These may be reviewed in the Vehicle Database dialog.

8. (Optional) Specify an impact factor or modify the **Multiple Presence Factors** as needed.
9. Specify the decks for consideration on the Decks tab and the load effects to dictate load placement on the Node Displacements, Support Reactions, Plate Center Stresses, and Beam End Forces tabs.
10. Click the **OK** button.

    The program places the selected loads in design lanes to produce the maximum or minimum effects requested. A text file containing a summary of the generated loads and corresponding effects is opened in a text editor for review.

Related Links

- *IRC Specific Parameters* (on page 2986)

AD.2007-07.3.6 STAAD.foundation V8i Integration

The export of support geometry and reactions to STAAD.foundation V8i can now be initiated from within STAAD.Pro using the Foundation Design mode. This feature is similar to the **Import STAAD.Pro File** capability included in STAAD.foundation.

When selected, the Foundation Mode opens the Foundation page which contains a view of the whole structure and the Foundation Design Options dialog.

From here, you can select to include all supports, you can graphically select supports, or you can specify a list of support numbers for exporting to a STAAD.foundation project. Similarly, the load cases from the analysis are listed for inclusion in the STAAD.foundation project.

> **Tip:** Models containing a large number of supported nodes or load cases may result in slow performance on older computer hardware. Exporting a limited set of data can be used to improve performance in STAAD.foundation in these cases.

Planned future enhancements also include the export of mat foundations modeled in STAAD.Pro for design in STAAD.foundation.

AD.2007-07.3.7 Additional Section Databases in RAM Connection mode

Steel section databases for the following countries are now available for use when design connection in RAM Connection mode:

- Indian
- European
- Japanese
- Australian

> **Note:** Connection design is only performed per the US and British codes available in RAM Connection.

AD.2007-07.4 Additional Features

The following features have yet to undergo testing and are presented "as is."

> **Note:** Items labeled with an asterisk (*) were added in the QA&R release of V8i (SELECTseries 2) (Build 20.07.07.32).

Beta Features

The following features have yet to undergo testing and are presented "as is."

- (None)

AD.2007-07.4.1* Design of Class 4 “Slender” Sections in IS800:2007

The design of slender classified sections (only rolled or welded I sections) per IS:800-2007 has been added to STAAD.Pro.

The IS:800-2007 code does not provide any clear guidelines about what method should be adopted for the design of slender section. The “Flange Only” concept has been adopted where it is assumed that flexure is taken by the flanges alone and the web will resist shear with adequate shear buckling resistance. This means that the flange elements must be non-slender with slender web element to qualify for slender section that can be designed. If any of the flanges become slender, the design will not be performed for Bending and a warning message is displayed.

Refer to D8.A. Indian Codes - Steel Design per IS 800 - 2007 (on page 1607) for additional information on the design procedures used for slender sections for IS800:2007 as well as a verification example problem.

V8*i* (SELECTseries 1)

The Software Release Report for STAAD.Pro contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8i (release 20.07.05) This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-06.1 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-06.1.1 CIS/2 Translator Update

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced for the transfer of models into 3D modeling, such as Intergraph SmartPlant® 3D (SP3D).

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models.

In previous versions of STAAD.Pro, CIS/2 files could not be imported into an existing STAAD file. The import process has now been updated so that new STAAD input files can be created or existing input files updated from a CIS/2 file.

STAAD.Pro will retain all relevant information generated by SP3D –including the object IDs (GUIDs)– when importing CIS/2 files. Further modeling operation will be done in STAAD.Pro which includes special purpose load generations, analysis, design and member selections and modifications. You may then export out to a CIS/2 STEP file and retain all information inherited from SP3D STEP file and addition/deletion/modification information performed in STAAD.Pro.

Using import and export of STEP files in SP3D, further modification can be made in SP3D and the STAAD.Pro model can be updated its model. Only geometry, member properties, boundary condition information are within the update scope of STAAD.Pro. While updating the STAAD.Pro model, no other information will be considered. This round-trip process can be repeated an unlimited number of times

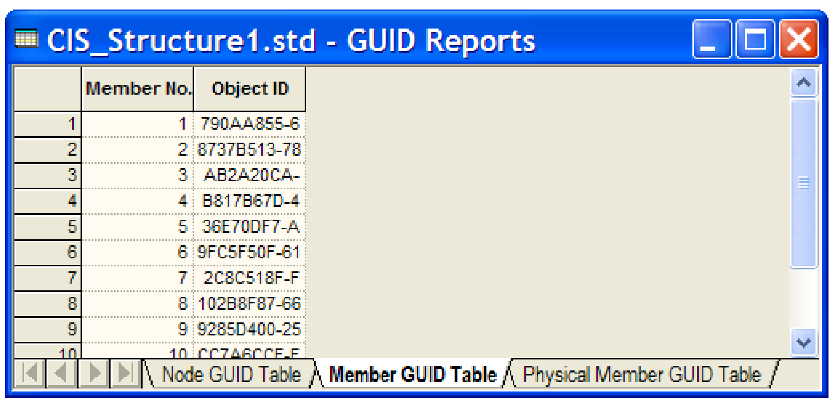
Additionally, two new VBS Macros have been included in the User Tools to aide in the verification of model integrity before and after update operations using the Import dialog. These may be found in the User Tools drop-down menu in the File toolbar or under **Tools** > **User Tools**.

New User Tools (VBS Macros)

Two VBS macro are developed to assist users to confirm the model integrity before and after the update operations.

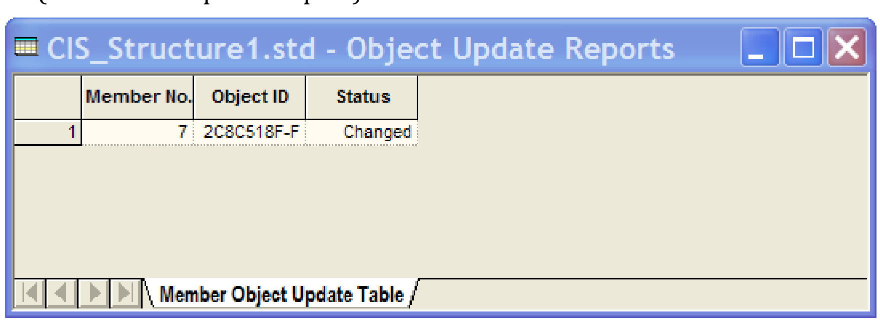
List Object GUIDs

Opens the GUID Report tables. This is useful to verify that the GUIDs are same as those in SP3D.



CIS/2 Object Update Report Tool

Used to capture model state before an update operation (Create Pre-Update Report) and then same tool can be invoked again after the update process to generate a report showing what exactly has been updated (Create Post-Update Report).



Related Links

- *I. To import a CIS/2 file* (on page 2078)
- *I. To export to a CIS/2 file* (on page 2080)

AD.2007-06.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-06.2.1 ANSI/AISC N690-1984 Design Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the *ANSI/ AISC N690-1984: Nuclear Facilities - Steel Safety-Related Structures for Design, Fabrication, and Erection*.

The parameter `CODE AISC N690 1984` is used to initiate code checking per ANSI/AISC N690-1984.

The full details for this code - including parameters, commands, and technical background - are in D1.K.2. ANSI/ AISC N690-1984 Code (on page 1303).

To use the ANSI/AISC N690 1984 code

1. In the modeling mode, select the **Design | Steel** page.
2. Select **AISC N690 1984** in the **Current Code** drop-down list.

AD.2007-06.2.2 Update to Russian Concrete Design

The Russian SNiP concrete design routines have been updated to accommodate new reinforcement and concrete class definitions. In order that these new classes can be assigned to members that are to be designed, the following changes have taken place in the RCL, BCL, and RHS parameters.

Refer to D13.A. Russian Codes - Concrete Design Per SNiP 2.03.01-84* (on page 1866) for additional information.

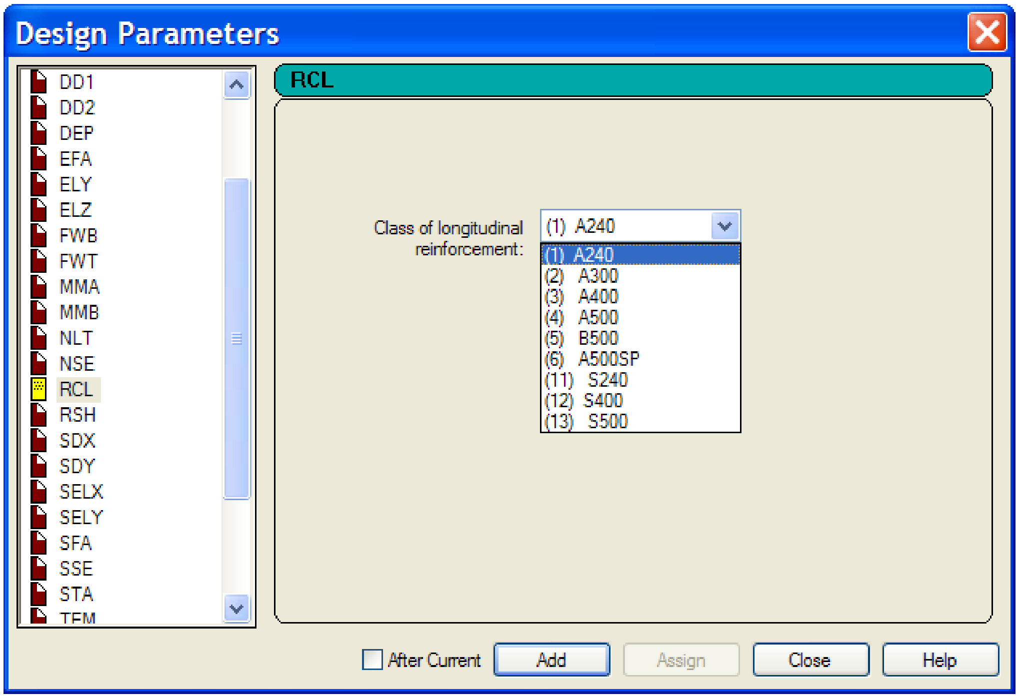
Additionally a few other minor updates have been incorporated to ensure axial tension is ignored in column design and that the provided area of steel in both directions is not less than the minimum.

To select reinforcement or concrete class definitions

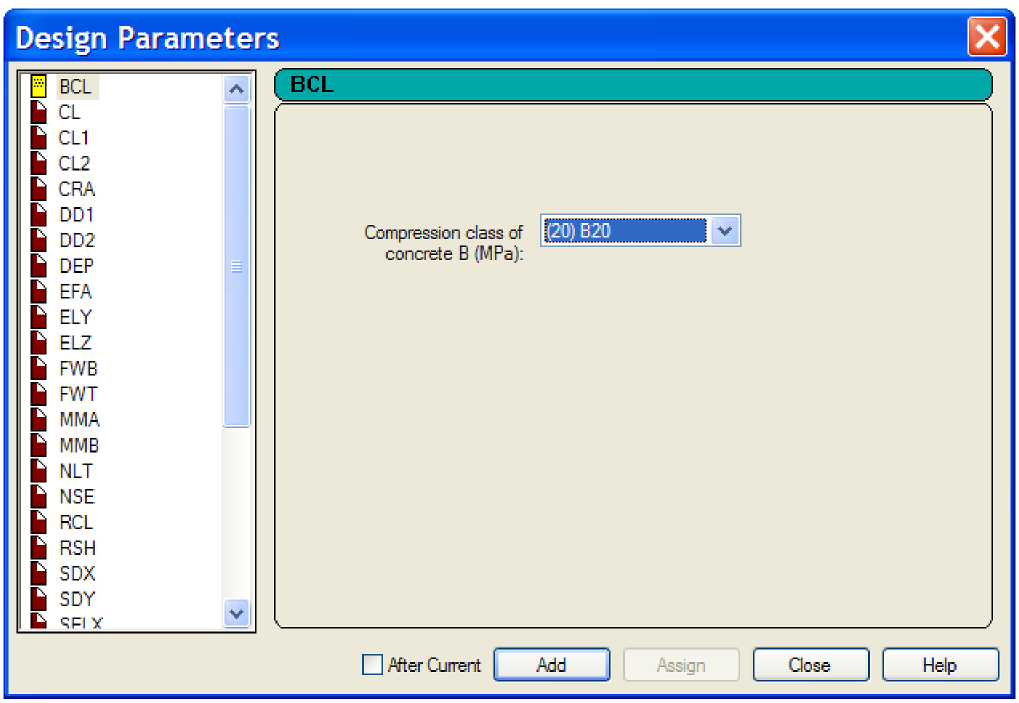
1. In the modeling mode, select the **Design | Concrete** page.

2. Select **SNiP 2.03 01-84** in the **Current Code** drop-down list.
3. Click the **Define Parameters** button below the model outline panel.
4. The **RCL**, **BCL**, and **RSH** parameters have the new definitions available. Refer to D13.A.2 Design Parameters (on page 1867) for additional information.

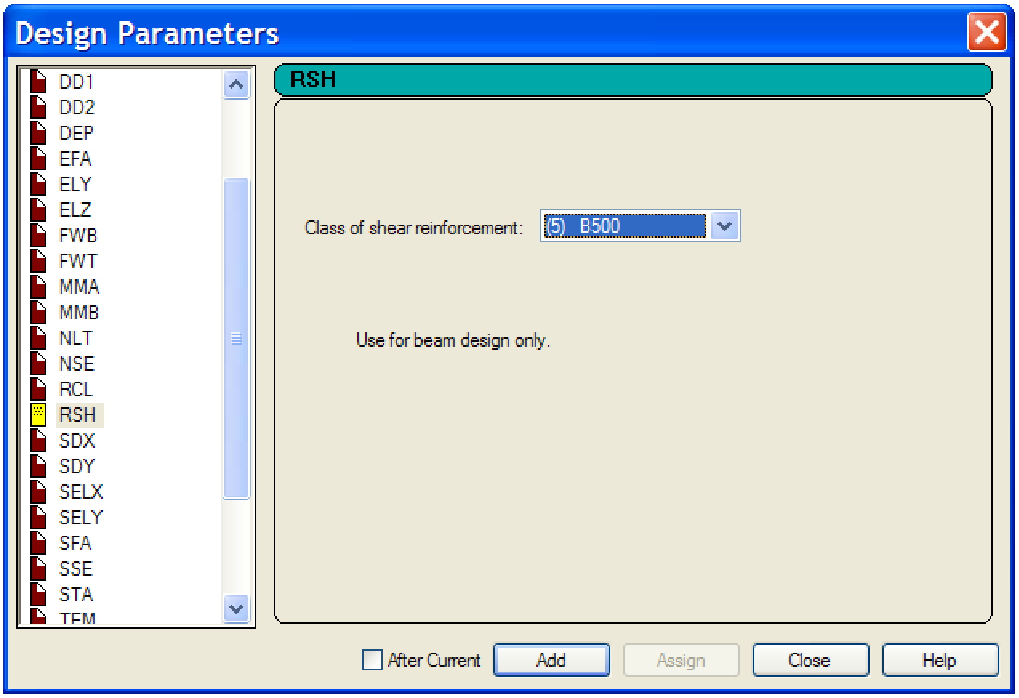
   Reinforcement Class for longitudinal reinforcement (RCL):



Compression class of concrete B (BCL):

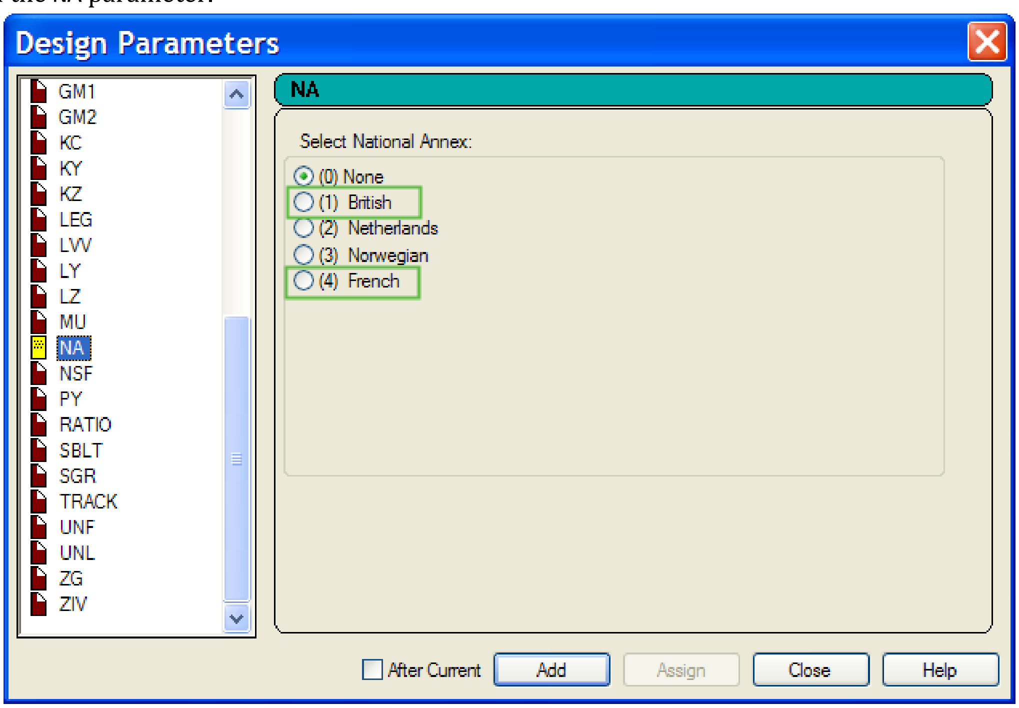


Reinforcement class for shear reinforcement (RSH):



AD.2007-06.2.3 Eurocode 3 National Annex

Two country's National Annex to Eurocode 3 have been incorporated into the Steel design module in STAAD.Pro: United Kingdom and France. As with the other National Annexes to EC-3, this implementation will make use of the NA parameter.



Caution: The GB1 parameter (which, in fact was common to the base EC-3 and was a reminiscent of the previous DD ENV implementation of EC-3) has been removed. Hence any legacy STAAD files that have the GB1 Parameter defined will need to be revised to take out this parameter as it is no longer valid as per the latest EN1993.

AD.2007-06.2.3.1 United Kingdom National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-UK NA Steel Design, use the following procedure.

The UK National Annex document referred to is "NA to BS EN 1993-1-1:2005".

1. In the Modeling mode, click the **Design | Steel** page.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters** button.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(1) United Kingdom**.
6. Click **Add**.

   This will insert the following command into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 1
```

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516)

AD.2007-06.2.3.2 French National Annex to Eurocode 3 (EN 1993-1-1:2005)

To Initiate a EC3-French NA Steel Design, use the following procedure.

The French National Annex document referred to is "Annexe Nationale a la NF EN 1993-1-1:2005".

1. In the Modeling mode, click the **Design | Steel** page.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click the **Define Parameters** button.
   The **Design Parameters** dialog opens.
4. Select the **NA** parameter in the list box.
5. Select the option for **(4) France**.
6. Click **Add**.

   This will insert the following command into the STAAD input file:

```
CODE EN 1993-1-1:2005
NA 4
```

Note: For additional information, please refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) and D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516)

AD.2007-06.2.4 Chinese Static Seismic Loading

A simplified base shear method of the seismic load generation for the Chinese GB50011-2001 code has been added to STAAD.Pro V8i.

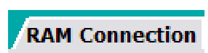
This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

Related Links

- *TR.31.2.5 Chinese Static Seismic per GB50011-2001* (on page 2369)
- *Add New Seismic Definitions dialog* (on page 2895)
- *M. To add a seismic load definition* (on page 857)

AD.2007-06.3 Features Affecting the RAM Connection Design Mode

Several new features have been added and existing features have been modified in the RAM Connection Design Mode. These are explained in the following pages.



Note: Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-06.3.1 RAM Connection V8*i* Support

The enhancements included in Bentley's RAM Connection V8*i* (release 5.5) are now available in STAAD.Pro. Additionally, the connection assignment and design process has been streamlined within STAAD.Pro. Now, joints and connections can be automatically assigned for a set of selected members and connection designs can be grouped together. Additionally, when a connection is edited using the RAM Connection pad, those changes will be saved in the STAAD.Pro model design.

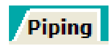
Connections are designed in the newly updated RAM Connection Mode by creating “Joints,” from the geometry, section properties and forces resulting from the analysis and assigning a design brief made up of connection templates. A suitable connection design, if one is available, will be reported once you have selected the appropriate connection templates.

In previous releases of STAAD.Pro, you were able to only select and assign a connection to individual joints. Now, any number of joints may be selected and designed. Further, connection design is performed automatically for you once appropriate templates have been selected for the selected joints. These enhancements greatly reduce the time required for connection design in models of all sizes.

Tip: The selected load envelope is now used for all connection designs, instead of per design brief as in previous versions of STAAD.Pro.

AD.2007-06.4 Features Affecting the Piping Mode

STAAD.Pro can utilize the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.



AD.2007-06.4.1 AutoPIPE Integration Enhancements

An enhancement to how pipe/structure connections are assigned has been made to the Piping module. A new Support Connection Wizard is available to allow you to add multiple supports to the entire model or a subset of a model, based on some general parameters.

Note: Pipe-structure links are not part of the undo system. Nodes created at the end of the wizard will be removed by an undo but the links are not changed.

Within the Piping mode, after the pipe model has been loaded, you will be able to call up a modeless connection wizard. The wizard will take you through the following steps:

1. Defining the set of pipe nodes to consider.
2. Defining the set of structural beams to consider.
3. Defining the set of structural nodes to consider.
4. Setting range and tolerance parameters.
5. Previewing and accepting the determined connections.

Potential connections will be determined after step 4 and fully created at the end of step 5.

Potential connections will be determined by finding the closest beams and closest nodes to each pipe node. In the previewing stage the closest five items, along with a “to ground” option, will be available as options to you, sorted by distance.

Parameter Page / Connection Finder

The connection finding routine runs in two parts. First looking at beams (length of perpendicular from beam) and then looking at nodes (straight line length). The five closest found connectable points are saved to be presented on the results page. In order to provide control over the connection finder, several parameters are available to you for editing. The default values of distance-based parameters will depend on the base unit of STAAD.Pro.

General

- Max. Range: double: default 2m / 6’ : Potential connections beyond Max. Range will be discarded.
- Insert nodes into beams at connection points: default true : This parameter effects the connection of the structures rather the point finding algorithm. If set true a new node will be created at each intermediate beam point and the connection made to that rather than to the beam itself. If the algorithm finds new node points within “End Tolerance” of each other then only one new node will be added.

Beam

- End Tolerance : double : default 5cm /2" : To allow for differences in precision and to avoid very short beam breaks this parameter will determine at what distance from the node the perpendicular will be considered to be at the node itself.

- Allow Non-Perpendicular Connection at End Nodes: Boolean : default true: This is only really relevant if the node subset does not explicitly include the nodes at the end of members in the beam subset. If set "true" the beam end nodes will be included in the node search. If set 'false' the end nodes of a given beam will only be considered for connections perpendicular to the beam, unless they have been explicitly added to the node subset.

This page has no effect on structure diagrams. The connection finding routine is run when advancing from this page.

Related Links

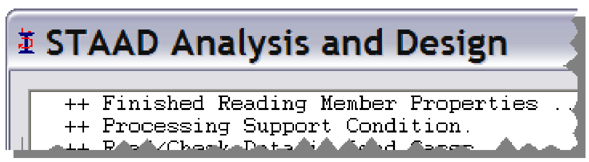
- *M. To use the Support Connection Wizard* (on page 904)
- *Support Connection Wizard* (on page 2964)
- *I. To export structure date to AutoPipe* (on page 2080)

V8*i* (release 20.07.05)

The latest What's New document for STAAD.Pro V8*i* contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro V8*i* (2007 build 04). This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-05.1 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.



AD.2007-05.1.1 Geometric Nonlinear Analysis

The range of analysis options has been supplemented with a new solution to account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

Note: The nonlinear analysis command is available with the Advanced Analysis license.

Related Links

- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *A. To specify a nonlinear analysis* (on page 954)

AD.2007-05.1.2 IS 800:2007 Steel Design

The Indian Bureau of Standards has released a new version of the design code for the design of steel structures, known as IS 800:2007. This replaces the previous version of the code (also supported in STAAD.Pro), IS 800:1984, which was versified again in 1998. This is a new approach to steel design and is based on the limit state design rather than a stress based design of the old code.

The Steel Design section has been enhanced to include design per IS 800:2007. Both section checking and selection routines are supported.

The design will follow the same process as used by all other steel design codes currently available in STAAD.Pro. The design of each member is controlled through a set of parameters that have been added to the GUI Steel Design Dialog under the title **IS 800:2007**.

The design engine will allow all standard section database sections and User Table sections to be designed.

The design process followed:

1. Check slenderness
2. Check classification
3. Check tension forces
4. Check compression forces
5. Check bending Forces
6. Check interaction

The results will be:

1. output to the `ANL` file
2. available in the member query
3. available in the post processing mode in the Design Results table

Note: *Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).

Related Links

- *D8.A.4 Design Parameters* (on page 1617)

AD.2007-05.1.3 Eurocode 3 Includes National Annex

A number of countries that have signed up to the replace their current steel design standards with the Eurocode, EN 1993-1-1:2005, known commonly as Eurocode 3, have published their National Annex documents. These documents make small changes to the base document and STAAD.Pro has been updated to incorporate some of these National Annex documents. Currently, the Dutch and Norwegian National Annexes have been added to the STAAD.Pro engine.

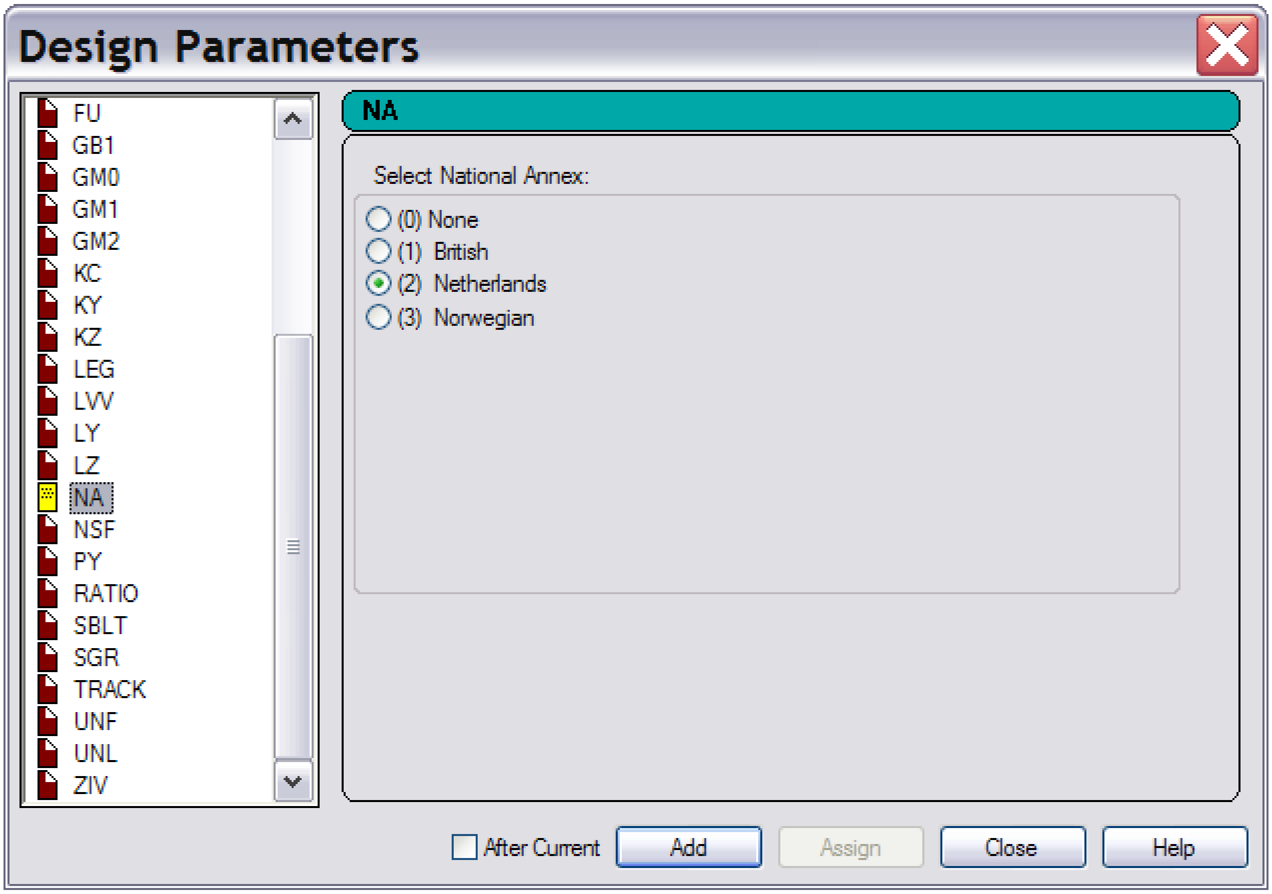
A new parameter, `NA`, that sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods.

The output file printout has been updated to indicate which National Annex (if any) has been used in a code check / select process. (For all `TRACK` settings)

To specify checks per a National Annex

In order to include additional check specified by a National Annex, do the following:

1. From the Modeling mode **Design | Steel** page, select **EN 1993-1-1:2005** from the current code list.
2. Click **Define Parameters** to launch the Design Parameters dialog.
3. Select the parameter **NA** from the list.



4. Select the radio button for the National Annex you wish to use; or leave as Basic in order to use EC3 without additional checks.
5. Click **Add** to add the NA parameter to the code check.
6. Click **Close** to dismiss the dialog once parameter definitions are complete.

A design performed to the new Eurocode 3 National Annex is displayed in the output file (`*.ANL`) with the following header, in addition to the base EC3 output:

```
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE NOTED)

MEMBER      TABLE        RESULT/   CRITICAL COND/      RATIO/      LOADING/
                           FX            MY              MZ        LOCATION
=======================================================================


  EC-6.2.5       0.723     1      0.0      50.0     0.0    50.0     0.0
  EC-6.2.6-(Y)   0.284     1      0.0      50.0     0.0    50.0     0.0
  EC-6.3.2 LTB   0.757     1      0.0      50.0     0.0    50.0     0.0

  ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NEN-EN 1993-1-1/NB] (units- kN,m):

  EC CLAUSE              NA-CLAUSE         RATIO    LOAD        FX        VY        VZ
  EC-6.2.8-(Y)   NEN-6770-Eq.11.3.1      0.689     1         0.0      50.0      0.0
  EC-6.2.10(Y)   NEN-6770-Eq.11.3.1      0.689     1         0.0      50.0      0.0
  EC-6.2.10(C)   NEN-6770-Eq.11.3-31     0.550     1         0.0      50.0      0.0
   Torsion and deflections have not been considered in the design.
```

Note: The previous, development edition of Eurocode 3 is included as the Code EN3 DD.

AD.2007-05.1.4 Eurocode 8

Eurocode 8: Part 1 [EN 1998-1-1:2004] contains specific requirements and recommendations for building structures that are to be constructed in seismic regions. Essentially, these fundamental requirements have been provided to ensure that the structures can sustain the seismic loads without collapse and also – where required – avoid suffering unacceptable damage and can continue to function after an exposure to a seismic event.

As with all Eurocodes, a National Annex Document should accompany the use of Eurocode 8 in each of the European nations.

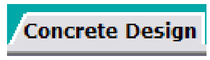
AD.2007-05.1.5 AIJ Concrete Design Update

An old version of Japanese concrete code based on the AIJ standard for structural calculation of Reinforced Concrete Structures (1985 edition) was previously implemented in STAAD.Pro batch mode design. Recently, critical errors in the column design and beam design were discovered and corrected. In addition, the AIJ concrete design has been udpated to incorporate the latest AIJ standard for structural calculation of Reinforced Concrete Structures (1991 edition). .

Generally, this implementation will be invisible to you. The 1985 edition of the AIJ has been completely replaced by the 1991 edition for the design of concrete members per the AIJ. The existing commands as described in D9.A. Japanese Codes - Concrete Design per 1991 AIJ (on page 1713) are applicable to the 1991 edition.

AD.2007-05.2 Features Affecting the Concrete Design Mode

An existing feature has been modified in the RC Designer section of the program, also known as the Concrete Design mode. This is explained in the following pages.



AD.2007-05.2.1 RC Designer Member and Envelope Import

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

V8*i* (release 20.07.04)

The Software Release Report for STAAD.Pro V8*i* contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2007 build 03. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-04.0 New Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-04.0.1 ProjectWise Integration

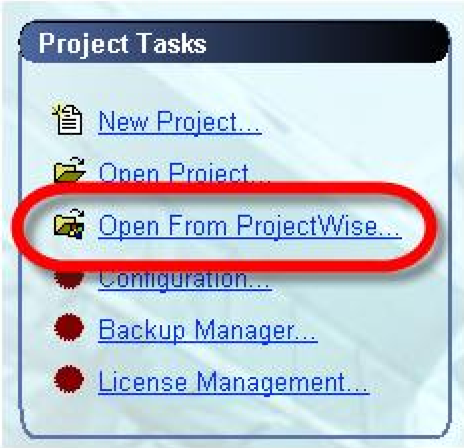
ProjectWise is an engineering project team collaboration system which is used to help teams improve quality, reduce rework, and meet project deadlines. One of the major pieces of functionality provided by ProjectWise is an Integration Server which allows data to be managed and shared across a distributed enterprise.

STAAD.Pro has been enhanced so that the model STD data file can be managed on a ProjectWise server.

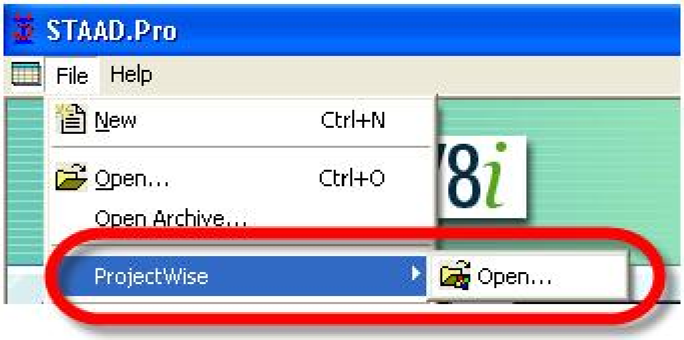
Installation and management of a ProjectWise server is beyond the scope of this document and should be obtained from the ProjectWise installation.

A local ProjectWise client should be installed which allows access to ProjectWise repositories.

When STAAD.Pro is launched, the option to open and check out a STAAD.Pro STD file from a ProjectWise repository is made available from the Project Tasks on the Start Page thus:



This is also available from the File menu while still on the Start Page prior to opening a model:

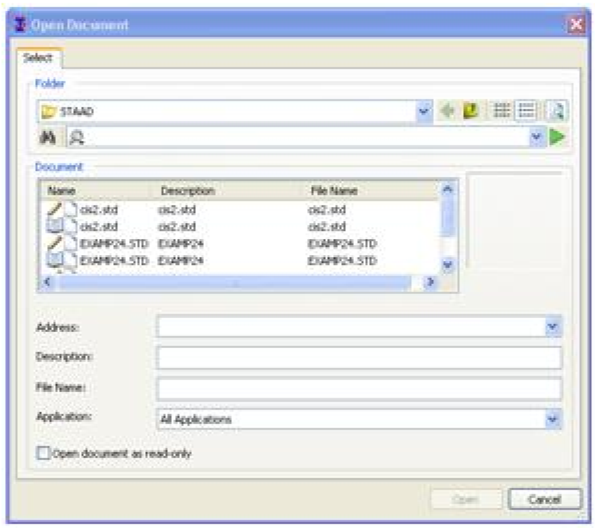


If a suitable ProjectWise client is not installed, then the link on the Start Page is shown as unavailable with a red line through the icon thus:

As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials thus:

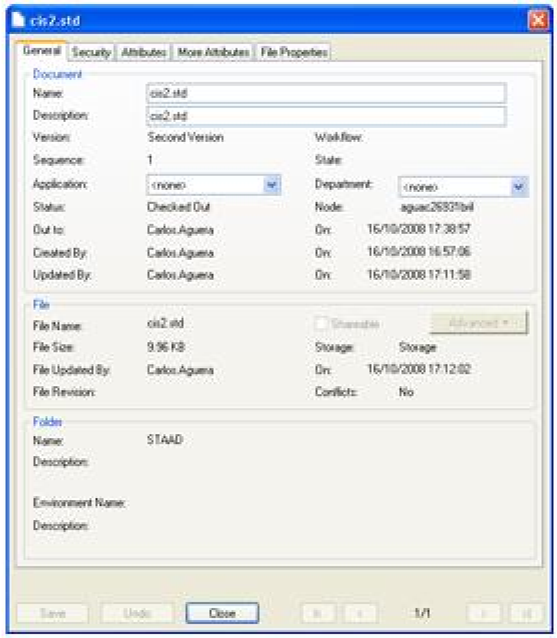
Files that are accessed from a ProjectWise server are “Checked Out” and stored locally during the STAAD.Pro session until the file is closed and then it is returned to the server.

The first time that a successful link to a ProjectWise server is established, a location in which check out files are to be stored locally and additionally, where all the auxiliary data files are stored whist STAAD is running is required. Afterwards and on all future occasions, the ProjectWise open dialog presented is then presented where the repository can be navigated and filtered as defined in the ProjectWise documentation.

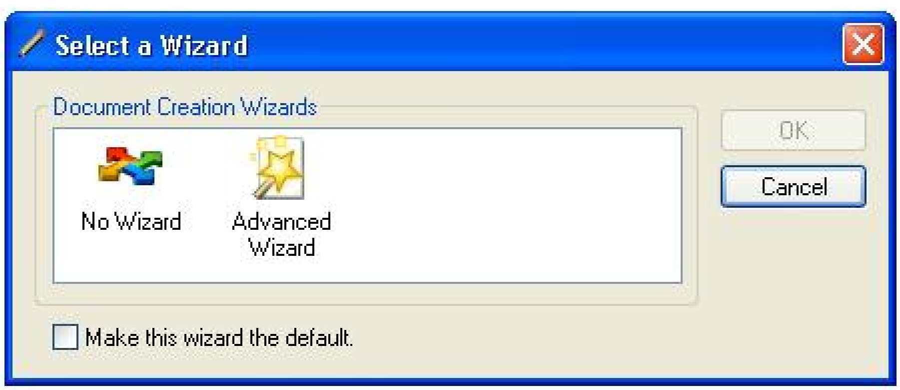


Note the significance of the icons next to the STAAD filenames. These indicate the status of the file such as the current document, checked out to you, or locked as checked out to some other user. Refer to the ProjectWise documentation for a full description of each icon.

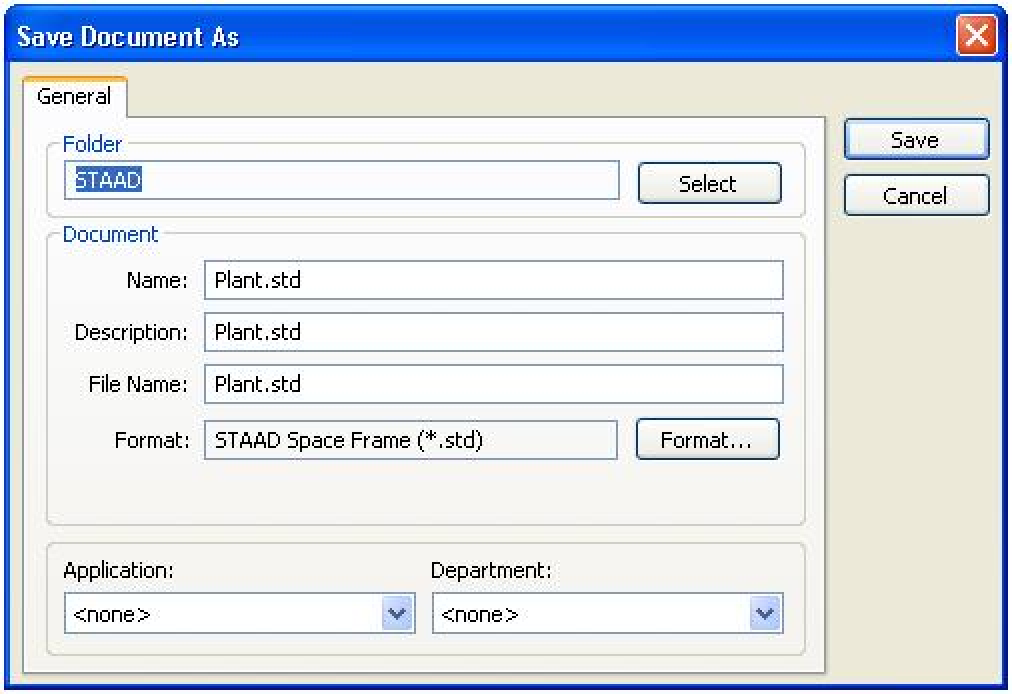
With a file checked out and loaded in STAAD.Pro, it is possible to see the ProjectWise Properties, by selecting the option from the ProjectWise toolbar or File menu:



If working on a STAAD model that has not originated from a ProjectWise server (e.g., starting a new file) and it has been decided that it needs to be added to a repository, then at any time whilst working in the STAAD.Pro environment, clicking on the toolbar option Add to ProjectWise server, or equivalent **File** > **ProjectWise** menu option will launch the following dialog:



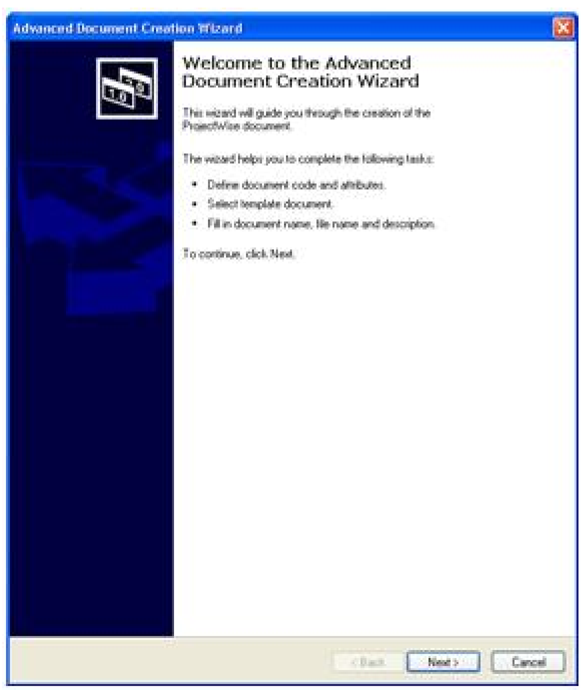
Selecting the option **No Wizard** offers the following dialog into which the file details can be entered.



Clicking on Save, adds this model to the repository, but indicates its status as checked out until the file is closed in STAAD.Pro and the model checked back in.

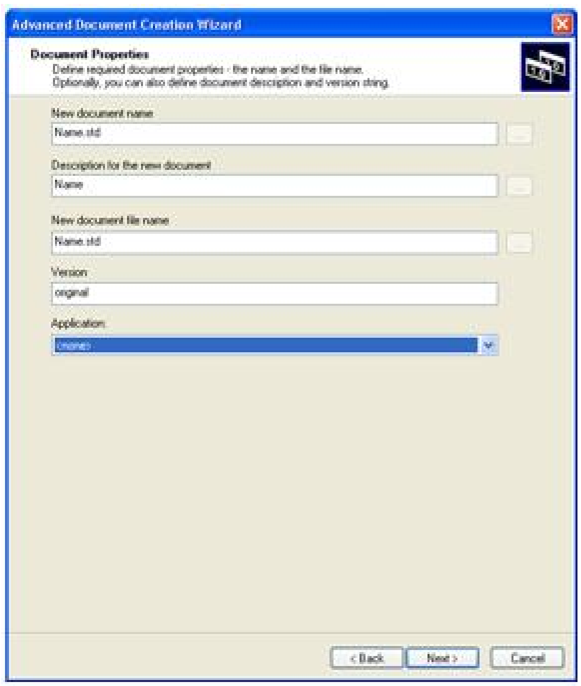
Alternatively, by selecting the **Advanced Wizard** option the data needed to define the ProjectWise data file is presented in the following four steps:

1. Advanced Wizard



2. Select target Folder

3. Document Properties



4. Create the document

Creates the document and marks it as checked out until the file is closed and checked in to the server.

Four integration functionalities have been added. These are:

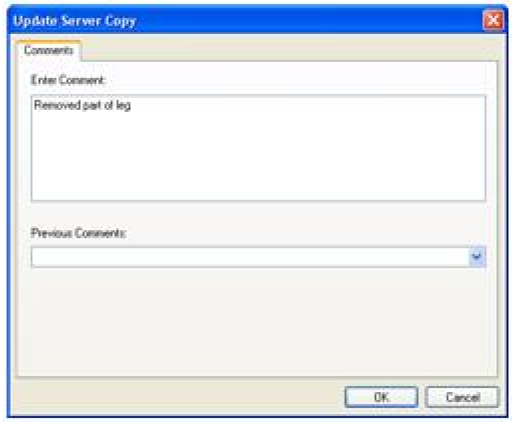
- Open a STAAD model from a ProjectWise repository.
- Save a local STAAD model into a ProjectWise repository.
- Update an existing model from ProjectWise.
- Review model properties (meta-data) which has been opened from a ProjectWise repository.

Note that access to all of these functionalities is available from ProjectWise sub-menu under the general **File** menu described below.

Saving changes of a Checked Out model Back on the Server

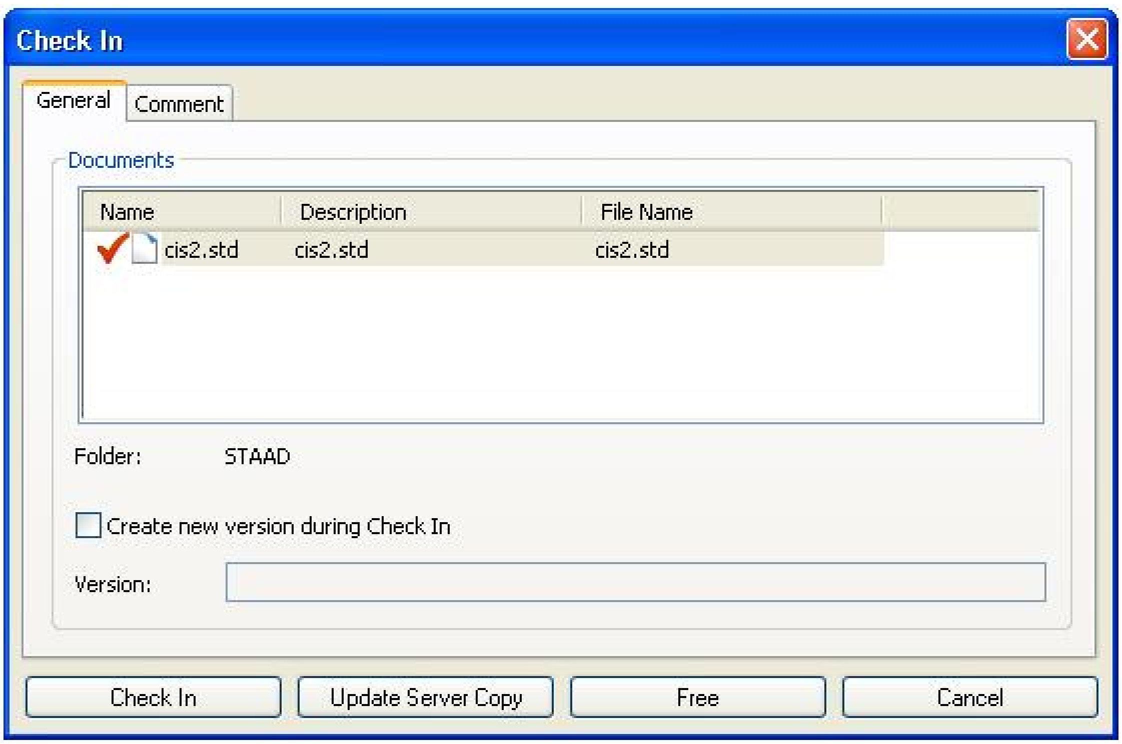
When a model is checked out from a ProjectWise server, selecting Save or Save As, only maintains a local copy of the model. There are two methods available to update a checked out model. Firstly, during a STAAD.Pro session, it is possible at any time to save any changes back on the server by selecting the Update Server Copy icon from the ProjectWise toolbar or from the **File** > **ProjectWise** menu.

First save updates to the file locally. If not, then this will be prompted. Then the following dialog is displayed which allows a comment to be added to this model, thus:



The file remains checked out and can be continued to be worked on.

The second method is automatically generated when a checked out model is closed. This launches the Check In dialog that displays the file(s) that are to be checked in and provides four actions:



Check In This copies the local version of the STAAD model back to the server and releases it so that it is available for others to modify. If the option Create New Version is selected, then a new copy of the file is created on the server which becomes the current version of the model. The status of the checked out model is changed to read only on the server and can be used as a reference to a stage in the development of the model.

Update Server Copy This updates the model on the server with the current local model, but does not change its status which remains as checked out.

Free	This changes the status of the model as checked in which allows other users to take control and check out the model, but does not update the server model. Thus local changes will be discarded
Cancel	This cancels any change to the status of the model or the model itself on the server.

Menu

The commands that drive the ProjectWise integration are defined in the main File menu thus:

File > **ProjectWise**

The Add... command is available when using a locally opened (not checked out from ProjectWise), which allows the current model to be saved into a ProjectWise repository.
The Open… command is available when a suitable ProjectWise client has been installed to allow access to a repository from which to check out a STD file.
The Update Server Copy command is available when working on a checked out file and the changes made on the model can be

Toolbar

A new toolbar named ProjectWise has been added that duplicates the commands from the **File** > **ProjectWise** menu, thus:

Notes

1. For more details on ProjectWise refer to the ProjectWise client installation documentation.
2. This functionality requires access to a version V8i or greater of ProjectWise.

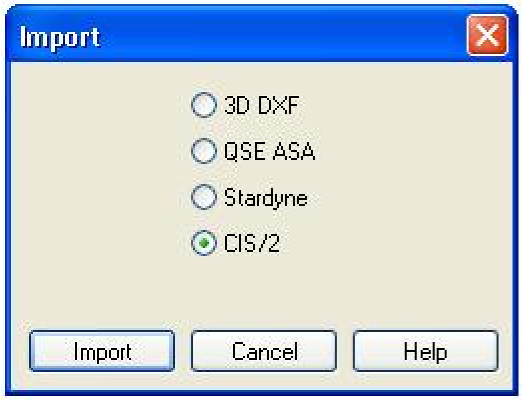
AD.2007-04.0.2 CIS/2 Update

The STAAD.Pro tool to import and export models with the CIS/2 translator has been enhanced to work with international models and transferring models into 3D modeling such as SmartPlant 3D.

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of steel models using a prescribed data standard in the STEP (Part 21) format. These files can contain different models including analysis models. The STAAD.Pro CIS/2 translator only operates on the analysis model within the file. While STAAD.Pro has supported a wide range of the steel sections that this standard can support, this enhancement allows a far greater range of sections to be imported/exported with this tool.

Import

The CIS/2 import can be initiated after starting a new model and before creating any model data, selecting the menu option, **File** > **Import...**

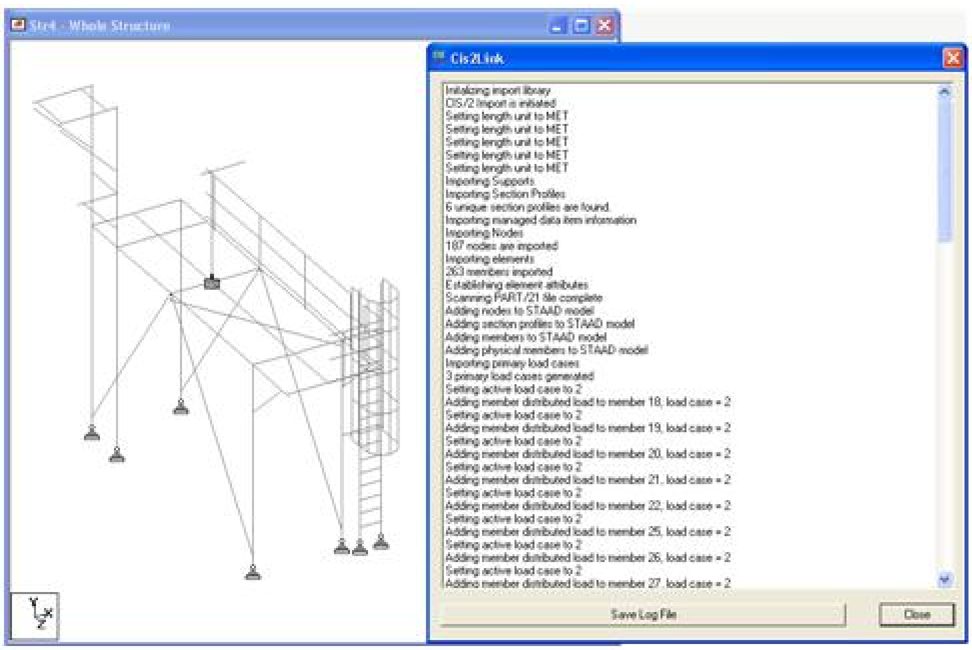


Selecting the CIS/2 option and clicking on the “Import” button allows selection of a suitable STEP file. This file is a text file and will be similar to the following extract:

```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer
* from STEP Tools, Inc. (www.steptools.com)
*/
FILE_DESCRIPTION(
/* description */ ('CIS2 Export File'),
/* implementation_level */ '2;1');
FILE_NAME(
/* name */ 'model',
/* time_stamp */ '2007-01-08T17:00:33+00:00',
/* author */ (''),
/* organization */ (''),
/* preprocessor_version */ 'ST-DEVELOPER v9',
/* originating_system */ 'STAAD.Pro 2007',
/* authorisation */ '');
FILE_SCHEMA (('STRUCTURAL_FRAME_SCHEMA'));
ENDSEC;
DATA;
#10=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#264);
#11=LOAD_COMBINATION_OCCURRENCE(1.39999997615814,#13,#265);
#12=LOAD_COMBINATION_OCCURRENCE(1.60000002384186,#13,#266);
#13=LOADING_COMBINATION('ULS',$,#2511);
#14=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member19UDLLoad1',$,#1089,
$,$,.F.,.GLOBAL_LOAD.,.TRUE_LENGTH.,#214,#214,#64);
#15=LOAD_ELEMENT_DISTRIBUTED_CURVE_LINE(#265,'Member20UDLLoad1',$,#1090,
```

Once a file has been selected, the data is ready to be imported with the CIS/2 import tool thus:

Clicking the Import button then runs the model import where the CIS/2 file is processed and the analysis model data is extracted to form the STAAD.Pro model.



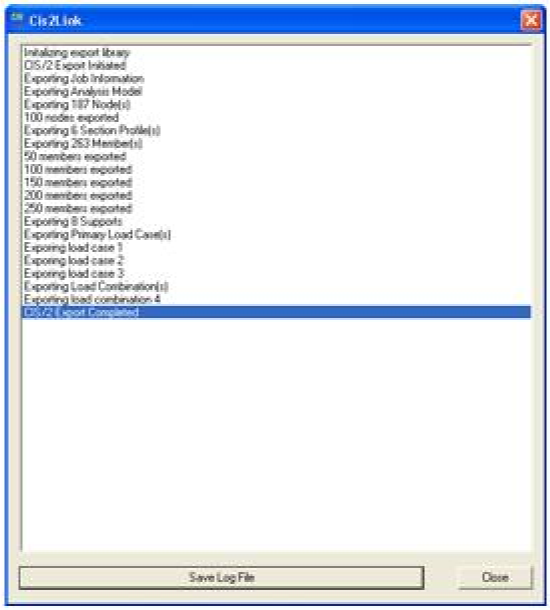
Note that as the CIS/2 file is being processed, a log file which identifies the data that has been utilized in the STAAD model is produced and displayed which can be saved as a text file for future reference.

Export

The CIS/2 Export is available for any model that has been created as an option in the menu item File>Export…

Selecting the CIS/2 option and clicking on the “Export” button allows the definition of a suitable STEP filename and folder to locate it. Once again the CIS/2 tool is presented, this time with an Export Model button at the bottom which when clicked creates the STP file.

Once again, when the export has completed, the user can save the log file which is produced as the model is converted into the STEP format.



Enhancements

The STAAD.Pro CIS/2 import now recognizes sections defined from standard databases such as those defined in Japanese, British, Indian, Australian and European tables.

The import/export has been enhanced to support the ability of STAAD.Pro to create double sections, such as back to back angles and channels or double wide flange sections. Additionally, it now supports the creation of T sections which are defined in STAAD.Pro as a wide flange section that is split at mid height.

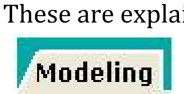
Although CIS/2 has been developed for the processing of steel models, the STAAD.Pro translator will now support the transfer of prismatic properties normally associated with concrete sections. This means that sections defined as PRISMATIC in a STAAD.Pro model will be included in exported STP file and can be imported if they exist.

Related Links

- *I. To import a CIS/2 file* (on page 2078)
- *I. To export to a CIS/2 file* (on page 2080)

AD.2007-04.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.



AD.2007-04.1.1 ASME NF Steel Design Codes

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:

- 1974
- 1977
- 1989
- 1998

Post analysis steel design code checking requirements for the required ASME NF code can be selected by entering the **Design | Steel** page and setting the ASME code / year in the **Current Code** option in the **Steel Design - Whole Structure** dialog:

The method for selecting design parameters and assigning them to the members to change them from the default values is exactly the same as for all other steel design codes. Additionally, choosing members that are to be checked or selected for maximum utilization follows exactly the same method as for all other steel design codes.

For more information on the technical requirements of this design code, including the full set of parameters and default values, see the new section in the D1.L. American Codes - Steel Design per ASME NF Codes (on page 1310).

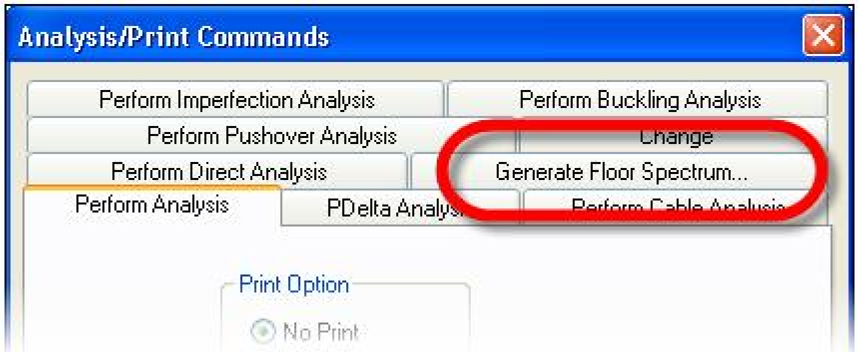
Note: In order to run a design check to any of the ASME NF design codes, then access to a STAAD Nuclear Code pack will be required.

AD.2007-04.1.2 Floor Response Spectrum

A new dynamic feature has been added that allows the extraction of a response spectrum from a collection of nodes that constitute a floor when subjected to a time history loading. This information can then be used in conjunction with equipment that will be supported by these floors and is often required by the equipment manufacturers.

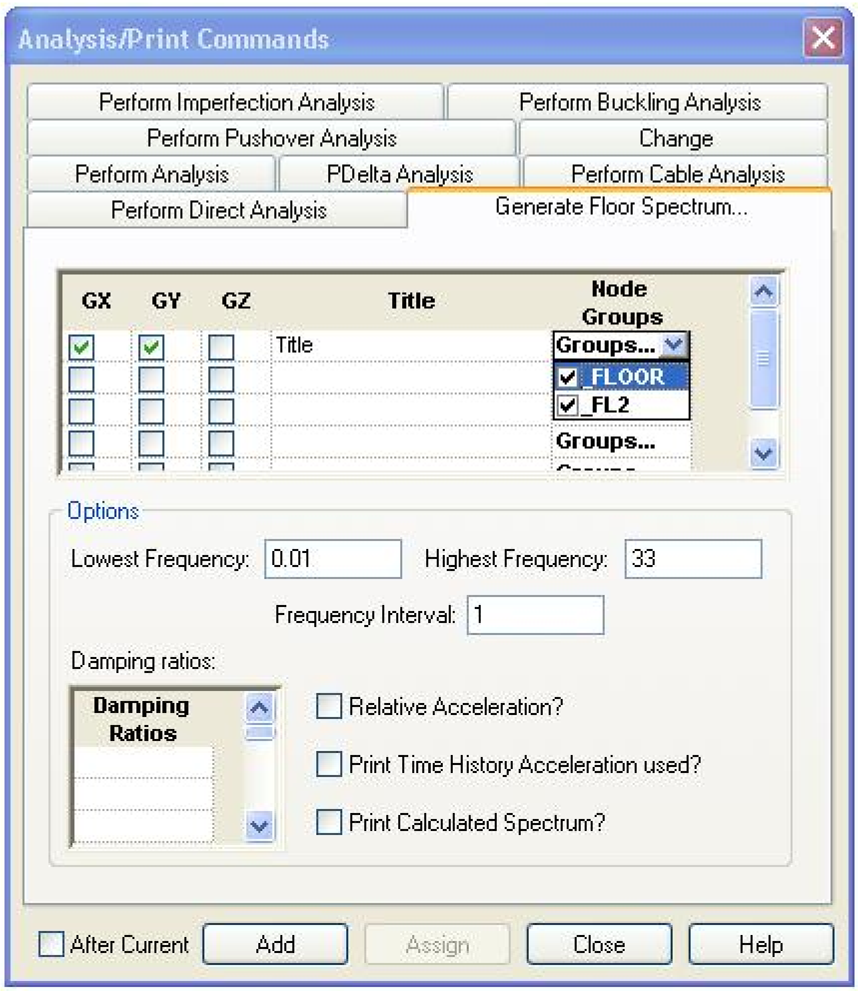
Users will require a license for the advanced analysis module to access this feature.

The required commands (see AD.2007-04.2.3 Floor Response Spectrum (on page 245) for more information) can be entered graphically after adding the analysis command by selecting the new analysis sheet **Generate Floor Spectrum** thus:



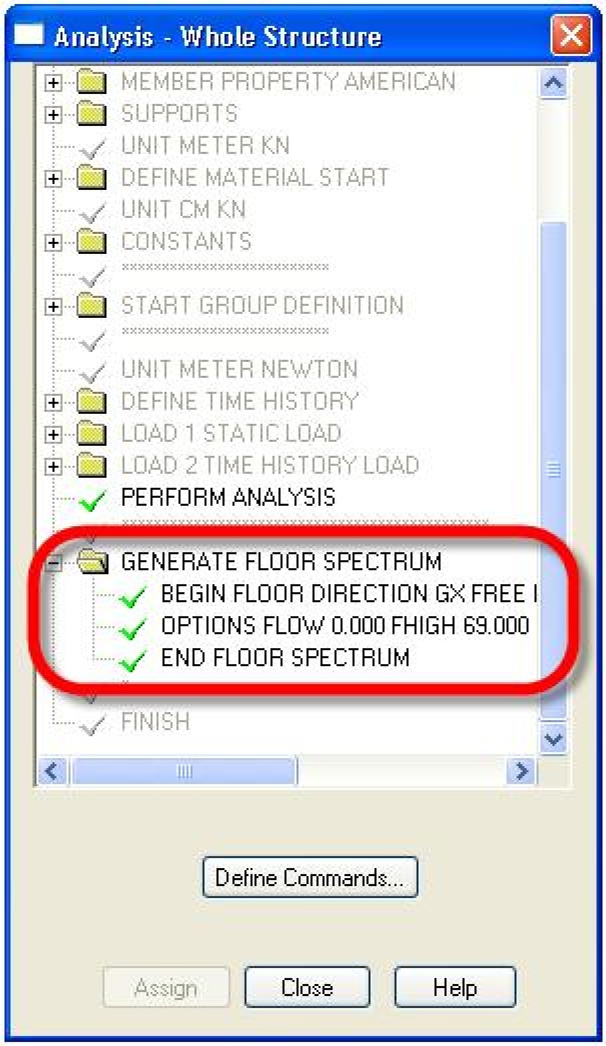
This command should follow immediately the definition of the analysis and will require defined groups of nodes which need to be defined first.

The following displays the layout of this sheet:

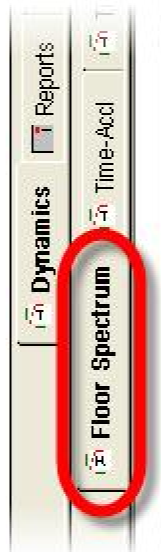


Note: Each line of settings constitutes one floor which can have one or more floor groups assigned. The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.

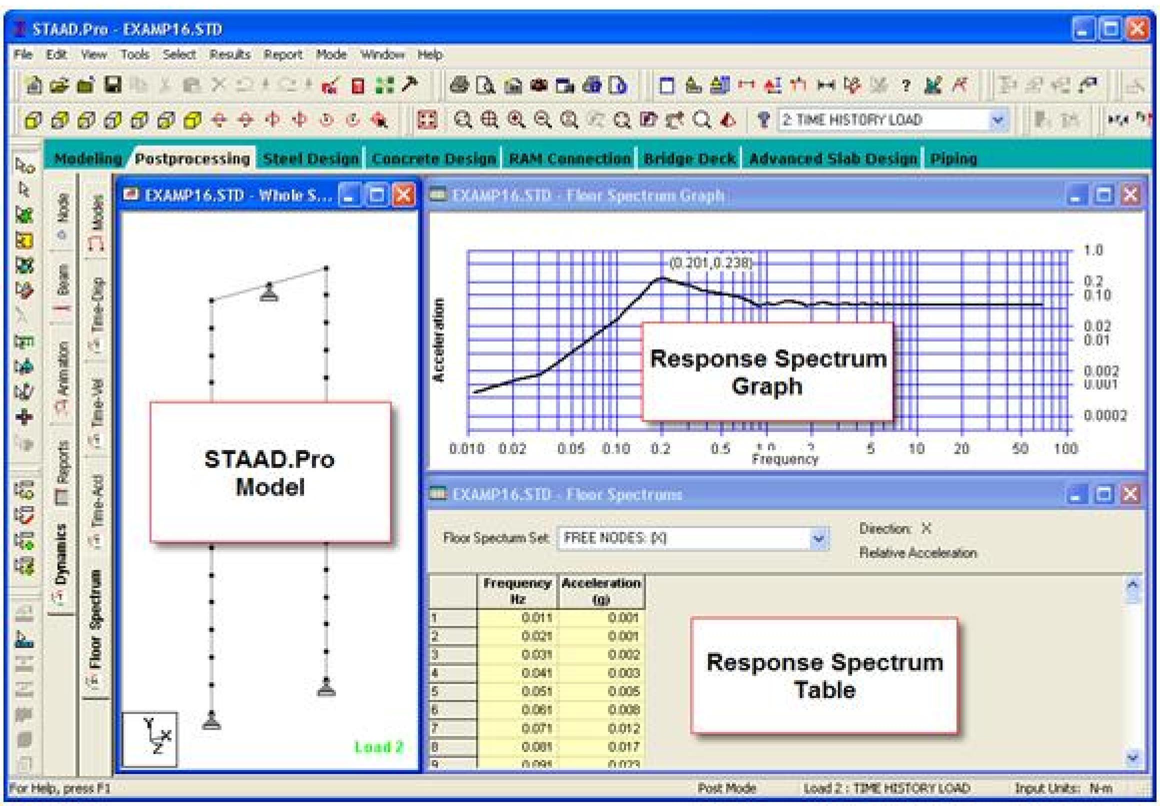
Once the required options have been set, click on the Add button to add the command set to the model which should appear in the Analysis Window thus:



Once the command has been added and the file saved, the analysis can be run which will generate a new sub-page in the Post-Processing Mode in the Dynamics Page called Floor Spectrum:



Entering this page, STAAD.Pro will display the floor spectrum thus:



To change graphs to that of another selection of node groups or damping ratio, then

select the required set from the drop list in the Response Spectrum Table.

The graph is initially set to display the results on a log/log graph. This can be changed to a linear graph by right clicking on the graph and selecting the “Linear Graph” option. Additionally, the calculated points on the graph can be added by again right clicking on the graph and selecting the option ‘Show Points’.

The data that has defined the graph can also be exported to a text file and used in a third party application by right clicking on the graph and selecting the option "Save Data in Text File…"

Related Links

- *TR.37.10 Floor Spectrum Command* (on page 2657)
- *A. To generate a floor spectrum* (on page 959)
- *P. To display floor spectrum results* (on page 1994)

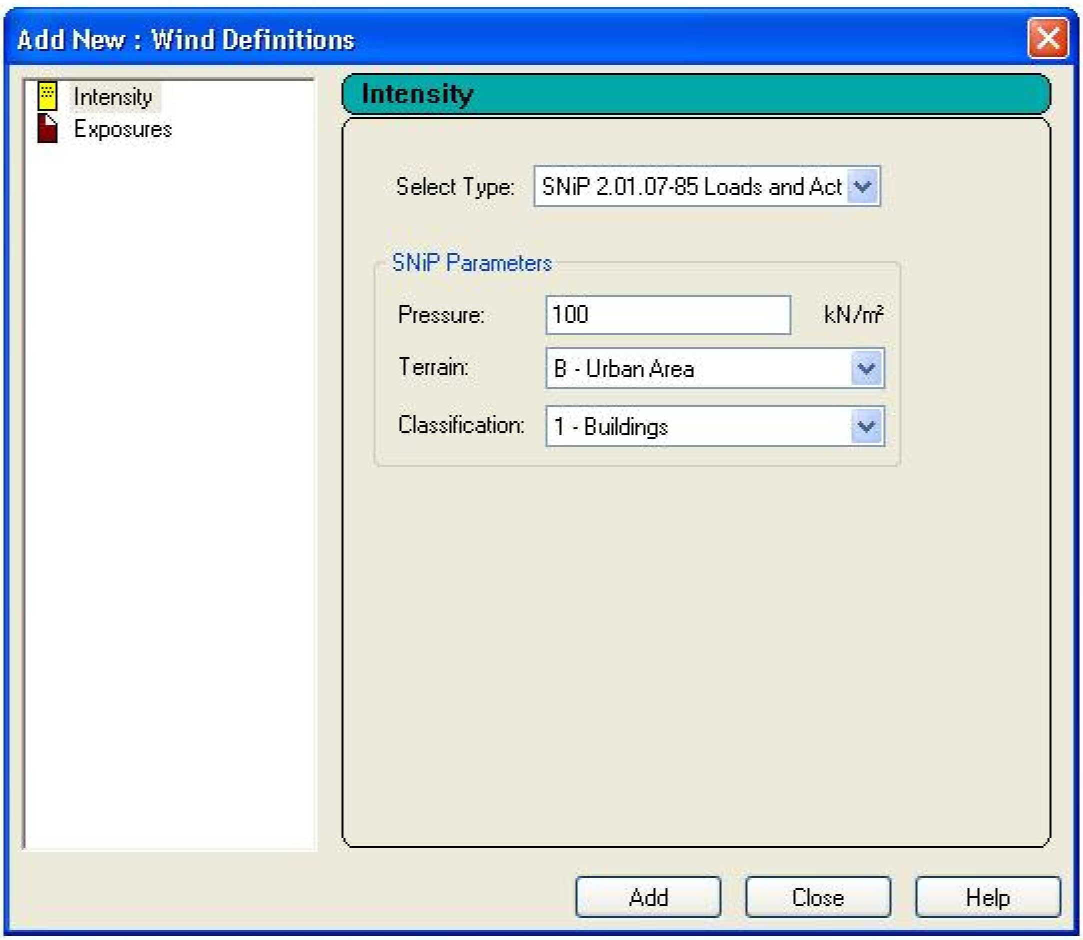
AD.2007-04.1.3 Russian Wind Loading

The wind loading as defined in the design code SNiP 2.01.07–85 "*Loads and Actions*" added in this version of STAAD.Pro can be added graphically to the model by modifications in the Loads Page.

There are two areas where the graphical user interface has been updated:

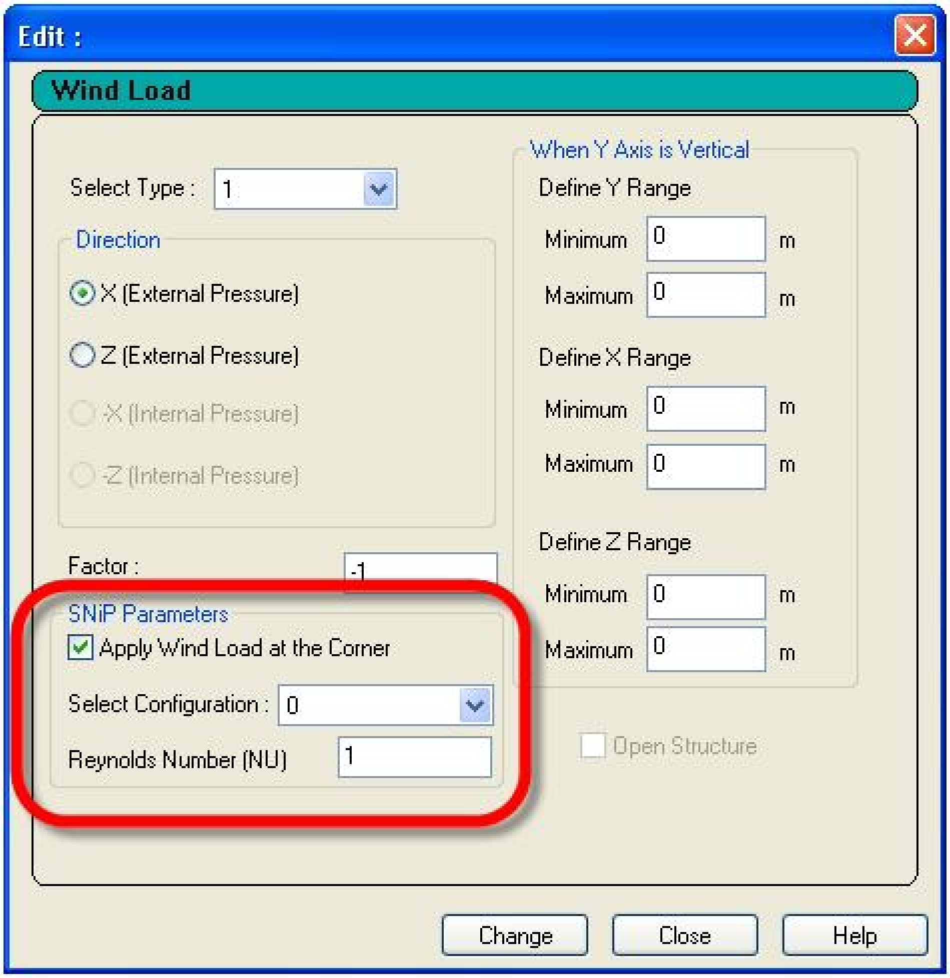
Wind Load Definition dialog

The dialog has been updated to allow the entry of the required parameters for a Russian Wind Load definition thus:-



This will create a definition which can be added to a wind load case.

Wind Load



Note: The first load case which has a Russian Wind Load command added to it will consider all other loads defined in it as the masses to be considered for calculating the dynamic effect which is required by this command.

For technical details of this wind loading see section AD.2007-04.2.2 Russian Wind Loading (on page 244) .

AD.2007-04.1.4 Additional Standard Profile Databases

An additional Australian cold formed database has been provided to complement the cold formed sections databases currently provided.

The following database and tables have been added from OneSteel. Duragal®, Galtube® and Tubeline®

Australian Cold Formed Steel Hollow Sections

Circular Hollow Sections:

- Galtube Plus®, 26.9mm to 76.1mm diameter
- Tubeline, grades C250L0 (AS1163) and 350L0 (AS1163), 26.9mm to 457.0mm

Rectangular Hollow Sections

- Duragal®, grade C450L0 (AS1163), 50x20mm to 150x50mm
- Galtube Plus®, grade C350L0, 50x20mm to 75x25mm
- Tubeline®, grade C350L0 (AS1163), 50x20mm to 250x150mm

Square Hollow Sections

- Duragal®, grade C450L0 (AS1163), 20x20mm to 100x100mm
- Galtube Plus®, grade C350L0, 20x20mm to 65x65mm
- Tubeline®, grade C350L0 (AS1163), 13x13mm to 250x250mm

AD.2007-04.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-04.2.1 ASME NF

The design of steel sections according to the requirements in the American Society of Mechanical Engineers (ASME) specifications, Rules for the Construction of Nuclear Power Plant Components, Section III – Subsection NF has been implemented and the steel design page has been updated to allow the design parameters to be defined and assigned.

The design requirements for the following years have been added:

- 1974
- 1977
- 1989
- 1998

For the list of parameters and commands including the default values, please refer to the D1.L. American Codes - Steel Design per ASME NF Codes (on page 1310).

For each steel member that is checked, the following checks are performed according to the clauses for that year:

1. Slenderness check
2. Tension
3. Compression
4. Bending (a) About major axis, (b) About major axis
5. Shear
6. Combined Stresses

AD.2007-04.2.2 Russian Wind Loading

The wind loading commands have been enhanced to allow the creation of wind loading as defined in the in Russia by the design code SNiP 2.01.07–85 *Loads and Actions*.

The basic quantity in the wind loading is the characteristic (normative in Russian terminology) wind pressure. The reference wind velocity pressure corresponds to a 10-minute time-averaged velocity pressure at 10-metres height in a flat terrain, based on a 5-year return period. This wind pressure is the static component of the wind load. The total wind pressure consists of static and fluctuating components. If the structure is sufficiently flexible, according to the code provisions, the dynamic structural response to the fluctuating wind component must be taken into account.

The updated wind loading commands automatically perform both the aerodynamic and structural load analysis of vibration-susceptible buildings and structures.

The wind loading commands have been updated to support the wind loading as defined in the Russian design code. This requires the creation of the following commands.

There are three parts to creating a Russian wind load on a STAAD Model.

1. Definition of the wind load requirements
2. Application of the wind load definition within a load case
3. Cut-off frequency or mode shape

Related Links

- *TR.30.1 Cut-Off Frequency, Mode Shapes, or Time* (on page 2344)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *M. To add a SNiP wind load definition* (on page 847)

AD.2007-04.2.3 Floor Response Spectrum

The following commands have been added in order to allow the response spectrum of floors to be extracted from a time history analysis.

This command is used to specify the calculation of floor and/or joint spectra from time history results.   The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case.

Related Links

- *TR.37.10 Floor Spectrum Command* (on page 2657)
- *A. To generate a floor spectrum* (on page 959)

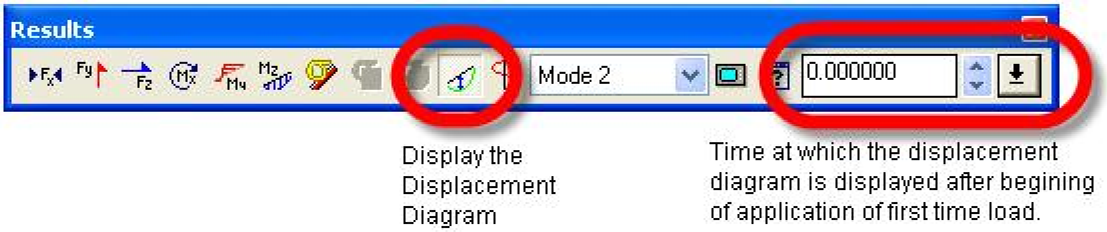
AD.2007-04.3 Features Affecting the Post-Processing (Results Mode)

Several new features have been added and existing features have been modified in the post-processing section of the program, also known as the Results Mode. These are explained in the following pages.

AD.2007-04.3.1 Time History Animation

In order to visualize the displacement that occurs on a model during the application of a time history load, a new toolbar icon has been added which allows the displacement at a specific time instance to be displayed.

The results toolbar has been updated to include a new option which is activated when the displacement diagram icon has been clicked and the current load case contains time history loading:



For a time history load case the displacement that is displayed will be defined by the time instance entered in the above edit box. Alternatively the time can be set from a sliding scale by clicking on the button with the down arrow icon which displays the following option setting:

The slider scale is based on the overall time for which the time history analysis has been performed.

The displacement is produced at the time at which the slider arrow is dragged to and the mouse button released.

If the **Apply Immediately** option is selected, then the application will attempt to render the displacement diagram dynamically as the slider is dragged up and down the scale of time. For large models this may prove to be too demanding on the graphics system and left un-checked which means that the displacement diagram will be produced at the time step at which the slider arrow is released.

Note that the displacement diagram will be set to the scale as defined in the Scales sheet in the **Structure Diagrams** dialog box.

Additionally note that it is possible to view the time v displacement of individual nodes by clicking on the Dynamics>Time-Disp in the post processing mode which is only available for models that include time history.

AD.2007-04.3.2 Enhanced Plate Stress Results

In order to provide additional understanding of stress distribution in finite element models, STAAD.Pro has expanded the sets of results that can be reported for each element both in the Plate Centre Stress Table and graphically using the Plate Stress Contour.

To view the results data of plate elements, enter the Post Processing (Results) Mode, and click on the Plate Contour Page on the left menu.

Plate Centre Stress Table

The new Combined Stresses sheet in the Plate Centre Stress table provides resolved stresses for the top (positive local Z) and bottom (negative local Z axis) surface for each plate element, referred to as the Top Combined Stresses and Bottom Combined Stresses respectively.

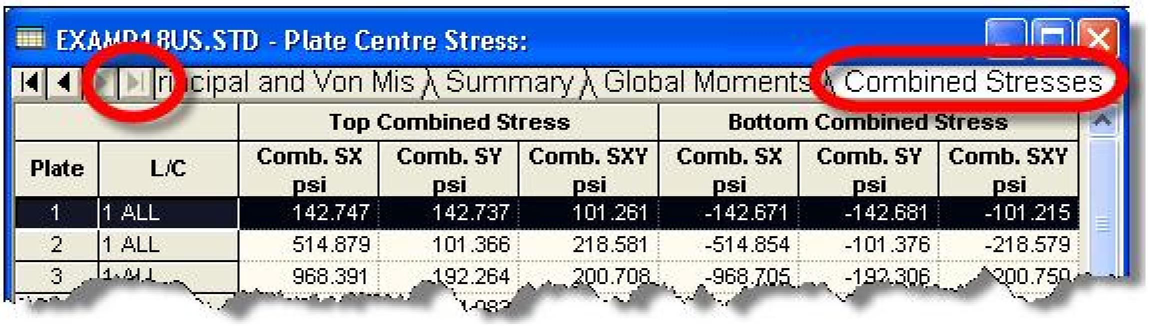


Plate	L/C	Top Combined Stress Comb. SX psi	Comb. SY psi	Comb. SXY psi	Bottom Combined Stress Comb. SX psi	Comb. SY psi	Comb. SXY psi
1	1 ALL	142.747	142.737	101.261	-142.671	-142.681	-101.215
2	1 ALL	514.879	101.366	218.581	-514.854	-101.376	-218.579
3	1 ALL	968.391	192.264	200.708	-968.705	-192.306	200.750

Note that if the sheet title Combined Stresses is not visible on the top of the table, then click on the right arrows displayed on the left of the table header to scroll the table sheets which will display the title.

The combined stresses are calculated thus:

Top:

SX_{top} = SX + MX/S

SY_{top} = SY + MY/S

SXY_{top} = SXY + MXY/S

Bottom:

SX_{bottom} = SX + MX/S

SY_{bottom} = SY + MY/S

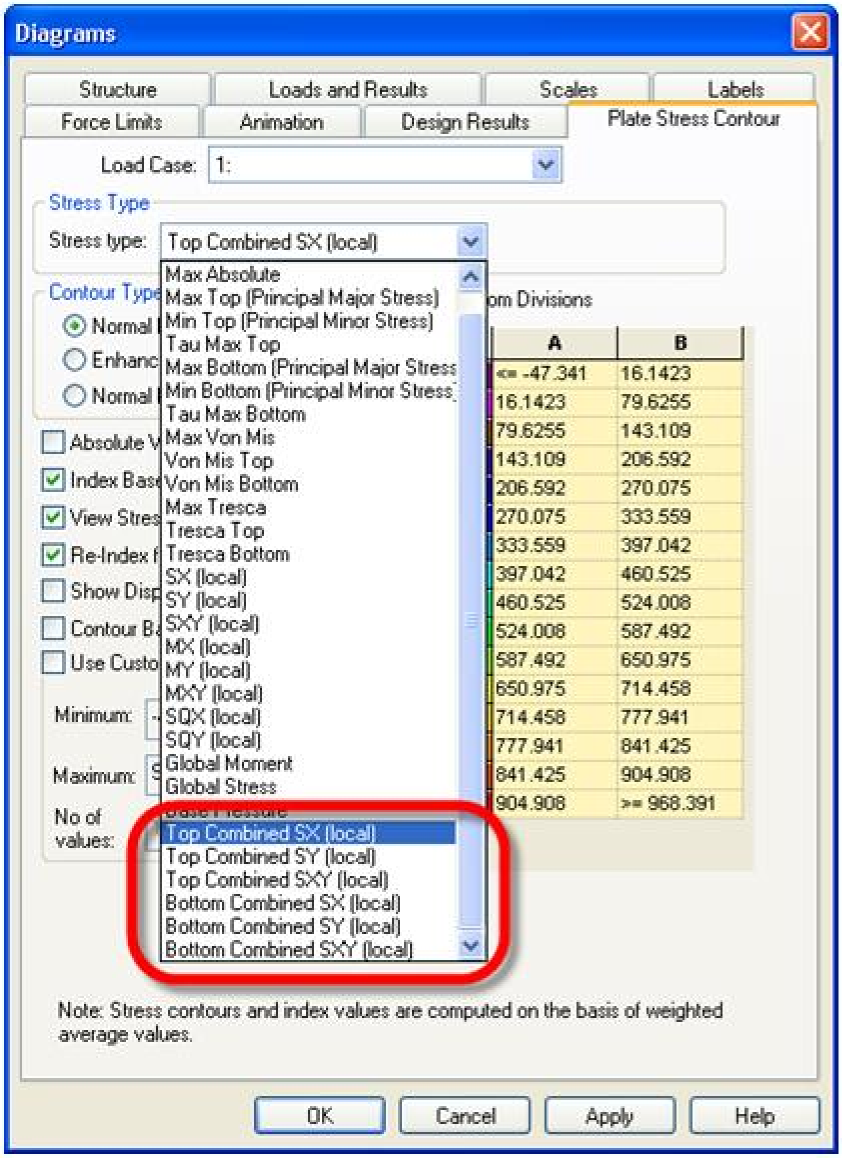
SXY_{bottom} = SXY - MXY/S

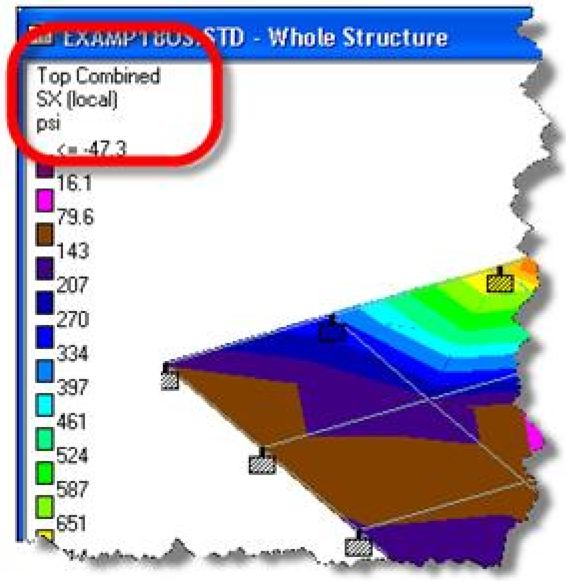
where

S	=	$t^2/6$ t
t*S*	=	average plate thickness

Plate Stress Contour

The Plate Stress Contour sheet of the Diagrams dialog has been enhanced to allow visualization of these stresses. The six new stress results are available in the Stress Type pull down menu thus:


Diagrams
Structure
Loads and Results
Scales
Labels
Force Limits
Animation
Design Results
Plate Stress Contour
Load Case:
1:
Stress Type
Stress type:
Top Combined SX (local)
Max Absolute
Max Top (Principal Major Stress)
Min Top (Principal Minor Stress)
Tau Max Top
Max Bottom (Principal Major Stress
Min Bottom (Principal Minor Stress
Tau Max Bottom
Max Von Mis
Von Mis Top
Von Mis Bottom
Max Tresca
Tresca Top
Tresca Bottom
SX (local)
SY (local)
SXY (local)
MX (local)
MY (local)
MXY (local)
SQX (local)
SQY (local)
Global Moment
Global Stress
Top Combined SX (local)
Top Combined SY (local)
Top Combined SXY (local)
Bottom Combined SX (local)
Bottom Combined SY (local)
Bottom Combined SXY (local)
Contour Type
A
B
Minimum:
Maximum:
No of values:
Note: Stress contours and index values are computed on the basis of weighted average values.
OK
Cancel
Apply
Help



STAAD.Pro 2007 Release Reports

This section of the Software Release Report contains detailed information on additions and changes that have been implemented in builds of STAAD.Pro 2007 since the release of STAAD.Pro 2006 build 1005. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro 2007 Build 03 Release Report

The Software Release Report for STAAD.Pro 2007 Build 03 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 Build 05. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

STAAD.Pro 2007 Build 02 was a maintenance update and only contained a number of minor updates which are listed in the Revision History text file.

AD.2007-03.0 Features Affecting the General Program

This section describes features that have been added that affect the general behavior of the STAAD.Pro application.

AD.2007-03.0.1 RSS Feed added to the Start Page

The STAAD.Pro Start Page has been updated to include a new section which has been configured to display the most current information about STAAD and Bentley that will be of use to you, such as program updates, seminars, and training courses. This has been done by adding an RSS (Really Simple Syndication) reader on the application front screen.

Each news item is identified with a title which is a link to a website which can be clicked on and will launch your web browser and load that website and a brief summary of the item.

The item is categorized with one of the following five categories:

 Important

 Bentley General

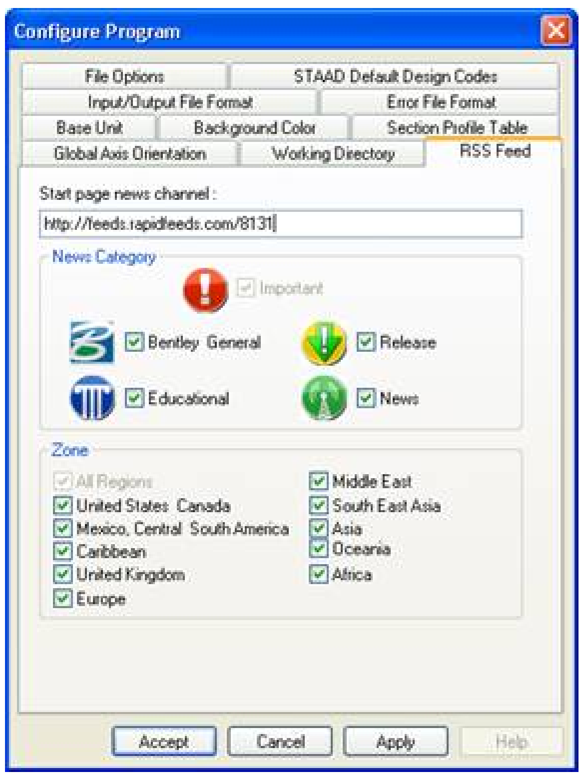
 Release

 Educational

 News

Additionally, each item is marked for specific regions thus:

To configure which messages are displayed and for which region, click on the **Configuration...** link on the Start Page. Select the RSS Feed and click in the categories and regions to remove those that are not required.



Note: Items that have been classed as Important and for All Regions cannot be disabled, but other issues can be by un-checking the boxes next to them

AD.2007-03.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

AD.2007-03.1.1 New Meshing Options

The Parametric Models section STAAD.Pro has been enhanced with additional meshing options to provide three methods for creating a finite element mesh. These methods are classified as **Basic**, **Standard**, and **Advanced**.

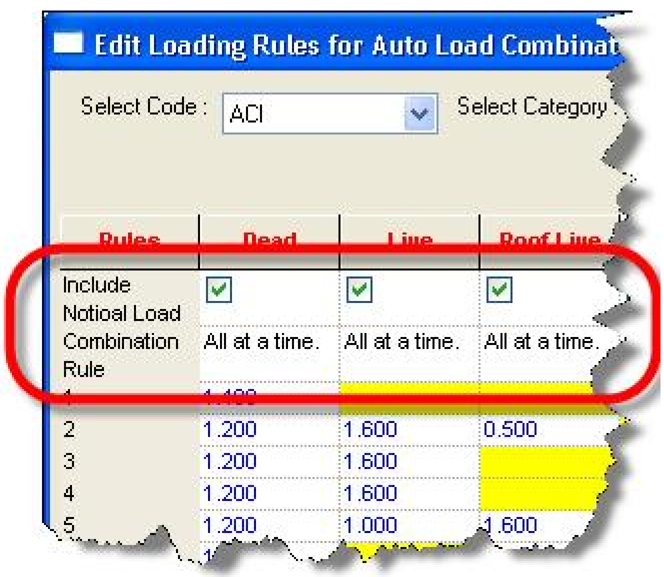
The original content of this topic has been superceeded by content in the main help.

Please refer to the Add Parametric Model dialog (on page 2737) in the User Interface help for current details.

AD.2007-03.1.2 Enhanced Automatic Load Combination Generation

The Automatic load combination generator has been enhanced with two options that define whether notional loads should be included in the combinations and how combinations are created to allow a far greater range of combination conditions to be considered.

The two new options are displayed on the top rows of the dialog box that is opened from the Pre-Processor menu item: **Commands** > **Loading** > **Edit Auto Rules**



Related Links

- *Edit Load Rules for Auto Load Combination Generator dialog* (on page 2875)
- *M. To define automatic load combination rules* (on page 887)

AD.2007-03.1.3 Generation of Primary Load Cases Using Repeat Load Commands

With the large number of combinations that are required by design codes, STAAD.Pro introduced a tool to allow the combinations to be created automatically based on a rule set. This functionality has been enhanced to allow the automatic creation of equivalent combinations, but using primary load cases and the `REPEAT` command.

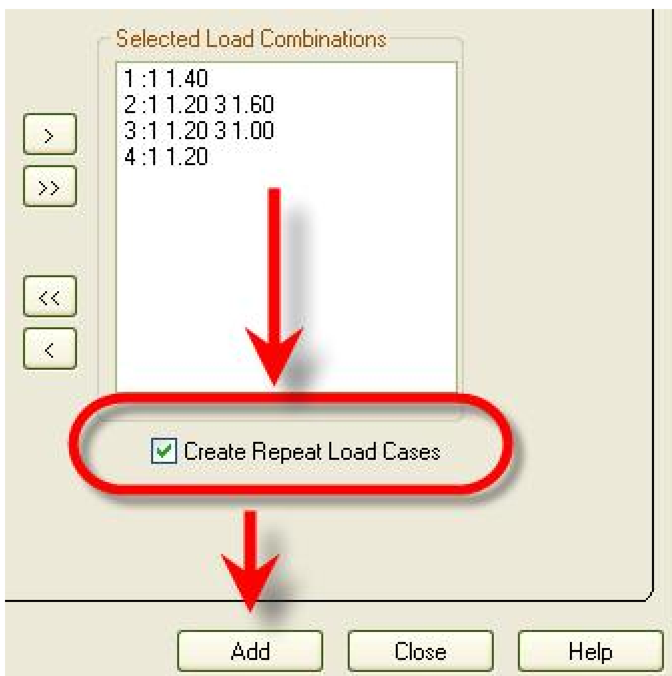
The use of the `COMBINATION` command to obtain results for a collection of load cases that occur together is entirely valid when using first order analysis and models that are not dependent upon the forces in them (such as tension only members or one-way supports). In these cases, the law of superposition allows the results of each load case to be multiplied by a scaling factor and added together. However, for models where this does not apply, such as using a P-DELTA analysis, TENSION ONLY members or ONE-WAY supports, then the law of superposition does not necessarily apply. For this reason, STAAD.Pro has the ability to create primary load cases which replicate the effect of a combination, but in a primary load case. This means that the analysis will solve the load case by using the applied loading rather than simply adding the results. This is achieved in a primary load case by using the `REPEAT` command.

For full details of the `REPEAT` command see TR.32.11 Repeat Load Specification (on page 2595).

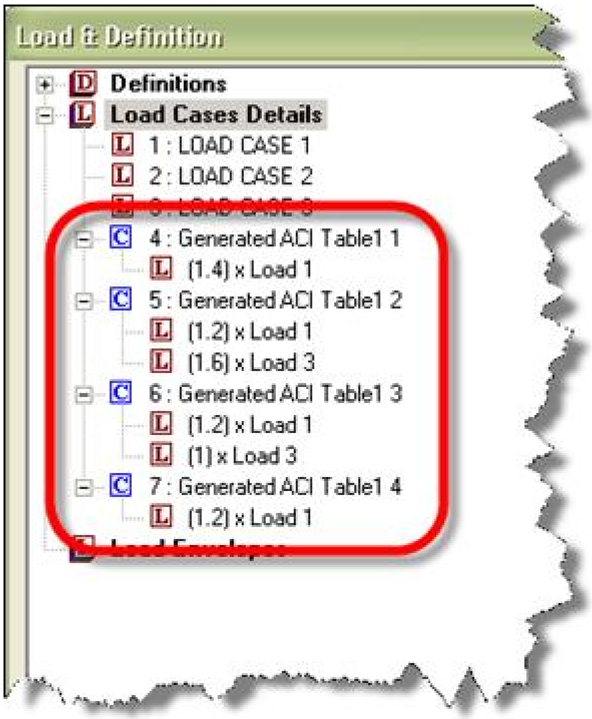
There are three parts to automatically creating primary load cases equivalent to combinations.

1. Define the rules that determine how load case types are to combined
2. Define the individual primary load cases classified with their type
3. Create the combinations.

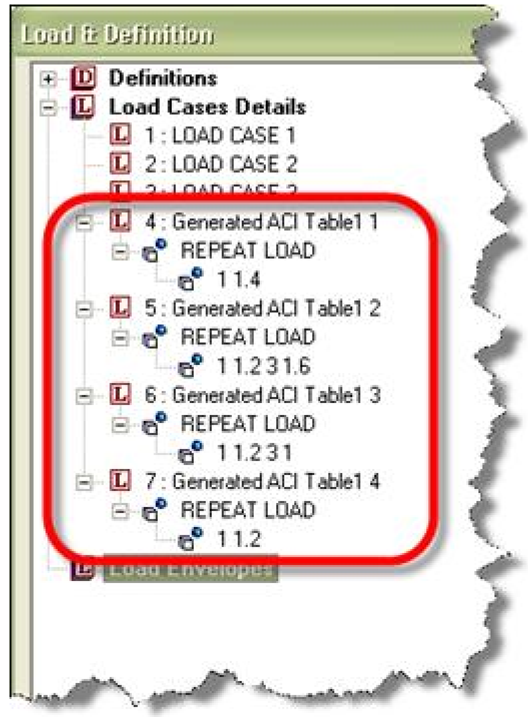
By selecting the option **Create Repeat Load Cases** all the load case combinations in the Selected Load Combinations list will be created into primary load cases:



Note the difference in the **Load & Definition** dialog when comparing a generation of Combination load cases rather than Primary load cases with `REPEAT` commands:



Combination Load Cases



Primary Load Cases with Repeat

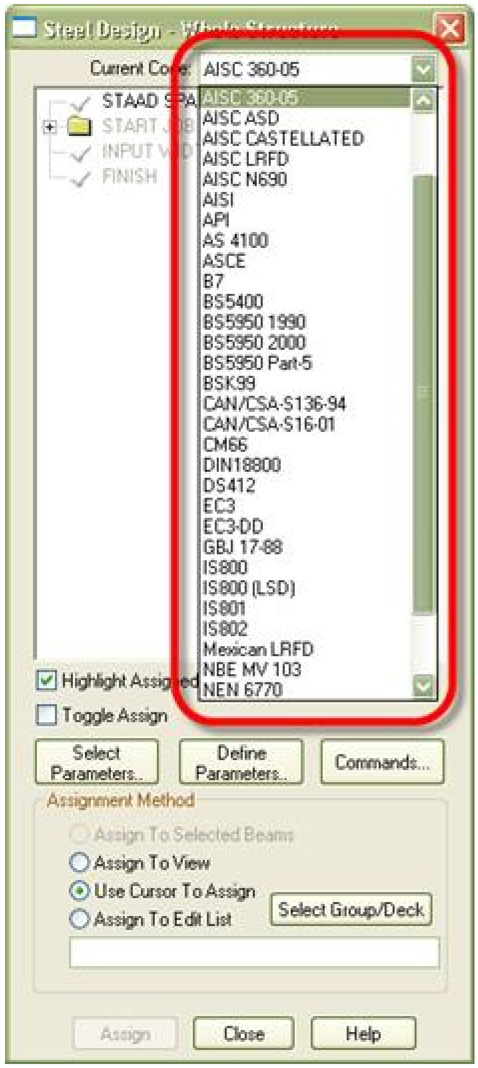
Related Links

- *Edit Load Rules for Auto Load Combination Generator dialog* (on page 2875)
- *M. To define automatic load combination rules* (on page 887)

AD.2007-03.1.4 Design Code List

The list of supported design codes in the design dialog boxes has been updated to represent the actual document rather than the country where this is possible.

The list of codes displayed in the Steel, Concrete, Timber, and Shearwall Design dialog boxes has been updated such as here for the Steel **Design** dialog box:



Note that there is no change to the input or output generated from the design unless noted in the section, AD. 2007-03.2 Features Affecting the Analysis and Design Engine (on page 262).

AD.2007-03.1.5 Additional Standard Profile Databases

An additional Japanese cold formed database has been provided to complement the cold formed sections databases currently provided.

The following database and tables are available.

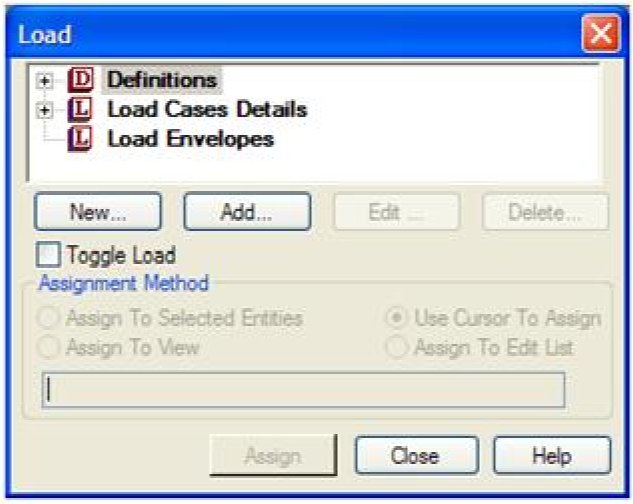
Japanese Cold-Formed Steel

Box Column Press
Box Column Press (Tough)
Box Column Roll

AD.2007-03.1.6 IBC 2006

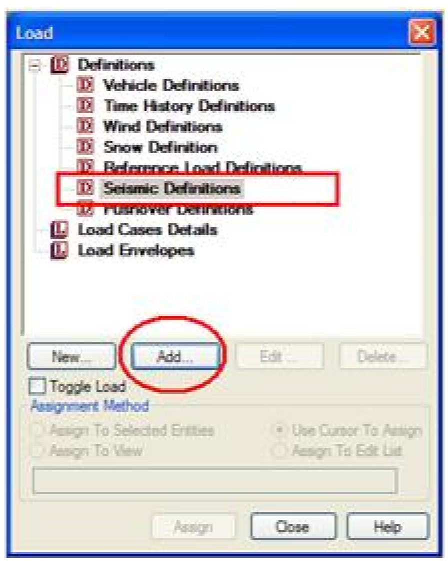
The static equivalent method for performing dynamic analysis as per the IBC 2006 code has been implemented. This follows the principals and methods as in the IBC 2000 and IBC 2003 codes previously implemented.

When the **General | Load** Page is selected, the right hand side of the screen will display the following if no load cases exist in the model.



Definitions contains the options through which one creates the **Define** block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases.

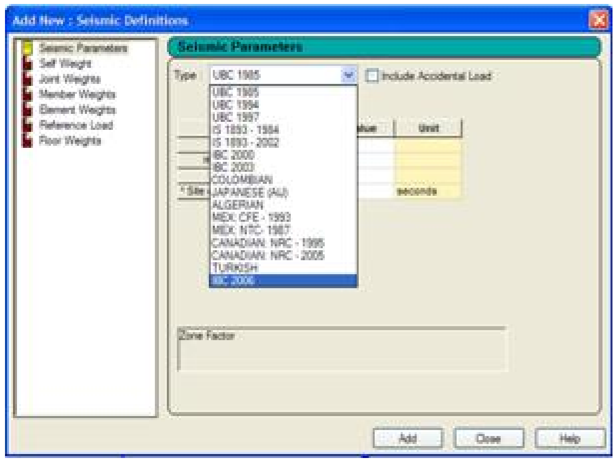
When the tree view is expanded, it will display thus:



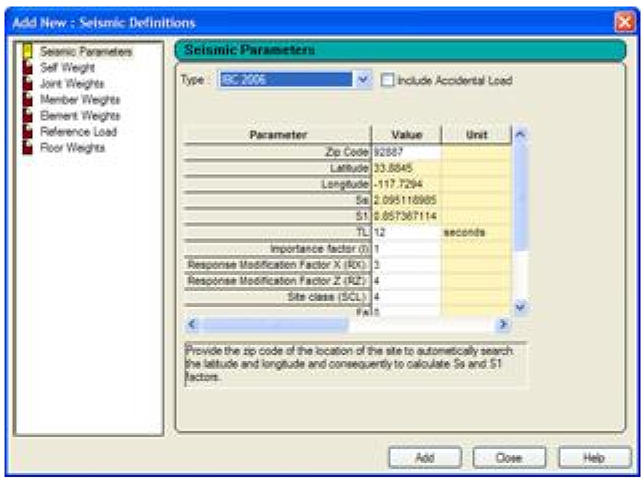
To define the definition of the IBC 2006 select the Seismic Definitions and click on Add. As with other seismic definitions, there are three parts to the exercise,

1. Defining the code and parameters
2. Defining the weights to be considered
3. Include the definition in a primary load case

In the dialog box that is displayed, select **IBC 2006** from the drop-down list.



In this dialog box, the required data is entered into the parameters as described below.

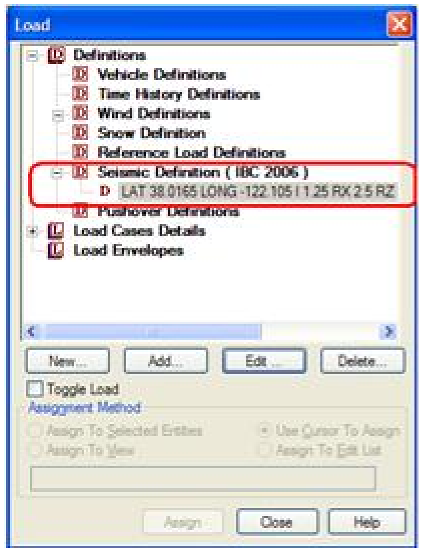


There is an option to **Include Accidental Load**, if this is selected the analysis will include an additional accidental torsion component as described in section 12.8.4.2 of ASCE 7-05.

Refer to the Add New Seismic Definitions dialog (on page 2895) and TR.31.2.14 IBC 2006/2009 Seismic Load Definition (on page 2405) for details.

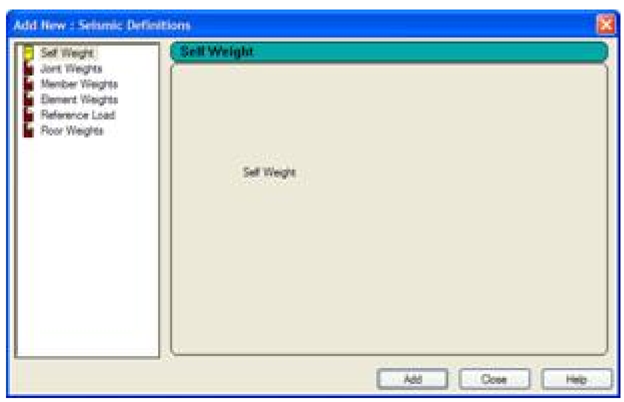
After specifying the values for the parameters, click on the **Add** button.

Note that the **Load** dialog box has been updated with the new command.



As described earlier, the second part of the process is to identify the structural weights that should be considered. These can be added once the defined parameters have been created by clicking on the **Add** button. The dialog box is updated to display the various weights that can be added thus:

Selfweight



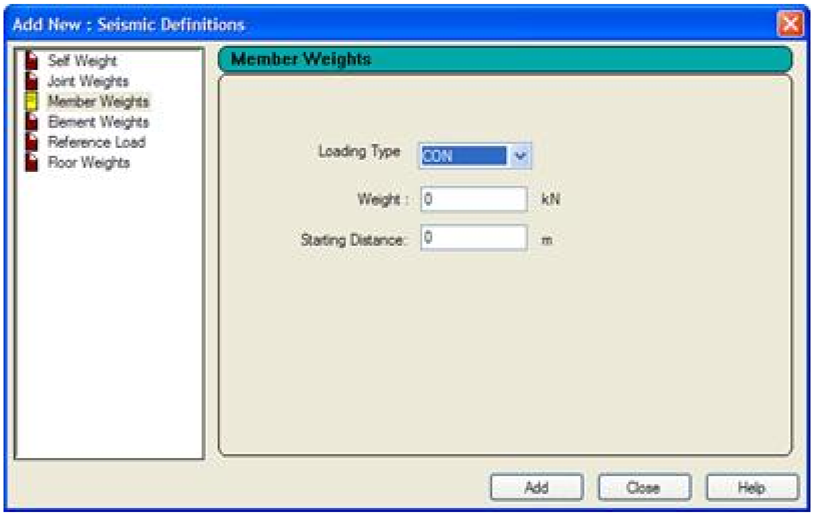
This is the selfweight of the structure.

Joint Weights

These are the concentrated weights acting at one or more joints.

Member Weights

Distributed and concentrated weights acting on member spans are specified through this option. After clicking the Member weight button, the Member weight dialog box appears, as shown below.



Select the **Concentrated** or **Uniform** load type from the **Loading Type** drop-down list. Enter the intensity of the distributed weight or magnitude of the concentrated weight as the case may be, along with the location of the load.

Element Weights

If the structural model consists of plate elements representing entities like floor slabs, the pressure loads on those slabs can be considered for weights calculation for lateral load generation per UBC/IBC/other codes. This is done with the help of the Element Weights option. Its parameters include the magnitude of the uniform pressure, and the elements they are applied on. Since it is a weight, it is a quantity without a sign.

Reference Load

Reference Load cases which are described in AD.2007-1001.1.12 of the STAAD.Pro 2007 Software Release Report can be referred to using this option. Loads which are specified under Reference Loads can be used as weights for IBC.

Floor Weights

In many situations, a user may decide not to include the structural slabs in his/her analytical model. Hence, the model may be solely the skeleton framing system consisting of the beams, columns and bracing members.

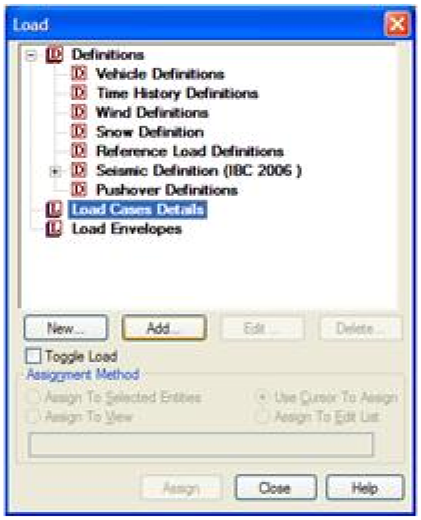
Under these circumstances, the loads which act on the slab can no longer be applied on the structure using the ELEMENT PRESSURE options. This is because there are no elements to represent the slab. So, an alternative is to apply the load using the FLOOR LOAD option. It is described in detail in TR.32.4 Area, One-way, and Floor Load Specifications (on page 2475).

Within a UBC/IBC/other codes definition, the FLOOR WEIGHT is the counterpart for the FLOOR LOAD just as MEMBER WEIGHT is the counterpart for a MEMBER LOAD, and an ELEMENT WEIGHT is the counterpart for an ELEMENT LOAD.

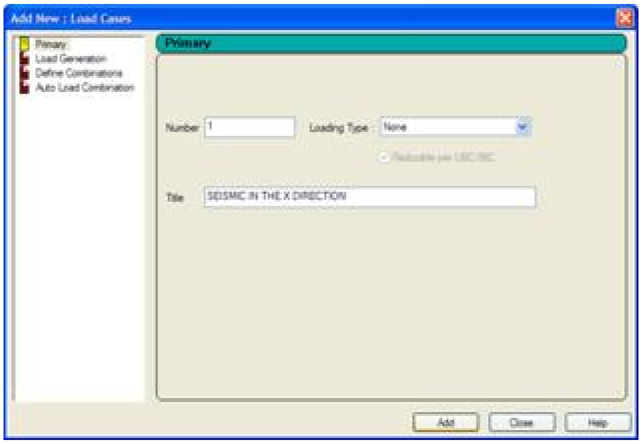
Note: Its parameters are hence very similar to what are found in a normal FLOOR LOAD definition. XRANGE, YRANGE and ZRANGE options allow the user to narrow in on panels at specific regions of the building. The pressure value is provided as a quantity without sign because it is contributing to the overall weight - a numerically positive term.

Once the seismic definition, including weights, has been specified, it should be included in a one or more primary load cases.

Select **Load Cases Details** and click on **Add**.

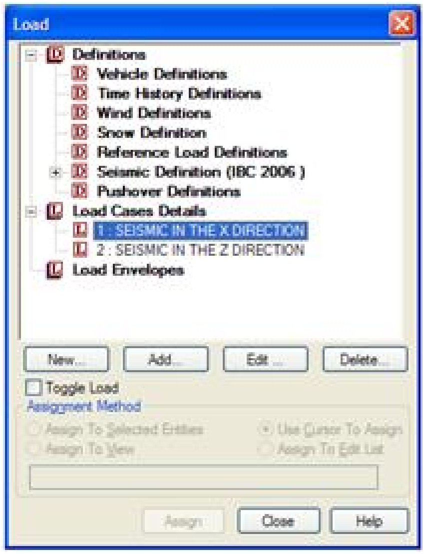


In the dialog box that is displayed, provide a **Title**, select the loading type as ‘seismic’ (although this is not strictly required, just useful for future reference) and click on **Add.**

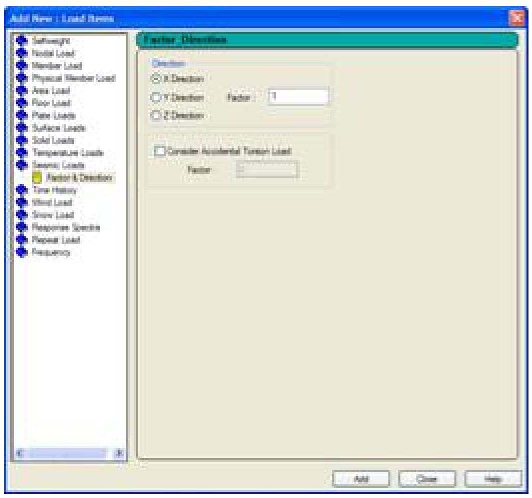


More load cases can be added in this manner.

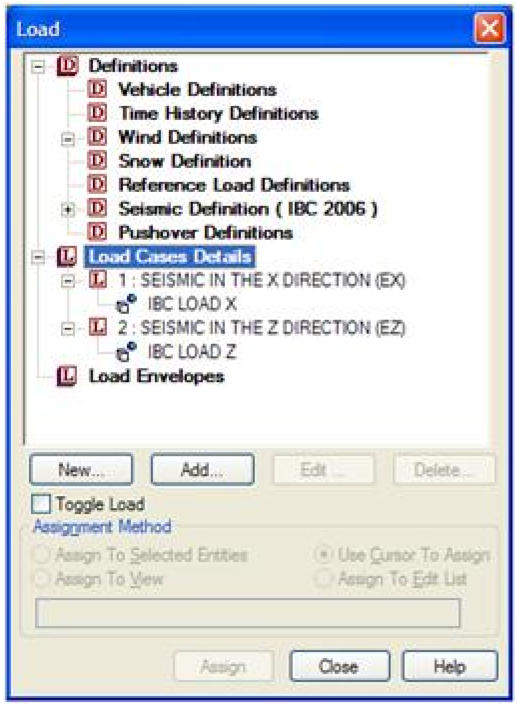
To add load items to our first load case, keep the expression **1: SEISMIC IN THE X DIRECTION** highlighted and click on the **Add** button.



Enter the **Factor**, **Direction**, etc. and click on **Add**.



The **Load** dialog box will display the new load item.



We can continue adding other load items to this load case in a similar fashion.

For more information on other UBC and IBC load definitions, see TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2349). To find more information about including the seismic loads in a primary load case, see TR.32.12 Generation of Loads (on page 2596).

AD.2007-03.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2007-03.2.1 Selfweight Command with a Member List

When considering stage construction, it is occasionally necessary to create load cases which include the effects only of the additional parts of structure. In order to do this the SELFWEIGHT command has been enhanced to allow a member list to be added which will be processed and only selfweight on that list will be considered.

This command may be used to calculate and apply the SELFWEIGHT of the structure for analysis. This allows you to enter body weight of the structural components. In other words, it allows different parts of the structure to be excited with different accelerations.

Refer to TR.32.9 Selfweight (on page 2496) for details on using member lists with the Selfweight command.

Example

```
LOAD 1 DEAD AND LIVE LOAD
SELF WEIGHT X 1.4 LIST 4 TO 10
SELF X 1.0 _PLATEGRP1
SELF
SELF Y -1.4
SELF X 1.4 ALL
SELF WEIGHT Z 1.4 _SOLIDGRP1
SELF Y -1.4 YR 10.5 10.51
SELF Y -1.4 X
```

In above example, the first specification includes factored weight of member (/plate/solid) 4 through 10 along global X direction and the second includes weight of all members (/plates/solids) associated with `group _PLATEGRP1`. The third command includes weight of all structural components towards gravity direction (`SELF WEIGHT Y -1.0`).

`SELFWEIGHT` specification for definition of static method of seismic load generator also is capable of accepting list. Only there is no direction specification for this command.

Similarly, the `SELFWEIGHT` specification when used in a Reference Load is also capable of handling a list.

AD.2007-03.2.2 Direct Analysis

The AISC 360-05 Appendix 7 describes a method of analysis, called Direct Analysis, which accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

This is a non-linear iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate, in each step changing the member characteristics until the maximum change in any Tau-b is less than the tau_tolerance, If the maximum change in any Tau-b is less than 100*tau_tolerance *and* the maximum change in any displacement degree of freedom is less than the disp_tolerance; then the solution has converged for this case.

There are two steps involved in setting up a Direct Analysis.

1. Specify the definition with a `DEFINE DIRECT` command.
2. Specify a direct analysis method with the command `PERFORM DIRECT ANALYSIS`

Note: Like all other analysis methods, by specifying the direct analysis parameters and only including a PERFORM ANALYSIS command, will result in only a first order elastic analysis, not a direct analysis to be performed.

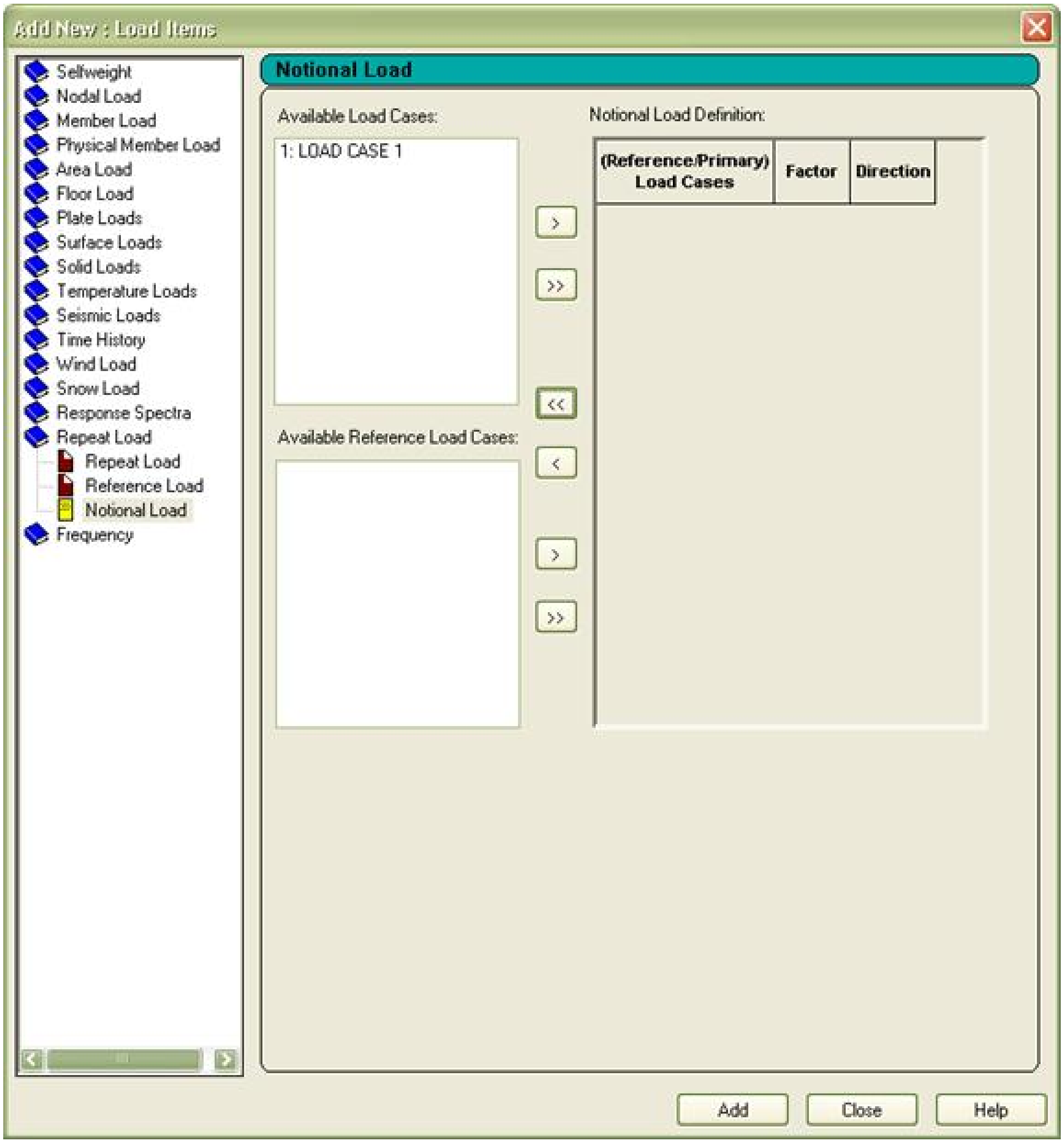
Related Links

- *M. To define direct analysis parameters* (on page 871)
- *A. To specify a direct analysis* (on page 953)

AD.2007-03.2.3 Notional Loads

A number of design codes require that a notional load be considered. Typically this is defined as a percentage of the gravity loads. What has been added to STAAD.Pro is an enhancement on the `REPEAT LOADS` command so that Primary and /or Reference load cases can be selected and a percentage of them can be applied in the appropriate global direction.

The addition of the Notional Loads command in the GUI is as an additional option on the **Repeat Load** option of the New Load Items, thus:



Both Principal and Reference load cases can be selected and moved into the Notional Load Definition where the required factor and direction can be specified.

The notional loads are calculated and applied as joint loads. Note that the actual values of the applied loading are not displayed in the GUI until after the analysis has been performed.

AD.2007-03.2.4 STP Parameter Added to AISC 360-05 Design

The AISC 360-05 design of wide flange depends on whether the section is formed from a rolled process or from plates welded together.

The parameter STP has been added to the list of parameters to indicate whether the section is built up from welded plates or from a rolling process. This is then used to determine the appropriate slenderness ratio as defined in table B44.1 from AISC 360-05. This parameter takes the following values:

1. Rolled Section
2. Built-up section

The default value is 1.0

Related Links

- *D1.A.6 Design Parameters* (on page 1100)

AD.2007-03.2.5 Updated and Additional Standard Steel Grades in Eurocode 3

The standard steel grades are referenced in the Eurocode 3 have been provided as standard steel grades in order that the appropriate buckling curves can be selected.

The steel grade can be specified using the parameter SGR which accept values of 0 to 5. This translates to the following:

Grade	SGR
S 235	0 (default)
S 275	1
S 355	2
S 450	3
S 460	4

Note: If a user-defined yield stress PY is specified, the buckling curve will still be determined from the SGR setting (i.e., if not specified, it will be based on the default SGR parameter). Therefore, if setting a user PY, this should be *preceded* in the file by the appropriate steel grade command.

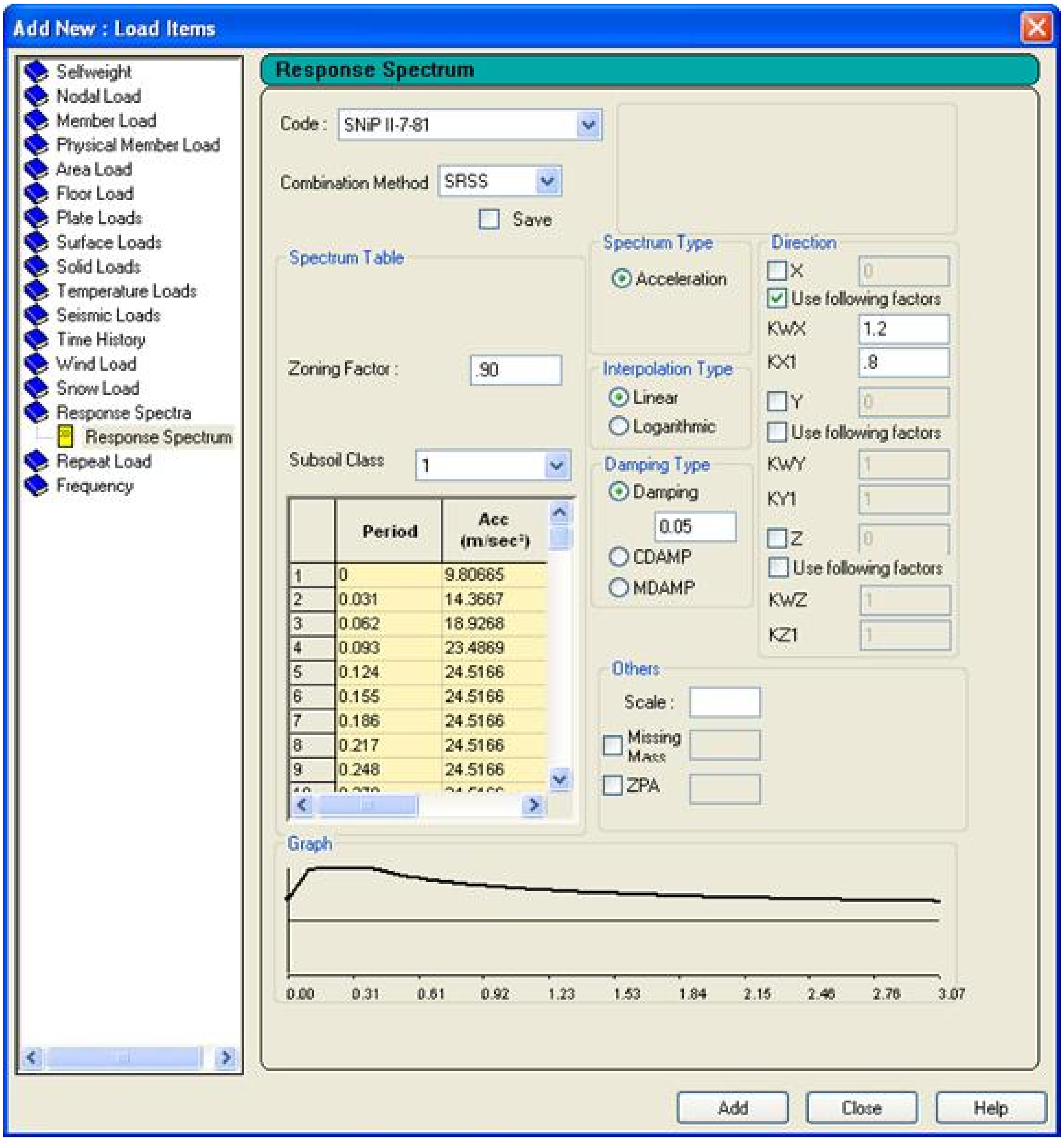
Related Links

- *D5.C.6 Design Parameters* (on page 1502)
- *D5.C.6 Design Parameters* (on page 1502)

AD.2007-03.2.6 Russian Response Spectrum

The Response Spectrum functionality has been enhanced with the ability to create response spectra as defined in the SNiP II-7-81.

To create a response spectrum as defined in the SNiP code, enter the **General | Load & Definition** Page and create or select the load case which is to be used. Click on the **Add...** button and select the **Response Spectra** option from the left panel.



Enter the required data in the provided fields and click on the Add button to include the command in the selected load case of the data file.

Related Links

- *TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81* (on page 2584)

AD.2007-03.2.7 Eurocode 3 Updated to Support Design of Slender Box Sections

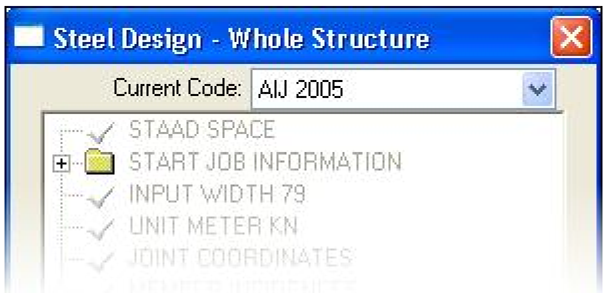
Both the ENV and DD versions of Eurocode 3 steel design need to design box sections that have slender elements.

Members that are specified with steel hollow box section properties that contain slender elements can now be designed to Eurocode 3. Previously, these members would not be designed and result in a warning in the output stipulating that slender members are limited to I section properties only.

AD.2007-03.2.8 AIJ 2005 Steel Design

The AIJ has updated the Japanese steel design code, AIJ 2005. This design code has been implemented and complements the existing Japanese steel design code AIJ 2002. The changes are minor and the code checking process remains the same as the previous AIJ design.

The design parameters are the same and apply as defined for the current AIJ, the only exception, however, is the **Current Code** parameter which must be set to **AIJ 2005.** This setting is available from the GUI list of codes.



Note that a design performed to the new AIJ 2005 standard is displayed in the output file (`*.ANL`) with the following header:

```
                         STAAD.PRO CODE CHECKING - (    AIJ 2005      )

************************************************
```

The equivalent header for a code check (or member selection) to the older standard is displayed thus:

```
STAAD.PRO CODE CHECKING - (    AIJ 2002      )

************************************************
```

AD.2007-1003.2.9 IBC 2006

In AD.2007-03.1.6 IBC 2006 (on page 256), the graphical screens containing the implementation of the seismic loading chapters of the IBC 2006 and ASCE 7-05 codes were described. In this section, the technical details and command syntax for the parameters of that feature are described.

Related Links

- *TR.31.2.14 IBC 2006/2009 Seismic Load Definition* (on page 2405)

AD.2007-03.2.10 IBC 2006 Response Spectrum

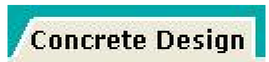
This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2006 edition of the IBC specification “International Building Code (IBC),” for dynamic analysis. The graph of frequency–acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

Related Links

- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)

AD.2007-03.3 Features Affecting the Concrete Design Mode

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.



AD.2007-03.3.1 ACI 2005 Beam, Column, and Slab Design

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-1003.3.2 AS 3600 Beam and Column Design

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-1003.3.3 EC2 2004 Beam and Column Design

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-03.4 Features Affecting the RAM Connection Design Mode

Several new features have been added and existing features have been modified in the RAM Connection Design Mode. These are explained in the following pages.

Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-03.4.1 Support of British Sections

To allow models created with British steel sections to be designed with RAM Connection, these sections need to be added to the supported US section database.

The ranges of supported sections are:

US Sections

Wide Flange

W Shape - W
M Shape - M
S Shape - S
HP Shape - HP

Tee

Tees cut from W sections - WT
Tees cut from S sections - ST
Tees cut from M sections - MT

Channel

Channel - C
MC Channel - MC

Angle

Angle - Equal and unequal, L and LU
(Double angle, T2L and T2LU, but with fixed spacing)

Hollow

Tube - TUBE
Pipe - PIPE
HSS Rectangular - HSS_RECT
HSS Round - HSS_RND

Not supported

B Shape
Castellated
Solid Round
Cable

British Sections

Wide Flange

UB Shape - UB
UC Shape - UC
JO Shape - Joist

Tee

Tees cut from UB sections - TUB
Tees cut from UC sections - TUC

Channel

Channel - PFC

Angle

Angle - Equal and unequal, EA and UEA

Hollow

Tube - Rectangular and square, SHS and RHS

Pipe - CHS

Not supported

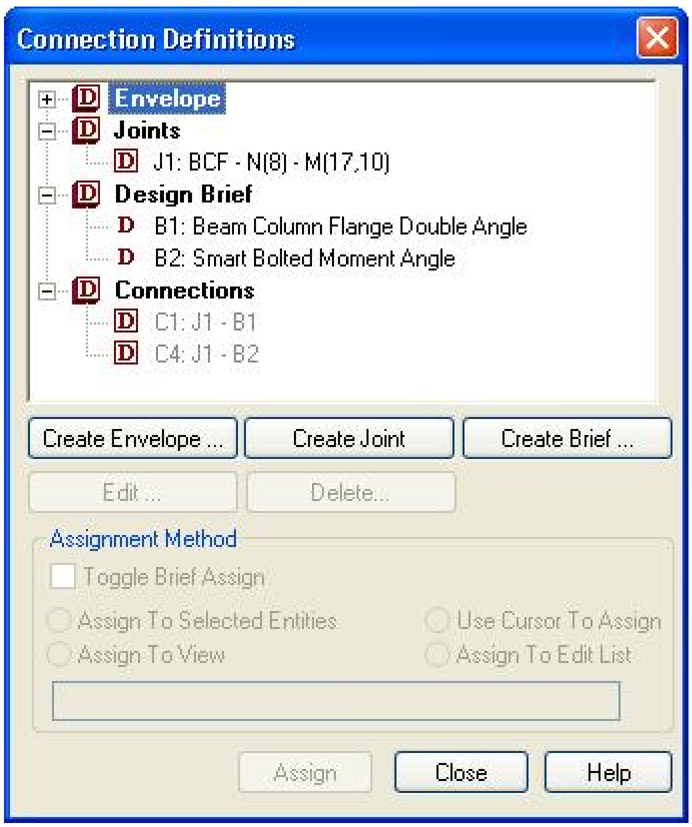
UP Shape

AD.2007-03.4.2 Support of Multiple Connections at a Joint

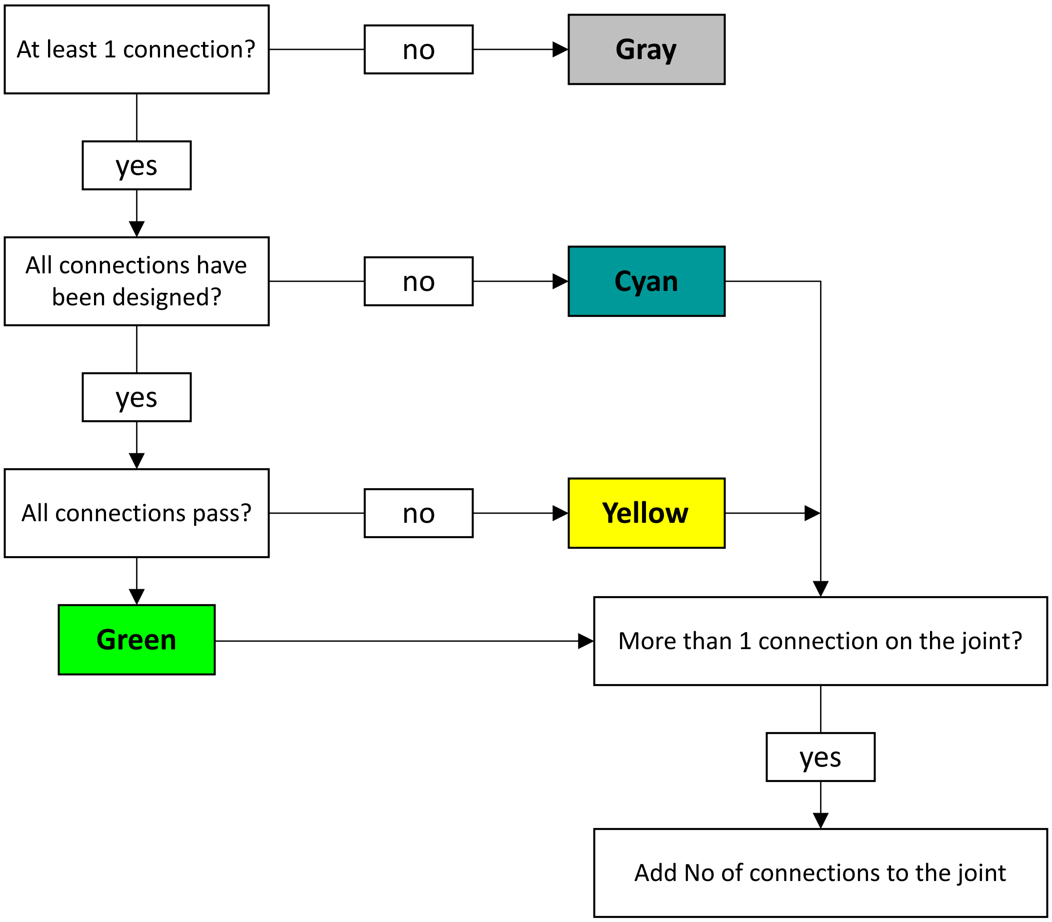
RAM Connection designs separate parts of a connection with different templates such as a Shear Plate Connection and a Moment Connection. Therefore the RAM Connection mode has been enhanced to allow multiple Connections to be assigned to a single joint.

The following is an enhancement of the new feature added in STAAD.Pro 2007 Build 1001. Refer to AD. 2007-1001.5.1 RAM Connection Design Mode.

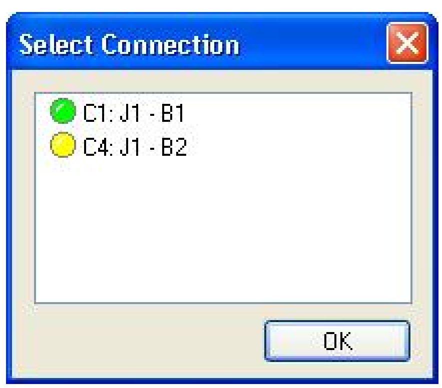
If a design brief is assigned to a joint that already has one assigned, then rather than replacing the existing design brief, an additional connection will be defined.



When multiple connections exist at a joint, the information is displayed such that the colour coding of the connection uses the following logic:



When selecting a joint that has multiple connections associated with it, the following dialog box is displayed that shows the associated connections from which the required connection should be selected:



Note: Each connection is designed and detailed independently

STAAD.Pro 2007 Build 1001 Release Report

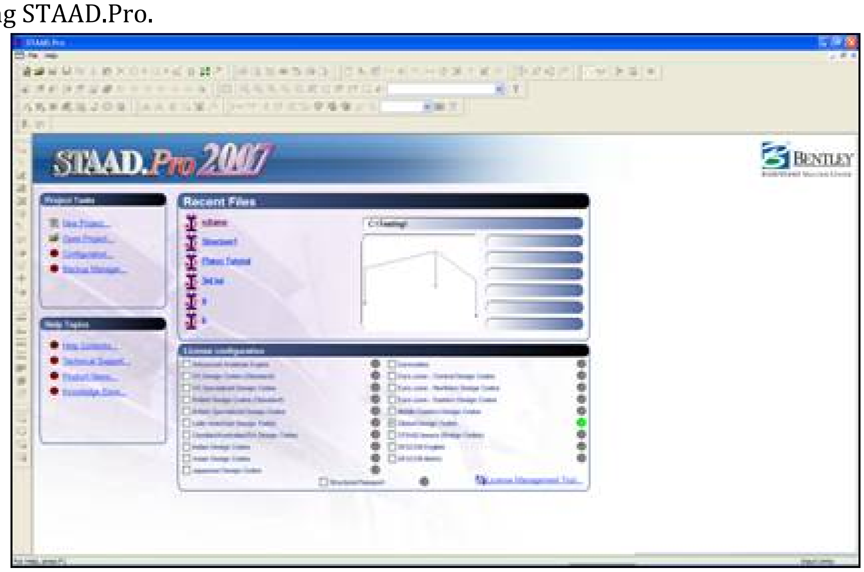
The Software Release Report for STAAD.Pro 2007 Builds 01 + 02 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1005. This document should be read in conjunction with all other STAAD.Pro manuals, including the Revision History document.

AD.2007-1001.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

AD.2007-1001.1.1 New Start Page

STAAD.Pro now includes a start page with access to the functions normally required when first starting STAAD.Pro, including shortcuts for starting a new file, accessing the recently accessed files, launching the help file and configuring STAAD.Pro.

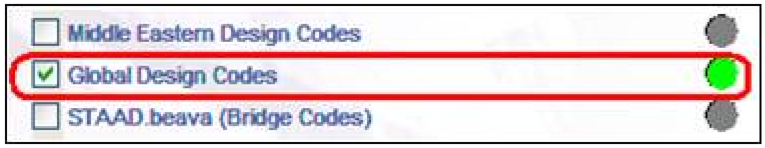


The new Start Page is divided into 4 sections that can be used to achieve the following:

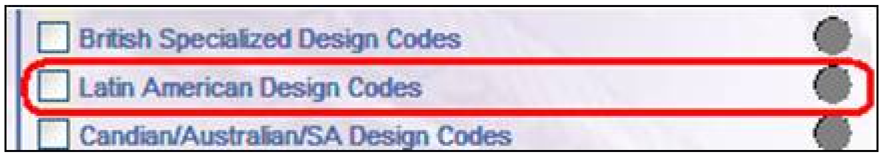
1. **Project Tasks**, to:
   a. Start a New Project using the STAAD.Pro wizard.
   b. Open an existing file using the traditional windows browse dialog enhanced with a model preview window.
   c. Set the program behavior with the Configuration options.
   d. Setup the automatic Backup configuration requirements.
2. **Recent Files** to preview and access the last 6 models opened.
3. **Help Topics**,
   a. Quick access to the online Help document.
   b. Locate the technical support centers and contact details.

c. Find out the latest information on the program online from the Product News link.
d. Access the growing STAAD.Pro online knowledge base.

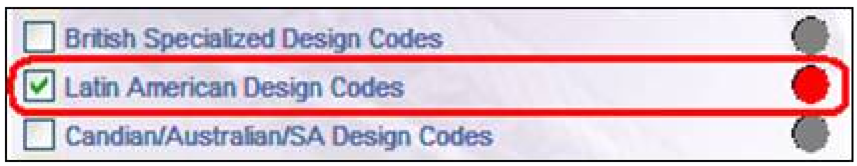
4. **License Configuration**. To quickly identify which SELECT licenses are being used by the current session of STAAD.Pro, they are displayed and color coded on the Start Page thus:
   - If the license is available it is marked with a green circle thus:



   - Licenses that have not been selected are marked with a grey circle thus:



   - If the selected license can not be obtained or is not available from the server will be shown with a red circle, thus:



Additional configuration of the Bentley SELECT license, such as specifying the server name and activation key, can be viewed and set using Bentley License Management Tool which can be accessed from the link at the bottom right of the License Configuration section.

AD.2007-1001.1.2 Enhanced Grid Tool

The Snap/Grid Node tools have been enhanced to

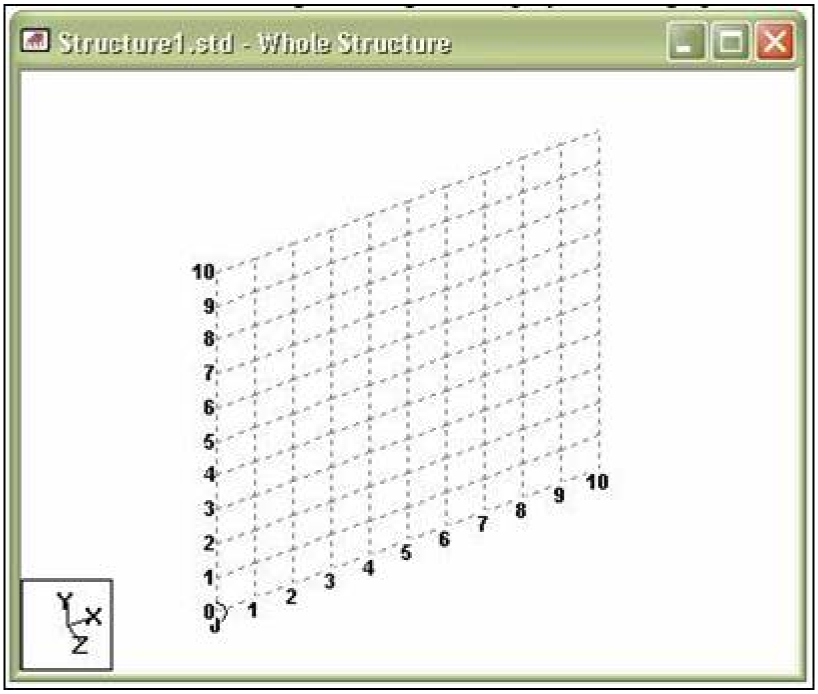
1. Allow multiple different grids to be created.
2. Import a DXF file and use it as a template.
3. Import grid files created in another STAAD.Pro model

Beams, plates (both triangular and quadrilateral) and 8 noded solid elements can be generated using the appropriate Snap/Grid Node tool.

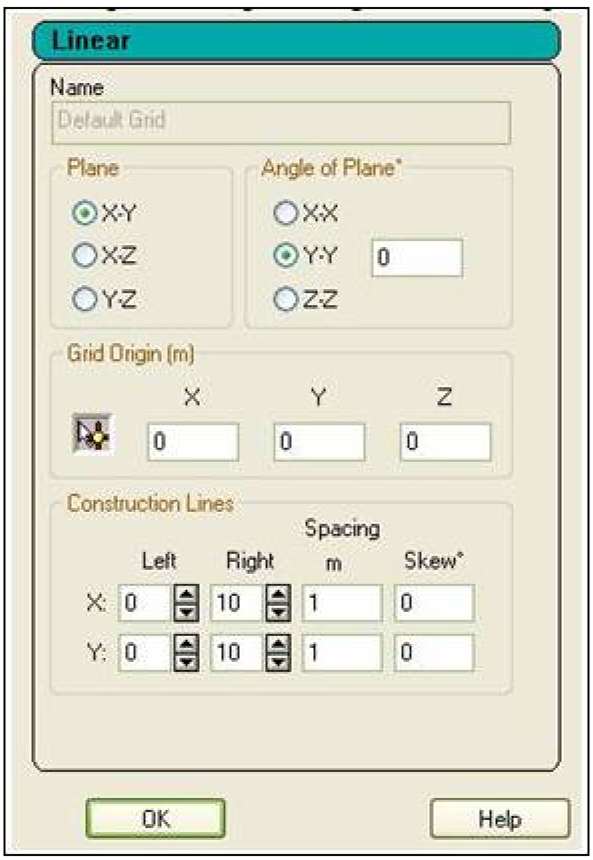
When this function is launched, the following dialog is opened which will include a Default Grid. This grid will be of type ‘Linear’, there are also options to create Radial, Irregular and imported DXF grids which will be described later.

As new grids are added or modified, the information is stored in the STAAD.Pro data folder with a GRD extension that allows other STAAD files to re-use these defined grids.

The effect of the current grid settings are displayed in the graphics window, thus:

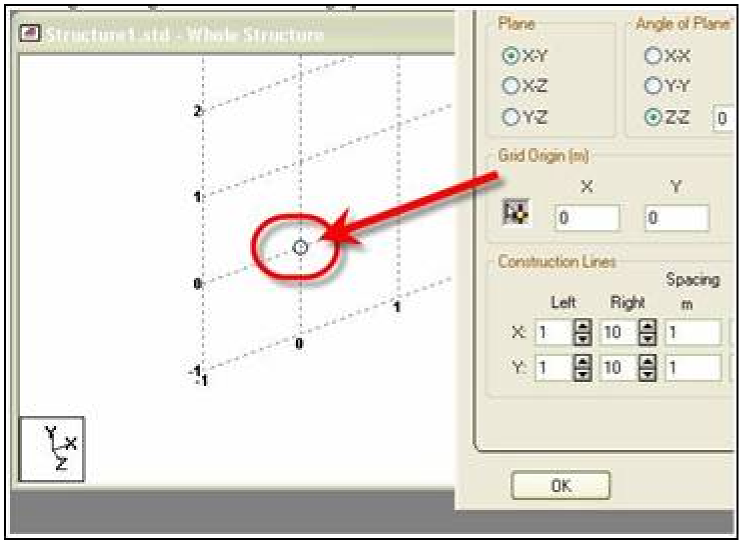


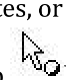
To change the settings of this grid click on the **Edit** button to display the current grid properties, e.g.:



The current plane of the grid is set by selecting the required option. This can be rotated about one of the global planes by selecting the axis of rotation and setting the angle.

The origin of the grid is marked on the graphics with a small circle:



The location of the origin, specified in global co-ordinates, can either be defined explicitly in the given X, Y and Z co-ordinates, or it can be set to the co-ordinates of an existing node by clicking on the icon,  (the cursor changes to ) and then on the node itself in the graphical window. Note that at this point the origin co-ordinate is updated.

The construction lines are used to specify how many gridlines are created either side of the origin, the spacing between the gridlines and if there should be a skew in degrees along either axis.

Click on the **OK** button to accept these settings.

Additional grids can be defined by clicking on the [Create…] button. Three different types of standard grid can be created:

- Linear
- Radial
- Irregular

The type of grid required should be selected from the drop list of types available at the top of the property sheet.

Each new grid should be identified with a unique name for future reference. The functionality for each type of grid is thus:

Linear

The Default Grid defined above is a Linear Grid and thus see above for the settings of a Linear Grid.

Radial

The settings for a Radial grid are defined in the following window:

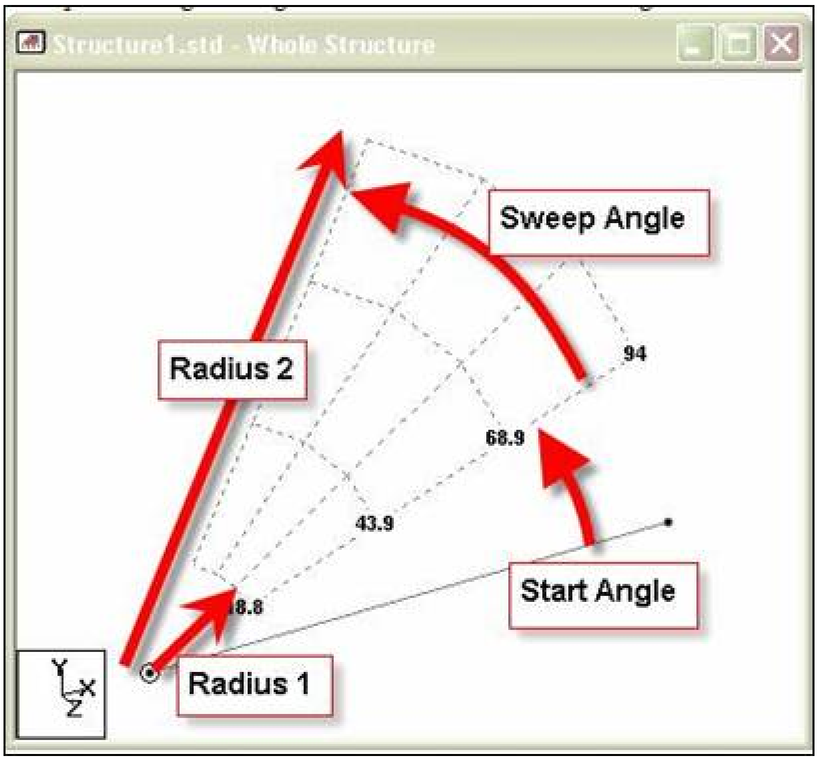
Modeling

The Plane, Angle of Plane and Grid origin options are as for the Linear (or Default Grid).

The construction lines options

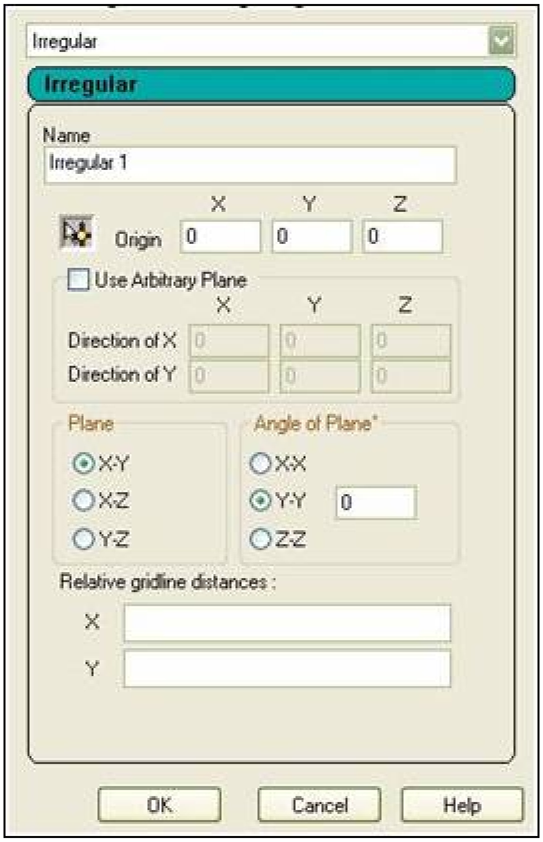
Start Angle, is the angle in degrees about the orthogonal axis to the plane from the axis first referred to in the definition of the plane. For examle, if the selected plane is X-Y, then the angle is measured about the Z axis (using the right hand rule) from the axis parallel to the X axis.

Sweep is the angle in degrees measured from the start angle which is divided into the selected number of Bays, thus:



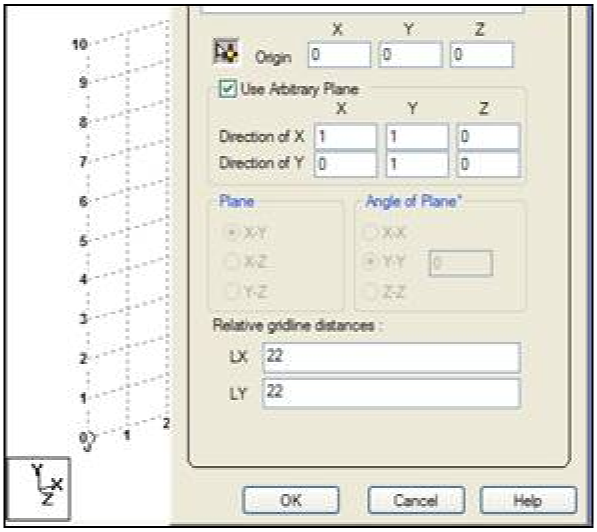
Irregular

The settings for an Irregular grid are defined in the following window:



The origin is set as described above for both Regular and Radial grids.

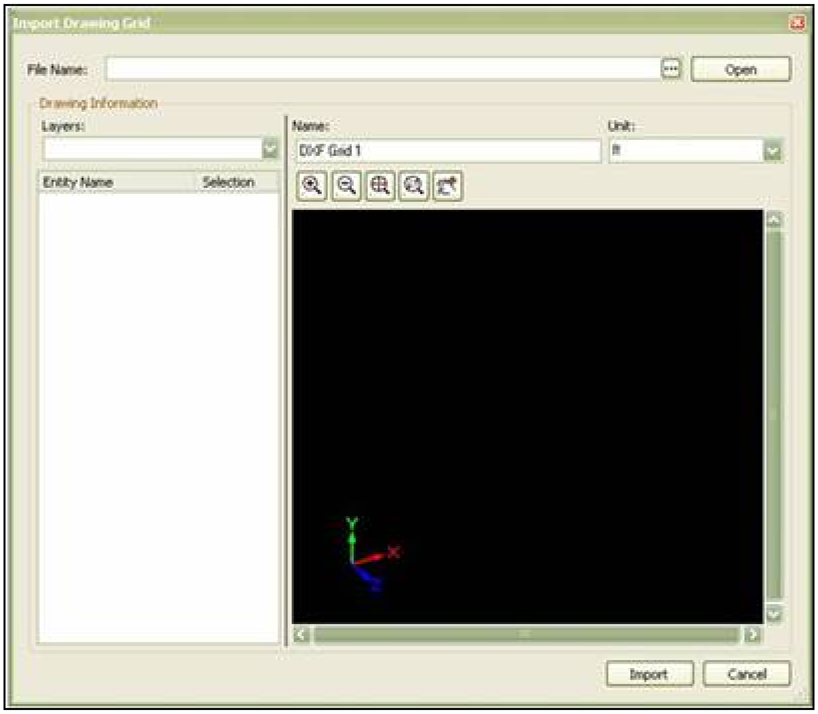
The plane of the grid can either be set in one of the global planes X-Y, X-Z or Y-Z and rotated about one of the global axes. This method is identical to that described for the Regular or Radial Grids. Alternatively, the directions of the two axes can be specified as relative co-ordinates from the origin:



The gridlines are defined by the distance to the next gridline and the numbers separated with a space.

There are two import options that can be selected that can allow either DXF files or grids defined in another STAAD.Pro model (all but the default will be imported).

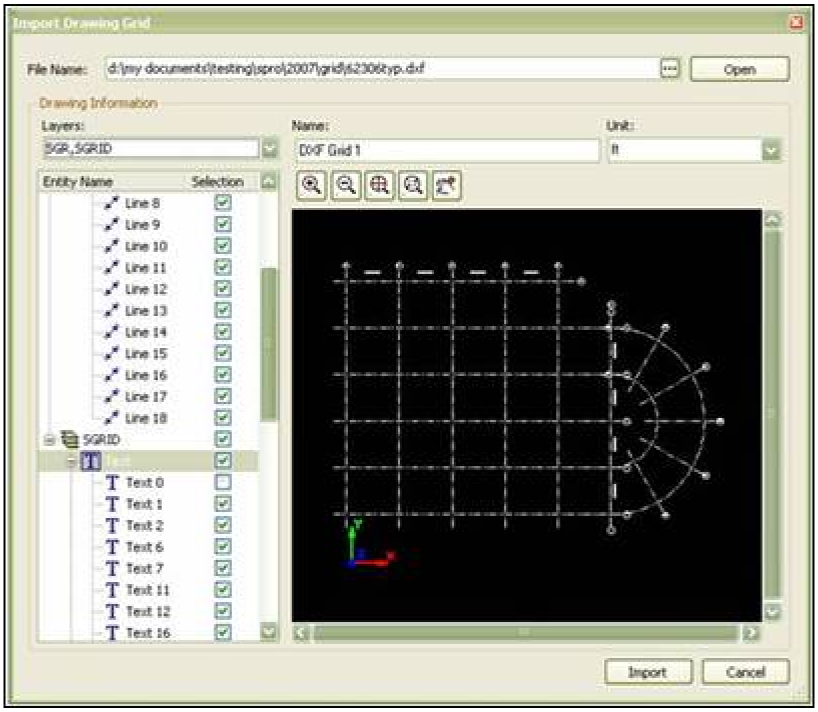
The option to import a DXF file will open the following dialog:



To select a DXF file click on the **[...]** button and navigate to the required file.

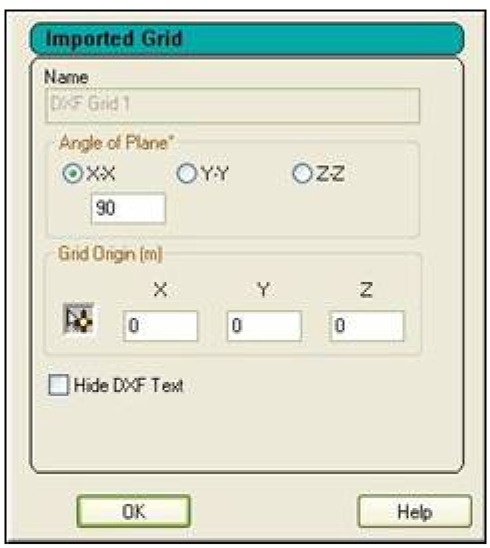
The file will be opened and displayed in the preview window. Individual layers can be turned on and off from the Layers droplist. The individual entities in the selected layers are displayed and can be toggled on or off for import.

By clicking on an entity in the graphical window, the entity is highlighted in the table so that it can be turned off if required.

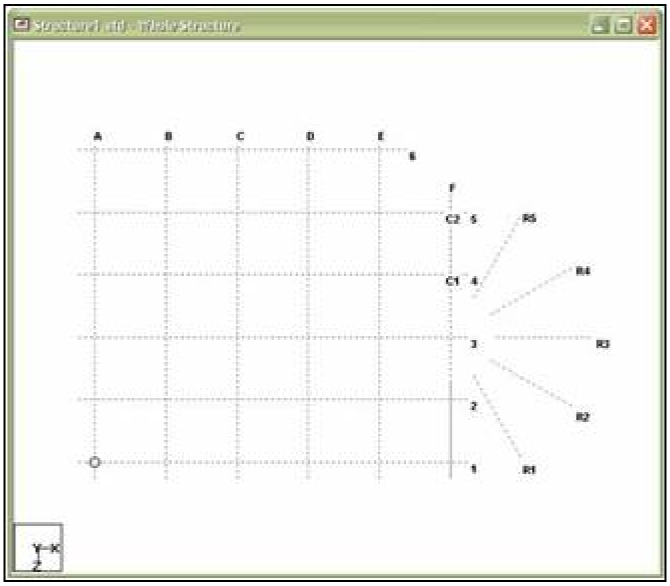


With the required entities selected, a suitable reference name supplied and unit selected, click on the **Import** button.

The data will be imported in the plane in which it was defined in the original DXF. However, if required this can be rotated about any of the global axes. Also, the origin of the grid can be located at any 3D co-ordinate.



The option to Hide DXF text can be used to toggle the display of grid labels if they start clashing with the rest of the model. The grid is displayed thus (Note curved lines are currently not imported):



The DXF grid operates as the other forms of grid in that when the **Snap Node/...** button is clicked, nodes can be created at the ends and intersections of grid lines.

The second import option is to Import Grids previously defined in another STAAD.Pro model. Selecting this option opens a browse dialog box to identify a GRD file created by the Snap Node Grid tool. Note that GRD files are only created by STAAD.Pro 2007 (or later).

Icons on the **Geometry** toolbar are:

 **Snap/Node Beam**

 **Snap/Node Triangular Plate**

 **Snap/Node Quadrilateral Plate**

AD.2007-1001.1.3 Fly-out Toolbars

The amount of screen space occupied by a number of toolbar icons has been recovered by collapsing a number of similar icons into a single icon.

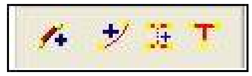
The active icon can be changed by holding down the left mouse button when clicking on the button. Icons that have this property are identified with a black triangle in their lower right corner:

There are four Geometry Toolbar icons that have this propery:

- Add Beam
- Add Plate
- Add Solid

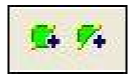
- Snap Node Grid

The **Add Beam** icon supports four commands:



- Add Beam from Node to Node
- Add Curved Beam
- Add Beam Between Mid Points
- Add Beam Using Perpendicular Intersection

The **Add Plate** icon supports two commands:

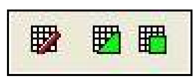


- Add Quadrilateral Plate
- Add Triangular Plate

The **Add Solid** icon supports five commands:

- Add 8 Noded Solid
- Add 7 Noded Solid
- Add 6 Noded Solid
- Add 5 Noded Solid
- Add 4 Noded Solid

The **Snap Node Grid** icon supports three icons:



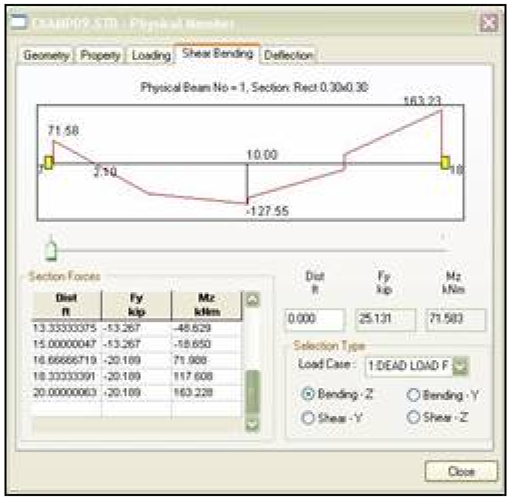
- Snap Node Beam
- Snap Node Quadrilateral Plate
- Snap Node Triangular Plate

AD.2007-1001.1.4 Physical Member Query

The addition of the Physical Member to the range of objects supported by STAAD.Pro requires that there is access to a query tool to get information about the Physical Member.

To access information on a Physical Member, use the **Physical Member Selection** Tool  to highlight the required object and then select the menu option: **Tools** > **Query** > **Physical Member**

to highlight the required object and then select the menu option:



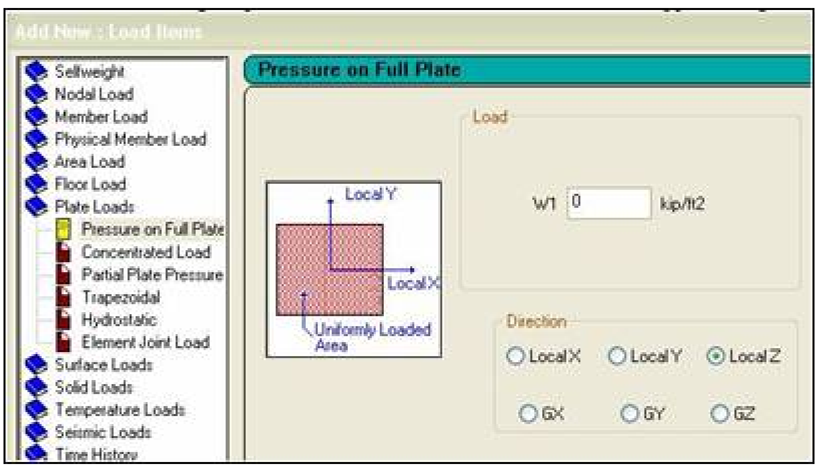
This dialog box can also be accessed from

- Double clicking on a physical member with the **Physical Member Cursor**
- Selecting the physical member, right-clicking and selecting **Properties...** from the pop up menu.

AD.2007-1001.1.5 In-Plane Area Loads on Plates

STAAD.Pro can now apply plate loads in the local X and Y directions to represent in plane friction loads.

As shown in the next figure, pressure on the full element can now be applied along local X and Y axes.



Thus a -2 kip/ft^2 load applied to an element that is 3 ft^2 will result in -6 kips being applied in the plane of the element.

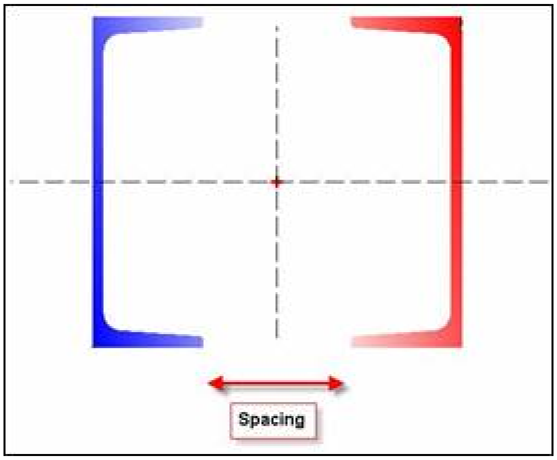
This applies only to Pressure on Full Plate.

Related Links

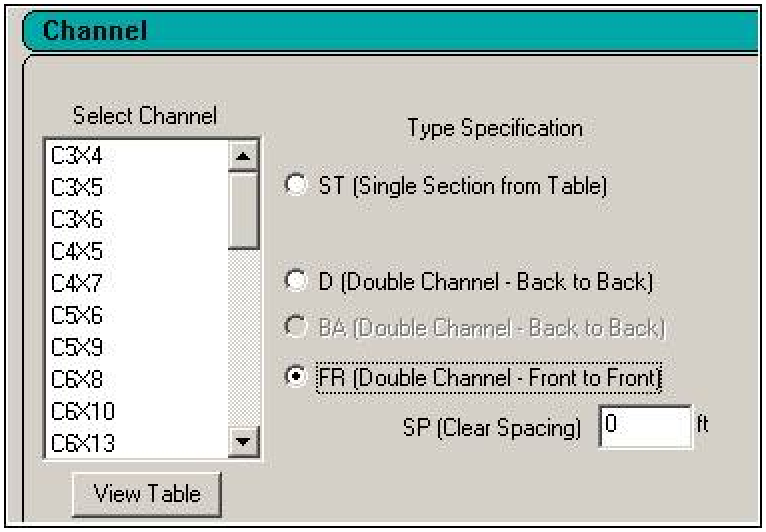
- *TR.32.4.1 Area Load Specification* (on page 2476)

AD.2007-1001.1.6 Front-to-Front Channels

STAAD.Pro has been enhanced to support the definition of steel channel sections being defined in an arrangement with the toes pointing to each other with a given spacing.



Front to front channel steel sections can be defined in the Properties dialog box by selecting the required channel section and choosing the **FR (Double Channel – Front to Front)** option and any spacing required between the channels.



The command that appears in the STAAD.Pro STD data file is:

```
<Member List> TABLE FR C4X5 SP 0.5
```

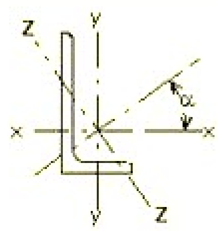
Note: The SP parameter is optional, but if it is not set, the section will *not* be assumed to be a closed box for torsional calculations.

AD.2007-1001.1.7 Automatic Property Calculations for User-Provided Table Angle Sections

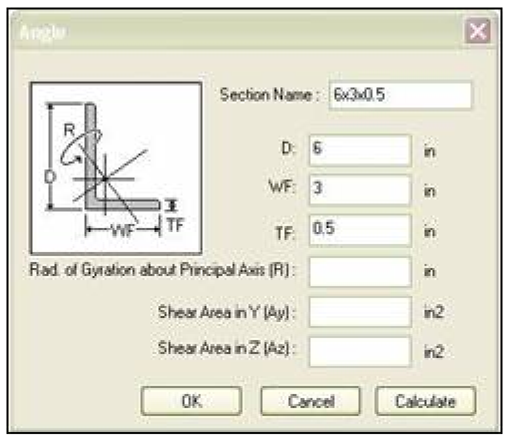
User-Provided Tables (UPTs) of Angle sections require section properties that can now be calculated when the section dimensions have been defined.

Angle Section UPTs require the following data to be entered for each section:

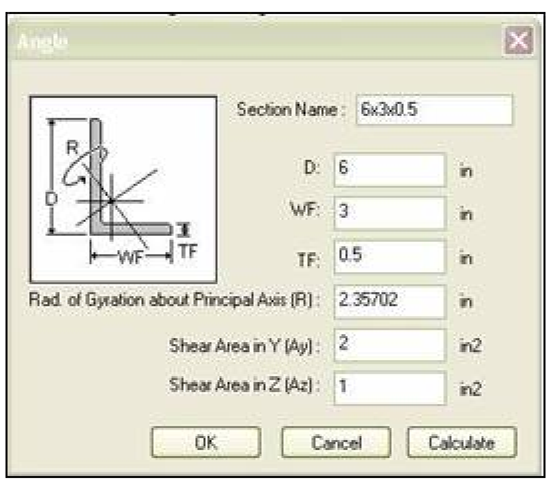
1. **D, Depth of angle**
2. **WF, Width of angle**
3. **TF, Thickness of flanges**
4. R, radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual (see below). This must not be set to zero.
5. AY, Shear area long Y axis
6. AZ, Shear area along Z axis



Define the dimensions of the angle (shown in bold above):



Then click on the **Calculate** button to have STAAD.Pro calculate the remaining properties provided:



Note: When these sections are added to a STAAD.Pro model, the angle will be oriented such that the Z-Z axis as defined in the AISC manual will align with the local Z axis.

AD.2007-1001.1.8 Consolidation of Multiple Property References

When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command.

STAAD models that have the same property defined multiple times can be consolidated by clicking on the new menu item, **Tools** > **Merge Properties**.

A warning is given and thus an opportunity to cancel the property collation if not required or selected by mistake.

All instances of a given section property will be collated into a single property reference.

Notes:

- Properties references with differing additional parameters will not be collated.
- Properties references with differing assigned material properties will not be collated.

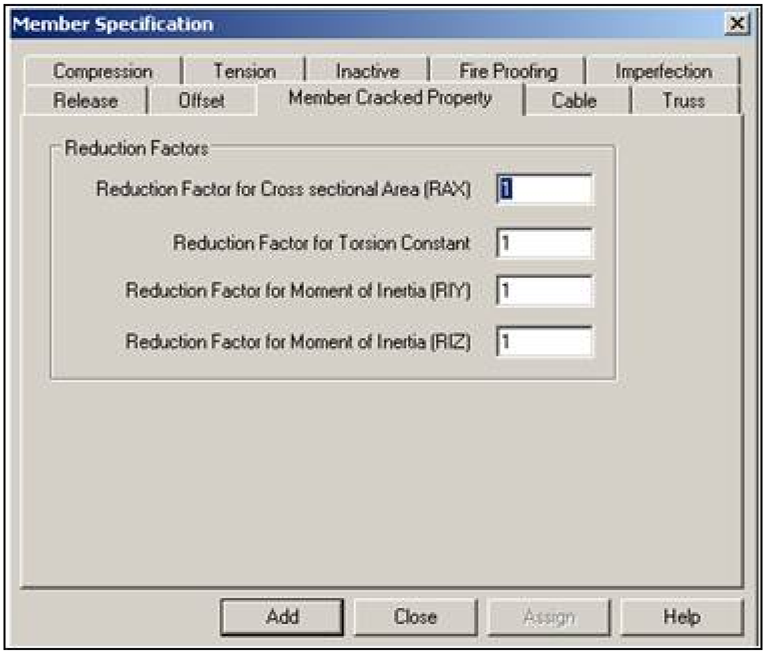
AD.2007-1001.1.9 Section Property Reduction in Analysis to Account for Cracking

Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. Though the methodology to handle cracked section properties is non-linear in nature, i.e. the section capacities should be checked and modified depending upon the section forces the section is handling. The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

In STAAD.Pro this approach has been simplified as per the recommendations in the ACI-318: 2005 standard which suggests a user input of reduction factors for the individual members. Section 10.11.1 of ACI-318 has provided a list of suggested reduction factors of section properties dependent upon the nature of stresses the member is subjected to.

An additional sheet has been added to the Beam Specifications dialog to allow the reduction of section properties for analysis to be created and assigned. The method is identical to that for creating any other beam specification.

The Specifications dialog can be accessed from the **General | Specifications Page** in the Modeling Mode:



The reduction factor should be a fraction of unity; hence a factor of 0.5 defined for RAX will reduce a defined cross sectional area of 0.5 ft^2 to 0.25 ft^2.

The format of the command that is generated in the STAAD.Pro STD file is:

```
MEMBER CRACKED
<Member List> REDUCTION { RAX | RIX | RIY | RIZ} factor
```

Multiple factors can be assigned on the same line

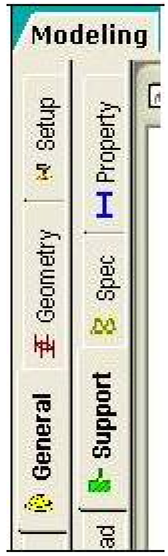
Note: The reduction factor is considered only for analysis but not for design.

AD.2007-1001.1.10 Tension/Compression-Only Spring Support

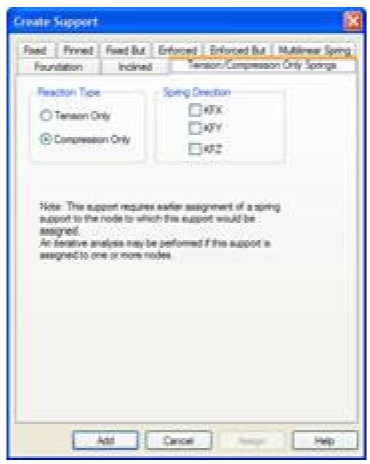
STAAD.Pro can now graphically define spring supports that are to allow tension only or compression only forces.

The Tension/Compression Only Support command is created from the **Supports** dialog box.

The **Supports** dialog can be accessed from the **General | Support** page in the Modeling mode.



Click on the **Create** button and select the **Tension/Compression Only Spring** sheet thus



The selection of Reaction Type indicates that if, after any of the cycles of analysis, the direction of the force in the spring is of the wrong 'type', then the support will be removed from that direction and a new analysis performed.

Reaction Type	Tension Only	Compression Only
Support will remain if the reaction is	- ve	+ ve
Support will be removed and a new analysis flagged if the reaction is	+ ve	- ve

This support definition should be either 'added' to the Supports dialog (by clicking on the **Add** button) or assigned to the currently selected nodes (by clicking on the **Assign** button) that have previously been assigned with spring supports. If the support is not assigned as it is created, it can be assigned later from the Support dialog. However, note that if it is not assigned to at least one spring support when STAAD.Pro is closed, then the definition will not be saved in the STD file.

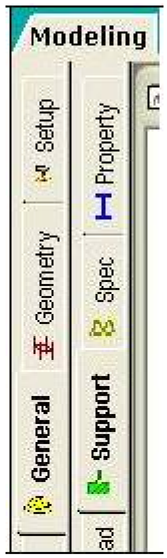
AD.2007-1001.1.11 Enhanced Elastic Mat Plate Mat Options

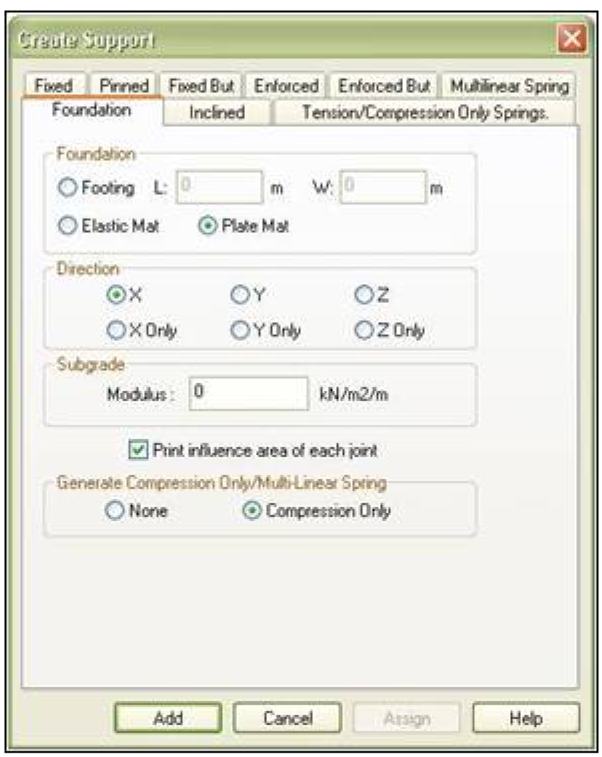
The Elastic Mat and Plate Mat commands can be set to behave as compression only springs and include the influence area that each node has been subjected to in the ANL output file.

In order to allow Elastic Mat and Plate Mat commands to perform as compression only supports to model lift of support situations, the command has been enhanced with an additional parameter which can be set graphically from the command definition.

The Elastic Mat and Plate Mat commands are created from the Foundation sheet in the **Supports** dialog box.

The **Supports** dialog can be accessed from the **General | Support** page in the Modeling mode.





When the Compression Only option is set, then if after any of the cycles of analysis, the force at a node included in the command range (in the elastic mat range or used to define a plate in the plate mat range) is found to be tensile (i.e., negative reaction), then the load case is marked for a re-analysis with that support removed.

There is also a new option to include in the output file, the area that has been used in the calculation of the spring stiffness of each joint used when defining a Plate Mat or Elastic Mat command.

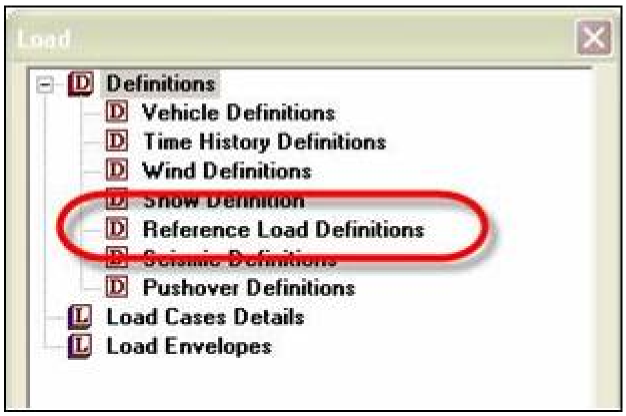
Related Links

- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2319)

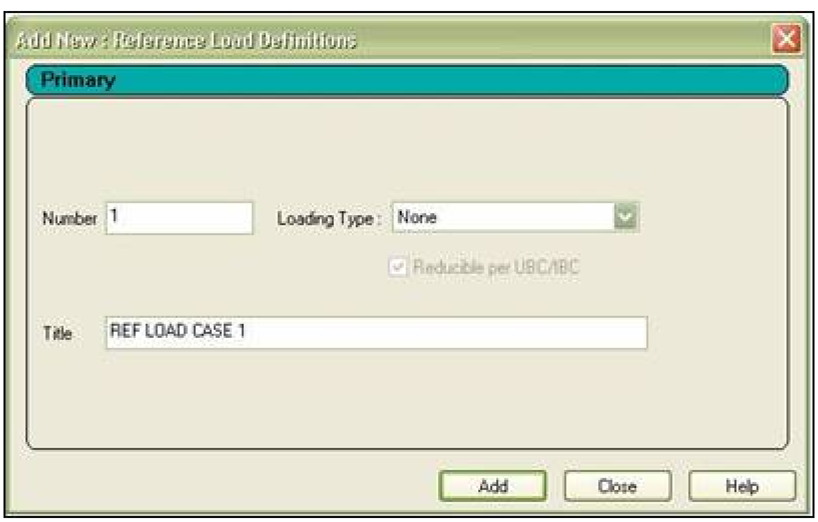
AD.2007-1001.1.12 Reference Load Cases

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. This is similar to a `REPEAT LOAD` command, but has the added benefit of not being solved in its own right.

A reference load case is listed in the Load dialog box of the General Load Page of the modeling mode. A Reference load case is listed in the Definitions section of the data file and displayed thus:

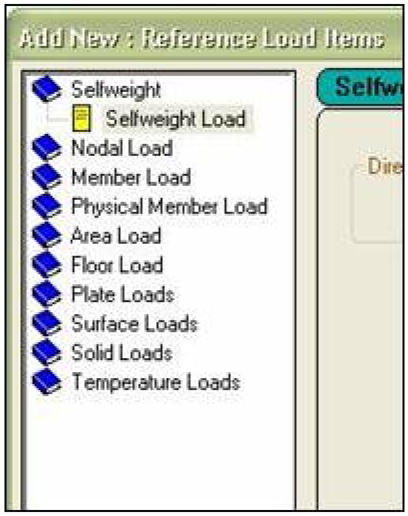


To add a new Reference load case, highlight the **Reference Load Definitions** in the **Load** dialog and click on the **Add** button. Provide the title for the load case and number (however, similar to that of creating Primary load cases, the next available reference load case number. Note that these reference load case numbers can be the same as a primary or combination load case number (however a combination load case number cannot be the same as a primary load case number).



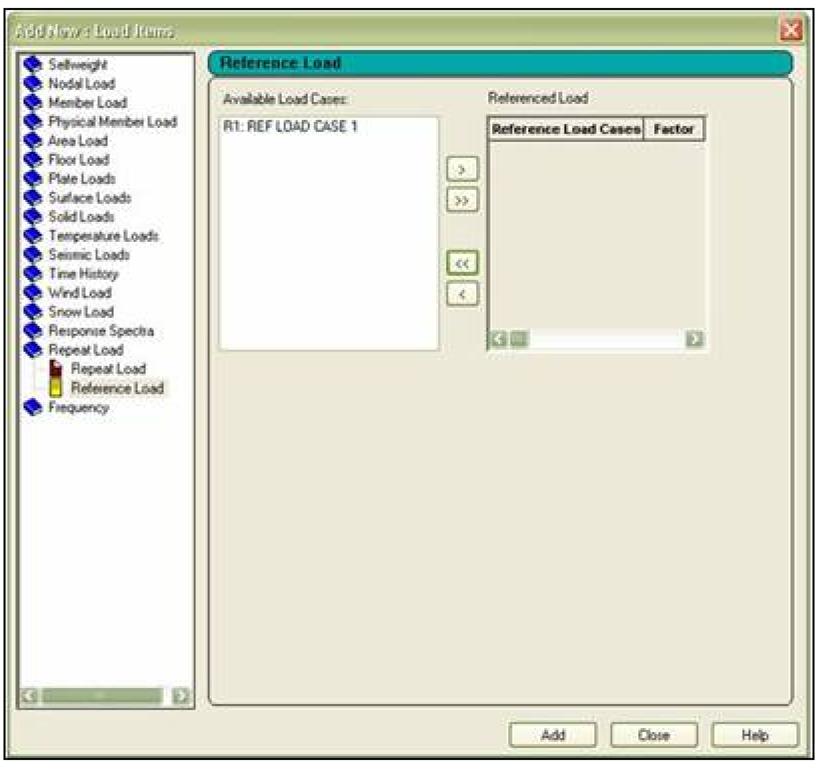
The Reference load case names are shown in the **Load** dialog box. Load items should then be added to this load case in exactly the same way as adding load items into a primary load case.

Select the Reference load case and click on the **Add** button to display all the load items that can be added to this load case, thus:



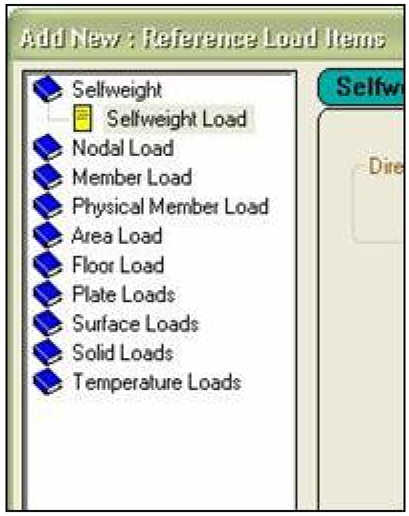
To be analyzed, the loads defined in a Reference load case must be added to one or more Primary load cases. This is done by selecting the required primary load case and clicking on the **Add** button.

Select the load item **Repeat Load** > **Reference Load**:



Select the required defined Reference load cases and click on the **[>]** button to select them for inclusion in the current load case. The **>>** button will select all defined Reference Load cases.

These are added with a factor of 1.0, however, each can be modified with its own specific multiplication factor:



The format of the definition of a Reference Load (i) in the data file is thus:

```
DEFINE REFERENCE LOADS
LOAD R(i) LOADTYPE (type)  TITLE REF LOAD CASE 1
(Load items)
…
END DEFINE REFERENCE LOADS
```

The format of a reference to a Reference Load in a primary load (j) case is thus:

```
LOAD (j) LOADTYPE (type) TITLE LOAD CASE 1
REFERENCE LOAD
R(i) 1.0
…
```

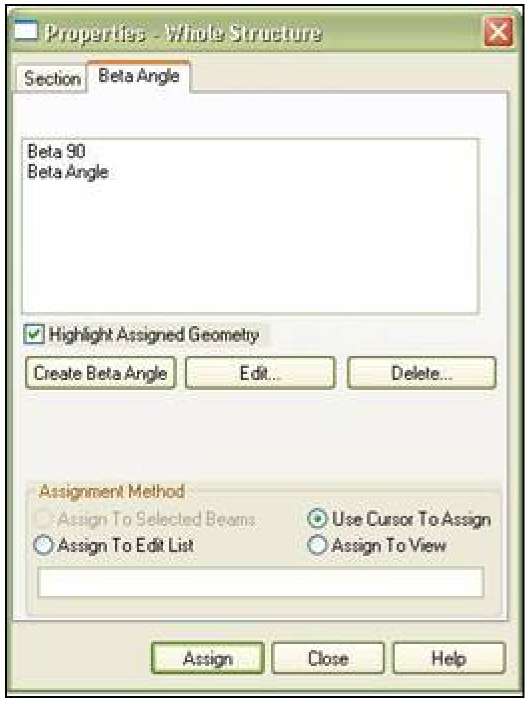
Related Links

- *Repeat Load tab* (on page 2863)
- *TR.31.6 Defining Reference Load Types* (on page 2453)
- *M. To create a reference load* (on page 893)

AD.2007-1001.1.13 Enhanced Beta Angle Definition and Assignment

Assigning beta angles on members has been improved visually by displaying the commands that are stored in the STAAD data file, making them easier to manage.

Beta angle definitions are created and listed in the Beta Angle sheet of the Property dialog box. Each new command is listed in the dialog as shown below and can be assigned using the assignment methods commonly used throughout STAAD.Pro:



The creation of beta angle commands includes three options:

1. Define specific angle to rotate the beams about their local X axis.
2. Specify the command 'Angle'(*)
3. Specify the command 'RAngle' (*)

(*) The Angle and RAngle commands are specifically for equal and unequal angle sections. All sections are aligned with their principal axes aligned with the global axes, however, angle sections are often required to align their flanges with the global axes. By assigning either the Angle or RAngle command, the section will be rotated to align the flanges. For more information, see TR.26.2 Specifying Constants for Members and Elements (on page 2305).

Related Links

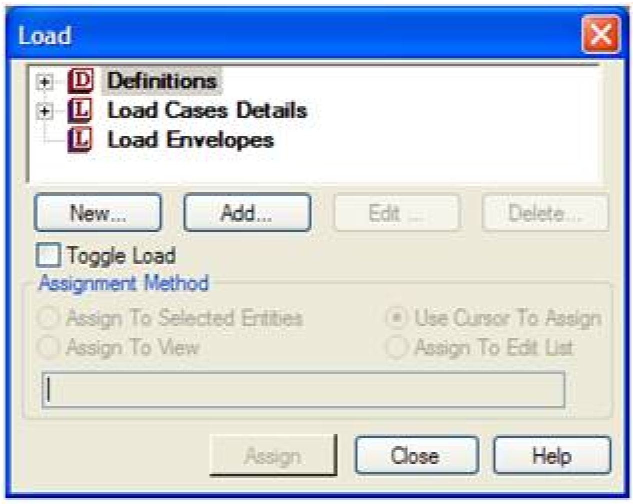
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

AD.2007-1001.1.14 IBC 2006 Equivelant Lateral Force Procedure

The static equivalent method for performing dynamic analysis per the IBC 2006 code has been implemented in STAAD.Pro 2007 Build 02.

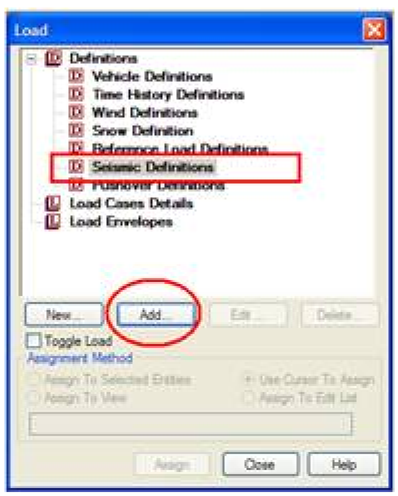
This option can be accessed from the **General | Load** page as explained below.

When the **General | Load** Page is selected, the right hand side of the screen will display the following if no load cases exist in the model.



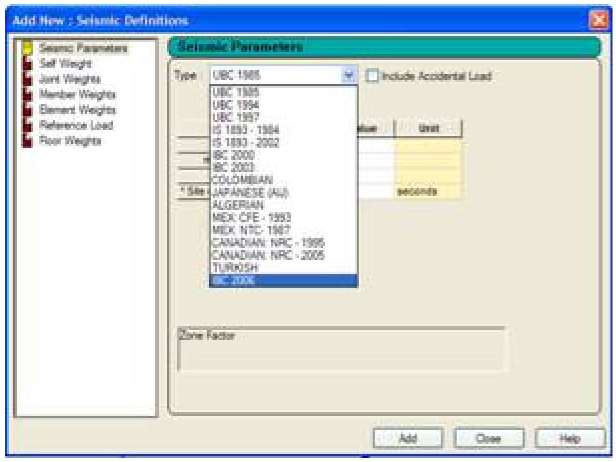
Definitions contains the options through which one creates the “Define” block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases.

When the tree view is expanded, it will look as shown below.

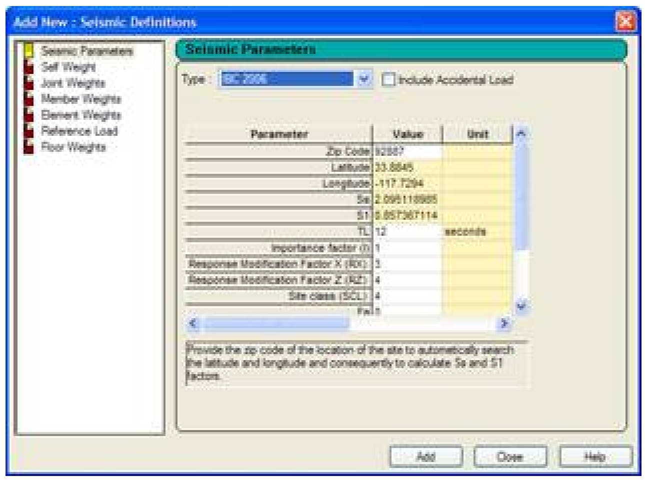


Select Seismic Definitions and click on **Add.**

In the dialog box that comes up, select IBC 2006 from the drop-down list.



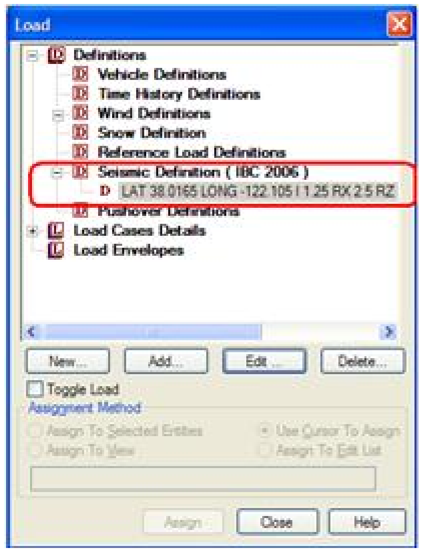
In this dialog box, we can specify the various parameters as described below.



Include Accidental Torsion	Check this box to calculate the accidental torsion component described in section 12.8.4.2 of ASCE 7-05.
Parameter	The various parameters for the IBC 2006 code, such as the Occupancy Importance factor IE, Response modification factors RX and RZ, spectral response accelerations SDS, SD1 and S1, etc., are described in detail in TR.31.2.14 IBC 2006/2009 Seismic Load Definition (on page 2405).

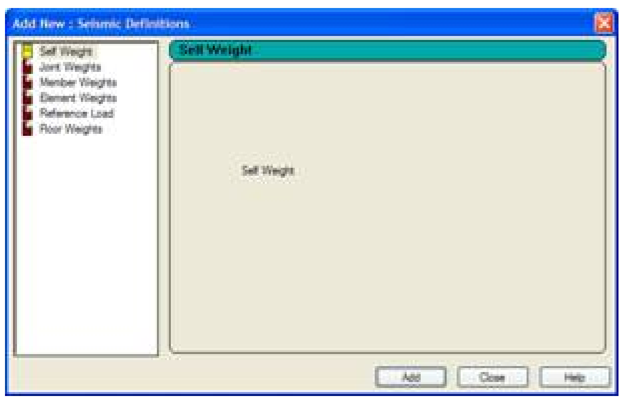
After specifying the values for the parameters, click on the **Add** button.

We will see that the Load dialog box has now been updated.



Next, we should define the structural weights for calculating Base Shear.

After highlighting the expression **LAT 38.0165……**, click on **Add**. A new dialog box titled **Add New Seismic Definitions** will come up.



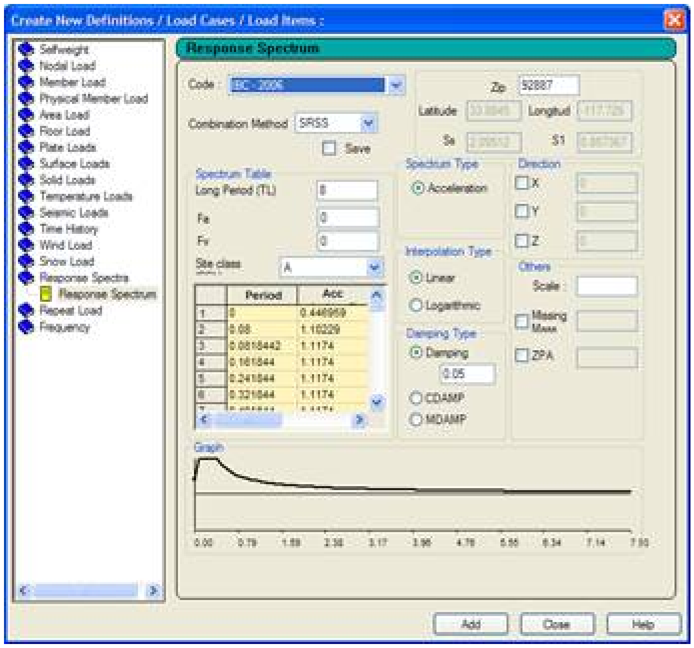
Related Links

- *M. To add a seismic load definition* (on page 857)

AD.2007-1001.1.15 IBC 2006 Response Spectrum Specification GUI

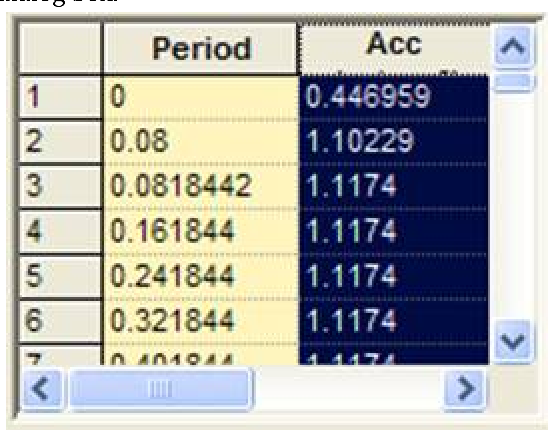
This tab allows the user to apply response spectrum loads on the structure.

This option can also be accessed from the **General | Load** page.



Period vs. Acceleration

Provide the values of period (seconds) and corresponding acceleration (current length units/sec^2) or displacement (current length unit). Spectrum pairs should be provided in ascending value of period. As we provide the curve points, the program displays the curve at the bottom of the dialog box.



Related Links

- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)
- *M. To add an IBC 2006 response spectrum* (on page 866)

AD.2007-1001.2 Features Affecting the Analysis and Design Engine

The following section describes the new features that have been added to the analysis and design engine and existing features that have been updated or modified.

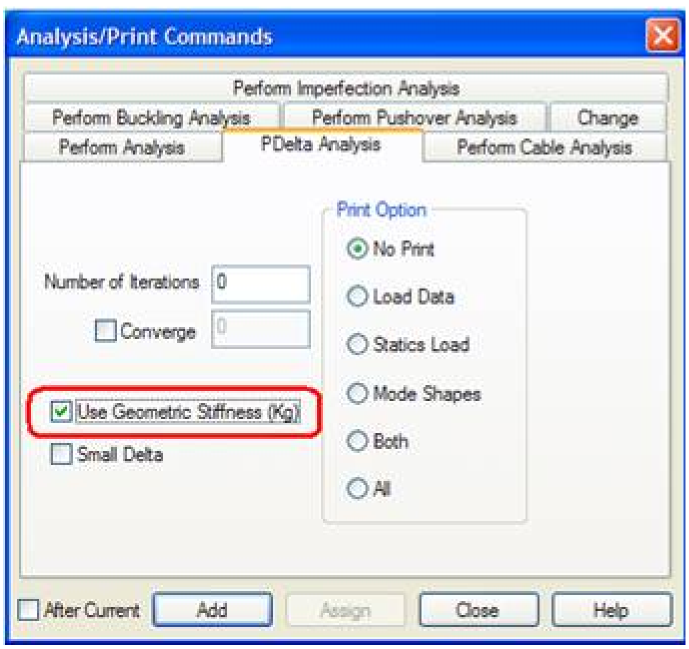
AD.2007-1001.2.1 P-Delta Analysis Including Stiffening Effect of the KG Matrix

The P-Delta analysis capability has been enhanced with the option of including the stress stiffening effect of the Kg matrix into the member / plate stiffness.

A regular STAAD P-Delta Analysis performs a first order linear analysis and obtains a set of joint forces from member/plates based on the large P-Delta effect. These forces are added to the original load vector. A second analysis is then performed on this updated load vector (5 to 10 iterations will usually be sufficient).

In the new P-Delta KG Analysis, that is, with the Kg option selected, the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included (1 or 2 iterations will usually be sufficient).

The KG option is activated by selecting the option on the P-Delta Analysis dialog thus:



Related Links

- *A. To specify a P-Delta analysis* (on page 952)

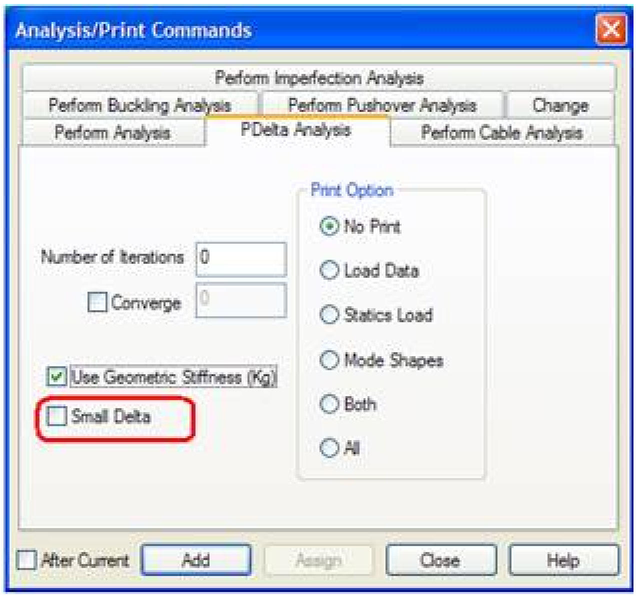
AD.2007-1001.2.2 P-Delta Analysis Including Small Delta

A regular STAAD P-Delta Analysis can now account for the small P-Delta effect whilst performing a P-Delta analysis

Without the Small Delta option, i.e. a regular STAAD P-Delta analysis, STAAD performs a first order linear analysis and obtains a set of joint forces, from members/plates based on the large P-Delta effect, which are then added to the original load vector. A second analysis is then performed on this updated load vector.

With the Small Delta option selected, both the large & small P-Delta effects are included in calculating the end forces, (5 to 10 iterations will usually be sufficient).

The option is activated by selecting the option on the P-Delta Analysis dialog thus:



Related Links

- *A. To specify a P-Delta analysis* (on page 952)

AD.2007-1001.2.3 Buckling Load Analysis

STAAD.Pro can now identify the factor by which the loads in the selected load case should be increased (or decreased if less than 1) such that Euler buckling would occur.

Two methods have been introduced to do buckling analysis. One method is introduced in the standard solver, the other in the advanced solver, as described below.

Related Links

- *A. To specify buckling analysis* (on page 957)

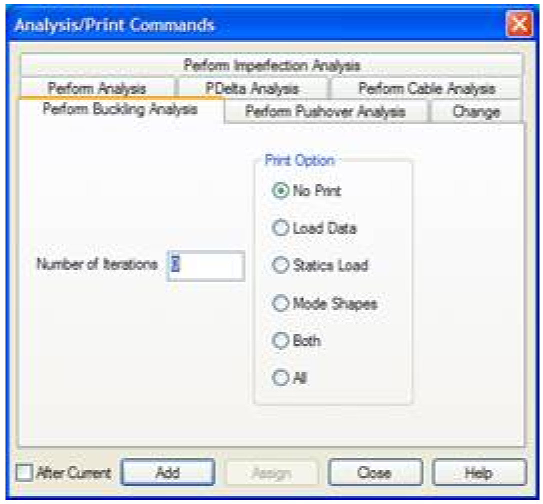
AD.2007-1001.2.3.1 Buckling Analysis Using the Basic Solver

Note: The Basic solver is deprecated as of STAAD.Pro CONNECT Edition V22 Update 4 (v22.04).

AD.2007-1001.2.3.2 Buckling Analysis Using the Advanced Solver

This buckling method is automatically activated if an Advanced Analysis license is available. When using the Advanced Solver, the corresponding “buckling modes” are included in the output file.

The option is activated using the new option in the **Analysis/Print** dialog thus:

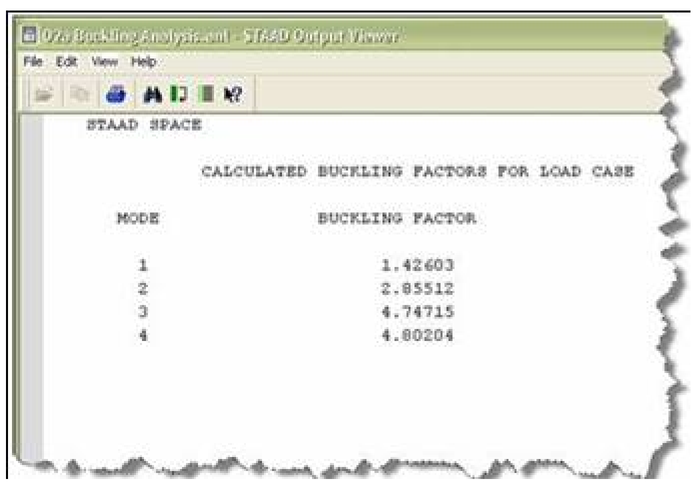


The program performs a P-Delta analysis including Kg Stiffening (geometric stiffness of members and plates) due to large and small P-Delta effects.

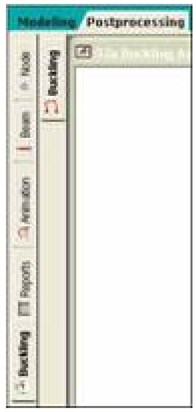
The eigensolution,

$$| [K] - BF [Kg] | = 0$$

is solved for the buckling factors and buckled mode shapes. The first 4 buckling factors and buckled shapes are calculated and included in the output file:



The buckling modes and shapes are available to be viewed in the Post Processing Mode in a new Buckling Page.



This page includes both a Buckling Factors table:

02a Buckling Analysis.std...

Mode	Buckling Factor
1	1.426
2	2.855
3	4.747
4	4.802

and a Buckling Modes table:

02a Buckling Analysis.std - Buckling Modes:

Node	Mode	X	Y	Z	rX	rY	rZ
1	1	0.000	0.000	0.000	0.000	0.000	0.000
	2	0.000	0.000	0.000	0.000	0.000	0.000
	3	0.000	0.000	0.000	0.000	0.000	0.000
	4	0.000	0.000	0.000	0.000	0.000	0.000
2	1	0.006	0.001	0.964	0.000	-0.000	-0.000
	2	0.001	0.001	-0.002	0.000	0.000	-0.000
	3	0.000	0.000	-0.000	0.000	-0.000	-0.000
	4	0.004	0.000	-0.008	0.000	-0.000	-0.000
3	1	0.006	0.000	0.985	0.000	-0.000	-0.000
	2	0.001	0.001	-0.009	0.000	0.000	-0.000
	3	0.000	-0.000	0.001	0.000	-0.000	-0.000
	4	0.004	-0.002	-0.002	-0.000	-0.000	-0.000
4	1	0.000	0.000	0.000	0.000	0.000	0.000
	2	0.000	0.000	0.000	0.000	0.000	0.000
	3	0.000	0.000	0.000	0.000	0.000	0.000

Only the primary load case just prior to the `PERFORM BUCKLING` command is used. The number of iterations entered is ignored. The buckling factor result is reported in the output file and in post processing.

AD.2007-1001.2.4 Modal Analysis Including Stress Stiffening Effect of KG Matrix

STAAD.Pro can include the stress stiffening effect (geometric stiffness) based on the axial member forces/plate in-plane stresses from a selected load case when calculating the modes & frequencies of a structure.

Position the selected load case from which the axial stresses are to be used to modify the stiffness matrix, such that it is the last static case before the dynamic case which is in turn immediately followed by a `PDELTA KG` command.

The dynamic load case should contain mass data followed by one of the following:

a. A `MODAL CALCULATION REQUESTED` command.
b. A response spectrum definition, i.e., set of `SPECTRUM` command data.
c. A reference to a time history definition, i.e., include `TIME LOAD` commands.
d. Valid Steady State data.

Example

```
...
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
******* This is static loading case from which the “axial stress” is
******* used to compute the stress stiffening effects (P-Delta)
******* This case will be solved as a PDelta case with large & small
******* P-Delta effects
SELFWEIGHT Y -1.0
JOINT LOAD
2 3 6 7 9 TO 12 FY -3
LOAD 2 LOADTYPE None TITLE LOAD CASE 2
******* Enter masses in weight units
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
2 3 6 7 9 TO 12 FX 10 FY 10 FZ 10
******* Declare this to be a modes/freq analysis
******* Note that dynamic cases use the factored matrix from the last *******
load case; which is a (K+Kg) case
MODAL CALCULATION REQUESTED
PDELTA KG ANALYSIS
```

AD.2007-1001.2.5 Enhanced Control/Dependent Command Processing

The internal processing of any Control/Dependent command has been enhanced to allow an automatic bandwidth reduction to take place.

The analysis engine now performs a bandwidth reduction on files that include Control/Dependent commands which must occur in the input file after the definition of supports. In previous versions of STAAD.Pro, for the bandwidth reduction to take place, the data of Control/Dependent would need to be repeated before the support definitions. This requirement is now no longer required.

Note: Control / Dependent was formerly referred to as master / slave.

Related Links

- *TR.28.1 Control/Dependent Specification* (on page 2327)

AD.2007-1001.2.6 Advanced Solver

A new substantially faster analysis engine has been produced which can provide solutions of large structures in a fraction of the time currently required by the standard STAAD engine. The Advanced Solver generally uses less disk and memory as well.

The Advanced solver is a new addition to the STAAD Analysis Engine which can be used for solving both static and dynamic problems. It is part of the STAAD engine with no special command required to run it.

The engine can operate in two modes: *in-core* and *out-of-core*. The in-core solver will be used for models with under 20,000 joints and the out-of-core solver for models over 20,000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be overridden using the `SET STAR` command.

The full set of overrides for the advanced engine is:

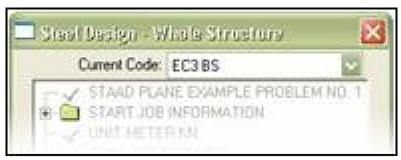
`SET STAR 3` – default
`SET STAR 4` – use out-of-core solver regardless of size

AD.2007-1001.2.7 Eurocode 3:2005

The latest UK release of *Eurocode 3: Design of steel structures – part 1-1: General Rules and rules for buildings* has been implemented.

This code is an update to our current Eurocode 3. The changes are minor and the code checking process remains the same as the previous Eurocode 3 design.

The design parameters are the same and apply as defined for the current Eurocode 3, the only exception, however, is the 'Code' parameter which must be set to 'EC3 BS'. This setting is available form the GUI list of codes.



Note that a design performed to the new Eurocode 3 standard is displayed in the output file (`*.ANL`) with the following header:

```
                       STAAD.PRO CODE CHECKING - (BS EN 1993-1-1:2005)
  **************************

 PROGRAM CODE REVISION V1.1 BS_EC3_2005/1
```

The equivalent header for a code check (or member selection) to the older standard is displayed thus:

```
              STAAD.PRO CODE CHECKING - (DD ENV)
  ***********************

  PROGRAM CODE REVISION V1.14_EC3_94/1
```

AD.2007-1001.2.8 IBC 2006 Equivalent Lateral Force Procedure – Syntax for the Command File

In AD.2007-1001.1.14 IBC 2006 Equivelant Lateral Force Procedure (on page 294), the graphical screens containing the implementation of the seismic loading chapters of the IBC 2006 and ASCE 7-05 codes were presented.

Related Links

- *M. To add a seismic load definition* (on page 857)

AD.2007-1001.2.9 Response Spectrum Specification in Accordance with IBC 2006 - Syntax for the Command File

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2006 edition of the ICC specification “International Building Code (IBC),” for dynamic analysis. The graph of frequency–acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

Related Links

- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)
- *M. To add an IBC 2006 response spectrum* (on page 866)

AD.2007-1001.3 Features Affecting the Post-Processor (Results Mode)

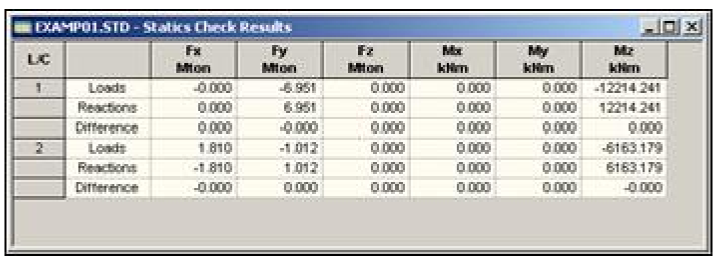
A new feature has been added to the post-processor section of the program, also known as the Results Mode. It is explained in the following pages.

AD.2007-1001.3.1 Statics Check Table

The equilibrium check that has been available in the output file is now automatically included in the Post Processing Mode.

The **Node | Reactions** Page has been enhanced with a new table that displays the results of an equilibrium check. This is the same information that in the past would have been only available by including a `PRINT STATICS CHECK` in the perform analysis command.

The **Statics Check Results** table can be accessed from the **Nodes | Reactions** page in the Post-Processing mode.



EXAMP01.STD - Statics Check Results

L/C		Fx Mton	Fy Mton	Fz Mton	Mx kNm	My kNm	Mz kNm
1	Loads	-0.000	-6.951	0.000	0.000	0.000	-12214.241
	Reactions	0.000	6.951	0.000	0.000	0.000	12214.241
	Difference	0.000	-0.000	0.000	0.000	0.000	0.000
2	Loads	1.810	-1.012	0.000	0.000	0.000	-6163.179
	Reactions	-1.810	1.012	0.000	0.000	0.000	6163.179
	Difference	-0.000	0.000	0.000	0.000	0.000	-0.000

AD.2007-1001.4 Features Affecting the Concrete Design Mode

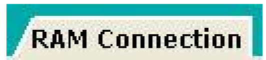
The enhancement made in the RC Designer section of the program, also known as the Concrete Mode, is explained in the following pages.

AD.2007-1001.4.1 Beam and Column Designs to the Russian Concrete Code SP52

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2007-1001.5 Features Affecting the Connection Design Mode

A new mode has been added that allows quick design of connections using the RAM Connection application. This is displayed on the Modes toolbar with the above icon.



Full use of the RAM Connection Mode requires access to a valid RAM Connection license. If you do not possess a license, contact your Bentley account manager to have it added to your SELECT licenses. Without a valid license, only a small subset of the full range of available RAM connections can be utilized.

AD.2007-1001.5.1 RAM Connection Design Mode

There is now a new mode in STAAD.Pro to dynamically link structural model data, including section properties and analysis results, to the RAM Connection application to check connection designs for code compliance. The resulting data and diagrams of the connection can also be included in the User Report.

Connections are designed in the RAM Connection Mode by creating 'Joints', from the geometry, section properties and forces resulting from the analysis and assigning a 'Design Brief' of connection templates from which a suitable connection, where available, is reported.

Refer to the D. Connection Design workflow (on page 1014) for current details on this feature.

AD.2007-1001.6 Features Affecting the Piping Mode

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Piping Mode. These are explained in the following pages.

AD.2007-1001.6.1 Persistency of Pipe Models

When a pipe model is loaded in the Piping Mode, the location of the data is retained along with the connections defined in the Supports Page. Thus, if STAAD.Pro is closed and re-opened, it is no longer necessary to redefine the pipe model or the support relationships.

This feature is an enhancement of the functionality of the Piping Mode.

STAAD.Pro 2006 Release Reports

The Software Release Report for STAAD.Pro 2006 contains detailed information on additions and changes that have been implemented since the final build of STAAD.Pro 2005. This document should be read in conjunction with all other STAAD.Pro manuals.

STAAD.Pro 2006 Build 1004 Release Report

The Software Release Report for STAAD.Pro 2006 Build 1004 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1002. This document should be read in conjunction with all other STAAD.Pro help.

AD.2006-1004.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modelling Mode. These are explained in the following pages.

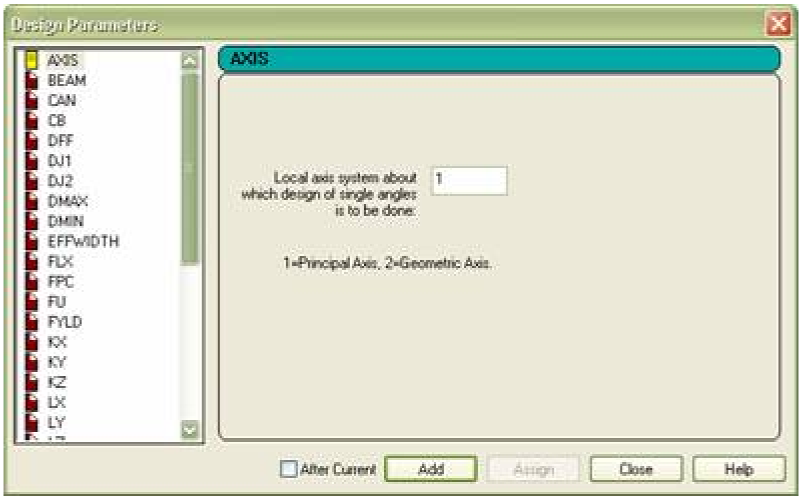
AD.2006-1004.1.1 New AISC Unified Code

A new steel design code module has been added to STAAD.Pro, which allows steel beam sections to be designed to the rules defined in AISC specification, *Specification for Steel Buildings*, March 9 2005, ANSI/AISC 360-05. This is also referred to as the AISC Unified Code. The GUI has been updated to allow you to enter the design information for this new design code.

To perform a design of steel sections to the AISC Unified Code, a design parameter block containing the command `CODE AISC UNIFIED` followed by the command to perform a `CODE CHECK` or `MEMBER SELECTION` must be added to the STAAD.Pro data file.

The GUI has been updated by adding **AISC UNIFIED** in the Steel Design dialog:

When this design code option is selected, clicking on the **Define Parameters** button at the base of the dialog box will in turn open the Design Parameters dialog that will allow parameters to be added or added and assigned to specific steel members.



Note: Design to this module is only possible if your security allows design to US design codes.

Related Links

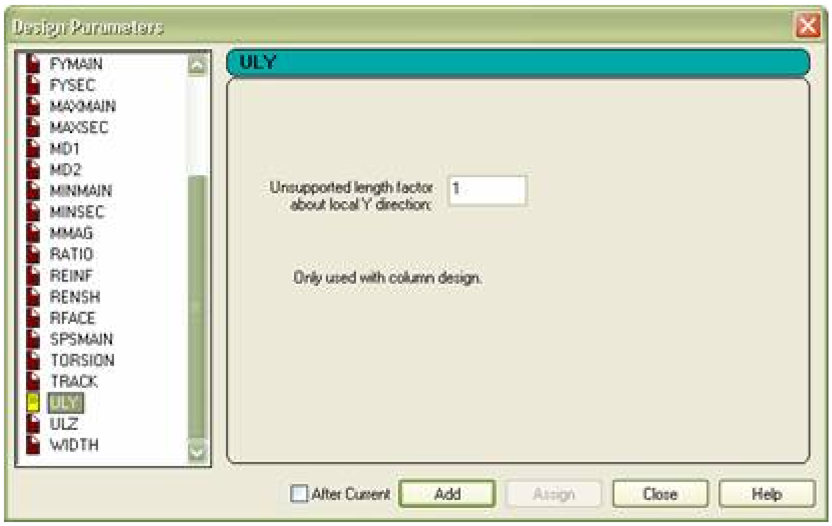
- *D1.A. American Codes - Steel Design per AISC 360 Unified Specification* (on page 1086)

AD.2006-1004.1.2 Update Indian IS 456 Concrete Code

Two new parameters have been added to the IS 456 design code to assist in the design of concrete columns. ULY and ULZ are used to define the ratio of unsupported to actual column length about the local Y and Z axes.

To include the design parameters for a IS456 design check, click on the Design, Concrete Page, and select IS456 from the drop-down list in the concrete design dialog.

When this design code option is selected, clicking on the **Define Parameters** button at the base of the dialog box will in turn open the Design Parameters dialog that will allow parameters to be added or added and assigned to specific steel members.



The definition of the parameters and their default values are as defined below:

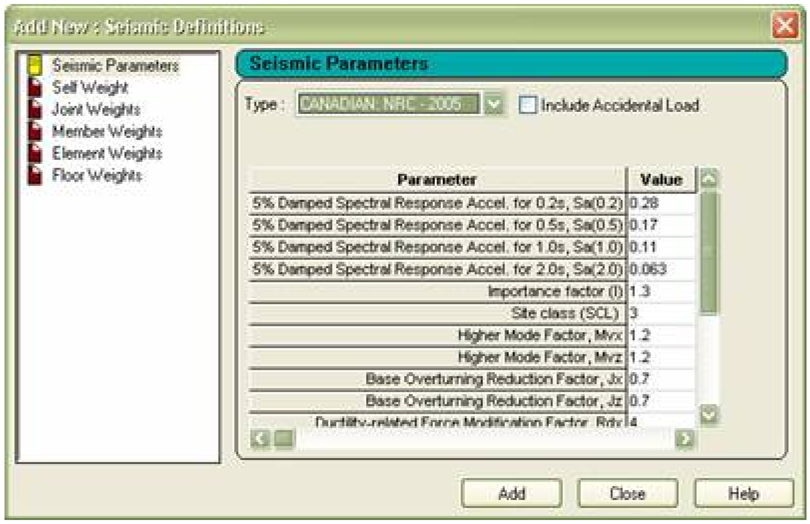
Parameter	Default Value	Description
ULY	1.0	Ratio of unsupported length to actual length of column about minor axis.
ULZ	1.0	Ratio of unsupported length to actual length of column about major axis.

AD.2006-1004.1.3 New NRC 2005 Seismic Code

A new seismic definition module has been added to the STAAD.Pro Loading Definitions.
This is to provide the user a tool to calculate and distribute the equivalent static loads produced by an earthquake that should be considered in a building as defined in the 'National Building Code of Canada 2005 – Volume 1', Section 4.1.8. This is also known as the NRC 2005 Seismic provisions.

This option has been added to be in addition to the previously available 1995 version of the code.

The NRC 2005 seismic provisions can be added using the **Seismic Definitions** dialog from the Definitions option in the main Load dialog box and selecting **CANADIAN NRC – 2005** from the type pull-down list.



Once all the required values for the parameters have been defined, then clicking on the **Add** button will add the data to the model such that it is ready to be used in seismic load cases.

For full details of the command structure for the NRC 2005 parameters see TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1 (on page 2358).

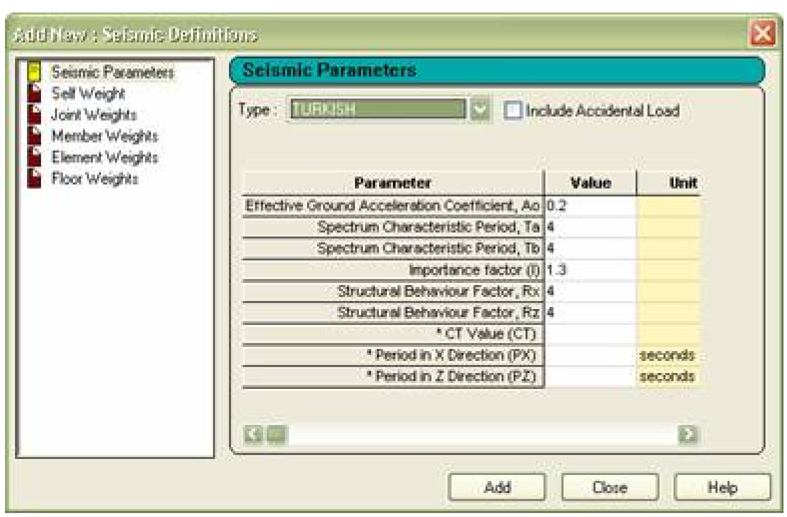
Related Links

- *TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1* (on page 2358)

AD.2006-1004.1.4 New Turkish Seismic Code

A new seismic definition module has been added to the STAAD.Pro Loading Definitions. This is to provide the user a tool to calculate and distribute the equivalent static loads produced by an earthquake that should be considered in a building as defined in the '*Specifications for Structures to be Built in Disaster Areas. Part III – Earthquake Disaster Prevention. Amended on 2.7.1998, Official Gazette No. 23390 (English Translation)*'. This is referred to as the Turkish Seismic Provisions.

The Turkish seismic provisions can be added using the Seismic Definitions dialog from the Definitions option in the main Load dialog box and selecting **TURKISH** from the type pull-down list.



Once all the required values for the parameters have been defined, then clicking on the **Add** button will add the data to the model such that it is ready to be used in seismic load cases.

For full details of the Turkish seismic load command structure see TR.31.2.21 Turkish Seismic Code (on page 2426).

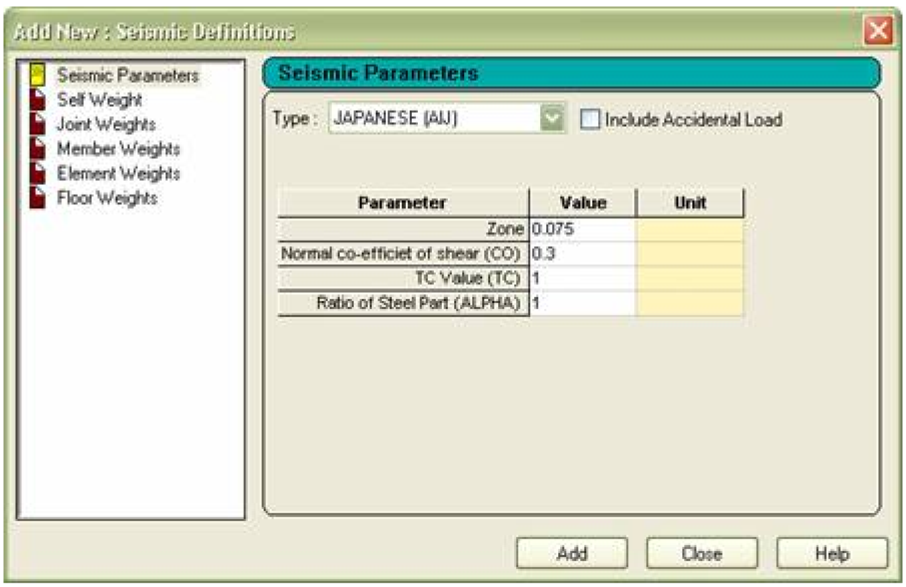
Related Links

- *TR.31.2.21 Turkish Seismic Code* (on page 2426)

AD.2006-1004.1.5 Update Japanese Seismic Definition

The current version of the Japanese Seismic Loading Definition has been enhanced to allow the user to specify a value of 'α (alpha)' which is the ratio of frame height to the overall building height which previously been set to 0.8 and not possible for the user to change.

The Japanese seismic provisions can be added using the Seismic Definitions dialog from the Definitions option in the main Load dialog box and selecting **Japanese (AIJ)** from the Type pull-down list.



Once all the required values for the parameters have been defined, then clicking on the **Add** button will add the data to the model such that it is ready to be used in seismic load cases.

Note: The implementation of the Japanese Seismic Definition is as per Article 88 in the 'Building Codes Enforcement Ordinance 2006.'

For full details of the Japanese seismic load command structure see TR.31.2.18 Japanese Seismic Load (on page 2418).

Related Links

- *TR.31.2.18 Japanese Seismic Load* (on page 2418)

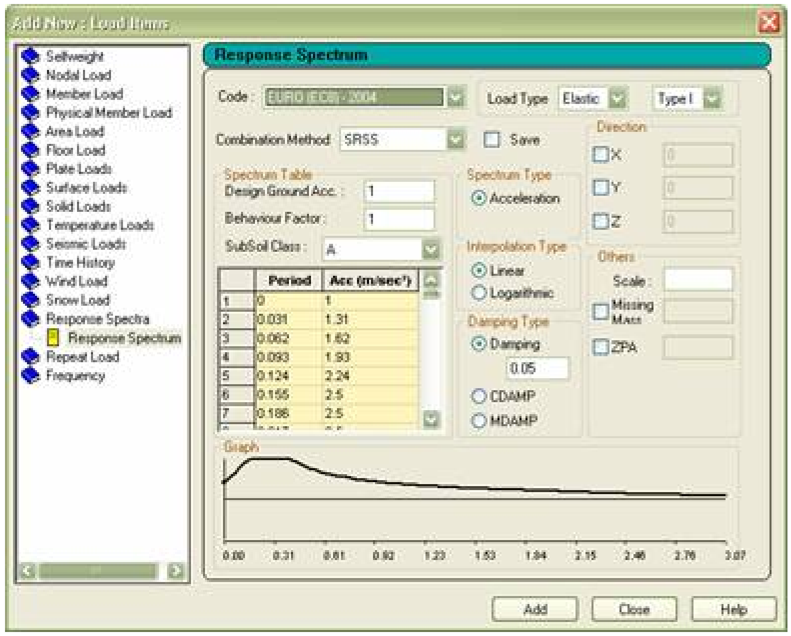
AD.2006-1004.1.6 New Eurocode 8 Response Spectrum

A new response spectrum specification has been added to STAAD.Pro, 'Eurocode 8. Design of structures for earthquake resistance'. General rules, seismic actions and rules for buildings', BS EN 1998-1:2004.

The provisions for Eurocode 8- 2004 have been added to the **Response Spectrum** dialog box.

Enter the General, Loads Page of the Modelling Mode, and select the load case in the **Load** dialog box. Select the required load case name from the Load Case Details group and click on the **Add** button.

In the dialog box that is displayed, select the **Response Spectra** option from the list of items on the left and select the code **Euro (EC8)-2004** to display the following:



Select the required parameters such as the Load Type and Combination Method and the direction that the spectrum is to be defined and click on the **Add** button to add the command to the input file.

Note: This is response spectrum definition has been provided in addition to the ENV 1998-1-1:1994 version of the code that was previously implemented. The older version of the code is still available, although recommended for comparison purposes only.

For a full description of both implemented versions of the Eurocode 8 Response Spectrum command structure, see TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2525).

Related Links

- *TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004* (on page 2525)
- *M. To add an EC8 response spectrum* (on page 867)

AD.2006-1004.1.7 Update IS 1893 Dialog to Support Command Structure

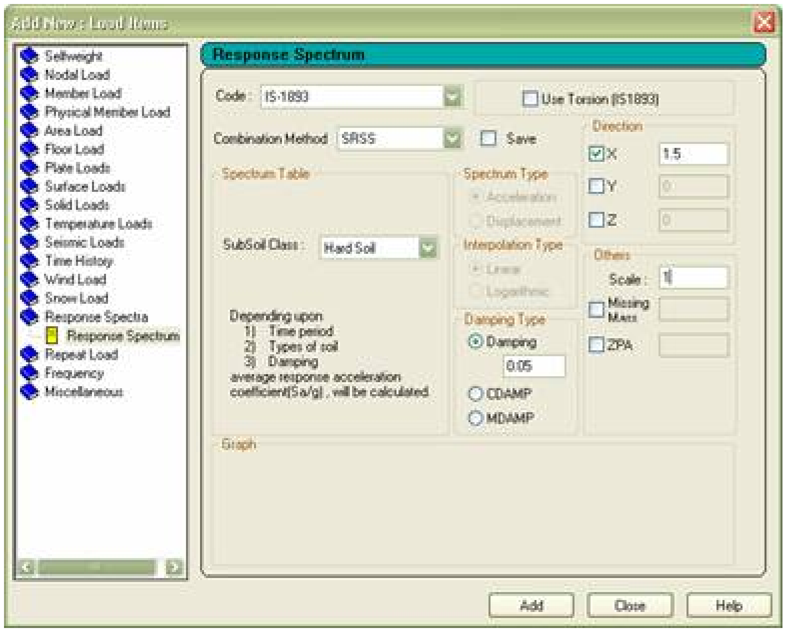
The **Response Spectrum** dialog for defining a Response Spectrum to be calculated as per the IS-1893 code by the analysis engine has been enhanced so that it reflects the input requirements of this command.

Enter the **General | Loads** Page of the Modelling Mode and select the load case in the **Load** dialog box. Select the required load case name from the Load Case Details group and click on the **Add** button.

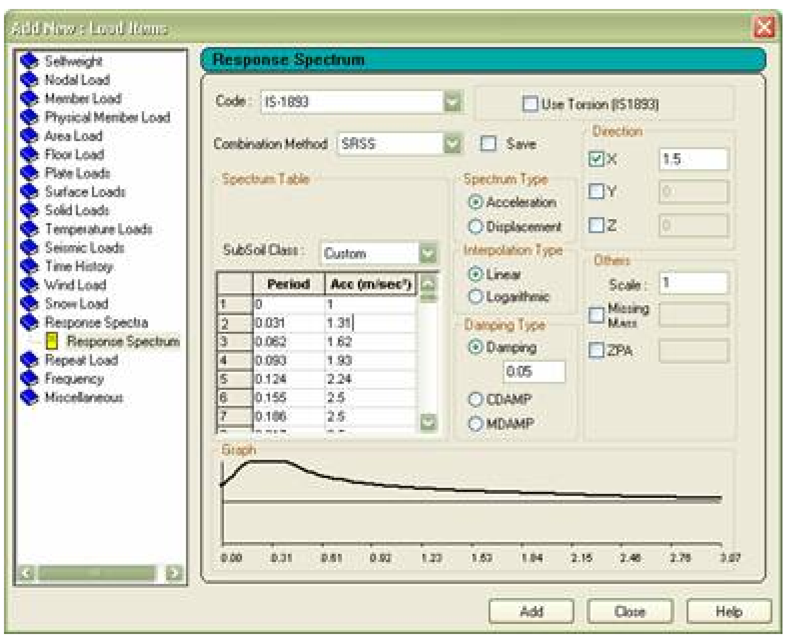
In the dialog box that is displayed, select the **Response Spectra** option from the list of items on the left and select the code **IS-1893**. There are essentially 2 methods that can be employed to define the response spectrum:

a. Select a SubSoil Class
b. Define a Custom Reponse Graph

This choice is made from the **SubSoil Class** in the Spectrum Table. If a defined class is selected, then the Response Spectrum Graph is automatically defined and specified during the analysis, however the following dialog is presented to allow other options to be set:



However, if a custom Response Spectrum Graph is required and selected from the SubSoil drop list, then the various points on the graph of period/acceleration or period/displacement must be entered, thus:



For a full description of the scope of the IS-1893 Response Spectrum command, refer to TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534).

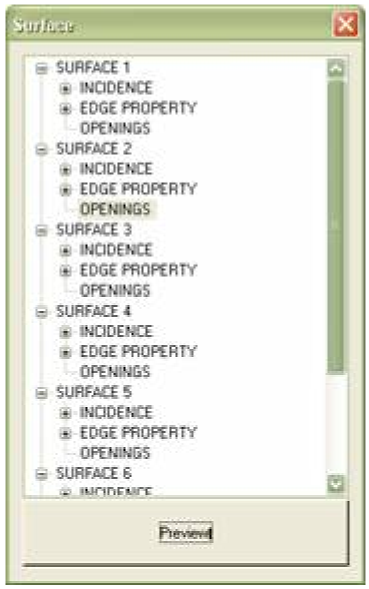
Related Links

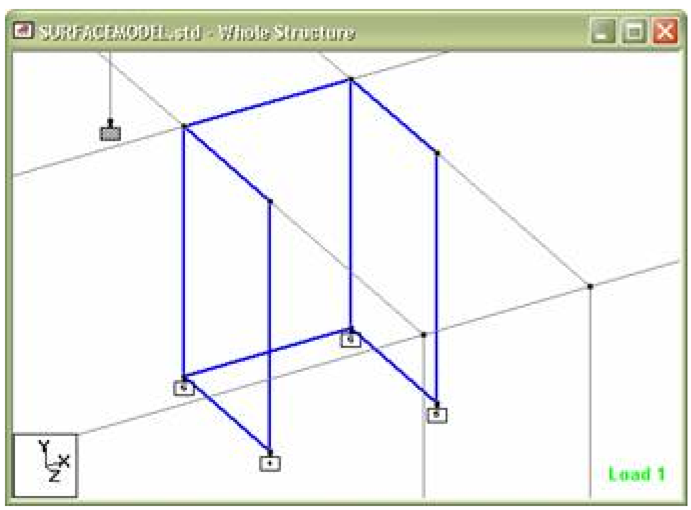
- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)
- *M. To add an IS 1893 response spectrum* (on page 864)

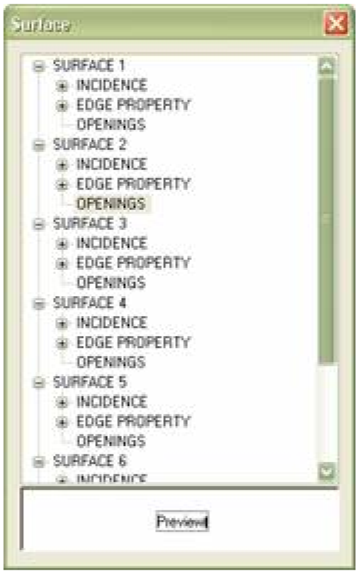
AD.2006-1004.1.8 Preview of Surface Element Meshes

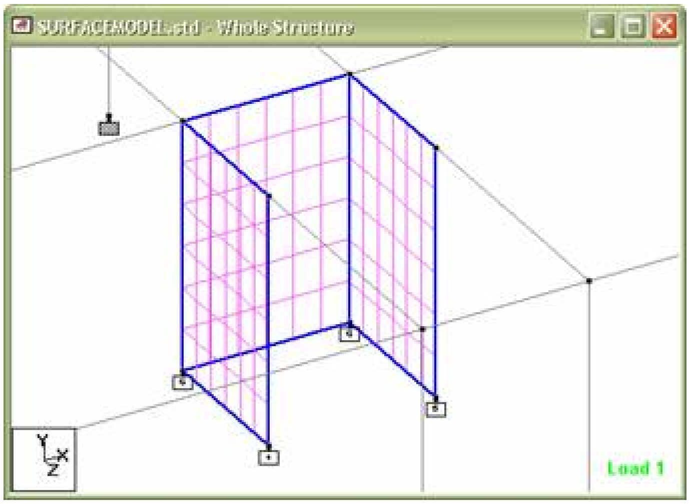
Models that contain large numbers of surfaces, or have complex geometrical requirements can result in extensive analyses which may be un-necessary if appropriate settings are applied to the surface. In order to the mesh that will be generated during the analysis, the option to 'Preview' the surface meshes has been added.

After creating one or more surface elements, go to the Geometry, Surface Page which will display the **Surface** dialog. By clicking on the **Preview** button at the base of the dialog, the mesh that will be generated by the defined parameters will be displayed thus:









AD.2006-1004.1.9 Key File Option Added to Configuration Settings

The output file has the option to list all the key files that were used with the analysis including the date of the principal files. However, this can produce unwanted extra printout data and thus an option to specify whether or not this is to be used has been set in the Configuration Settings.

To access the File Configuration Settings, start STAAD.Pro, but close any open file or the New Wizard if it is open. Click on the menu option, **File** > **Configure** to open the **Configuration Settings** dialog and click on the sheet **Input/Output File Format**, thus:

To include the key file information in the output file, set the option **Add key file information to output**.

If the option is set, the output file contains information similar to the following at the end of the file.

```
EXAMPLE PROBLEM NO. 1                                       -- PAGE NO.   18


    Information about the key files in the current distribution


    Modification Date   CRC     Size (Bytes)        File Name


------------------------------------------------------------------------------
-----------

    10/26/2006      0x3cc1      13266944        SProStaad.exe

    10/25/2006      0x1cc0      05836800        SProStaadStl.exe

    09/19/2003      0x2fc0      00081970        CMesh.dll

    10/25/2006      0x4601      02486272        dbSectionInterface.dll

    …
```

Note: If the general case is that the Key Information is displayed, it can be tuned off in a particular analysis if the data file contains the command `SET NOFILE`.

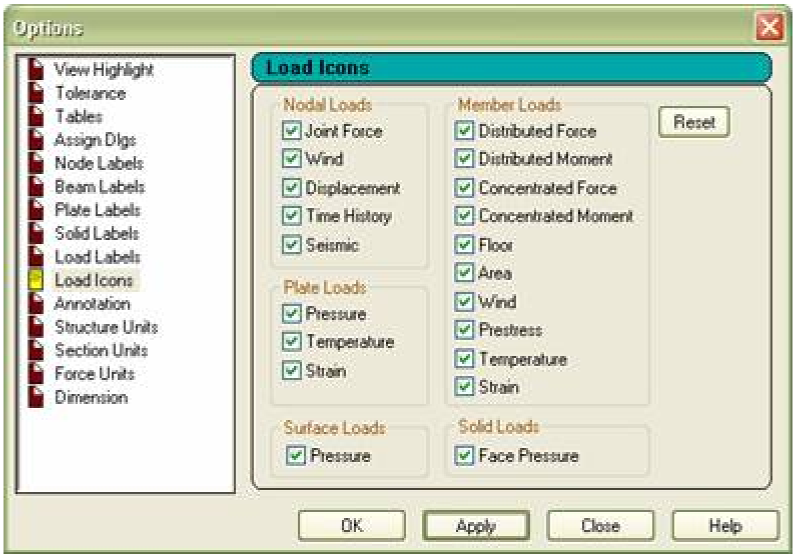
Related Links

- *GS. Application Configuration dialog* (on page 59)

AD.2006-1004.1.10 Load Icons Display Option

Certain models have large numbers of loads defined in any given load case. It is now possible to select which 'types' of load are displayed when the load case data is displayed.

To select which load items are displayed when the load is displayed, select the Options from the menu item **View** > **Options** and select **Load Icons** thus:



To return to the default of displaying all load types, click on the button **Reset**.

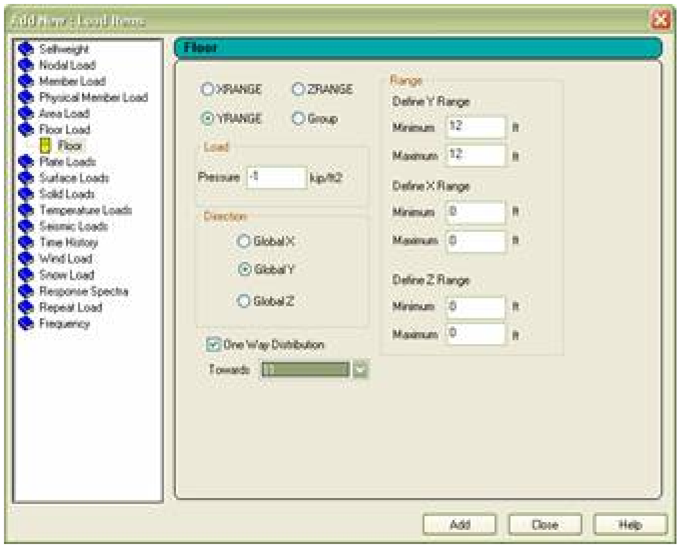
AD.2006-1004.1.11 Update Oneway Loading

The `ONEWAY` command has been enhanced with the option to define the direction in which the load is spanning by selecting a member in the loaded zone onto which the load is to be directed.

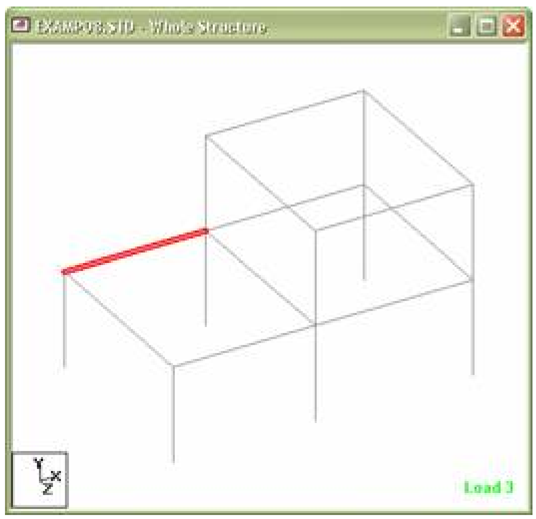
Enter the **General | Loading** Page and create or select the required load case.

Click on the **Add** button on the Load dialog box.

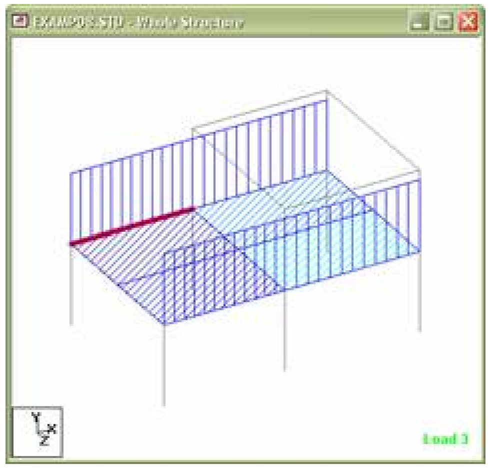
From the options on the left, select the **Floor Load** option and enter the Floor load in the normal way defining the direction, Load intensity and range. This defines a two way spanning system, to indicate a one-way span, click on the **One Way Distribution** option. This will then span in the shorter direction. However, now a beam number can be selected from the **Towards** drop list which is populate with the beams that exist in the defined range. As shown blow:



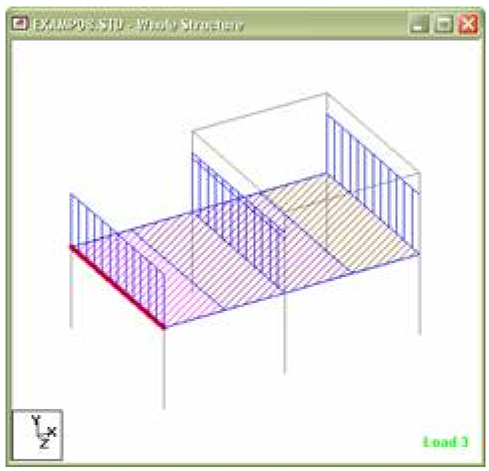
The selected beam is highlighted on the structure so that the spanning direction can be confirmed, thus:



Once the command is created, the load and contributing areas can be displayed by highlighting the command in the Load dialog box, thus:



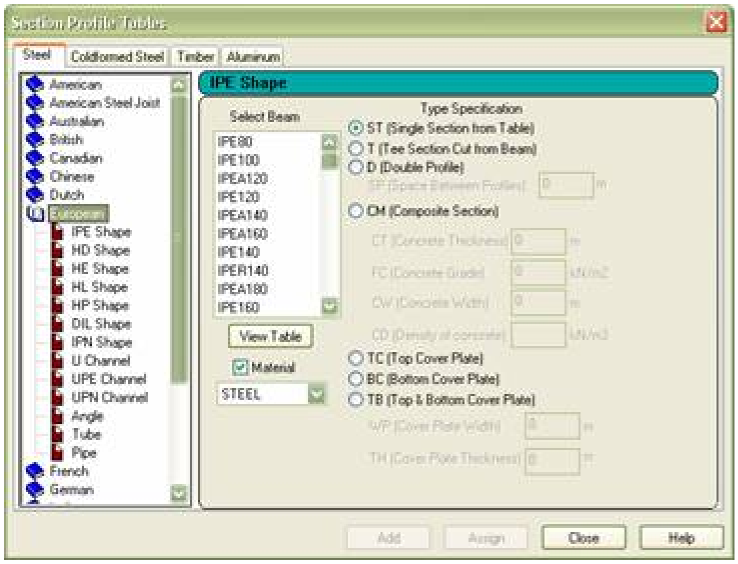
By changing the beam number that defines the direction that the load is towards, the change in the loading can be viewed, thus:



AD.2006-1004.1.12 Additional Standard European Steel Section Types

Additional steel section shapes produced by European Steel manufactures have been added to the standard steel database libraries. Section shapes, HD, HL, UPE have been added.

Click on the **General | Property** page and on the **Properties** dialog box, click on the **Section Database** button to view steel section databases available. Click on the **European** label to view the section types that are now available.



AD.2006-1004.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2006-1004.2.1 AISC Unified Code

A new steel design code module has been added to STAAD.Pro, which allows steel beam sections to be designed to the rules defined in AISC specification, *Specification for Steel Buildings*, March 9 2005, ANSI/AISC 360-05. This is also referred to as the AISC Unified Code. The GUI has been updated to allow you to enter the design information for this new design code.

Related Links

- *D1.A. American Codes - Steel Design per AISC 360 Unified Specification* (on page 1086)

AD.2006-1004.2.2 ACI 2005

The ACI concrete design module has been updated to support the current 2005 version of the ACI code, ACI 318-05.

When performing the concrete design using the parameter CODE ACI 2005, the design will be performed according to the requirements of the American Concrete Institute document, 'Building Code Requirements for Structural Concrete (ACI 318-05)'. If it is required to perform the design to the older versions of the code, specifically, the 1999 or 2002 version of the code, then the commands:

```
CODE ACI 1999
```

or

```
CODE ACI 2002
```

The commands `CODE ACI` or `CODE ACI 2005` will set the program ready to perform the design to ACI 2005 when it encounters the commands `DESIGN BEAM`..., `DESIGN COLUMN`... or `DESIGN ELEMENT`....

For more information on the refer to the updated [D1.F. American Codes - Concrete Design per ACI 318](#) (on page 1198).

AD.2006-1004.2.3 Enhancement of Indian Concrete Design

Two new parameters have been added to the Indian concrete design code, IS 456.

ULY and ULZ which are the unsupported lengths in the local Y and Z directions. These commands have been added to assist in the calculation of the minimum eccentricities.

Related Links

- *D8.E. Indian Codes - Concrete Design per IS 456* (on page 1669)

AD.2006-1004.3 Features Affecting the Post Procesing (Results) Mode

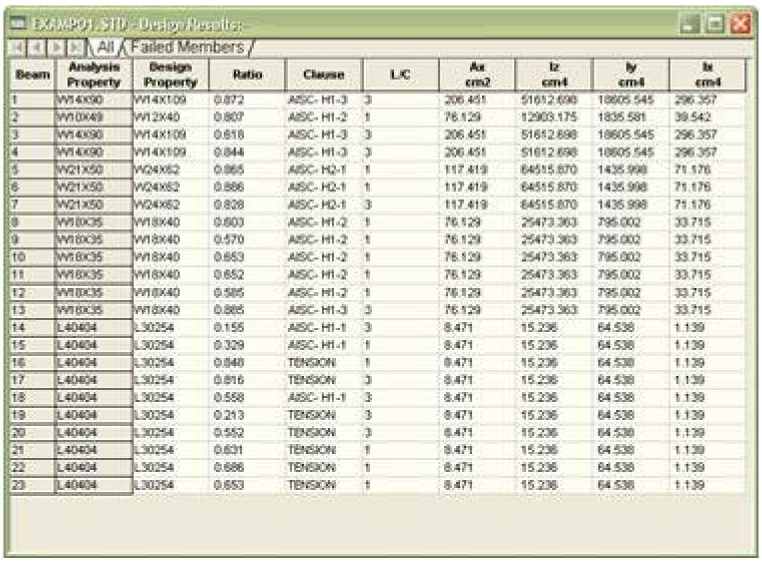
Several new features have been added and existing features have been modified in the Steel Designer section of the program, also known as the Steel Designer Mode. These are explained in the following pages.

AD.2006-1004.3.1 Unity Check Page Update

The Design Results table has been updated so that not only can the maximum utilisation ratio be displayed, but also the governing clause and combination. The superfluous section dimension information has been removed.

Additionally, the table can be sorted by design ratio.

Perform an analysis and design and enter the Post Processing Mode. To view the **Design Results** table, click on the **Beam | Unity Check** Page. The table is displayed on the right of the window:



Beam	Analysis Property	Design Property	Ratio	Clause	L/C	Ax cm2	Iz cm4	Iy cm4	Ix cm4
1	W14X90	W14X109	0.872	AISC- H1-3	3	206.451	51612.698	18605.545	296.357
2	W10X49	W12X40	0.807	AISC- H1-2	1	76.129	12903.175	1835.581	39.542
3	W14X90	W14X109	0.618	AISC- H1-3	3	206.451	51612.698	18605.545	296.357
4	W14X90	W14X109	0.844	AISC- H1-3	3	206.451	51612.698	18605.545	296.357
5	W21X50	W24X62	0.865	AISC- H2-1	1	117.419	64515.870	1435.998	71.176
6	W21X50	W24X62	0.886	AISC- H2-1	1	117.419	64515.870	1435.998	71.176
7	W21X50	W24X62	0.828	AISC- H2-1	3	117.419	64515.870	1435.998	71.176
8	W18X35	W18X40	0.603	AISC- H1-2	1	76.129	25473.363	795.002	33.715
9	W18X35	W18X40	0.570	AISC- H1-2	1	76.129	25473.363	795.002	33.715
10	W18X35	W18X40	0.653	AISC- H1-2	1	76.129	25473.363	795.002	33.715
11	W18X35	W18X40	0.652	AISC- H1-2	1	76.129	25473.363	795.002	33.715
12	W18X35	W18X40	0.585	AISC- H1-2	1	76.129	25473.363	795.002	33.715
13	W18X35	W18X40	0.885	AISC- H1-3	3	76.129	25473.363	795.002	33.715
14	L40404	L30254	0.155	AISC- H1-1	3	8.471	15.236	64.538	1.139
15	L40404	L30254	0.329	AISC- H1-1	1	8.471	15.236	64.538	1.139
16	L40404	L30254	0.848	TENSION	1	8.471	15.236	64.538	1.139
17	L40404	L30254	0.816	TENSION	3	8.471	15.236	64.538	1.139
18	L40404	L30254	0.558	AISC- H1-1	3	8.471	15.236	64.538	1.139
19	L40404	L30254	0.213	TENSION	3	8.471	15.236	64.538	1.139
20	L40404	L30254	0.552	TENSION	3	8.471	15.236	64.538	1.139
21	L40404	L30254	0.631	TENSION	1	8.471	15.236	64.538	1.139
22	L40404	L30254	0.686	TENSION	1	8.471	15.236	64.538	1.139
23	L40404	L30254	0.653	TENSION	1	8.471	15.236	64.538	1.139

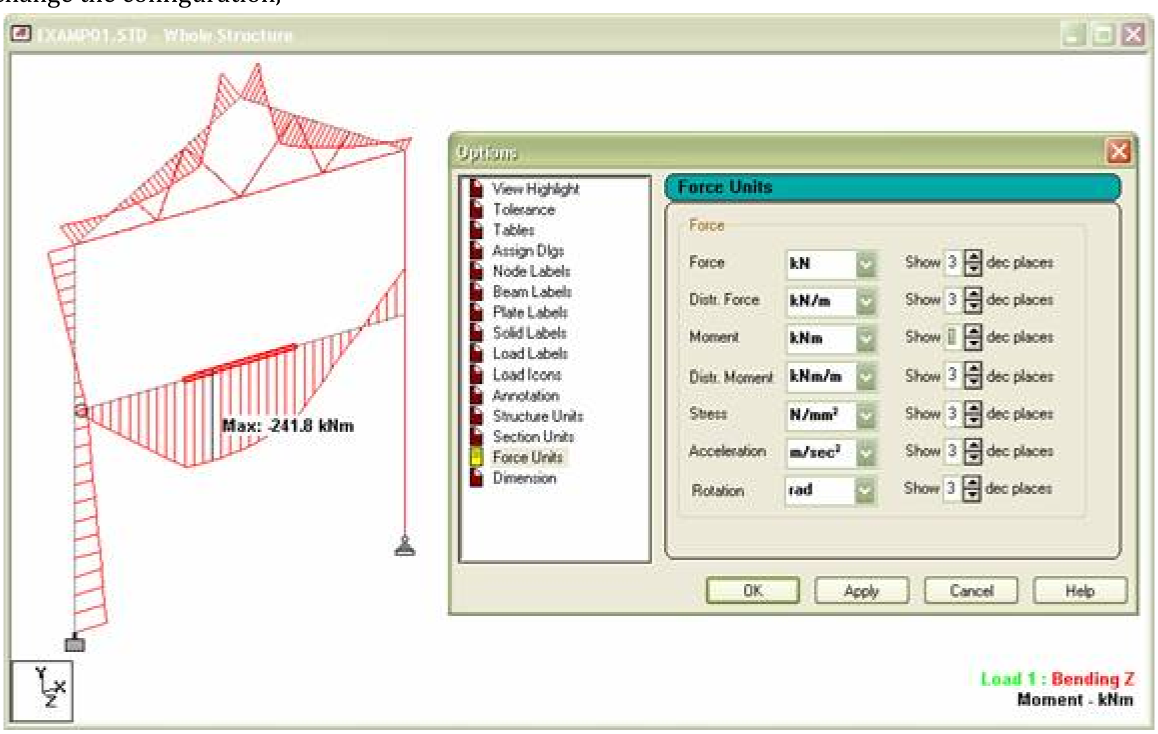
To sort the table by ratio, click on the **Ratio** column heading. To return to the un-sorted list, click on the **Beam** column heading.

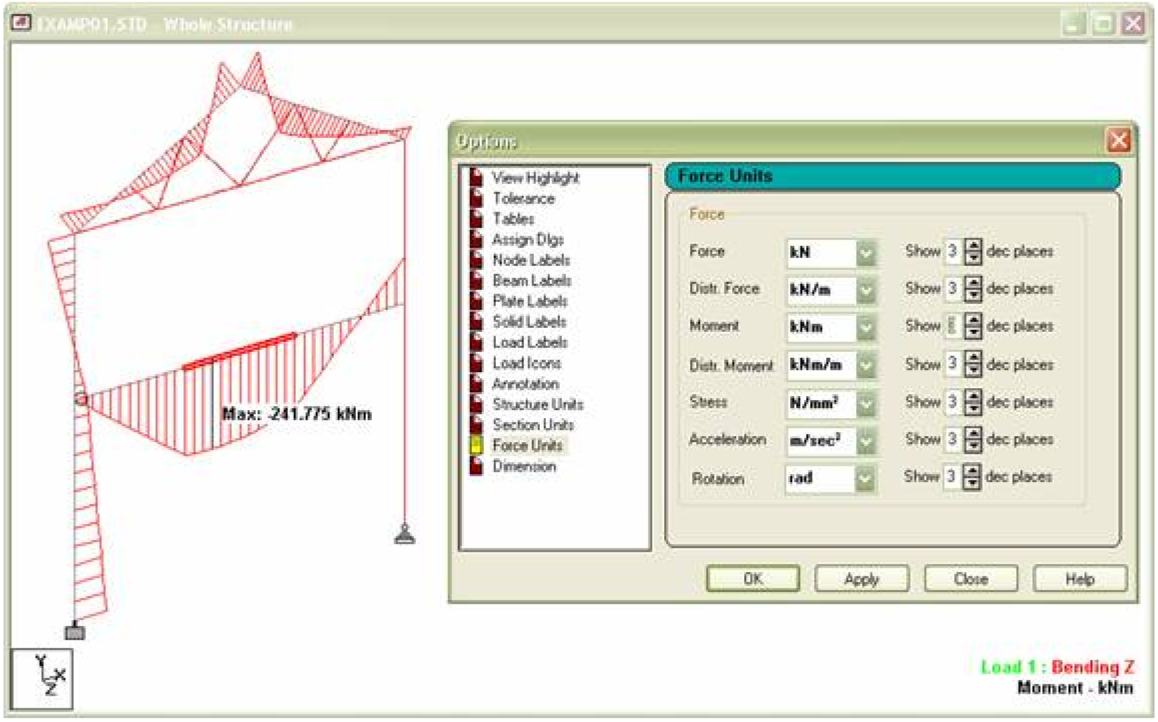
AD.2006-1004.3.2 View Value Annotation Enhanced

The annotation of forces and displacements has been enhanced to allow the number of decimal places viewed on the diagrams to be configured.

1. Open a model with analysis results or run an analysis to obtain a set of valid analysis results.
2. Enter the Post Processing mode and click on the **Beam | Forces** Page to display the bending moment on the graphic view of the structure.
3. Click on the menu item **Results** > **View Value...** to annotate the bending moment diagram and define the items that are required to be annotated in the **Ranges** sheet, the list of items to be annotated such as **Maximum Bending Moments**.

4. To change the configuration,





AD.2006-1004.4 Features Affecting the Concrete Design Mode

Several new features have been added and existing features have been modified in the RC Designer section of the program, also known as the Concrete Design Mode. These are explained in the following pages.

AD.2006-1004.4.1 Bentley Rebar Export from RC Designer

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006-1004.4.2 Addition of Amendments 1,2, and 3 to the BS 8110 Modules

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006-1004.5 Features Affecting the Piping Mode

STAAD.Pro can utilise the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.

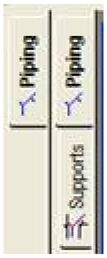
AD.2006-1004.5.1 Update of Importing Piping Models

The import stage of the Piping Mode has been enhanced by allowing the user to better relate the co-ordinate systems used by the two models, the structural model and the piping model. There are two issues that are addressed:

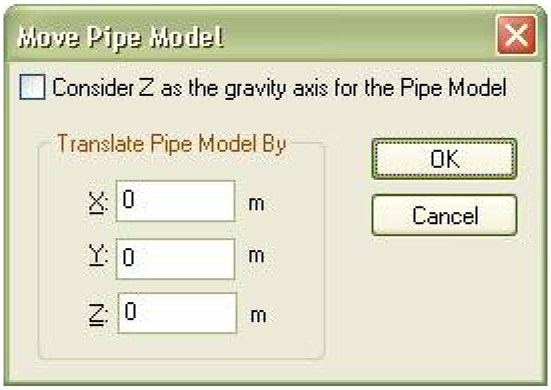
a. Locating the origin
b. Allowance for Y or Z axis as vertical in the piping model

Enter the Piping Mode by either selecting the menu option **Mode** > **Piping** or from the Mode Bar thus:

This displays the principal page and two sub pages available in this mode thus:



To select a pipe model to import, click on the **Open File…** button in the Information dialog box and navigate to the folder that has the required ADLPIPE or AutoPipe (`*.ADI`) model. Select the required file and click on the **Open** button. The following dialog is then displayed:



This allows the direction that was used as vertical in the pipe model system to be identified by clicking the option **Consider Z as the gravity axis for the Pipe Model** if required. Additionally, the X, Y, and Z values allows the origin of the pipe model to be placed at a different location than the origin of the STAAD.Pro structure.

Refer to the M. Piping workflow (on page 902) in the User Interface help

AD.2006-1004.5.2 Pipes Support by Member

Supports defined in the piping model may not have corresponding nodes defined in the STAAD.Pro model. Therefore the pipe support function has been enhanced to allow members to be selected so that the support load will be transferred as a member load rather than node load.

Enter the Piping Supports page. Select the menu item 'Support>Support Node'. Using the Pipe Support cursor, select the pipe support node then click on the supporting member.

Note: For a member to be valid as a pipe support beam, then a line from the pipe support node, that is perpendicular to the support beam, must lie between the start and end nodes of that beam.

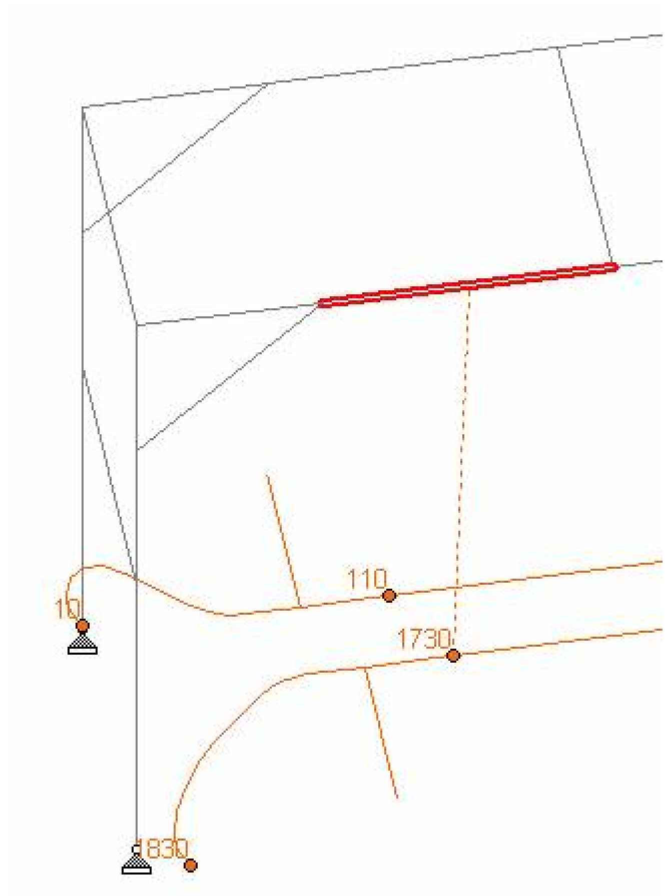


Figure 11: A pipe support node is supported by the highlighted beam

AD.2006-1004.6 Additional STAAD.Pro 2006 Build 1004 Features

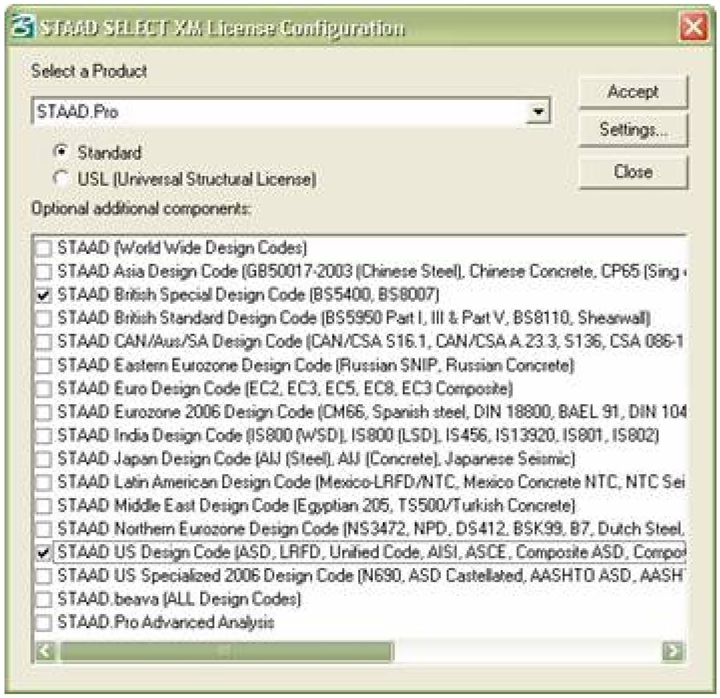
The following pages outline the additional features that have been added to STAAD.Pro in this version.

AD.2006-1004.6.1 Modifications of the SELECT XM Security System

With the SELECT security system, STAAD.Pro can be configured to use either a **STAAD.Pro** license or a **Universal Structural License** (USL), which can also be used to access other structural products in the Bentley portfolio. However, to perform a design, the USL needs to be configured along to specify which design code packs are to be accessed during the operation of STAAD.Pro. Additionally, once the configuration of the STAAD.Pro SELECT license has been established, it is now possible to review the current settings and update it as required.

To review and/or update the current STAAD SELECT security configuration, click on the Start menu item: **Start** > **Programs** > **STAAD.Pro 2006** > **Select License Tools** > **STAAD SELECT Configuration**

1. Select STAAD.Pro from the Product drop list.
2. Select whether a Standard or USL license should be obtained to run STAAD.Pro.
3. Select which code packs should also be obtained when STAAD.Pro is run.



Notes:

1. This configures what licenses are to be obtained. This does not confirm that those features are available. To confirm what features are available, click **Settings**. This launches the **License Management Tool** which displays the list of products and licences available on the currently defined SELECT XM server and those that are currently 'Checked Out' onto the local machine.
2. If a feature or design code requires a specific license during a session of STAAD.Pro, then it is important to check that the license for that feature is available using the License Management Tool and setting the STAAD SELECT XM License Configuration to obtain that license as STAAD.Pro starts.

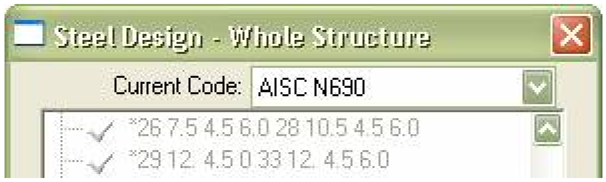
For a complete description of the SELECT security system, please refer to the *Quickstart & Troubleshooting Guide* (`STAAD.Pro_InstallGuide_en.pdf`) document distributed with the installation setup.

AD.2006-1004.6.2 AISC N690

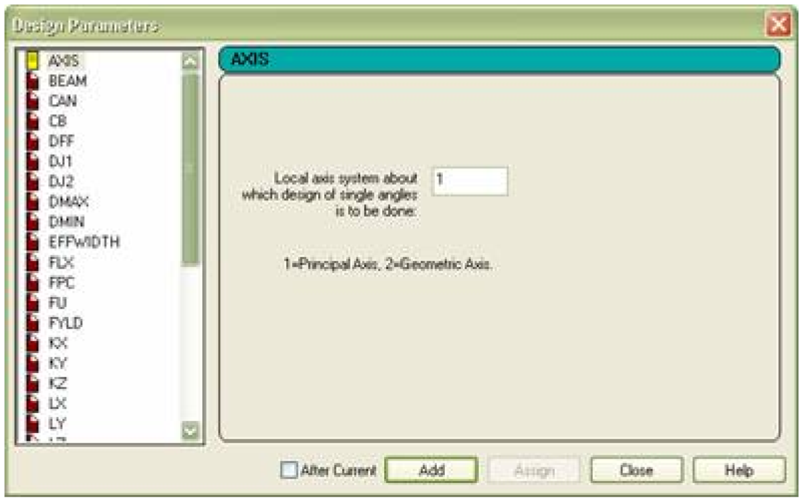
In addition to the standard ASD design method, STAAD.Pro can perform a design to incorporate the design requirements of *Supplement No. 1 to the Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities* ANSI/AISC N690 – 1994s1. This additional code has been added to the **Steel Design** dialog.

To perform a design of steel sections to the AISC N690, a design parameter block containing the command `CODE AISC N690` followed by the command to perform a `CODE CHECK` or `MEMBER SELECTION` must be added to the STAAD.Pro data file.

The GUI has been updated by adding **AISC N690** in the **Steel Design** dialog:



When this design code option is selected, clicking on the **Define Parameters...** button at the base of the dialog box will in turn open the **Design Parameters** dialog that will allow parameters to be added or added and assigned to specific steel members.



For a full explanation of the features of the design engine see D1.K. American Codes - Steel Design per ANSI/AISC N690 Design Codes (on page 1296).

Note: Design to this module is only possible if your security allows design to US Specialised design codes.

STAAD.Pro 2006 Build 1002 Release Report

The Software Release Report for STAAD.Pro 2006 Build 1002 contains detailed information on additions and changes that have been implemented since the release of STAAD.Pro 2006 build 1001. This document should be read in conjunction with all other STAAD.Pro help, including the Revision History document.

AD.2006-1002.1 Features Affecting the Pre-Processor (Modeling Mode)

Several new features have been added and existing features have been modified in the pre-processor section of the program, also known as the Modeling Mode. These are explained in the following pages.

AD.2006-1002.1.1 Automatic calculation of the Response Spectrum as per IBC

STAAD.Pro previously allowed response spectra to be defined by the user, or calculated as per the Indian IS 1893 or Eurocode EC8.
The graphical user interface (GUI) has been enhanced so that it can now automatically generate a response spectrum as per the guidelines of the IBC/ASCE code

(*). This new feature saves users the time and effort involved in establishing the 1 second and short period accelerations indicated in the IBC/ASCE maps and then converting them into spectral data based on the equations provided in the code.

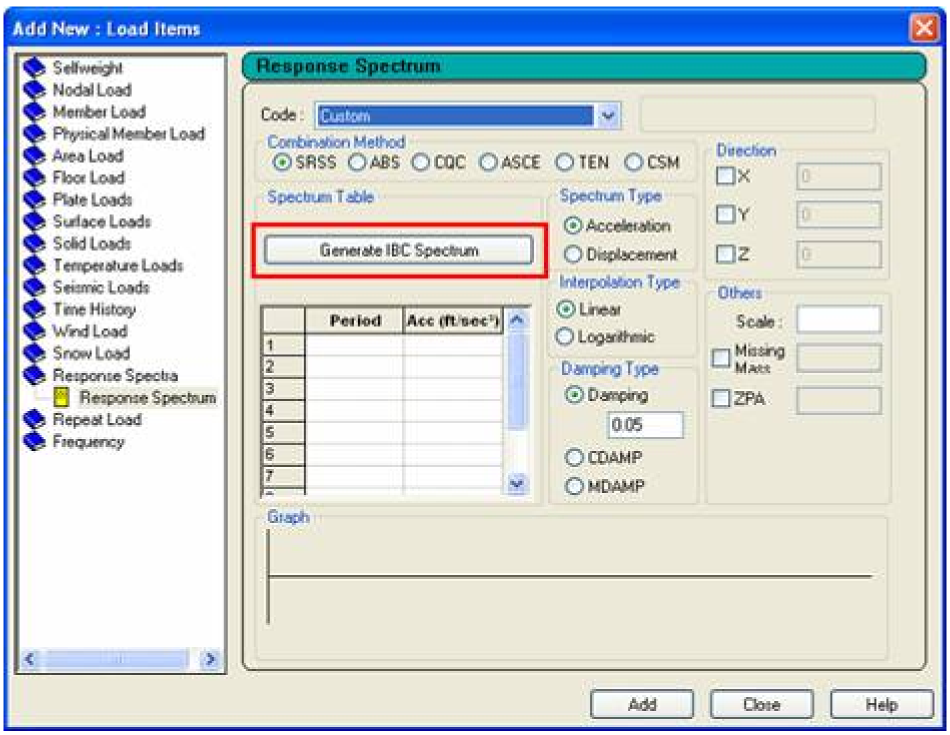
The user enters the latitude and longitude (or select a US zip code), STAAD.Pro then establishes the site coefficients, S1 and SS. The user then selects the appropriate site class (A-F) and with this information, STAAD.Pro calculates the MCE spectral acceleration and design spectral acceleration as per sections 11.4.3 and 11.4.4 of the ASCE7-05 code. Finally the design response spectrum will be generated based on section 11.4.5 of the ASCE7-05 code.

Note: There is no change to the input command structure.

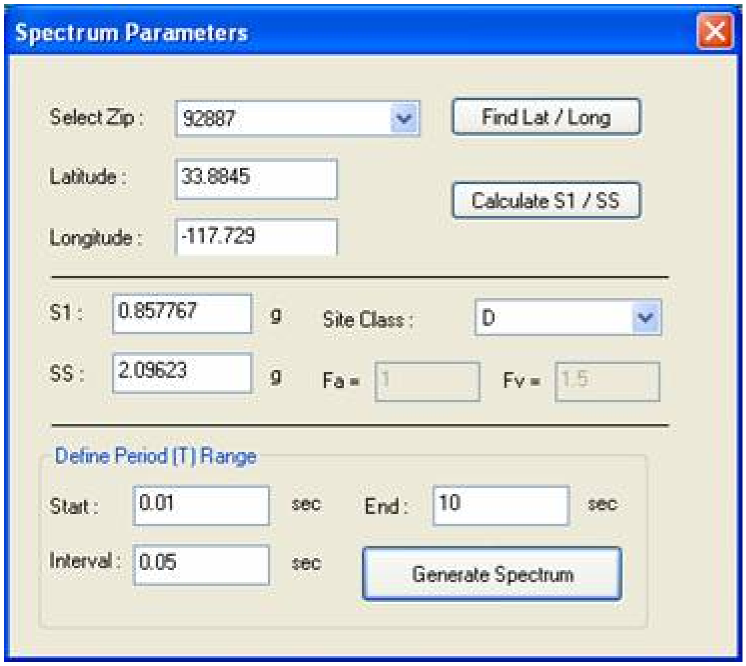
Description

Go to the **General | Load** page and in the **Load** dialog box, open the **Load Case Details** and select the appropriate primary load case with the masses (defined as load items) defined or create a new primary load case and add the masses, if required. Click on the **Add** button which will bring up the **Add New: Load Items** dialog box. Select the **Response Spectrum** option from the list on the left.

With the code option set to **Custom**, a **Generate IBC Spectrum** button will now be available as shown thus



Click on the **Generate IBC Spectrum** button to open the **Spectrum Parameters** dialog box thus



Refer to Spectrum Parameters dialog (on page 2862) for details on the input fields in this dialog.

To complete the generation of the IBC response spectrum pairs, click on the **Close** button. The completion of the Response Spectrum command continues as for all other Custom defined Response Spectrum commands. Note that the details of the spectrum Period, Acceleration pairs are displayed in the Response Spectrum window, but cannot be modified at this time.

For more details on the Response Spectrum loading command, refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499).

Related Links

- *M. To add an IBC 2000 response spectrum* (on page 862)

AD.2006-1002.1.2 Enhancement to Physical Member Query Window

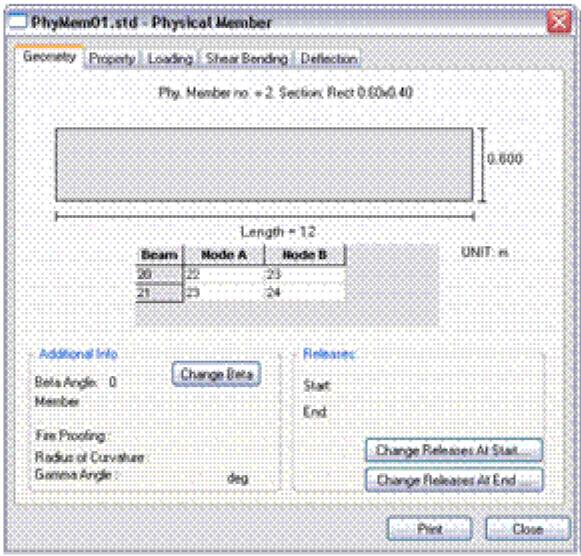
With the enhancement of the physical Member in the Pre-Processing Mode, the Query dialog box obtained when a physical member is double clicked.

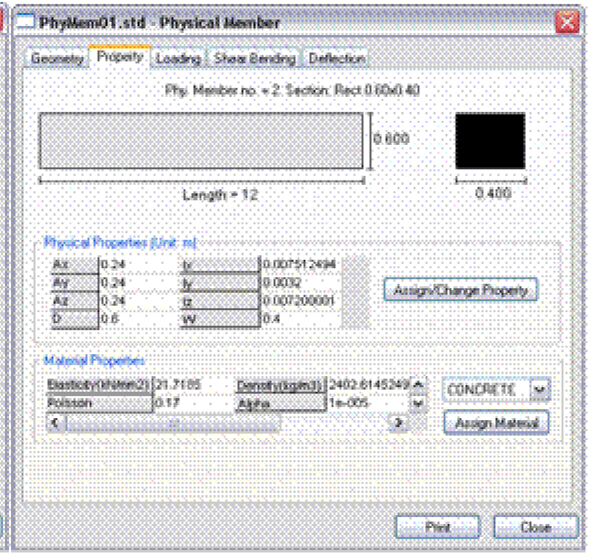
Select a Physical Member by using the Physical Member cursor,  and double-click on that to view the **Physical Member Query** dialog box.

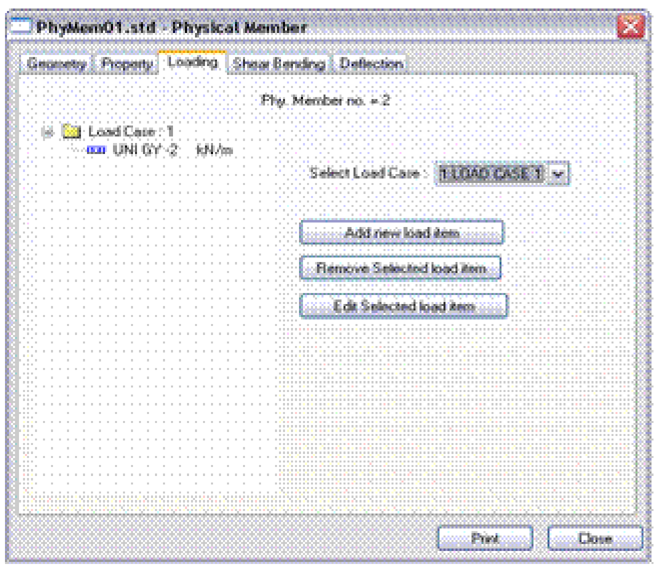
Pre-Processing Information

Prior to any analysis being performed, the physical member information available is displayed in the following three sheets **Geometry**, **Property**, and **Loading**.

All those common physical member attributes, assigned to that particular physical member (i.e., to all the analytical members comprising that physical member) are displayed in these information fields, as shown below:







Post Processing Information

If an analysis has been performed and there are results available, the following post processing results are made available in two additional sheets **Shear Bending**, to view the Shear Force diagram, Bending Moment Diagram, and **Deflection** to display deflected shape information.

Note: The design of Physical Members is only performed in the Steel Design Mode and note the batch method in the current version of STAAD.Pro.

Related Links

- *Physical Member dialog* (on page 3053)

AD.2006-1002.1.3 AAHSTO (LRFD) Design Code

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD). The Graphical User Interface (GUI) has been updated to allow the design parameters to be entered and applied as required.

Go to the **Design | Steel** page and in the **Steel Design** dialog box, select the option **AASHTO (LRFD)** from the Current Code drop-list. Click on the button **Select Parameters** to display the parameters available to be used with this design code. With all parameters displayed in the right Selected Parameters' panel, close the dialog box. Click on the **Define Parameters** to set the values for the parameters which can be added or assigned to selected members.

Click on the **Commands** button to add or assign the commands to the file to instruct a check or selection of sections based on the parameter settings.

The AASHTO design parameters listed in previous versions of STAAD.Pro as **AASHTO** in the Current Code drop-list of the **Steel Design** dialog box, have been renamed as 'AASHTO (ASD)

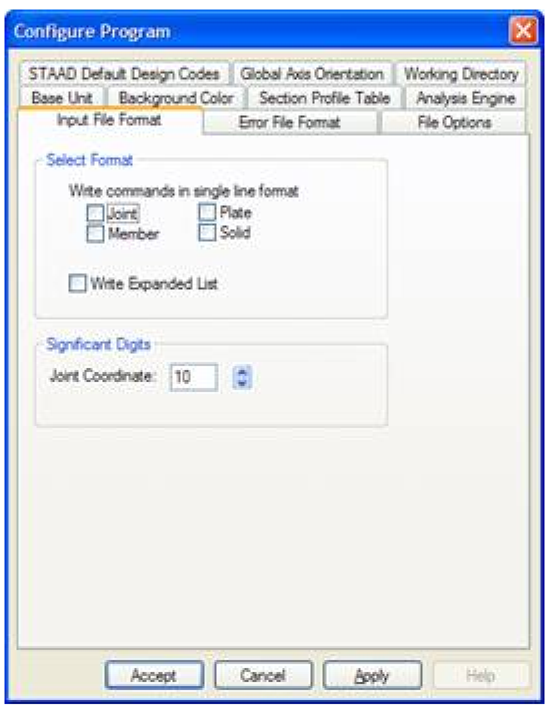
Note that access to design codes is controlled by your licence agreement and security settings. Not all users have access to all design codes. If this code is unavailable, see your STAAD.Pro supplier to obtain the updates needed to activate this or any other design code.

AD.2006-1002.1.4 Node Precision

A number of models require a higher than usual precision on the node co-ordinates that are used by the analysis engine.

With STAAD.Pro running, but without having any structural model loaded, select **File** > **Configure**.

In the resulting dialog, click on the **Input File Format** sheet, there is a new section Significant Digits which allows the required value to be set



Related Links

- *GS. Application Configuration dialog* (on page 59)

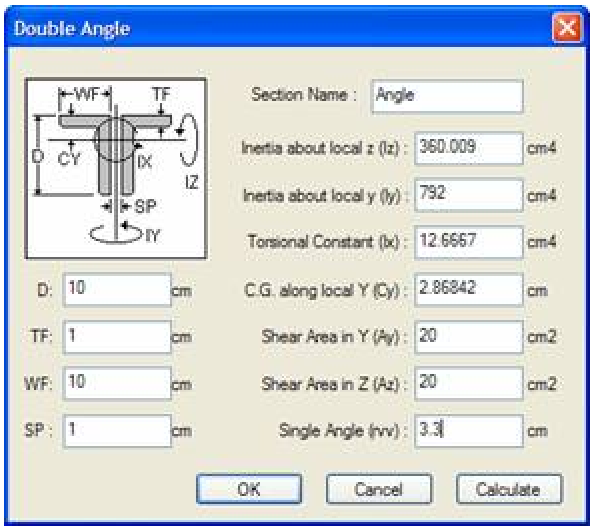
AD.2006-1002.1.5 User Steel Table - Double Angles

To design a beam that has been defined as a Double Angle from a User table, then an 11th parameter must be provided in the section parameters which defines the RVV value. The GUI has been enhanced to support this additional parameter.

Description

Select **Tools** > **Create User Table**.

Start a new table which is of type **DOUBLE ANGLE** and click on the **Add New Property** button which will display the following dialog which allows the value of Rvv to be included in the section property definition.



Related Links

- *TR.19.4 Double Angle* (on page 2248)

AD.2006-1002.2 Features Affecting the Analysis and Design Engine

The following section describes the new features have been added to the analysis and design engine and existing features that have been updated or modified.

AD.2006-1002.2.1 AASHTO Design Code

The STAAD.Pro analysis and design engine has been enhanced with the addition of a new design code, AASHTO (LRFD).

AD.2006-1002.2.2 Updated Parameters on Indian IS 1893 Static Seismic Loading Command

The static equivalent method for creating loads as per the Indian seismic code, IS 1893 has been enhanced with the option for checking soft stories.

Related Links

- *TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load* (on page 2387)

AD.2006-1002.2.3 Updated Parameters on Indian IS 1893 Response Spectrum Command

The method for creating a response spectrum to the Indian seismic code, IS 1893 has been enhanced with the option for checking soft stories and documentation on the torsion option added.
Related Links

- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)

AD.2006-1002.3 Features Affecting the Steel Design Mode

Several new features have been added and existing features have been modified in the Steel Designer section of the program, also known as the Steel Designer Mode. These are explained in the following pages.

AD.2006-1002.3.1 Access to Physical Members Created in Pre-Processing Mode

Physical members can be created in the Pre-Processor mode and their definition stored in the STAAD.Pro input file (`*.STD`). Physical members that have been created in the Pre-Processing Mode are now available in the Steel Designer Post Processing Mode. Previous versions of STAAD.Pro would require these members being re-defined in the Steel Designer Mode.

Create Physical Members as defined in the manual AD.2006.1.1 Physical Member Interface (on page 333). Perform an analysis and enter the Steel Designer Mode. Go to the Member Design | Member Setup Page, and note that the members that were created in the Pre-Processing Mode are ready to be utilised in the Steel Designer Mode.

AD.2006-1002.4 Features Affecting the Concrete Design Mode

Several new features have been added and existing features have been modified in the RC Designer section of the program, also known as the Concrete Design Mode. These are explained in the following pages.

AD.2006-1002.4.1 DXF Output of Beam and Column Designs

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006-1002.5 Features Affecting the Advanced Slab Design Mode

The Mode previously known as the ADAPT Slab design has been redefined as the Advanced Slab Design and has been enhanced to support integration with RAM Concept.

Note: Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

AD.2006-1002.5.1 Advance Slab Design Mode

Refer to the D. Advanced Slab Design (on page 1073) of the User Interaface help for details.

AD.2006-1002.6 Features Affecting the Piping Mode

STAAD.Pro can utilise the pipe layout and reactions created in the applications ADLPipe or AutoPipe. The pipe model can be imported in the Piping Mode. The following describes the method of using the Piping Mode and features recently added to this mode.

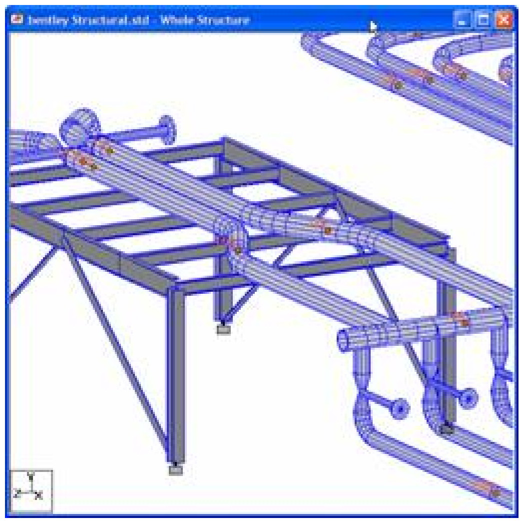
AD.2006-1002.6.1 Piping Mode

Refer to the M. Piping workflow (on page 902) in the User Interface help

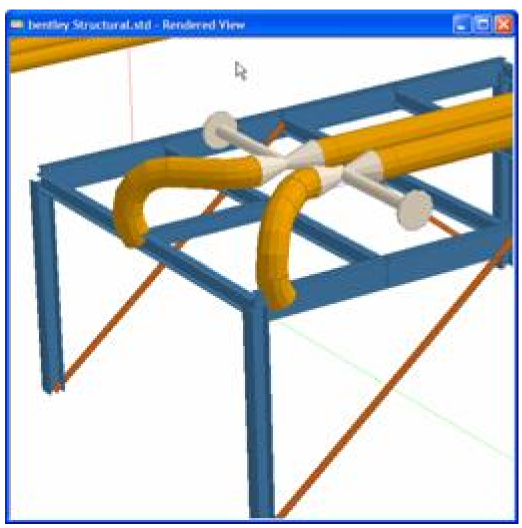
AD.2006-1002.6.2 3D Rendered View of Piping Model

In order to perform quick, visual clash detections, the 3D properties of the piping model have been added to the 3D section view and Rendered 3D View.

To view the solid 3D shapes in the graphical window, select the menu option, View > Structure Diagrams… This opens the Diagrams dialog in the Structures sheet. In the 3D Sections group, select the option 'Full Sections' to display a solid 3D shape of the structure model and piping arrangement, thus:



To view a rendered 3D view of the structural model with the piping arrangement, select the menu option, View > 3D Rendering… This creates a new window and displays a 3D perspective view of the model and piping thus:



Note: The piping model is only displayed in the Piping Mode.

AD.2006-1002.7 Additional STAAD.Pro 2006 Build 1002 Features

The following pages outline the additional features that have been added to STAAD.Pro in this version.

AD.2006-1002.7.1 Modifications in REI Security System

For details on the current security system for STAAD.Pro, please refer to the *STAAD.Pro Installation Guide* (`STAAD.Pro_InstallGuide_en.pdf`).

AD.2006-1002.7.2 Online Help

In order to assist the user operate STAAD.Pro, a number of context sensitive links have been added to the GUI such that clicking on a **Help** button or pressing the **<F1>** key will go directly to an appropriate help topic within the online documentation.

STAAD.Pro 2006 Build 1001 Release Report

The Software Release Report for STAAD.Pro 2006 Build 1001 contains detailed information on additions and changes that have been implemented since the final build of STAAD.Pro 2005. This document should be read in conjunction with all other STAAD.Pro help.

AD.2006.1 Features Affecting the Pre-Processor

Several new features have been added and existing features have been modified in the pre-processor section of the program. They are explained in the following pages.

AD.2006.1.1 Physical Member Interface

STAAD.Pro will allow grouping analytical predefined members into physical members using a special member group `PMEMBER`. `PMEMBER` defines a group of analytical collinear members, with same cross section and material property.

To model using PMEMBER, one needs to model regular analytical members and the start grouping those together.

While creating a PMEMBER, the following are the pre-requisites,

- Existence of the analytical members in the member-list.
- Selected members should be interconnected.
- The selected individual members are collinear.
- Local axis of the individual members comprising the physical member should be identical (i.e., x, y and z are respectively parallel and in same sense).
- A member in one Physical Member Group should not occur in any other Physical Member Group.

Related Links

- *TR.16.2 Physical Members* (on page 2238)

General Format

PMEMBER can be created either in modeling mode or Steel-Designer mode. Modeling mode and Steel-Designer mode physical members will be labeled as M and D, respectively. Modeling mode PMEMBER will allow variable cross-sections. Steel-Designer mode will allow importing PMEMBER-s created in modeling mode.

General Format

Following STAAD commands related to PMEMBER are implemented.

```
DEFINE PMEMBER
PMEMBER PROPERTY
PMEMBER CONSTANT
PMEMBER LOAD
PRINT PMEMBER FORCE
```

To define a Physical Member, the following command is used after the MEMBER INCIDENCE Command:

```
DEFINE PMEMBER
member list PMEMBER pmember-no
```

Example

```
 JOINT COORDINATE
1 0 0 0 6 10.0 0 0
MEMBER INCIDENCE
1 1 2 5
DEFINE PMEMBER
1 TO 5 PMEMB 1
```

To define the member property of a Physical Member, the following command is used:

```
PMEMBER PROPERTY
pmember-list PRIS …
```

The Physical Member supports all types of member properties available in STAAD.

If multiple definitions of member properties for a particular analytical member is encountered (e.g. analytical member properties is defined twice, once via PMEMBER PROP command and again via MEMBER PROP command, then MEMBER PROP command will override PMEMBER PROP definition.

To define the Material constants of a Physical Member, the following command is used:

```
PMEMBER CONSTANT
E  CONCRETE pmember-list
DEN CONCRETE pmember-list
…
```

Any member, which is a part of any PMEMNER is not allowed to be assigned constants explicitly.

Note: Loads are applied directly to PMembers using the PMEMBER LOAD command. See “PMember Load Specification” for details.

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

```
PRINT PMEMBER FORCE
```

This command will produce member forces for all the analytical members in the group.

Graphical User Interface for Physical Member

The following steps are to be followed for creation of a Physical Member

Post-Processing Features Related to Physical Members

In the post processing results, to view the Shear Force diagram, Bending Moment Diagram and Deflected shape, use the member query dialog box.

Select a Physical Member by using the Physical Member cursor, and double-click on that to view the member query dialog box.

Design of Physical Members

In the current version of STAAD.Pro, the design of a Physical Member is not handled.

To Create a Physical Member

Make a frame model in STAAD.Pro, comprising of two or more consecutive beams.

1. Select two or more colinear, interconnected beams in th emodel.

   Refer the pre-requisites of the analytical members for details.
2. Either:

   Select the **Form Member** tool in the **Physical Member** toolbar

   or

   Right-click and select **Form Member** from the pop-up context menu.

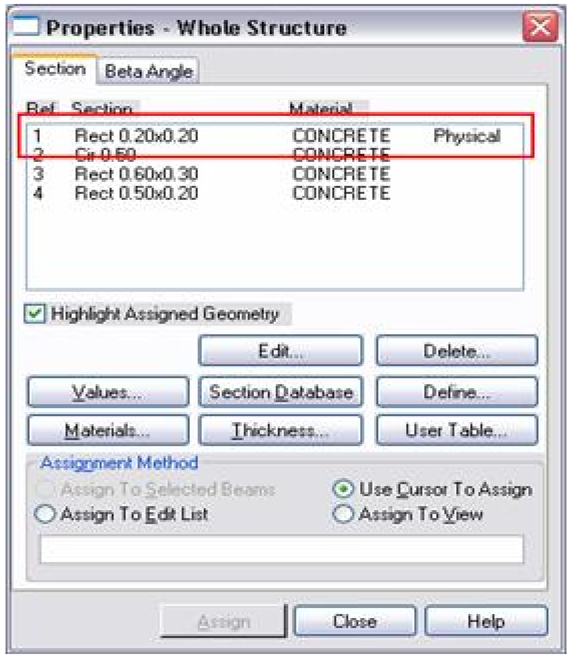
The details of the formed Physical Member can be seen in the **General | Physical Member** page at the right pane of the window.

To assign properties to a physical member

You must first create one or more physical member properties and then these can be assigned to physical members.

1. Select the **General | Property** page.
2. Select the **Toggle Physical Member mode** tool in the **Physical Member** toolbar.
3. Create a physical property:
   a. In the **Properties - Whole Structure** dialog, click **Define**.
      The **Propety** dialog opens.
   b. Select a property class and define the required property parameters.
   c. Click **Add**.

   A new property is added to the list in the **Properties - Whole Structure** dialog. This property is marked as “Physical”.



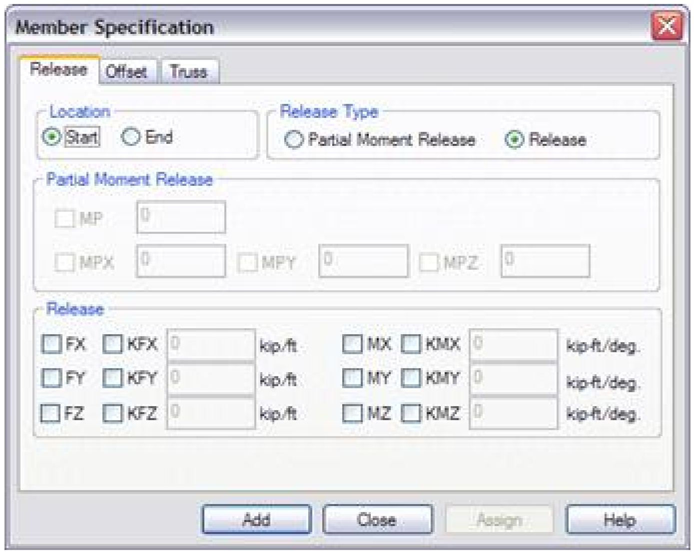
4. Assign this property to the physical member.

Note: For a Physical Member whose physical member property is already assigned, the individual analytical members in that Physical member will adopt the same member property that of the physical member. However, in case, where the analytical member property is assigned to any member in that physical member, then this analytical member property will supersede physical member property.

To assign specifications to a physical member

Only the Release, Offset, and Truss member specifications are available for physical members.

1. Select the **General | Spec** page.
2. Select the **Toggle Physical Member mode** tool in the **Physical Member** toolbar.
3. Create the member specification:
   a. Click **Beam** in the **Specifications - Whole Structure** dialog.
   The **Member Specification** dialog opens.



b. Select the dialog tab corresponding to the specificaiton type you want to add.
c. For Release or Offset specifications, specify the parameters to define the specificaiton.
d. Click **Add**.

A new specification is added to the list in the **Specifications - Whole Structure** dialog. Release and Offset specifications are marked as "Physical" where as Truss specifications are labeled **PMEMBER TRUSS**.

4. Assign this specification to the physical member.

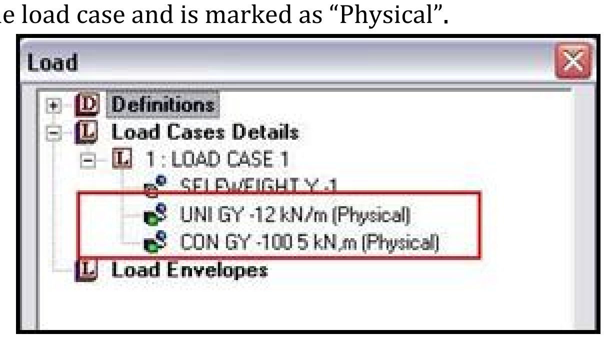
Note: For a Physical Member whose physical member specification is already assigned, the individual analytical members in that Physical member will adopt the same member specification that of the physical member. However, in case, where the analytical member specification is assigned to any member in that physical member, then this analytical member specification will supersede physical member specification.

To assign loads to a physical member

1. Select the **General | Load & Definition** page.
2. Select a load case in the **Load & Definition** dialog and click **Add**.

   Physical member loads can be assigned to a new or existing load case.

   The **Add New : Load Items** dialog opens.
3. Select the **Physical Member Loads** tab.
4. Specify the load item paramters for a load type and click **Add**.
   The load item is added to the load case and is marked as "Physical".



5. Assign the physical load items to physical members.

Note: For a Physical Member whose physical member Load is already assigned, the individual analytical members in that Physical member will adopt the member loads internally depending on the physical member load. However, in case, where the analytical member Load is assigned to any member in that physical member, then this analytical member Load will superimpose on the physical member Load.

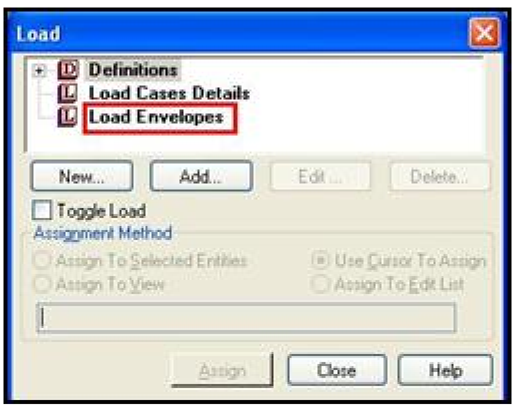
AD.2006.1.2 Load Envelopes

Load page will allow creation of Envelopes that can be saved as part of the STAAD input file. The user will be able to define multiple load envelopes each consisting of groups of predefined load cases. These envelopes can latter be used for post-processing. For example post analysis results may be viewed for a selected load envelope. As far as the STAAD engine is concerned, ENVELOP command will translate in to a LOAD LIST command.

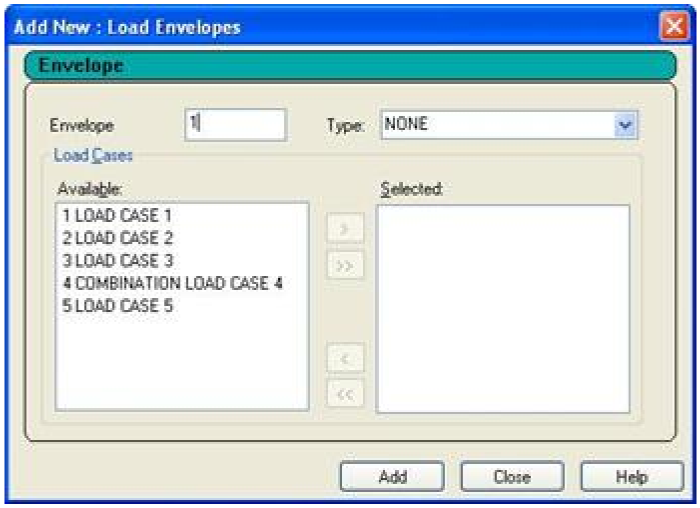
The envelope can be tagged with optional key words to specify qualitative nature of the load or load combination cases included in the envelop definition. Based on the nature of the load cases in the envelope, the users can define appropriate design parameters for each envelope. For example, for design under wind load condition, most of the design codes allow increase of allowable stresses. Design routine can increase the allowable stress used in interaction equation, when it does the design for the envelope. Another application of this feature can be to specify separate load groups for serviceability check, working stress and limit state checks.

Description

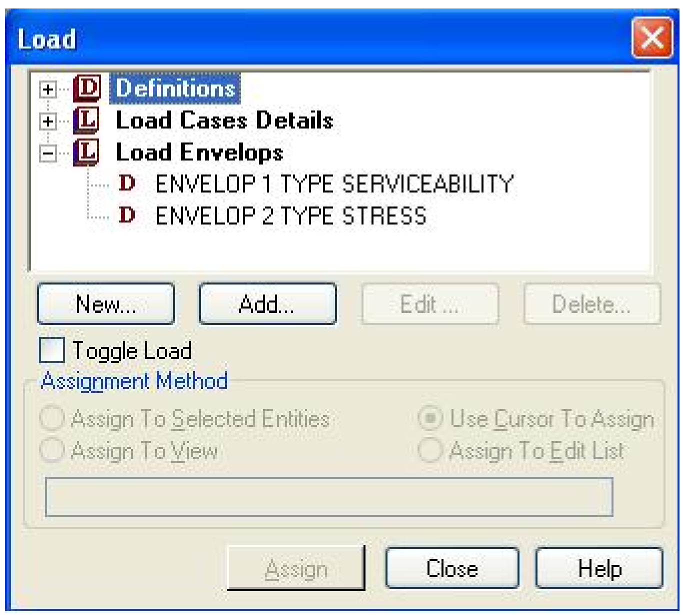
The option for defining **Load Envelopes** appears within the **Load** dialog box in the **General | Load** page as shown in the next figure.



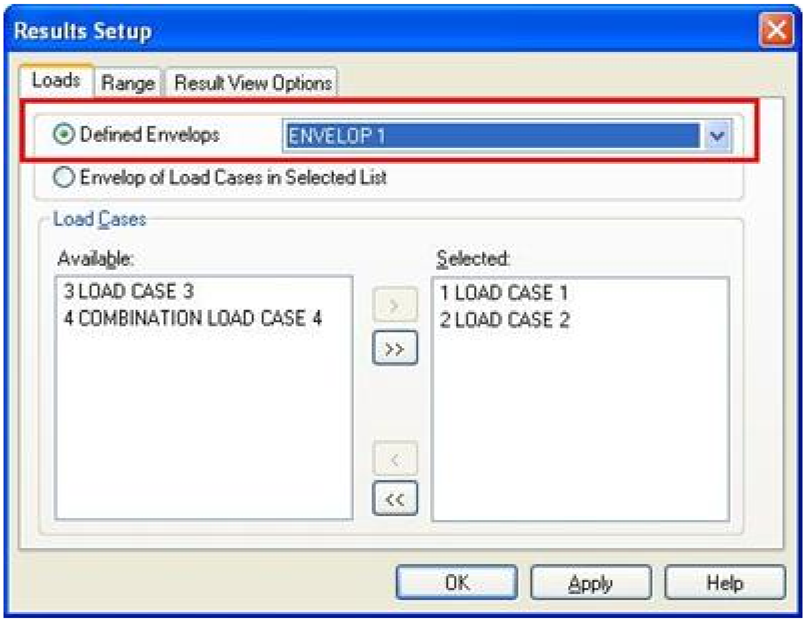
To define an envelope, select the Load Envelopes and click on the Add button and the Add New : Load Envelopes dialog box will come up as shown in the next figure.



The Envelope will be identified by the number that appears in the Envelope edit box. One can select an appropriate type for the load envelope like stress, serviceability etc. depending on the nature of the loads selected for the envelope. All the predefined primary load cases and combination cases will appear inside the **Available** window. One can now select one or more of these cases, bring them to the **Selected** window on the right hand side and click on the **Add** button to create the envelope. Once an envelope is created, it will be displayed in the **Load** dialog box as shown in the next figure.



In the post processing mode, the **Results Setup** dialog box will include option for selecting **Defined Envelopes** for the purpose of displaying results as shown in the next figure. Once an envelope is selected the corresponding loads can be seen within the **Selected** window.



General Format

The load envelope commands are written in the STAAD input file as shown below.

```
DEFINE ENVELOP
load-case-list ENVELOP # TYPE { NONE | STRESS | SERVICEABILITY | COLUMN | CONNECTION |
STRENGTH }
END DEFINE ENVELOP
```

Example

```
DEFINE ENVELOP
1 2 ENVELOP 1 TYPE SERVICEABILITY
3 5 ENVELOP 2 TYPE STRESS
END DEFINE ENVELOP
```

The first line within DEFINE ENVELOPE command means that load cases numbered 1 and 2 make up the serviceability type load envelope 1. Similarly load cases 3 and 5 define the stress type load envelope 2.

To print out the support reactions corresponding to load envelope 1, that includes load cases 1 and 2, the following commands will be defined in the input file

```
 LOAD LIST ENV 1
PRINT SUPPORT REACTIONS
```

If ENV keyword is encountered, the list will be interpreted as list of envelopes rather than a list of load cases.

Note: Please refer to the example file `Load_envelopes.std`, which is available under the different country folders.

Related Links

- *TR.40 Load Envelope* (on page 2663)

- *M. To create a load envelope* (on page 901)

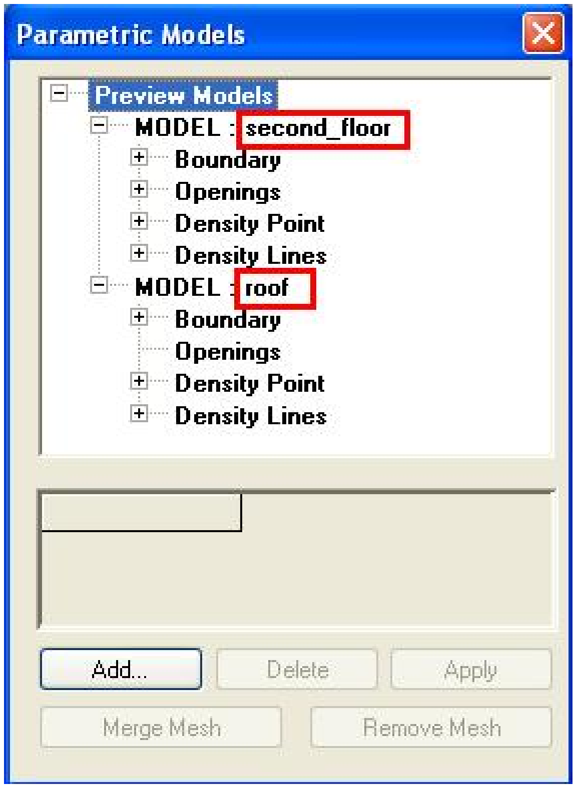
AD.2006.1.3 Persistency of Parametric Mesh Model in STAAD Input File

In the past, once the parametric mesh model was merged with the base model, no information about the parametric mesh was retained by STAAD. So, if any modification was required at a later stage, the parametric mesh had to be created afresh. Parametric model feature has now been enhanced and multiple parametric mesh models can now be saved as part of the STAAD model. This would allow the users the flexibility to come back to the saved mesh models at any time and make modifications to it like adding an opening or adding a density line.

The parametric mesh model data is now saved as part of the STAAD input file. Special tag based commands has been introduced to support saving of parametric mesh models as part of the STAAD input file as shown below.

```
2072 1114 1113 1160; 2073 1045 1160 1113;
   ELEMENT PROPERTY
   810 TO 1779 1821 TO 2073 THICKNESS 1
   <! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
   PARAMETRIC MODEL SLAB
   MESH PARAM 0 3
   MESH ORG 3 5 8
   BOUNDARY 10
   11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
   OPENING CIRC 72 360 96 43.2666 12
   OPENING POLY 5
   216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360 100.8 2 216 360 100.8 2
   DENSITY POINTS 2
   180 360 168 1 360 360 168 1
   DENSITY LINE  0 360 168 100 180 360 168 200
   DENSITY LINE 180 360 168 1 360 360 168 1
   DENSITY LINE 360 360 168 1 540 360 168 1
   DENSITY LINE 180 360 0 1 180 360 168 1
   DENSITY LINE 180 360 168 1 180 360 336 1
   DENSITY LINE 360 360 0 1 360 360 168 1
   DENSITY LINE 360 360 168 1 360 360 336 1
   DENSITY LINE 54 360 302.4 1 162 360 201.6 1
   DENSITY LINE 216 360 201.6 1 324 360 235.2 1
   GENERATED PLATES ALL
   END
   <! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
   PARAMETRIC MODEL FIRST_FLOOR_SLAB
   MESH PARAM 60 3
   MESH ORG 2 3 5
   BOUNDARY 6
   36 1 65 1 66 1 53 1 52 1 51 1
   GENERATED PLATES ALL
   END
   !> END GENERATED DATA BLOCK
   !> END GENERATED DATA BLOCK
   DEFINE MATERIAL START
   ISOTROPIC STEEL
```

Go to the **Geometry | Parametric Models** page and the saved parametric mesh models will appear within the **Parametric Models** dialog box as shown in the next figure.



There are two parametric mesh models named **second_floor** and **roof** inside the **Parametric Models** dialog box as shown in the previous figure.

Note: Please refer to the example file `parametric_models.std`, which is available under various country folders.

Related Links

- *Parametric Models dialog* (on page 2734)

AD.2006.1.4 Persistency of Parameters User to Generate ASCE Wind Load in STAAD Input File

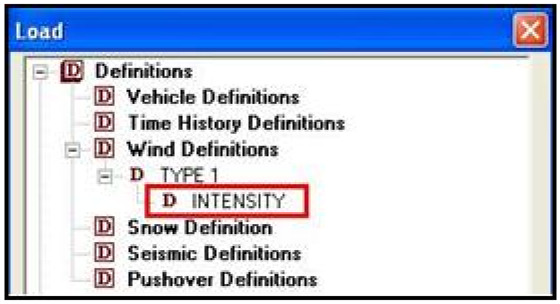
In the past, once the wind load as per the ASCE-7 code was generated automatically using the wind load generator, no information about the parameters was retained by STAAD. So, there was no way to check the parameters, based on which the generation was done. Moreover, if any modification was required at a later stage, all the parameters had to be defined afresh. Automatic wind load generation feature has now been enhanced and the parameters can now be saved as part of the STAAD.Pro model.

Special tag based commands has now been introduced to support saving of ASCE parameters as part of the STAAD input file as shown below.

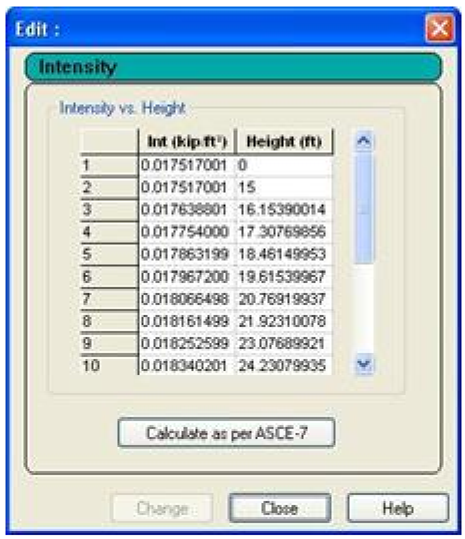
```
DEFINE WIND LOAD
   TYPE 1
   INT 0.017517 0.017517 0.0176388 0.017754 0.0178632 -0.0179672 0.0180665 0.0181615
0.0182526 0.0183402  0.018506 -0.0184246 0.0185846 0.0186607 0.0187345 -
   HEIG 0 15 16.1539 17.3077 18.4615 19.6154 20.7692 21.9231 -23.0769 24.2308 25.3846
26.5385 27.6923 28.8462 30
   <! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
   ASCE-7-2002:PARAMS 85.000 MPH 0 2 2 0 0.000 FT 0.000 FT -0.000 FT 1 1 30.000 FT
```

```
80.000 FT 60.000 FT 2.000 0.010 0 -
   0 0 0 0 0.982 1.000 1.150 0.850 0 0 0 0 0.631 0.800 0.550
   !> END GENERATED DATA BLOCK
```

When a user goes back to the Load page and goes to **Definitions | Wind Definitions** page and double clicks on the **Intensity** as shown in the next figure,



the intensity versus height data shows up as shown below.



One can now click on the **Calculate as per ASCE-7** button and the parameters that were initially defined are going to appear in the resulting dialog boxes.

Note: Please refer to the example file `ASCE_WIND_load_generation.std`, which is available under various country folders.

Related Links

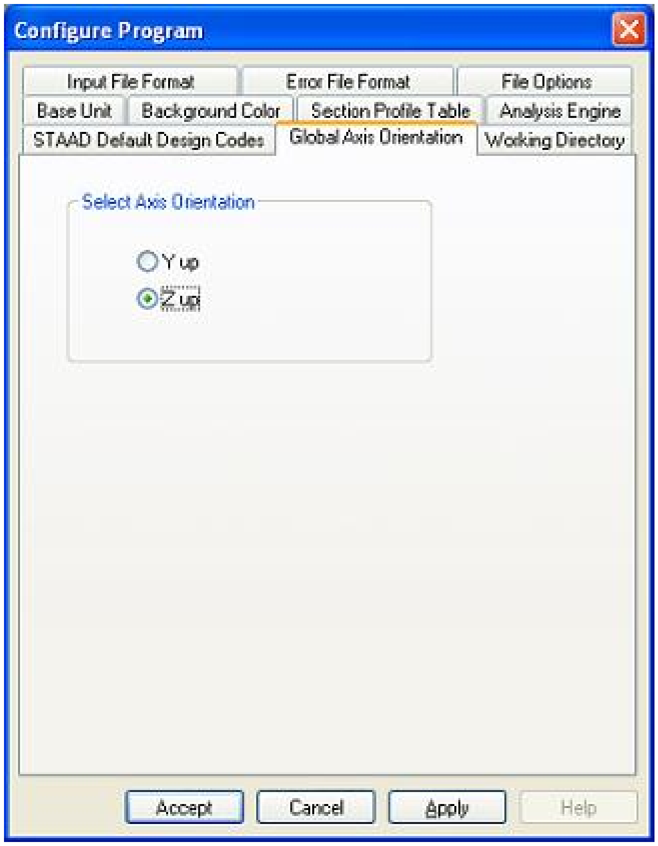
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *ASCE 7 Wind Load dialog box* (on page 2882)
- *Wind Load tab* (on page 2850)
- *M. To add an ASCE 7 wind load definition* (on page 845)

AD.2006.1.5 Enhancement of Z UP System

Earlier, when geometry created using a CAD software that used Z axis as the vertical axis, had to be imported into STAAD.Pro, the users had to reorient the model to make the Y axis the vertical axis before getting the model into STAAD.Pro. This was necessary because a number of STAAD.Pro load generation commands did not work when the Z axis was defined as the vertical axis. In STAAD.Pro 2006 these limitations has been addressed. All the load generations are going to work with the Z axis up coordinate system.

The default coordinate system for a model can be set by going to **File** > **Configure**. The **Configure** option is only available when no STAAD model is open. If a STAAD model is open, one has to click on **File** > **Close** to close the file first.

The **Configure Program** dialog comes up as shown in the next figure.



The **Z up** option can then be selected and applied to the model. This would set the Z axis to be the vertical axis instead of the default Y axis. Subsequently all modeling, analysis and postprocessing items would be based on this coordinate system.

Related Links

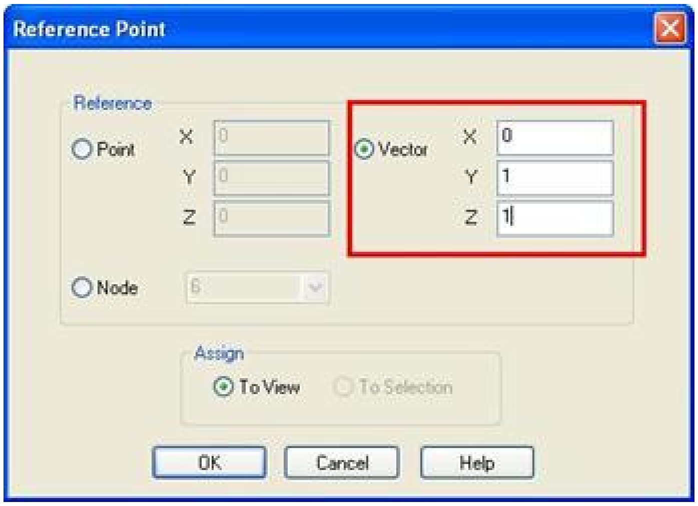
- *GS. To use Z as the vertical axis* (on page 50)

AD.2006.1.6 Specification of Member Orientation Using Reference Vector

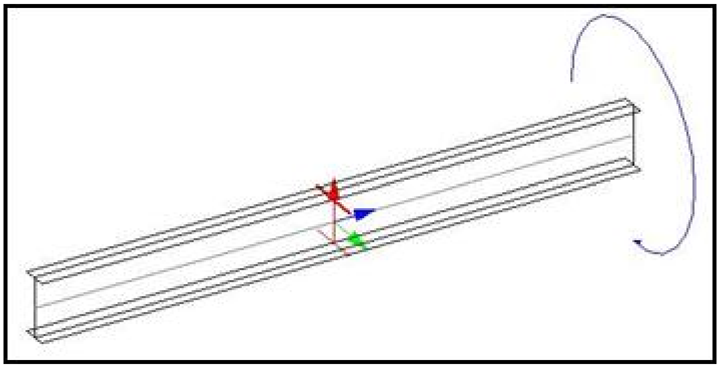
This feature will allow users to orient members by specifying a direction vector. Beta angles will be calculated by the software by itself based on the direction vector specification. The new reference vector will be defined with respect to the local coordinates of the member. This is going to make the task of orienting members much easier.

To access the feature go to **Commands** > **Geometric Constants** > **Member Reference Point**.

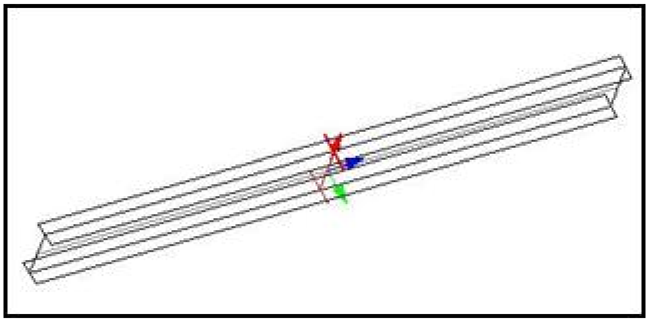
The **Reference Point** dialog box comes up where the vector for the reference axis can be specified as shown in the following figure. The local Y axis for the member is going to oriented along the vector. This X, Y, Z values are going to be based on the local axis system of the member.



The feature is explained in the next section with an example. Let us consider the beam shown in the next figure. The requirement is to rotate the cross section about the axis of the member in the direction as shown, such that the angle of rotation is governed by a slope of 1 horizontal to 2 vertical.



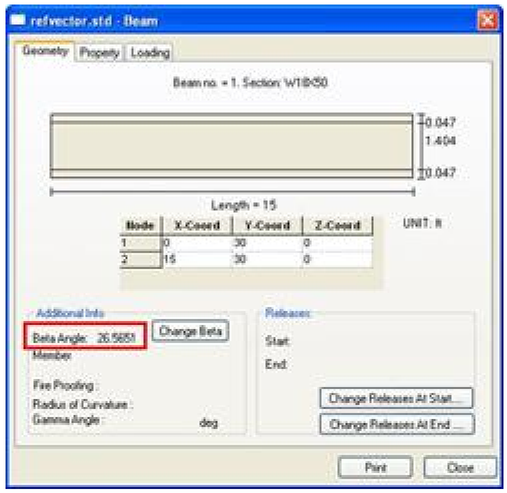
1. Select **Commands** > **Geometric Constants** > **Member Reference Point**. The **Reference Point** dialog opens.
2. Select the **Vector** option and specify the reference vector so that it is oriented as per the required slope.
3. Click **OK**. The member is rotated to the referenc vector.



The member gets oriented correctly without the user having to take the trouble of calculating the beta angle. In the STAAD input file, the following command lines get written.

```
CONSTANTS
REFVECTOR 0 2 1 ALL
```

Beta angle for the member can be easily figured out by simply double clicking on the member that brings up the **Beam** dialog box as shown next. The correct beta angle of 26.5651 is automatically calculated by STAAD.Pro based on the reference vector specification.



Related Links

- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *M. To align a member to a reference point* (on page 801)

AD.2006.1.7 Single File Archive to Save All STAAD Input & Ouput Files

Based on user configurations, file open and save dialogs will allow user to select .stz (std archive) files. The archive will be expanded to TEMP folder or user configurable working folder. All input/output will be created at that location.

While exiting the program/session, all of those files will be archived again under same name and copied to original source folder.

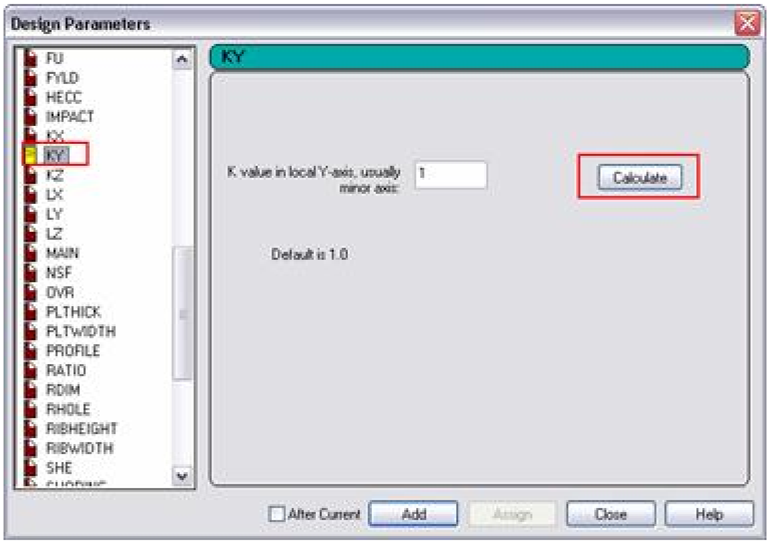
Archive related commands can be found on the **File** menu.

AD.2006.1.8 Auto Calculation of Effective Length Factors of Members as per AISC Code

STAAD.Pro has a new feature to auto-calculate the Effective Length Factors (KY, KZ etc), for the members to be designed as per AISC-ASD Code.

To make use of this feature, the user needs to go to the **Design | Steel** page. Select the Country Code as **AISC ASD**. Select the member for which the Effective Length Factors are to be calculated. Click on the **Define Parameters** button and select the parameter **KY** or **KZ**.

The following dialog box will appear.



The only difference from the previous version of STAAD.Pro is the addition of the **Calculate** button as marked in the figure.

On clicking the **Calculate** button, the program will ask whether the braced or un-braced effective length of the selected member is required.

After getting confirmation from the user, the program will calculate and display the value of the Effective Length Factor in the Edit box.

Now, click on the **Assign** button to associate this value of the respective Effective Length Factor with the selected member.

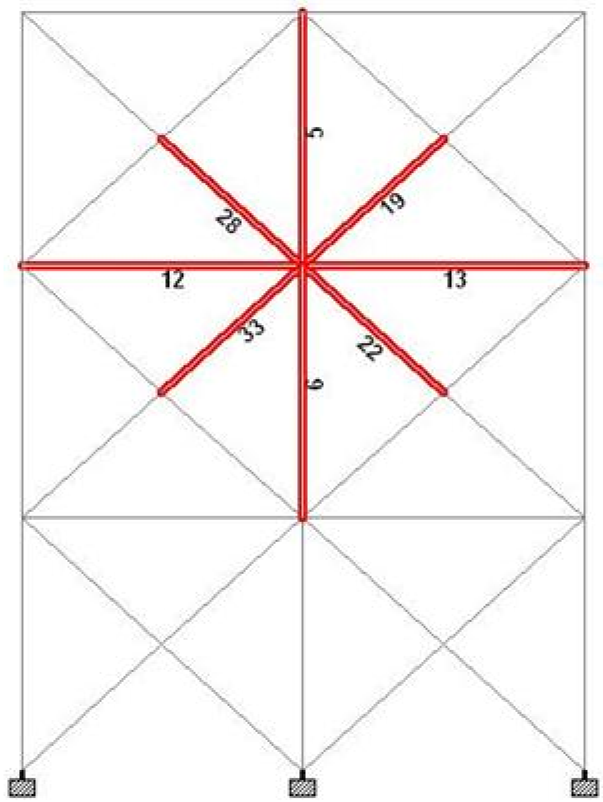
Note: The calculation of the effective length factors are done as per a paper titled *Compression Members* presented by George Tsiatas.

AD.2006.2 Features affecting the Post-Processor (Results Mode)

Several new features have been added and existing features have been modified in the post-processor section of the program. They are explained in the following pages.

AD.2006.2.1 Generation of Transfer Force Report for Connection Design

STAAD.Pro can now calculate the “Transfer force” or “pass through force” that can be used for connection design. This feature is based on a paper on the subject by Dr. William. A. Thornton. Refer to the next figure which shows beams and bracing members connected to either side of the column.

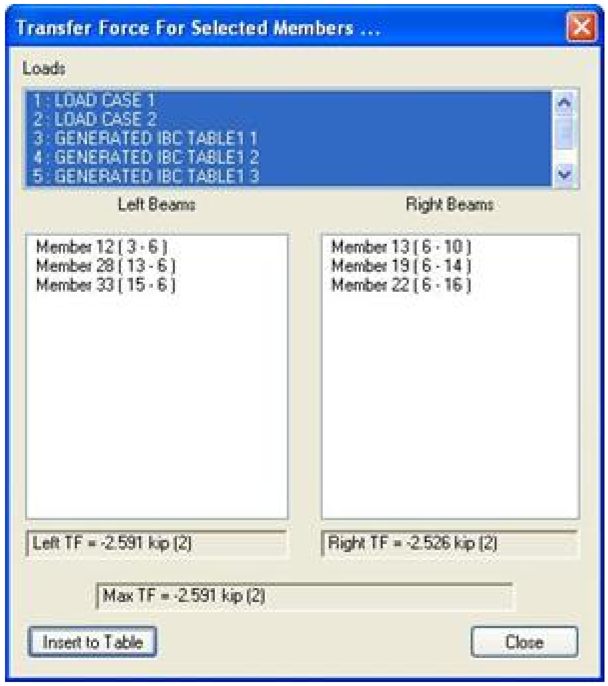


Transfer force is simply the maximum net horizontal force that gets transferred from the one side of the column to the other through the connection. So STAAD.Pro checks the forces in the members framing into each side of the column and finds out the resultant horizontal force for either side. Typically the resultant forces on the two sides would not be equal as some amount of force will be taken up by the column in shear. The greater of the two resultants is reported by STAAD.Pro as the transfer force. The option to determine transfer force automatically, will save engineers considerable time and effort as in most cases, they have to report the transfer forces in the design drawings. The transfer force feature only works for a beam column bracing system lying in a vertical plane.

Description

Go to the Post processing mode by selecting **Mode | Postprocessing** or by clicking on the **Postprocessing** tab above the graphics window. Select all the members that frame into any connection as shown in the previous figure. Then select **Report** > **Column Transfer**.

A dialog box titled **Transfer Force for Selected Members** comes up as shown in the next figure. The **Loads** box displays the load cases that has been considered to calculate the transfer force. By default all load cases are considered but one can exclude a few of them by simply clicking on the load case within the **Loads** box. The boxes **Left Beams** and **Right Beams** list all the members on the respective sides of the column along with their incidences. The boxes **Left TF** and **Right TF** shows the resultant horizontal force from either side. **Max TF** reports the transfer force.



Click on the **Insert to Table** button and a table will be generated containing all the transfer force information as shown in the next figure.

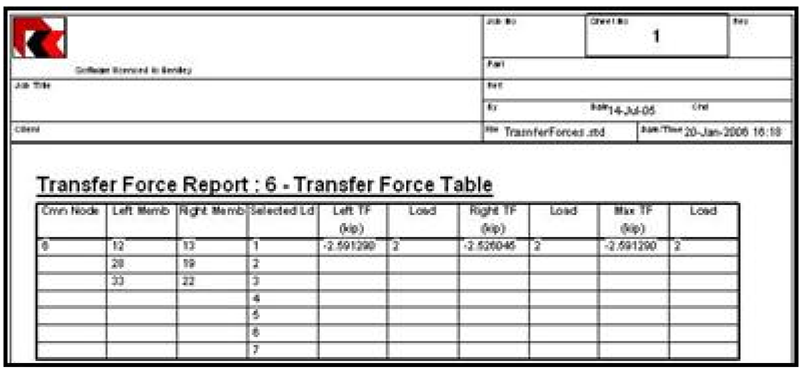
TrasnferForces.std - Transfer Force Report : 6

	Cmn Node	Left Memb	Right Memb	Selected Ld	Left TF kip	Load	Right TF kip	Load	Max TF kip	Load
1	6	12	13	1	-2.591290	2	-2.526045	2	-2.591290	2
2		28	19	2						
3		33	22	3						
4				4						
5				5						
6				6						

Transfer Force Table

The transfer force information can also be included as part of a report by selecting **File** > **Report Setup** or by selecting the **Report Setup** tool  which will open the **Report Setup** dialog box as shown in the next figure.

By default, the **Transfer Force Report** appears in the selected list of items within the dialog box. A report with the transfer force data can then be generated as shown next.



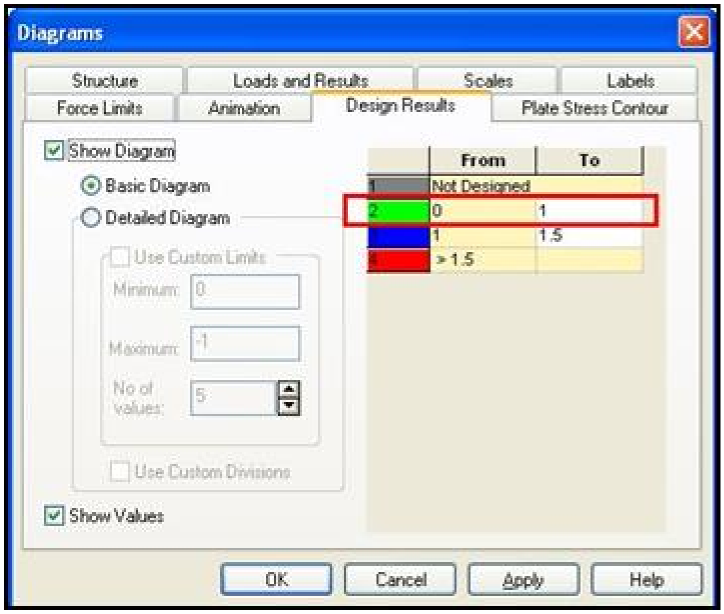
Related Links

- *Transfer Forces for Selected Members dialog* (on page 3002)
- *P. To generate transfer forces report* (on page 1996)

AD.2006.2.2 Customizable Color to Display Unity / Check Utilization Ratio

View steel design results.

Default color for display of passed members was black. It has now been changed to green. The colors representing the various ranges for utilization ratios can be set by the user.



AD.2006.2.3 Enhancement to Steel Designer BS 5950 Calculation Sheets

The design calculation sheets of a steel section that has been designed with a BS 5950-1:2000 design brief has been modified with the following enhancements.

1. The terminology for elastic and plastic sections updated to use same terminology as in the design code.
2. A summary has been added at the end of the calculation sheet.
3. The section classification summary has been improved.
4. The reported dimensions of the section in the 'Compression Flange Buckling Check' have been improved.
5. The status for various checks has been re-titled.
6. Details of check to 4.8.3.3.1 added when required.
7. There are improved details of the clauses 4.8.2., 4.8.3. and Annex I1 calculations.
8. The 4.3.6. Lateral Torsional Buckling Check has been enhanced.
9. Additional detailed calculations for axially loaded members have been added to the document.
10. A number of spelling issues have been addressed.

AD.2006.3 Features affecting Analysis and Design

Several new features have been added and existing features have been modified in the analysis and design section of the program. They are explained in the following pages.

AD.2006.3.1 Pushover Analysis

This is a set of procedures to implement a Pushover Analysis as defined in the document FEMA 356:2000.

STAAD Pushover analysis in STAAD is a static, non-linear procedure in accordance with FEMA 356 specification. Basically, in this method, the magnitude of the lateral push load is increased progressively according to a predefined loading pattern until either loading or the deflection reaches the described level.

Please refer to M. Pushover Loads (on page 881) for details on using this feature.

Note: Refer to the example files `ExampPush01.std`, `ExampPush02.std`, `ExampPush03.std`, and `ExampPush04.std` in the folder `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Pushover\`.

AD.2006.3.2 Steady State Analysis

Please refer to TR.37.6 Steady State and Harmonic Analysis (on page 2632) for details.

Note: Also refer to the example files `SVM33.std`, `SVM32.std`, `SS-beam2.std`, `SS-beam3.std`, `Exam07.std`, and `Exam14b.std` in the folder `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\SteadyState\`.

AD.2006.4 Features affecting the RC Designer Mode

Several new features have been added and existing features have been modified in the RC Designer mode of the program. They are explained in the following pages.

Refer to the RC Designer Manual for full details. Note that references made here are for sections in the RC Designer Manual.

AD.2006.4.1 Slab Design to BS 8110

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.4.2 BAEL Beam Design Enhancement

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.4.3 French GUI

Recent changes to the program's language facilities have depricated these changes.

AD.2006.4.4 DIN 1045-1 Beam and Column Design

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.4.5 Use of Generated Load Cases

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.4.6 Use of Primary Load Cases in Column Designs

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.4.7 BS8110 Beams Torsion Check Added

Note: The legacy Concrete Design workflow (included in V8i and earlier as the RC Designer application) has been deprecated in favor of the Advanced Concrete Design workflow using RCDC.

AD.2006.5 Additional STAAD.Pro 2006 Features

The additional enhancements made to STAAD.Pro 2006 are explained in the following pages.

AD.2006.5.1 STAAD.Pro Language Application

Recent changes to the program's language facilities have depricated these changes.

AD.2006.5.2 Section Wizard databases

Section Wizard Section Builder module and FreeSketch modules make use of defined standard steel databases. 3 new additional databases have been added and 3 have been updated.

New Additional Databases

INDIAN.PRF - Indian
JAPANESE.PRF - Japanese
SAFRICA.PRF - South African

Updated Databases

BRITISH1.PRF – UK Corus
DIN.PRF - German
USA.PRF - ASTM

Note, all other databases are unchanged.

AD.2006.6 Features Added in STAAD.Pro 2005 Previously Undocumented

The following pages explain features that have been introduced into STAAD.Pro but were not fully documented in previous Release Reports.

AD.2006.5.1 DESCON, Advanced Connection Design Mode

Recent changes to the program's language facilities have depricated these changes.

RAM Connection is now the supported facility for steel connection design in STAAD.Pro.

AD.2006.5.2 ADAPT Slab Design Mode

Recent changes to the program's language facilities have depricated these changes.

Refer to the User Interface help for use with the D. Advanced Slab Design (on page 1073).

Note: Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

AD.2006.6.3 BS5950 Part 5 - Cold Formed Steel Design

Refer to D3.E. British Codes - Design per British Cold Formed Steel Code (on page 1405) for details.

AD.2006.6.4 EC4 Timber Design

Refer to D5.E. European Codes - Timber Design Per EC 5: Part 1-1 (on page 1582) for details.

AD.2006.6.5 Canadian Timber Design

Refer to D4.D. Canadian Codes - Timber Design per CAN/CSA-086-01 (on page 1439) for details.

AD.2006.6.6 South African Steel Design

Refer to D14.B. South African Codes - Steel Design per SANS10162-1:1993 (on page 1954) for details.

AD.2006.6.7 South African Concrete Design

Refer to D14.A. South African Codes - Concrete Design per SABS-0100-1 (on page 1949) for details.

AD.2006.6.8 EC8 Earthquake Loading

Refer to TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2520) for details.

AD.2006.6.9 Additional Kingspan Cold Formed Steel Database

Refer to TR.20 Member Property Specification (on page 2255) for details.

AD.2006.6.10 Imperfection Analysis

Refer to TR.26.6 Member Imperfection Information (on page 2313) for details.

AD.2006.6.11 Tapered Steel Design Added to BS 5950

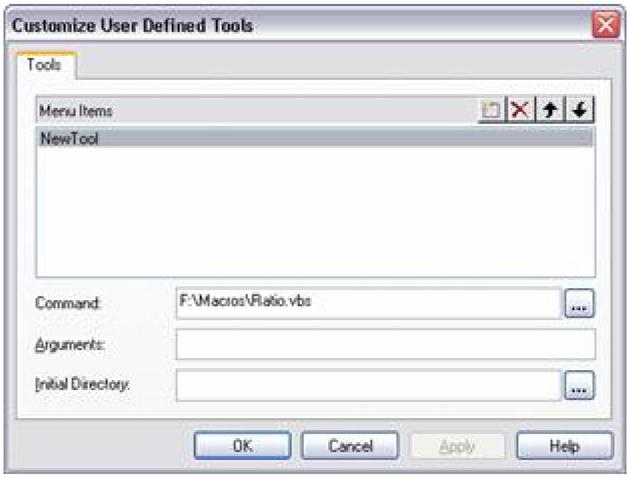
Refer to D3.B. British Codes - Steel Design per BS5950:2000 (on page 1378) for details.

AD.2006.6.12 User Tools

The users can utilize their own customized tools, in form of VBS Macro, to operate on STAAD.Pro models.

The procedure of associating a user tool in STAAD.Pro is furnished below:

1. Run STAAD.Pro and Open an input file.
2. Select **Tools** > **Configure User Tools**.
   The **Customzie User Defined Tools** dialog opens.



3. In the **Menu-Items**, add a name of the Tool, and in the **Command**, locate the Tool (VBS Macro) using the browser. Click **OK** to confirm.
4. Now the Tool name will appear in **Tools** > **User Tools** and also by selecting the Tools tool. 
5. To Run the Tool (Macro), click on the Tool name in any of the places as described in step 4.

Related Links

- *OS. To add the macro to the list of user tools* (on page 7040)

STAAD.Pro 2005 Release Report

This Software Release Report for STAAD.Pro Version 2005 contains detailed information on the additions and enhancements made to the program since the final build of STAAD.Pro 2004.

AD.2005.1 Features affecting the Pre-Processor

Several new features have been added and existing features have been modified in the pre-processor section of the program. They are explained in the following pages.

AD.2005.1.1 Generation of Wind Pressure profile per ASCE 7-02

STAAD.Pro is now capable of generating the wind pressure profile for a structure in accordance with the ASCE-7-02 code. The pressure profile is the table of values of wind intensity versus height above ground.

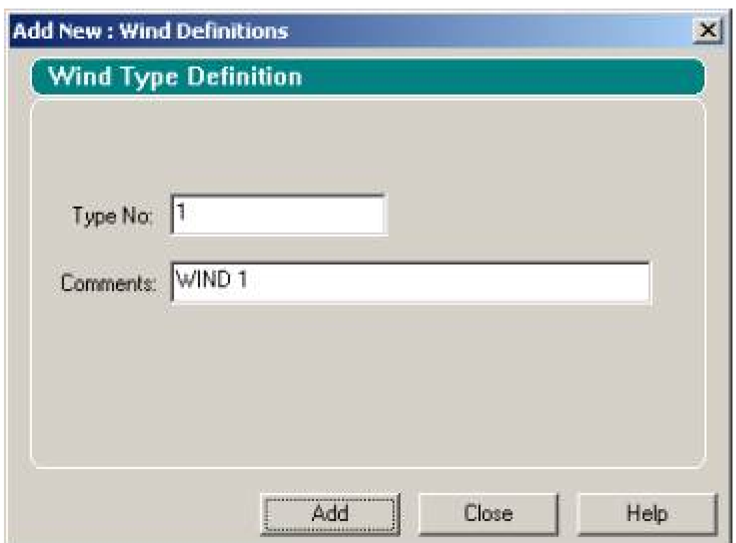
The calculated pressure may then be applied on the structure to compute loads on members using the program's built-in wind load generation algorithm for the closed as well as open-lattice type structures.

Description

The steps required to generate the wind pressure profile are as follows:

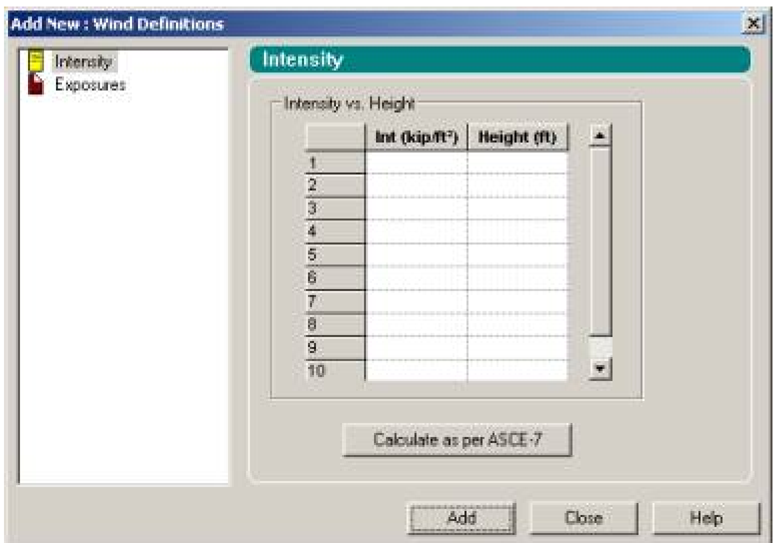
In the **General | Load** page (from the vertical tabs on the left-hand side), go to the Load dialog box on the right and select **Wind Definitions** as shown in the next figure.

Click on the **Add** button to instantiate the **Add New Wind Definitions** dialog box shown in the next figure. Enter the **Type No.** which denotes a number by which the wind load type will be identified. Multiple wind types can be created in the same model. Click on the **Add** button within this dialog box and then click on **Close**.



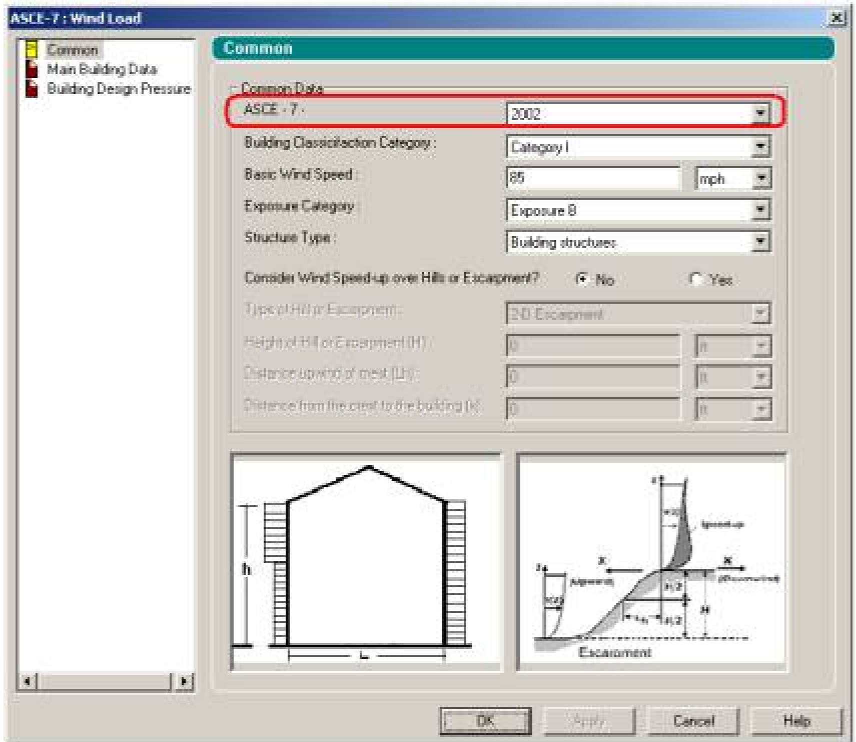
The newly created TYPE 1 wind definition will appear underneath Wind Definitions in the **Load** dialog box as shown below.

Select the TYPE 1 name in the tree control and click on the Add button. The dialog box shown below will prompt for the pressure profile (intensity) for this wind definition.



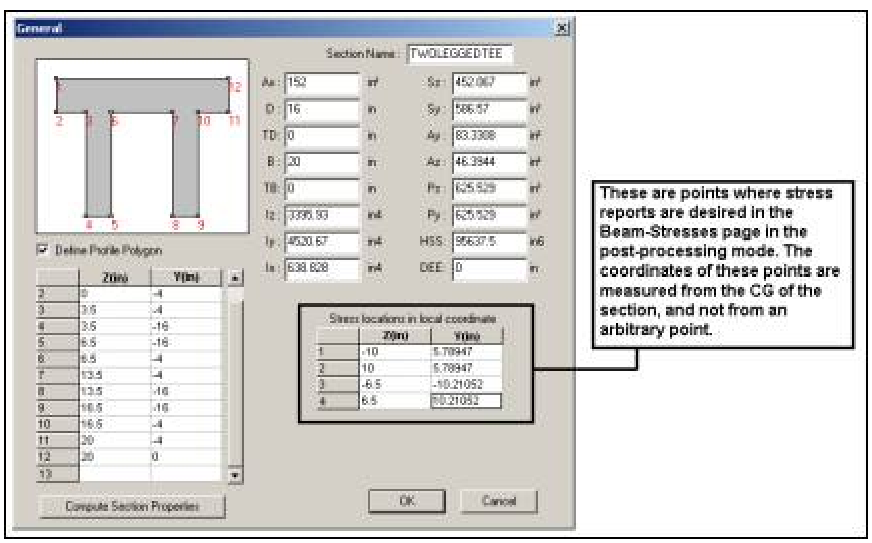
As we said earlier, the pressure profile is the table of wind intensity versus height above ground. If we know that, that information can be typed into the box above. But, our goal is to calculate that. Hence, we click on the button **Calculate as per ASCE-7**.

The ASCE-7: Wind Load dialog box shown below will appear.



The options shown in the dialog box are explained below:

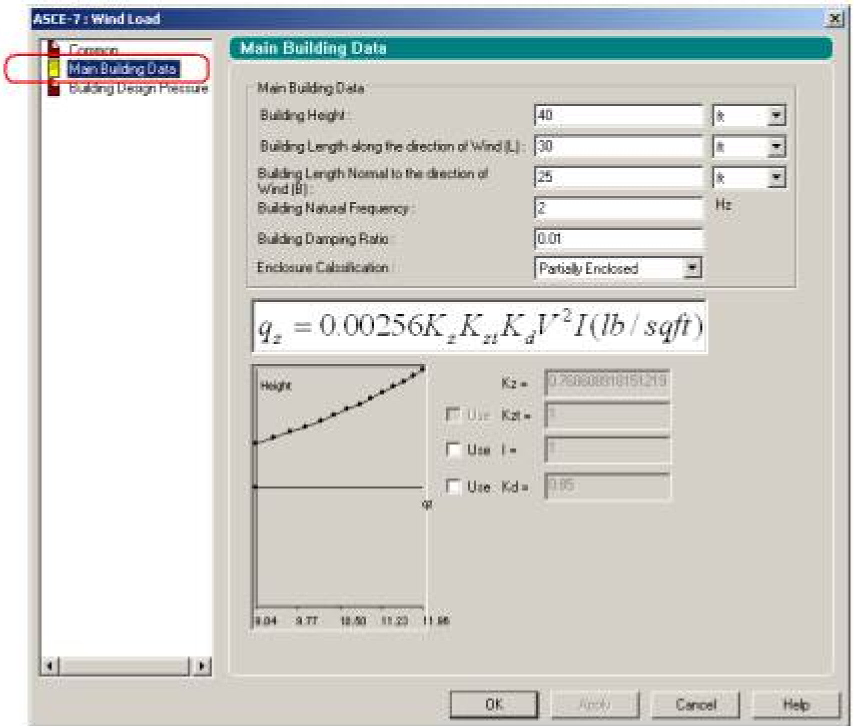
Common Data



ASCE - 7 -	Depending on which version of the code to use, choose either 1995 or 2002.
Building classification category	Building classification category as obtained from Table 1-1 in SEI/ASCE 7-02. Category can be I, II, III or IV
Basic Wind Speed	Basic Wind Speed as described in section 6.5.4 of the SEI/ASCE 7-02 code.

Exposure Category	Exposure category as described in section 6.5.6.3 of the SEI/ASCE 7-02 code.
Structure Type	Select the type of structure that best fits the model from the available choices: • Building structures • Chimneys, Tanks and similar structures • Solid Signs • Open Signs • Latticed Framework • Trussed tower The associated input pages will change depending on the selection of the type of structure.
Consider Wind speed-up over hill or escarpment	If there are isolated hills and escarpments that constitute abrupt changes in the general topography, the increase in speed can be considered as per section 6.5.7 in the SEI/ASCE 7-02 code. Select 'Yes' to consider wind speed-up over a hill or an escarpment and 'No' to ignore it.
Type of Hill or Escarpment	Select the type of hill, ridge or escarpment on which the structure is located, based on figure 6-4 of the SEI/ASCE 7-02 code. The options available are 2-D Escarpment, 2-D Ridge and 3-D Axisymmetric Hill.
Height of Hill or Escarpment (H)	Specify the height of the hill or escarpment relative to the upwind terrain (H in Figure 6-4 of the SEI /ASCE-7-02 code).
Distance upwind of crest (L_h)	Specify the distance upwind of crest to where the difference in general elevation is half the height of the hill or escarpment (L_h in Figure 6-4 of the SEI/ASCE-7-02 code).
Distance from the crest to the building (x)	Specify the distance from the crest to the building site. A negative value signifies that the distance is in the downwind direction (X in Figure 6-4 of the SEI/ ASCE-7-02 code).

Main Building Data



Building Height	The height above ground to the highest point on the roof surface.
Building Length along the direction of Wind (L)	Length of building measured along the direction of wind
Building Length Normal to the direction of Wind (B)	Length of building measured normal to the direction of wind
Building Natural Frequency	Specify the natural frequency of the building to calculate Gust Effect Factor.
Building Damping Ratio	Specify the damping ratio to calculate Gust Effect Factor.
Building Damping Ratio	Specify the damping ratio to calculate Gust Effect Factor.
Enclosure Classification	Classify the building as open building, partially enclosed or enclosed as per the provisions of section 6.2 of SEI/ASCE-7-02.
Kz	Velocity pressure exposure coefficient that is calculated by STAAD.Pro as per Table 6-3 of the SEI/ASCE-7-02 code.
Kzt	When wind speedup is considered, Kzt is calculated as per Eq 6-4 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and providing the required value in the edit box for Kzt.
I	Importance Factor I is considered as per section 6.2 of the SEI/ASCE-7-02 code. Users can enter their own value by checking the Use box and typing the required value in the edit box for I.

Kd	Wind directionality factor is calculated as per Table 6-4 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for Kd.

The list of parameters explained earlier corresponds to the structure type called “Building Structures”. When structure type is varied, some of those parameters change. Those parameters that are different are explained below for each type of structure.

Structure type: Chimney, Tank, and similar Structures

Height (H)	Height of the structure as defined by the term ‘h’ in Figure 6-19 of the SEI/ASCE 7-02 code.
Least Horizontal Dimension (W)	Smaller of the plan dimensions. In case the cross section of the structure in plan is circular, the diameter needs to be specified.
Horizontal Cross-Section Type	This is the cross section of the structure in plan as defined in Figure 6-19 of the SEI/ASCE 7-02 code. The available options include square with wind being normal to face or acting along the diagonal, hexagonal, octagonal and round.
Depth of protruding elements such as ribs and spoilers (D’)	For round type cross sections, depth of protruding elements need to be defined which is a measure of the surface roughness as indicated in Figure 6-19 of the SEI/ASCE 7-02 code.
C_f	Force coefficient that is calculated by STAAD.Pro as per Figure 6-19 of the SEI/ASCE 7-02 code. The parameter is used for calculation of design pressure. If desired, users can enter their own value for C_f by checking the **Use** box and typing the required value in the appropriate edit box.

Structure type: Solid Signs

Height (H)	Height of the structure which is used for calculating the height to width ratio as defined by the term ‘n’ in Figure 6-20 of the SEI/ASCE 7-02 code.
Horizontal Dimension of Sign (M)	Horizontal dimension of the solid sign
Vertical Dimension of Sign (N)	Vertical dimension of the solid sign. If the sign is at the ground level, the height (H) and vertical dimension (N) should both be specified the same value.
C_f	Force coefficient that is calculated by STAAD.Pro as per Figure 6-20 of the SEI/ASCE 7-02 code. The parameter is used for calculation of design pressure. If desired, users can enter their own value for C_f by checking the **Use** box and typing the required value in the appropriate edit box.

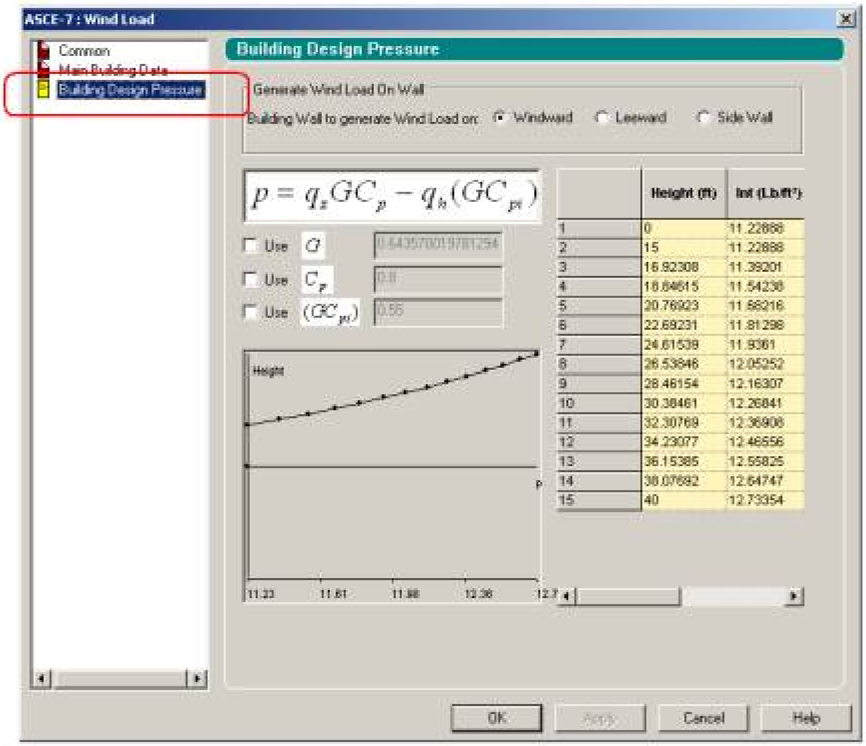
Structure type: Open Signs / Lattice Frame Work

Ratio of Solid Area to Gross Area	Ratio of solid area to gross area as indicated by the term e in Figure 6-21of the SEI/ASCE 7-02 code.
Orientation of the members exposed to wind	The type of member surfaces which are exposed to wind. Select flat-sided members or rounded members in Figure 6-21 of the SEI/ASCE 7-02 code.
Diameter of typical round member	Diameter for round members as defined by the term ‘D’ in Figure 6-21 of the SEI/ASCE 7-02 code.

Structure type: Trussed Tower

Horizontal Cross Section	The type of cross section of the tower in plan as defined in Figure 6-22 of the SEI/ASCE 7-02 code. The available options include square and triangle.

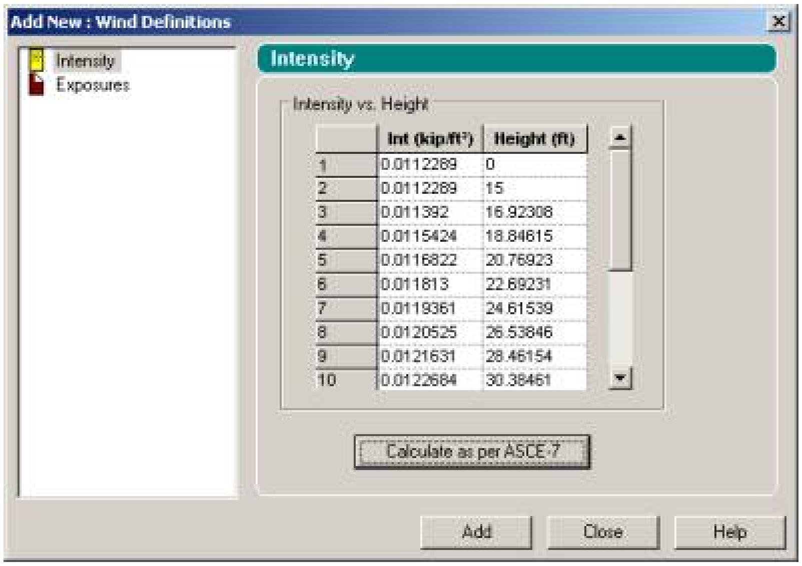
Building Design Pressure



Building Wall to generate Wind Load on:

Windward	To generate the design wind pressure for the windward side, choose this option. The pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation is also displayed in the box just beneath the radio buttons.
Leeward	To generate the design wind pressure for the leeward side, choose this option. The pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation will be displayed in the box just beneath the radio buttons.
Side Wall	To generate the design wind pressure for the side wall, choose this option. As before, the pressure will be calculated as per Equation 6-23 of the SEI/ASCE-7-02 code. The relevant equation will be displayed in the box just beneath the radio buttons.
G	Gust effect factor calculated as per section 6.5.8 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for G.
Cp	External pressure coefficient. The product of external pressure coefficient & gust effect factor is considered as per Figures 6-11 through 6-17 of the SEI/ASCE-7-02 code. Users can modify the value of Cp by checking the Use box and typing the required value in the edit box for Cp.
GCpi	Internal pressure Coefficients as per Figure 6-5 of the SEI/ASCE-7-02 code. Users can modify the value by checking the Use box and typing the required value in the edit box for GCpi.

The table on the right displays the intensities at various heights. Click on OK to arrive at the dialog box shown below.



Click on **Add** to add the load definition.

The defined wind loading can then be applied to the structure following the same procedure as in prior versions of the program. For details on the command syntax for generation of wind loads, refer to section 5.32.12: Generation of Loads from the STAAD Technical Reference Manual and example problem 15 in the Examples manual.

Note: The option we encountered earlier regarding the windward side, leeward side and side walls tells us that the pressure profile for each of those has to be individually determined under a unique type number. Thus, generating the profile for the 3 sides of the building constitutes 3 separate steps and thus, 3 separate types. Each type can then be applied with one load case or separate load cases and then applied in the relevant direction with the appropriate direction factor. Examples illustrating wind load generation can be found in the examples manual.

Related Links

- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Wind Load tab* (on page 2850)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *ASCE 7 Wind Load dialog box* (on page 2882)
- *M. To add an ASCE 7 wind load definition* (on page 845)

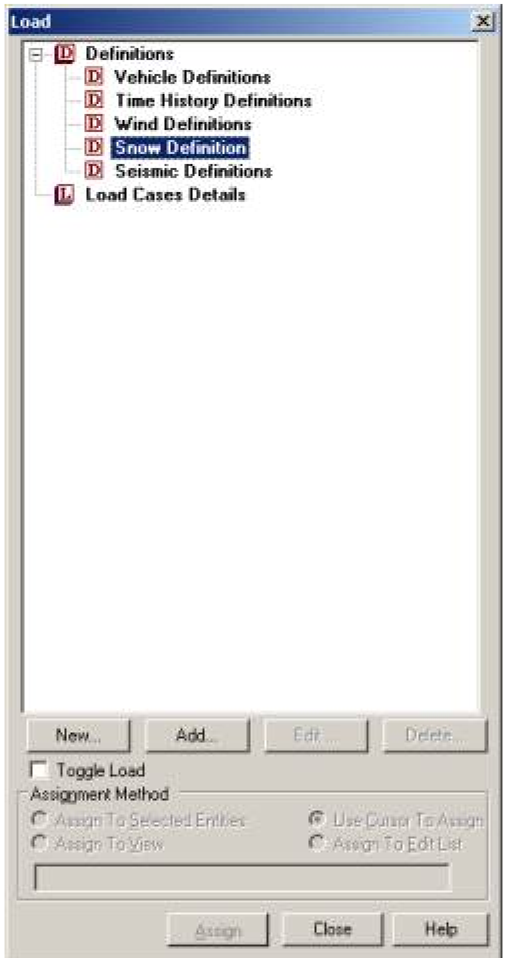
AD.2005.1.2 Generation of Snow Load Per SEI-ASCE 7-02

STAAD.Pro is now capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

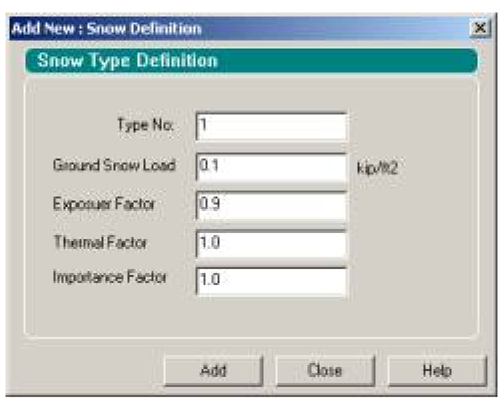
There are two parts to the command specification for snow load generation.

Part I

In this step, we define the parameters. In the **General | Load** page (from the vertical tabs on the left-hand side), go to the **Load** dialog box on the right and select **Snow Definition** as shown in the next figure.



Click on the **Add** button to instantiate the **Add New: Snow Definition** dialog box shown below. The **Type No.** is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model.

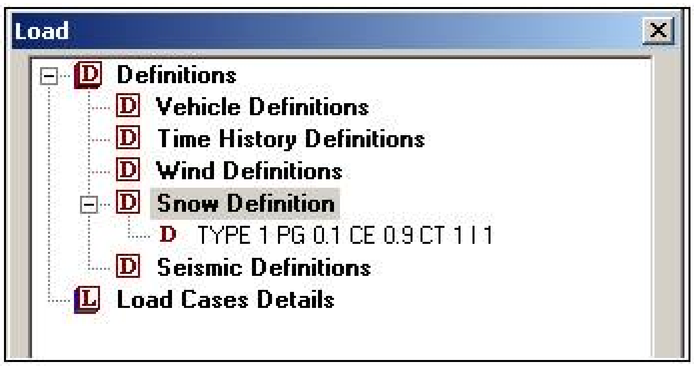


The fields in the above dialog box are:

Ground Snow Load	The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.
Exposure Factor	Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.
Thermal Factor	Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.
Importance Factor	Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Click on the Add button within the dialog box and then click on Close.

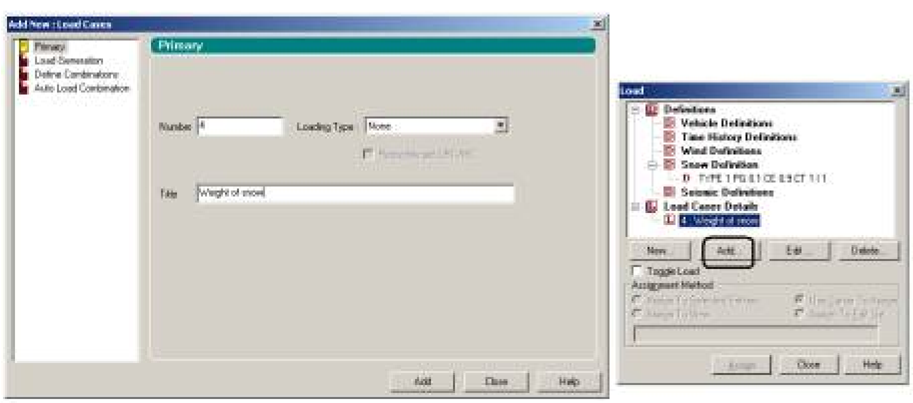
The newly created data will appear underneath Snow Definition in the Load dialog box as shown below.



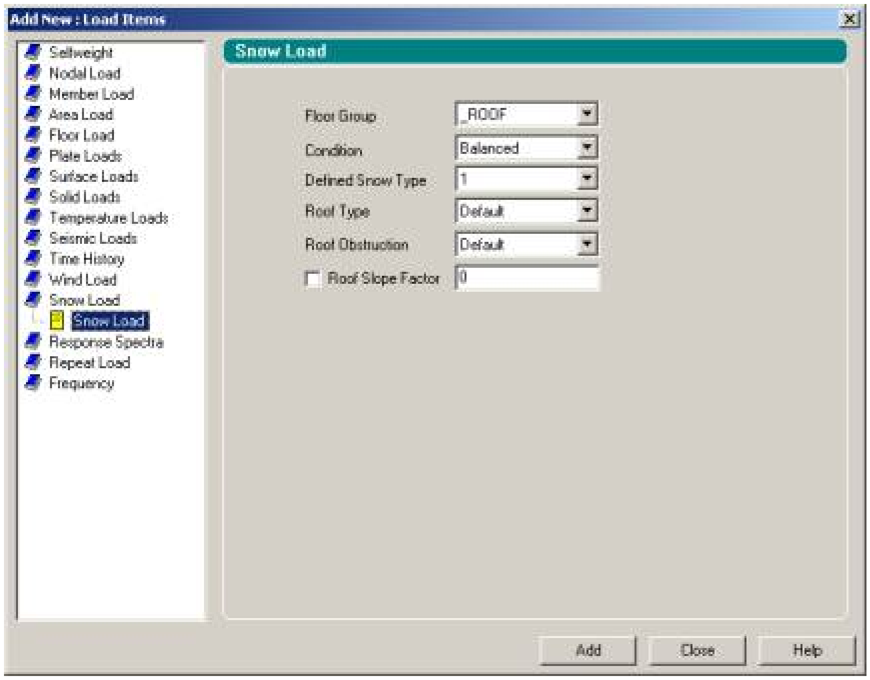
The members for which the snow load has to be generated have to be clustered together into FLOOR GROUPs. That is because, as we saw at the beginning of this section, the load generation is based on panel areas, not the exposed width of individual members. For details on creation of groups, refer to section 5.16 of the STAAD.Pro Technical Reference manual, and AD.2004.11 of the Software Release report for STAAD.Pro 2004.

Part II

Create a new primary load case. After the load case number and load title have been specified, click on Add.



Select it in the Load dialog box and click on the Add button to bring up the Add New: Load Items dialog box as shown below.



Select **Snow load** from the available list of load types. Various options corresponding to data required for snow load generation will appear on the right hand side, as shown in the figure above. The options are explained as follows:

Floor Group Select the floor group on which the snow load is to be applied.

Condition Specify whether the load is “” or “Unbalanced”. These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.

Define Snow Type Select the snow load type number. This is the number specified against the **Type No.** field while creating the snow load definition in Part I described earlier in this section.

Roof Type Specify the roof type from the available choices:

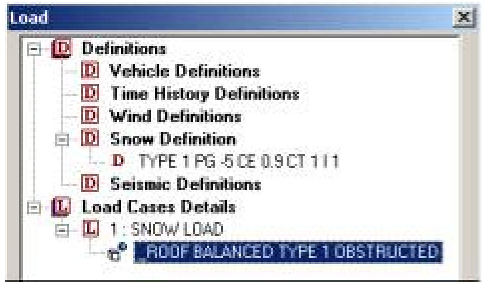
- Default (if the roof type is not Mono, Hipped or Gable, it is referred to as Default)
- Mono (see mono-sloped roof shown in figure 6-6 of the code)
- Hipped (see figures 6-3 and 6-6 of the code)
- Gable (see figures 6-3 and 6-6 of the code)

These choices are described in section … of ASCE 7-02.

Roof Obstruction Specify whether the roof is “obstructed” or “unobstructed”. This also is a term described in section … of ASCE 7-02.

Roof Slope Factor For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

Click on **Add** to complete the data input for Part II. The snow load command will appear underneath the snow load case in the **Load** dialog box as shown below.



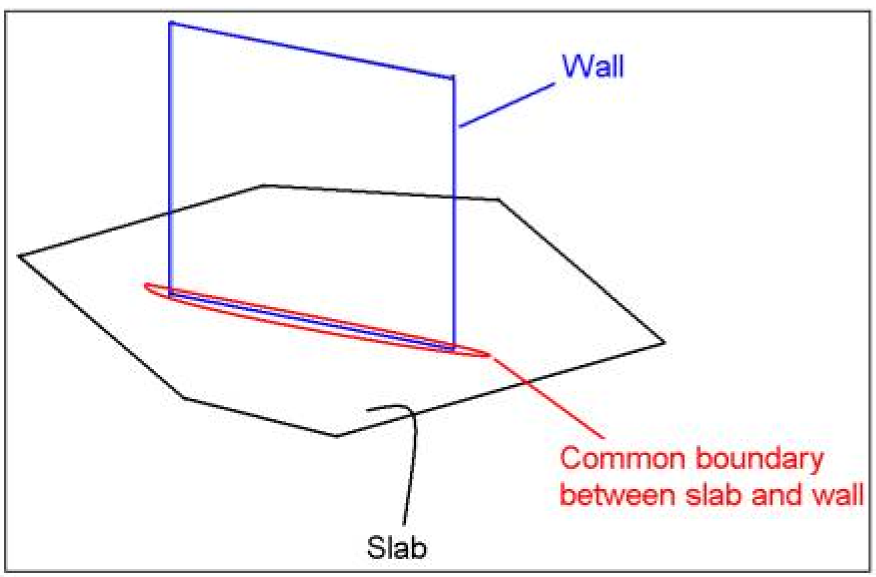
Related Links

- *Add New Snow Definition dialog* (on page 2894)
- *TR.31.5 Definition of Snow Load* (on page 2452)
- *M. To add an ASCE 7-02 snow load* (on page 870)
- *Snow Load tab* (on page 2854)
- *TR.32.12.4 Generation of Snow Loads* (on page 2609)
- *M. To add an ASCE 7-02 snow load* (on page 870)

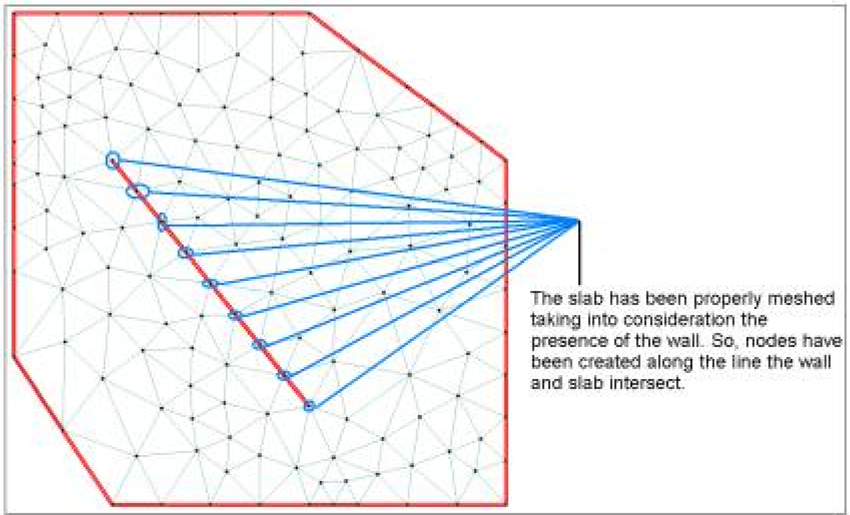
AD.2005.1.3 Wall-slab Interface Considerations in Finite Element Meshing

In the plate element mesh generation for a panel type of entity like a wall or slab, STAAD.Pro now provides a facility for the consideration of boundary conditions at the interface of the panel and any other panel on whose surface one of its edge lies.

To illustrate the problem that one faces if the facility did not exist, consider the wall and slab shown in the figure below:



The Parametric Models facility may be used to mesh the slab, and use the wall boundary as a density line, resulting in the following mesh for the slab.



If the wall is to be meshed subsequently, the nodes along its common boundary with the slab, as shown in the above figure, must automatically be "control points" in the wall meshing process. There was no simple way to do this until now.

Related Links

- *M. To define a slab/wall connection* (on page 675)

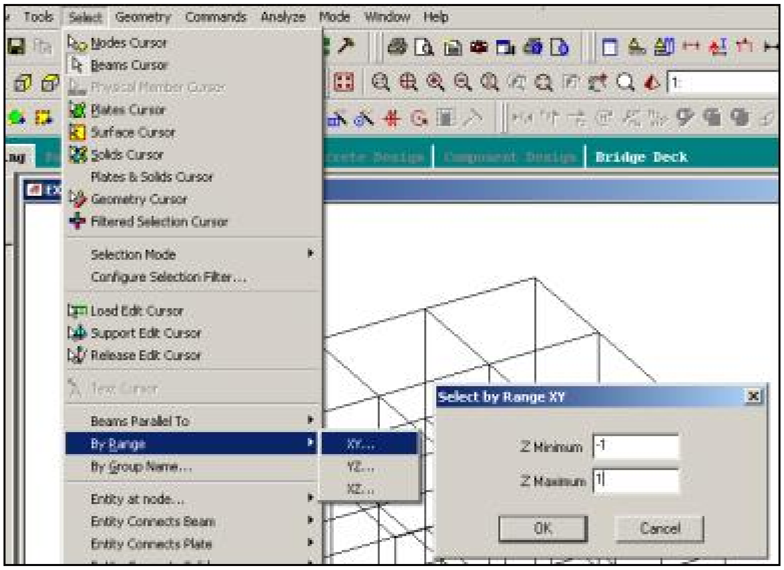
AD.2005.1.4 Enhancements to Renumbering of Entities

The capabilities of the program in renumbering entities such as joints, members, plates, etc. have been enhanced. In the past, only a limited amount of control was available in the manner in which renumbering was to be done. The current enhancements enable a user to set multiple criteria and assign them an order of priority.

Description

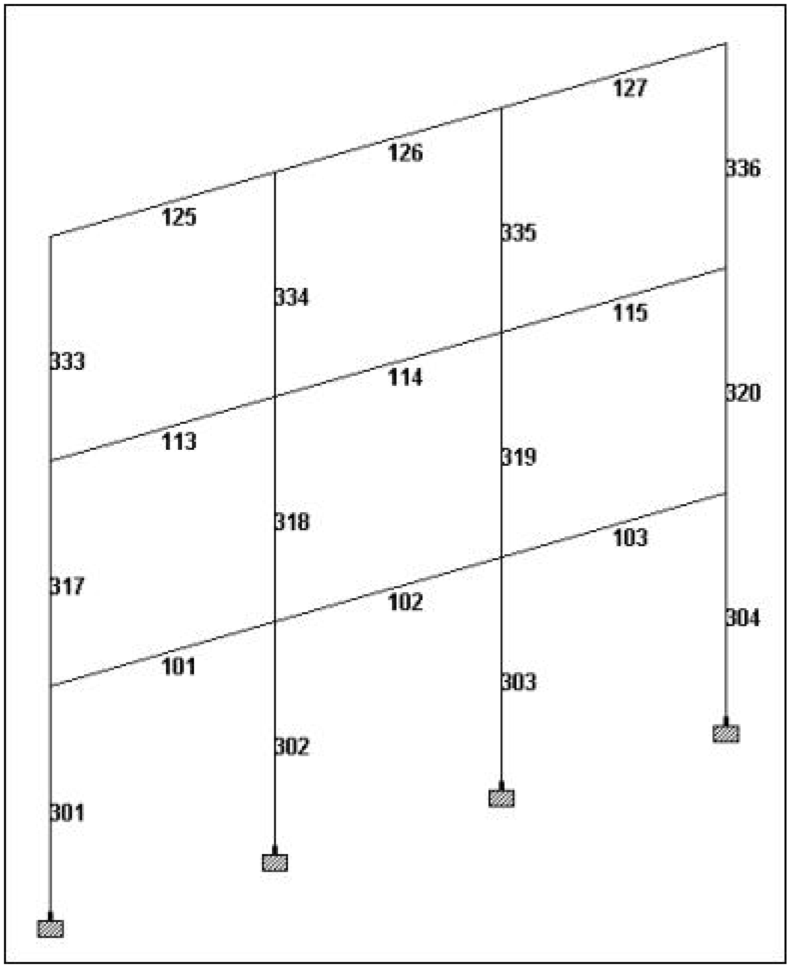
We will use example problem 14 in the US examples folder to illustrate this. We will renumber the columns of one frame of this multi-story building. If you do not wish to modify the file supplied with the program, you may make a copy of the file before you start this exercise.

After opening the model, go to **Select** > **By Range** > **XY** > , and set the **minimum** and **maximum** Z coordinate values to `-1` ft and `1` ft respectively.

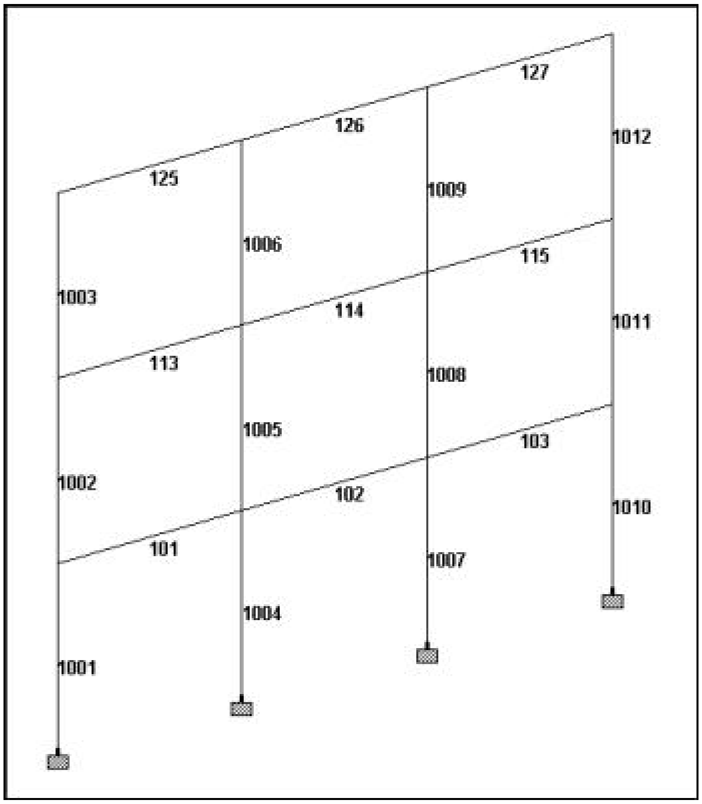


The members at the rear of the structure will be highlighted. Select **View** > **New View** > **Display the view in the active window** > .

Switch on the beam numbers and we get the following.

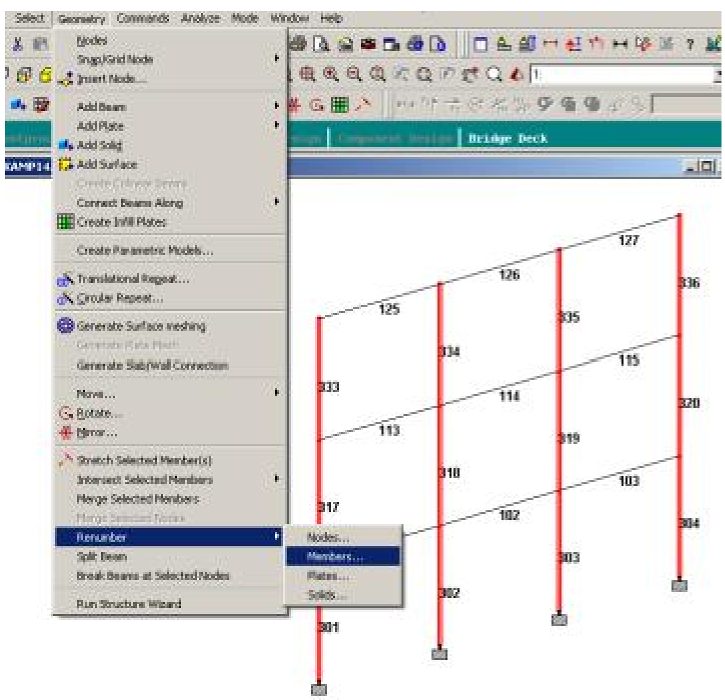


Our goal is to renumber the columns in such a manner that we will have the following once the renumbering is complete.

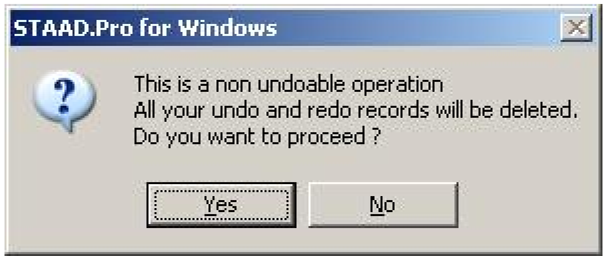


To accomplish that, do the following:

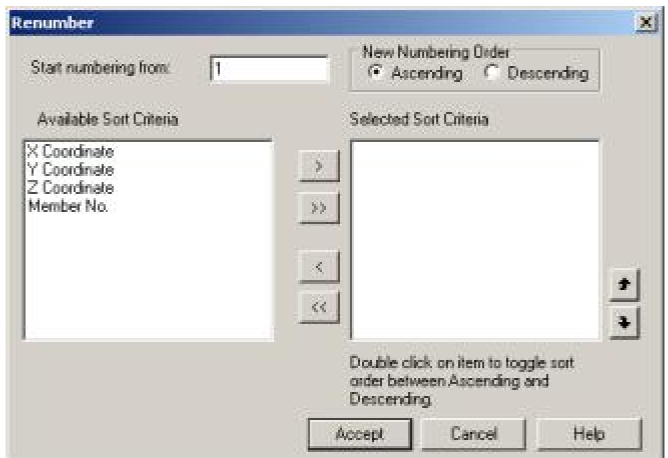
Select **Select** > **Beams Parallel to** > **Y**. The columns will be highlighted. Select **Geometry** > **Renumber** > **Members**.



The following message will be displayed.



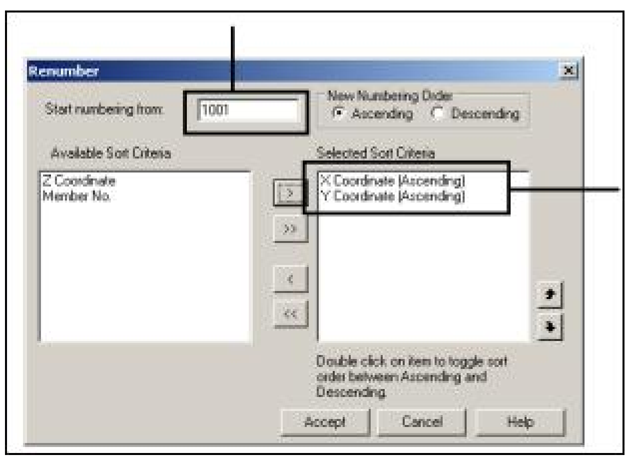
Choose **Yes**, and the **Renumber** dialog box will appear.



It is apparent from the earlier figure that if we set the renumbering criteria to be based on X coordinate first and Y coordinate next, the program will internally create the following sequence for renumbering:

301, 317, 333, 302, 318, 334, 303, 319, 335, 304, 320, 336

So, the settings for the dialog box should be as follows:



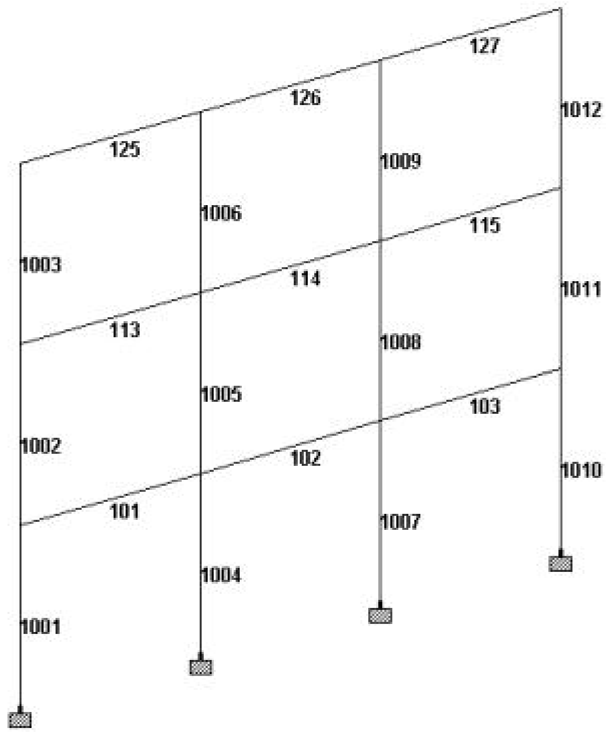
Start numbering from = `1001`

Select Sort Criteria = **X Coordinate (Ascending)**, **Y Coordinate (Ascending)**

Click on **Accept**.

A message indicating the successful completion of the operation will appear.

Subsequently, the following view will appear on the drawing.



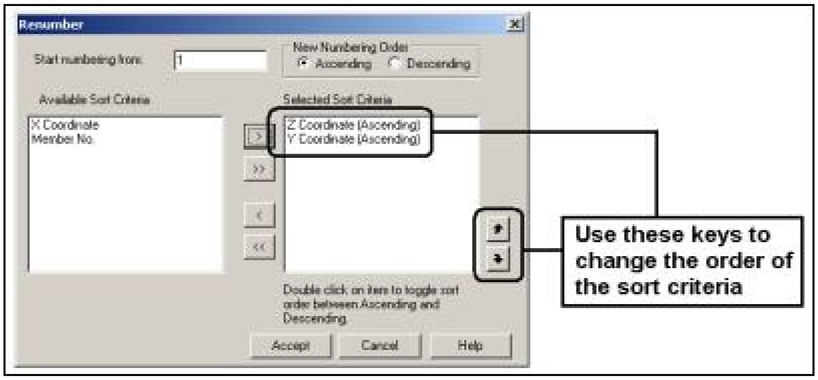
In past versions, 4 separate renumbering operations, each involving one line of columns, would have been required to achieve the same thing.

It is also worth noting that if the user wishes to renumber all the columns of all the frames in sequence, the Z coordinate can be included as a third criteria for sorting.

Notes

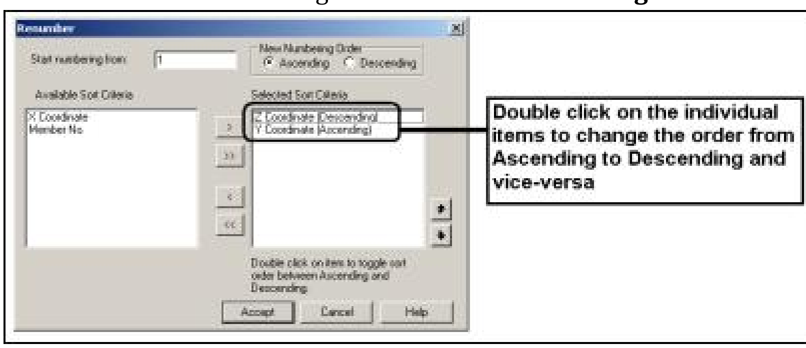
1. The order of importance of the sort criteria can be changed by clicking on the Up and the Down arrow keys as shown below.

Use these keys to change the order of the sort criteria



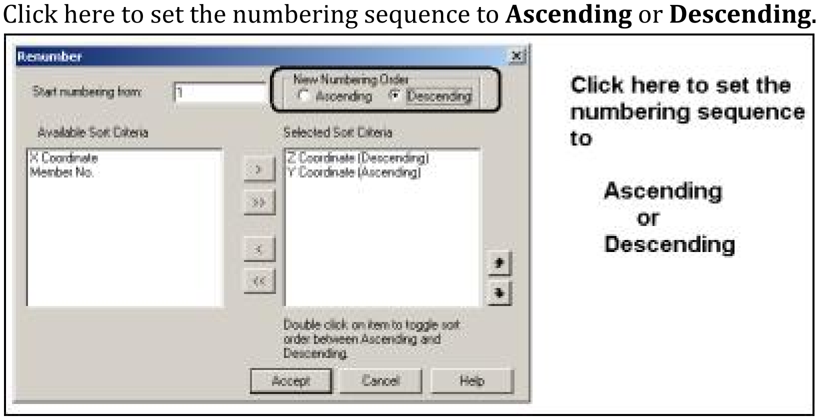
2. For the individual criteria, if you wish to change the order from Ascending to Descending or vice versa, double click on the item in the list as shown. For example, if you want the extreme right columns to have a lower number than the extreme left, set the criteria for X coordinate to Descending.

   Double-click on the individual items to change the order from **Ascending** to **Descending** and vice-versa.



3. If you want the entity to be numbered in the descending order instead of ascending order (as in 99, 98, 97, 96, …. , etc.), choose the appropriate button shown in the next figure.

   Click here to set the numbering sequence to **Ascending** or **Descending**.



Related Links

- *Renumber dialog* (on page 2728)
- *M. To renumber selected beams* (on page 663)

AD.2005.1.5 Property Calculator for User Table General Sections

The user table section type called General has been enhanced in the following manner:

a. The shape of the section can be defined in terms of coordinates of the corner points of its cross section outline

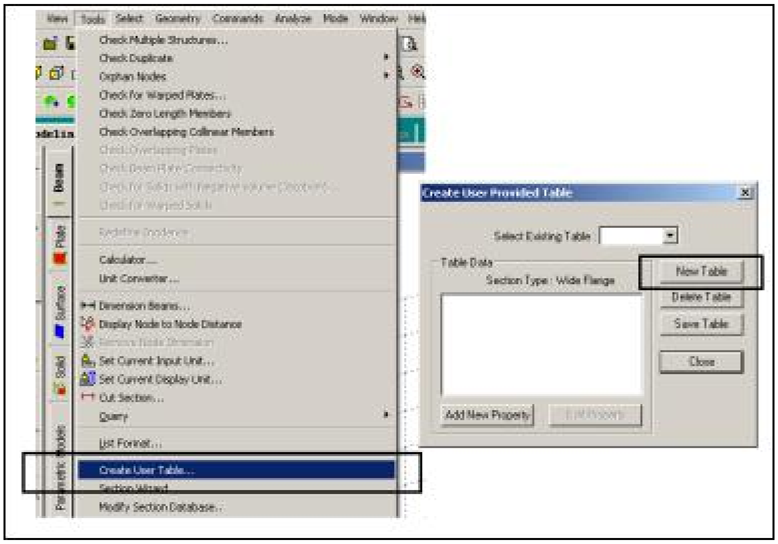
b. A section property calculator built into that dialog box enables the values to be calculated instantaneously. Any values the user wishes to override can be changed by typing over them.

c. Points can be defined at which stresses are to be reported in the Beam-Stresses page of the post-processing mode.

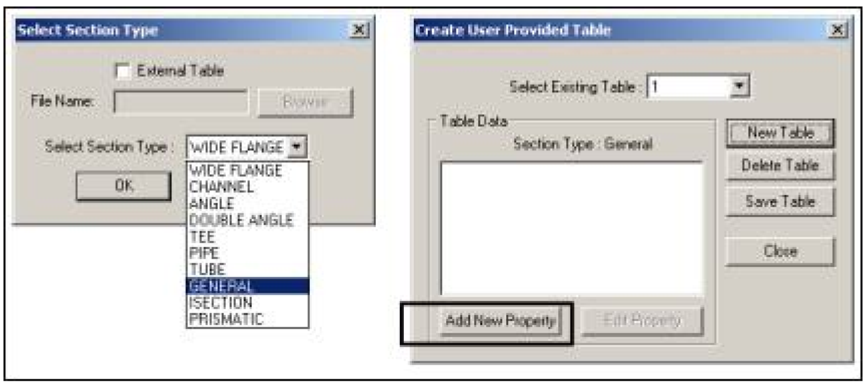
Description

Select **Tools** > **Set Current Input Units** > , and choose the length unit in which to specify the dimensions of the cross section. Let us choose inches for this exercise.

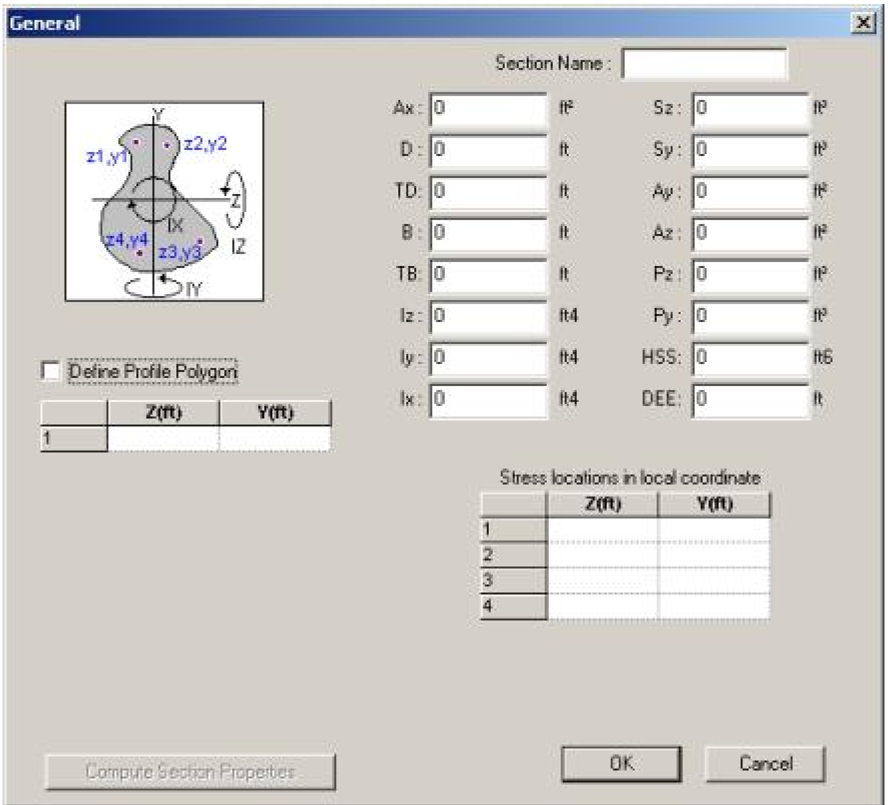
Next, select **Tools** > **Create User Table.** In the **Create User Provided Table** dialog box, click on **New Table.**



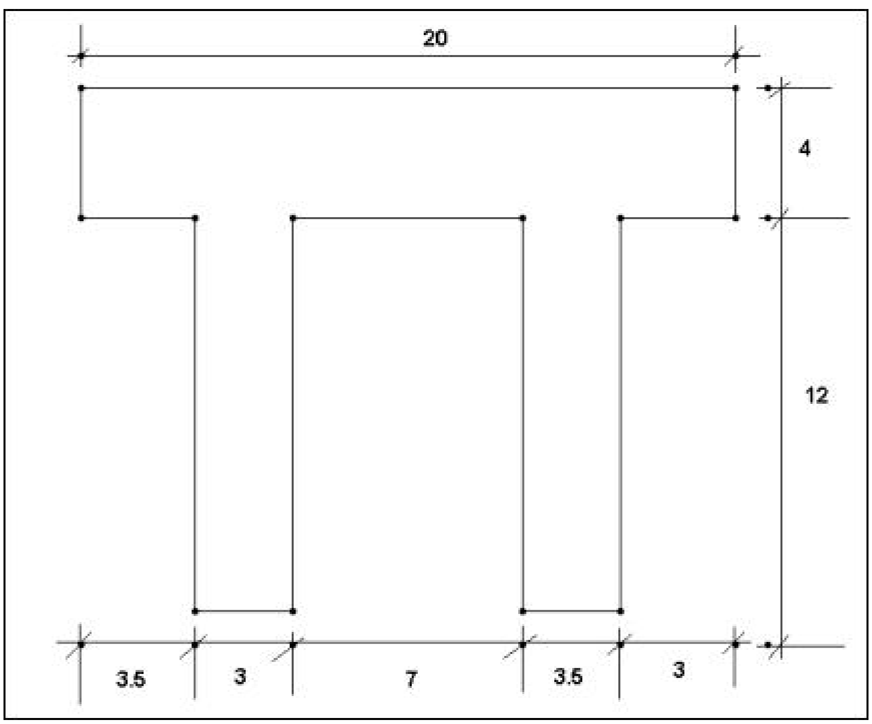
Select **General** for the **Select Section Type.** Click on **OK.**



The table number will appear as 1. Click on **Add New Property.** The following dialog box will appear.



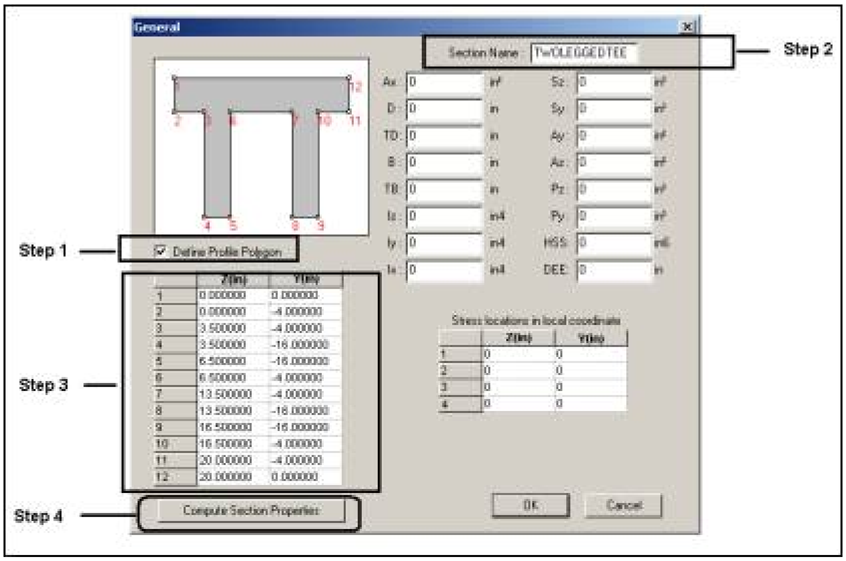
Let us say that we would like to define a cross section as shown below.



Assuming the top left corner of the cross section to be (0,0) for (Y,Z), , Z being along left-right and Y along top-bottom, specify the data as shown in the next figure. The sequence of steps is circled in the next figure. Once the points are specified, click on Compute Section Properties and observe the values being computed and filled in the relevant boxes (step 4 in the next diagram)

Steps

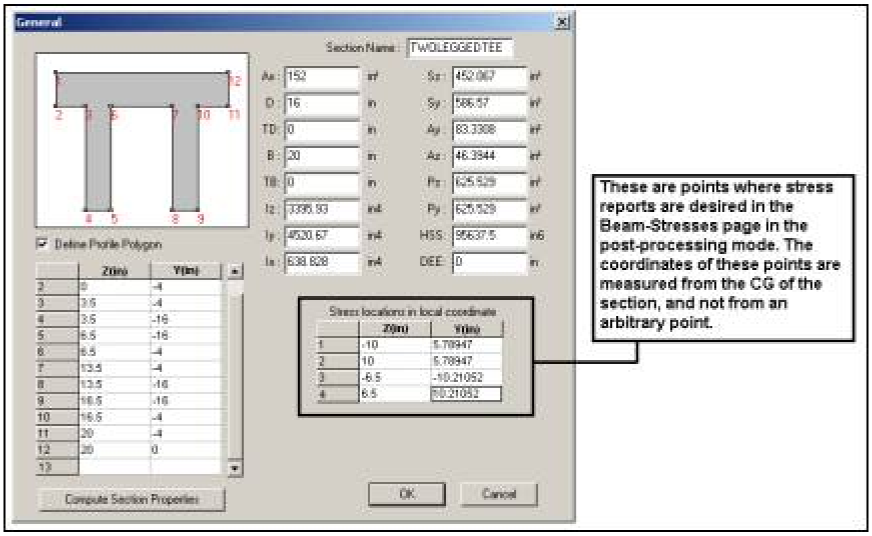
1.	Select **Define Profile Polygon**
2.	Type `TWOLEGGEDTEE` in the **Section Name** field.
3.	Type the vertex coordinates in the table.
4.	Click **Compute Section Properties**.



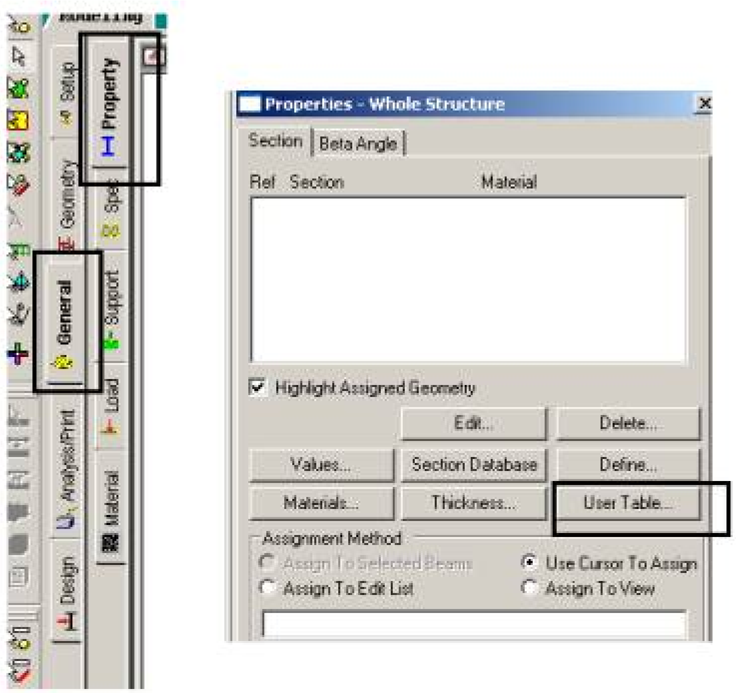
You will observe that when we clicked on Compute Section Properties, the Y and Z coordinates of the corner points that we typed earlier will be replaced by values that are measured from the center of gravity of the section.

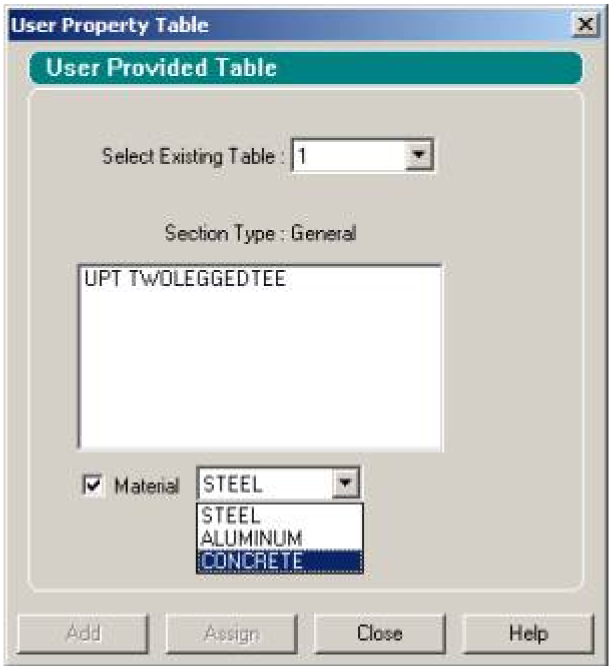
If you wish to specify the locations on the cross section where stresses are desired, they must be provided too. Note that the coordinate locations for that box should be based not on the arbitrary datum point we started out with initially, but on the basis of the center of gravity of the section.

These are points where stress reports are desired in the **Beam | Stresses** page in the post-processing mode. The coordinates of these points are measured from the CG of the section and not from an arbitrary point.



Once the section is defined, it may be assigned from the General-Properties page in the conventional way. Some of those steps are shown in the next 2 figures.





For the case of the example above, the following command syntax will appear in the STAAD input file.

```
START USER TABLE
TABLE 1
UNIT INCHES KIP
GENERAL
TWOLEGGEDTEE
152 16 20 0 3395.93 4520.67 638.828 452.067 586.57 83.3308 -
46.3944 625.529 625.529 95637 .5 0
PROFILE POINTS
0 0 0 -4 3.5 -4 3.5 -16 6.5 -16 6.5 -4 13.5 -4 13.5 -16 16.5 -16 16.5 -4 20 -4 20 0
STRESS LOCATIONS
-10 5.78947 10 5.78947 -6.5 -10.2105 6.5 10.2105
END
```

Related Links

- *M. To create a general section* (on page 743)

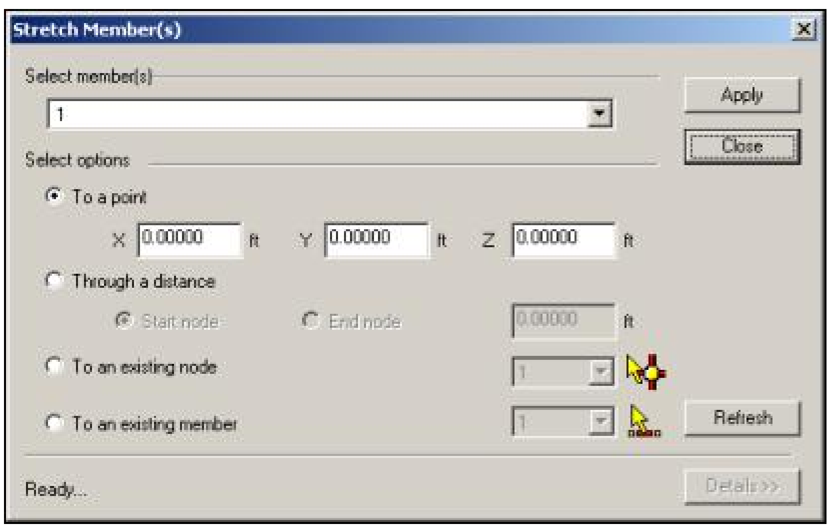
AD.2005.1.6 Stretch Members

This new facility allows one to increase the length of a member in various ways. The benefit it offers is that a member may be extended even if one does not readily know the coordinates of its ends joints in its stretched condition, something a user may appreciate in the case of members whose axis lie at an inclination to the global planes.

Description

Select the member which is to be extended. In the **Geometry** menu at the top of the screen, the option called **Stretch Selected Member(s)** becomes active.

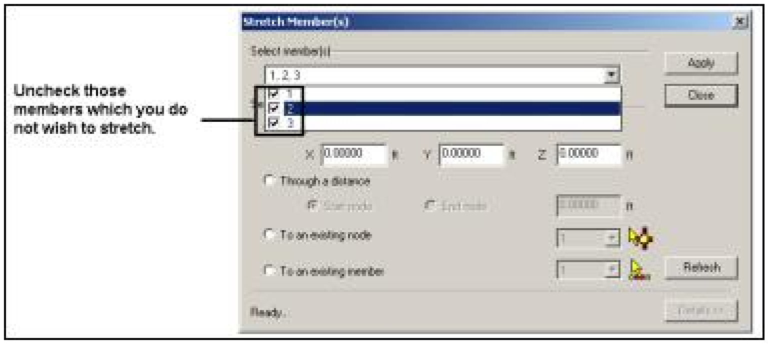
Click on it and the following dialog box will appear.



The various options of the dialog box are explained below.

Select member(s) Multiple members can be selected simultaneously for stretching. However, whether they will get stretched or not depends upon the type of method used in stretching as described later. If you wish to remove one or more members from the list, uncheck the corresponding boxes.

Uncheck those members which you do not wish to stretch.



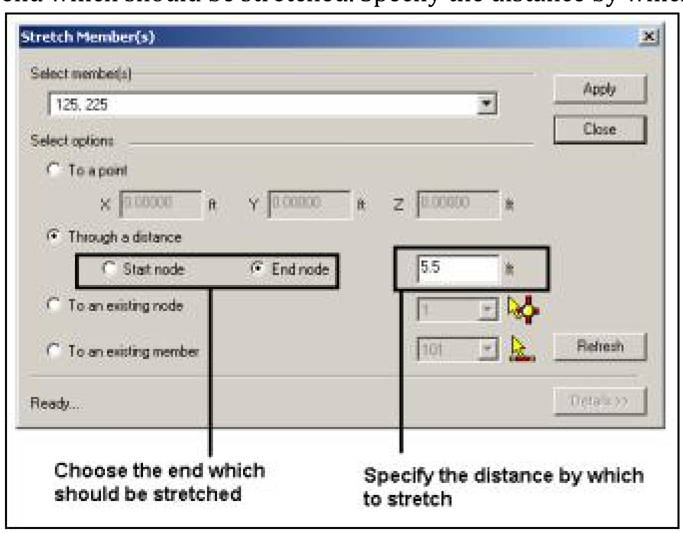
The methods available for stretching are:

To a point Specify the coordinates in current length units of the point to which one of the ends of the member are to be moved. The point must lie on the axis of the member being stretched, or else, the stretching will not be performed. The program automatically determines which of the two ends of the member is to be moved. So, this method involves

- **i.** determining if the point lies on the axis of the member(s) being stretched.
- **ii.** determining which end to move for the member(s) which satisfy criteria (i)
- **iii.** replacing the corresponding nodal coordinates with those of the desired point.

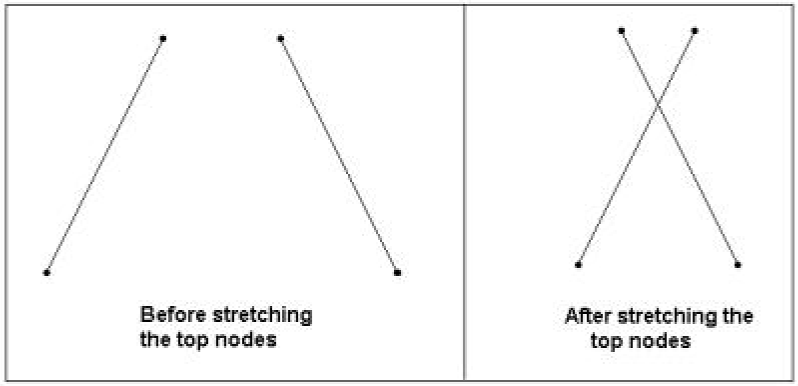
Through a distance In this method, the user has to merely specify the distance by which the start or end node must be stretched.

Choose the end which should be stretched. Specify the distance by which to stretch.



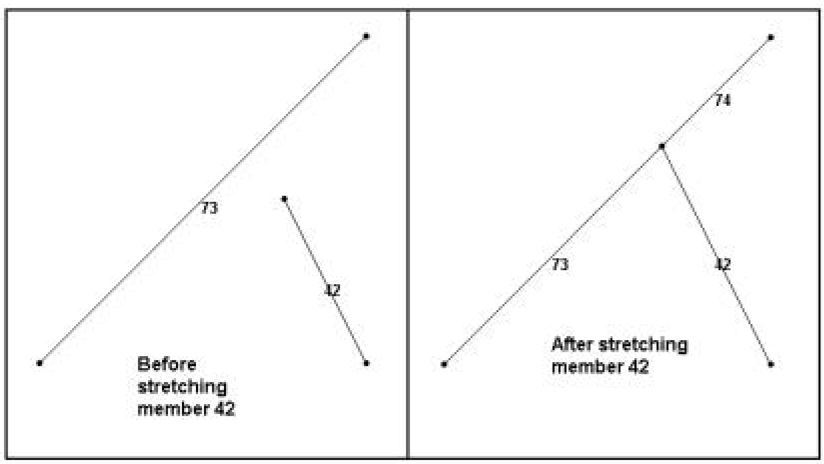
In this method, all members selected for this operation will see the change in length. The next diagram illustrates this. If this causes member to cross each other, the user must create the intersection point using the Geometry – Intersect Selected Members tool as the program does not automatically create it in the Stretch operation.

Before stretching the top node (left). After stretching the top nodes (right).

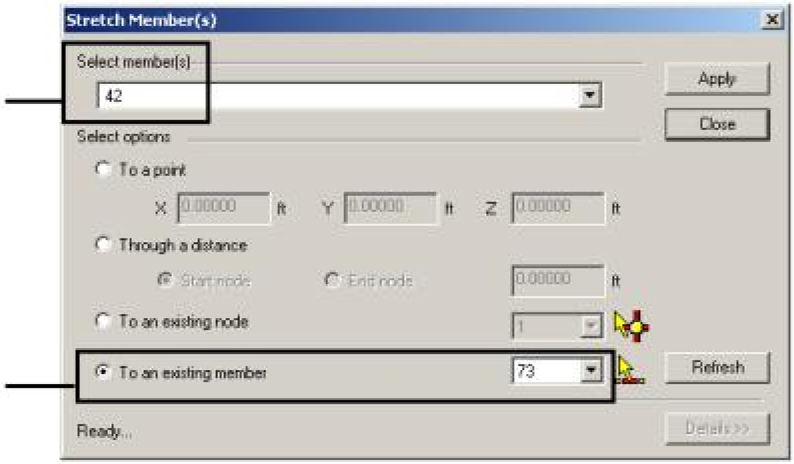


To an existing node This is very similar to the **To a point** method described earlier. The only difference is that instead of explicitly specifying the coordinates of the desired point, that point is already available for identification through its node number.

To an existing member If 2 members are oriented in such a manner that the local X-axis of one of those members can potentially intersect a second member within the span of that member, this method may be used to stretch the first member to meet the second member. In the next figure, member 42 may be stretched so it meets member 73. The second member will be automatically split up at the intersection point into two segments.



The dialog box settings required to achieve this are as follows:



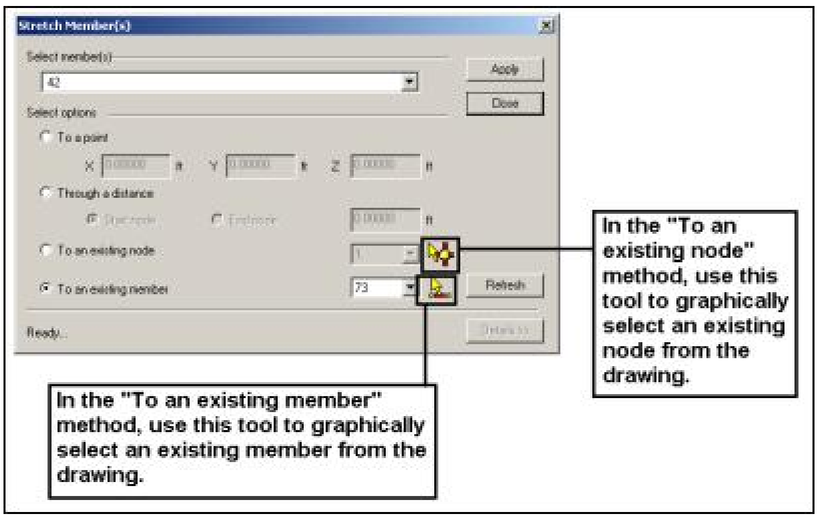
Notes

In the **To an existing node** method, a graphical tool is available for selecting the node from the drawing as shown in the next figure.

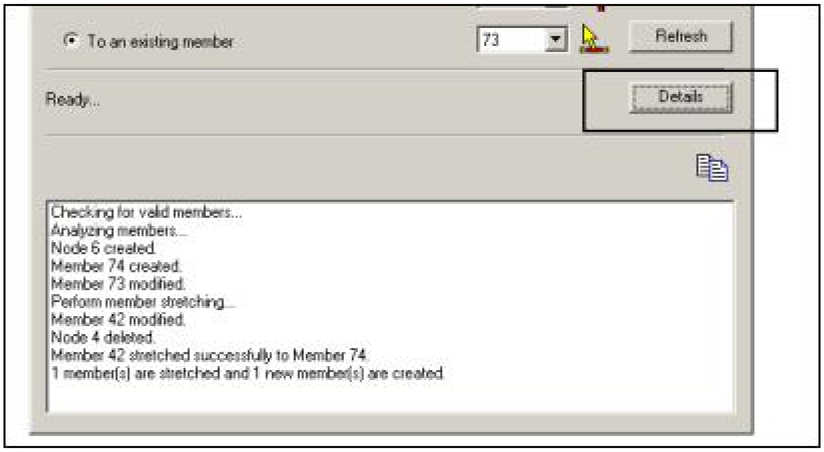
In the **To an existing member** method, a graphical tool is available for selecting the member from the drawing as shown in the next figure.

In the **To an existing node** method, use this tool graphically select an existing node from the model.

In the **To an existing member** method, use this tool graphically select an existing node from the model.



The **Details** button may be switched on to view the internal details of the stretch operation.



Related Links

- *Stretch Member(s) dialog* (on page 2732)
- *M. To stretch a member* (on page 662)

AD.2005.2 Features Affecting the Post-Processor

The additions and enhancements to the post-processor section of the program are explained in the following pages.

AD.2005.2.1 Floor Vibration Analysis

The adequacy of a floor system from the standpoint of its vibration serviceability due to human activity, specifically walking excitation, can now be assessed using STAAD.Pro. The procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11 - Floor Vibrations due to Human Activity - have been implemented.

Related Links

- *Floor Vibration Output dialog* (on page 3002)
- *P. To generate a floor vibration report* (on page 1996)

AD.2005.3 Features Affecting Analysis and Design

Some new features have been added and existing features have been modified in the analysis and design part of the program. They are explained in the following pages.

AD.2005.3.1 Designing I-beams w Web Openings per AISC ASD

Design of steel members with web openings per AISC Steel Design Guide 2 - ASD specifications is now available in STAAD. The facility is available for members whose yield strength is 65 ksi or less.

Note: In the current implementation, the web openings are given consideration only during the design phase. The reduction in section properties caused by the presence of the openings is not considered automatically during the analysis phase. Hence, the analysis is performed as if the full section properties are effective for such members.

During the design process, the program first determines the utilization ratio (U.R.) at the location of the opening as though it is an unreinforced opening. If the U.R. is less than 1.0, the member is presumed to have passed the requirements at that location. If the U.R. exceeds 1.0, then it determines the U.R. as though it is a reinforced opening. If it fails this too, the cause of the failure along with the associated numerical values is reported.

3 Tutorials

T.1 – Steel Portal Frame

This chapter provides a step-by-step tutorial for creating a 2D portal frame using STAAD.Pro.

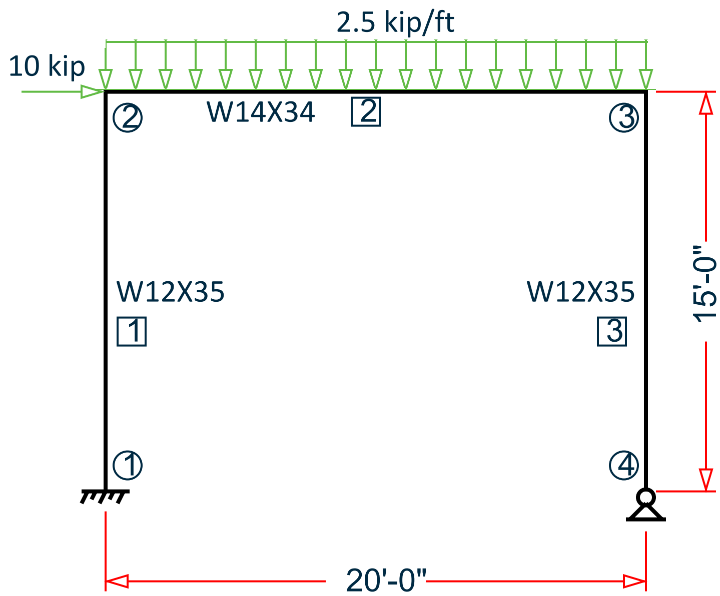


Figure 12: Portal Frame model

A copy of the completed input file is typically installed at `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Tutorials\Tutorial 1 - Steel Portal Frame.std`.

Related Links

- *EX.Tutorials* (on page 6255)

T.1 Methods of creating the model

There are three methods of creating the structure data:

1. T.1 Creating the Model using the Physical Modeler (on page 391),
2. T.1 Creating the model using the analytical user interface (on page 399),
3. T.1 Creating the model using the command file (on page 442)

The physical model is used to draw structural elements as they are physically constructed. The program will then decompose this into an analytical modeling which is passed to the STAAD.Pro analysis and design engine when you run your model.

The analytical model is a finite element model of the structure which is typically processed directly by the analysis and design engine.

The command input file is a text file which contains the data for your structural model. This file consists of simple English language-like commands. This file is created for you behind the scenes for you when you model your structure using either the physical modeler interface or the analytical model interface.

The physical model will be linked to the command input file so any changes to your physical model *must* be made using the physical modeler interface. This ensures data integrity when the program decomposes the physical model. You can “break” this link to make changes to the underlying analytical model directly, either using the STAAD.Pro analytical modeling interface or using the command input file editor.

You can switch back and forth between the analytical modeling interface and the command input file editor for models created by either of those methods.

All three methods are described in this tutorial, with a detailed focus on the STAAD.Pro Physical Modeler interface.

T.1 Description of the tutorial problem

The structure for this project is a single bay, single story steel portal frame that will be analyzed and designed.

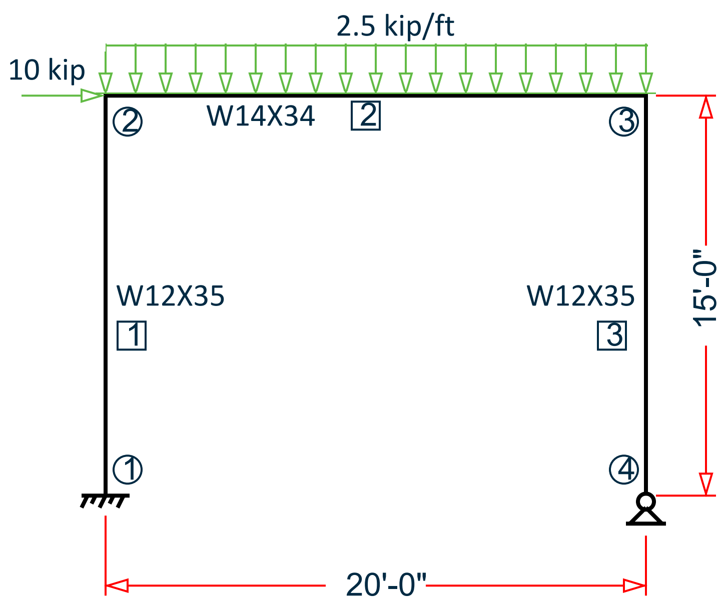


Figure 13: Portal Frame model

Table 22: Basic Data for the Structure

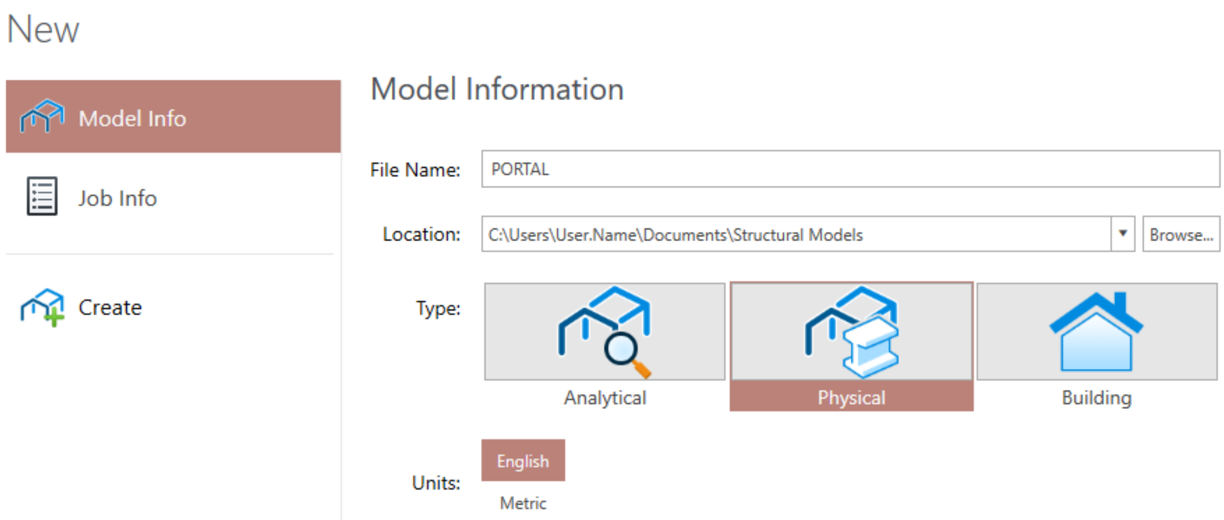
Attribute	Data
Member properties	Members 1 & 3 : W12X35 Member 2 : W14X34
Material Constants	Modulus of Elasticity : 29,000 ksi Poisson's Ratio : 0.30
Member Offsets	6.0 inches along global X for member 2 at both ends
Supports	Node 1 : Fixed Node 4 : Pinned
Loads	Load case 1 : Dead + Live Beam 2 : 2.5 kips/ft downward along global Y Load case 2 : Wind From Left 10 kips point force at Node 2 Load case 3 : 75 Percent of (DL+LL+WL) Load Combination - L1 X 0.75 + L2 X 0.75
Analysis Type	Linear Elastic (PERFORM)
Steel Design	Consider load cases 1 and 3 only. Parameters: Unsupported length of compression flange for bending : 10 ft for members 2 and 3, 15 ft for member 1. Steel Yield Stress : 50 ksi Perform member selection for members 2 and 3

T.1 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.
2. Type `PORTAL` in the **File Name** field.



3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.
4. Select **Physical** for the **Type** of model.

   This option selects the modeling method you want to use:

Analytical	For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
Physical	For creating a model using the STAAD.Pro Physical Modeler interface.
Building	For creating a building model structure using the Building workflow.
5. Select **English** as the system of **Units**.

   Tip: The units can be changed later if necessary, at any stage of the model creation.
6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.
7. Click **Create**.

The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose **Analytical**, click **Create**, and then proceed to either “ T.1 Creating the model using the analytical user interface (on page 399) ” or “ T.1 Creating the model using the command file (on page 442),” respectively.

T.1 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

> **Tip:** Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

T.1 Generating the model geometry

1. On the **Model** ribbon tab, select the **Grid** tool in the **Create** group.

   The **Create Grid** dialog opens.
2. Supply the grid details:
   a. Type `Grid 1` in the **Name** field.
   b. Leave the **Plane** selection as **XY** and the **Creation Method** as **By Spacing**.
   c. Type `20` in the **Number of spaces in X** and `1 ft` in the **Grid spacing X** fields.
   d. Type `15` in the **Number of spaces in Y** and `1 ft` in the **Grid spacing Y** fields.
   e. Click **OK**.
      The grid is created.

   You can control the visibility of individual grids by selecting the **Grids** tool on the **Spreadsheet** ribbon tab, where you can also change the settings for each grid. You can have as many grids as necessary displayed at one time.
3. On the **Model** ribbon tab, select the **Member** tool in the **Create** group.

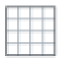
   

   The tool is highlighted to show it is active. A yellow circle highlights the grid intersection the cursor "snaps" to when you hover in the view area.
4. Click on the lower, right corner of the grid (i.e., the origin at 0,0,0).
   A dashed line is "rubber banded" from this point to your cursor.
5. Click on the following two points around the edge to draw two members:

   (0,15,0)
   (20,15,0)
6. Double-click on the final point at (20,0,0).

   This instructs the program you are finished drawing connected members.

   > **Tip:** The tool remains active until so you can continue drawing a new set of connected members elsewhere if necessary.
7. Select the **Member** tool again to make it inactive.
8. On the **View** ribbon tab, select the **Grids** tool in the **Reference** group.



The tool is no longer highlighted and all grids are hidden in the display.

9. Switch on the node and beam labels:
   a. On the **View** ribbon tab, select the drop-down list below the **Numbering** tool in the **Model** group.

   b. Select **Node** and **Member** on this list.
      They are both marked with a check.
   c. Select the **Numbering** tool.

You have now created the nodes and members for this model.

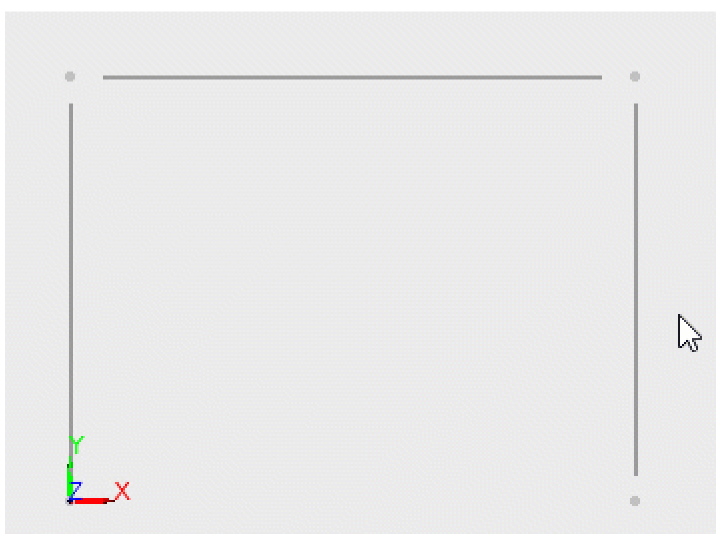
You can review the geometry of each by selecting the **Nodes** tool in the **Nodes** group and **Members** tool in the **Members** group on the **Spreadsheet** ribbon tab, respectively.

T.1 Specifying member properties

Tip: In STAAD.Pro Physical Modeler, you will select a model object and then assign properties, loads, etc. to that object. This fundamental workflow is import to understand as it applies to essentially all actions once you have drawn your model objects.

1. Select the two column members (M1 and M3):
   a. Click anywhere in the empty are in the view window.
      All model objects (members and nodes) are deselected.
   b. Click-and-drag a line horizontally from right to left across the columns to select them.

      This method (i.e., right to left) will select any model object that your drag window crosses.



Note: The **Member** ribbon tab reappears any time you have one or more members in your current selection.

2. On the **Member** ribbon tab, select the **Section** tool in the **Assign properties** group.

   The **Assign Section** dialog opens.
3. Specify the section data for the columns:
   a. Leave all the defaults:

      Type: Standard,
      Country: United States,
      Material: STEEL,
      Category: Hot Rolled,
      Specification: AISC,
      Version: 14 Edition,
      Manufacturer: Generic
   b. Select **W** from the **Table** drop-down list.
      The list of all section names in this table are populated.
   c. Scroll to and select **W12X35** in the list.
   d. Click **OK**.

   The section is assigned to the columns.
4. Click on the beam member.
   It is selected and all other members are deselected.
5. Repeat steps 2 and 3 except to assign a **W14X34** to the beam (M2).
6. Select all three members by either:

   click-and-drag a window around them

   or

   press **<Ctrl+A>**

   or

   on the **Data** ribbon tab, select the **Select all** tool in the **Model** group

Tip: There are numerous ways in which to select all the beams in a model. These are some examples.

7. On the **Member** ribbon tab, select the **Material** tool in the **Assign properties** group.

   The **Assign Material** dialog opens.
8. Specify the material data for the members:

a. Leave the initial defaults:

Source: Catalog
Type: Standard
Country: United States

b. Select **ASTM_STEEL** from the **Specification** drop-down list.
c. Select **A992** from the Name list.
d. Click **OK**.

The material is assigned to the members.

T.1 Specifying member offsets

1. Click the beam member (M2) to select it.
2. On the **Member** ribbon tab, select the **End Offset** tool in the **Edit** group.

   The **Modify Member End Offsets** dialog opens.
3. Specify the start (in this case, left) node offset:
   a. Leave the **Direction** as **Local.**

   > **Tip:** Given the orientation of this beam, the local 1 and global X axis are parallel, but it's best practice to use local coordinates when the offset is taken with respect to the local member's ends.

   b. Type `6 in` in the **x** direction.

   Local axis 1, 2, and 3 correspond to the member's localized X, Y, and Z axis.

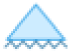
   > **Tip:** Notice that even though the default input units are in feet, you are able to type different units and the program converts to the input units for you.
4. Specify the end (in this case, right) node offset:
   a. Leave the **Direction** as **Local.**
   b. Type `-6 in` in the **x** direction.

   > **Note:** The negative sign for the offset value at the end.
5. Click **OK**.
   The end offsets are drawn as red lines from the nodes in their indicated direction.

T.1 Specifying supports

1. Click on the lower-left node (N1) to select it.
   Any other selected items are removed from the current selection set. The node is selected. The node label appears if you have the labeling turned on.
2. On the **Node** ribbon tab, select the **Fixed** tool in the **Supports** group.

   A fixed nodal support type is assigned to this node.
3. Select the lower-right node (N4).
4. On the **Node** ribbon tab, select the **Pinned** tool in the **Supports** group.



A pinned nodal support type is assigned to this node.

Tip: You can use the **Custom** tool in the **Supports** group to generate any restraint or spring nodal boundary condition necessary.

You can review the nodal boundary conditions by selecting either the **Supports** tool or the **Springs** tool in the **Nodes** group on the **Spreadsheet** ribbon tab.

T.1 Viewing the model in 3D

1. Select all model objects by either:

   Right-click in the view area and select **Select All** from the pop-up menu

   or

   Click-and-drag a window area around all model objects

   or

   Press **<Ctrl+A>**
2. On the **View** ribbon tab, select the **3D Rending** tool in the **Model** group.

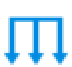
   The tool is highlighted to indicate it is active. The view displays the rendered model using the member's shapes, materials, and properties such as end offsets.

You can continue to model and work with this view option turned on. However, it is typically clearer to work with line drawings. Select the **3D Rendering** tool again to deactivate it.

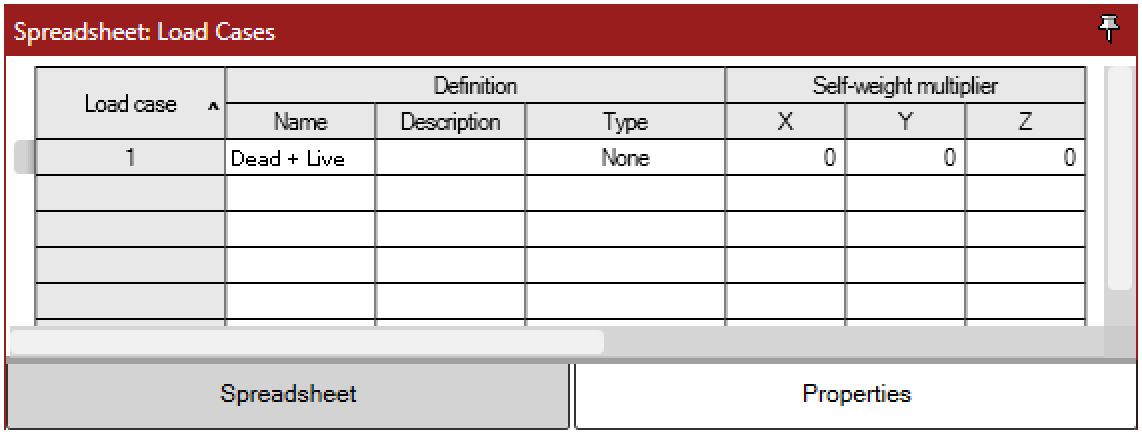
T.1 Specifying loads

The STAAD.Pro Physical Modeler creates a default Load Group 1 with the no specified load type. You will add the dead and live loads to this load group and then create a new load group for the wind load.

1. On the **Spreadsheet** ribbon tab, select the **Load cases** tool in the **Reference** group.

   The Load Cases spreadsheet opens.
2. For Load case 1, type `Dead + Live` in the **Name** cell to rename the load case.

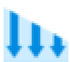


Spreadsheet: Load Cases

Load case	Definition			Self-weight multiplier		
	Name	Description	Type	X	Y	Z
1	Dead + Live		None	0	0	0

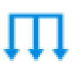
Spreadsheet | Properties

3. Click the beam member (M2) to select it.
4. On the **Member** ribbon tab, select the **Distributed** tool in the **Loads** group.

   The **Add Member Distributed Load** dialog opens.
5. Specify the load:
   Leave the **Load Group** as **LC: Dead + Live** and the **Load Type** as **Uniform**.
   a. Select **Global Y** as the **Direction**.
   b. Type `-2.5 kip/ft` in the **Magnitude** field.
   c. Click **OK**.

   The load is applied to the beam.
6. On the **Model** ribbon tab, select the **Load Case** tool in the **Loads** group.

   The **Add Load Case** dialog opens.
7. Specify the wind load case details:
   a. Type `Wind From Left` in the **Name** field.
   b. Select **Wind** for the **Type**.
   Leave the other items for Self-weight data empty (default).
   c. Click **OK**.

   A new load case is added to the **Load Cases** spreadsheet and the load is automatically selected in the program window status bar as the current load.

   Note: When load items are created they are added to the current load case or load group by default.
8. Select the upper-left node (N2).
9. On the **Node** ribbon tab, select the **Nodal Load** tool in the **Loads** group.

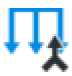
   

The **Add Nodal Load** dialog opens.

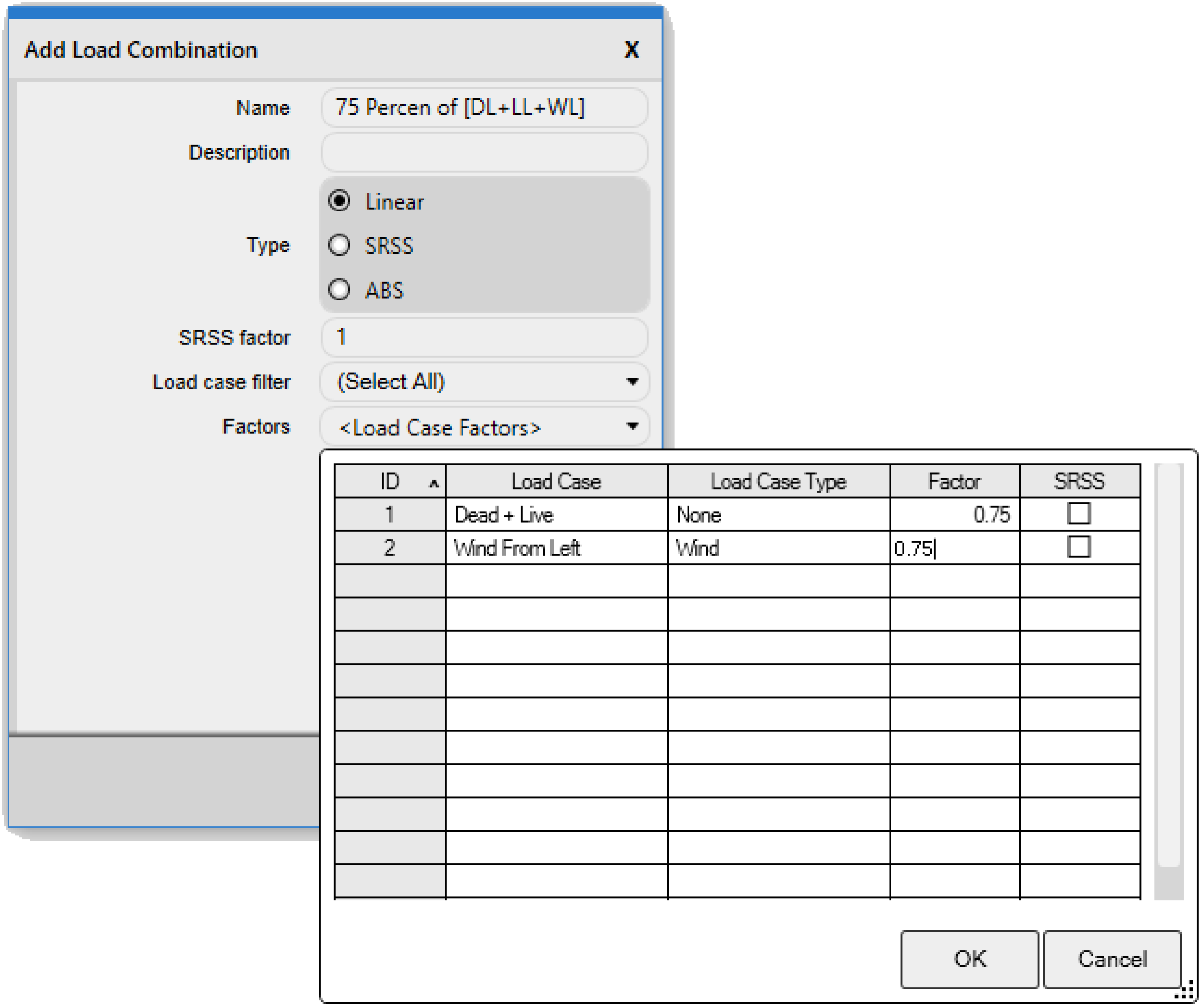
10. Specify the lateral wind load:
    a. Leave the **Load group** selection as **LC:Wind from left**.
    b. Type `10 kip` in the **Fx** field.
    c. Click **OK**.

T.1 Creating a load combination

1. On the **Model** ribbon tab, select the **Load Combination** tool in the **Loads** group.

   The **Add Load Combination** dialog opens.
2. Type `75 Percent of [DL+LL+WL]` in the **Name** field.
3. Select **Linear** as the load combination **Type**.
4. Click in the **Load case filter** field and then check the **(Select All)** option.
5. Assign the load case factors:
   a. Click in the **Factors** field.
      The load case factors table opens. All the load cases in the model are included from the previous step.
   b. For load case 1 (the first row in the table), click in the **Factor** cell and type `0.75`.
   c. Repeat step 5a to enter a load case factor of 0.75 for the wind load (load case 2).
   d. Click **OK**.



6. Click **OK**.

T.1 Generate the analysis model

You have now completed the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will create load combinations, add output commands, select analysis criteria, and specify design parameters.

For this tutorial, you will use the default options for analytical model generation. However, for advanced models you can review and change those options in the **Configuration** dialog.

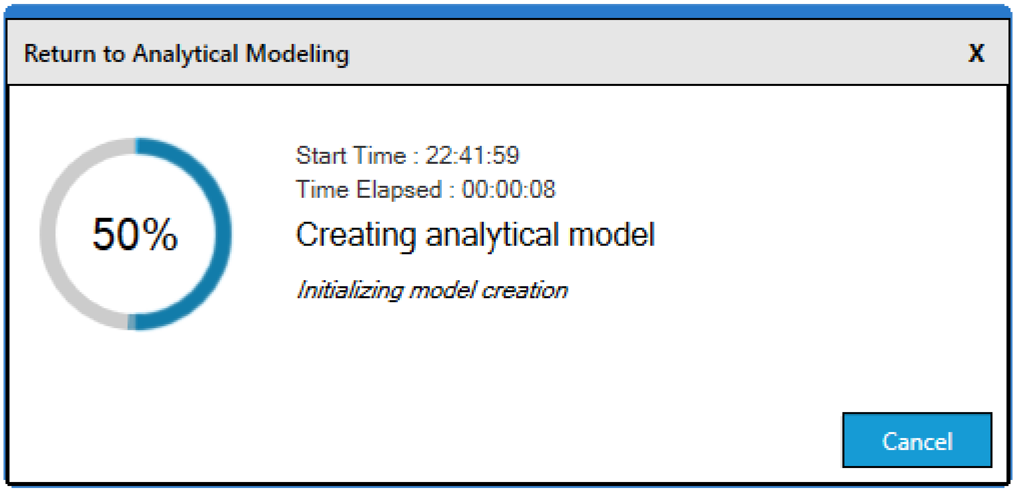
1. Select **Save** to save the physical model.
2. Either:

   On the **Model** ribbon tab, select the **Return to Analytical Modeling** tool in the **STAAD.Pro** group

or

On the **File** ribbon tab, select **Create analysis model** in the backstage tabs

The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.



3. Click **OK**.
   If you have not previously associated this model with a CONNECT Project, you are asked to do so. This is will not be necessary for this tutorial.
4. Click **Cancel** in the **Assign Project** dialog.

The STAAD.Pro Physical Modeler window closes and the model is loaded into the STAAD.Pro analytical modeling interface.

T.1 Creating the model using the analytical user interface

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

Tip: Refer to GS. Application Window Layout (on page 54) for reference on the application window.

Note: If you completed the model using the **Physical Modeling** worfkow, then instead proceed to T.1 Analysis and Design (on page 426).

T.1 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

The STAAD input file commands generated are:

```
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```

When starting in the Analytical Modeling workflow, a grid is initially displayed in the main view window. This grid is controlled by the **Snap Node/Beam** dialog. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area.

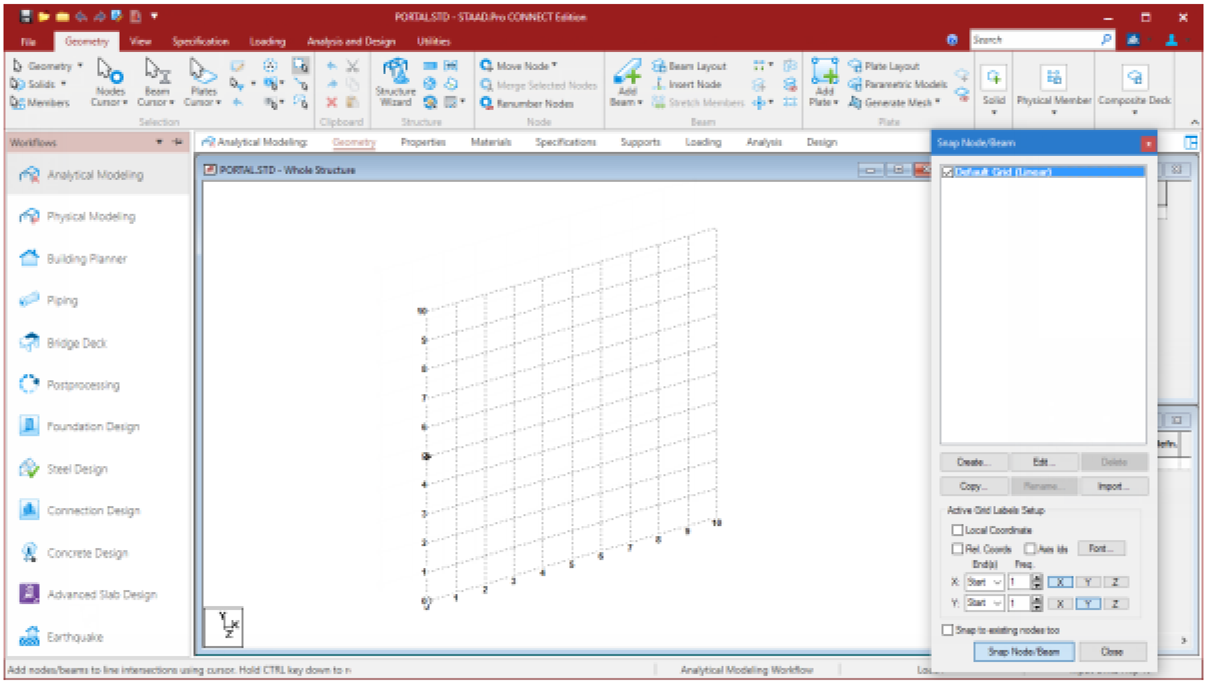
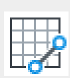


*Figure 14: The STAAD.Pro window with the **Snap Node/Beam** dialog open*

Note: The **Snap Node/Beam** dialog opens on the right side of the screen. You can reopen this dialog by selecting the **Snap Grid Beam** tool  in the **Structure** group on the **Geometry** ribbon tab.

1. On the **Snap Node/Beam** dialog, click **Create**.
   A dialog opens which will enable us to set up a grid.

   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.

Linear	used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board
Radial	used to place construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modeled as piece-wise linear straight line segments
Irregular	used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane

2. Select **Linear**, which is the **Default Grid.**

   In this structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. Leave **X-Y** as the **Plane** of the grid. The size of the model that can be drawn at any time is controlled by the number of **Construction Lines** to the left and right of the origin of axes, and the **Spacing** between adjacent construction lines.
3. Type a **Name** of `Grid 1`.
4. Type `20` as the number of lines to the **Right** of the origin along **X** and `15` above the origin along **Y**

Leave the default spacing of **1** feet between lines along both **X** and **Y**.

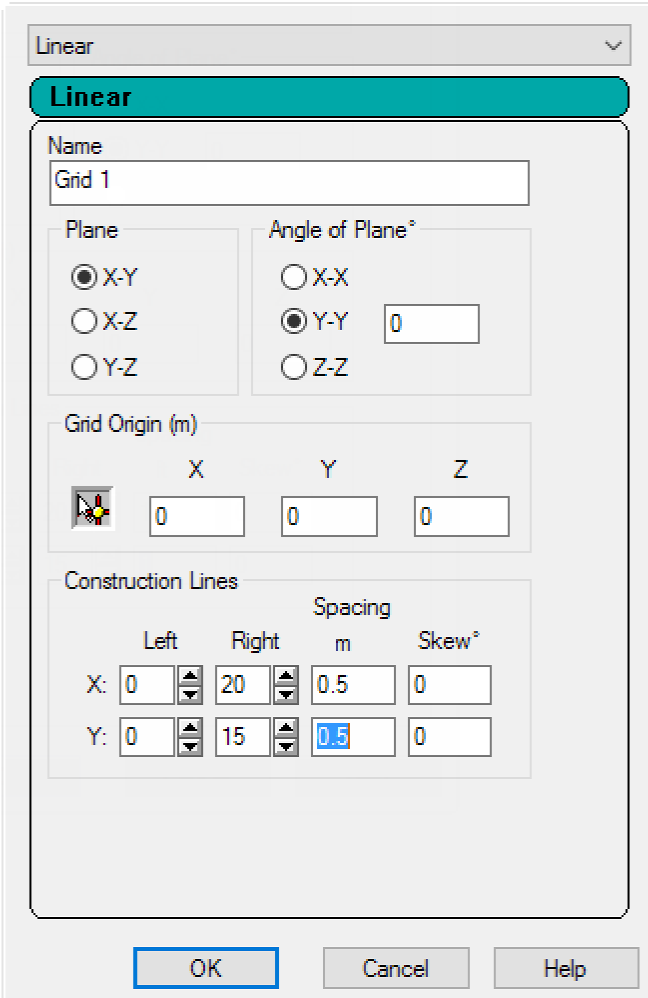


Figure 15:

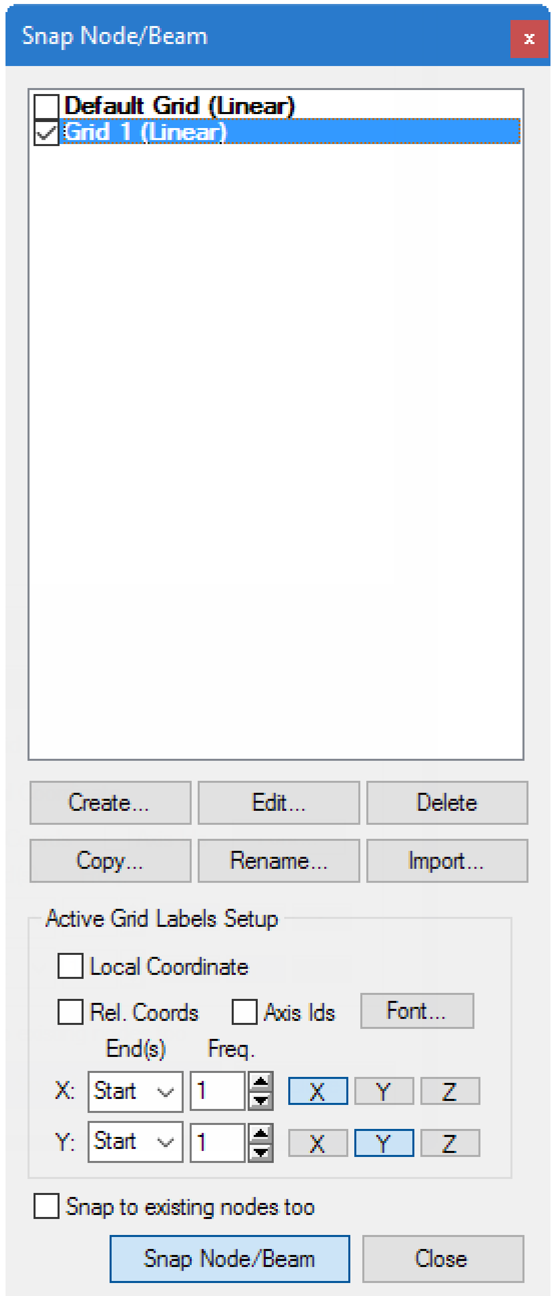
5. Click **OK**.
6. In the **Snap Node/Beam** grids list, check the new **Grid 1** option.

   You can create any number of grids. By providing a name, each new grid can be identified for future reference.

Tip: Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

Tip: To change the settings of this grid, select the name in the **Snap Node/Beam** dialog and then click **Edit**.

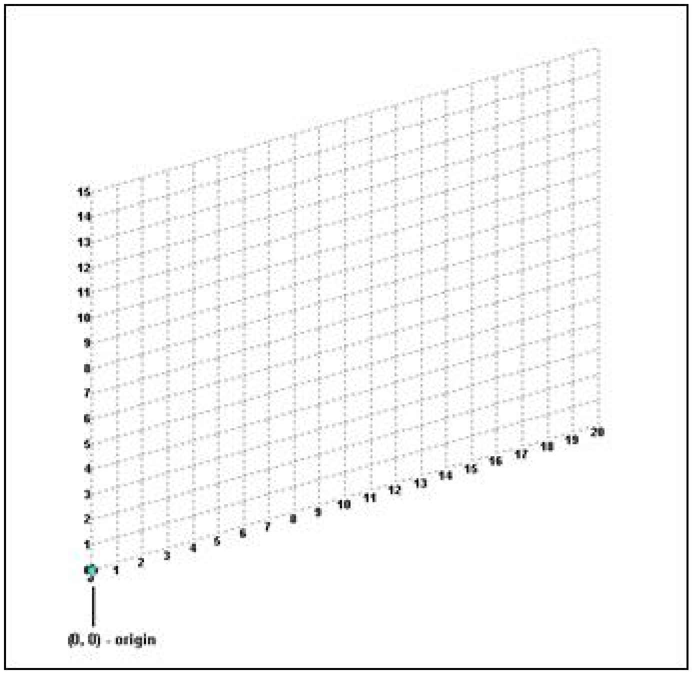
The check by **Default Grid** is automatically cleared. STAAD.Pro will only display a single grid at a time.



7. In the View window, click at the origin (0, 0) to create the first node.

   A line is "rubber-banded" between this node and the mouse pointer, which previews the member to placed with the next mouse click.

   The **Snap Node/Beam** feature is active by default.



8. Click on the following points to create nodes and automatically join successive nodes by beam members.

(0, 15)
(20, 15)
(20, 0)

When steps 1 through 5 are completed, the structure will be displayed in the drawing area as shown below.

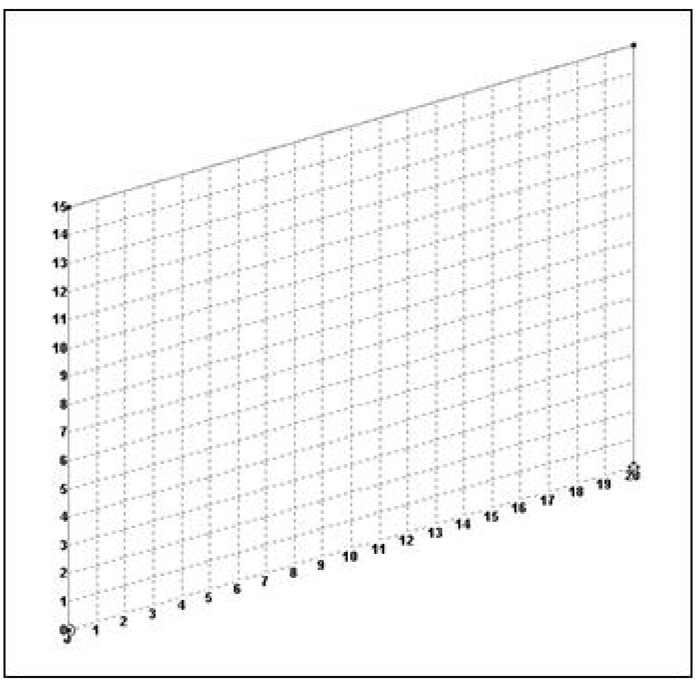


Figure 17:

9. Click **Close** in the **Snap Node/Beam** dialog.
   The grid is hidden.

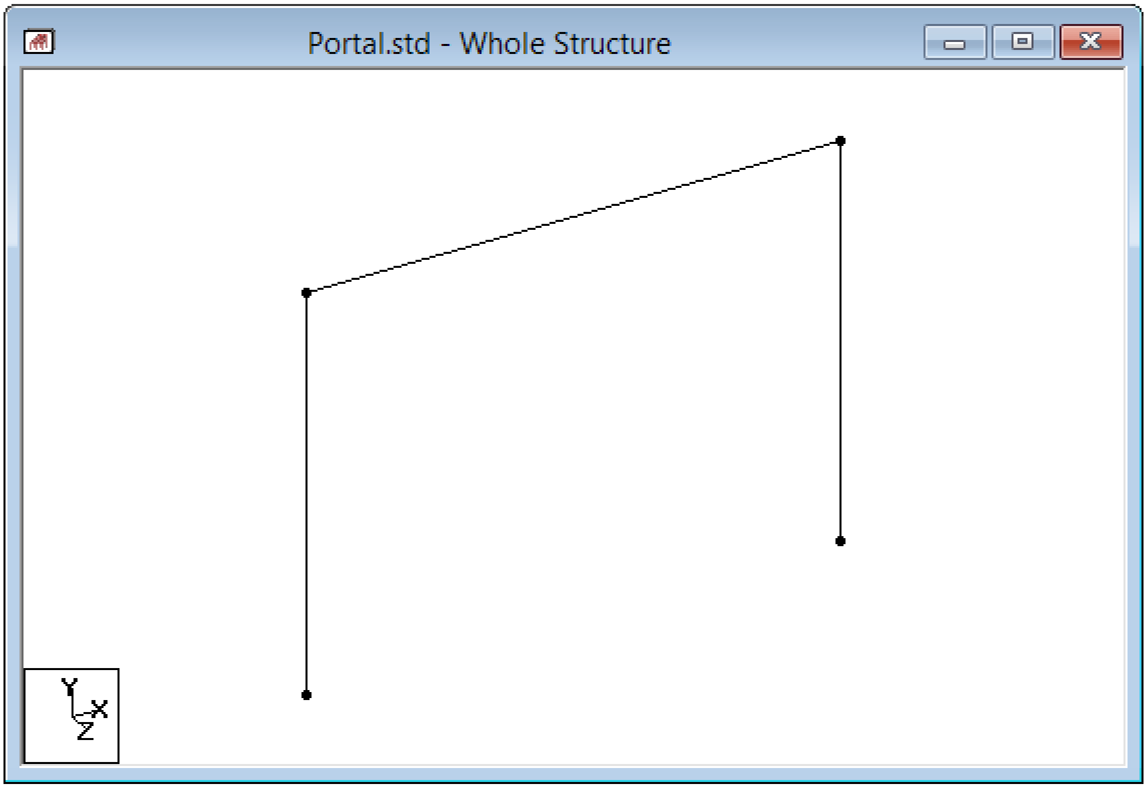


Figure 18: The portal frame members drawn

It is important to save your work often. This helps to avoid loss of data and protect your investment of time and effort against power interruptions, system problems, or other unforeseen events.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Switching on node and beam labels

Node and beam labels are a way of identifying the entities in the View window.

1. Right click anywhere in the view window and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens to the **Labels** tab.
2. Set the **Node Numbers** and **Beam Numbers** on and then click **OK**.

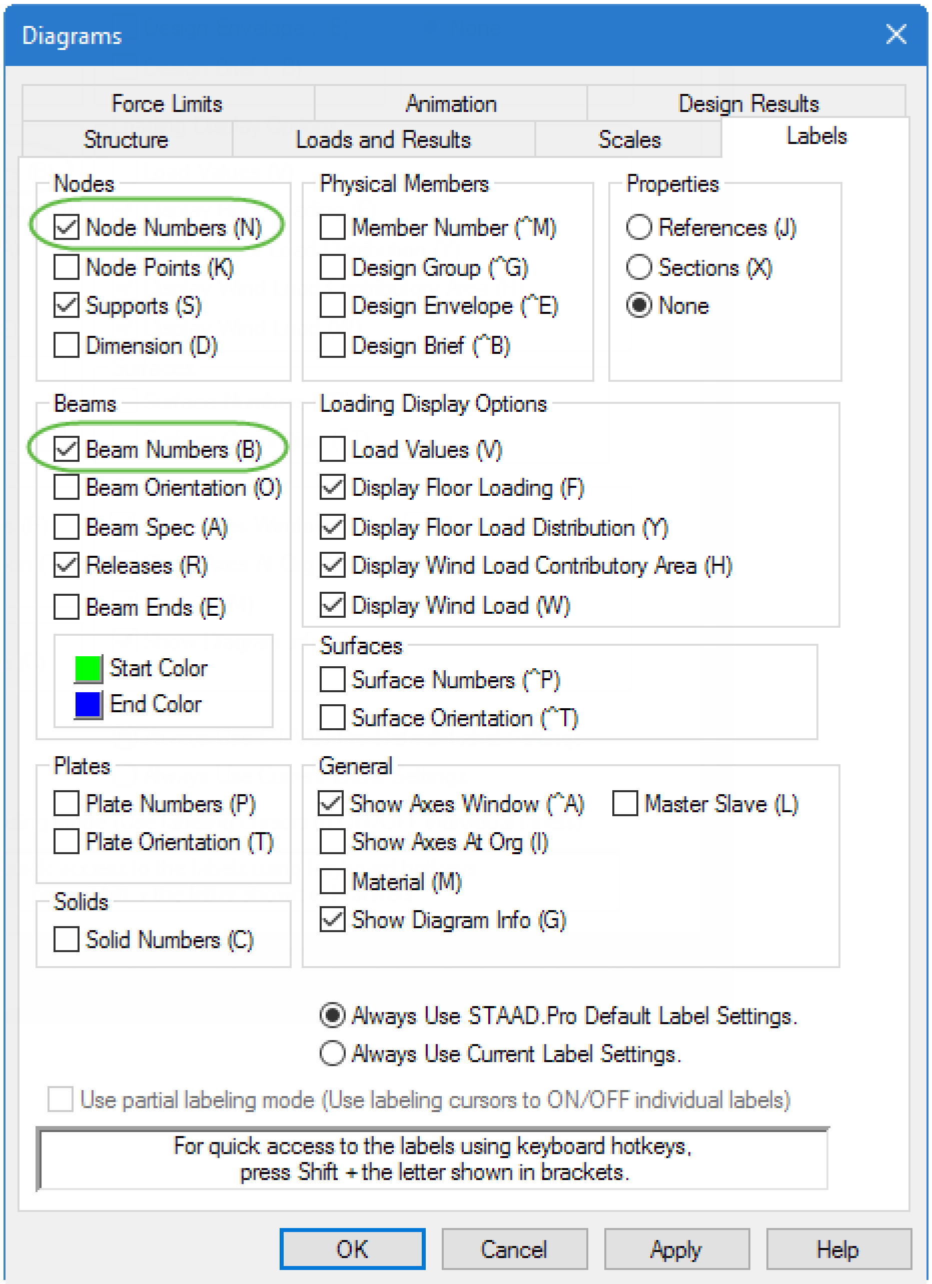


Figure 19: Select the Node and Beam numbers

Tip: Alternatively, you can press **<Shift+N>** and then **<Shift+B>** to quickly toggle the same labels. The letters in parenthesis on the **Labels** tab indicate these **<Shift>** key shortcuts.

Tip: You can change the font of the node/beam labels by selecting the **File** ribbon tab and then selecting **Settings** > **Display Options**. Select the appropriate tab (Node Labels / Beam labels) from the **Options** dialog.

T.1 Specifying member properties

In this procedure, you will assign cross section properties to the beams and columns.

The STAAD input file commands generated are:

```
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X35
2 TABLE ST W12X34
```

1. Select the **Properties** page in the Analytical Modeling page control bar.

The **Properties - Whole Structure** dialog opens.

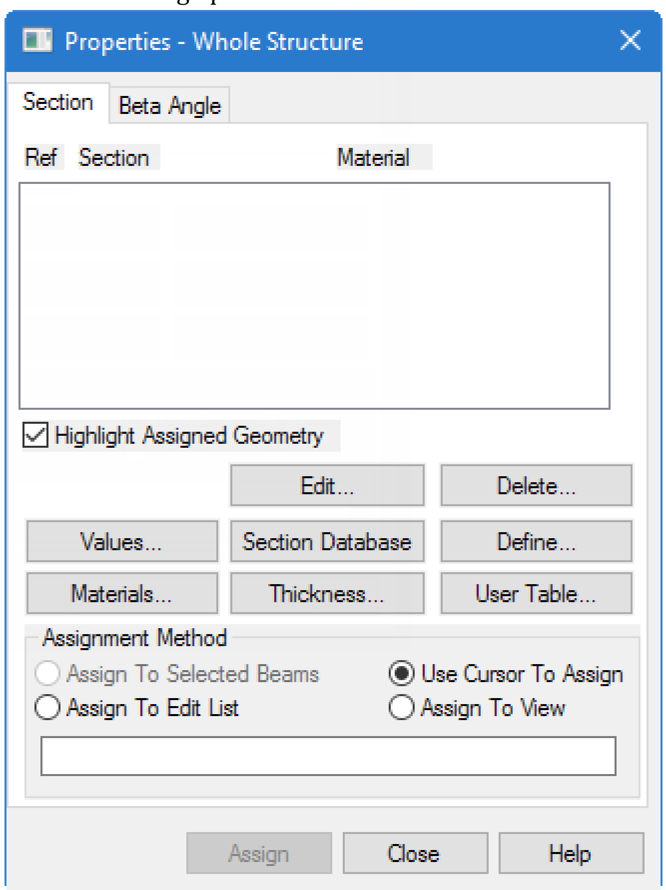


Figure 20:

2. Click **Section Database**.

   The **Section Profile Tables** dialog opens.

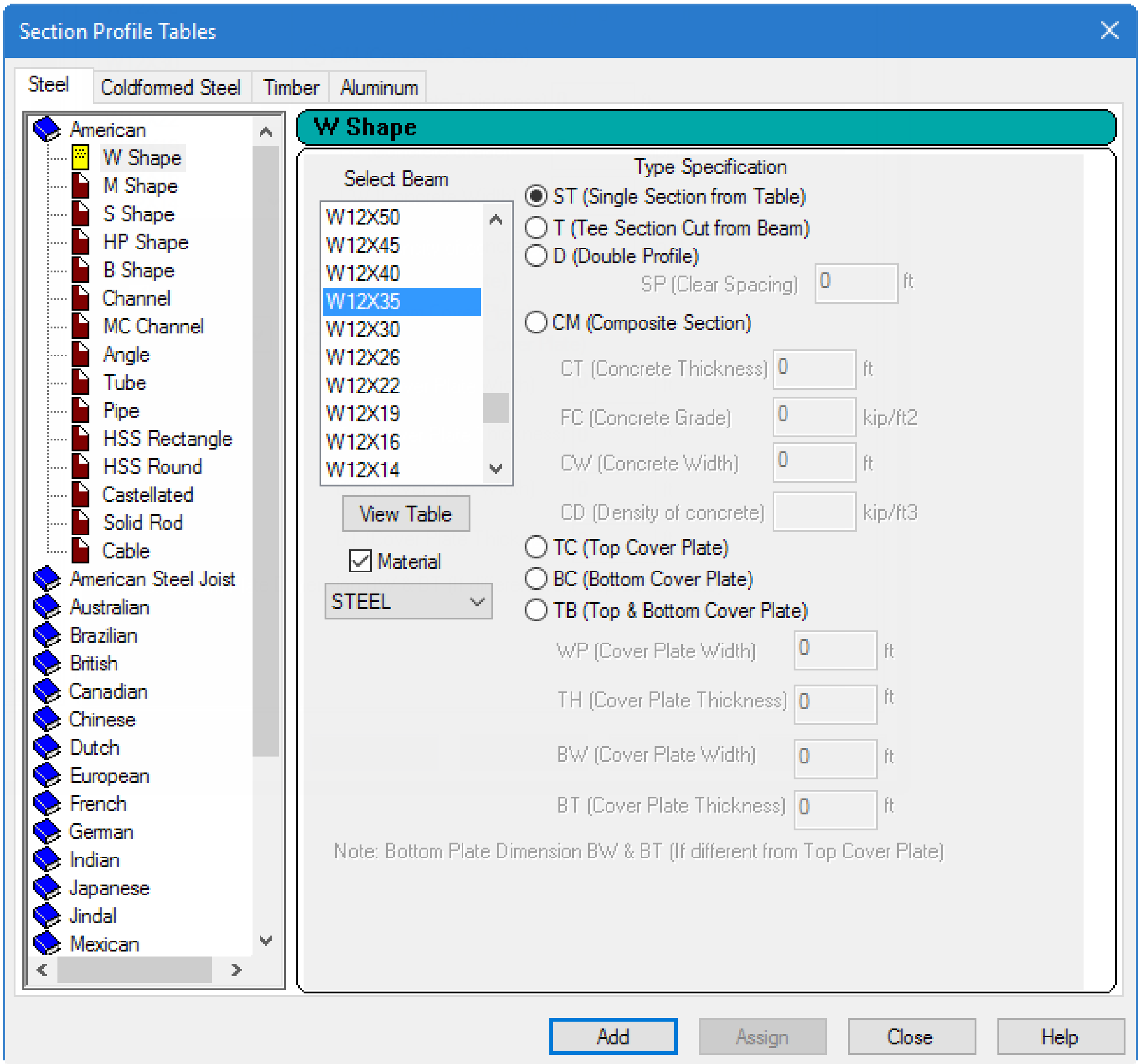


Figure 21:

3. Select the **W Shape** tab under the **American** option.

   The property type we wish to create is the W shape from the AISC table.
4. Choose **W12X35** as the beam size and then select the **ST** option as the section type.

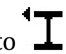
Tip: The **Material** check box is set. Leave this set and the selection set to **Steel** as it will be used to assign the built-in steel material properties when this section is assigned.

5. Click **Add**.

Tip: Detailed explanation of the terms such as ST, T, CM, TC, BC, etc. is available in TR.20.1 Assigning Properties from Steel Tables (on page 2257).

The W12X35 section is added to the Sections list in the **Properties - Whole Structure** dialog.

6. To create the second member property (ST W14X34), select the **W14X34** shape and click **Add**.
7. Click **Close** in the **Section Profile Tables** dialog.
8. Assign these properties with selected members in our model:
   a. Select the first property reference in the **Properties** dialog (W12X35).
   b. Select the **Use Cursor to Assign** option in the Assignment Method group.
   c. Click **Assign**.

      The mouse pointer changes to 
   d. Click on members 1 and 3.
   e. To stop assigning properties, either:

      click **Assigning**

      or

      press the **<Esc>** key.
9. Repeat step 8 except to assign the second property reference (W14X34) to member 2.

   After the properties are assigned to the respective members, the member labels will indicate the section reference numbers.

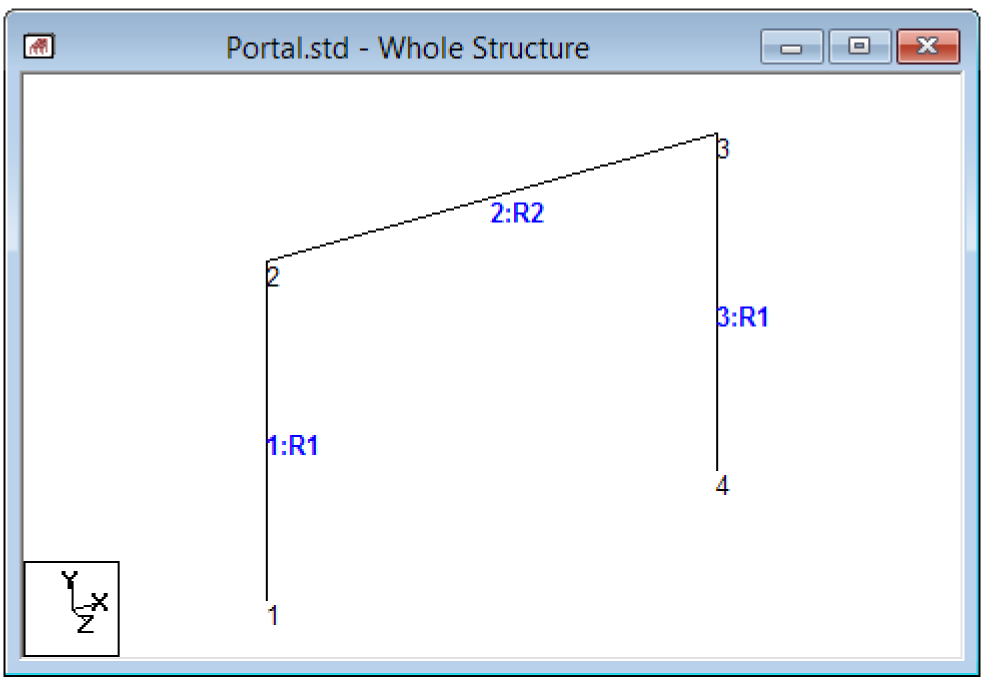


Figure 22:

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Specifying material definition

When T.1 Specifying member properties (on page 406), you kept the **Material** option checked. Consequently, the material definitions were assigned to the members along with the sections.

The STAAD input file commands generated are:

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
...
CONSTANTS
MATERIAL STEEL ALL
```

T.1 Changing the input units of length

For specifying member offset values and for some material definition values, as a matter of convenience, use length units of inches instead of feet.

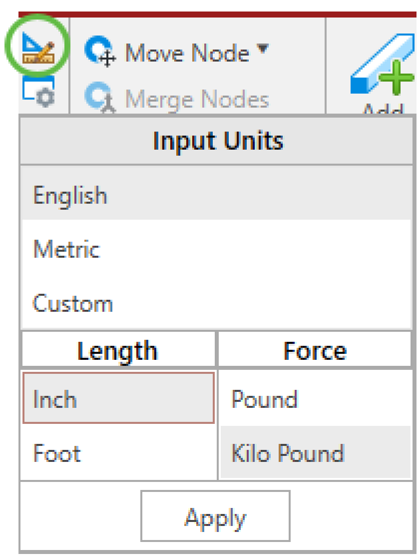
The STAAD input file commands generated are:

```
UNIT INCHES KIP
```

1. On the **Geometry** ribbon tab, select the **Input Units** tool in the **Structure** group.

   The **Set Input Units** pop-up opens.



2. Select **English** as the **Input Units**.
3. Set the **Length** input units to **Inch**.
4. Click **Apply**.

T.1 Specifying member offsets

Member 2 (the beam) actually spans only the clear distance between the column faces rather than the center to center distance. This is modeled by specifying offsets.

Member 2 is offset at the start joint by 6 inches in the global X direction (and 0.0 and 0.0 in Y and Z directions). The same member is offset by negative 6.0 inches at its end joint.

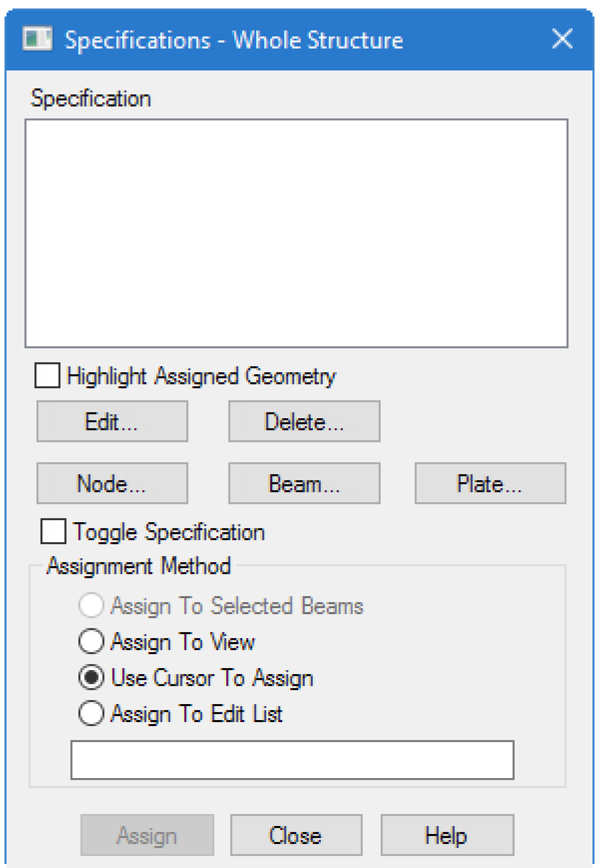
The STAAD input file commands generated are:

```
MEMBER OFFSET
2 START 6.0 0.0 0.0
2 END -6.0 0.0 0.0
```

1. Select member 2 by clicking on it in the view window.

   Note: If the beam cursor isn't currently active, select the **Beam Cursor** tool in the **Selection** group on the **Geometry** ribbon tab. 

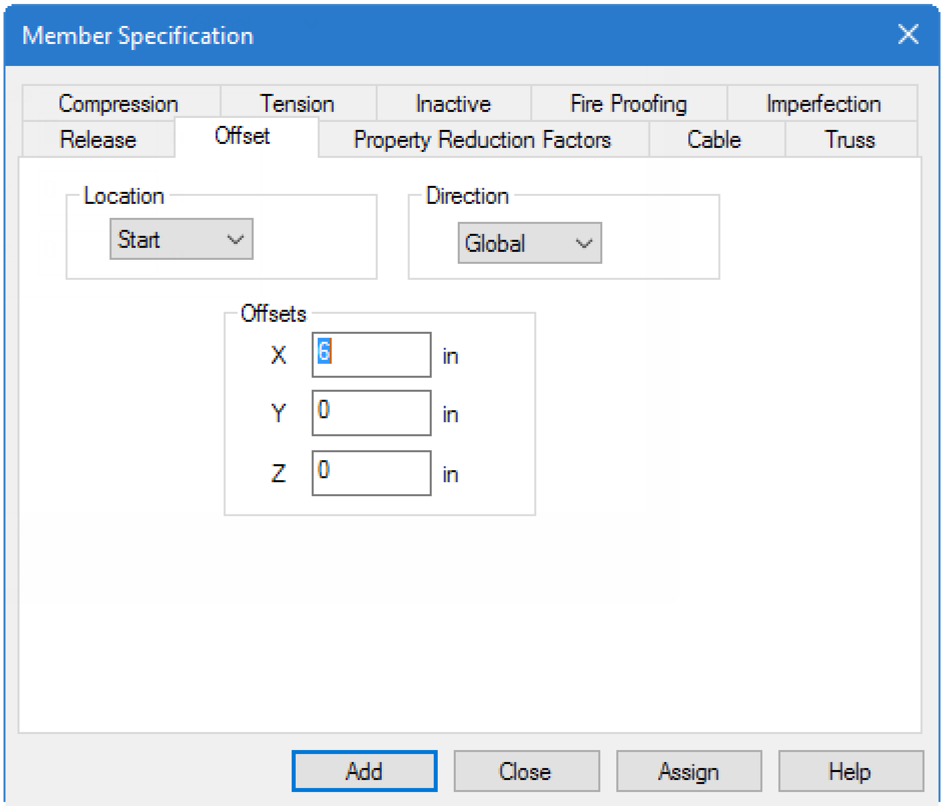
   The selected member will be highlighted.
2. Select the **Specifications** page on the Analytical Modeling page bar.
   The **Specifications - Whole Structure** dialog opens. Member releases, offsets, and other beam specifications are defined here.



3. Click **Beam**.

The **Member Specification** dialog opens.

4. Specify the member start offset:
   a. Select the **Offset** tab.



   b. Select **Start** as the **Location**.
   c. Type `6.0`(in.) in the **X** offset field.

      Leave the Direction as Global. It is convenient to define the offset at the start node in the X direction. Since the beam member is aligned with the global X axis, there is no difference between choosing a global or local frame of reference.
   d. Click **Assign**.

   The dialog closes and the offset specification is added to the start of member 2. This is displayed visually on the member as well as listed in the Specification list in the **Specifications - Whole Structure** dialog.
5. On the **Specification** ribbon tab, select the **Beam** > **Offset** tool in the **Specifications** group.

   The **Member Specification** dialog opens to the **Offsets** tab.
6. Specify the member end offset:
   a. Select **End** as the **Location**.
   b. Type `-6.0`(in.) in the **X** offset field.
   c. Click **Assign**.

   The dialog closes and the offset specification is added to the start of member 2.

Tip: Alternatively to steps 5 and 6, you can repeat steps 3 and 4, except for selecting the **End** option and providing a value of `-6.0` for **X**. This procedure is used to demonstrate two methods to complete a similar task.

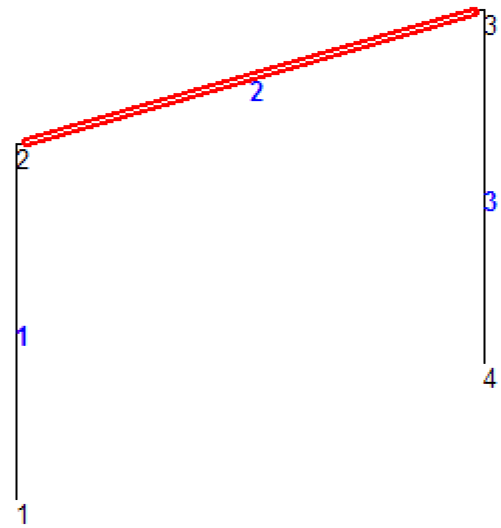


Figure 23: The model after both start and end offsets are assigned

Click anywhere in the drawing area to deselect the member.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Printing member information in the output file

It is useful to view the member information used by the program for analysis. You can add a report in the STAAD output file consisting of information about all the members including start and end joint numbers (incidences), member length, beta angle, and member end releases.

The STAAD input file commands generated are:

```
PRINT MEMBER INFORMATION ALL
```

1. Select all the beam members by one of the following methods:

   on the **Select** ribbon tab, select the **All** tool in the **Beams** group

   or

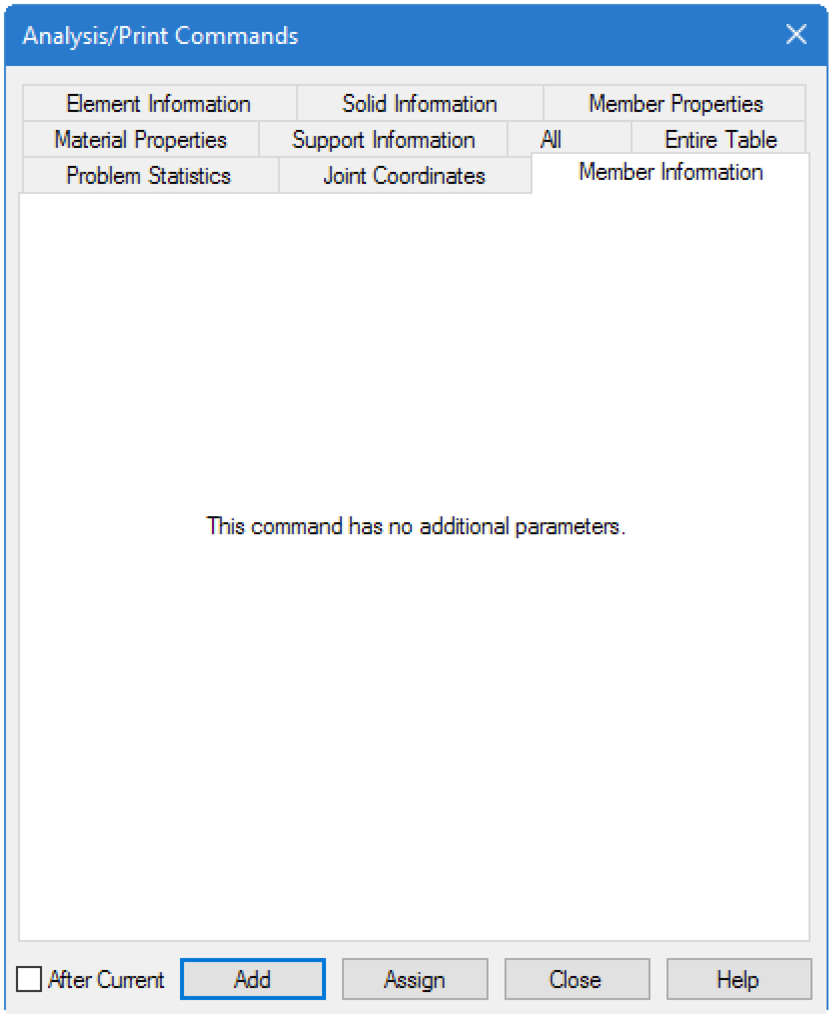
   click-and-drag a window around all members in the View window

   or

   press **<Ctrl+A>**
2. On the **Analysis and Design** ribbon tab, select the **Pre-Analysis Commands** tool in the Analysis Data group.

The **Pre Analysis Print - Whole Structure** dialog opens.

3. In the **Pre Analysis Print - Whole Structure** dialog, click **Define Commands**.
   The **Analysis/Print Commands** dialog opens.
4. Select the **Member Information** tab and then click **Assign**.



5. Click **Close**.

Click anywhere in the drawing area to deselect the member.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Specifying Supports

The boundary conditions of this problem call for restraining all degrees of freedom at node 1 (FIXED support) and a pinned type of restraint at node 4 (restrained against all translations, free for all rotations).

The STAAD input file commands generated are:

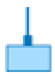
```
SUPPORTS
1 FIXED ; 4 PINNED
```

1. On the **Select** ribbon tab, select the **Node Cursor** tool in the **Cursors** group.

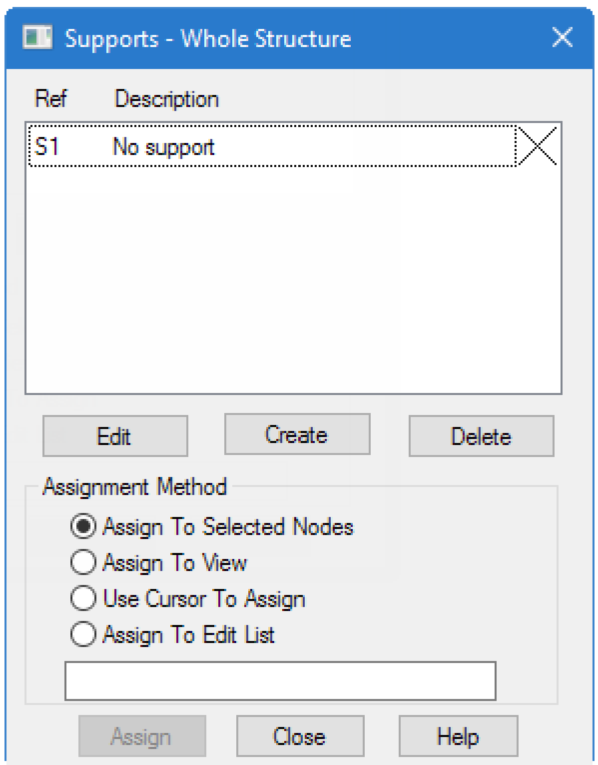
2. In the View window, click on nodes 1 to select it.

Note: The **Node Tools** ribbon tab opens when nodes are selected.

3. On the **Node Tools** ribbon tab, select the **Assign Supports** tool in the **Model** dialog.

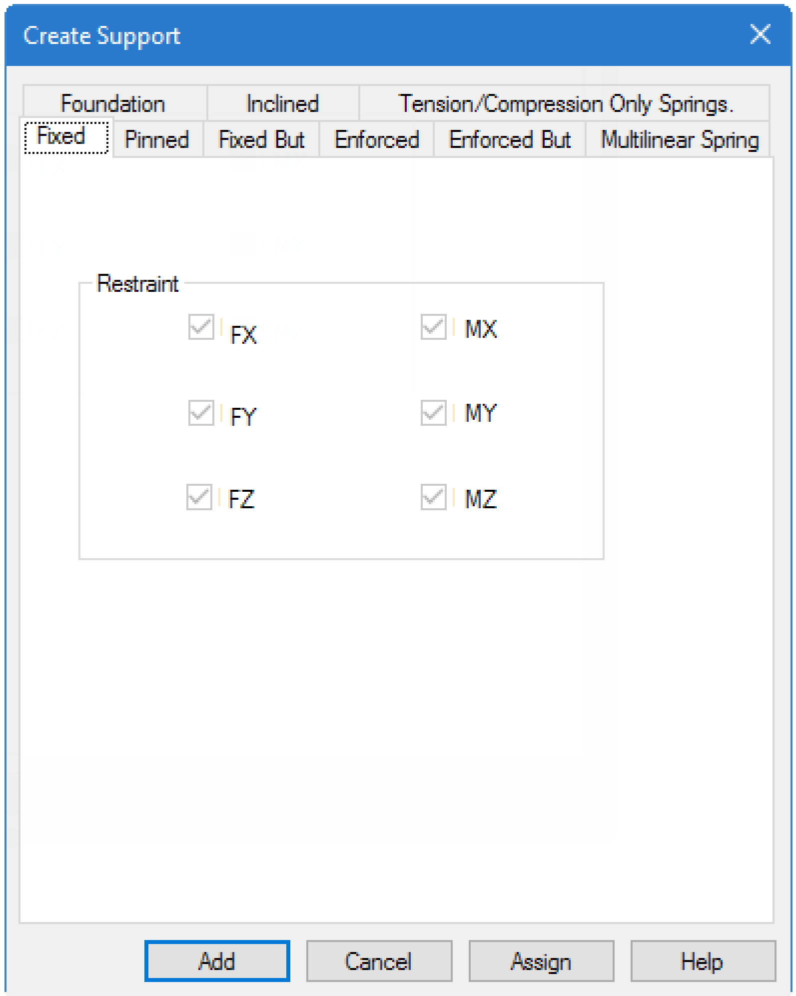


The **Supports - Whole Structure** dialog opens.



4. In the **Supports** dialog, click **Create**.

The **Create Support** dialog opens.



5. Select the **Fixed** tab (selected by default) and then click **Assign**.

   This creates a FIXED type of support at node 1 where all 6 degrees of freedom are restrained.
6. To create a PINNED support at node 4, repeat steps 2 through 5, except for selecting node 4 and selecting the **Pinned** tab in the **Create Support** dialog.

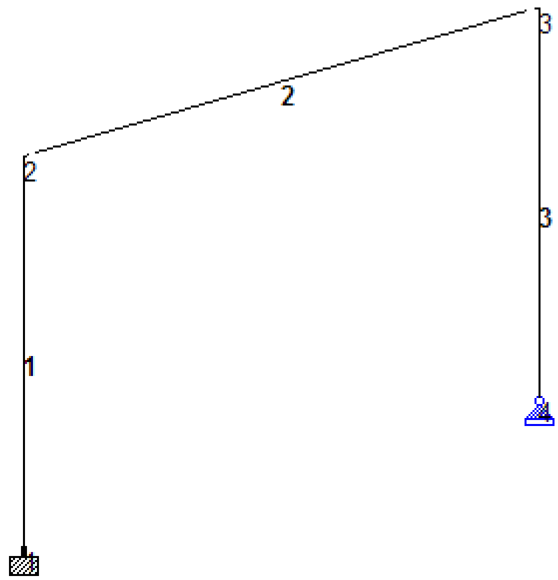


Figure 24: The portal frame with supports assigned

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Viewing the model in 3D

You can review the model in 3D in the STAAD.Pro user interface.

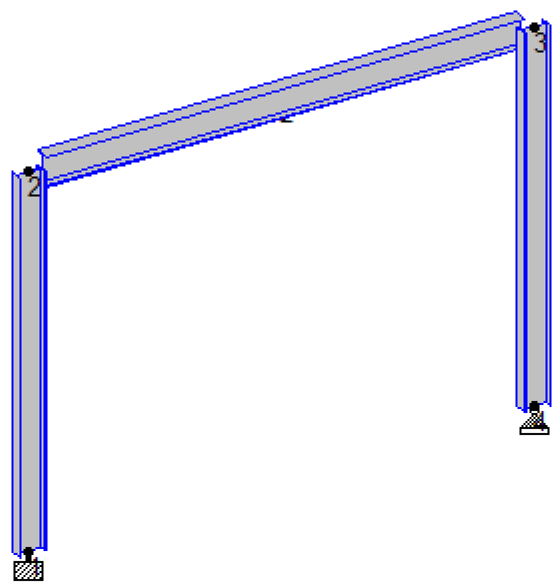
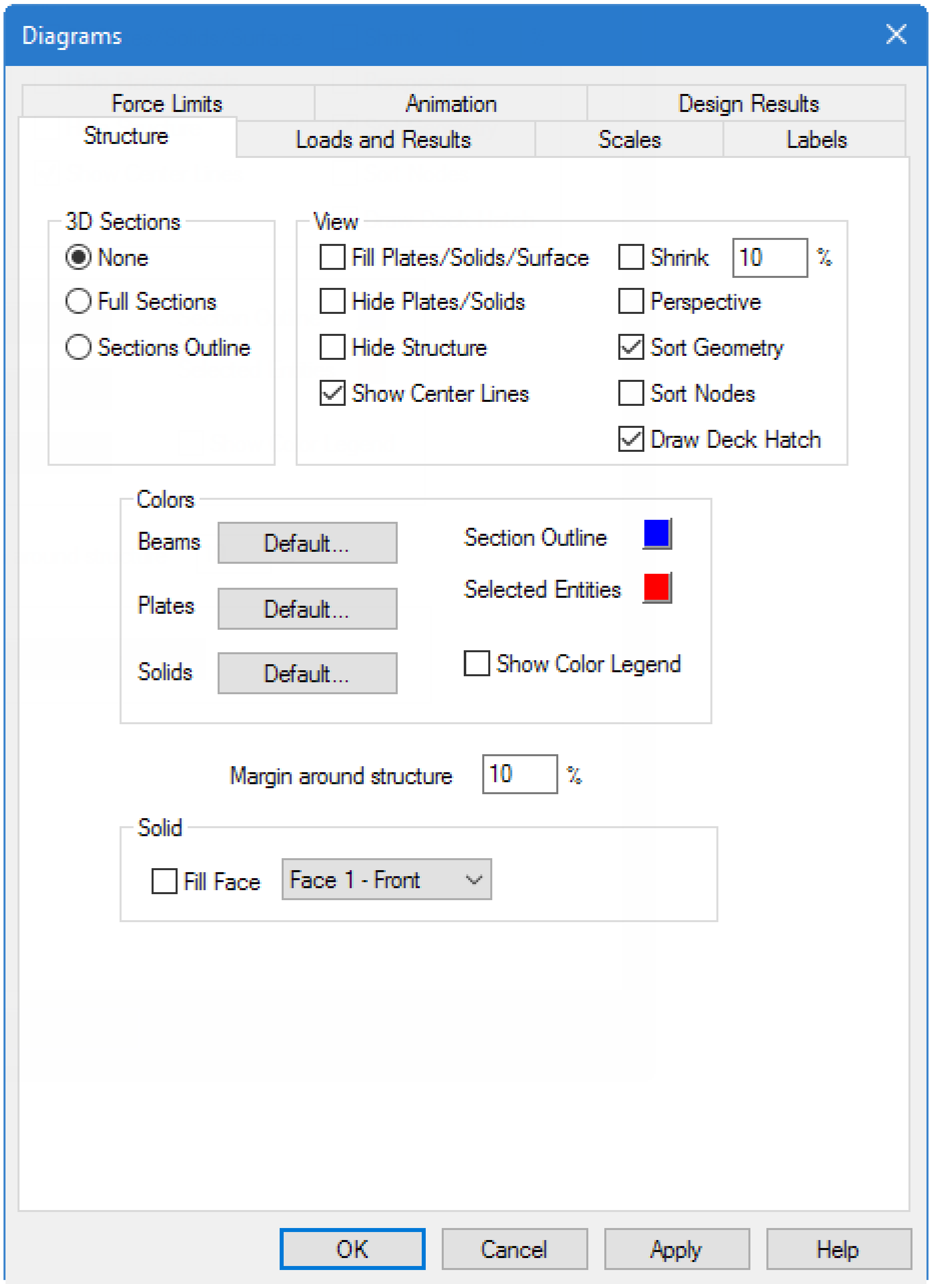


Figure 25: The model rendered in three dimensions

1. Right-click in the view window and select **Structure Diagrams** from the pop-up menu.

The **Diagrams** dialog opens to the **Structure** tab.



The options under **3D Sections** control how the members are displayed.

None	displays the structure without displaying the cross-sectional properties of the members and elements.
Full Sections	displays the 3D cross-sections of members, depending on the member properties.
Sections Outline	displays only the outline of the cross-sections of members.

2. Select **Full Sections** and then click **OK**.

> **Tip:** You can also change the color of the sections by clicking on the **Section Outline** color button in the **Colors** group.

Alternatively, you can quickly render the model in 3D in a new window either by:

selecting the **3D Rendering** tool in the **Windows** group on the **View** ribbon tab  or selecting **View** > **3D Rendering**.

This method has no further settings for colors.

T.1 Specifying Loads

Three load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial.

The STAAD input file commands generated are:

```
UNIT FEET KIP
LOADING 1 DEAD + LIVE
MEMBER LOAD
2 UNI GY -2.5
LOADING 2 WIND FROM LEFT
JOINT LOAD
2 FX 10.
LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL)
1 0.75 2 0.75
```

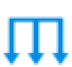
The creation and assignment of load cases involves the following steps:

I. Create the two basic load cases
II. Create a load combination as a third load case
III. Assign these loads to the respective members/nodes.

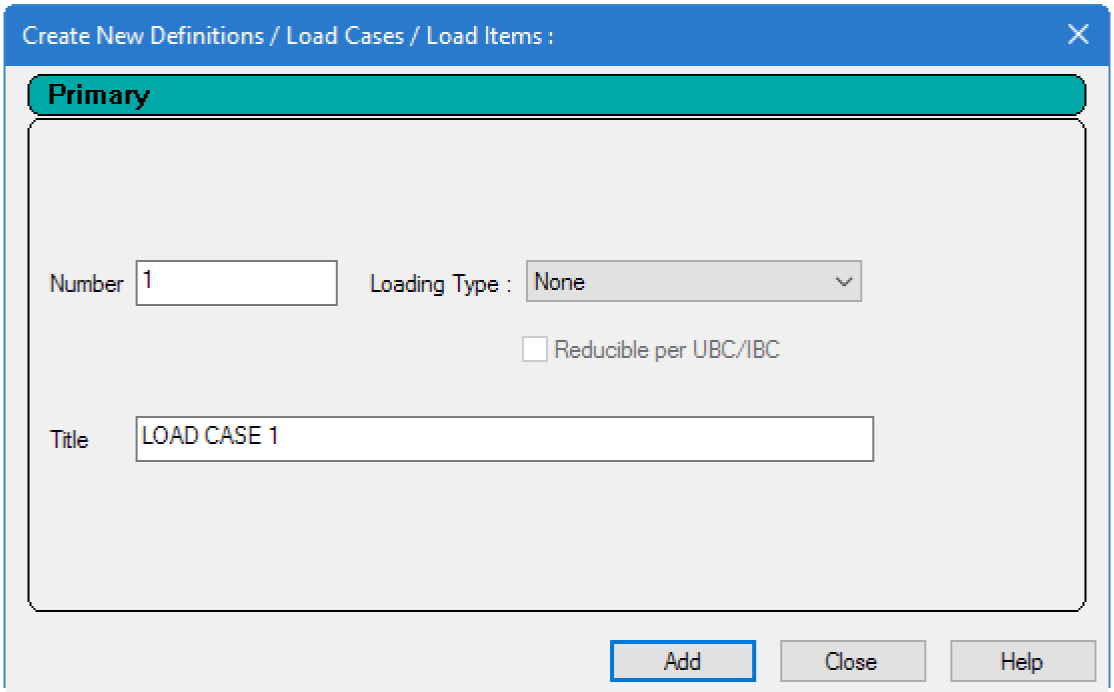
T.1 Creating Load Cases 1 and 2

Load cases 1 and 2 are primary load cases. These load cases are used during analysis of the structure.

1. Change the length units to **Feet** by selecting the length unit in the application window status bar.
   See T.1 Changing the input units of length (on page 409).
2. On the **Loading** ribbon tab, select the **Primary Load Case** in the **Loading Specifications** group.

   The **Create New Primary Load Cases** dialog opens.

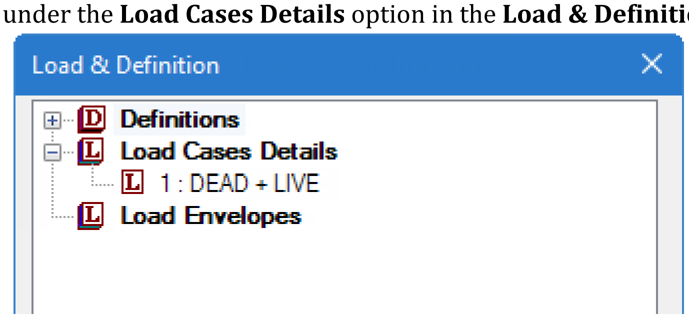


3. Enter the properties for the first load case:
   a. Type `DEAD + LIVE` in the **Title** field.

      Leave the **Number** as the default value of 1.

      Note: The **Loading Type** list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those codes. Note that there is a check box labeled **Reducible per UBC/IBC**. This feature is active only when the load case is assigned a Loading Type called **Live** when you create that load case.

      Since this tutorial does not use the automatic load combination generation feature, leave the **Loading Type** as **None**.
   b. Click **Add**.
      The load case appears under the **Load Cases Details** option in the **Load & Definition** dialog.



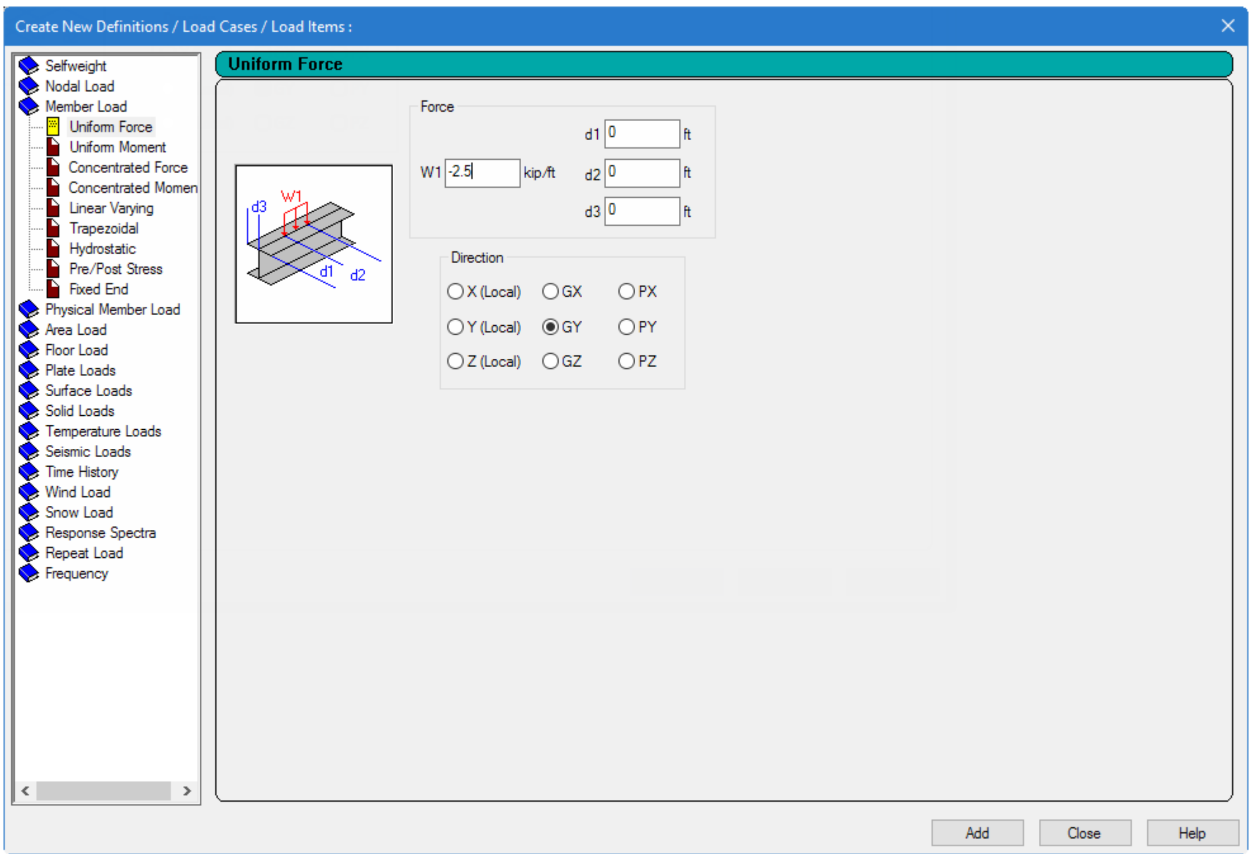
   c. Click **Close**.
4. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

The **Create New Load Items** dialog opens with entries for adding load items to the current load case.

5. Define the uniform load for this load case:
   a. In the **Create New Load Items** dialog, select the **Uniform Force** tab under the **Member Load** item.
   b. Select **GY** as the **Direction**.
   c. Type `-2.5` in the **W1** field (magnitude of the uniform force).

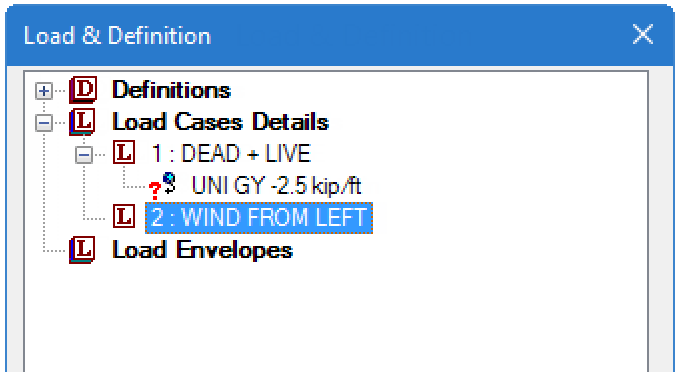
      The negative value indicates the load acts "down".

      Leave the offset fields (d1 through d3) as zero.
   d. Click **Add**.
   e. Click **Close**.

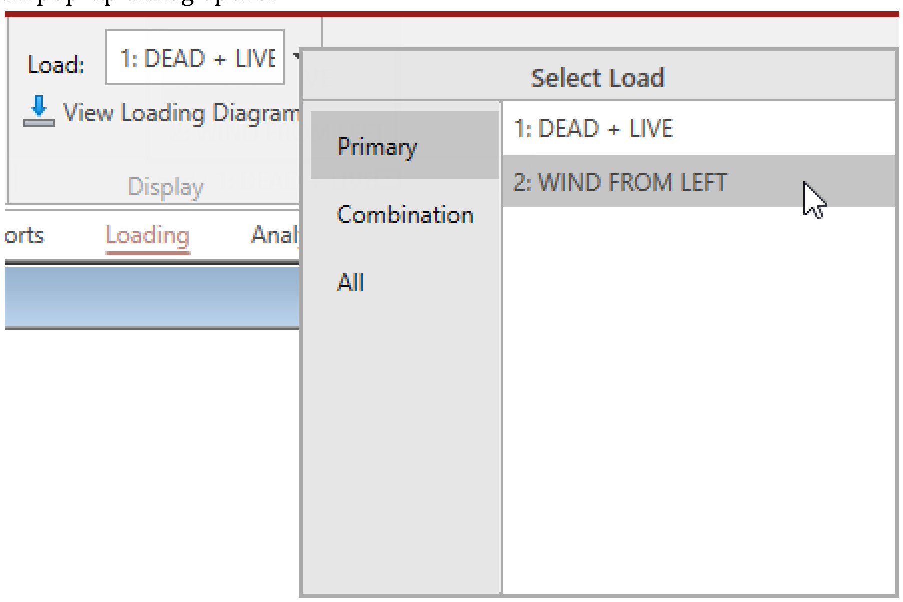


6. Repeat steps 2 and 3 to create the second load case, except title the load case as `WIND FROM LEFT`.

   Again, leave the Loading Type as **None**.



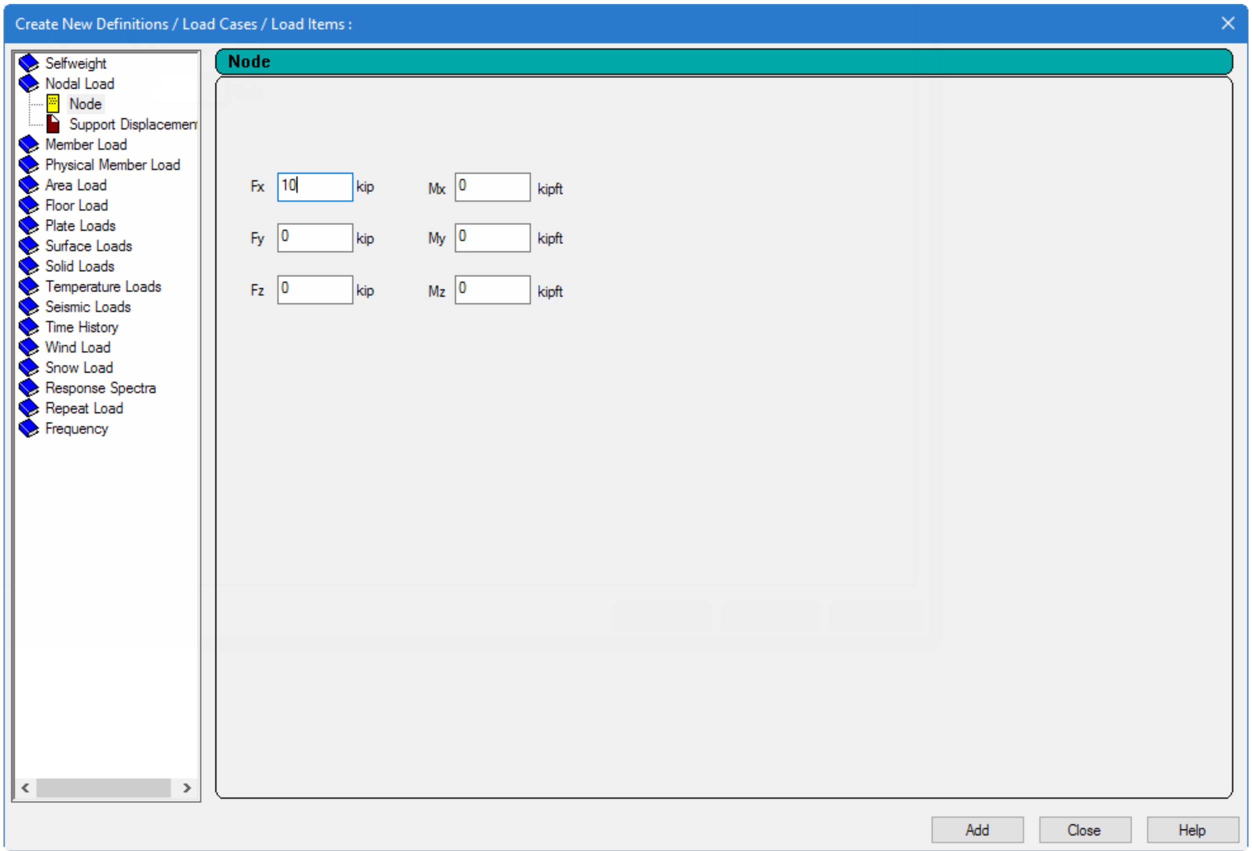
7. Set the wind load case as the current load case:
   a. On the **Loading** ribbon tab, click the down arrow on the **Load** tool in the **Display** group.
   The **Select load** pop-up dialog opens.



   b. Select the **2: Wind From Left** primary load case.

   Load items created using the ribbon tab tools are added to the current load case.
8. On the **Loading** ribbon tab, select the **Load Items** in the **Loading Specifications** group.

   The **Create New Load Items** dialog opens with entries for adding load items to the selected load case.
9. Define the wind load at the joint:
   a. In the **Create New Load Items** dialog, select the **Node** tab under the **Nodal Load** item.
   b. Type `10` in the **Fx** field (magnitude of the force along the global X direction).
   c. Click **Add**.
   d. Click **Close**.

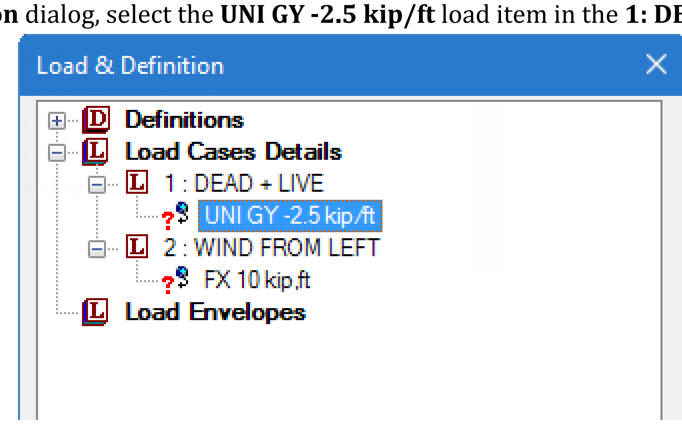


T.1 Assigning load cases to members

Load item assignments are made using the **Load & Definition** dialog, which opens when most of the Loading tools are selected. If this dialog is not currently open, select the Loading page in the Analytical Modeling page control bar to re-open it.

You may notice that the load items in the Load & Definition dialog are listed with an ? icon. This indicates that these load items are not yet assigned to any model objects (i.e., beams or nodes).

1. In the **Load & Definition** dialog, select the **UNI GY -2.5 kip/ft** load item in the **1: DEAD + LIVE** load case .



2. Select the **Use Cursor to Assign** option in the **Assignment Method** group.

3. Click **Assign**.

   The mouse pointer changes to 
4. Click on member 2.

   The uniform load is drawn on member 2 and the load item icon changes to  in the **Load & Definition** dialog.

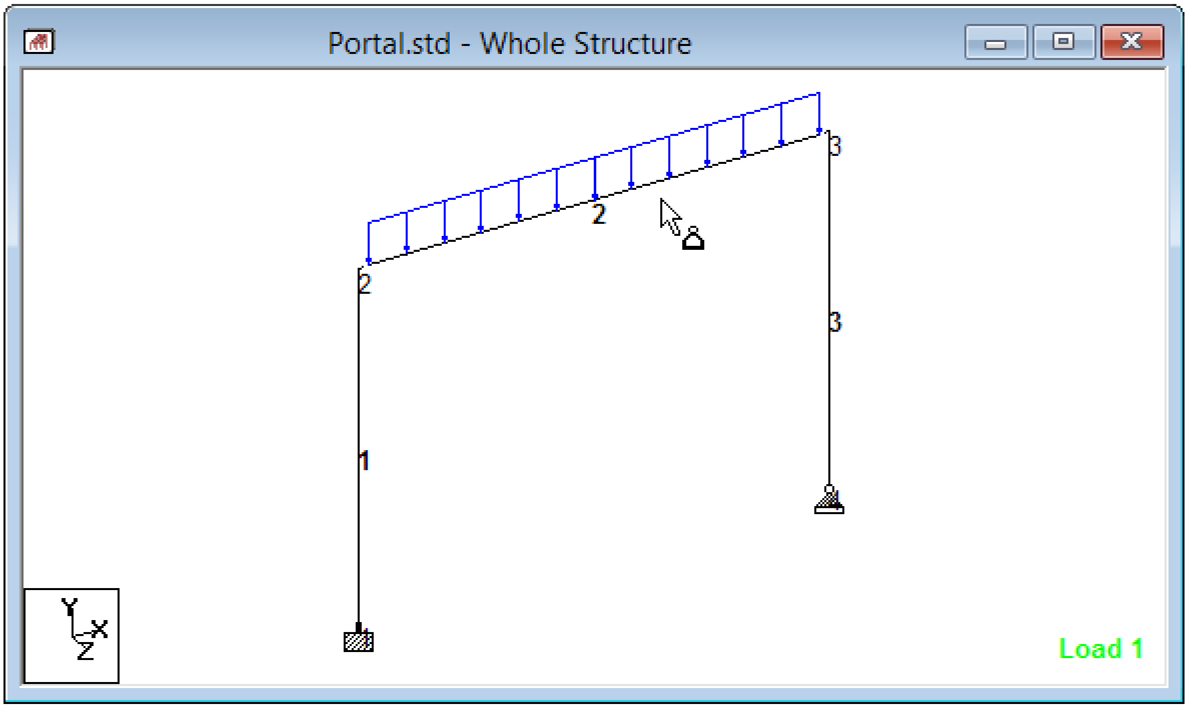


Figure 26: Uniform load item assigned to member 2

5. Repeat steps 1 through 4, except to assign the nodal load item in the second load case (**FX 10 kip, ft**) to Node 2.

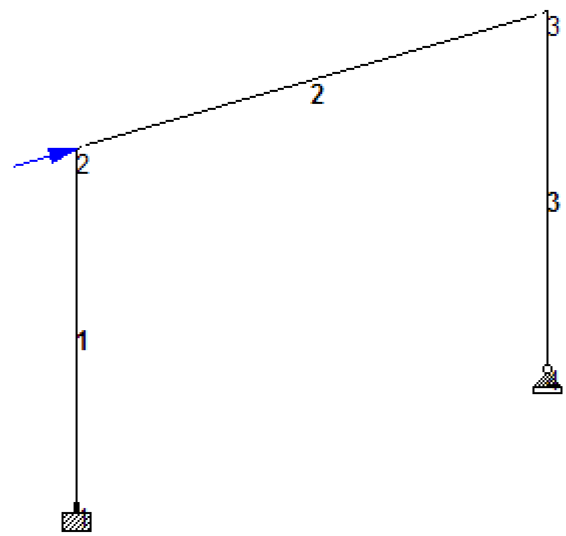


Figure 27: Model with the nodal load item assigned to node 2

6. To stop assignment load items:

   click **Assigning**

   or

   Press the **<Esc>** key.

What about the load combinations? These do not need to be assigned to any model objects. As you may recall, these represent a factored combination of the *results* of the primary load cases and thus are not directly analyzed.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Creating Load Case 3

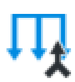
In this procedure, you will add a load combination of the first two primary load cases.

If you started your model in STAAD.Pro Physical Modeler, first select the **Loading** page in the Analytical Modeling page control bar. The **Load & Definition** dialog opens on the right side of the program window.

Here you will see that the two load groups you created inSTAAD.Pro Physical Modeler have each been added as primary load cases to the analytical model. The loads you added to the physical model are also transferred as load items in the respective load cases.

Load case 3 is a load combination. This means that the results from the primary load cases will be combined as directed by this load case definition.

1. On the **Loading** ribbon tab, select the **Combination Load Case** tool in the **Loading Specifications** group.

   The **Create New Load Combinations** dialog opens.

2. Type `75 Percent of [DL+LL+WL]` in the **Name** field.
3. Set the **Type** option to **Normal**.

   This combines the loads algebraically.

   The other combination types available are called SRSS (square root of sum of squares) and ABS (Absolute). The SRSS type offers the flexibility of part SRSS and part Algebraic. That is, some load cases are combined using the square root of sum of squares approach, and the result is combined with other cases algebraically, as in:

   $$A + \sqrt{B^2 + C^2}$$

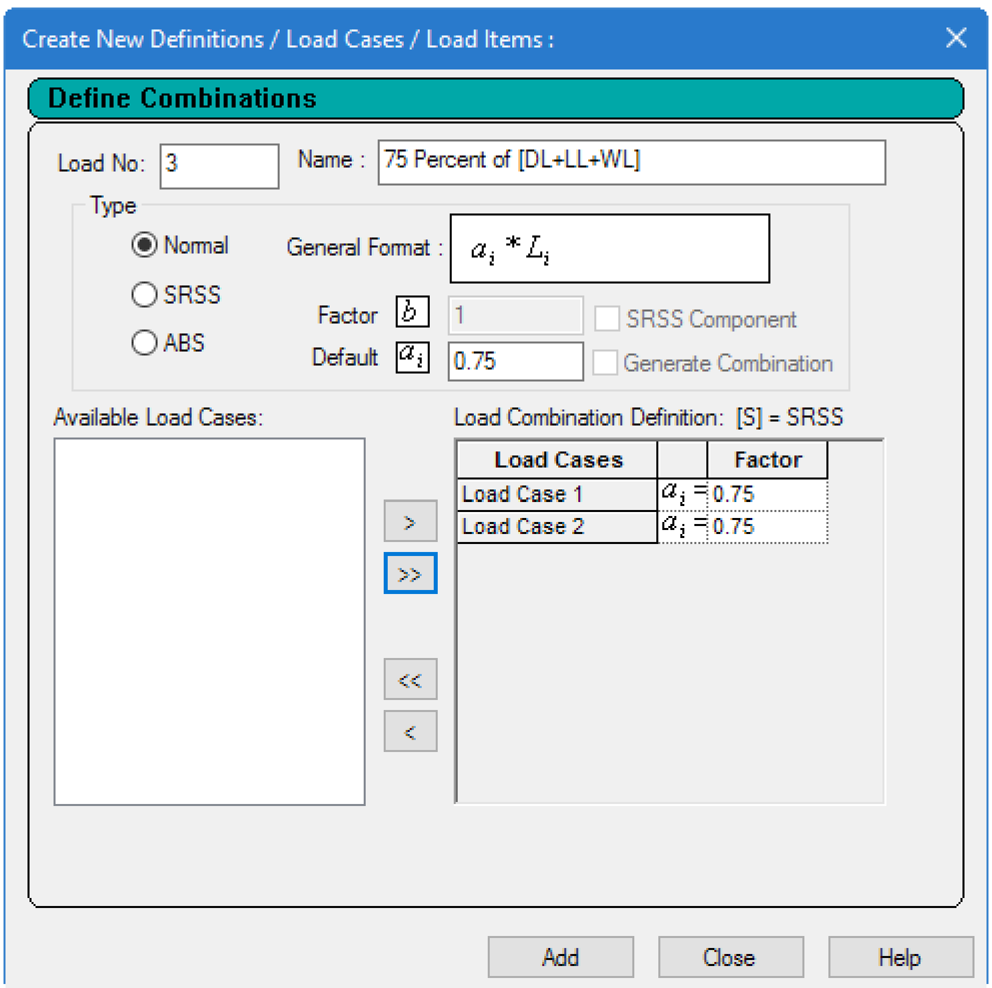
   where

   A, B, and C = the individual primary load cases
4. Type `0.75` in the **Default α1** field.

   This is used as the default factor for normal load combinations.
5. Click **[>>]**.

   Both primary load cases are added to the **Load Case Definition** list using the default factor.

   This indicates that the results of the two load cases will be multiplied by 0.75 and then summed to produce load combination results. Note that such a load combination method should only be used for linear analysis.



6. Click **Add**.
   The load combination is added to the **Load Cases Details** section in the **Load & Definition** dialog as load 3.
7. Click **Close**.

T.1 Analysis and Design

The following procedures describe how to add analysis and design commands to the STAAD input file.

Note: The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

T.1 Specifying the analysis type

A linear, static analysis is required for this model. You can also instruct STAAD.Pro to provide a static equilibrium report.

The STAAD input file commands generated are:

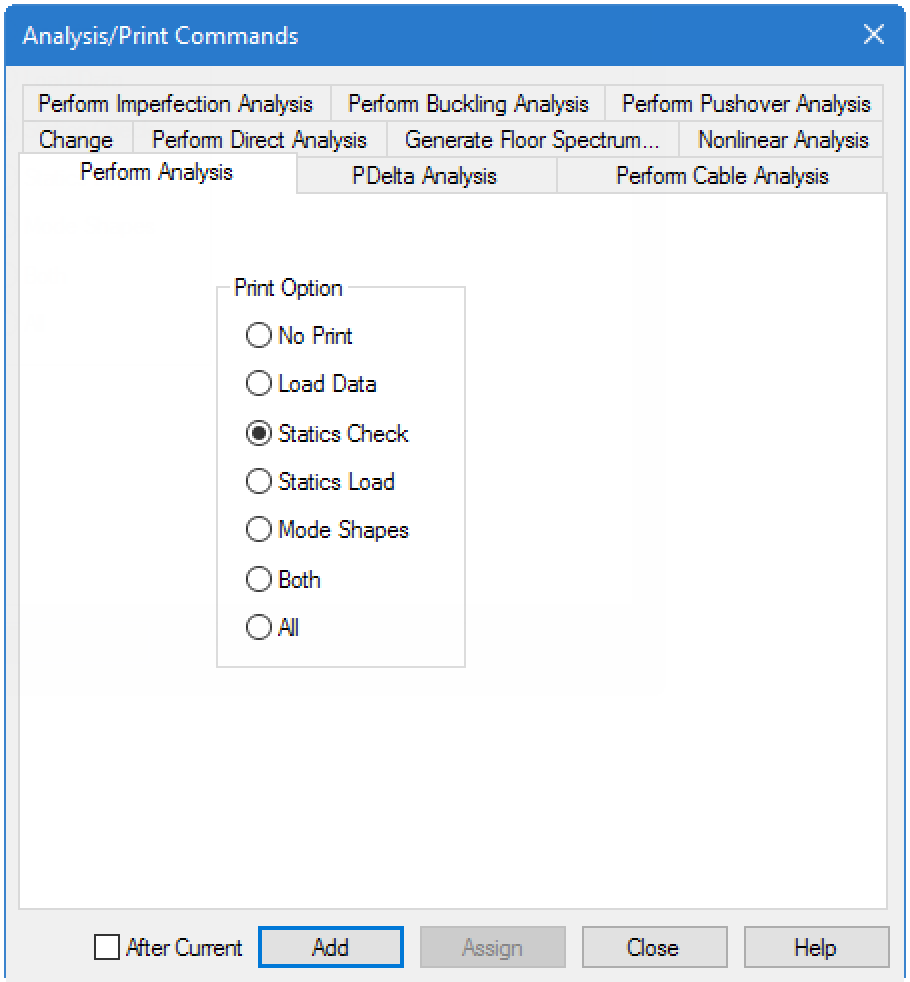
```
PERFORM ANALYSIS PRINT STATICS CHECK
```

1. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

   The **Analysis/Print Commands** dialog opens.

   Tip: If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.
2. In the **Analysis/Print Commands** dialog, select the **Perform Analysis** tab.



3. Select the **Statics Check** option.

 Note: In response to this option, a report consisting of the summary of applied loading and summary of support reactions, for each load case, will be produced in the STAAD output file.

4. Click **Add**.

 Tip: The command is added to the list of commands outlined in the **Analysis - Whole Structure** dialog on the right-hand side of the window. The command has a green checkmark to indicate that it is valid. You can review the outline of the command input structure here.

5. Click **Close**.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Specifying post-analysis print commands

Once an analysis has been completed, the program can then provide results of the analysis in the output. In particular, you will request member end forces and support reactions written to the output file.

The STAAD input file commands generated are:

```
PRINT MEMBER FORCES ALL
PRINT SUPPORT REACTION LIST 1 4
```

1. Select all the beam members by one of the following methods:

   on the **Select** ribbon tab, select the **All** tool in the **Beams** group

   or

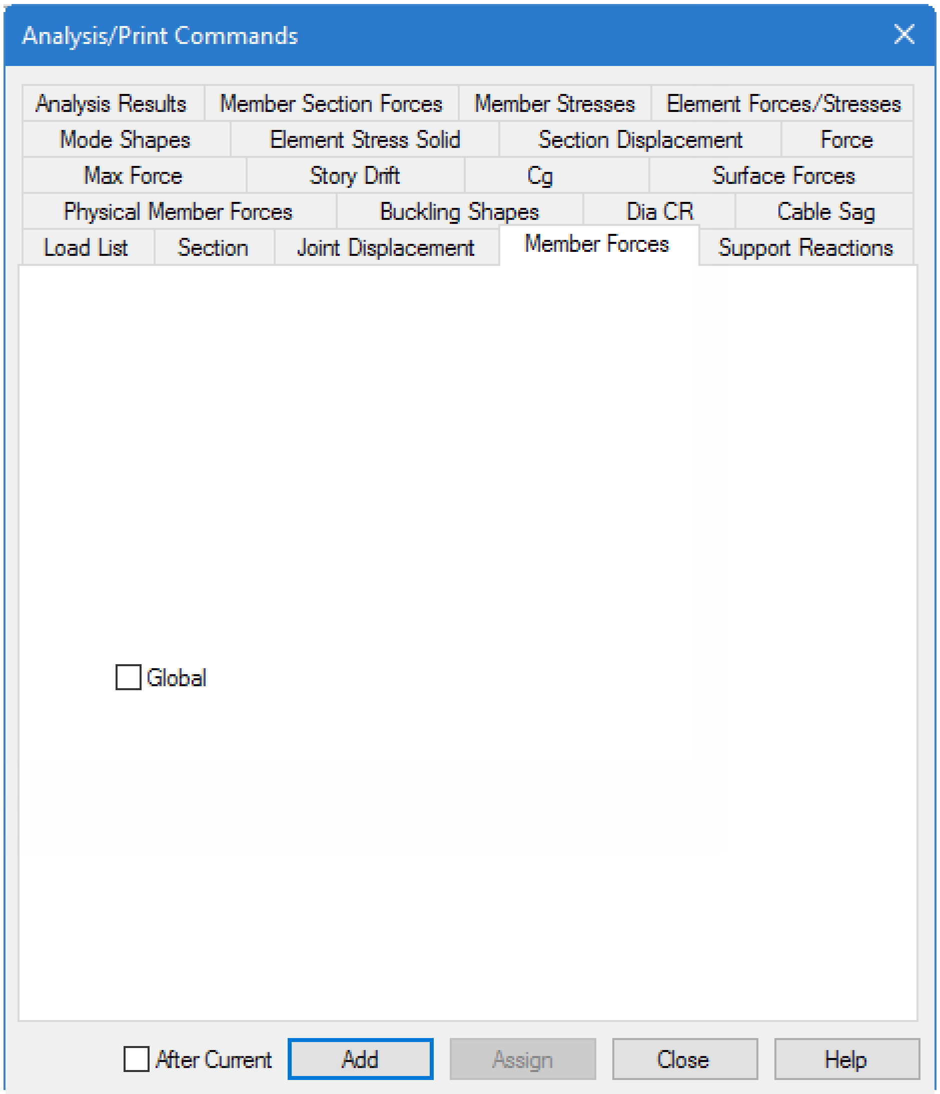
   click-and-drag a window around all members in the View window

   or

   press **<Ctrl+A>**
2. On the **Analysis and Design** ribbon tab, select the **Post-Analysis Commands** tool in the **Analysis Data** group.

3. On the **Post Analysis Print - Whole Structure** dialog, click **Define Commands**.
   The **Analysis/Print Commands** dialog opens.
4. Add the member forces results to the output:
   a. Select the **Member Forces** tab.



b. Click **Assign**.

Tip: Since you first selected the members, you can assign the command to those members directly from this dialog. Otherwise, you would add the command and then assign it —similar to how you assigned load case items— to those members.

c. Click **Close**.

5. On the **Select** ribbon tab, select the **Supports** tool in the **Nodes** group.

The supported nodes (nodes 1 and 4) are both selected.

6. Repeat steps 2 and 3 to open the **Analysis/Print Commands** dialog for the post-print commands.
7. Add the support reactions to the output:

a. Select the **Support Reactions** tab.

b. Click **Assign**.
c. Click **Close**.

The command is added to the list of commands.

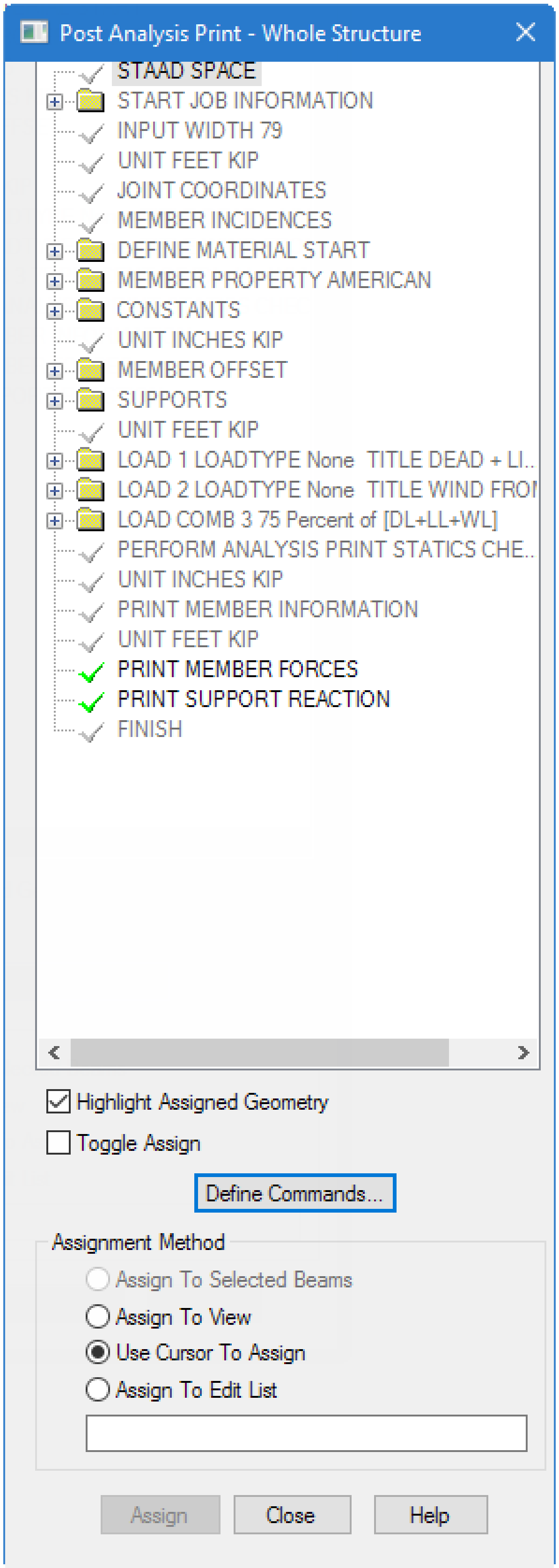


*Figure 28: The **Post Analysis Print** dialog after the post-analysis print commands are added and assigned.*

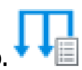
Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Short-listing the load cases to be used in steel design

The steel design has to be performed for load cases 1 and 3 only per the specification at the beginning of this tutorial. To instruct the program to use just these cases, and ignore the remaining, you will use the `LOAD LIST` command.

The STAAD input file commands generated are:

```
LOAD LIST 1 3
```

1. On the **Anlaysis and Design** ribbon tab, select the **Load List** tool in the **Analysis Data** group. 

   The **Load List** dialog opens.

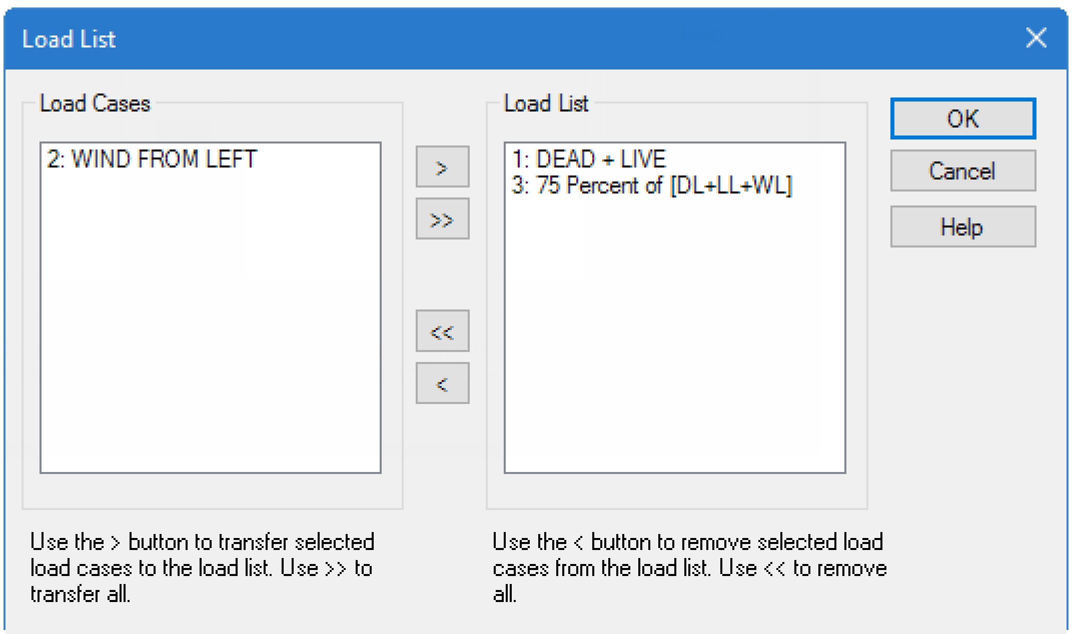


Figure 29:

2. From the **Load Cases** list box on the left, double-click on `1: DEAD + LIVE` and `3: 75 Percent of [DL+LL +WL]` to add these loads to the **Load List** box on the right.
3. Click **OK**.

T.1 Specifying steel design parameters

Designing members per a building code as specified for this tutorial require you to provide values for some of the terms where the default values do not match the problem requirements.

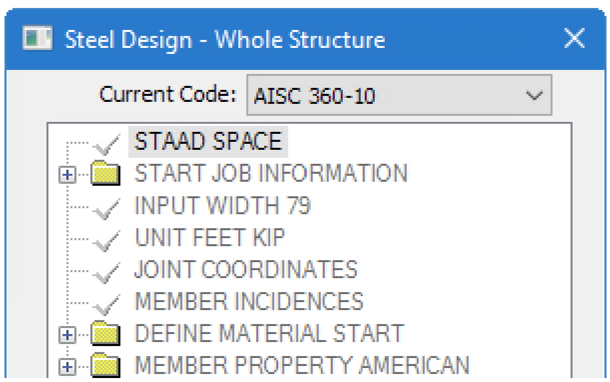
Note: This tutorial uses the 2010 edition of the American Institute of Steel Construction 360 specification for design (AISC 360-10). Details on the code parameters can be found in D1. American Codes (on page 1086) section.

The STAAD input file commands generated are:

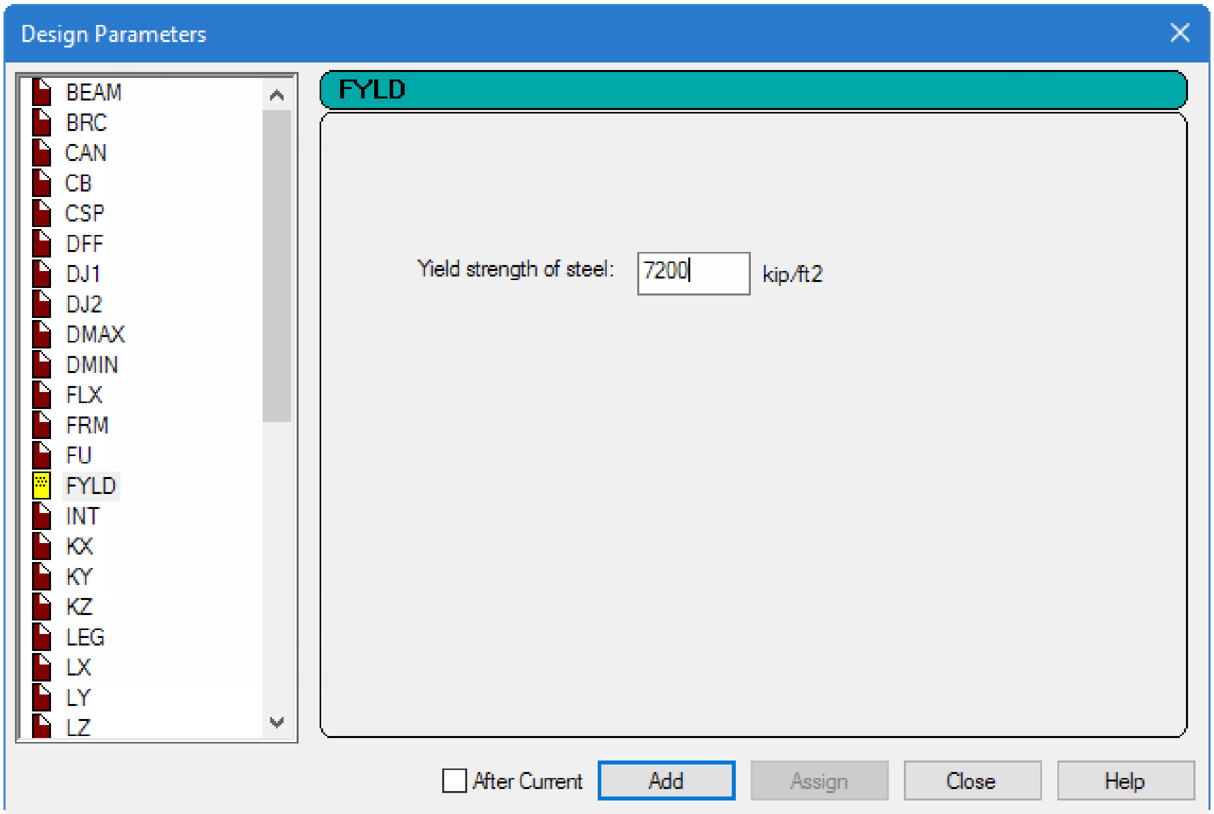
```
PARAMETER
CODE AISC UNIFIED 2010
FYLD 7200 ALL
TRACK 2 MEMB 2 3
UNB 10.0 MEMB 2 3
UNT 10.0 MEMB 2 3
SELECT MEMB 2 3
```

1. On the **Analysis and Design** ribbon tab, select **Steel** in the **Design Commands** gallery.

2. In the **Steel Design - Whole Structure** dialog, select **AISC 360-10** in the **Current Code** list.



3. In the **Steel Design - Whole Structure** dialog, click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Specify the yield strength parameter:
   a. Select the **FYLD** parameter tab.



b. Type `7200` (kip/ft^2) [50 (kip/in^2)]in the **Yield strength of steel** field.

> **Tip:** This equates to a 50 ksi steel, which is the common strength for American wide flange shapes. You can change the input units before and after this command to use inches for convenience, but this tutorial uses a non-standard unit here for brevity.

c. Click **Add**.
The parameter list is added to the list of commands in the **Steel Design - Whole Structure** dialog, including the selected design code and the yield strength value.

5. To define the remaining parameters, repeat step 4 except for selecting the parameters and providing the values listed below.

Parameter	Value
TRACK	**2** (select the option)
UNB	`10` (ft) [120 (in)]
UNT	`10` (ft) [120 (in)]

6. Click **Close** in the *Design Parameters* dialog.

> **Note:** The steel design parameters are all marked with an icon. This indicates that they need to be assigned to steel members.

7. Assign the yield strength to all members:

a. Select the **FYLD 7200** parameter in the command list.

   b. Select the **Assign to View** option in the Assignment Method group.
   c. Click **Assign**.
   A dialog opens confirming you want to assign to the view (all members currently displayed in the view window).
   d. Click **Yes**.

8. Assign the output detail (TRACK) parameter to members 2 and 3:
   a. Select the **TRACK 2** parameter in the command list.
   b. Select the **Use Cursor to Assign** option in the Assignment Method group.
   c. Click **Assign**.
   The mouse pointer changes to a .
   d. Click on each member in the Frame.
   e. Click on **Assigning** to stop assigning design parameters.

9. Repeat step 8 to assign the other two parameters (UNB and UNT) to members 2 and 3, as well.

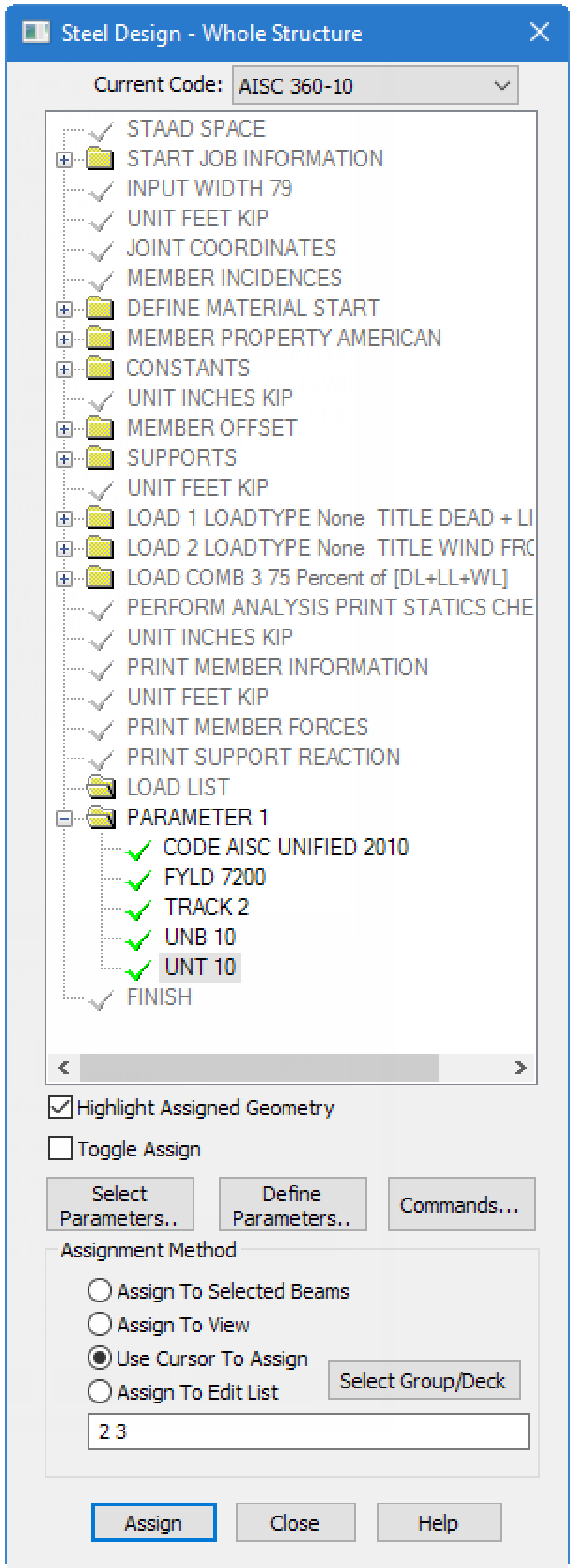


*Figure 30: The **Steel Design - Whole Structure** dialog after the design parameters have been added and*

assigned

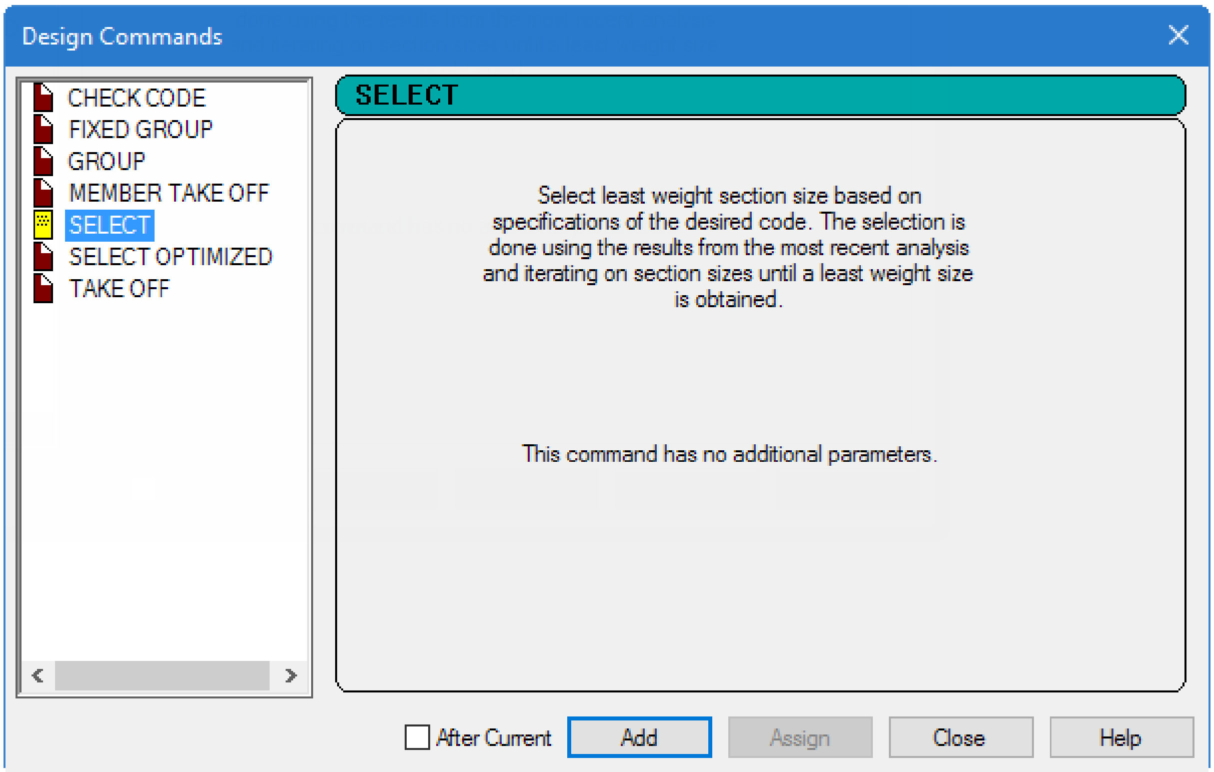
10. Add a command to instruct select sizes for members 2 and 3:

The select command is an instruction to the program to assign the least-weight cross-section which satisfies all the code requirements for the member.

a. In the **Steel Design - Whole Structure** dialog, click **Commands**.

The **Design Commands** dialog opens.

b. Select the **Select** tab.



c. Click **Add**.

d. Click **Close**.

11. Assign the select command to members 2 and 3:

a. Select the **SELECT** parameter in the command list.

b. Select the **Assign to Edit List** option in the Assignment Method group.

c. Type 2 3 in the list.

Notice this is a space-separated list of member numbers.

d. Click **Assign**.

Tip: You may also use either the method of selecting the members first or using the cursor to assign the command to the members.

After the parameters are assigned, click anywhere in the drawing area to deselect the members.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Re-specifying the analysis command

When the analysis and design engine executes the member selection operation you specified in the previous step, a new set of properties will end up being assigned to those members. This changes the stiffness distribution for the entire structure. Since the structure is statically indeterminate, it should be analyzed again to determine the correct the nodal displacements, member forces, etc. to reflect this new stiffness distribution.

The STAAD input file commands generated are:

```
PERFORM ANALYSIS
```

1. Select the **Select** command in the **Analysis - Whole Structure** dialog outline of the model commands.
2. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

   The **Analysis/Print Commands** dialog opens.

   > **Tip:** If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.
3. Select the **Perform Analysis** tab.
4. Select the **No Print** option.

   > **Tip:** The statics check report does need to be repeated.
5. Check the **After Current** option.

   This allows you to specify that where the new command is added. This will place the new Perform Analysis *after* the command selected in Step 1, instead of in the first valid point within the model file.
6. Click **Add**.
7. Click **Close**.

> **Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Re-specifying the TRACK parameter

The final calculation we need to do is make sure the current set of member properties pass the code requirements based on the up-to-date member forces.

This will require that we do a code checking operation again. To restrict the output produced to a reasonable level, we specify the TRACK parameter again but use a different value.

The STAAD input file commands generated are:

```
PARAMETER
CODE AISC UNIFIED 2010
TRACK 0 ALL
```

1. On the **Analysis and Design** ribbon tab, select **Steel** in the **Design Commands** gallery.

2. In the **Steel Design - Whole Structure** dialog, click **Define Parameters**.
   The **Design Parameters** dialog opens.
3. Specify the track parameter:
   a. Select the **TRACK** parameter tab.
   b. Select option **0**.

   > **Note:** The TRACK 0 command instructs the program to only provide the controlling limit state check for each member assigned to this command.

   c. Click **Add**.
   The parameter list is added to the list of commands in the **Steel Design - Whole Structure** dialog, including the selected track parameter and value.
   d. Click **Close**.
4. Select the **TRACK 0** parameter entry in the list of commands.
5. Select all the beam members by one of the following methods:

   on the **Select** ribbon tab, select the **All** tool in the **Beams** group

   or

   click-and-drag a window around all members in the View window

   or

   press **<Ctrl+A>**
6. In the **Steel Design - Whole Structure** dialog, select the **Assign To Selected Beams** option.
7. Click **Assign**.
   A message dialog prompting you to confirm the assignment.
8. Click **Yes**.

After the parameters are assigned, click anywhere in the drawing area to deselect the members.

> **Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Specifying the CHECK CODE command

As part of the iterative process of analysis and design, you must perform a code check on the selected members after they have been analyzed. This ensures that the new set of member forces do not exceed the capacity of the selected members.

A code checking operation which uses the up-to-date cross sections of the members and the latest member forces is used to evaluate that the members have sufficient capacity per the code specifications.

The STAAD input file commands generated are:

```
CHECK CODE ALL
```

1. In the **Steel Design - Whole Structure** dialog, click **Commands.**

   The **Design Commands** dialog opens.
2. Select the **Check Code** tab.

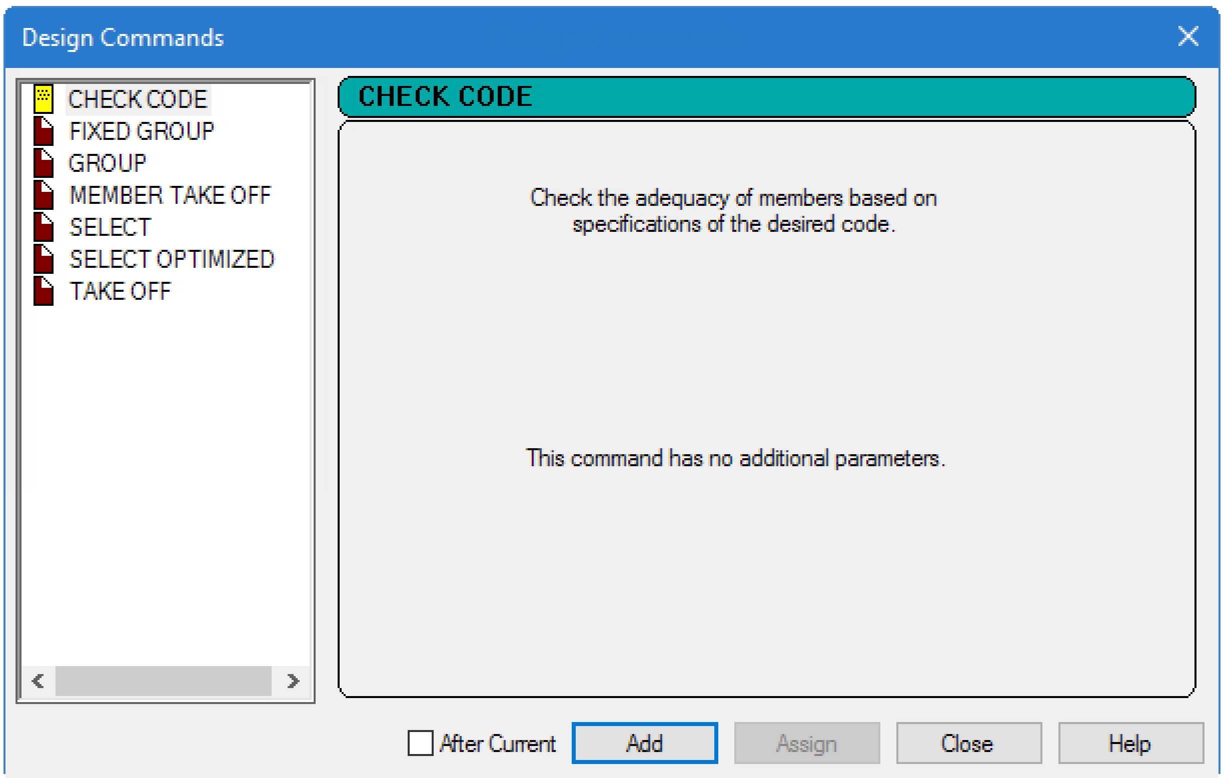


Figure 31:

3. Click **Add**.
4. Click **Close**.
5. Select the unassigned **CHECK CODE** entry in the **Steel Design - Whole Structure** dialog.
6. Select the **Assign to View** option in the **Assignment Method** group.
7. Click **Assign.**
   A message dialog prompting you to confirm the assignment.
8. Click **Yes**.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.1 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the **Utilities** ribbon tab, select the **Command File** tool in the **Edit** group.

The STAAD.Pro Editor opens with the contents of the input file.

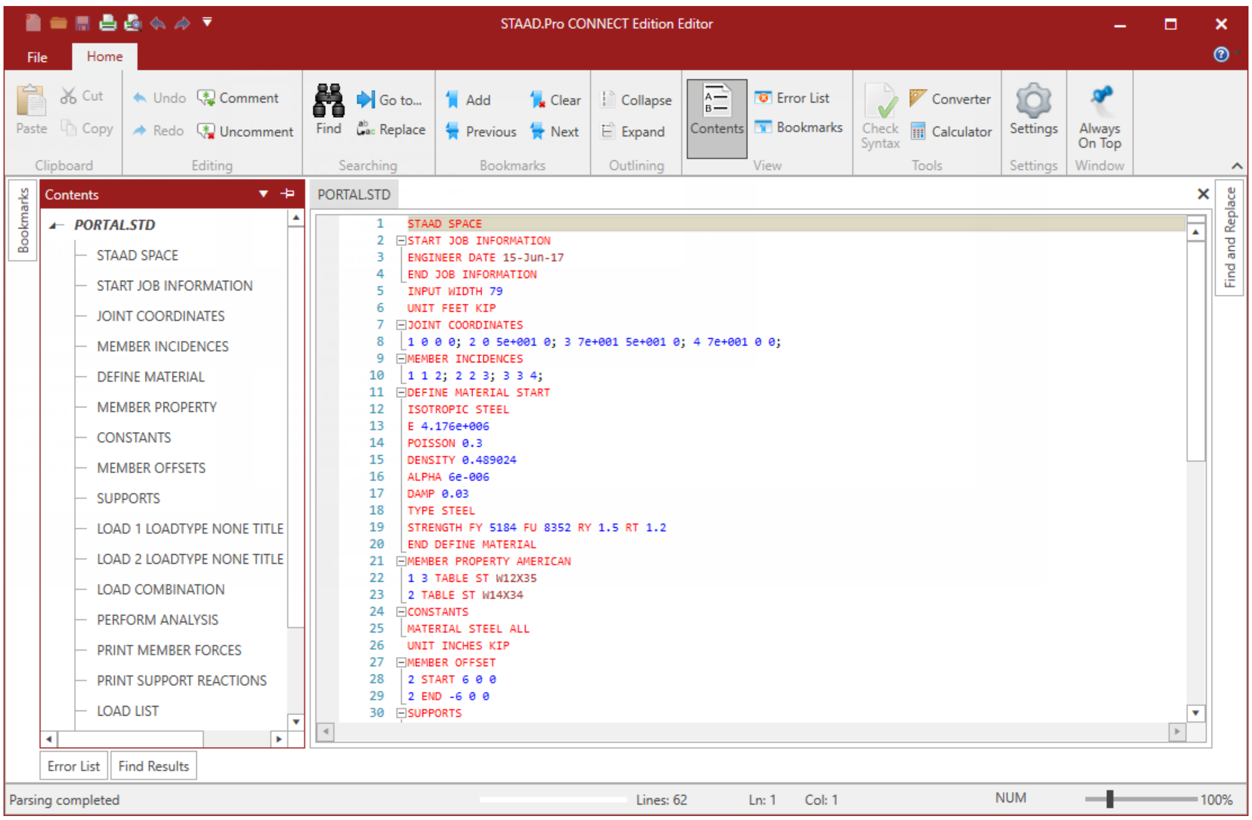


Figure 32: The STAAD.Pro Editor window

2. (Optional) You modify the data of the structure in this editor if necessary.
3. Select **File** > **Exit Editor** in the STAAD.Pro Editor window to close.

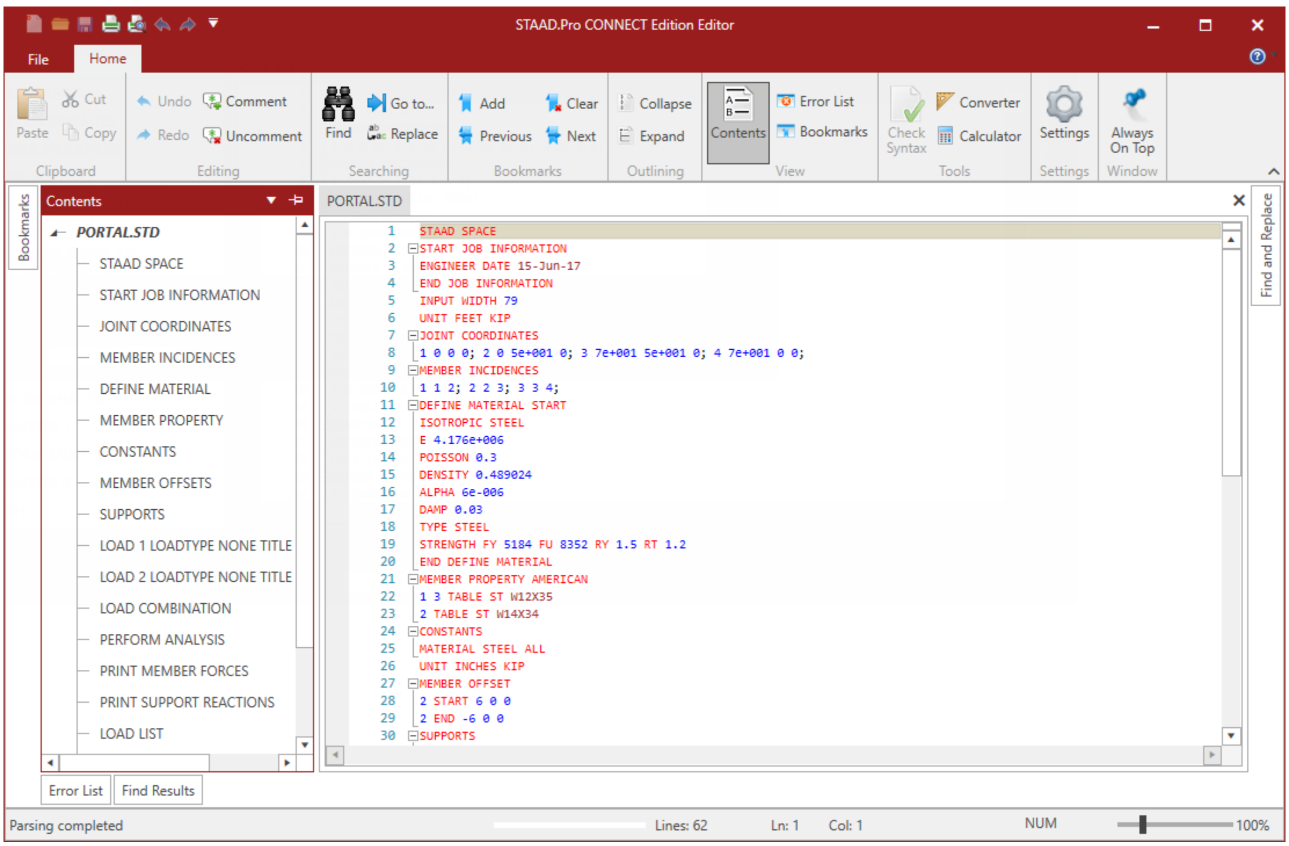
As stated in " T.1 Methods of creating the model (on page 387)," you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section.

If you want to skip that part, proceed to " T.1 Performing Analysis/Design (on page 446) " where you will perform the analysis and design on this model.

T.1 Creating the model using the command file

As an alternatively to the procedures described in the proceeding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

Note: A STAAD input file is a plain text file that uses the `.std` file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.



To start a new STAAD input file using the STAAD.Pro Editor, follow the procedure described in T.1 Creating a new structure (on page 389). Then select the **Command File** tool in the **Edit** group on the **Utilities** ribbon tab. The STAAD.Pro Editor window opens with the basic commands for your model entered.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are *not* case sensitive (i.e., they may be typed in upper *or* lower case letters). By convention, this and most input files use all caps, though.

For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain

English for later reference. By convention, the required letters in a command or keyword are underlined here ("PLANE" = "PLA" = "plane" = "pla").

```
STAAD PLANE PORTAL FRAME
```

Every STAAD input file has to begin with the word STAAD. The word PLANE signifies that the structure is a plane frame (in the XY plane). The remainder of this line is the title of the problem, which is optional.

Note:

If a line is typed with an asterisk in the first column, it signifies that the line is a comment line and should not be executed. For example, one could have put the optional title above on a separate line as follows:

```
* PORTAL FRAME
```

```
UNIT FEET KIP
```

Specify the force and length units for the commands to follow.

```
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
```

Joint numbers and their corresponding global X and Y coordinates are provided above. For example, 3 20 15. indicates that node 3 has an X coordinate of 20 ft and a Y coordinate of 15 ft. Note that the reason for not providing the Z coordinate is because the structure is a plane frame. If this were a space frame, the Z coordinate would also be required.

Semicolons (;) are used as line separators. In other words, data which is normally put on multiple lines can be put on one line by separating them with a semicolon.

```
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```

The members are defined by the joints to which they are connected.

```
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X35
2 TABLE ST W14X34
```

Members 1 and 3 are assigned a W12X35 section from the built-in AMERICAN steel table. Member 2 has been assigned a W14X34. The word ST stands for standard single section. Refer to subsections 1 through 5 of TR.20 Member Property Specification (on page 2255) for details on the convention for assigning member property names.

```
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

And to use this material definition for all members, add the following:

```
CONSTANTS
MATERIAL STEEL ALL
```

The length unit is changed from FEET to INCHES to use familiar units for most of the material definition. For this example, you use a set of built-in values for steel, so the units are only shown for convenience here. In the user interface, units were not changed until the following commands were generated. See TR.26.1 Define Material (on page 2303) for more information.

```
MEMBER OFFSET
2 START 6.0 0. 0.
2 END -6.0 0. 0.
```

The beam member is physically connected to the 2 columns at the face of the column, and not at the column centerline. This creates a rigid zone, about half the depth of the columns, at the 2 ends of the beam 2. This rigid zone is taken advantage of using member offsets (It is you choice whether or not you wish to use these). So, the above commands define that member 2 is eccentrically connected or OFFSET at its START joint by 6 inches in the global X direction, 0.0 and 0.0 in Y and Z directions. The same member is offset by negative 6.0 inches at its END joint. See TR.25.1 Member Offset Specification (on page 2296) for more information.

```
PRINT MEMBER INFORMATION ALL
```

The information that is printed by this command includes start and end joint numbers (incidence), member length, beta angle and member end releases.

```
SUPPORTS
1 FIXED ; 4 PINNED
```

A fixed support is located at joint 1 and a pinned support (fixed for translations, released for rotations) at joint 4. More information on the support specification is available in TR.27 Support Specifications (on page 2315).

```
UNIT FT
```

The length unit is changed to FEET to facilitate input of loads.

```
LOADING 1 DEAD + LIVE
MEMBER LOAD
2 UNI GY -2.5
```

The above commands identify a loading condition. DEAD + LIVE is an optional title to identify this load case. A Uniformly distributed MEMBER LOAD of 2.5 kips/ft is acting on member 2 in the negative global Y direction. Member Load specification is explained in TR.32 Loading Specifications (on page 2461).

```
LOADING 2 WIND FROM LEFT
JOINT LOAD
2 FX 10.
```

The above commands identify a second load case. This load is a JOINT LOAD. A 10 kip force is acting at joint 2 in the global X direction.

```
LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL)
1 0.75 2 0.75
```

This command identifies a combination load with an optional title. The second line provides the components of the load combination case - primary load cases and the factors by which they should be individually multiplied.

```
PERFORM ANALYSIS PRINT STATICS CHECK
```

This command instructs the program to proceed with the analysis and produce a report of static equilibrium checks. TR.37 Analysis Specification (on page 2620) offers information on the various analysis options available.

```
PRINT MEMBER FORCES ALL
PRINT SUPPORT REACTION LIST 1 4
```

The above print commands are self-explanatory. The member forces are in the member local axes while support reactions are in the global axes.

```
LOAD LIST 1 3
PARAMETER 1
CODE AISC UNIFIED 2010
FYLD 7200 ALL
TRACK 2.0 MEMB 2 3
UNB 10.0 MEMB 2 3
UNT 10.0 MEMB 2 3
SELECT MEMBER 2 3
```

The above sequence of commands is used to initiate the steel design process. The command PARAMETER is followed by the various steel design parameters. Parameters are specified typically when their values differ from the built-in program defaults. Specifications of the AISC Unified Code LRFD specification are to be followed. A parameter list for the AISC code is available in D1.A.6 Design Parameters (on page 1100). Member numbers 2 and 3 have 10 ft unsupported length for the top and bottom flange (UNT and UNB).

UNT and UNB are used to compute the allowable compressive stress in bending. The yield strength, FYLD, of steel is specified as 7,200 ksf (50 ksi) since it is different from the default value of 36 ksi and it is assigned to ALL members. The TRACK parameter controls the level of description of the output, 2.0 being the most detailed. The LOAD LIST command lists the load cases (1 and 3) to be used in the design. The SELECT MEMBER command asks the program to evaluate the most economical section for members 2 and 3 in the context of the above analysis.

```
PERFORM ANALYSIS
```

When the analysis and design engine executes the member selection operation specified in the member selection , a new set of properties will end up being assigned to those members. This has the effect of changing the stiffness distribution for the entire structure. Since the structure is statically indeterminate, it is best practice to re-analyze it to determine the accurate nodal displacements, member forces, etc. which reflect this new stiffness distribution. The above command instructs the program to do another cycle of analysis.

```
PARAMETER 2
TRACK 0 ALL
```

The TRACK parameter is specified again in a new PARAMETER set. It controls the level of information produced in the steel design output. This time, the value of 0 is used to provide a pass or fail status for each member.

```
CHECK CODE ALL
```

The analysis operation carried out earlier will create a new set of member forces. These forces will very likely be different from those which were used in the member selection operation. Consequently, you should verify that the structure is safely able — from the standpoint of the design code requirements — to carry these new forces. A code checking operation, which uses the up-to-date cross sections of the members, and the latest member forces, will provide a status report on this issue.

```
FINISH
```

A STAAD run is terminated using the FINISH command.

Save the input file and close the editor. The model is opened in the STAAD.Pro interface.

This concludes the session on generating the model as a command file using the built-in editor. If you wish to perform the analysis and design, you may proceed to the T.1 Performing Analysis/Design (on page 446). The post-processing facilities are explained in T.1 Post-Processing (on page 455).

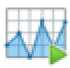
Caution: Remember that without successfully completing the analysis and design, the post-processing facilities will not be accessible.

T.1 Performing Analysis/Design

STAAD.Pro performs Analysis and Design simultaneously.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group. .



The **STAAD Analysis and Design** dialog opens and displays messages of the analysis progress.

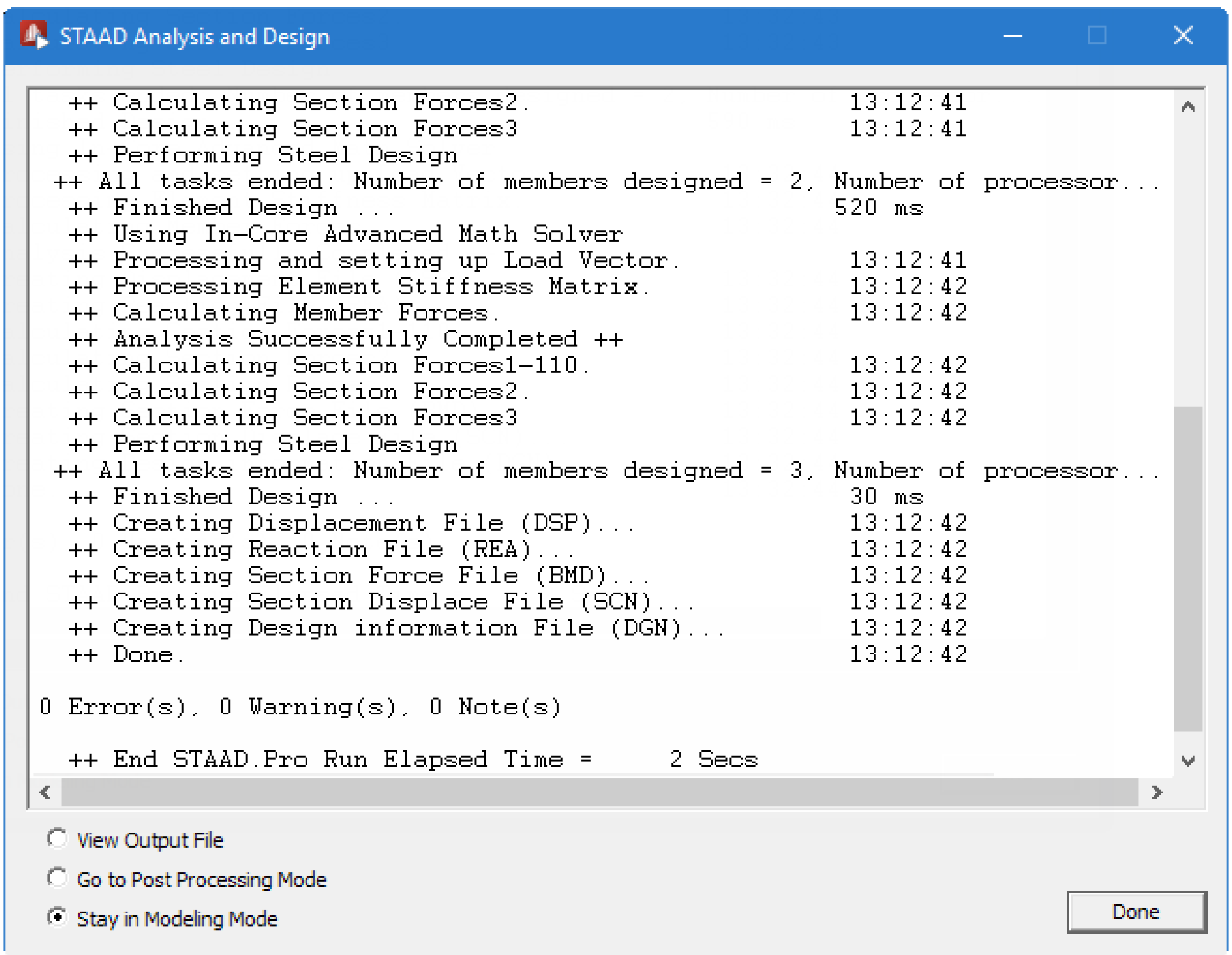


Figure 33:

2. Select the **View Output File** option once the analysis and design are complete.

   The three options are indicative of what will happen after you click **Done.**

View Output File This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the model generation process. It also provides you with important messages of any errors were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 447) for details on viewing and understanding the contents of the output file.

Go to Post Processing Mode This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 455) for details on the Post processing mode.

Stay in Modeling Mode This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click **Done**.
   The STAAD.Pro Output Viewer window opens.

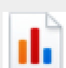
T.1 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

Note:

If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the **Utilities** ribbon tab, select the **Analysis Output** tool in the **View** group.



Tip: By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Set Echo** tool option from the menu bar and the select the **Echo Off** option in the **Set Echo** dialog.

It is *strongly recommended* that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
                                                      PAGE NO.    1
          ****************************************************
          *                                                  *
          *          STAAD.Pro CONNECT Edition               *
          *          Version  22.10.00.***                   *
          *          Proprietary Program of                  *
          *          Bentley Systems, Inc.                   *
          *          Date=    MAR 24, 2022                   *
```

```
            *           Time=     9:43:54                        *
            *                                                    *
            *  Licensed to: Bentley Systems Inc                  *
            ****************************************************
     1. STAAD PLANE PORTAL FRAME
INPUT FILE: Tutorial 1 - Steel Portal Frame.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 15-FEB-02
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT FEET KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 15 0; 3 20 15 0; 4 20 0 0
     9. MEMBER INCIDENCES
    10. 1 1 2; 2 2 3; 3 3 4
    11. MEMBER PROPERTY AMERICAN
    12. 1 3 TABLE ST W12X35
    13. 2 TABLE ST W14X34
    14. UNIT INCHES
    15. DEFINE MATERIAL START
    16. ISOTROPIC STEEL
    17. E 29000
    18. POISSON 0.3
    19. DENSITY 283E-006
    20. ALPHA 6E-006
    21. DAMP 0.03
    22. TYPE STEEL
    23. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    24. END DEFINE MATERIAL
    25. CONSTANTS
    26. MATERIAL STEEL ALL
    27. MEMBER OFFSET
    28. 2 START 6 0 0
    29. 2 END -6 0 0
    30. PRINT MEMBER INFORMATION ALL
  MEMBER   INFORMAT ALL
      PORTAL FRAME                                              -- PAGE NO.    2
   MEMBER INFORMATION
   ------------------
   MEMBER    START    END         LENGTH     BETA
             JOINT   JOINT        (INCH)     (DEG)     RELEASES
      1        1       2        180.000     0.00
      2        2       3        228.000     0.00
      3        3       4        180.000     0.00
   ************ END OF DATA FROM INTERNAL STORAGE ************
    31. SUPPORTS
    32. 1 FIXED
    33. 4 PINNED
    34. UNIT FEET KIP
    35. LOAD 1 DEAD + LIVE
    36. MEMBER LOAD
    37. 2 UNI GY -2.5
    38. LOAD 2 WIND FROM LEFT
    39. JOINT LOAD
    40. 2 FX 10
    41. LOAD COMB 3 75 PERCENT OF (DL+LL+WL)
    42. 1 0.75 2 0.75
    43. PERFORM ANALYSIS PRINT STATICS CHECK
```

```
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       7
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
     PORTAL FRAME                                             -- PAGE NO.    3
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
         DEAD + LIVE
          CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.100000003E+02
                       Y =  0.150000004E+02
                       Z =  0.000000000E+00
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =         -47.50
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=           0.00  MZ=        -475.00
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          -0.00
      SUMMATION FORCE-Y =          47.50
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=           0.00  MZ=         475.00
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      1)
            MAXIMUMS    AT NODE
     X =  1.82361E-01       2
     Y = -1.46578E-02       3
     Z =  0.00000E+00       0
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ= -4.82523E-03       2
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
         WIND FROM LEFT
          CENTER OF FORCE BASED ON X FORCES ONLY (FEET).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.000000000E+00
                       Y =  0.150000004E+02
                       Z =  0.000000000E+00
     PORTAL FRAME                                             -- PAGE NO.    4
TOTAL APPLIED LOAD     2
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =          10.00
      SUMMATION FORCE-Y =           0.00
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=           0.00  MZ=        -150.00
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =         -10.00
      SUMMATION FORCE-Y =           0.00
```

```
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=           0.00  MZ=         150.00
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      2)
            MAXIMUMS    AT NODE
     X =  7.27292E-01      2
     Y =  2.47270E-03      2
     Z =  0.00000E+00      0
     RX=  0.00000E+00      0
     RY=  0.00000E+00      0
     RZ= -5.48837E-03      4
  ************ END OF DATA FROM INTERNAL STORAGE ************
   44. PRINT MEMBER FORCES ALL
 MEMBER   FORCES   ALL
     PORTAL FRAME                                             -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1     23.18     -3.99     0.00      0.00      0.00     -11.48
                2    -23.18      3.99     0.00      0.00      0.00     -48.40
          2     1     -4.10      7.68     0.00      0.00      0.00      67.93
                2      4.10     -7.68     0.00      0.00      0.00      47.32
          3     1     14.30      2.77     0.00      0.00      0.00      42.34
                2    -14.30     -2.77     0.00      0.00      0.00      -0.81
     2    1     2      3.99     23.18     0.00      0.00      0.00      36.81
                3     -3.99     24.32     0.00      0.00      0.00     -47.72
          2     2      2.32     -4.10     0.00      0.00      0.00     -45.27
                3     -2.32      4.10     0.00      0.00      0.00     -32.69
          3     2      4.73     14.30     0.00      0.00      0.00      -6.34
                3     -4.73     21.32     0.00      0.00      0.00     -60.31
     3    1     3     24.32      3.99     0.00      0.00      0.00      59.88
                4    -24.32     -3.99     0.00      0.00      0.00       0.00
          2     3      4.10      2.32     0.00      0.00      0.00      34.74
                4     -4.10     -2.32     0.00      0.00      0.00      -0.00
          3     3     21.32      4.73     0.00      0.00      0.00      70.97
                4    -21.32     -4.73     0.00      0.00      0.00       0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   45. PRINT SUPPORT REACTION LIST 1 4
 SUPPORT  REACTION LIST     1
     PORTAL FRAME                                             -- PAGE NO.    6
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      3.99     23.18      0.00      0.00      0.00    -11.48
          2     -7.68     -4.10      0.00      0.00      0.00     67.93
          3     -2.77     14.30      0.00      0.00      0.00     42.34
     4    1     -3.99     24.32      0.00      0.00      0.00      0.00
          2     -2.32      4.10      0.00      0.00      0.00      0.00
          3     -4.73     21.32      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   46. LOAD LIST 1 3
   47. PARAMETER 1
   48. CODE AISC UNIFIED 2010
   49. FYLD 7200 ALL
   50. TRACK 2.0 MEMB 2 3
   51. UNB 10.0 MEMB 2 3
   52. UNT 10.0 MEMB 2 3
```

```
   53. SELECT MEMBER 2 3
STEEL DESIGN
     PORTAL FRAME                                             -- PAGE NO.    7
                    STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                    ***********************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
      2 ST   W10X22                  (AISC SECTIONS)
                          PASS   Eq. H1-1b           0.967         1
                       3.99 C          0.00        -70.55        9.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    172.030  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     9.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.967(PASS)                 |
|          Loc  :    9.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   3.992E+00(C ) Fy:  -5.738E-01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -7.055E+01               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.140E+00     Ayy:   2.448E+00     Cw:  2.760E+02              |
| Szz:   2.314E+01     Syy:   3.965E+00                                 |
| Izz:   1.180E+02     Iyy:   1.140E+01     Ix:  2.390E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        8639.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        19.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     7.99     N/A     13.49    T.B4.1(a)-1  |
|                Slender        37.00     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         7.99     9.15    24.08    T.B4.1(b)-10 |
|                Compact        37.00    90.55   137.27    T.B4.1(b)-15 |
     PORTAL FRAME                                             -- PAGE NO.    8
                    STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                    ***********************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.92E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   2.92E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    4.73E+00   2.37E+02   0.020      Eq. E7-1      3     0.00 |
```

```
| Min Buck    4.73E+00   4.95E+01   0.095      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    4.73E+00   1.53E+02   0.031      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.51E-02    53.47   5.84E+03   1.44E+04   2.63E+02        |
| Min Buck    4.51E-02   172.03   1.22E+03   1.39E+03   5.50E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    4.51E-02   3.77E+03   1.70E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.12E+02   0.000      Eq. G2-1      1     0.00 |
| Local-Y    -2.43E+01   7.34E+01   0.331      Eq. G2-1      1    19.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.88E-02     1.00     1.20     7.99   1.24E+02            |
| Local-Y     1.70E-02     1.00     0.00    37.00   7.34E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       7.05E+01   9.75E+01   0.724      Eq. F2-1      1     9.50 |
| Minor       0.00E+00   2.29E+01   0.000      Eq. F6-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.08E+02   0.00E+00                                       |
| Minor       2.54E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       7.05E+01   7.62E+01   0.926      Eq. F2-2      1     9.50 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       8.46E+01   0.00E+00    1.00     4.68    13.84    10.00    |
     PORTAL FRAME                                          -- PAGE NO.    9
                    STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                    ***********************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.967      Eq. H1-1b     1     9.50             |
| Flexure Tens          0.926      Eq. H1-1b     1     9.50             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         7.62E+01   7.05E+01   4.95E+01                   |
|                      2.29E+01   0.00E+00   3.99E+00                   |
| Flexure Tens         7.62E+01   7.05E+01   2.92E+02                   |
|                      2.29E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
| CHECK FOR IN-PLANE/OUT-OF-PLANE FLEXURE COMP INTERACTION              |
|                       RATIO      CRITERIA    L/C     LOC              |
| In-Plane Flx Comp     0.906      Eq. H1-1b     1     7.92             |
| Out Plane Flx Comp    0.923      Eq. H1-2      1     7.92             |
| Intermediate          Mr         Mc         Pr         Pc            |
| In-Plane Flx Comp    6.83E+01   7.62E+01   3.99E+00   2.37E+02        |
| Out Plane Flx Comp   6.83E+01   7.62E+01   3.99E+00   4.95E+01        |
```

```
|--------------------------------------------------------------------------|
     PORTAL FRAME                                             -- PAGE NO.   10
                    STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                    ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       3 ST   W10X26                  (AISC SECTIONS)
                          PASS   Eq. H1-2             0.863         3
                      21.32 C          0.00         70.97        0.00
|--------------------------------------------------------------------------|
| SLENDERNESS                                                              |
| Actual Slenderness Ratio    :    132.238  L/C  :     3                   |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      3    Ratio     :     0.863(PASS)                    |
|          Loc  :    0.00   Condition :   Eq. H1-2                         |
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   2.132E+01(C ) Fy:   4.731E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   7.097E+01                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   5.078E+00     Ayy:   2.678E+00     Cw:  3.427E+02                 |
| Szz:   2.796E+01     Syy:   4.887E+00                                    |
| Izz:   1.440E+02     Iyy:   1.410E+01     Ix:  4.020E-01                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:        7199.999   Fu:        8639.999                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        15.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                 |
| Compression :  Non-Slender     6.56     N/A     13.49    T.B4.1(a)-1    |
|                Non-Slender    33.92     N/A     35.88    T.B4.1(a)-5    |
| Flexure     :  Compact         6.56     9.15    24.08    T.B4.1(b)-10   |
|                Compact        33.92    90.55   137.27    T.B4.1(b)-15   |
     PORTAL FRAME                                             -- PAGE NO.   11
                    STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                    ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   3.42E+02   0.000      Eq. D2-1      1     0.00   |
| Rupture     0.00E+00   3.42E+02   0.000      Eq. D2-2      1     0.00   |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    2.43E+01   3.02E+02   0.081      Eq. E3-1      1     0.00   |
| Min Buck    2.43E+01   9.83E+01   0.247      Eq. E3-1      1     0.00   |
```

```
| Flexural                                                              |
| Tor Buck    2.43E+01   2.20E+02   0.110      Eq. E4-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    5.28E-02    41.38   6.35E+03   2.41E+04   3.36E+02        |
| Min Buck    5.28E-02   132.24   2.07E+03   2.36E+03   1.09E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    5.28E-02   4.63E+03   2.45E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.37E+02   0.000      Eq. G2-1      1     0.00 |
| Local-Y     4.73E+00   8.03E+01   0.059      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv      h/tw      Vn             |
| Local-Z     3.53E-02     1.00     1.20     6.56   1.52E+02            |
| Local-Y     1.86E-02     1.00     0.00    33.92   8.03E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -7.10E+01   1.17E+02   0.605      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   2.81E+01   0.000      Eq. F6-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.30E+02   0.00E+00                                       |
| Minor       3.13E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -7.10E+01   9.47E+01   0.749      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       1.05E+02   0.00E+00    1.00     4.81    14.89    10.00    |
     PORTAL FRAME                                          -- PAGE NO.   12
                        STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                        ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.341      Eq. H1-1a     1    12.50             |
| Flexure Tens          0.749      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         9.47E+01  -9.98E+00   9.83E+01                   |
|                      2.81E+01   0.00E+00   2.43E+01                   |
| Flexure Tens         9.47E+01  -7.10E+01   3.42E+02                   |
|                      2.81E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
| CHECK FOR IN-PLANE/OUT-OF-PLANE FLEXURE COMP INTERACTION              |
|                       RATIO      CRITERIA    L/C     LOC              |
| In-Plane Flx Comp     0.784      Eq. H1-1b     3     0.00             |
| Out Plane Flx Comp    0.863      Eq. H1-2      3     0.00             |
| Intermediate         Mr          Mc         Pr         Pc            |
| In-Plane Flx Comp   -7.10E+01   9.47E+01   2.13E+01   3.02E+02        |
| Out Plane Flx Comp  -7.10E+01   9.47E+01   2.13E+01   9.83E+01        |
|-----------------------------------------------------------------------|
```

```
   54. PERFORM ANALYSIS
   ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
   55. PARAMETER 2
   56. TRACK 0 ALL
   57. CHECK CODE ALL
STEEL DESIGN
     PORTAL FRAME                                                  -- PAGE NO.   13
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      1 ST   W12X35                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.531         1
                      23.77 C          0.00          65.68       15.00
      2 ST   W10X22                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.787         1
                       4.35 C          0.00         -59.21        9.50
      3 ST   W10X26                   (AISC SECTIONS)
                           PASS   Eq. H1-2             0.808         1
                      23.73 C          0.00          65.28        0.00
   58. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:43:55 ****
     PORTAL FRAME                                                  -- PAGE NO.   14
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

T.1 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the **Post Processing** Mode. The **Post Processing** mode is used to verify the analysis and design results and generate reports.

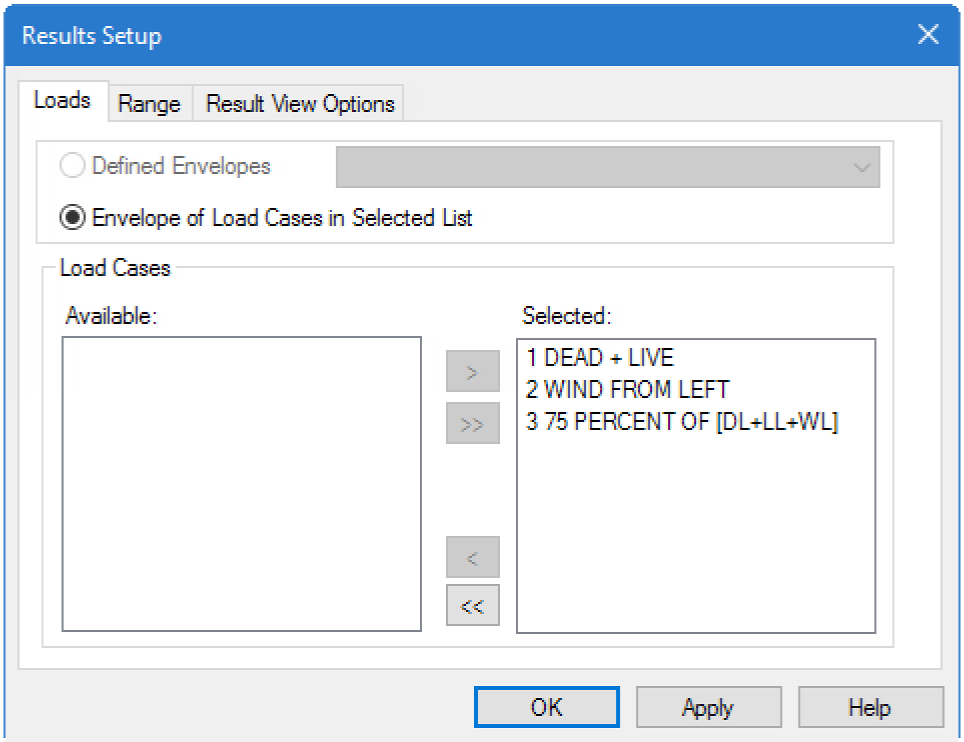
T.1 Opening the postprocessing workflow

You can open the Postprocessing workflow anytime there are current analysis results for your model.

If you selected to open the Postprocessing workflow after T.1 Performing Analysis/Design (on page 446), then you can skip to step 2.

1. On the **Workflows** panel, select **Postprocessing**.

The **Results Setup** dialog opens.



2. (Optional) Select the load cases for which to display the results.

   Tip: All load cases are selected by default.

   For this tutorial, we will use all load cases.
3. Select the **Result View Options** tab and then check the **Enable Automatic Scaling** option.
4. Click **OK**.

Note:

Notice that in the Postprocessing workflow, the **Postprocessing** page control bar is opens above the view window. The ribbon bar updates to include the **Results** ribbon tab.

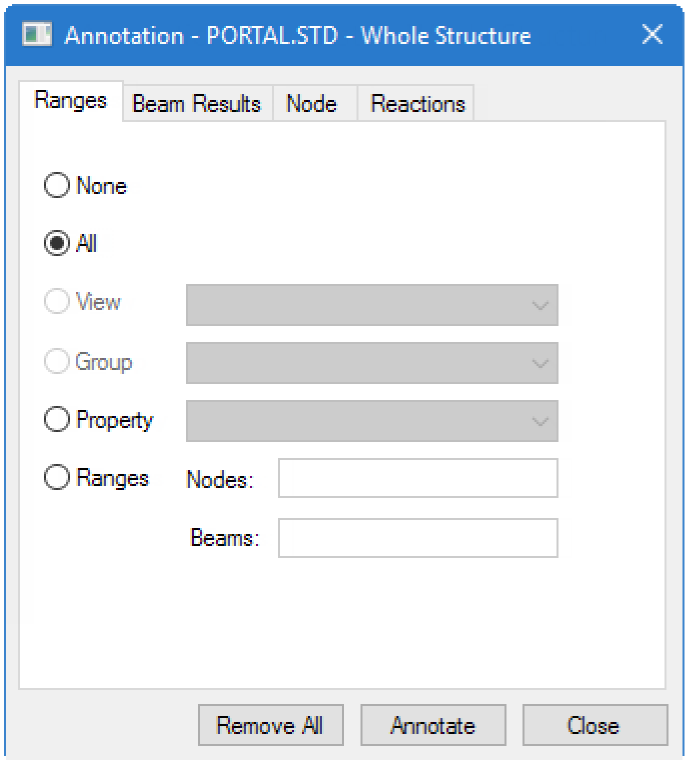
T.1 Annotating the displacements

Annotation is the process of displaying the displacement values on the screen.

1. Select the **Displacements** page in the Postprocessing page control bar.

   The diagram displayed is the node deflection diagram for load case 1 (DEAD + LIVE).
2. On the **Results** ribbon tab, select the **Annotate** tool in the **Configuration** group.

The **Annotation** dialog opens.



3. From the **Ranges** tab, select **All**.

Tip: If you want to annotate deflection for just a few nodes, specify the node numbers in the node list.

4. Select the **Node** tab and then check the **Resultant** option.

   The resultant is the square root of sum of squares of values of X, Y and Z displacements.

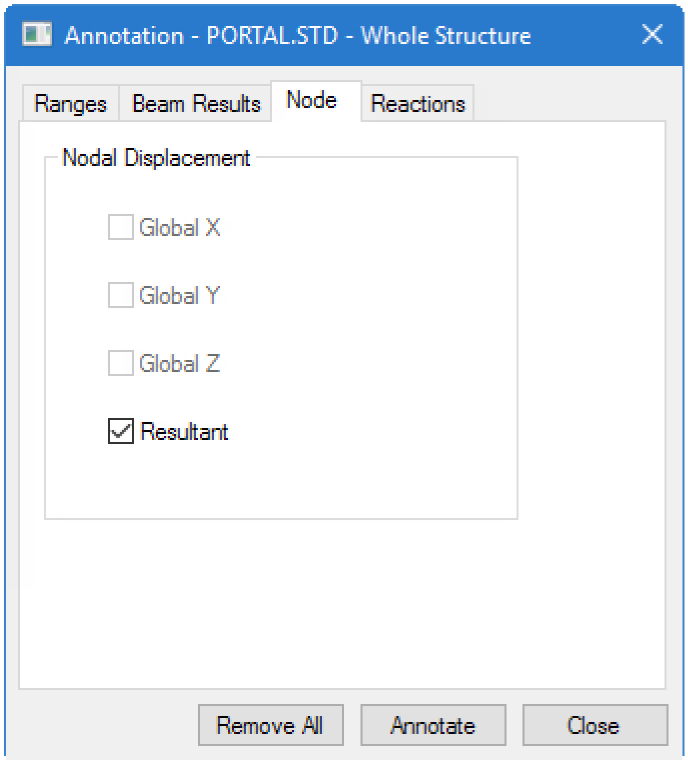


Figure 34:

5. Click the **Annotate**.
   The values appear on the structure.
6. Click **Close**.

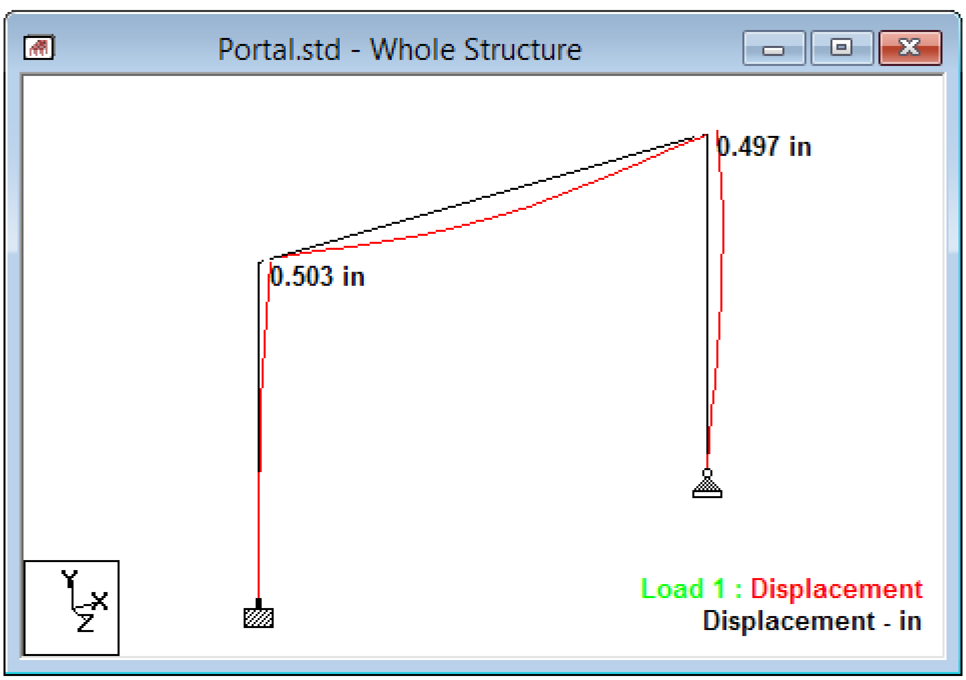


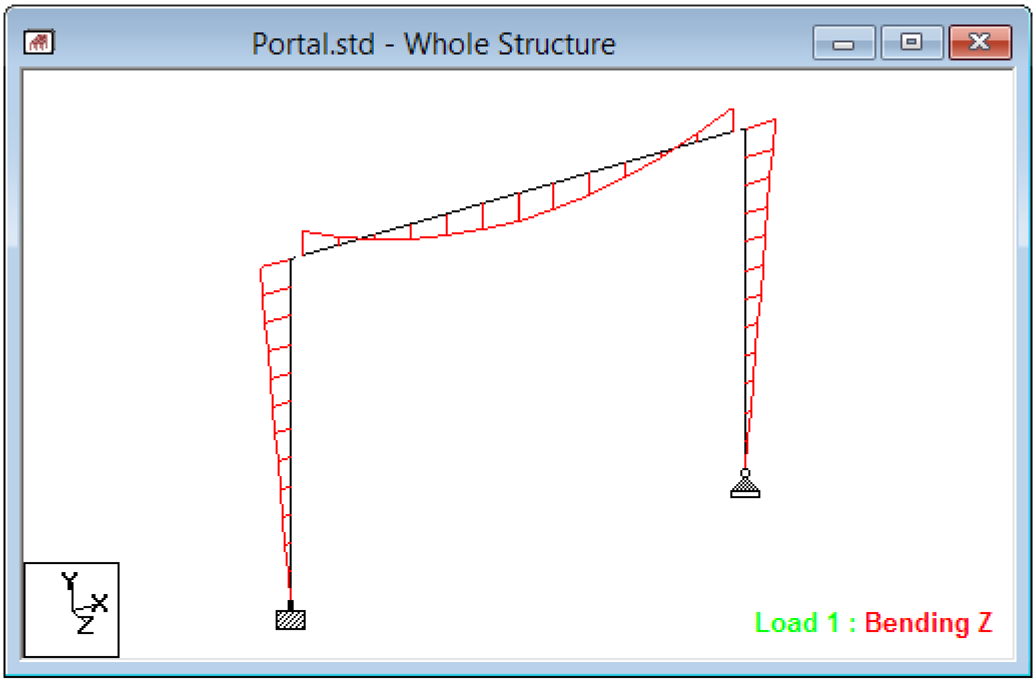
Figure 35: The deflected shape diagram for load case 1

Tip: You can exaggerate the deflected shape by *reducing* the **Deflection** scale in the **Diagrams** dialog **Scales** tab.

T.1 Displaying force and moment diagrams

1. Select the **Beam Results** page in the **Postprocessing** page control bar.

   The bending moment MZ is plotted by default. Note that the **Mz** tool  in the **View Results** group on the **Results** ribbon tab is depressed (active).

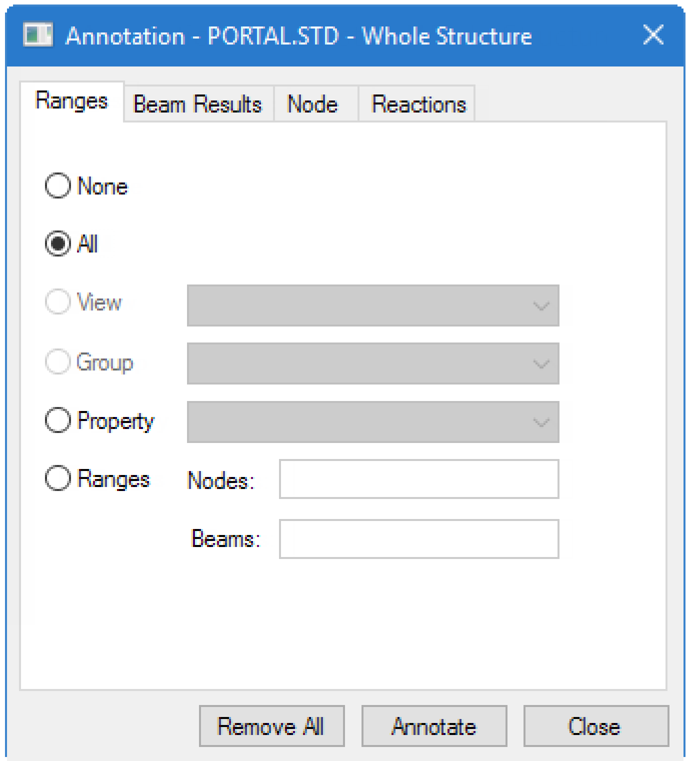
   

2. (Optional) Change the force or moment diagram drawn on the structure in the **Diagrams** dialog:
   a. Right-click in the view window and then select **Structure Diagrams** from the pop-up menu.
   The **Diagrams** dialog opens.
   b. Select the **Loads and Results** tab.
   c. Select a different **Load Case** from the drop-down list.
   d. Select one or more options in the **Beam Forces** group.
   For example, select the Shear yy to display the shear force diagrams in the local y direction on the members.
   e. (Optional) Click on the color block adjacent to any of the force or moment options to change the color for that diagram.
   f. (Optional) The **Beam Forces Diagram** options control how the diagrams are filled.

Tip: Alternatively, you can select which diagrams are displayed and for which load case using the tools and load cases selection in the **View Results** group on the **Result** ribbon tab.

3. On the **Results** ribbon tab, select the **Annotate** tool in the **Configuration** group.

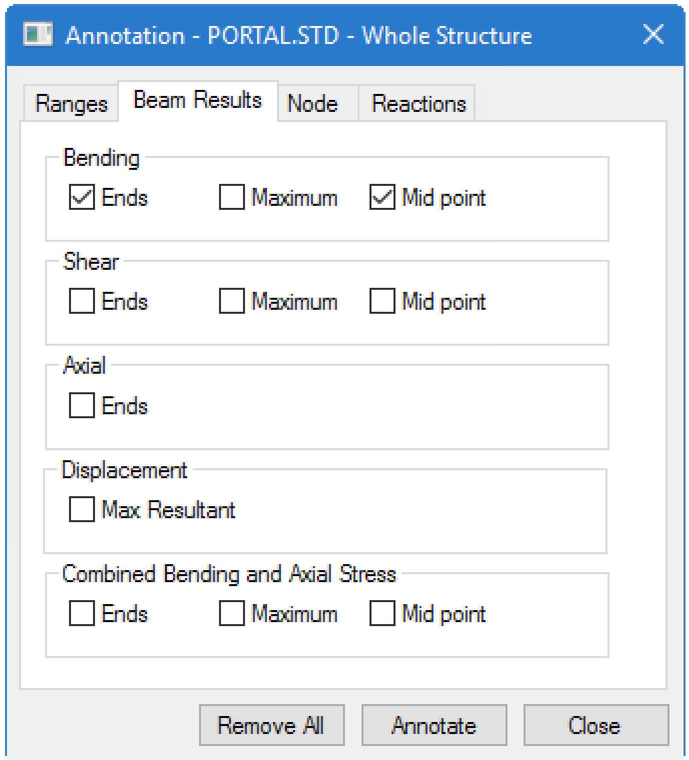
The **Annotation** dialog opens.



4. From the **Ranges** tab, select **All**.

> **Tip:** If you wish to annotate the force/moment for just a few members, specify the beam numbers in the beam list.

5. Select the locations where you want to display results:
   a. Select the **Beam Results** tab
   b. Check the **Ends**and **Mid point** options in the **Bending** group.



6. Click **Annotate**.

   The diagram is updated with annotated values displayed at the selected locations.

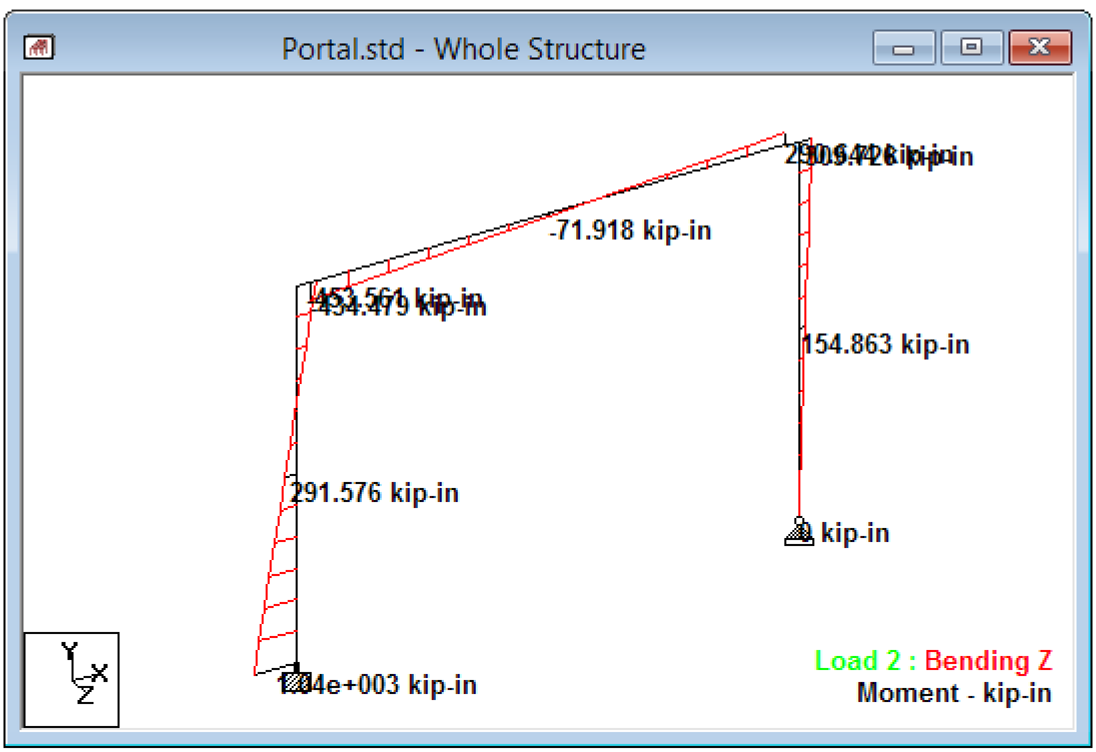


Figure 36: The Mz bending diagram for Load 2 with annotated values

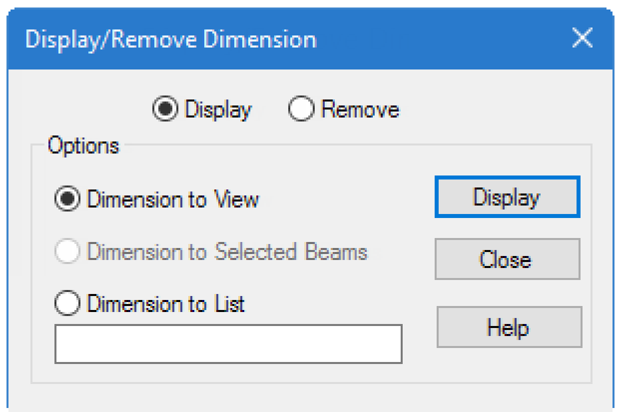
7. Click **Close**.

T.1 Displaying the dimensions of the members

1. On the Utilities ribbon tab, select the **Beam Tools** > **Dimension Beams** tool in the **Geometry Tools** group.

   The **Display/Remove Dimension** dialog opens.

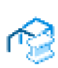
2. Select the **Dimension to View** option.
3. Click **Display**.
4. Click **Close**.
   The dimensions of the members will appear alongside the members.

T.1 Update physical model with design results

If you generated the model geometry using a physical model, then you need to update the physical model with design results.

Since you performed a member selection as part of the member design in the analytical model, the sections need to be updated in the physical model to keep it synchronized. STAAD.Pro Physical Modeler will detect that design results are present and prompt you to do this.

1. Select **Physical Modeling** workflow in the **Workflows** panel.

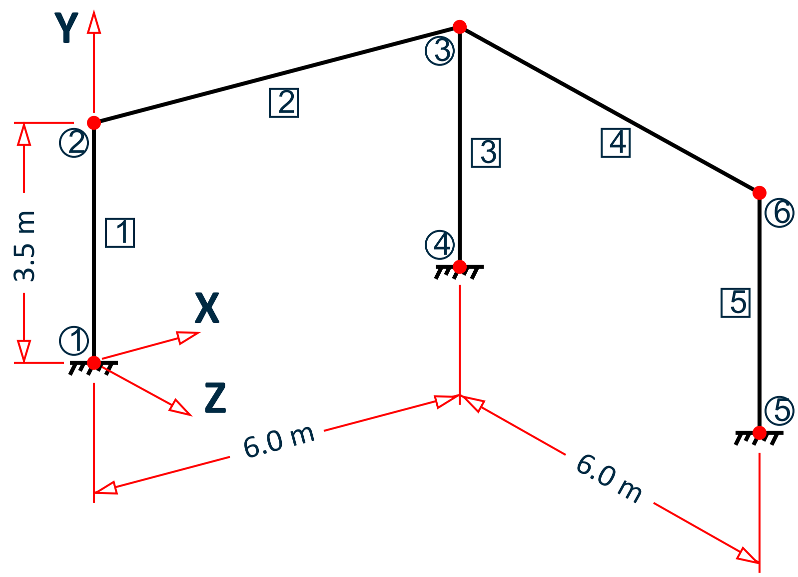
   The STAAD.Pro Physical Modeler window opens and the **Member Design Selection** dialog opens.
2. Click the **Profile** selection drop-down.
   The member design results table opens.

> **Note:** Here you can review the different analytical member design results for each physical member. Recall that a physical member may be decomposed into multiple analytical members and each of those may have different design results in STAAD.Pro (unless the `FIXED GROUP` command is used).

3. Click **Accept**.
   The table closes.
4. Click **Commit**.
   The dialog closes and the physical model is updated with the new sections from the STAAD.Pro steel design results.

T.2 - RC Framed Structure

This tutorial provides step-by-step instructions for creating the model of a reinforced concrete framed structure using STAAD.Pro.



A copy of the completed input file is typically installed at `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Tutorials\Tutorial 2 - RC Frames Structure.std`.

Related Links

- *EX.Tutorials* (on page 6255)

T.2 Methods of creating the model

As explained in " T.1 Methods of creating the model (on page 387) ", there are three methods of creating the structure data:

1. T.2 Creating the Model using the Physical Modeler (on page 466),
2. T.2 Creating the model using the analytical user interface (on page 472),
3. and T.2 Creating the model using the command file (on page 509).

All three methods are explained in this tutorial.

T.2 Description of the tutorial problem

The structure for this project is a 2 bay, 2 story reinforced concrete frame. The figure below shows the structure. The purpose of this tutorial is to create the model, assign all required input, and perform the analysis and concrete design.

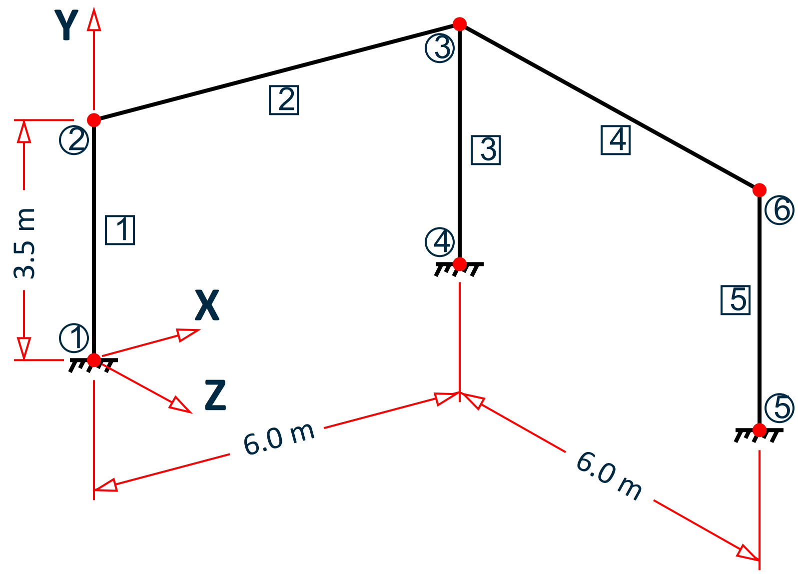


Figure 37:

Basic Data for the Structure

Attribute	**Data**
Member properties	Beams 2 & 5 : Rectangular, 275 mm width × 350 mm depth Columns 1 & 4 : Rectangular, 275 mm width × 300 mm depth Column 3 : Circular, 350 mm diameter
Member Orientation	All members except column 4 : Default Column 4 : Rotated by 90 degrees with respect to default condition
Material Constants	Modulus of Elasticity : 22 kN/mm^2 Density : 17 kn/m^3 (lightweight concrete) Poisson's Ratio : 0.17
Supports	Base of all columns : Fixed

Attribute	Data
Loads	Load case 1 : Dead Load Selfweight of the structure. Beams 2 & 5 : 400 kg/m in global Y downward Load case 2 : Live Load Beams 2 & 5 : 600 kg/m in global Y downward Load case 3 : Wind Load Beam 1 : 300 kg/m along positive global X Beam 4 : 500 kg/m along positive global X Load Case 4 : DEAD + LIVE L1 X 1.2 + L2 X 1.5 (Use REPEAT LOAD, not Load Combination) Load Case 5 : DEAD + WIND L1 X 1.1 + L2 X 1.3 (Use REPEAT LOAD, not Load Combination)
Analysis Type	PDELTA
Concrete Design	Consider load cases 4 and 5 only. Parameters: Ultimate Strength of Steel: 415 N/mm^2 Concrete Strength: 25 N/mm^2 Clear cover for top: 25 mm Clear cover for bottom: 30 mm Clear cover for side: 25 mm Design beams 2 and 5 Design columns 1, 3 and 4

T.2 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.
2. Type as `RCFrame` in the **File Name** field.
3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.

4. Select **Physical** for the **Type** of model.
   This option selects the modeling method you want to use:

Analytical	For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
Physical	For creating a model using the STAAD.Pro Physical Modeler interface.
Building	For creating a building model structure using the Building workflow.

5. Select **Metric** as the system of **Units**.

   Tip: The units can be changed later if necessary, at any stage of the model creation.

6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.
7. Click **Create**.

The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose **Analytical**, click **Create**, and then proceed to either “ T.1 Creating the model using the analytical user interface (on page 399) ” or “ T.2 Creating the model using the command file (on page 509),” respectively.

T.2 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

Tip: Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

T.2 Generate the model geometry

1. On the **Model** ribbon tab, select the **Grid** tool in the **Create** group.

   The **Create Grid** dialog opens.
2. Supply the grid details:
   a. Type `Grid 1` in the **Name** field.
   b. Leave the **Plane** selection as **XY** and the **Creation Method** as **By Spacing**.
   c. Type `6` in the **Number of spaces in X** and `1 m` in the **Grid spacing X** fields.
   d. Type `7` in the **Number of spaces in Y** and `0.5 m` in the **Grid spacing Y** fields.

e. Click **OK**.
The grid is created.

3. Use the **Member** tool to create the first portal frame:

a. On the **Model** ribbon tab, select the **Member** tool in the **Create** group.

b. Click on the following points on Grid 1, in sequence.

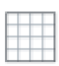
Double-click the final point to stop creating connected members.

(0,0,0)
(0,3.5,0)
(6,3.5,0)
(6,0,0)

c. Select the **Member** tool again to deactivate the tool.

Note: Refer to T.1 Generating the model geometry (on page 391) in Tutorial 1 for detailed steps on how to draw a portal frame.

4. On the **View** ribbon tab, select the **Grids** tool in the **Reference** group.



The tool is no longer highlighted and all grids are hidden in the display.

5. Switch on the node and beam labels:

a. On the **View** ribbon tab, select the drop-down list below the **Numbering** tool in the **Model** group.

b. Select **Node** and **Member** on this list.
They are both marked with a check.

c. Select the **Numbering** tool.

6. On the **Model** ribbon tab, select the **Circular Copy** tool in the **Edit** group.

The **Copy Model Circular** dialog opens.

7. Specify the circular copy details:

a. Leave the **Method** selection as **Global**.
b. Select **Y** as the **Axis of rotation**.
c. Type `6 m` in the Pivot point **X** field.
d. Type `90 deg` in the **Arc angle** field.
e. Type `1` in the **Copies** field.
f. Click **OK**.

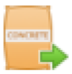
The dialog closes and the selection elements of the portal frame are copied at 90° from the right end.

8. In the lower, right corner of the view window, click the **Switch to isometric view** tool to display the frame in an isometric view.

T.2 Assign user-defined concrete material

If you don't still have all the members selected, press **<Ctrl+A>**.

1. On the **Member Tools** ribbon tab, select the **Material** tool in the **Assign properties** group.



   The **Assign material** dialog opens.
2. Select the concrete specification:
   a. Select **Custom** from the **Type** drop-down list.

      Leave the **Source** as **Catalog** and the **Country** as **United States**.
   b. Select **CONCRETE_ACI318** from the **Material type** drop-down list.
3. Define the custom lightweight concrete material:
   a. Type `0.17` in the **Mu** field (Poisson's ratio).
   b. Type `0.1E-6` in the **Alpha** field (coefficient of thermal expansion).
   c. Type `0.05` in the **CDAMP** field (composite damping ratio)
   d. Type `25 N/mm2` in the **Fck** field (concrete ultimate strength).

      Tip: You can type any appropriate unit in any unit system into the field and it will automatically be converted to the current default units.
   e. Type `19080 N/m3` in the **Gamma** field (material density).
   f. Type `CONCRETE-LTW` in the **Name** field.
   g. Click **OK**.

   The **CONCRETE-LTW** material is assigned to all the members. This material is also added to the model catalog for later use if necessary.

On the **Model** ribbon tab, select the **Catalog** tool in the **Catalog** group.  The model catalog window opens.

Select the **Material** tool so it is highlighted in the display control toolbar found in the lower-left corner. The **CONCRETE-LTW** material is shown as a tile in the Materials section. Select this tile to review the material values. You can use this interface to make changes to the custom materials and sections in your model.

Once you are done reviewing, click the back arrow in the top-left corner of the window.

T.2 Assign the member properties

1. While holding **<Ctrl>**, click on both the end columns (M1 and M4).
2. On the **Member** ribbon tab, select the **Section** tool in the **Assign properties** group.

The **Assign Section** dialog opens.

3. Define the rectangular column section:

   Leave the Source selection as Catalog.

   a. Select **Prismatic** in the **Type** drop-down list.

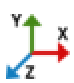
      Leave the **Source** selection as **Catalog**.

   b. Select **Solid Rectangle** in the **Shape** drop-down list.
   c. Select **RECT** in the **Template** drop-down list.
   d. Type `300 mm` in the **YD** field and `275 mm` in the **ZD** field.
   e. Type `RECT-COL` in the **Name** field.
   f. Click **OK**.
4. Repeat steps 1-3 except to define a 350mm (depth, YD) by 275 mm (width, ZD) rectangular prismatic section named `RECT-BEAM` to the beam members (M2 and M5).
5. Repeat steps 1-3 again except to define a 350mm diameter (YD) solid circular prismatic section named `CIRC-COL` to the middle column (M3).
6. Rotate the right column by 90 deg (Beta angle):

   a. Select the right column (M4).
   b. On the **Member Tools** ribbon tab, select the **Rotate** tool in the **Edit** group and then select **Rotate 90°** from the list.

      The column is rotated about its local 1 axis by 90°.
   c. On the **View** ribbon tab, select the **Local Axes** tool in the **Model** group.

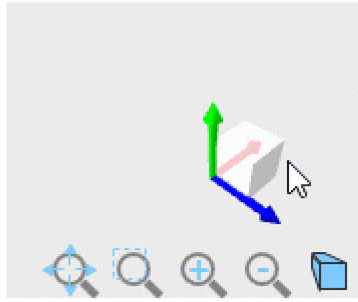
      

      You can confirm that the local z axis of M4 is now rotate 90° away from the global Z axis.

T.2 Assign supports

1. Orient the view to front view:

   a. (Optional) If the rotation controls are hidden, click the rotation widget in the lower-right corner of the view window.
   b. Select the **Switch to front** view tool.
   c. (Optional) Click the rotation widget again to hide the controls.



Tip: Alternatively, you can press and hold the right mouse button and drag the mouse pointer to quickly rotate the view so it is easy to select the column bases.

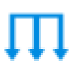
2. Click and drag a box from the right side of the to the left to enclose the column base nodes (N1, N4, and N5).
3. On the **Node** ribbon tab, select the **Fixed** tool in the **Supports** group.

   A fixed support is assigned to the selected nodes.

T.2 Assign loads to load cases

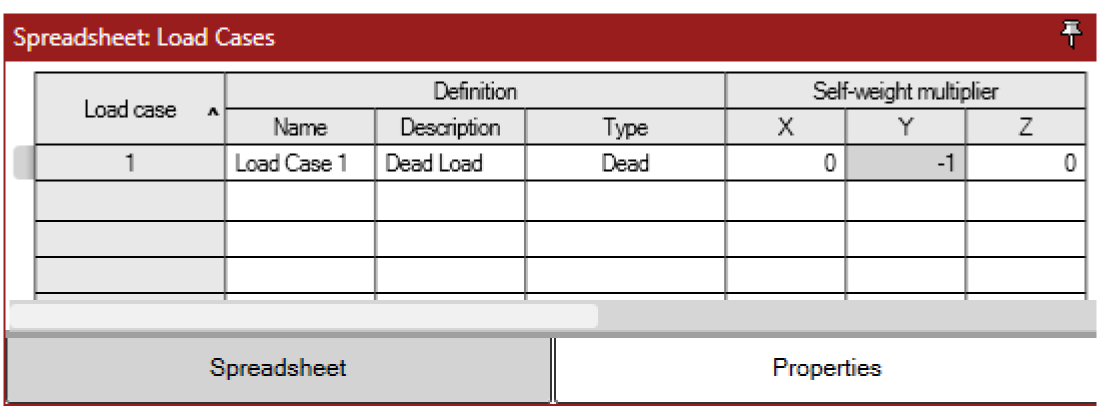
1. On the **Spreadsheet** ribbon tab, select the **Load Cases** tool in the **References** group.

   The Load Cases spreadsheet opens.
2. For Load 1:
   a. Type `Dead Load` in the description field of Load case 1.
   b. Select **Dead** as the **Type**.
   c. Type `-1` in the **Self-weight multiplier** in field **Y**.

   Note: This value is negative.

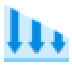
   d. Press **<Enter>**.



Spreadsheet: Load Cases

Load case	Definition			Self-weight multiplier		
	Name	Description	Type	X	Y	Z
1	Load Case 1	Dead Load	Dead	0	-1	0

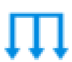
3. Select the beam members (M2 and M5).
4. Add the uniform distributed dead load:
   a. On the **Member** ribbon tab, select the **Distributed** tool in the **Loads** group.

   The **Add Member Distributed Load** dialog opens. The Load group is already selected to use Load case 1.
   b. Select the **Load type** as **Uniform**.
   c. Select **Global Y** as the **Direction**.
   d. Type `-400 kg/m` in the **Magnitude** field.

   > **Note:** This value is negative and uses different units than the default, so be sure to type out the units as indicated above. It will be automatically converted into the default units of kN/m.

   e. Click **OK**.
5. Create the live load group:
   a. On the **Model** ribbon tab, select the **Load Case** tool in the **Load** group.

   The **Add Load Case** dialog opens.
   b. Type `Live Load` in the **Description** field.
   Leave the Name as Load Case 2.
   c. Select **Live** as the **Type**.
   Leave the self weight modifiers all as zero.
   d. Click **OK**.

   The live load case is added and selected as the active load case as indicated in the application status bar.
6. Repeat steps 3 and 4 except to add a uniform distributed load of `-600 kg/m` (in the **Global Y** direction) to the beam members (M2 and M5).
7. Repeat steps 5 and 6 except to add a Load case 3 with a description of `Wind Load` and type of **Wind**.
   This load group will have the following to load items:
   - a uniform force of `300 kg/m` in the Global X direction and is applied to the left column (M1).
   - a uniform force of `500 kg/m` in the Global X direction and is applied to the right column (M4).

> **Note:** As this model uses a second-order, P-Delta analysis, load combinations are *not* recommended. Instead, repeat loads will be used to combine the load cases so that the combinations are processed by the analysis engine directly. These Repeat loads will be added in the Analytical Modeling workflow.
>
> Refer to TR.37.2 P-Delta Analysis Options (on page 2621) for further details on load types appropriate for P-Delta analysis.

T.2 Generate the Analysis Model

You have now completed the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will create load combinations, add output commands, select analysis criteria, and specify design parameters.

For this tutorial, you will use the default options for analytical model generation. However, for advanced models you can review and change those options in the **Configuration** dialog.

1. Select **Save** to save the physical model.
2. Either:

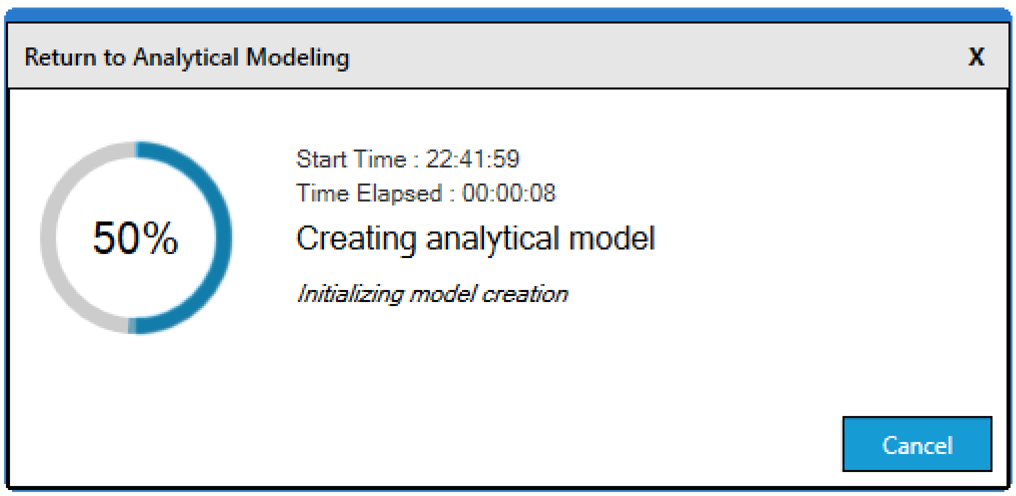
   On the **Model** ribbon tab, select the **Return to Analytical Modeling** tool in the **STAAD.Pro** group

   or

   On the **File** ribbon tab, select **Create analysis model** in the backstage tabs

   The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.

3. Click **OK**.
   If you have not previously associated this model with a CONNECT Project, you are asked to do so. This is will not be necessary for this tutorial.
4. Click **Cancel** in the **Assign Project** dialog.

The STAAD.Pro Physical Modeler window closes and the model is loaded into the STAAD.Pro analytical modeling interface.

T.2 Creating the model using the analytical user interface

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

Tip: Refer to GS. Application Window Layout (on page 54) for reference on the application window.

T.2 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

The STAAD input file commands generated are:

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.5 0.0
3 6.0 3.5 0.0 ; 4 6.0 0.0 0.0
5 6.0 0.0 6.0 ; 6 6.0 3.5 6.0
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4 ; 4 5 6 ; 5 3 6
```

When starting in the Analytical Modeling workflow, a grid is initially displayed in the main view window. This grid is controlled by the **Snap Node/Beam** dialog. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area.

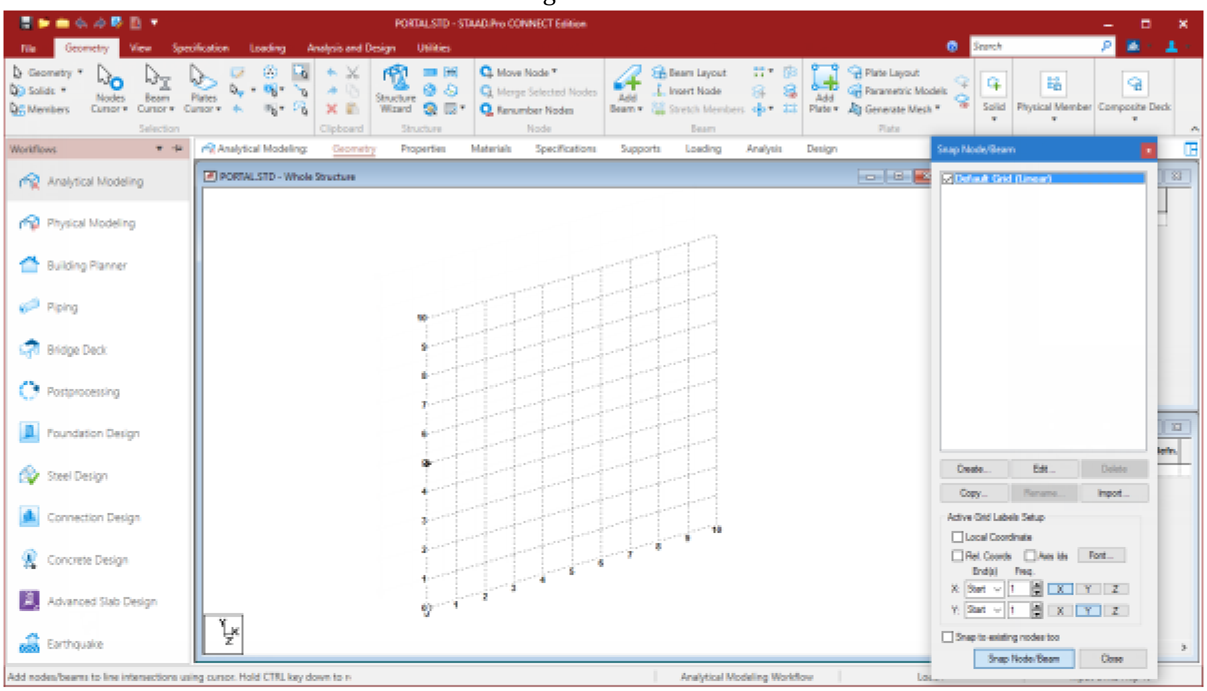


*Figure 38: The STAAD.Pro window with the **Snap Node/Beam** dialog open*

Note: The **Snap Node/Beam** dialog opens on the right side of the screen. You can reopen this dialog by selecting the **Snap Grid Beam** tool  in the **Structure** group on the **Geometry** ribbon tab.

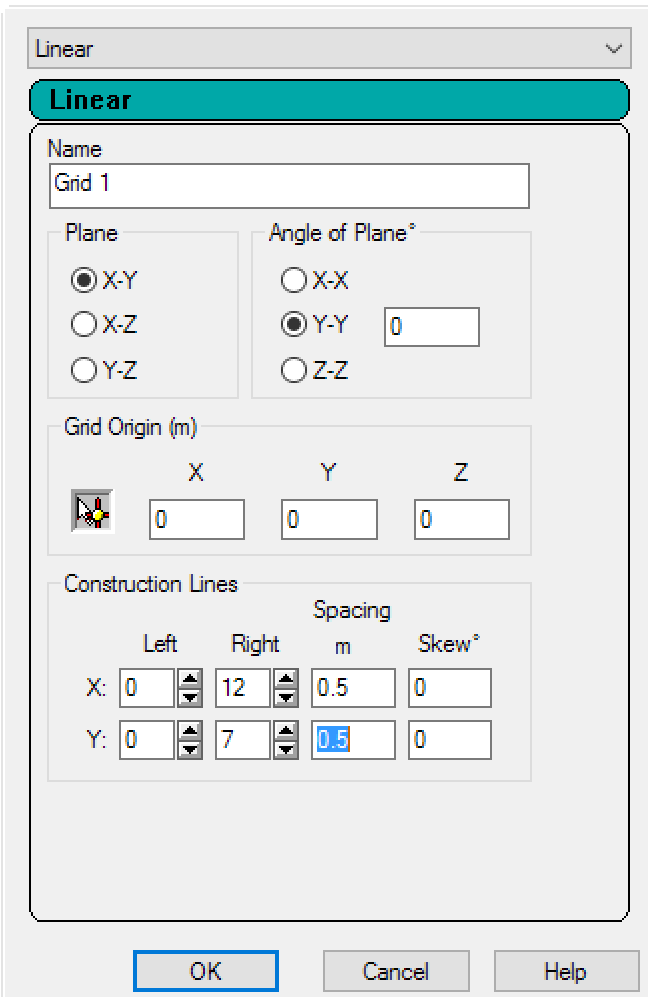
1. On the **Snap Node/Beam** dialog, click **Create**.
   A dialog opens which will enable us to set up a grid.

   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.

Linear	used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board
Radial	used to place construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modeled as piece-wise linear straight line segments
Irregular	used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane

2. Select **Linear**,which is the Default Grid.

   In this structure, the portion consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. Leave **X-Y** as the **Plane** of the grid. The size of the model that can be drawn at any time is controlled by the number of **Construction Lines** to the left and right of the origin of axes, and the **Spacing** between adjacent construction lines.
3. Type a **Name** of `Grid 1`.
4. Type `12` as the number of lines to the **Right** of the origin along **X** and `7` above the origin along **Y**.
5. Type a **Spacing** of `0.5` m between lines along both **X**and**Y** (see figure below) we can draw a frame 6m X 3.5m, adequate for our segment.



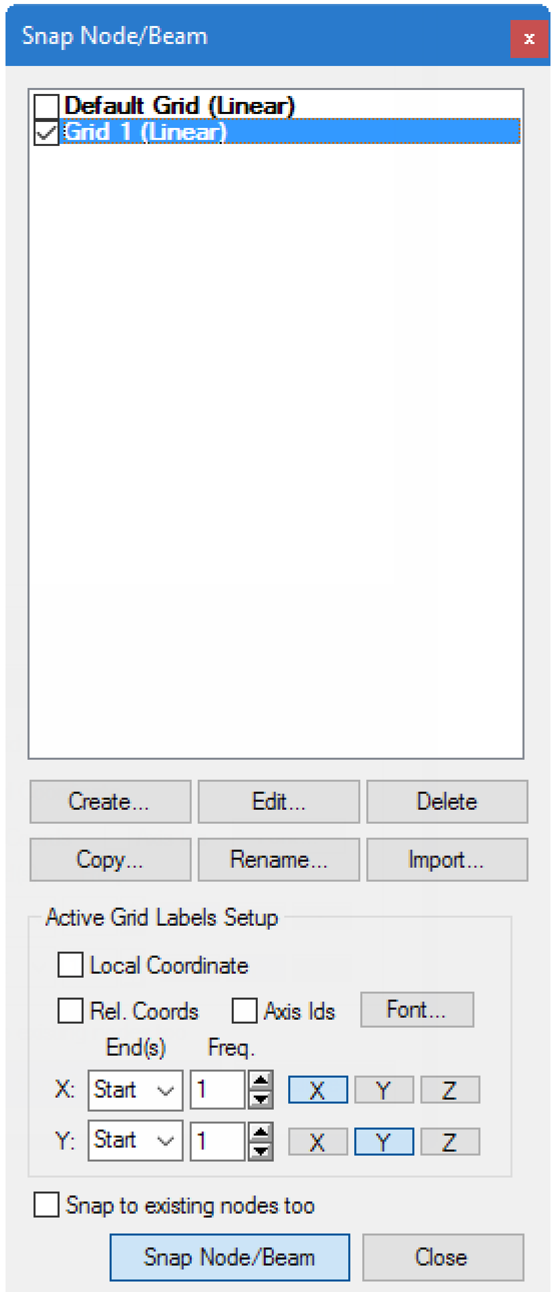
6. Click **OK**.
7. In the **Snap Node/Beam** grids list, check the new **Grid 1** option.

You can create any number of grids. By providing a name, each new grid can be identified for future reference.

Tip: Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

Tip: To change the settings of this grid, select the name in the **Snap Node/Beam** dialog and then click **Edit**.

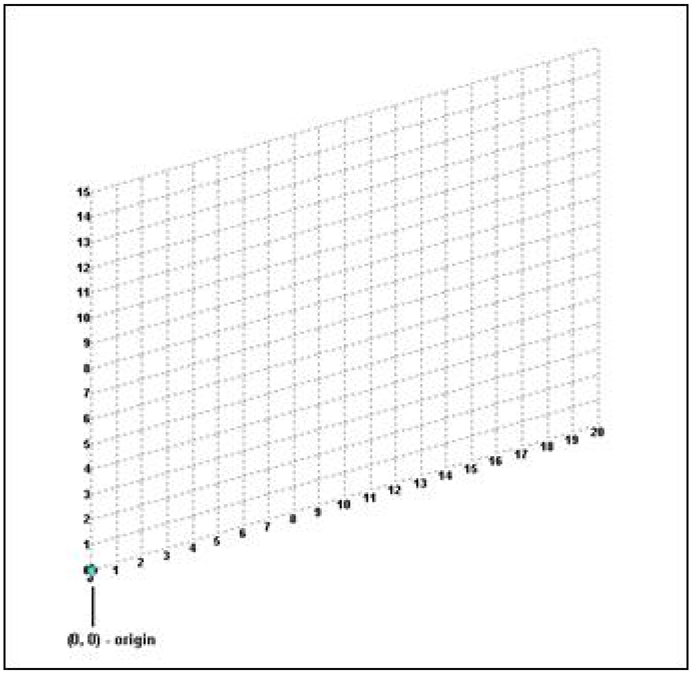
The check by **Default Grid** is automatically cleared. STAAD.Pro will only display a single grid at a time.



8. In the View window, click at the origin (0, 0) to create the first node.

A line is "rubber-banded" between this node and the mouse pointer, which previews the member to placed with the next mouse click.

The **Snap Node/Beam** feature is active by default.



9. Click on the following points to create nodes and automatically join successive nodes by beam members.

   (0, 3.5)
   (6, 3.5)
   (6, 0)

10. Click **Close** in the **Snap Node/Beam** dialog.
    The grid is hidden.

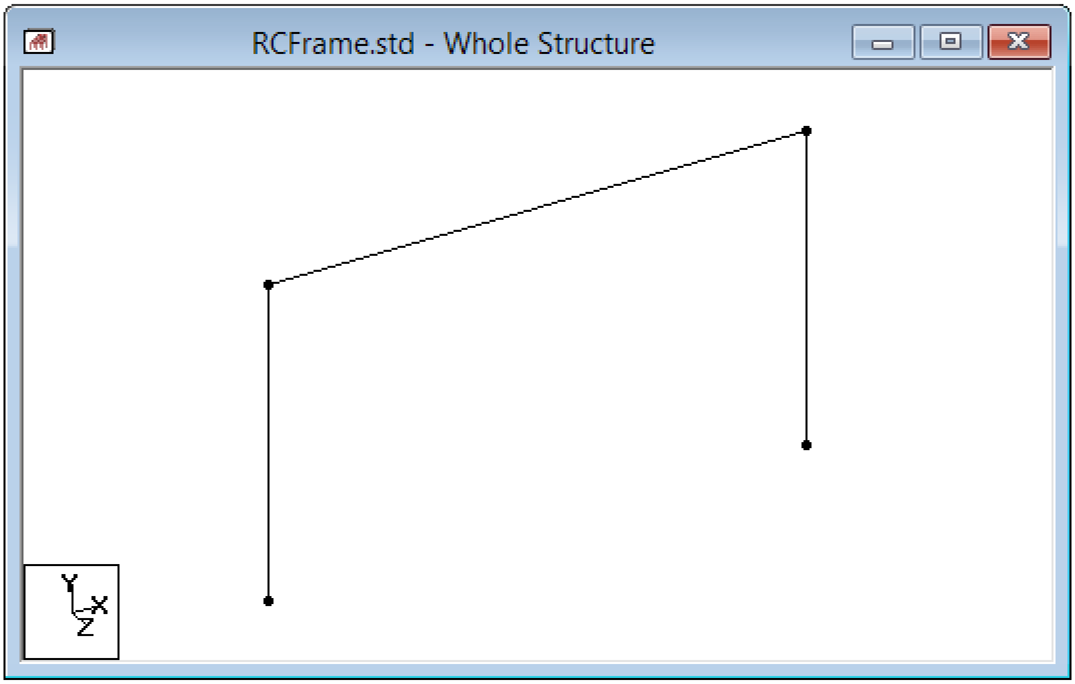


Figure 40: The first part of the frame drawn

It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Copying Model Objects by Circular Repeat

By examining the structure diagram for this tutorial exercise, you may have observed that members 4 and 5 can be easily generated if we could first create a copy of members 1 and 2 and then rotate those copied units about a vertical line passing through the point (6, 0, 0, that is, node 4) by 90 degrees. Fortunately, such a facility does exist which can be executed in a single step. It is called **Circular Repeat** and is available under the **Geometry** menu.

1. Right click anywhere in the view window and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens to the **Labels** tab.
2. Set the **Node Numbers** and **Beam Numbers** on and then click **OK.**

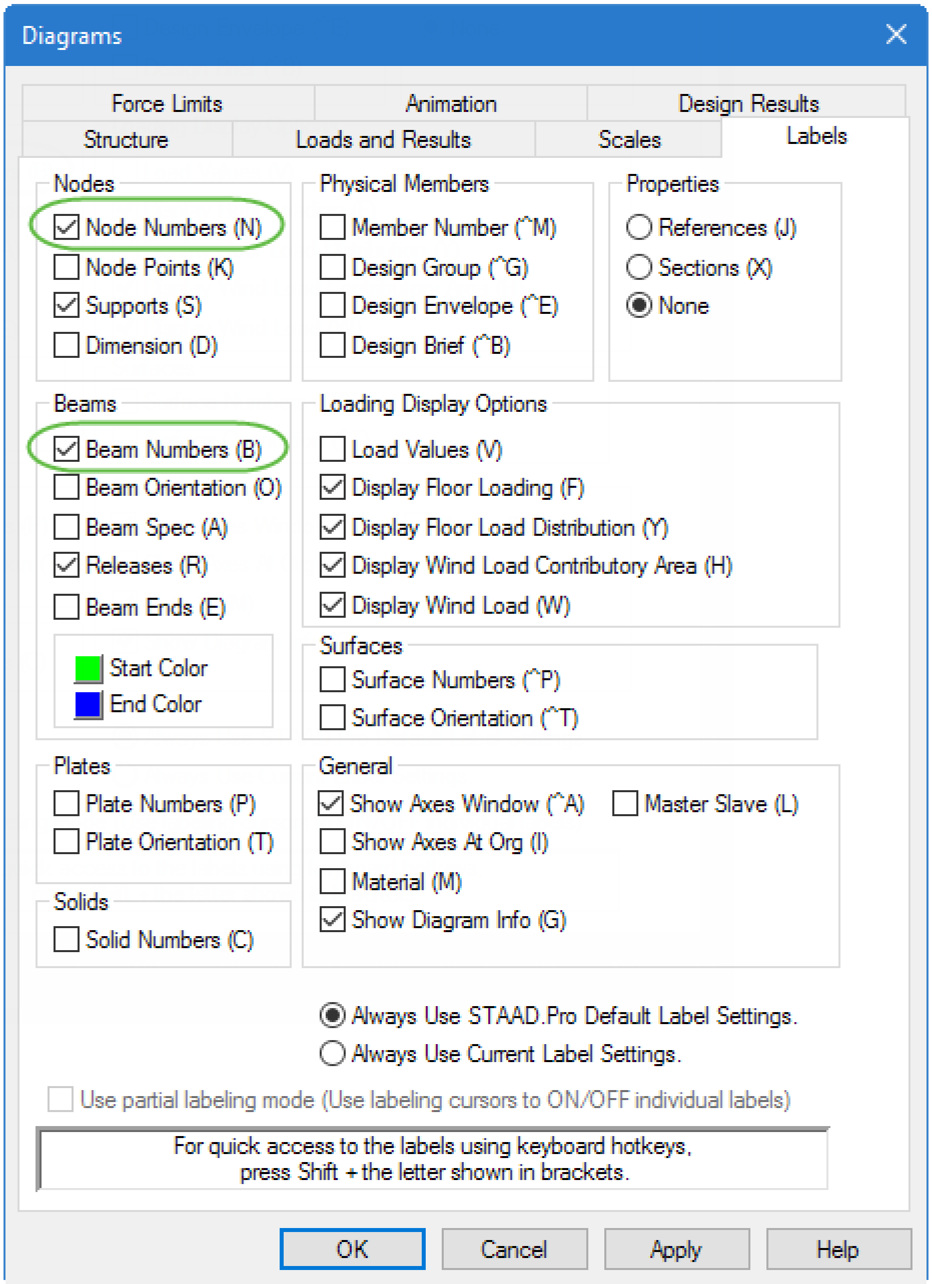


Figure 41: Select the Node and Beam numbers

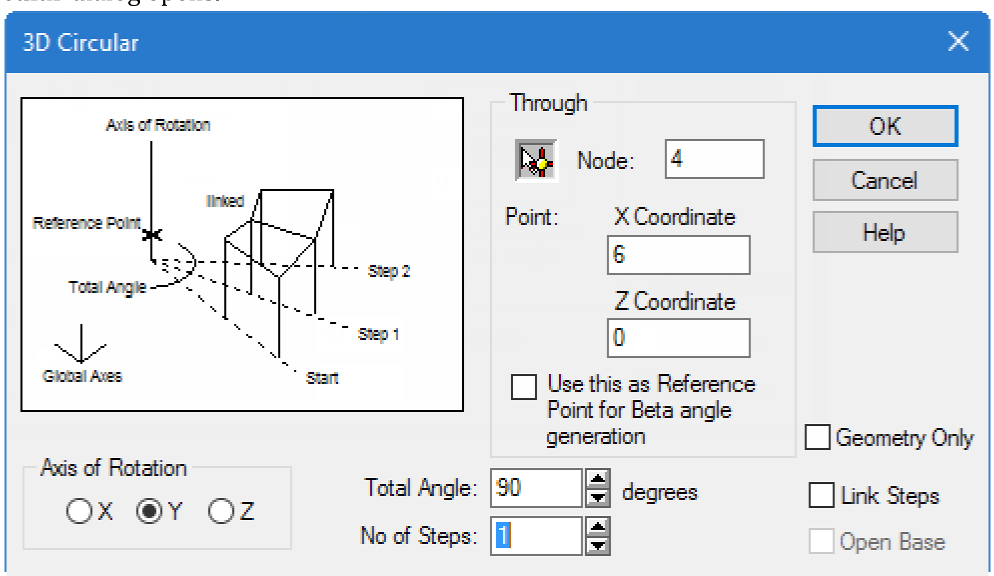
Tip: Alternatively, you can press **<Shift+N>** and then **<Shift+B>** to quickly toggle the same labels. The letters in parenthesis on the **Labels** tab indicate these **<Shift>** key shortcuts.

3. (Optional) You can change the font of the labels by selecting **File** > **Settings** > **Display Options** and then changing the corresponding label Font size in the **Options** dialog.
4. Select the first column and beam for copying:
   a. On the **Select** ribbon tab, select the **Beam Cursor** tool in the **Cursors** group.

b. Holding the **<Ctrl>** key, click on members 1 and 2.

5. On the **Geometry** ribbon tab, select the **Circular Repeat** tool in the **Structure** group.

The **3D Circular** dialog opens.



6. Specify the circular repeat details:
   a. Select **Y** as the **Axis of Rotation**.
   b. Type 4 in the **Node** field.

   This is the node number through which the rotation axis will pass.

   Tip: You can click the Node Selection tool here and then click on the node you want the rotation axis to pass through in the View window.

   c. Type `6` in the point **X Coordinate** field and type `0` in the point **Z Coordinate** field.
   d. Type `90` in the **Total Angle** field.
   e. Set the **No of Steps** value to `1`.
   f. Clear the **Link Steps**, **Geometry Only**, and **Use this as the Reference Point for the Beta angle generation** options.
   g. Click **OK**.
7. Click anywhere away from the members in the View window to deselect the members.

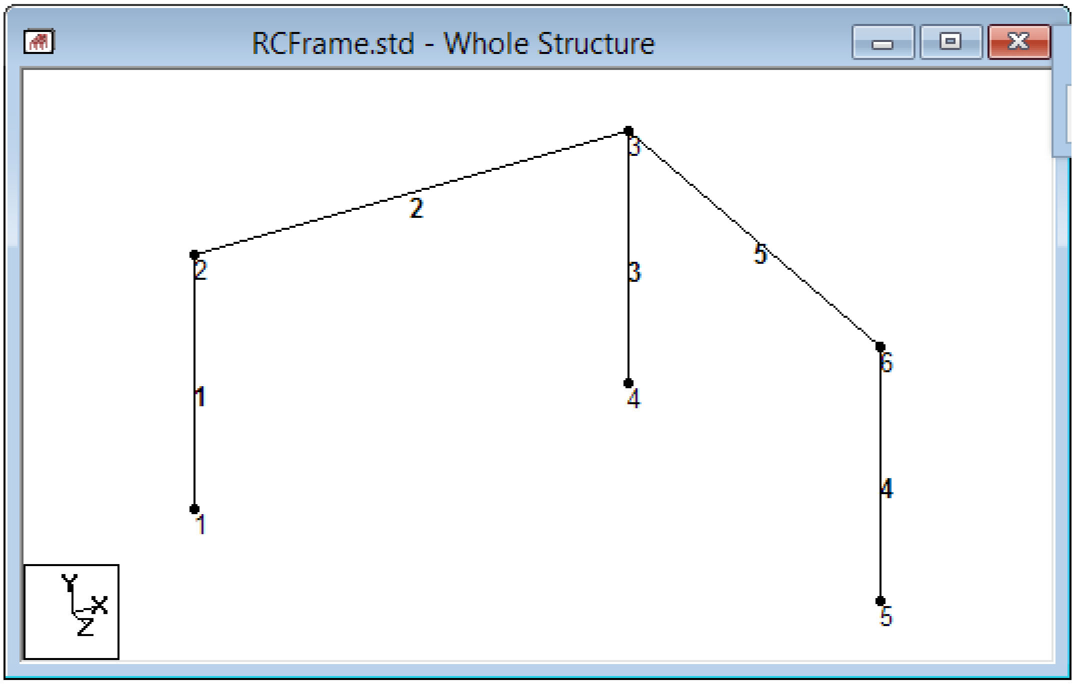


Figure 42: The frame members drawn

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Changing the input units of length

For specifying member properties for the structure, it is more convenient to use length units are millimeter instead of meter. This requires changing the current length units of input.

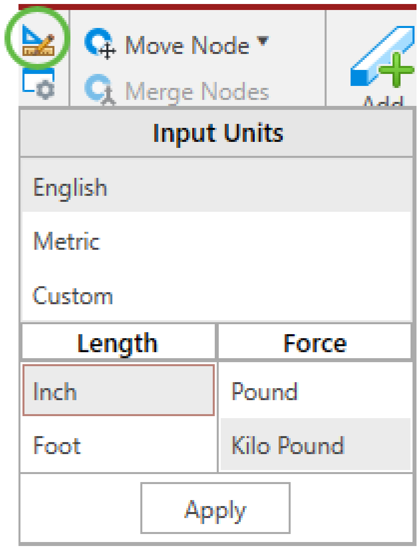
The STAAD input file commands generated are:

```
UNIT MMS KN
```

1. On the **Geometry** ribbon tab, select the **Input Units** tool in the **Structure** group.

   The **Set Input Units** pop-up opens.



2. Select **Metric** as the **Input Units**.
3. Set the **Length Units** to **Milimeter**.

   Leave the **Force Units** set to **KiloNewton**.
4. Click **Apply**.

T.2 User-defined concrete material

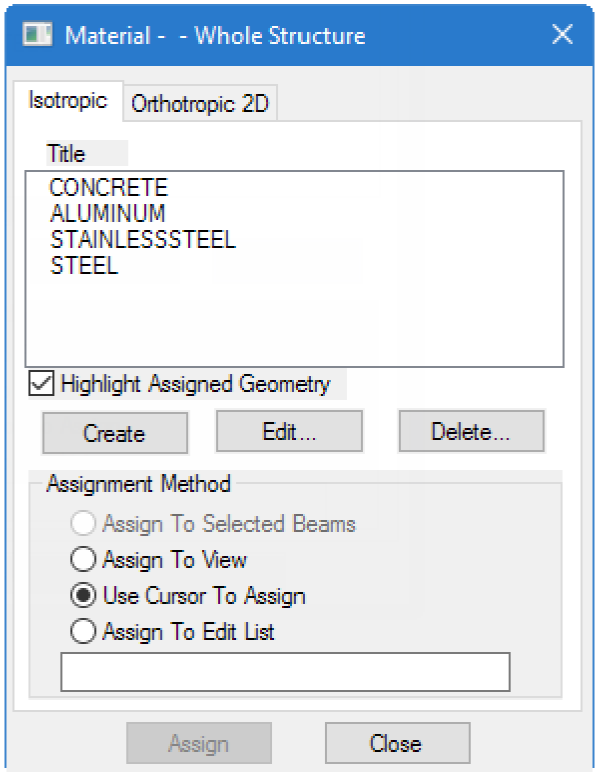
The STAAD input file commands generated are:

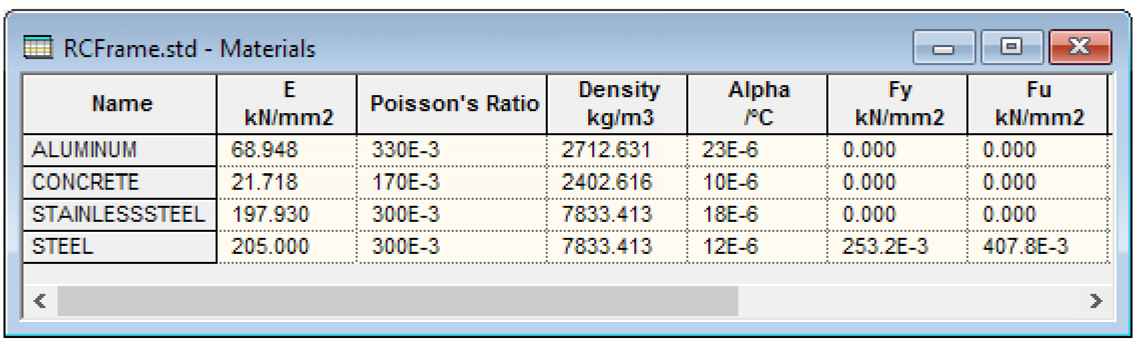
```
DEFINE MATERIAL START
ISOTROPIC CONCRETE-LTW
E 22
POISSON 0.17
DENSITY 1.9e-008
TYPE CONCRETE
STRENGTH RY 1 RT 1 FCU 0.025
END DEFINE MATERIAL
```

You will create a material definition for a lightweight concrete to use for all members.

1. Select the **Materials** page in the **Analytical Modeling** page control bar.

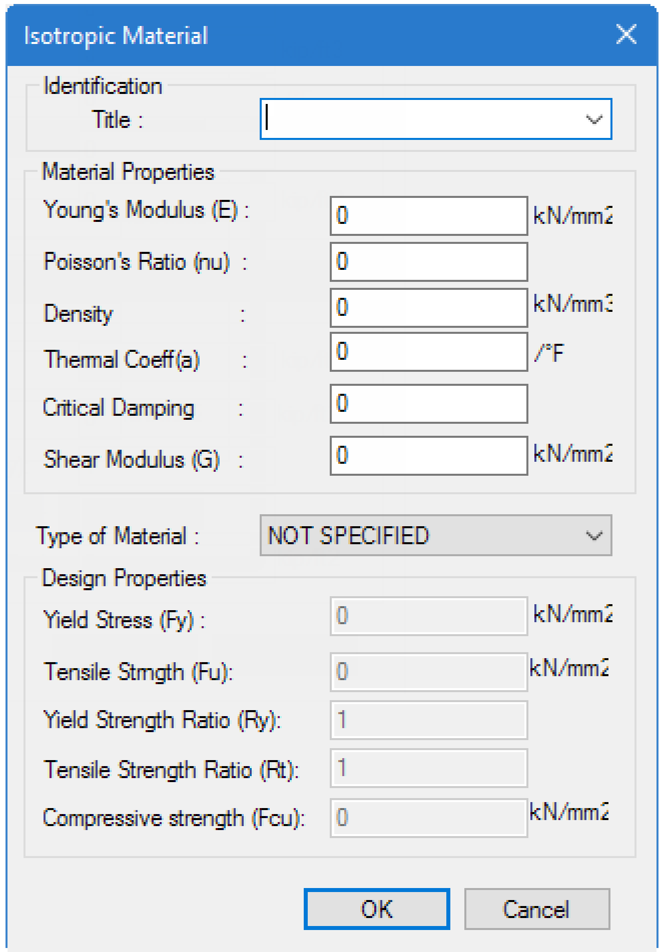
   The **Material - Whole Structure** dialog and the **Materials** table open.





2. In the **Material - Whole Structure** dialog **Isotropic** tab, click **Create**.

   The **Isotropic Material** dialog opens.



3. Input the required, user-defined material values for a lightweight concrete:
   a. Type `CONCRETE-LTW` in the **Title** field.
   b. Type 22 in the **Young's Modulus (E)** field.
   c. Type `0.17` in the **Poisson's Ratio (nu)** field.
   d. Type `1.9e-008` in the **Density** field.
   e. Select **CONCRETE** from **Type of Material** drop-down list.
   f. Type **0.025** in the **Compressive strength (Fcu)** field.

   Leave the remaining property fields as zero for this tutorial exercise. The shear modulus is calculated based on E and Poisson's ratio and the thermal coefficient and critical damping default to zero. Refer to the TR.26.1 Define Material (on page 2303) for details.

   Tip: The material definition can be edited later by selecting the material in the isotropic materials list and clicking Edit.

   The new material is added to the **Material** table and the list of isotropic materials.

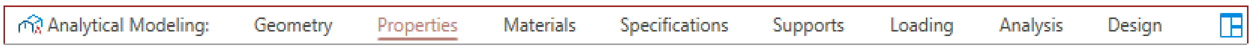
T.2 Specifying member properties

Next you will assign cross section properties for the beams and columns.

The STAAD input file commands generated are:

```
MEMB PROP
1 4 PRIS YD 300 ZD 275
2 5 PRIS YD 350 ZD 275
3 PRIS YD 350
...
MATERIAL CONCRETE-LTW ALL
```

1. Select the **Properties** page in the Analytical Modeling page control bar.



The **Properties - Whole Structure** dialog opens.

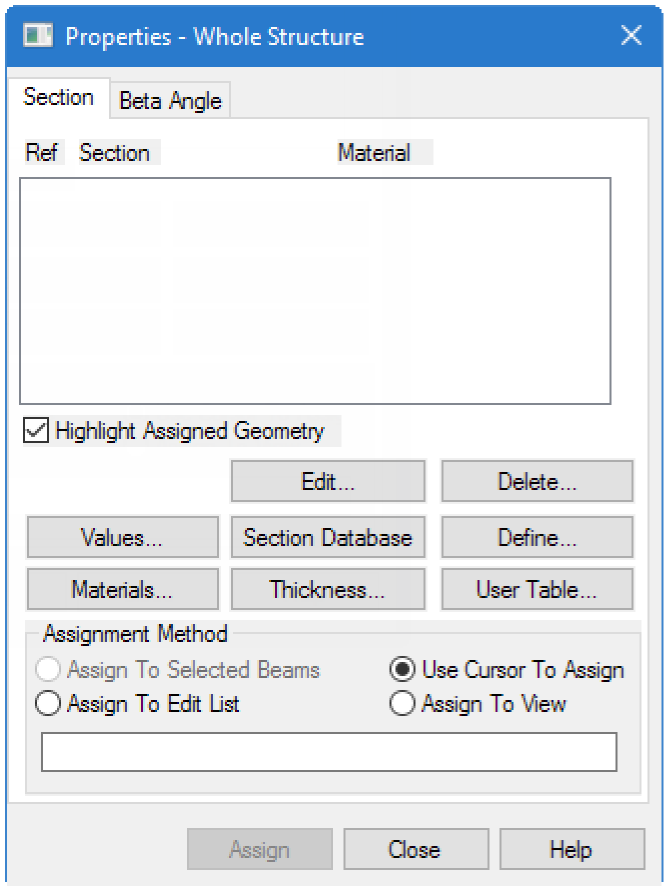


Figure 43:

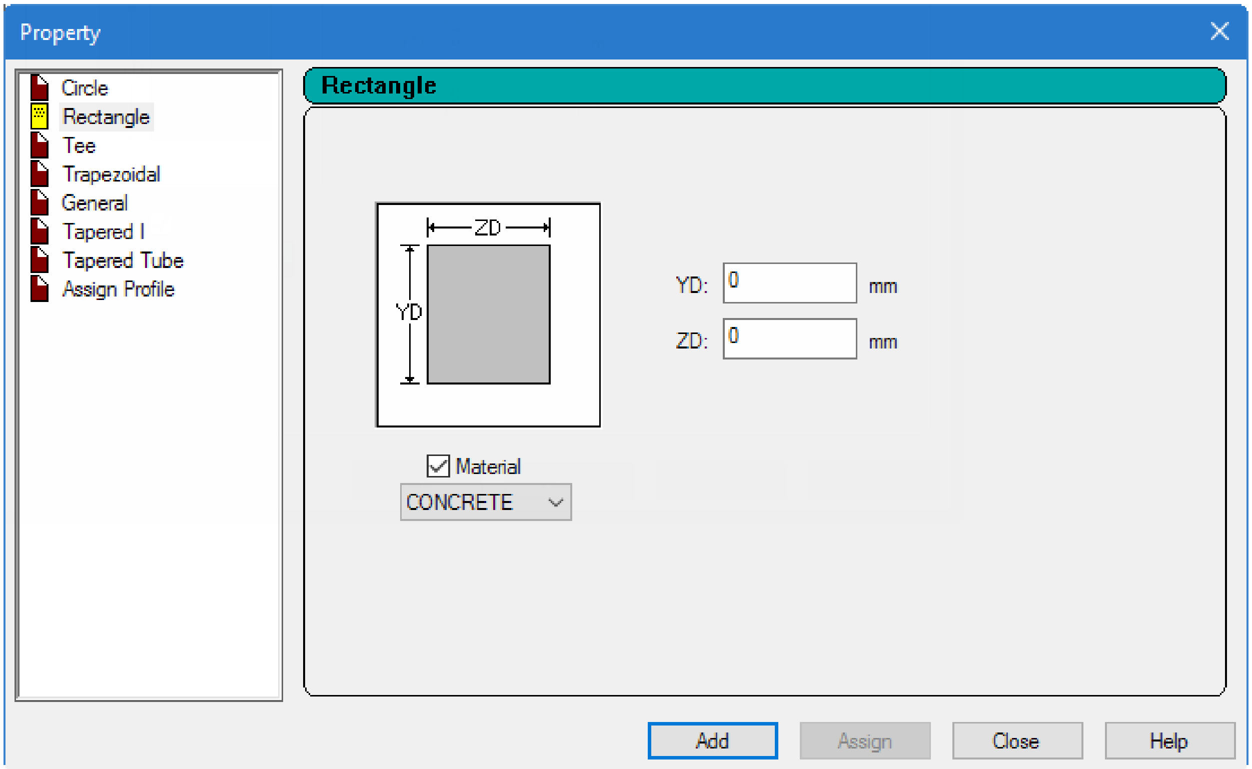
2. Either:

on the **Specifications** ribbon tab, click the **Prismatic** tool in the **Beam Profiles** group

or

click **Define**

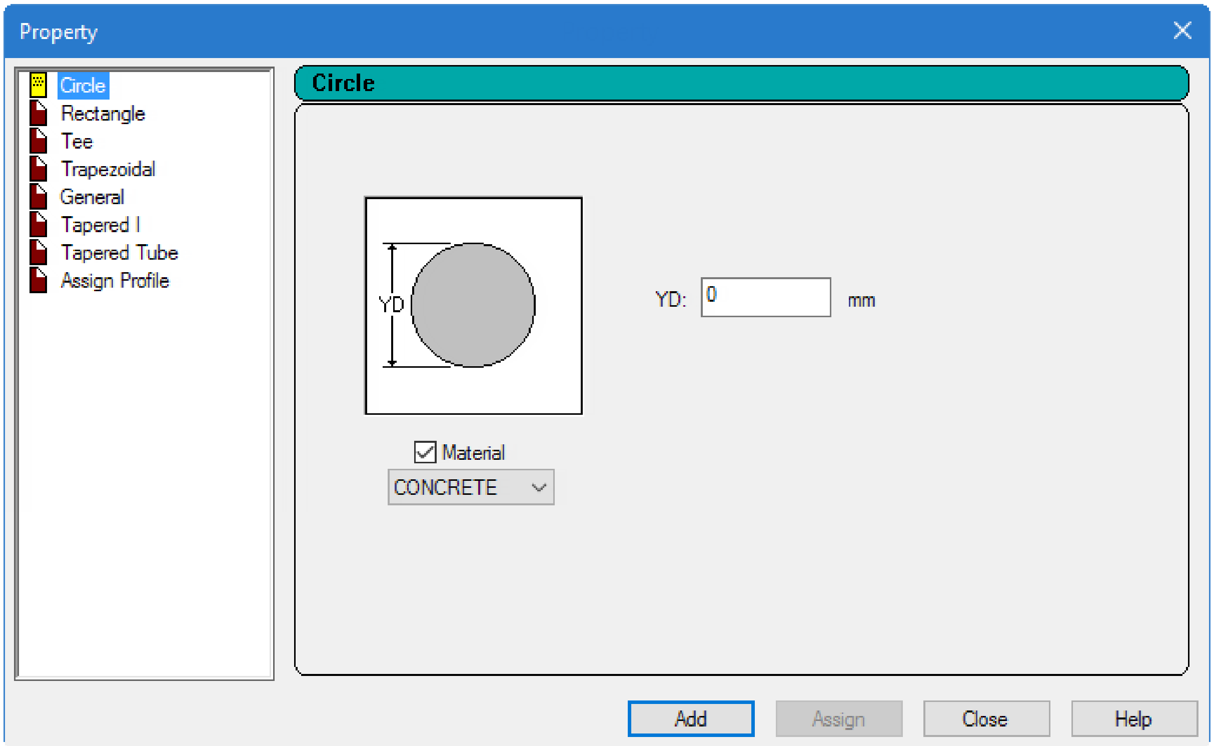
The **Property** dialog opens.



3. Define a 30 cm x 27.5 cm rectangular, concrete beam:
   a. Select the **Rectangle** tab.
   b. Check the **Material** option and select **CONCRETE-LTW** from the list.

   Note: This will automatically assign the user-defined material created in a previous procedure to the sections.

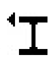
   c. Type `300` (mm) in the **YD** field.
   d. Type `275` (mm) in the **ZD** field.
   e. Click **Add**.
4. Repeat step 3 to define the beam member property, this time with a depth (**YD**) of `350` and a width (**ZD**) of `275`.
5. Define a 35 cm round, concrete column:
   a. Select the **Circle** tab.



b. Check the **Material** option and select **CONCRETE-LTW** from the list.
c. Specify the diameter (**YD**) as `350 mm`.
d. Click **Add**.

6. Click **Close**.
7. Assign the 30 cm x 27.5 cm shape to members 1 and 4 (end columns):

a. Select **Rect 0.30x0.28** in the list of sections.
b. Select the **Use Cursor to Assign** option in the Assignment Method group.
c. Click **Assign**.

The mouse pointer changes to 

d. In the view window, click on members 1 and 4.
e. To stop assigning members, either:

click **Assigning**

or

press the **<Esc>** key.

8. Repeat step 7 to assign the 0.35cm x 27.5 cm rectangular shape to members 2 and 5 (beams).
9. Repeat step 7 to assign the 0.35cm circular shape to members 3 (center column).

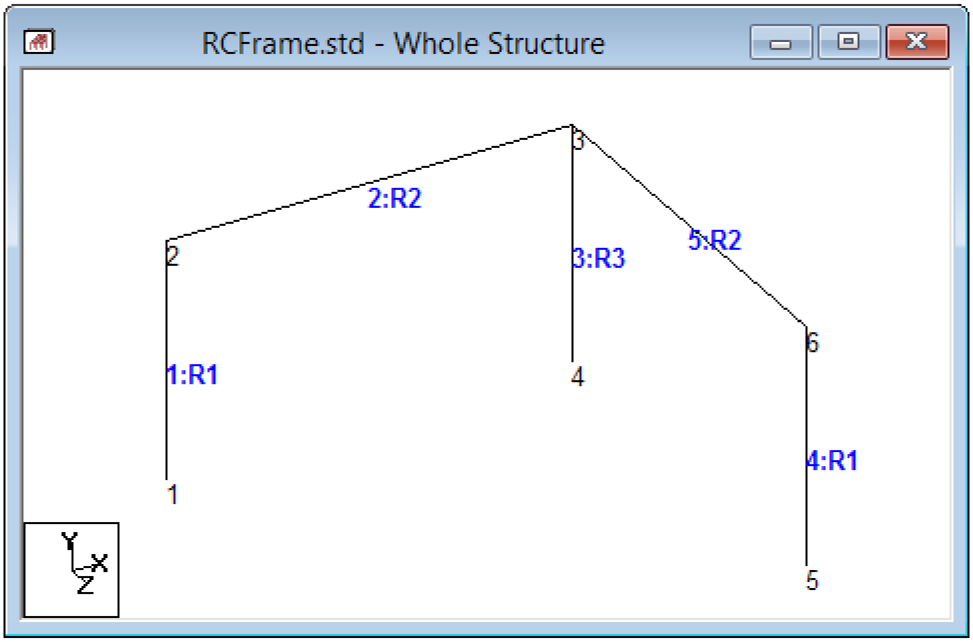


Figure 44: The frame with sections assigned

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Specifying geometric constants

You need to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis. This requires applying a beta angle of 90 degrees.

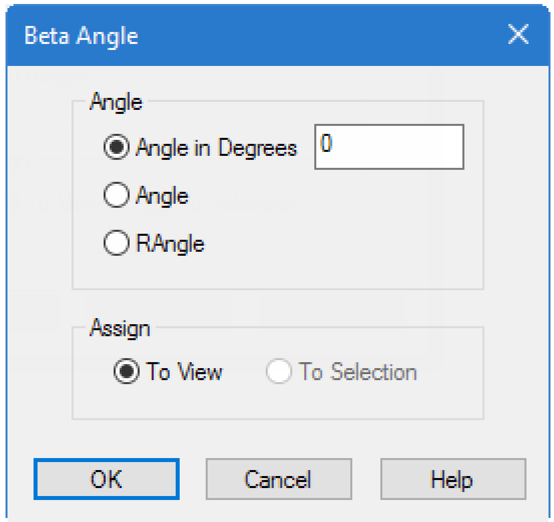
In the absence of any explicit instructions, STAAD.Pro will orient the beams and columns of the structure in a predefined way. Orientation refers to the directions along which the width and depth of the cross section are aligned with respect to the global axis system. The rules which dictate this default orientation are explained in General Engineering Theory (on page 2085).

The STAAD input file commands generated are:

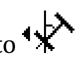
```
BETA 90 MEMB 4
```

1. Select the **Properties** page in the **Analytical Modeling** page control bar.
   The **Properties** dialog opens.
2. On the **Properties** dialog, select the **Beta Angle** tab.
3. Click **Create Beta Angle**.

   The **Beta Angle** dialog opens.



4. Type `90` in the **Angle in Degrees** field.
5. Click **OK**.
   The property **Beta 90** is added to the beta angles list.
6. Select the property **Beta 90**.
7. Assign the beta angle to member 4:
   a. Select the **Use Cursor to Assign** option in the Assignment Method group.
   b. Click **Assign**.

      The mouse pointer changes to 
   c. In the view window, click on member 4.
   d. To stop assigning members, either:

      click **Assigning**

      or

      press the **<Esc>** key.
8. To view the orientation of the member local axes:

   right-click in the view window and select **Labels** from the pop-up menu, check the option for **Beam Orientation**, and click **Apply**.

   or

   press **<Shift+O>**.

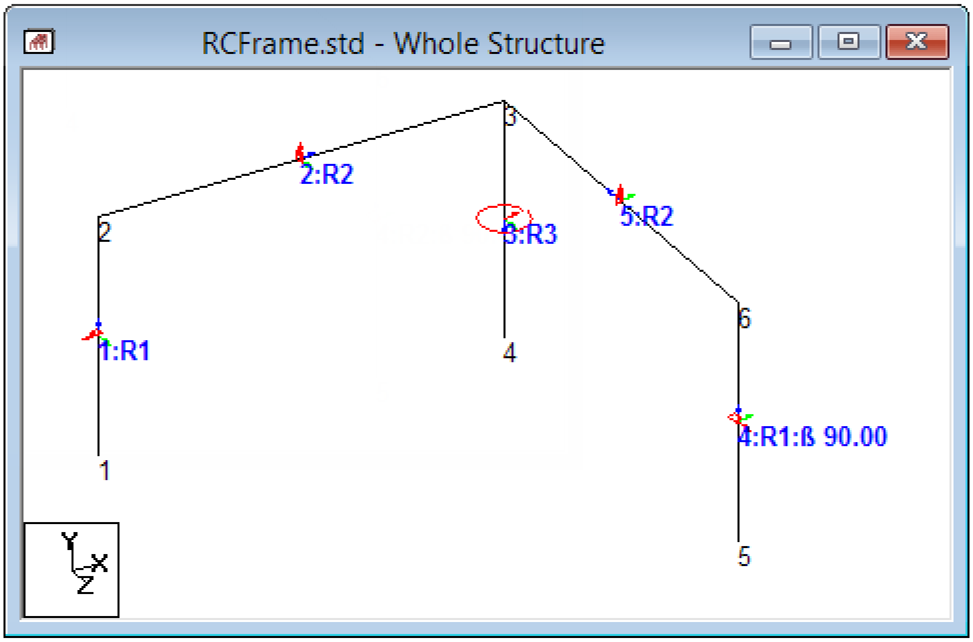


Figure 45: The Beta Angle applied to Member 4

Alternatively, you can assign a beta angle to a selected member set. On the **Specification** ribbon tab, select the **Beam** > **Beta Angle** tool in the **Specifications** group. Type the beta **Angle in Degrees** to use. Make sure that the **To Selection** option is selected and then click **OK**.

T.2 Specifying Supports

The base nodes of all the columns are restrained against translation and rotation about all the 3 global axes (i.e., fixed supports at those nodes).

The STAAD input file commands generated are:

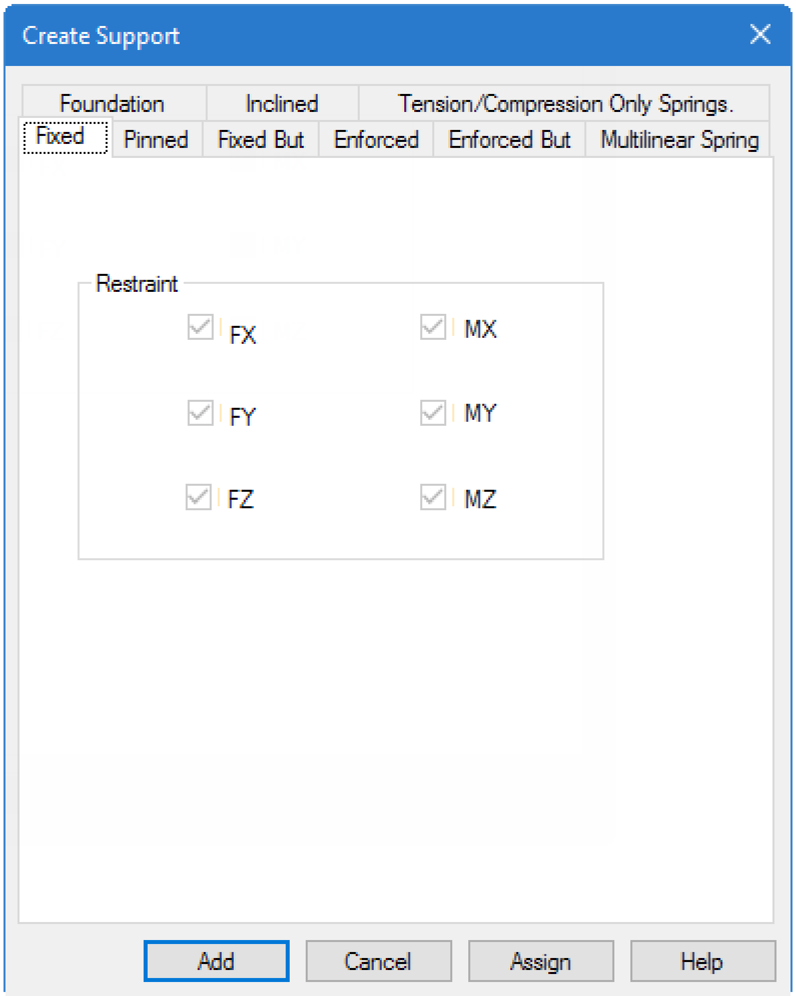
```
SUPPORTS
1 4 5 FIXED
```

1. On the **Select** ribbon tab, select the **Node Cursor** tool in the **Cursors** group.

2. In the View window, hold **<Ctrl>** and click on nodes 1, 4, and 5 to select them.
3. In the **Supports** dialog, click **Create**.

   The **Create Support** dialog opens.



4. Select the **Fixed** tab (selected by default) and then click **Assign**.

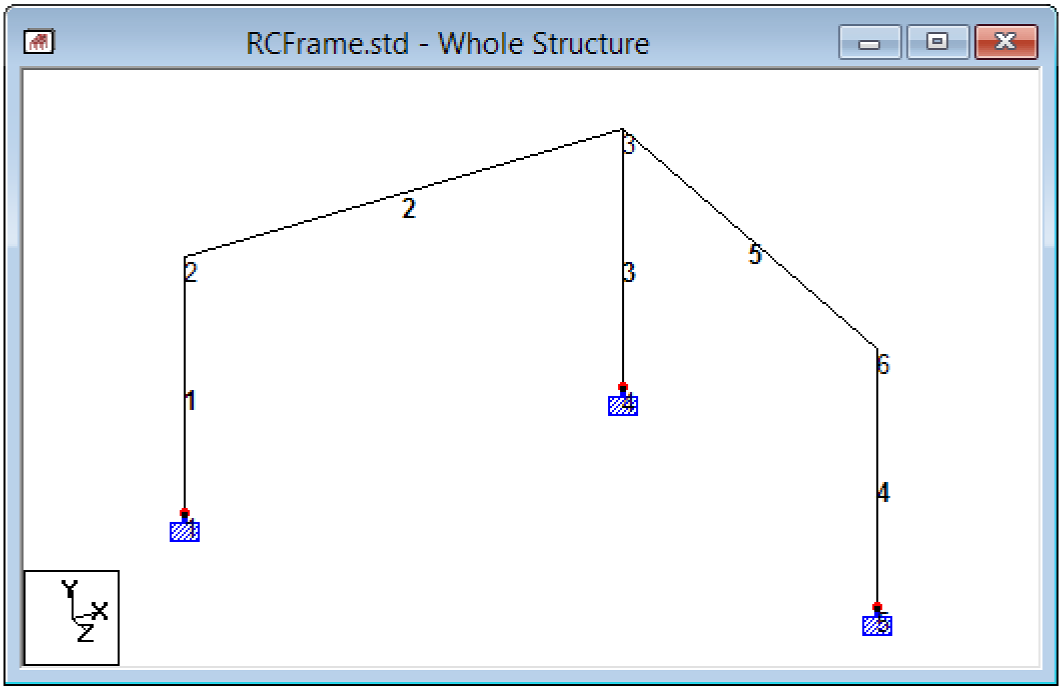


Figure 46: The structure with the supports assigned

5. Click any open space in the View window to deselect all selected nodes.

   This prevents accidental assignment of unwanted data to those nodes.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Specifying Loads

Five load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial.

The STAAD input file commands generated are:

```
UNIT METER KG
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1
MEMBER LOAD
2 5 UNI GY -400
LOAD 2 LIVE LOAD
MEMBER LOAD
2 5 UNI GY -600
LOAD 3 WIND LOAD
MEMBER LOAD
1 UNI GX 300
4 UNI GX 500
LOAD 4 DEAD + LIVE
REPEAT LOAD
1 1.2 2 1.5
LOAD 5 DEAD + WIND
```

```
REPEAT LOAD
1 1.1 3 1.3
```

T.2 Creating Load Cases 1, 2, and 3

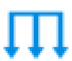
The load values are listed in the beginning of this tutorial in **kg** and **meter** units. Rather than convert those values to the current input units, we will conform to those units.

STAAD.Pro does not allow to change the units while editing load cases. An error message is displayed if this is attempted.

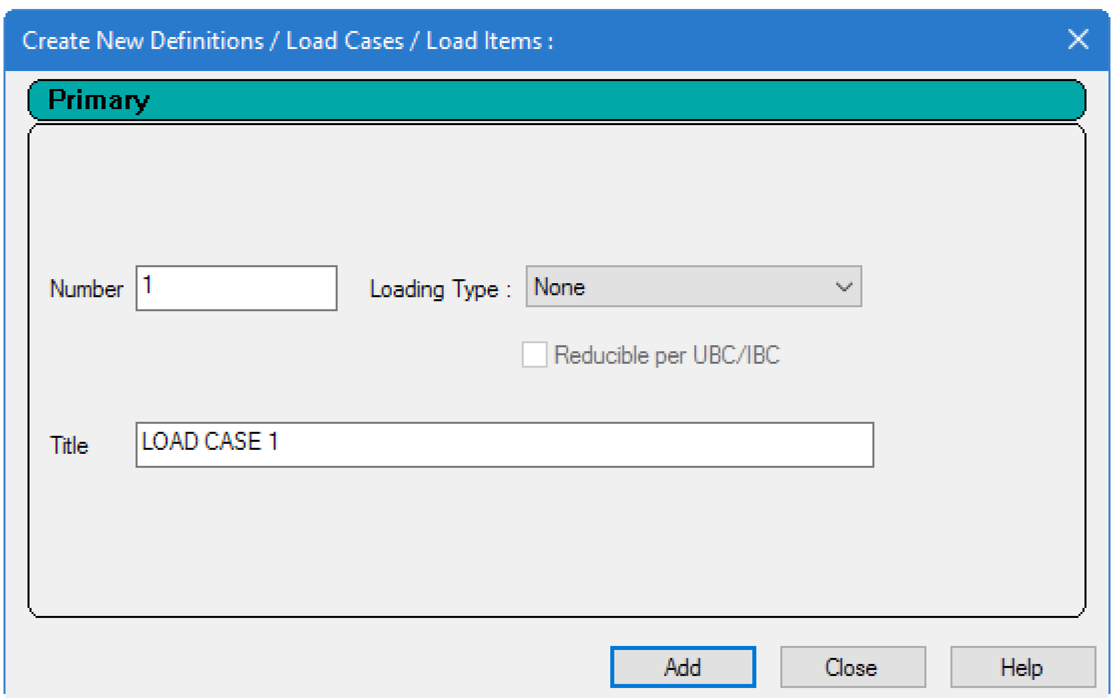
Create the primary load cases for this model.

1. Change the force unit to **Kilogram** and the length unit to **Meter**.

   Refer to T.2 Changing the input units of length (on page 480) for details.
2. On the **Loading** ribbon tab, select the **Primary Load Case** in the **Loading Specifications** group.

   The **Create New Primary Load Cases** dialog opens.

3. Enter the properties for the first load case:
   a. Type `DEAD LOAD` in the **Title** field.

      Leave the **Number** as the default value of 1.

> **Note:** The **Loading Type** list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those

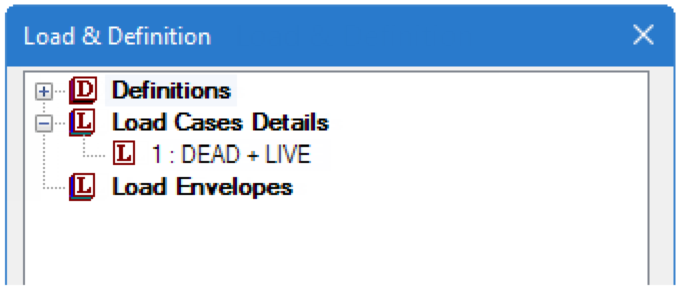
codes. Note that there is a check box labeled **Reducible per UBC/IBC**. This feature is active only when the load case is assigned a Loading Type called **Live** when you create that load case.

Since this tutorial does not use the automatic load combination generation feature, leave the **Loading Type** as **None**.

b. Click **Add**.

c. Click **Close**.

The load case appears under the **Load Cases Details** option in the **Load & Definition** dialog.



4. To create the selfweight dead load, either:

on the **Loading** ribbon tab, select the Load Items tool

a. Select the **1: DEAD LOAD** entry.

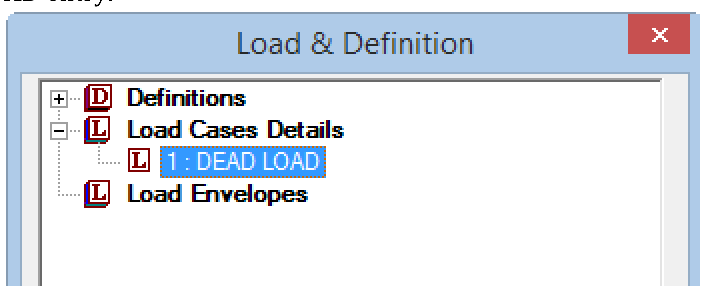


Figure 47:

b. Click **Add**.

The **Add New Load Items** dialog opens with entries for adding load items to the selected load case.

5. Define the selfweight load:

a. In the **Add New Load Items** dialog, select the **Selfweight Load** option under the **Selfweight** item.

b. Select **Y** as the **Direction**.

c. Type `-1.0` in the **Factor** field (multiplier for the calculated selfweight of members).

d. Click **Add**.

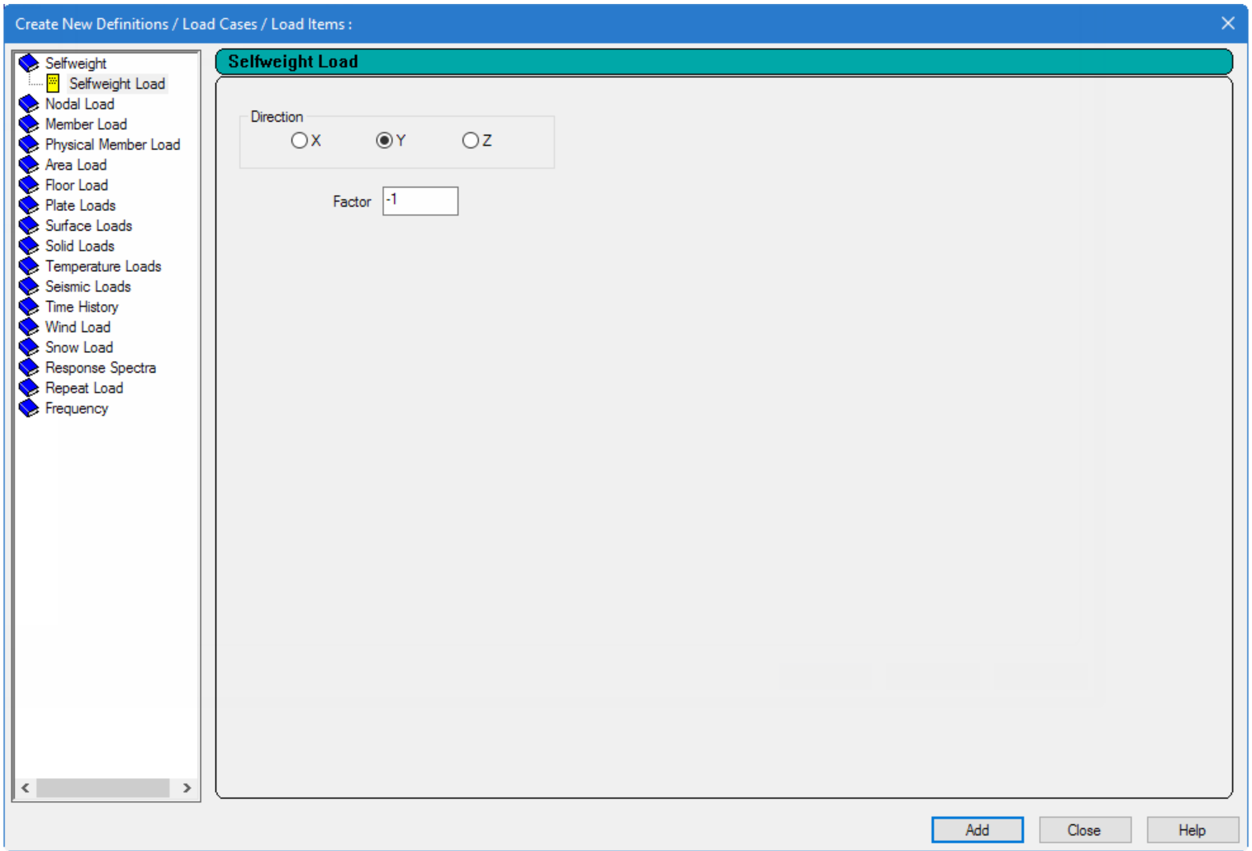


Figure 48:

6. With the **Add New Load Items** dialog still open, add the member loads to be applied to the beams:
   a. In the **Add New Load Items** dialog, select the **Uniform Force** tab under the **Member Load** item.
   b. Select **GY** as the **Direction**.

   Tip: For these members, since the local Y axis coincides with the global Y axis, selecting Y (local member Y) and GY (global Y direction) have the same effect. Press **<Shift+O>** to view the member orientation.

   c. Type `-400` (kg/m) in the **W1** field (magnitude of the uniform force).

      The negative value indicates the load acts "down".

      Leave the offset fields (d1 through d3) as zero.
   d. Click **Add**.
   e. Click **Close**.

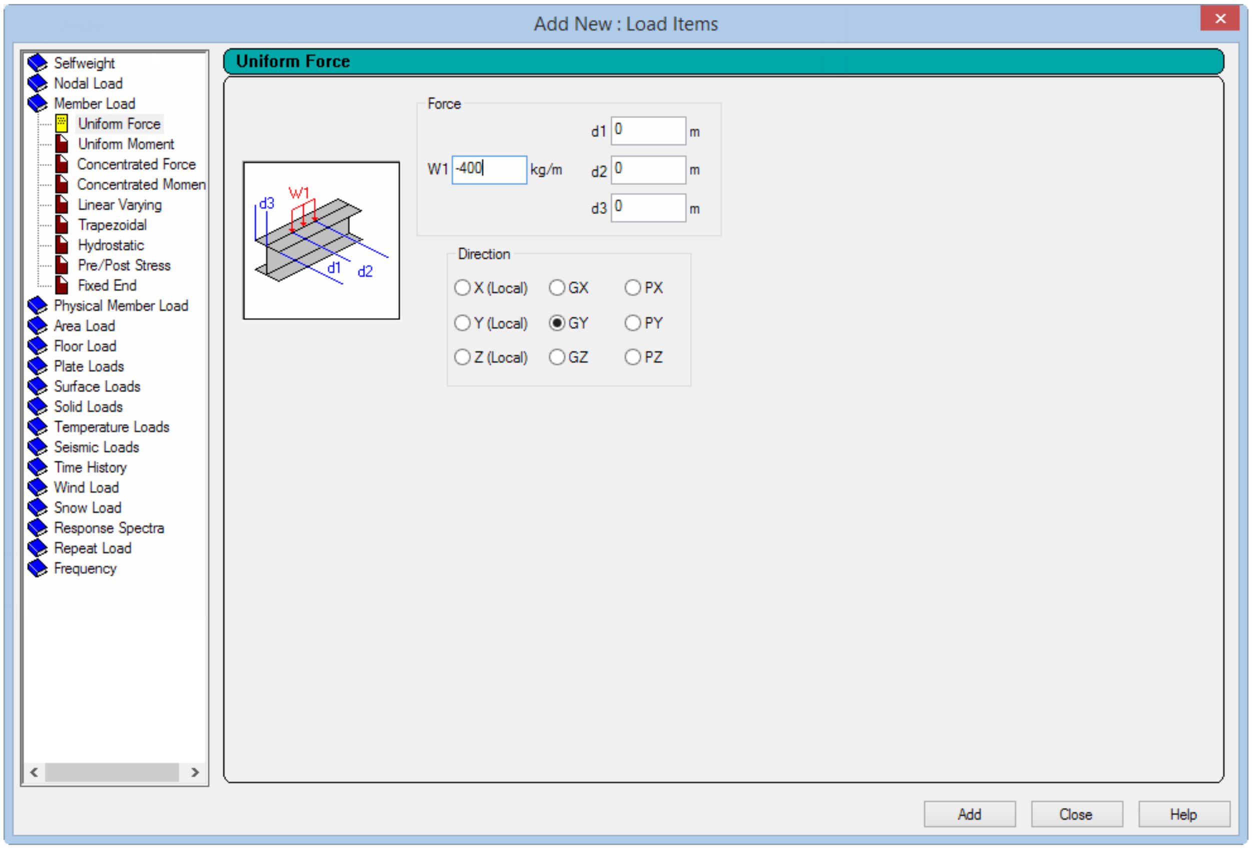


Figure 49:

7. Assign the selfweight load to all members:
   a. Select the **SELFWEIGHT Y -1** entry in the **Load & Definitions** dialog.

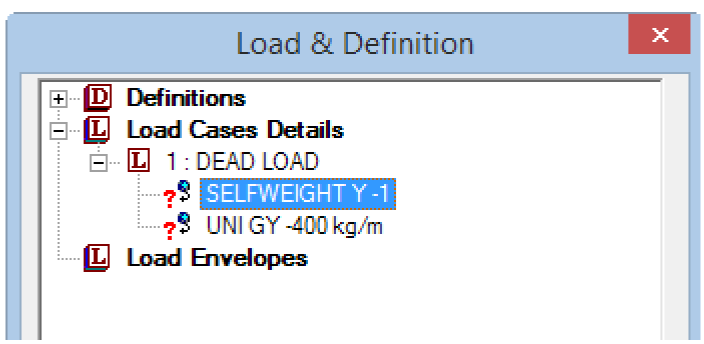


Figure 50:

   b. Select the **Assign to View** option.
   c. Click **Assign**.

      A message box opens to opens prompting you to confirm you want to make this assignment.
   d. Click **Yes**.
   e. Click anywhere in the View window away from the members to deselect all members.
8. Assign the member load to members 2 and 5:
   a. Select the **UNI GY -400 kg/m** entry in the **Load & Definitions** dialog.

b. Press and hold **<Ctrl>** and then select members 2 and 5 in the View window.

The **Beams Cursor** tool  is automatically selected when the **Load & Definition** dialog is opened.

c. Select the **Assign to Selected Beams** option.

d. Click **Assign**.

A message box opens to opens prompting you to confirm you want to make this assignment.

e. Click **Yes**.

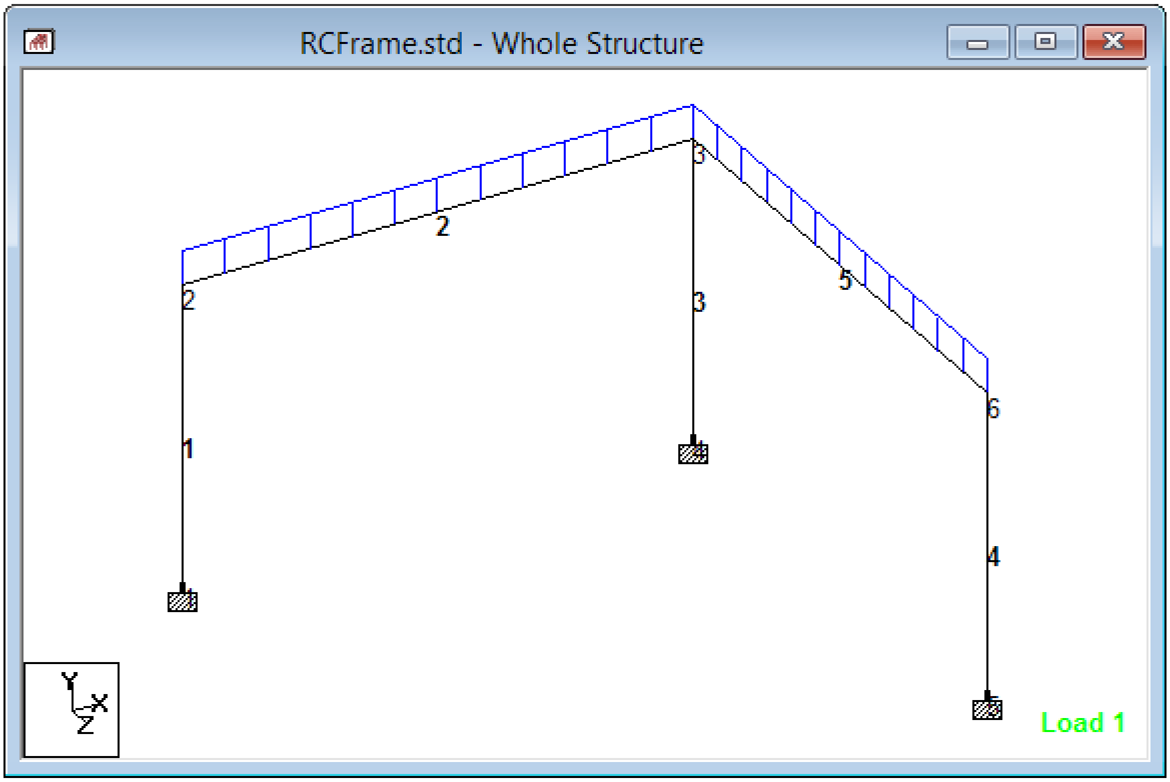


Figure 51: Structure after uniform force is assigned to beams

Tip: Use a scale of 6 kN/m per m for distributed forces if the loads are not visible or are too large for the View window when viewing the entire structure.

9. Repeat steps 3, 4, 6, and 8 to create a new load case titled `Live Load` which contains a uniform force of `-600` kg/m in the global Y direction and is applied to members 2 and 5.

Note: Be sure to select the **Live Load** on the **Loading** ribbon tab **Display** group to add load items to this load case.

10. Repeat steps 3 through 4 and then 6 through 8 to create a new load case titled `Wind Load` which contains the following two load items:

Note: Be sure to select the **Wind Load** on the **Loading** ribbon tab **Display** group to add load items to this load case.

a uniform force of `+300` kg/m in the **GX** (global X) direction and is applied to member 1
a uniform force of `+500` kg/m in the **GX** (global X) direction and is applied to member 4

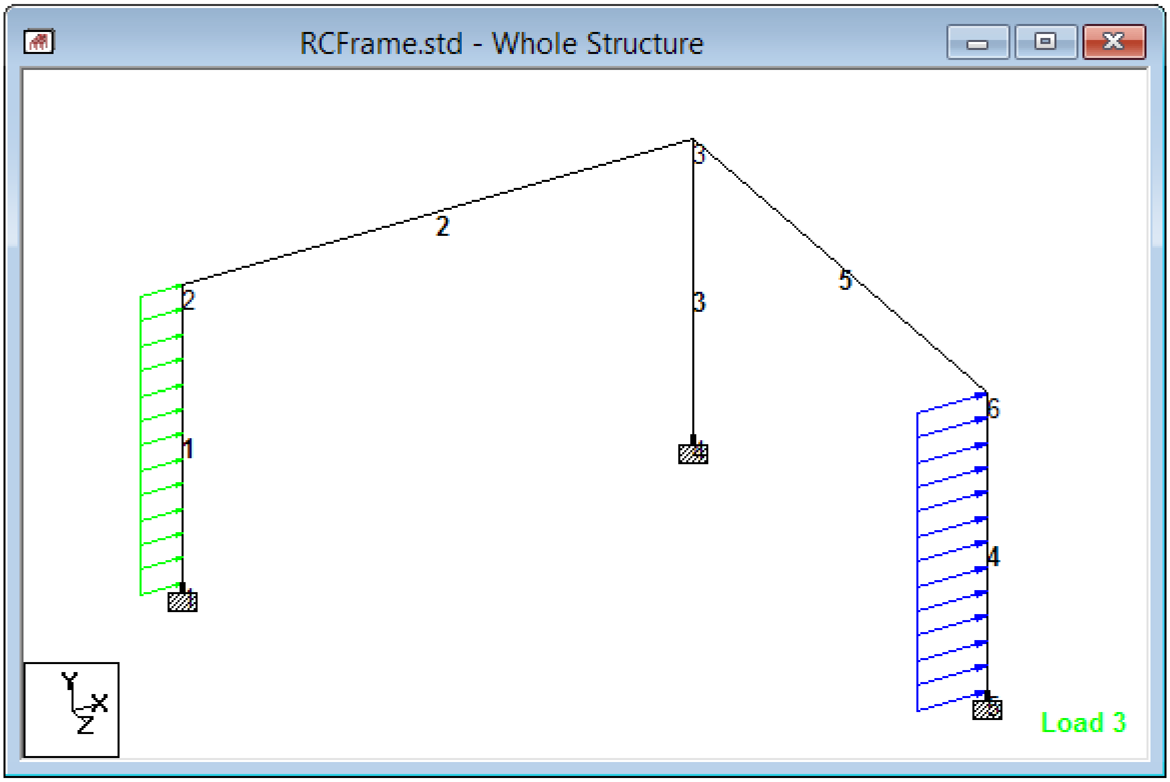


Figure 52: Structure after the wind load items are assigned

T.2 Analysis and Design

The following procedures describe how to add analysis and design commands to the STAAD input file.

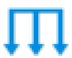
Note: The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

T.2 Creating Load Cases 4 and 5

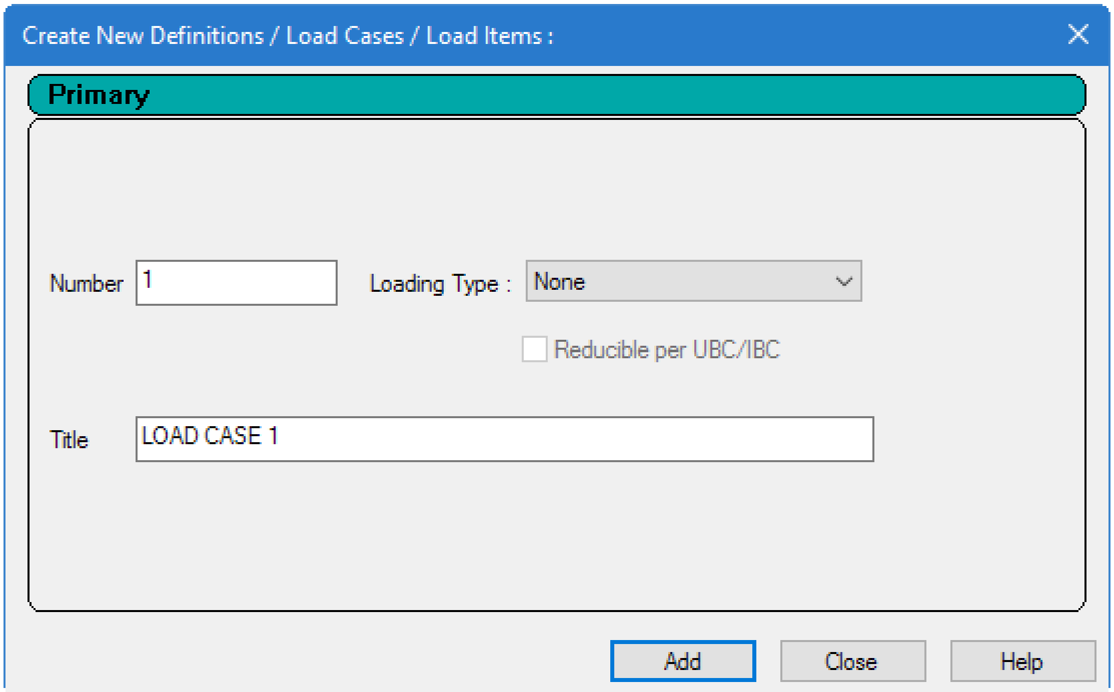
Use repeat load cases for the load combinations used in a nonlinear analysis.

Load cases 4 and 5 are to be generated using a combination load type called `REPEAT LOAD` (as opposed to a standard load combination). The instructions at the beginning of this tutorial specify using a PDelta analysis. A PDelta analysis is a non-linear type of analysis. In STAAD, to accurately account for the PDelta effects arising from the simultaneous action of previously defined horizontal and vertical loads, those previous cases must be included as components of the combination case using the `REPEAT LOAD` type.

1. On the **Loading** ribbon tab, select the **Primary Load Case** in the **Loading Specifications** group.

   The **Create New Primary Load Cases** dialog opens.



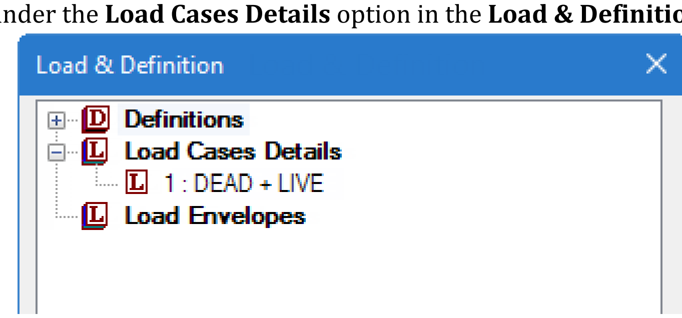
2. Enter the properties for the load case:
   a. Type `DEAD + LIVE` in the **Title** field.

   Leave the **Number** as the default value of 4.

   > **Note:** The **Loading Type** list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those codes. Note that there is a check box labeled **Reducible per UBC/IBC**. This feature is active only when the load case is assigned a Loading Type called **Live** when you create that load case.

   b. Click **Add**.

   The load case appears under the **Load Cases Details** option in the **Load & Definition** dialog.



3. To create the dead + live repeat load case:
   a. Select the **4: DEAD + LIVE** entry.

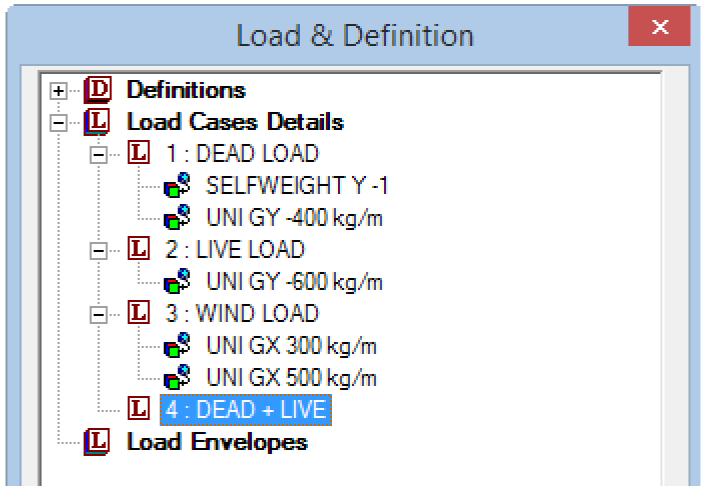


Figure 53:

b. Click **Add**.

The **Add New Load Items** dialog opens with entries for adding load items to the selected load case.

4. Define the repeat load:

a. In the **Add New Load Items** dialog, select the **Repeat Load** option under the **Repeat Load** item.

b. Select **1: DEAD LOAD** as the **Available Load Cases** list and then click **[>]**.
This load is added as Load Case 1 in the **Repeat Load Definition** list.

c. Type `1.2` in the **Factor** field for **Load Case 1** in the **Repeat Load Definition** list.

This indicates that the load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4.

d. Select **2: LIVE LOAD** as the **Available Load Cases** list and then click **[>]**.
This load is added as Load Case 2 in the **Repeat Load Definition** list.

e. Type `1.5` in the **Factor** field for **Load Case 1** in the **Repeat Load Definition** list.

f. Click **Add**.

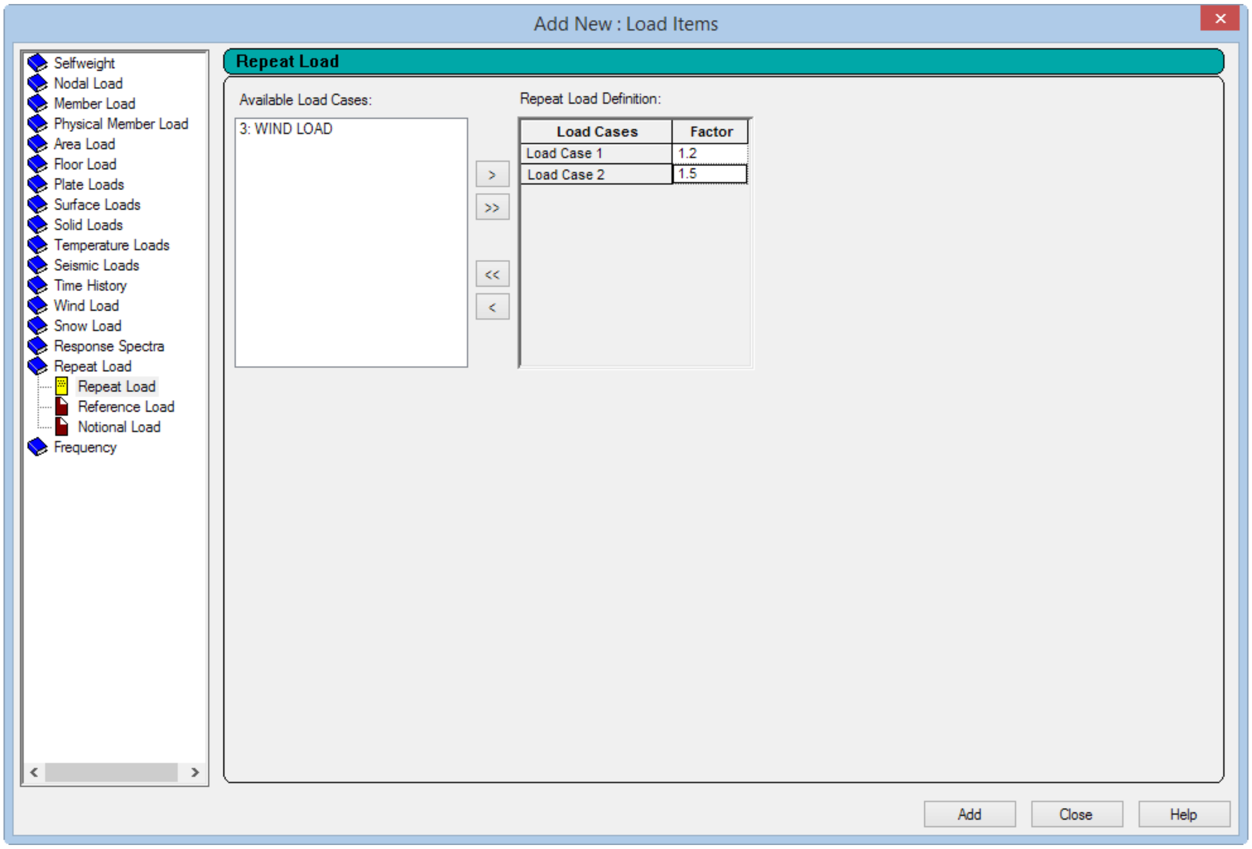


Figure 54:

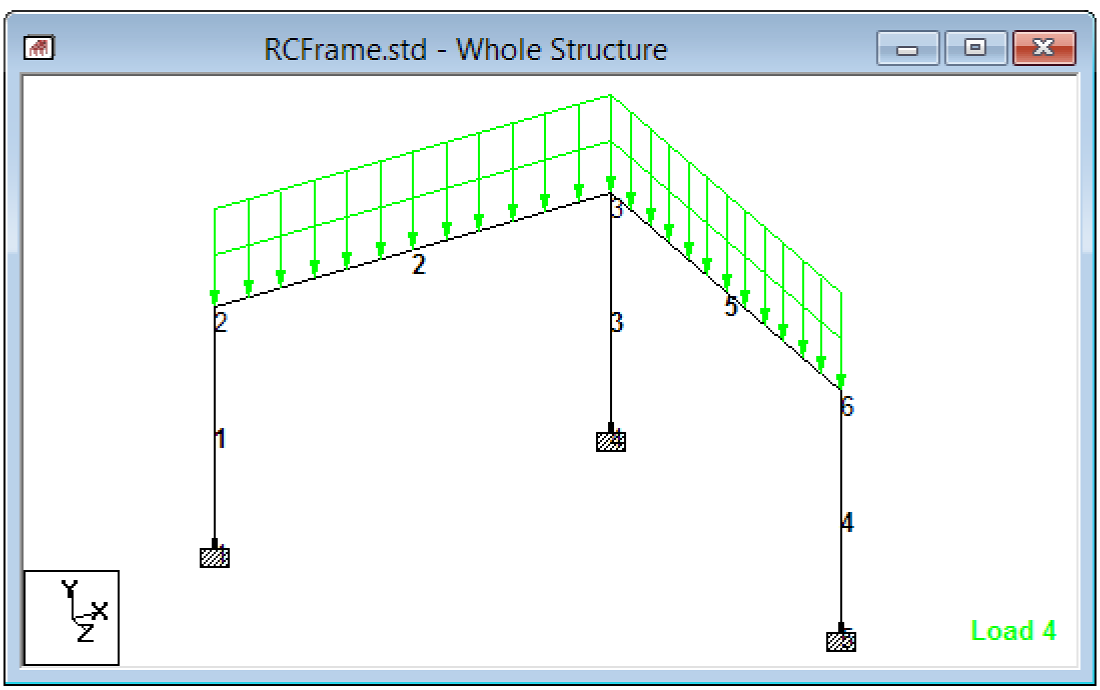


Figure 55: Structure with repeat load case selected

5. Click **Close**.
6. Repeat steps 3 and 5 to create another repeat load case for load case 5, except:

    title load case 5 **Dead + Wind**
    add the Dead Load case with a factor of 1.1
    add the Wind Load case with a factor of 1.3

Note: No member assignment is not necessary for repeat load cases. The members assigned to which the primary load cases are assigned are used for the repetition of the load.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Specifying the analysis type

The analysis type for this structure is called P-Delta.

Since this problem involves concrete beam and column design per the ACI code, second-order analysis is required and has to be done on factored loads acting simultaneously. The factored loads have been created earlier as cases 4 and 5. Now is the time to specify the analysis type.

The STAAD input file commands generated are:

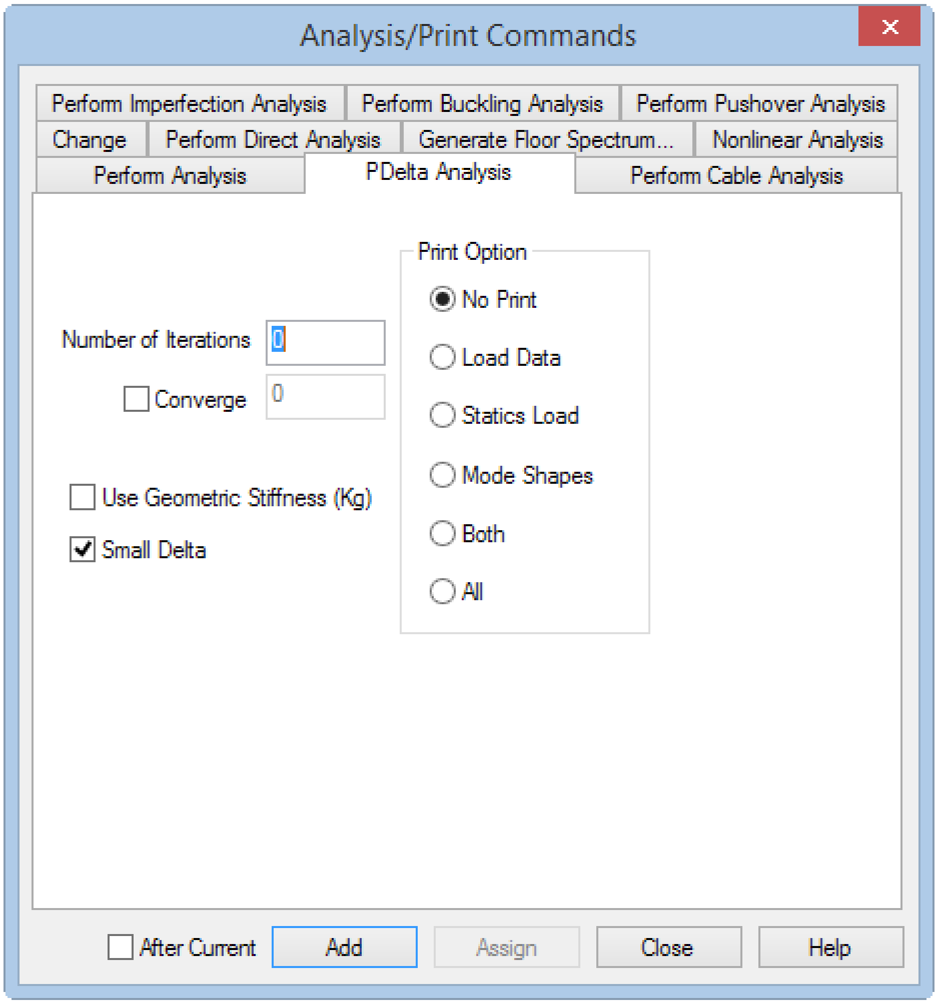
```
PDELTA ANALYSIS SMALLDELTA
```

1. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

    The **Analysis/Print Commands** dialog opens.

    Tip: If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.
2. Select the **PDelta Analysis** tab.



Note: Leave the parameters on this page as their default values and options.

3. Click **Add**.
4. Click **Close**.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Short-listing the load cases to be used in concrete design

The concrete design should only be performed for load cases 4 and 5 since those are the factored cases. Use the load list to instruct the program to use just these cases.

The STAAD input file commands generated are:

```
LOAD LIST 4 5
```

1. On the **Anlaysis and Design** ribbon tab, select the **Load List** tool in the **Analysis Data** group. 

   The **Load List** dialog opens.

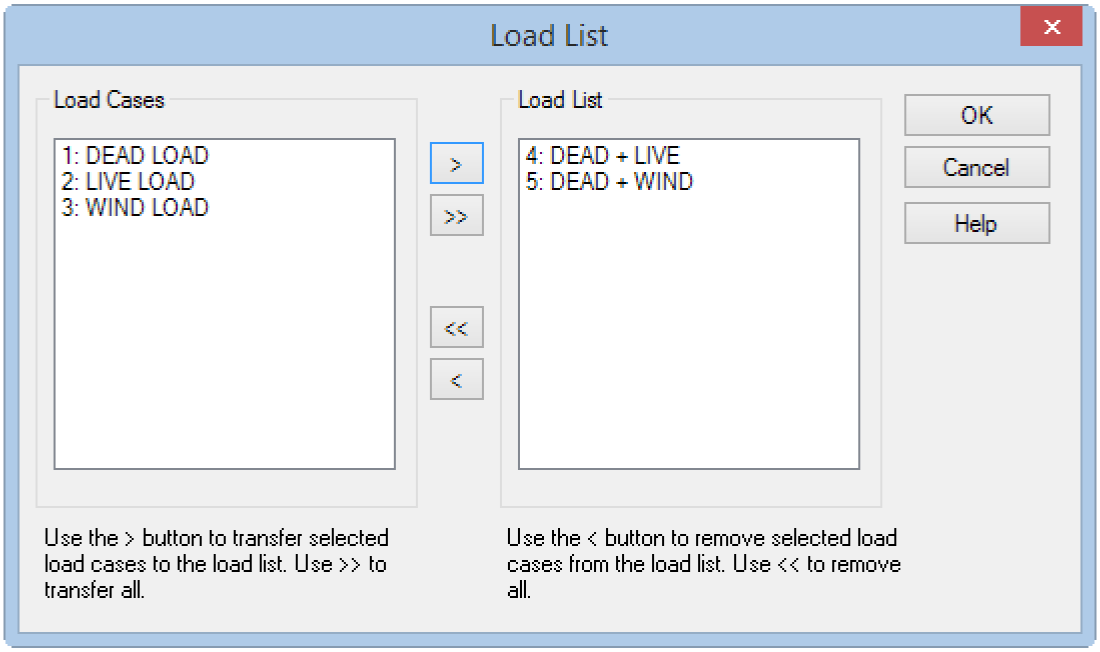


Figure 56:

2. Press and hold the **<Ctrl>** key and then select load cases **4: DEAD + LIVE** and **5: DEAD + WIND**.
3. Click **[>]**.
   Load cases 4 and 5 will be selected and placed in the **Load List** selection box.
4. Click **OK**.

T.2 Specifying concrete design parameters

Concrete design parameters allow you to directly specify some of the terms used in the equations for concrete member design.

For example, the grade of concrete or the maximum size of reinforcing bar you want to use.

Note: This tutorial uses the 2011 edition of the American Concrete Institute 318 specification for design (ACI 318-11). Details on the code parameters can be found in D1.F.3 Design Parameters (on page 1201).

The STAAD input file commands generated are:

```
UNIT MMS NEWTON
CODE ACI
CLB 30 ALL
CLS 25 ALL
CLT 25 ALL
FC 25 ALL
FYMAIN 415 ALL
TRACK 1 ALL
```

1. Set the force units as **Newton** and the length units as **Millimeter**.

   Refer to T.2 Changing the input units of length (on page 480) for details.
2. On the **Analysis and Design** ribbon tab, select **Concrete** in the **Design Commands** gallery.

The **Concrete Design - Whole Structure** dialog opens.

3. On the **Concrete Design - Whole Structure** dialog, select **ACI 318 2011** from the **Current Code** drop-down list.
4. Click **Define Parameters** in the **Concrete Design** dialog.

   The **Design Parameters** dialog opens.

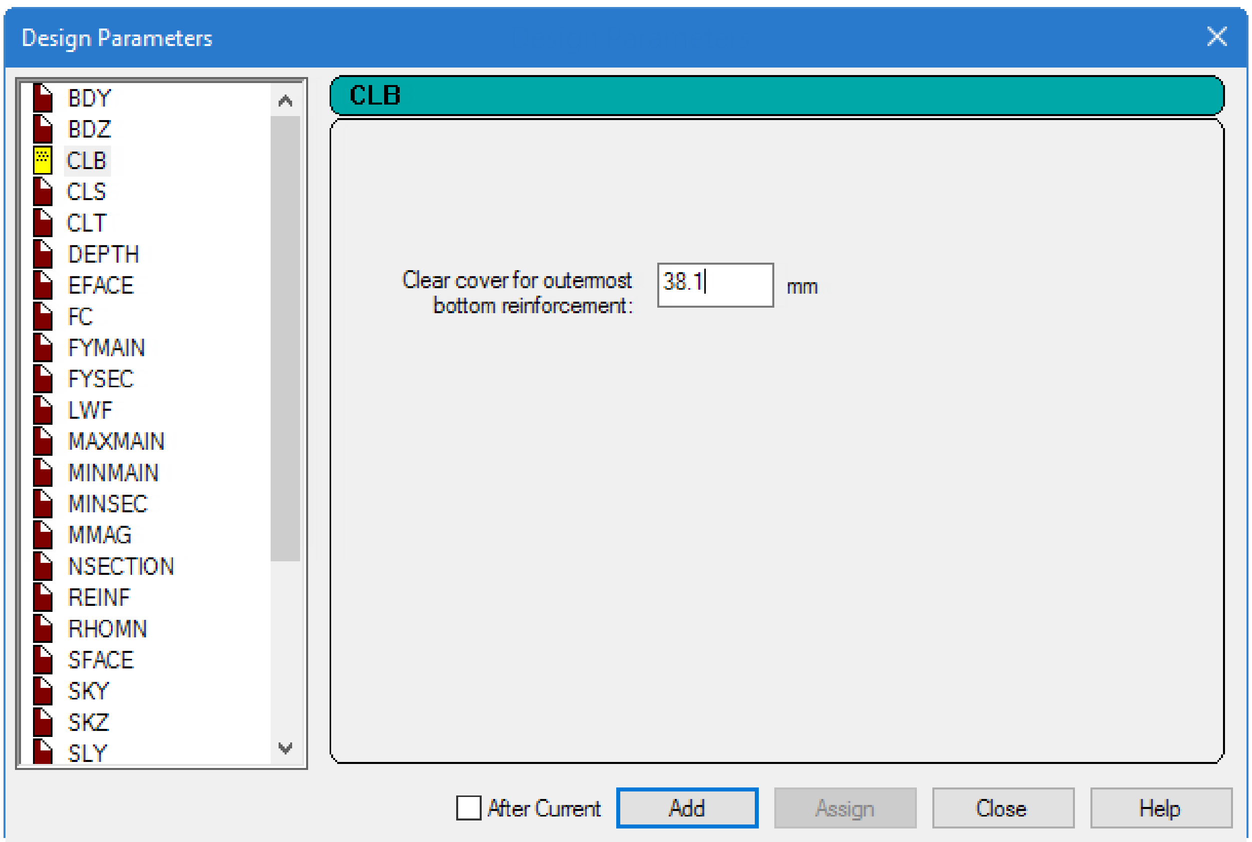


Figure 57:

5. Specify the clear cover on bottom:
   a. Select the **CLB** parameter tab.
   b. Type as `30` (mm) in the parameter value field.
   c. Click **Add**.
6. Repeat step 5 to define the remaining parameters:

Parameter	Value
CLS	25
CLT	25
FC	25
FYMAIN	415
TRACK	1 (select option from list)

7. Click **Close**.

   The **Concrete Design** dialog displays the parameters in the concrete design command.

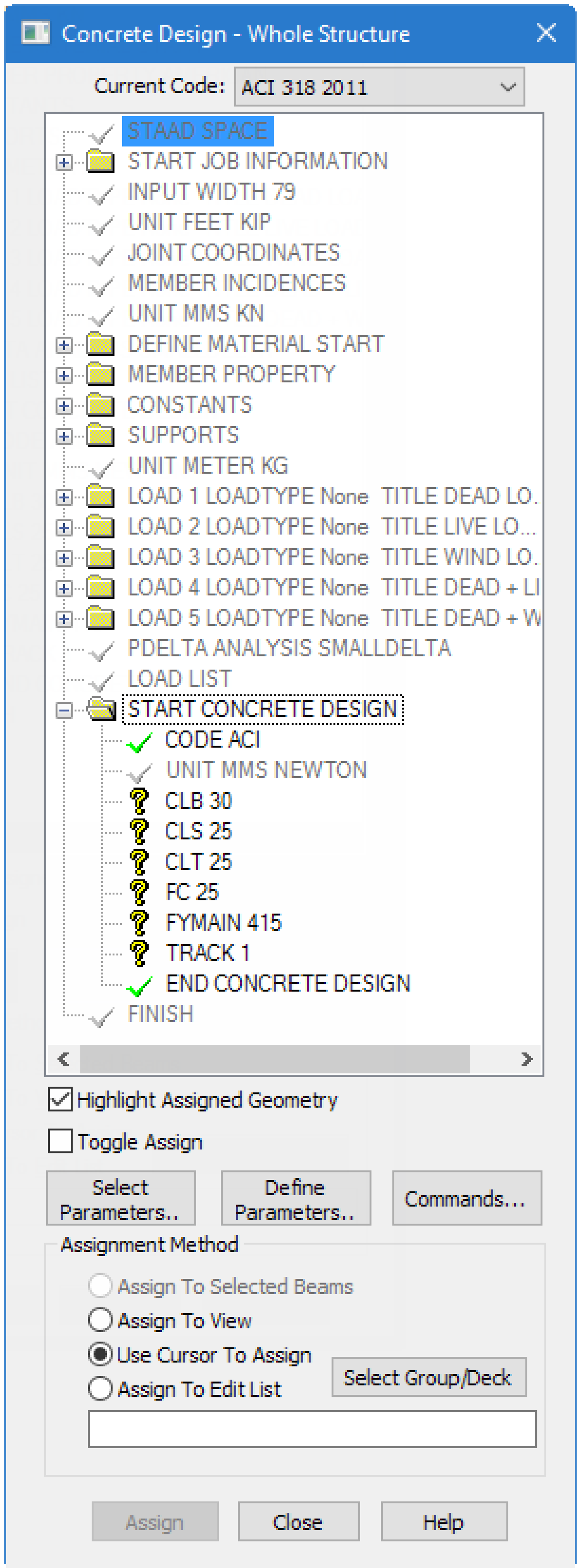

Concrete Design - Whole Structure
Current Code:
ACI 318 2011
STAAD SPACE
START JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
MEMBER INCIDENCES
UNIT MMS KN
DEFINE MATERIAL START
MEMBER PROPERTY
CONSTANTS
SUPPORTS
UNIT METER KG
LOAD 1 LOADTYPE None TITLE DEAD LO.
LOAD 2 LOADTYPE None TITLE LIVE LO...
LOAD 3 LOADTYPE None TITLE WIND LO.
LOAD 4 LOADTYPE None TITLE DEAD + LI
LOAD 5 LOADTYPE None TITLE DEAD + W
PDELTA ANALYSIS SMALLDELTA
LOAD LIST
START CONCRETE DESIGN
CODE ACI
UNIT MMS NEWTON
CLB 30
CLS 25
CLT 25
FC 25
FYMAIN 415
TRACK 1
END CONCRETE DESIGN
FINISH
Highlight Assigned Geometry
Toggle Assign
Select Parameters..
Define Parameters..
Commands...
Assignment Method
Assign To Selected Beams
Assign To View
Use Cursor To Assign
Assign To Edit List
Select Group/Deck
Assign
Close
Help

Figure 58:

8. Assign these parameters to all the members in the model:
   - **a.** Select the first unassigned parameter in the **Concrete Design | Whole Structure** dialog.
   - **b.** Select the **Assign to View** option.
   - **c.** Click **Assign**.

     A message dialog opens confirming you want to make this assignment.
   - **d.** Click **Yes**.
   - **e.** Repeat steps 8a though 8d for each unassigned parameter to assign them to all members.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.2 Specifying design commands

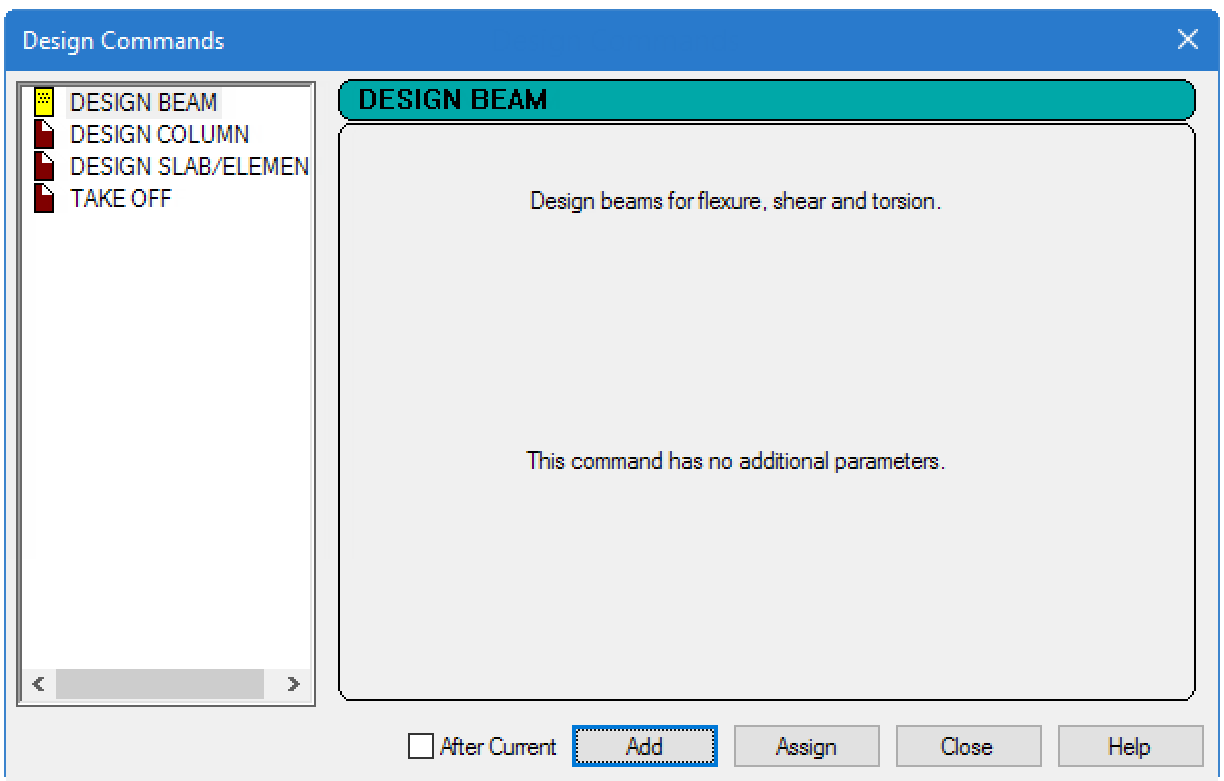
Design commands are the actual instructions for the design of beams and columns.

The STAAD input file commands generated are:

```
DESIGN BEAM 2 5
DESIGN COLUMN 1 3 4
```

1. On the **Concrete Design - Whole Structure** dialog, click **Commands**.

   The **Design Commands** dialog opens.



2. Select the **Design Beam** tab and then click **Add**.
3. Select the **Design Column** option and then click **Add**.
4. Click **Close**.

5. Assign the Design Beam command to the beams:
   a. Press and hold the **<Ctrl>** key and click members 2 and 5

   > **Tip:** The **Beams Cursor** tool  is selected when the **Concrete Design - Whole Model** dialog opens.

   b. Select the **Assign to Selected Beams** option.
   c. Click **Assign**.

      A message box opens to opens prompting you to confirm you want to make this assignment.
   d. Click **Yes**.
6. Repeat steps 5 to assign the **Design Column** command to members 1, 3, and 4.

> **Tip:** Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

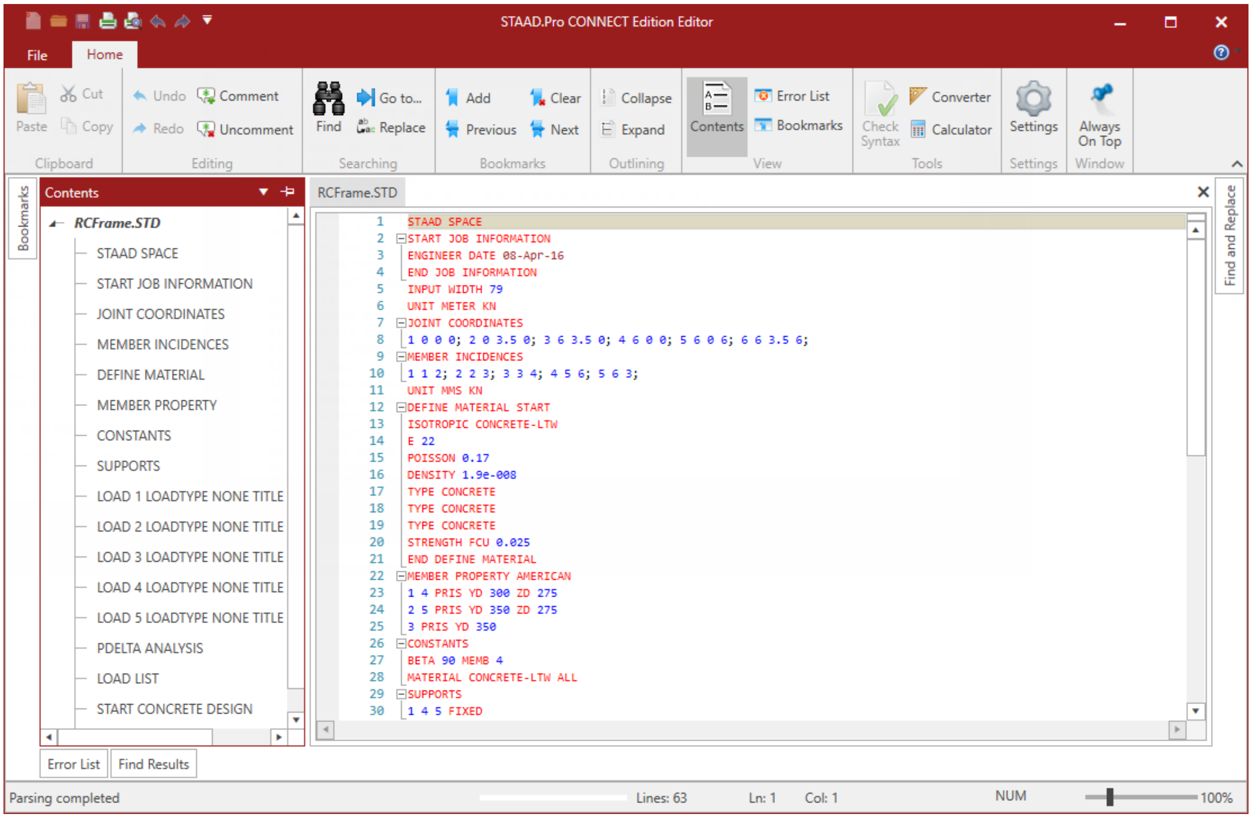
T.2 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the **Utilities** ribbon tab, select the **Command File** tool in the **Edit** group.

   The **STAAD Editor** window opens.



2. (Optional) You modify the data of the structure in this editor if necessary.
3. Select **File** > **Exit Editor** in the STAAD.Pro Editor window to close.

As stated in " T.2 Methods of creating the model (on page 463) ", you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to " T.2 Performing the analysis and design (on page 513) " where you will perform the analysis and design on this model.

T.2 Creating the model using the command file

As an alternatively to the procedures described in the proceeding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

Note: A STAAD input file is a plain text file that uses the `.std` file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.

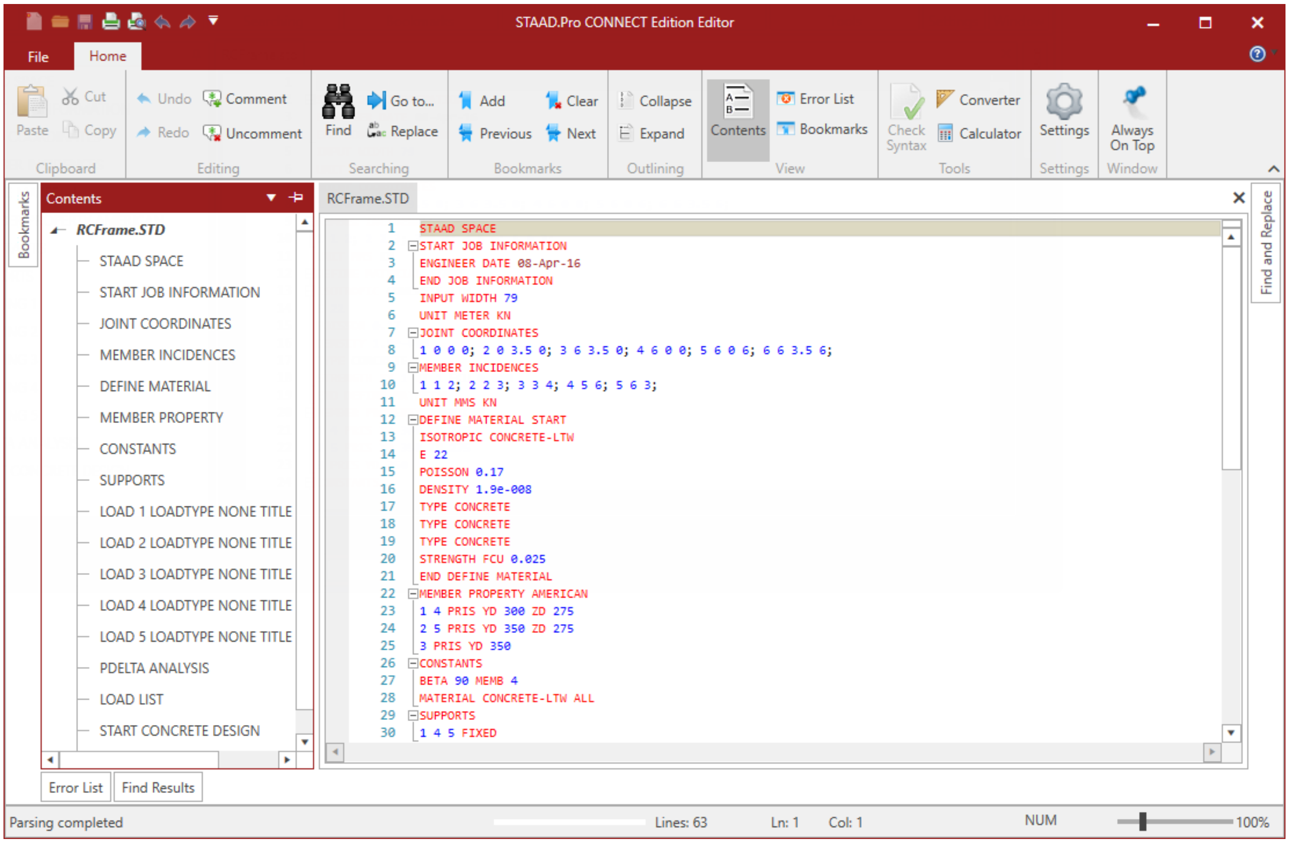


Figure 59:

To start a new STAAD input file using the STAAD.Pro Editor, follow the procedure described in T.2 Creating a new structure (on page 465). Then select the **Command File** tool in the **Edit** group on the **Utilities** ribbon tab. The STAAD.Pro Editor window opens with the basic commands for your model entered.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are *not* case sensitive (i.e., they may be typed in upper *or* lower case letters). By convention, this and most input files use all caps, though.

For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain English for later reference. By convention, the required letters in a command or keyword are underlined here (“`PLANE`” = “`PLA`” = “`plane`” = “`pla`”).

```
STAAD SPACE RC FRAMED STRUCTURE
```

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNIT METER KN
```

Specifies the unit to be used.

```
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 6 3.5 0
4 6 0 0 ; 5 6 0 6 ; 6 6 3.5 6
```

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables you to provide multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 ; 2 2 3 ; 3 3 4
4 5 6 ; 5 6 3
```

Defines the members by the joints they are connected to.

```
UNIT MMS
KN
MEMBER PROPERTY AMERICAN
1 4 PRIS YD 300 ZD 275
2 5 PRIS YD 350 ZD 275
3 PRIS YD 350
```

Member properties have been defined above using the PRISMATIC attribute for which YD (depth) and ZD (width) values are provided in MM unit. When YD and ZD are provided together, STAAD considers the section to be rectangular. When YD alone is specified, the section is considered to be circular. Details are available in Technical Reference of STAAD Commands (on page 2196).

```
CONSTANTS
E 22 MEMB 1 TO 5
```

Material constant E (modulus of elasticity) is specified as 22KN/sq.mm following the command CONSTANTS.

```
UNIT METER KN
CONSTANTS
DENSITY 25.0 ALL
POISSON 0.17 ALL
```

Length unit is changed from MMS to METER to facilitate the input of Density. Next, the Poisson's Ratio is specified.

```
BETA 90 MEMB 4
```

In the absence of any explicit instructions, STAAD will orient the beams and columns of the structure in a pre-defined way (see General Engineering Theory (on page 2085) for details.) In order to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis, you must apply a beta angle of 90 degrees.

```
SUPPORT
1 4 5 FIXED
```

Joints 1, 4 and 5 are defined as fixed supported.

```
UNIT METER KG
LOAD 1 DEAD LOAD
```

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is initiated along with an accompanying title.

```
SELFWEIGHT Y -1
```

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```
MEMBER LOAD
2 5 UNI
GY -400
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

```
LOAD 2 LIVE LOAD
```

Load case 2 is initiated along with an accompanying title.

```
MEMBER LOAD
2 5 UNI
GY -600
```

Load 2 also contains member loads. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

```
LOAD 3 WIND LOAD
```

Load case 3 is initiated along with an accompanying title.

```
MEMBER LOAD
1 UNI
GX 300
4 UNI
GX 500
```

Load 3 also contains member loads. GX indicates that the load is in the global X direction. The word UNI stands for uniformly distributed load. Loads are applied on members 1 and 4.

```
LOAD 4  DEAD + LIVE
```

Load case 4 is initiated along with an accompanying title.

```
REPEAT LOAD
1 1.2 2 1.5
```

Load case 4 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases specified earlier. This repeat load case instructs the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. The load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4. Similarly, the load data values from load case 2 are multiplied by a factor of 1.5, and the resulting values too are utilized in load case 4.

```
LOAD 5 DEAD + WIND
```

Load case 5 is initiated along with an accompanying title.

```
REPEAT LOAD
1 1.1 3 1.3
```

This repeat load case instructs the program to analyze the structure for loads from cases 1 and 3 acting simultaneously.

```
PDELTA ANALYSIS
```

The PDELTA ANALYSIS command is an instruction to the program to execute a second-order analysis and account for P-delta effects.

```
LOAD LIST 4 5
```

The above LOAD LIST command is a means of stating that all further calculations should be based on the results of load cases 4 and 5 only. The intent here is to restrict concrete design calculations to that for load cases 4 and 5 only.

```
START CONCRETE DESIGN
CODE ACI
UNIT MMS
NEWTON
CLT 25 ALL
CLB 30 ALL
CLS 25 ALL
FC 25 ALL
FYMAIN 415 ALL
TRACK 1 ALL
```

The first line is the command that initiates the concrete design operation. The values for the concrete design parameters are defined in the above commands. Design is performed per the ACI Code. The length units are changed from METER to MMS to facilitate the input of the design parameters. Similarly, force units are changed from KG to NEWTON. The TRACK value dictates the extent of design related information which should be produced by the program in the output. The parameters specified include CLT (Clear cover for top surface), CLB (Clear cover for bottom surface), CLS (Clear cover for sides), FC(Strength of concrete), and FYMAIN (Ultimate strength of steel). These parameters are described in D1.F.3 Design Parameters (on page 1201).

```
DESIGN BEAM 2 5
DESIGN COLUMN 1 3 4
```

The above commands instruct the program to design beams 2 and 5 for flexure, shear and torsion, and to design columns 1, 3 and 4 for axial load and biaxial bending.

```
END CONCRETE DESIGN
```

This command terminates the concrete design operation.

```
FINISH
```

This command terminates the STAAD run.

Save the input file and close the editor. The model is opened in the STAAD.Pro interface.

T.2 Performing the analysis and design

STAAD.Pro performs Analysis and Design simultaneously.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group. .

   As the analysis progresses, several messages appear on the screen as shown in the figure below.

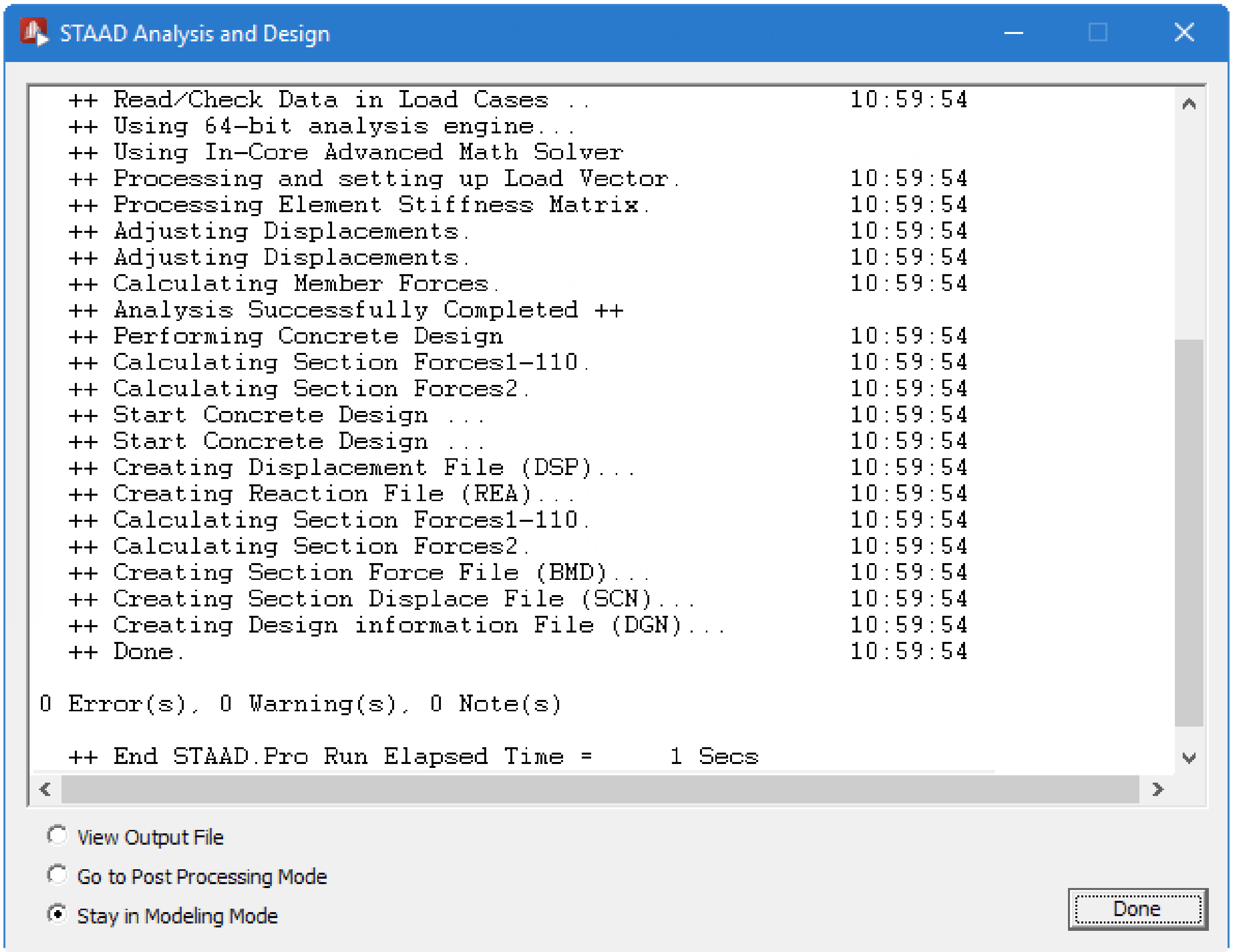


Figure 60:

2. Select the **View Output File** option once the analysis and design are complete.

   The three options are indicative of what will happen after you click **Done**.

View Output File	This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the model generation process. It also provides you with important messages of any errors were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 447) for details on viewing and understanding the contents of the output file.
Go to Post Processing Mode	This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 455) for details on the Post processing mode.
Stay in Modeling Mode	This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click **Done**.
   The STAAD.Pro Output Viewer window opens.

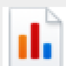
T.2 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

Note:

If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the **Utilities** ribbon tab, select the **Analysis Output** tool in the **View** group.



Tip: By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Set Echo** tool option from the menu bar and the select the **Echo Off** option in the **Set Echo** dialog.

It is *strongly recommended* that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:43:57                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE RC FRAMED STRUCTURE
INPUT FILE: Tutorial 2 - RC Frames Structure.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 16-FEB-02
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT METER KN
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 3.5 0; 3 6 3.5 0; 4 6 0 0; 5 6 0 6; 6 6 3.5 6
     9. MEMBER INCIDENCES
    10. 1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 3
    11. UNIT MMS KN
    12. MEMBER PROPERTY AMERICAN
    13. 1 4 PRIS YD 300 ZD 275
    14. 2 5 PRIS YD 350 ZD 275
```

```
    15. 3 PRIS YD 350
    16. CONSTANTS
    17. E 22 MEMB 1 TO 5
    18. UNIT METER KN
    19. CONSTANTS
    20. DENSITY 25.0 ALL
    21. POISSON 0.17 ALL
    22. BETA 90 MEMB 4
    23. SUPPORTS
    24. 1 4 5 FIXED
    25. UNIT METER KG
    26. LOAD 1 DEAD LOAD
    27. SELFWEIGHT Y -1
    28. MEMBER LOAD
    29. 2 5 UNI GY -400
    30. LOAD 2 LIVE LOAD
    31. MEMBER LOAD
    32. 2 5 UNI GY -600
    33. LOAD 3 WIND LOAD
    34. MEMBER LOAD
    35. 1 UNI GX 300
    36. 4 UNI GX 500
    37. LOAD 4 DEAD + LIVE
    38. REPEAT LOAD
      RC FRAMED STRUCTURE                                      -- PAGE NO.    2
    39. 1 1.2 2 1.5
    40. LOAD 5 DEAD + WIND
    41. REPEAT LOAD
    42. 1 1.1 3 1.3
    43. PDELTA ANALYSIS SMALLDELTA
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     5, TOTAL DEGREES OF FREEDOM =      18
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   ++ Adjusting Displacements.
   ++ Adjusting Displacements.
    44. LOAD LIST 4 5
    45. START CONCRETE DESIGN
 CONCRETE DESIGN
    46. CODE ACI
    47. UNIT MMS NEWTON
    48. CLT 25 ALL
    49. CLB 30 ALL
    50. CLS 25 ALL
    51. FC 25 ALL
    52. FYMAIN 415 ALL
    53. TRACK 1 ALL
    54. DESIGN BEAM 2 5
      RC FRAMED STRUCTURE                                      -- PAGE NO.    3
      RC FRAMED STRUCTURE                                      -- PAGE NO.    4
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************
Units: NEWTON, MMS    (Unless Noted Otherwise)
```

```
 Member :      2
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length: 6000.000 |
| Critical Ratio :   1.000        Criteria: Torsion                         |
| Critical Clause:   9.5.3/9.5.4                                            |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:  275.00 | Depth:  350.00 |
-------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |   Bars      |      Location      | Distance  |  Anchor  |
|       Position  | Nums | Size | Start  |    End    | From Face | Start End |
---------------------------------------------------------------------------
|         Bottom  |    2 | 13M  |    0.00|  6000.00 |    36.41  |  Yes  Yes |
|         Bottom  |    2 | 13M  |  335.44|  5609.93 |    36.41  |   No   No |
|            Top  |    4 | 13M  |    0.00|  6000.00 |    31.41  |  Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                                  |        Asv        |     Rebar Specification     |
| Zone | Dir.|  From    |    To    |  Reqd.  |  Prov.  | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
|    1 |  Y  |    0.00  | 6000.00  |   31.96 |   51.35 |   48 |  13M | 127.66  |   2  |
|    1 |  Z  |    0.00  | 6000.00  |   31.96 |   51.35 |   48 |  13M | 127.66  |   2  |
------------------------------------------------------------------------------------
------------------------ Member:     2 Design Ends ------------------------
      RC FRAMED STRUCTURE                                   -- PAGE NO.    5
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       **********************************************
Units: NEWTON, MMS    (Unless Noted Otherwise)
 Member :      5
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length: 6000.000 |
| Critical Ratio :   1.000        Criteria: Torsion                         |
| Critical Clause:   9.5.3/9.5.4                                            |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:  275.00 | Depth:  350.00 |
-------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |   Bars      |      Location      | Distance  |  Anchor  |
|       Position  | Nums | Size | Start  |    End    | From Face | Start End |
---------------------------------------------------------------------------
|         Bottom  |    2 | 13M  |    0.00|  6000.00 |    36.41  |  Yes  Yes |
|         Bottom  |    2 | 13M  |  335.44|  5609.93 |    36.41  |   No   No |
|            Top  |    4 | 13M  |    0.00|  6000.00 |    31.41  |  Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                                  |        Asv        |     Rebar Specification     |
| Zone | Dir.|  From    |    To    |  Reqd.  |  Prov.  | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
```

```
|     1 |  Y  |     0.01 | 1500.00 |    35.14 |    50.42 |    13 |   13M | 125.00   |    2  |
|     1 |  Z  |     0.01 | 1500.00 |    35.14 |    50.42 |    13 |   13M | 125.00   |    2  |
|     2 |  Y  | 1500.00 | 4500.00 |    31.96 |    52.44 |    25 |   13M | 125.00   |    2  |
|     2 |  Z  | 1500.00 | 4500.00 |    31.96 |    52.44 |    25 |   13M | 125.00   |    2  |
|     3 |  Y  | 4500.00 | 5999.99 |    35.14 |    50.42 |    13 |   13M | 125.00   |    2  |
|     3 |  Z  | 4500.00 | 5999.99 |    35.14 |    50.42 |    13 |   13M | 125.00   |    2  |
-----------------------------------------------------------------------------------
------------------------ Member:     5 Design Ends ------------------------
    55. DESIGN COLUMN 1 3 4
      RC FRAMED STRUCTURE                                      -- PAGE NO.    6
      RC FRAMED STRUCTURE                                      -- PAGE NO.    7
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       **********************************************
Units: NEWTON, MMS    (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length: 3500.000 |
| Critical Ratio :   0.634        Criteria: Flexure                          |
| Critical Clause:   10.5.2                                                   |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:  275.00 | Depth:  300.00 |
-------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |       Location      | Distance  |  Anchor  |
|        Position | Nums | Size | Start   |    End    | From Face | Start End |
---------------------------------------------------------------------------
|             Top |    3 | 13M  |     0.00|  3500.00 |    31.41   |   Yes  Yes |
|          Bottom |    3 | 13M  |     0.00|  3500.00 |    36.41   |   Yes  Yes |
|            Left |    1 | 13M  |     0.00|  3500.00 |    31.41   |   Yes  Yes |
|           Right |    1 | 13M  |     0.00|  3500.00 |    31.41   |   Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
-----------------------------------------------------------------------------------
|                                 |        Asv        |     Rebar Specification     |
| Zone | Dir.|  From    |    To    |  Reqd.  |  Prov.  | Nums | Size | Spacing | Legs |
-----------------------------------------------------------------------------------
|     1 |  Y  |     0.00 | 3500.00 |    31.96 |    33.71 |    19 |   13M | 194.44   |    2  |
|     1 |  Z  |     0.00 | 3500.00 |    31.96 |    33.71 |    19 |   13M | 194.44   |    2  |
-----------------------------------------------------------------------------------
------------------------ Member:     1 Design Ends ------------------------
      RC FRAMED STRUCTURE                                      -- PAGE NO.    8
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       **********************************************
Units: NEWTON, MMS    (Unless Noted Otherwise)
 Member :      3
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length: 3500.000 |
| Critical Ratio :   0.812        Criteria: Flexure                          |
| Critical Clause:   10.5.2                                                   |
-----------------------------------------------------------------------------
CROSS SECTION
------------------------------------
| Shape: Circular    | Dia:  350.00 |
```

```
-------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |       Location       | Distance  |  Anchor  |
|        Position | Nums | Size | Start  |    End      | From Face | Start End |
---------------------------------------------------------------------------
|   Evenly Dist. |     8 | 13M  |     0.00|  3500.00 |    31.41   |  Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                                 |         Asv         |     Rebar Specification      |
| Zone | Dir.|  From     |    To   |  Reqd.  |   Prov.  | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
|    1 |  Y  |    0.00 | 3500.00 |   31.96 |    33.71 |   19 |  13M | 194.44  |   2  |
|    1 |  Z  |    0.00 | 3500.00 |   31.96 |    33.71 |   19 |  13M | 194.44  |   2  |
------------------------------------------------------------------------------------
------------------------ Member:     3 Design Ends ------------------------
      RC FRAMED STRUCTURE                                      -- PAGE NO.    9
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************
Units: NEWTON, MMS    (Unless Noted Otherwise)
 Member :      4
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length: 3500.000 |
| Critical Ratio :   1.000        Criteria: Torsion                          |
| Critical Clause:   10.5.3/10.5.4                                           |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:  275.00 | Depth:  300.00 |
-------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |       Location       | Distance  |  Anchor  |
|        Position | Nums | Size | Start  |    End      | From Face | Start End |
---------------------------------------------------------------------------
|             Top |     3 | 13M  |     0.00|  3500.00 |    31.41   |  Yes  Yes |
|          Bottom |     3 | 13M  |     0.00|  3500.00 |    36.41   |  Yes  Yes |
|            Left |     1 | 13M  |     0.00|  3500.00 |    31.41   |  Yes  Yes |
|           Right |     1 | 13M  |     0.00|  3500.00 |    31.41   |  Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                                 |         Asv         |     Rebar Specification      |
| Zone | Dir.|  From     |    To   |  Reqd.  |   Prov.  | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
|    1 |  Y  |    0.00 | 3500.00 |   31.96 |    55.79 |   31 |  13M | 116.67  |   2  |
|    1 |  Z  |    0.00 | 3500.00 |   31.96 |    55.79 |   31 |  13M | 116.67  |   2  |
------------------------------------------------------------------------------------
------------------------ Member:     4 Design Ends ------------------------
    56. END CONCRETE DESIGN
    57. FINISH
      RC FRAMED STRUCTURE                                      -- PAGE NO.   10
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44: 0 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
```

```
         *   http://www.bentley.com/en/support/                    *
         *                                                         *
         *   Details about additional assistance from              *
         *   Bentley and Partners can be found at program menu     *
         *   Help->Technical Support                               *
         *                                                         *
         *           Copyright (c) Bentley Systems, Inc.           *
         *                http://www.bentley.com                   *
         ***********************************************************
```

T.2 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the **Post Processing** Mode. The **Post Processing** mode is used to verify the analysis and design results and generate reports.

T.2 Opening the postprocessing workflow

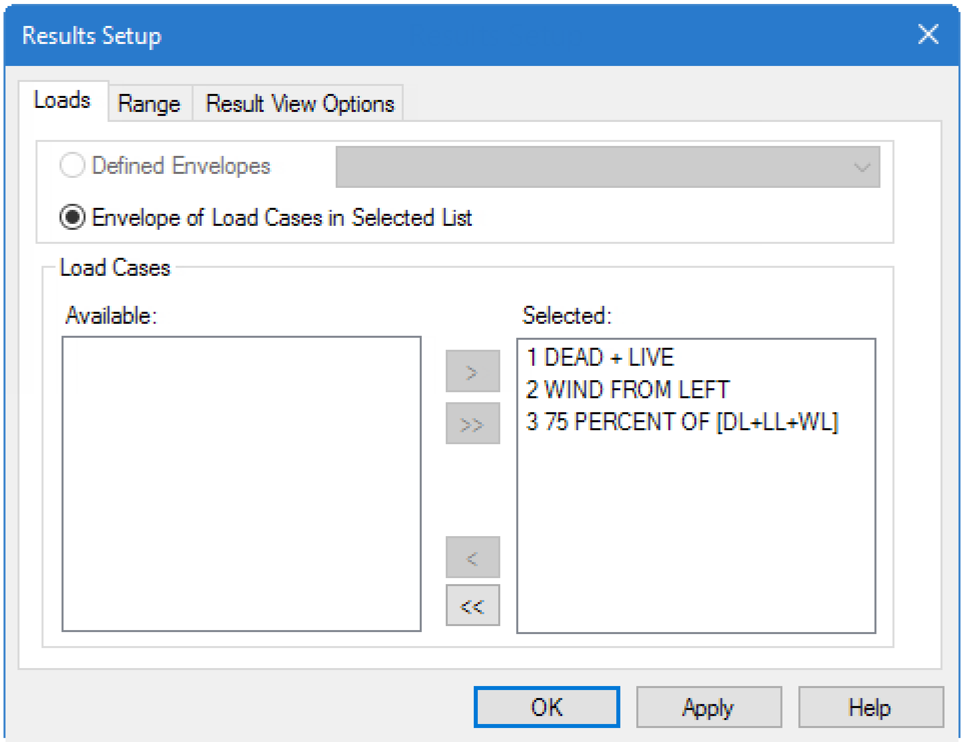
You can open the Postprocessing workflow anytime there are current analysis results for your model.

If you selected to open the postprocessing mode after T.2 Performing the analysis and design (on page 513), then you can skip to step 2.

1. On the **Workflows** panel, select **Postprocessing**.

   The **Results Setup** dialog opens.



2. (Optional) Select the load cases for which to display the results.

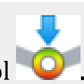
   Tip: All load cases are selected by default.

   For this tutorial, we will use all load cases.

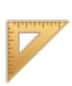
T.2 Viewing the deflection diagram

1. Select the **Displacement** page in **Postprocessing** page control bar.

   Tip: This page is the default page when you open the Postprocessing workflow.

   Note: You can display the Deflection diagram on most results pages by selecting the **Deflection** tool . The tool remains depressed when the displacement is displayed.

   The displacement diagram is drawn on the structure. The **Node Displacements table** and **Beam Relative Displacement Detail** table open. See T.2 The Node Displacements Table (on page 526) for details on these tables.
2. (Optional) If the deflection is not exaggerated enough to clearly identify, you can change the scale:
   a. On the Results ribbon tab, select the **Scale** tool in the **Configuration** group.

   The **Diagrams** dialog opens displaying the **Scales** tab.

> **Tip:** You can also open the **Diagrams** dialog by selecting the **Structure** tool or by right-clicking in the View area and then selecting **Structure Diagrams** from the pop-up menu.

b. Change the **Displacement** value.

> **Tip:** Smaller numbers exaggerate the deflected shape.

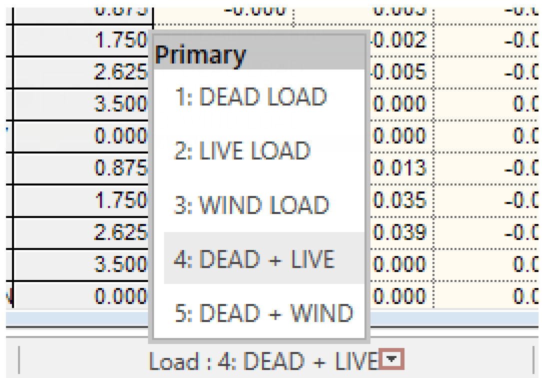
Type a value of 5 (mm per m) to clearly see the deflection due to load case 5.

c. Click **Apply**.

> **Tip:** You can set the **Apply Immediately** option on this tab to view the scale change as you use the up/ down arrows in the Displacement field.

d. Click **OK**.

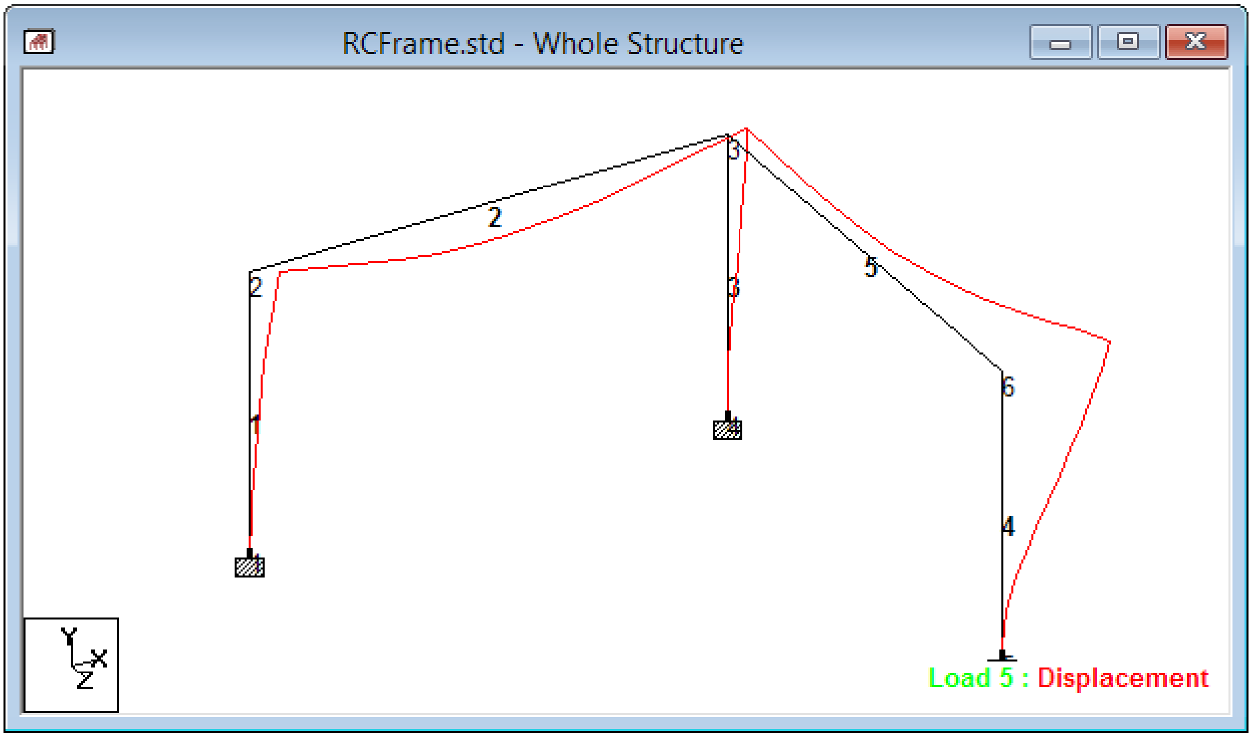
3. To change the load case being displayed, select from the current **Load** list in the window status bar.



> **Tip:** Alternatively, select the **Structure** tool  in the **Configuration** group on the **Results** toolbar. The active load case can be selected on the **Load and Results** tab in the **Diagrams** dialog.

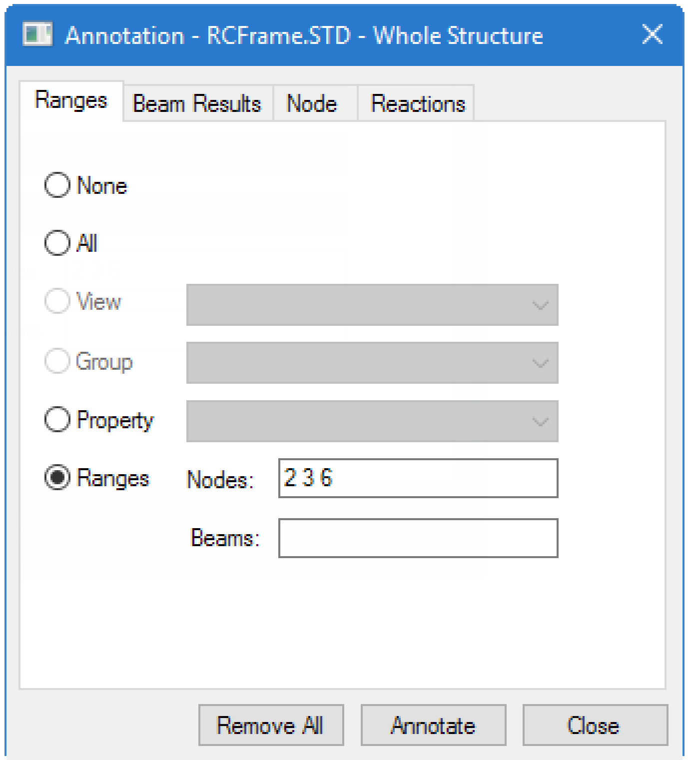
The results displayed are for the selected load case only.

Select **5: DEAD + WIND** for this tutorial.



4. Annotate the deflection at specific nodes:
   a. On the Results ribbon tab, select the **Annotate** tool in the **Configuration** group.

   The **Annotation** dialog opens.
   b. On the **Ranges** tab, select the **Ranges** option and then type 2 3 6 (spaced node number list) in the **Nodes** field.



c. Select the **Node** tab and then check the **Resultant** option.

Resultant stands for the square root of sum of squares of values of X, Y and Z displacements.

d. Click **Annotate** and then click **Close**.

The structure deflection diagram is annotated for load case 5, as in the following figure.

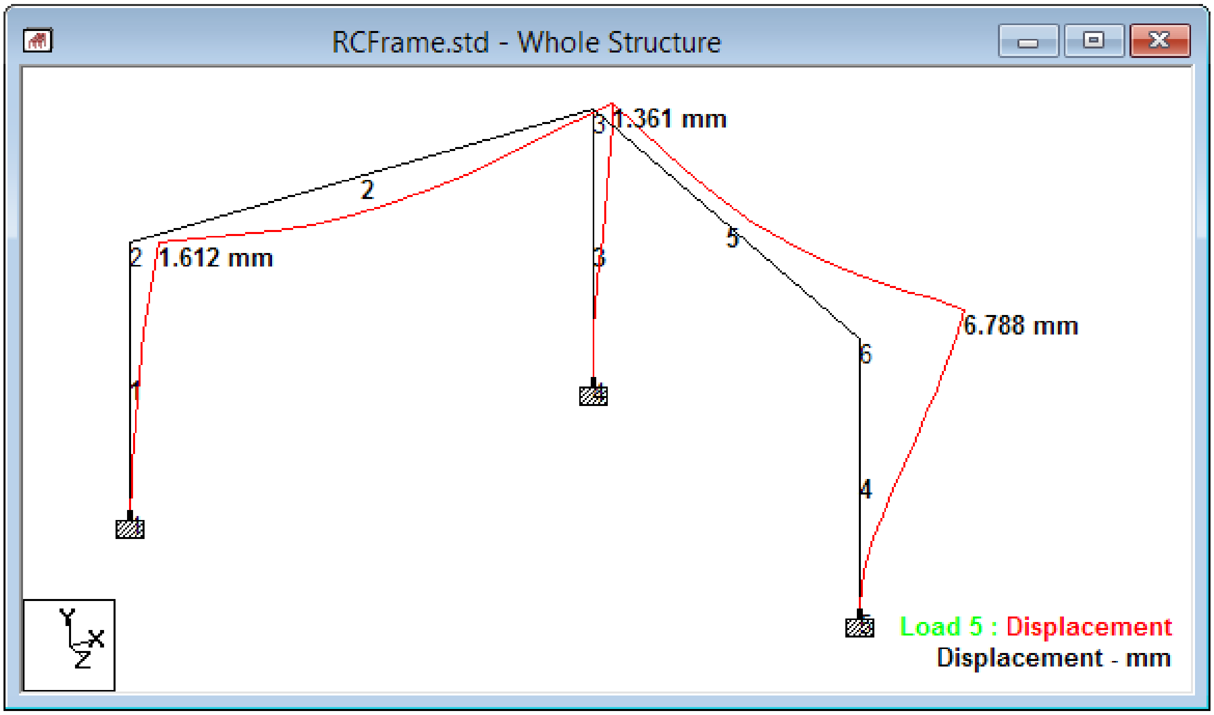


Figure 61:

5. Change the display units for displacement:

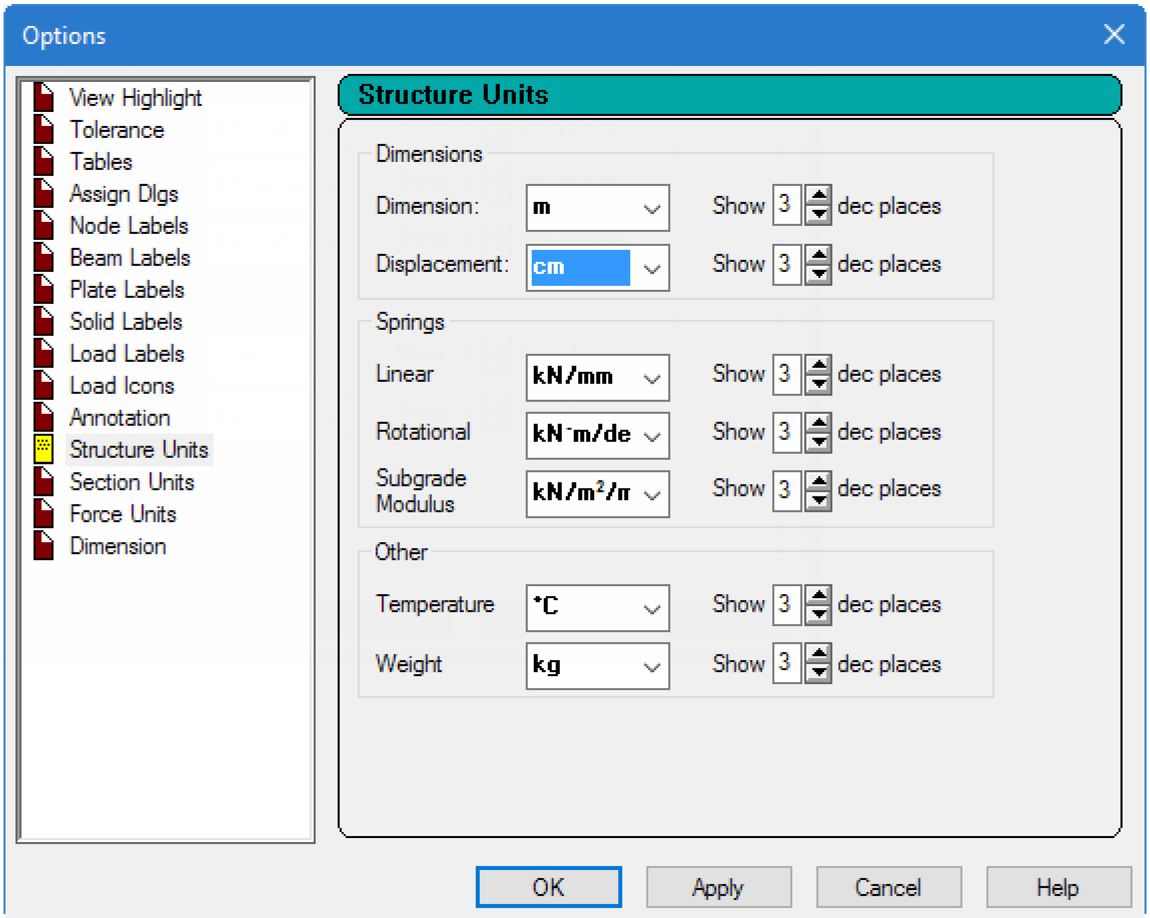
   The units in which displacement values are displayed in the post-processing mode are referred to as the display units.

   a. On the **File** ribbon tab, select **Display Options** on the **Settings** tab.

      The **Options** dialog opens.

   b. Select the **Structure Units** tab.

   c. Change **Displacement** selection from **mm** (millimeter) to **cm** (centimeter).



d. Click **OK**.

The diagram will be updated to reflect the new units.

T.2 The Node Displacements Table

When the **Displacements** page is selected, two tables open on the right side of the program window.

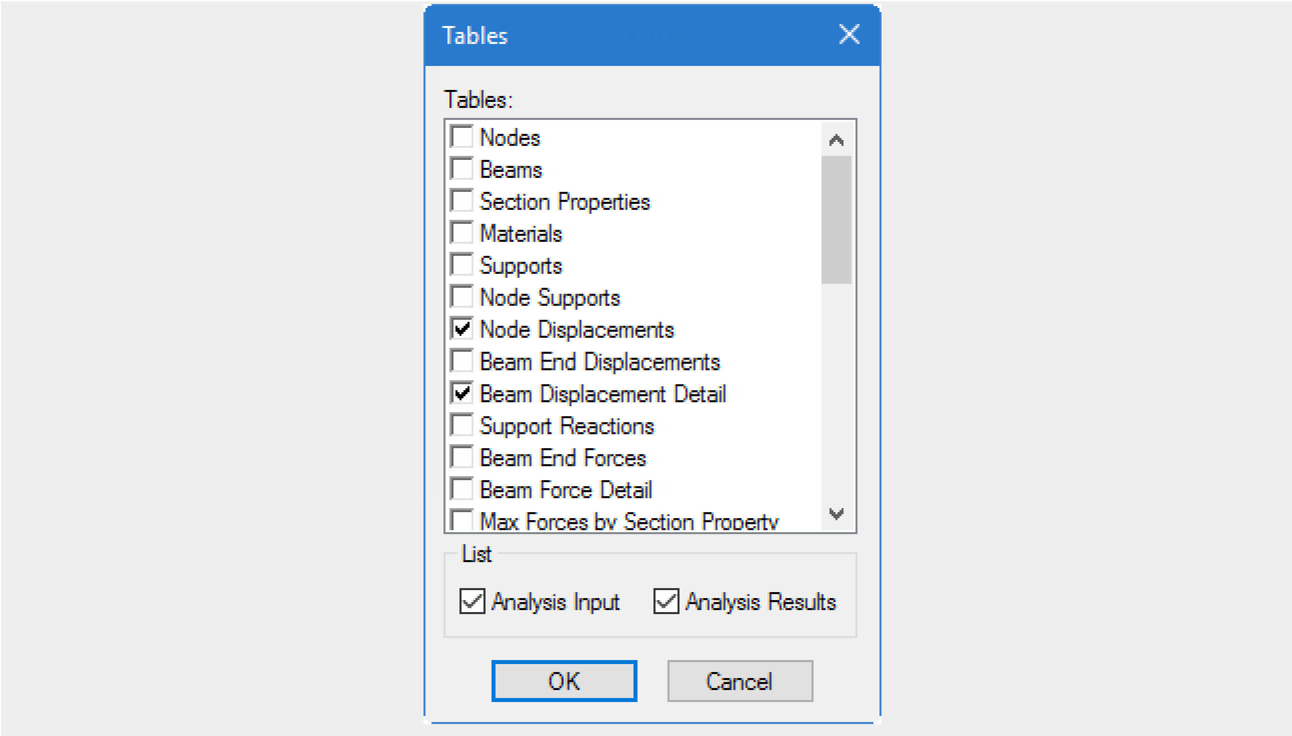
The **Node Displacements** table lists the displacement values for every node for every selected load case.

The **Beam Relative Displacement Detail** table displays the displacements along beams at intermediate points.

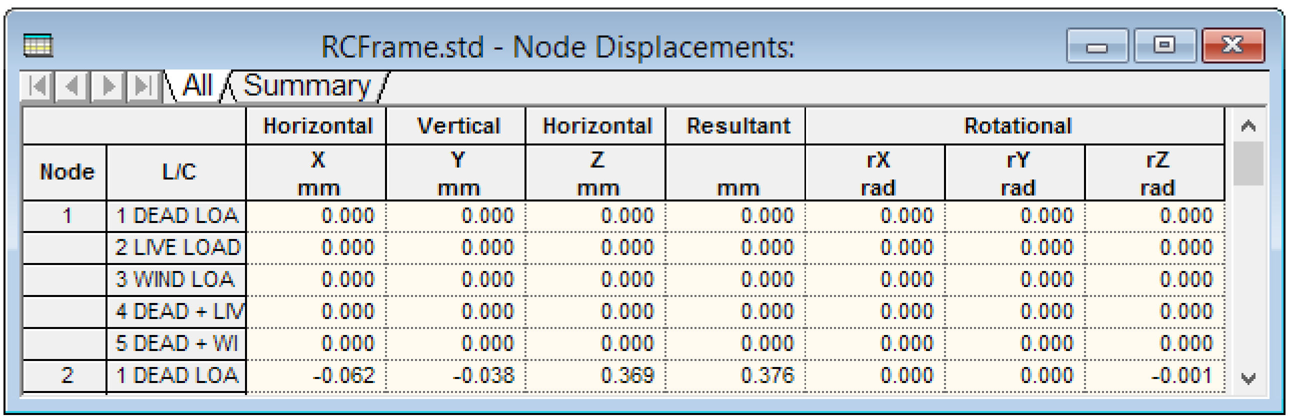
Note: The load cases included in results tables can be restricted using the **Results Setup** dialog. Refer to T.2 Restricting the load cases for results (on page 536) for details.

Tip:

You can reopen any closed tables from the **Tables** dialog, which is opened by selecting the **Tables** tool in the **Windows** group on the **View** ribbon tab.



The **Node Displacements** table



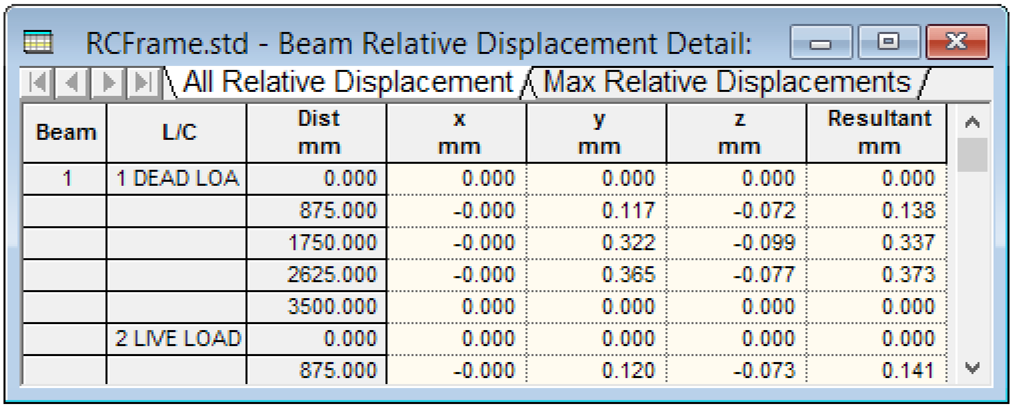
		Horizontal	Vertical	Horizontal	Resultant	Rotational		
Node	L/C	X mm	Y mm	Z mm	mm	rX rad	rY rad	rZ rad
1	1 DEAD LOA	0.000	0.000	0.000	0.000	0.000	0.000	0.000
	2 LIVE LOAD	0.000	0.000	0.000	0.000	0.000	0.000	0.000
	3 WIND LOA	0.000	0.000	0.000	0.000	0.000	0.000	0.000
	4 DEAD + LIV	0.000	0.000	0.000	0.000	0.000	0.000	0.000
	5 DEAD + WI	0.000	0.000	0.000	0.000	0.000	0.000	0.000
2	1 DEAD LOA	-0.062	-0.038	0.369	0.376	0.000	0.000	-0.001

Figure 62:

All This tab presents all nodal displacements in tabular form for all load cases and all degrees of freedom.

Summary This tab, shown in the figure below, presents the maximum and minimum nodal displacements (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

The Beam Relative Displacement Detail table



Beam	L/C	Dist mm	x mm	y mm	z mm	Resultant mm
1	1 DEAD LOA	0.000	0.000	0.000	0.000	0.000
		875.000	-0.000	0.117	-0.072	0.138
		1750.000	-0.000	0.322	-0.099	0.337
		2625.000	-0.000	0.365	-0.077	0.373
		3500.000	0.000	0.000	0.000	0.000
	2 LIVE LOAD	0.000	0.000	0.000	0.000	0.000
		875.000	-0.000	0.120	-0.073	0.141

Figure 63:

All	The **All** tab presents the displacements of members at intermediate section points. All specified members and all specified load cases are included. The table shows displacements along the local axes of the members, as well as their resultants.
Max Displacements	The **Max Displacements** tab presents the summary of maximum sectional displacements (see figure below). This table includes the maximum displacement values and location of its occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the member to the resultant maximum section displacement of the member.

T.2 Viewing the force and moment diagrams

Tip: You can display force or moment diagrams on most any page in the Post-Processing, but is recommended to use the Beam | Forces page to do so.

1. Select the **Beam Results** page on the **Postprocessing** page control bar.

   Note: You can display force and moment diagrams on most results pages by selecting corresponding tool from the **View Result** group on the **Results** ribbon tab. The corresponding tool remains depressed when the diagram is displayed.

   The bending moment MZ is drawn on the structure by default. The **Beam End Forces** table and **Beam Force Detail** table open. T.2 The Beam Forces Table (on page 533) for details on these tables.
2. To change the force or moment (i.e., degree of freedom) diagram displayed, either:

   select the corresponding diagram tool from the **View Result** group on the **Results** ribbon tab

   or

   select **Structure** tool in the **Configuration** group on the **Results** ribbon tab and then select the degrees of freedom to display from the **Beam Forces** group on the **Loads and Results** tab.

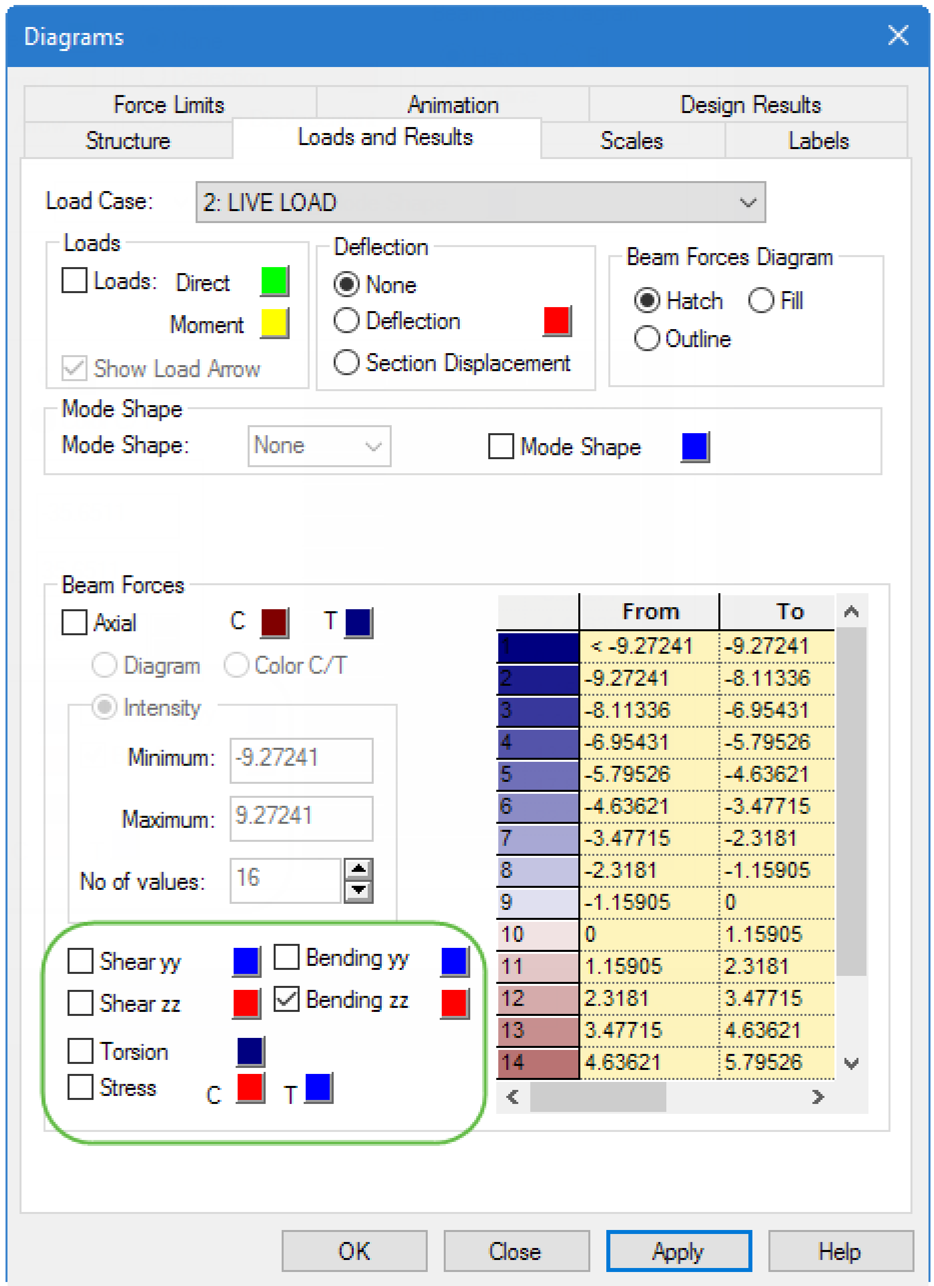
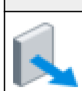
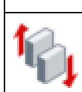


Table 23: Results diagram tools

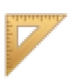
Tool	What it does
	Displays the axial force diagram
	Displays the shear Y diagram

Tool	What it does
	Displays the shear Z diagram
	Displays the torsion diagram
	Displays the bending Y diagram
	Displays the bending Z diagram
	Displays the beam stress (tension and compression) diagrams.

Note: The **Load and Results** tab on the **Diagrams** dialog can also be used to select the color for each diagram as well as the display method (i.e., hatched, filled, or outline).

Tip: You can display multiple diagrams simultaneously.

3. (Optional) If the force or moment diagram is not exaggerated enough to clearly identify, you can change the scale:
   a. On the Results ribbon tab, select the **Scale** tool in the **Configuration** group.

      The **Diagrams** dialog opens displaying the **Scales** tab.

      Tip: You can also open the **Diagrams** dialog by selecting the **Structure** tool or by right-clicking in the View area and then selecting **Structure Diagrams** from the pop-up menu.

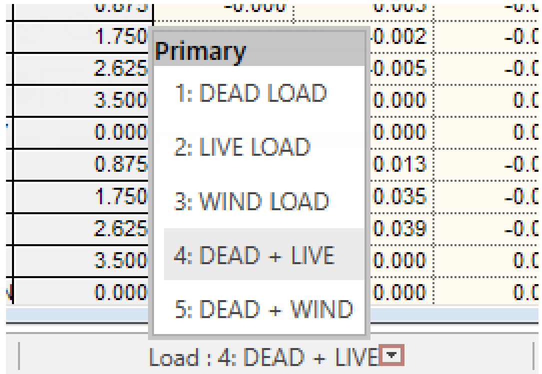
   b. Change the value in the corresponding **Results Scales** group to the diagram currently displayed.

      Tip: Smaller numbers exaggerate the diagram.

      Type a value of `50` (kN/m per m) to clearly see the **Bending Z** due to load case 5.
   c. Click **Apply**.

      Tip: You can set the **Apply Immediately** option on this tab to view the scale change as you use the up/down arrows in any of the **Result Scales** fields.

   d. Click **OK**.
4. To change the load case being displayed, select from the current **Load** list in the window status bar.



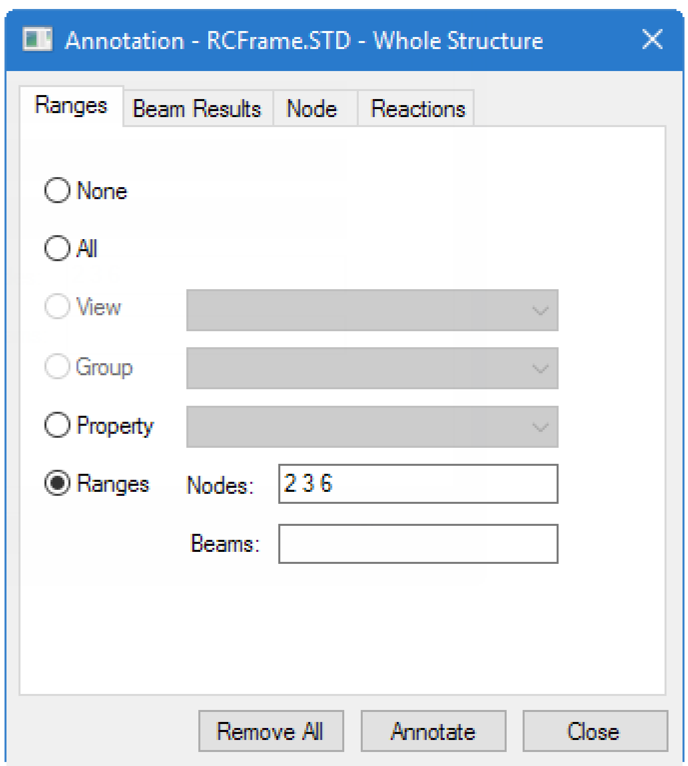
Tip: Alternatively, select the **Structure** tool in the **Configuration** group on the **Results** toolbar. The active load case can be selected on the **Load and Results** tab in the **Diagrams** dialog.

The results displayed are for the selected load case only.

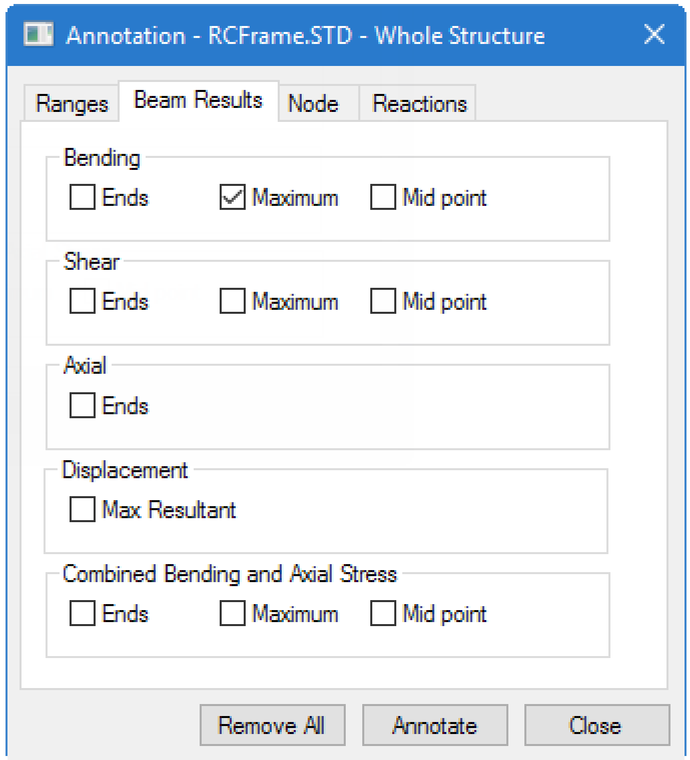
5. Annotate the bending Z moment diagram to display the maximum moment in the beams for load case 5:

a. Select **Results** > **View Value**.
The **Annotation** dialog opens.

b. On the **Ranges** tab, select the **Ranges** option and then type 2 5 (spaced node number list) in the **Beams** field.



c. Select the **Beam Results** tab and then check the **Maximum** option in the **Bending** group.



d. Click **Annotate** and then click **Close**.

The bending Z moment diagram is annotated for load case 5, as in the following figure.

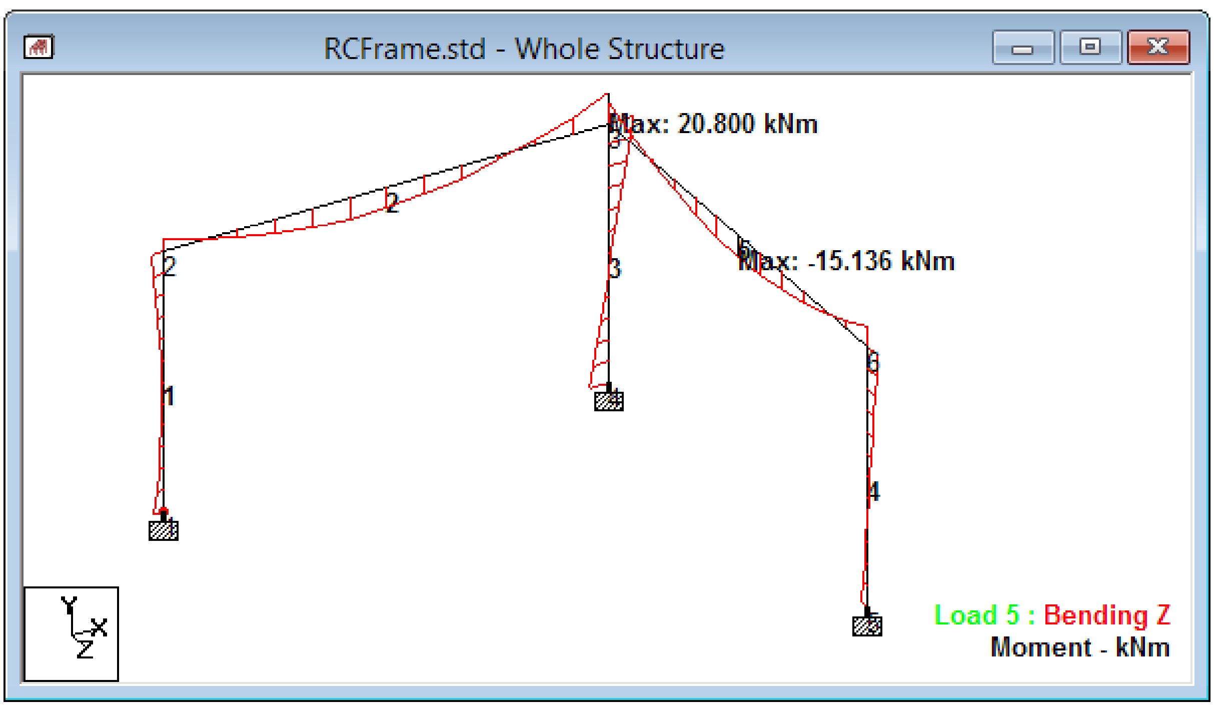


Figure 64:

6. Change the display units for bending moments:

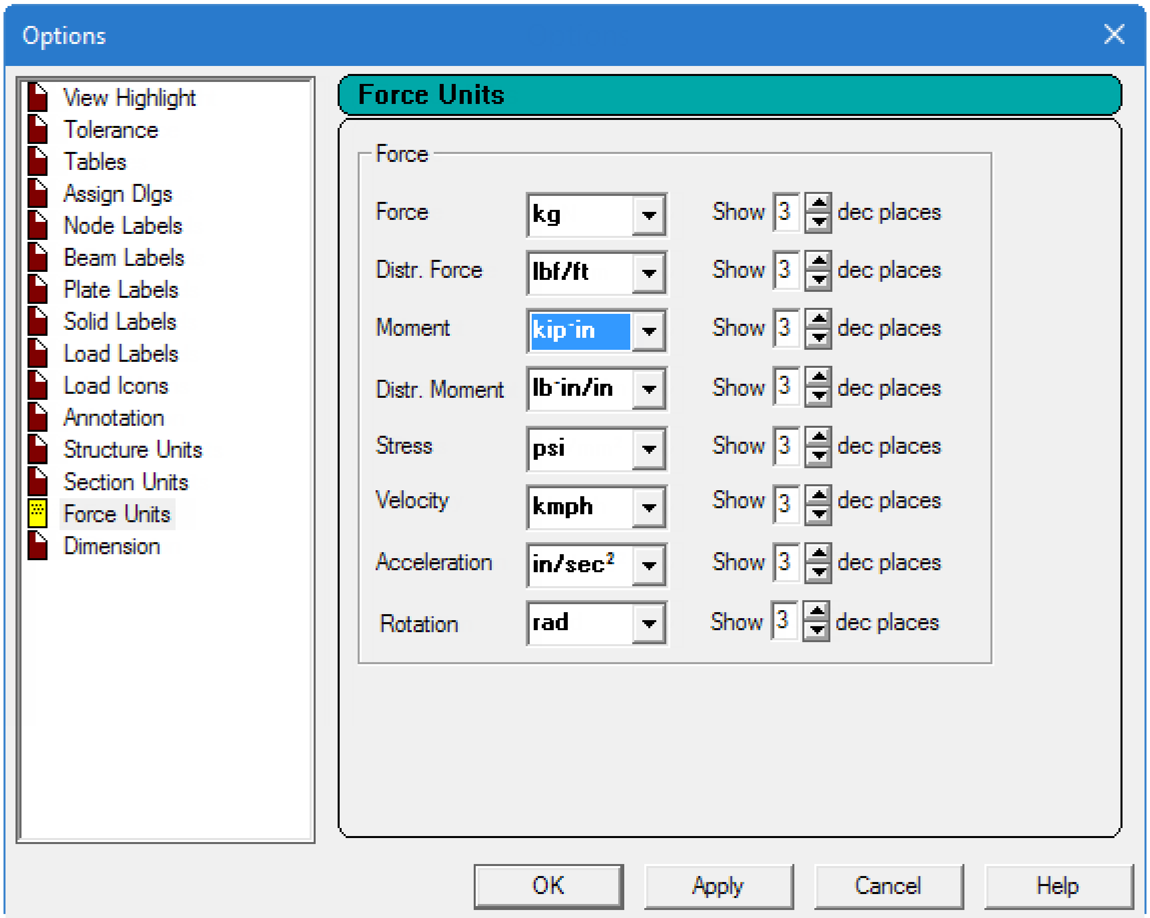
The units in which results values are displayed in the post-processing mode are referred to as the display units.

a. On the **File** ribbon tab, select **Display Options** on the **Settings** tab.

The **Options** dialog opens.

b. Select the **Force Units** tab.

c. Change **Moment** selection from **kNm** (kilonewton meters) to **kip-ft** (kilopound feet).



d. Click **OK**.

The diagram updates to reflect the new units.

T.2 The Beam Forces Table

When the **Beam Results** page is selected in the **Postprocessing** workflow, two tables open on the right side of the program window.

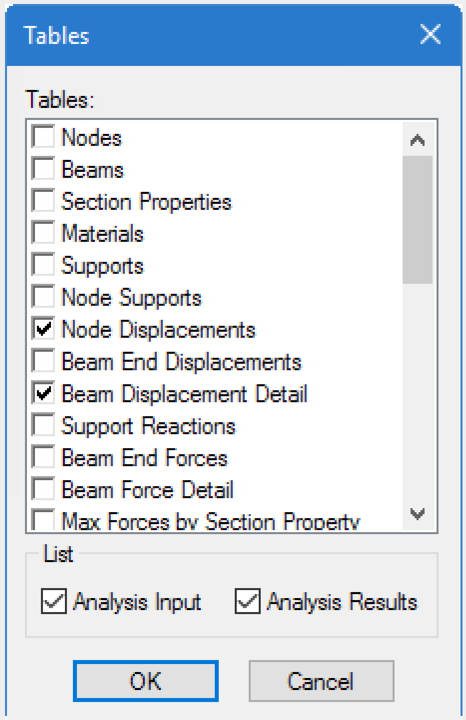
The **Beam End Forces** table lists the axial forces and shear forces, bending and torsional moments in all selected beams for all selected load cases are displayed in a tabular form along the right half of the screen.

The **Beam Force Detail** table lists force and moment values for every node for every selected load case.

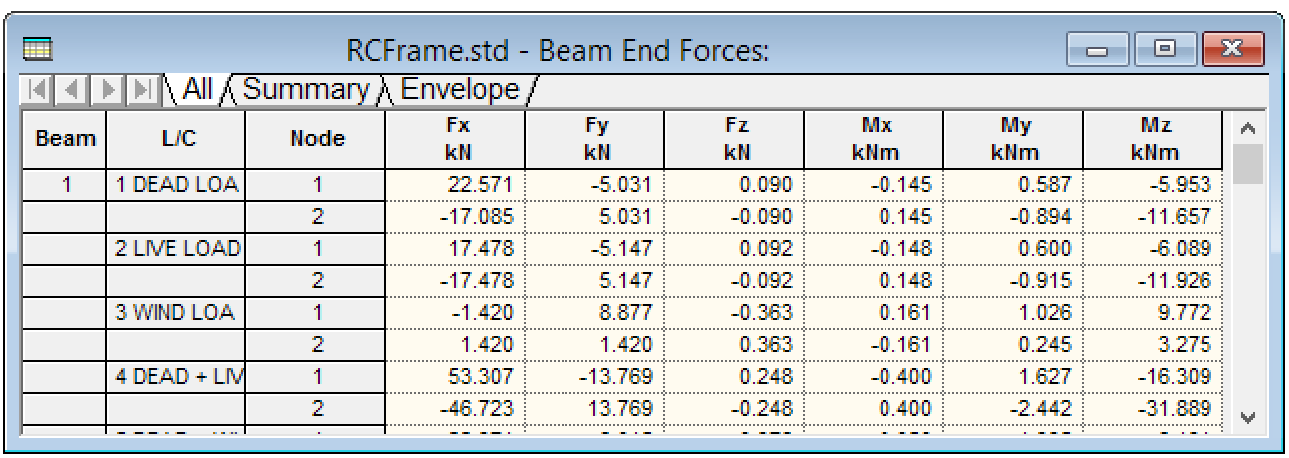
Note: The load cases included in results tables can be restricted using the **Results Setup** dialog. Refer to T.2 Restricting the load cases for results (on page 536) for details.

Tip:

You can reopen any closed tables from the **Tables** dialog, which is opened by selecting the **Tables** tool in the **Windows** group on the **View** ribbon tab.



The **Beam End Forces** table



RCFrame.std - Beam End Forces:

All | Summary | Envelope

Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx kNm	My kNm	Mz kNm
1	1 DEAD LOA	1	22.571	-5.031	0.090	-0.145	0.587	-5.953
		2	-17.085	5.031	-0.090	0.145	-0.894	-11.657
	2 LIVE LOAD	1	17.478	-5.147	0.092	-0.148	0.600	-6.089
		2	-17.478	5.147	-0.092	0.148	-0.915	-11.926
	3 WIND LOA	1	-1.420	8.877	-0.363	0.161	1.026	9.772
		2	1.420	1.420	0.363	-0.161	0.245	3.275
	4 DEAD + LIV	1	53.307	-13.769	0.248	-0.400	1.627	-16.309
		2	-46.723	13.769	-0.248	0.400	-2.442	-31.889

Figure 65:

All This tab presents all forces and moments corresponding to all 6 degrees of freedom at the start and end of each selected member for all selected load cases.

Summary This tab, shown in the next figure, presents the maximum and minimum values (forces and moments) for each degree of freedom. All beams and all Load Cases specified during the Results

Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

Envelope This tab shows a table consisting of the maximum and minimum for each degree of freedom for each member, and the load case responsible for each of those values.

T.2 Viewing the force and moment graphs

The **Graphs** results in the Postprocessing workflow are used to view moment and force graphs such as Axial, Bending zz, Shear yy and Combined Stresses for individual members.

1. Display the beam results graphs:
   a. On the **Results** ribbon tab, select the **Tables** tool in the **View Results** group.
   b. In the **Beam Results** group, select **Graphs**.

   The View window shows the loading on the structure. On the right side of the screen, the force/moment diagrams appear.
2. Select a member in the View window.

   The graphs are plotted for that member in the data area.

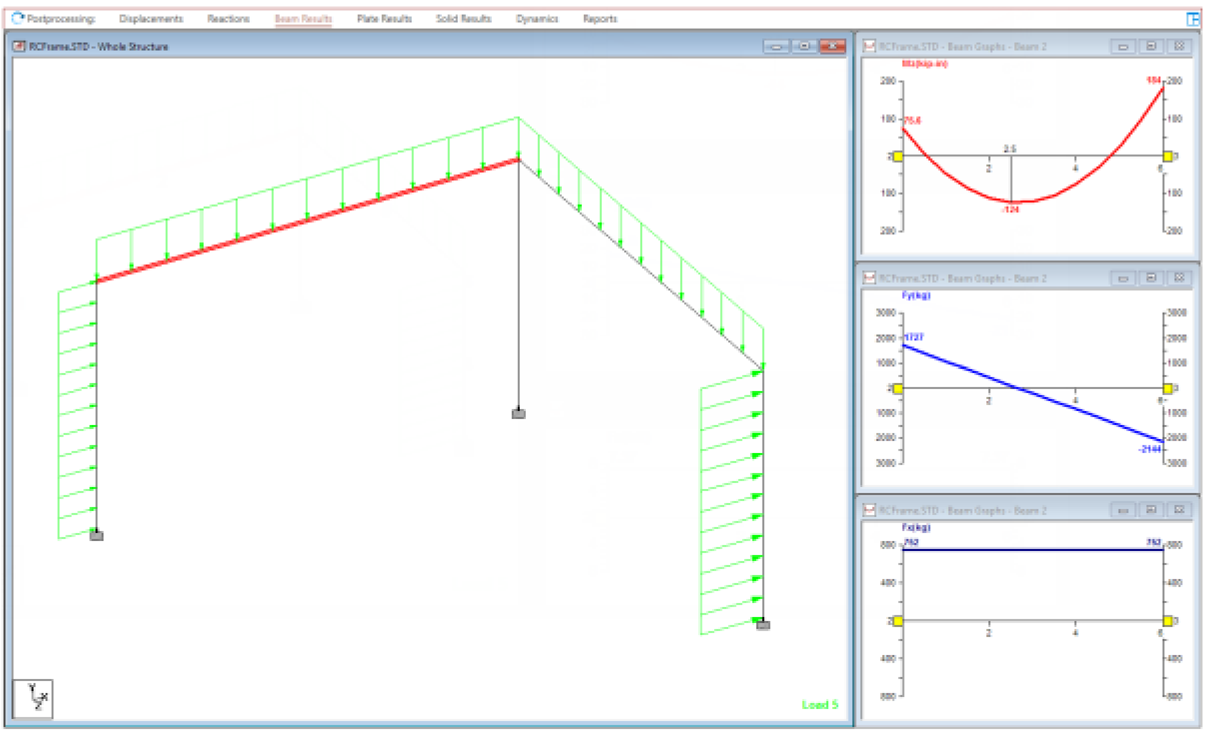


Figure 66: Bending, shear, and axial graphs for Beam 2 under load case 5: DEAD + WIND.

3. To change the diagrams displayed for the current beam and load case selection:
   a. Right-click on any graph window and select **Diagrams** from the pop-up menu.
      The **Diagram** dialog opens.
   b. Set the check box for the degrees of freedom you wish to view in the diagram.

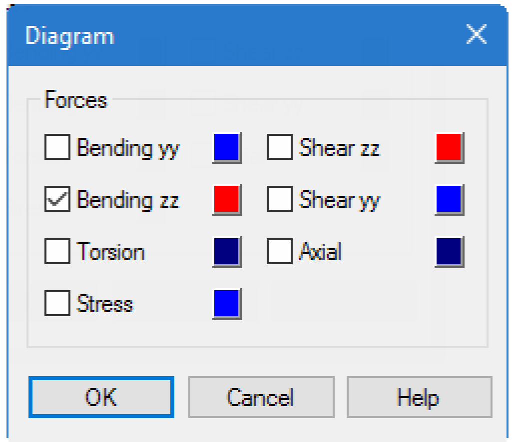


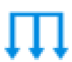
Figure 67:

c. Click **OK** .
The selected degree of freedom is plotted in that window.

T.2 Restricting the load cases for results

To restrict the load cases for which results are viewed, use the following procedure.

1. On the **Results** ribbon tab, select the **Select Load Case** tool in the **Configuration** group.



   The **Results Setup** dialog opens.
2. Click **[<<]**.

   All load cases are moved from the Selected list to the Available list.

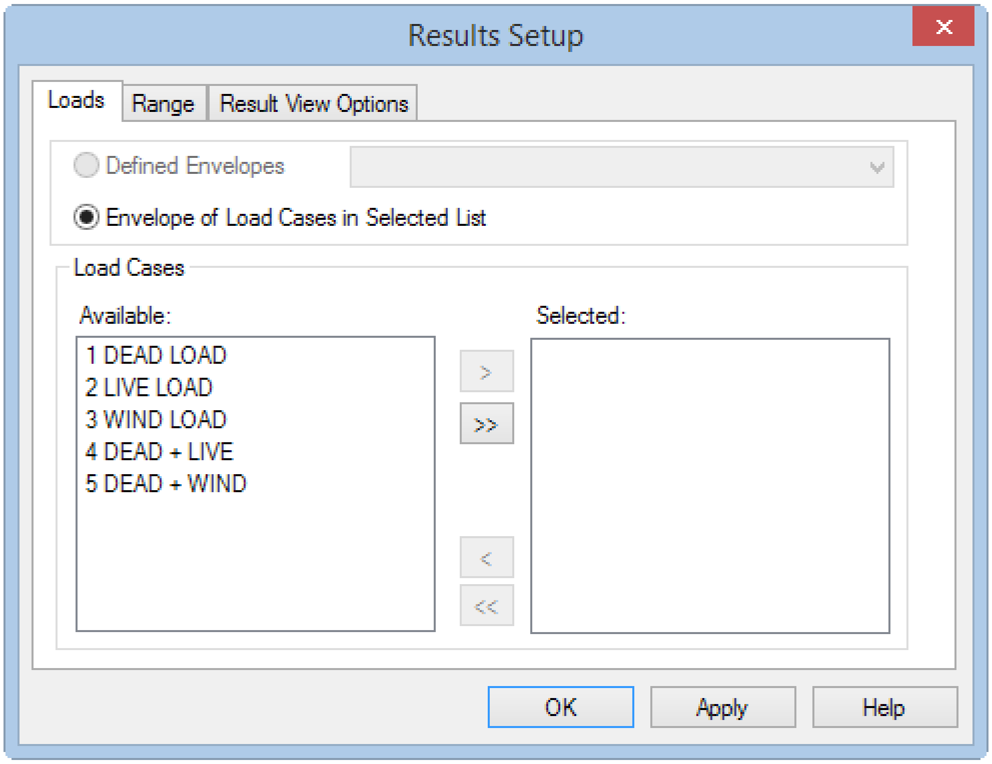


Figure 68:

3. Select two load cases to use for viewing results:
   - **a.** Press and hold the **<Ctrl>** key and then select load cases **1 DEAD LOAD** and **3 WIND LOAD**.
   - **b.** Click **[>]**.
   - **c.** Click **OK**.

T.2 Using Member Query

Member query is a facility where several results for specific members can be viewed at the same time from a single dialog. It is also a place from where many of the member attributes such as the property definition, specifications (releases, truss, cable, etc.) and beta angle can be changed for input purposes.

Note: The **Beam Cursor** tool  is active when the **Beam Results** page is selected.

1. Double-click a member in the View window.

   Tip: Alternately, if you have selected a member you can then select **Tools** > **Query** > **Member**.

   For this example, double-click on member 4.

   The **Beam** dialog opens.

   In the Modeling mode, you can actively edit members using this dialog. In the Post-Processing mode, the member data is presented for review only.

Note: The tabs in the member query dialog reflect the properties of the member along with any analysis and design results available. Changing the model will void any results and thus remove these tabs until another analysis is performed.

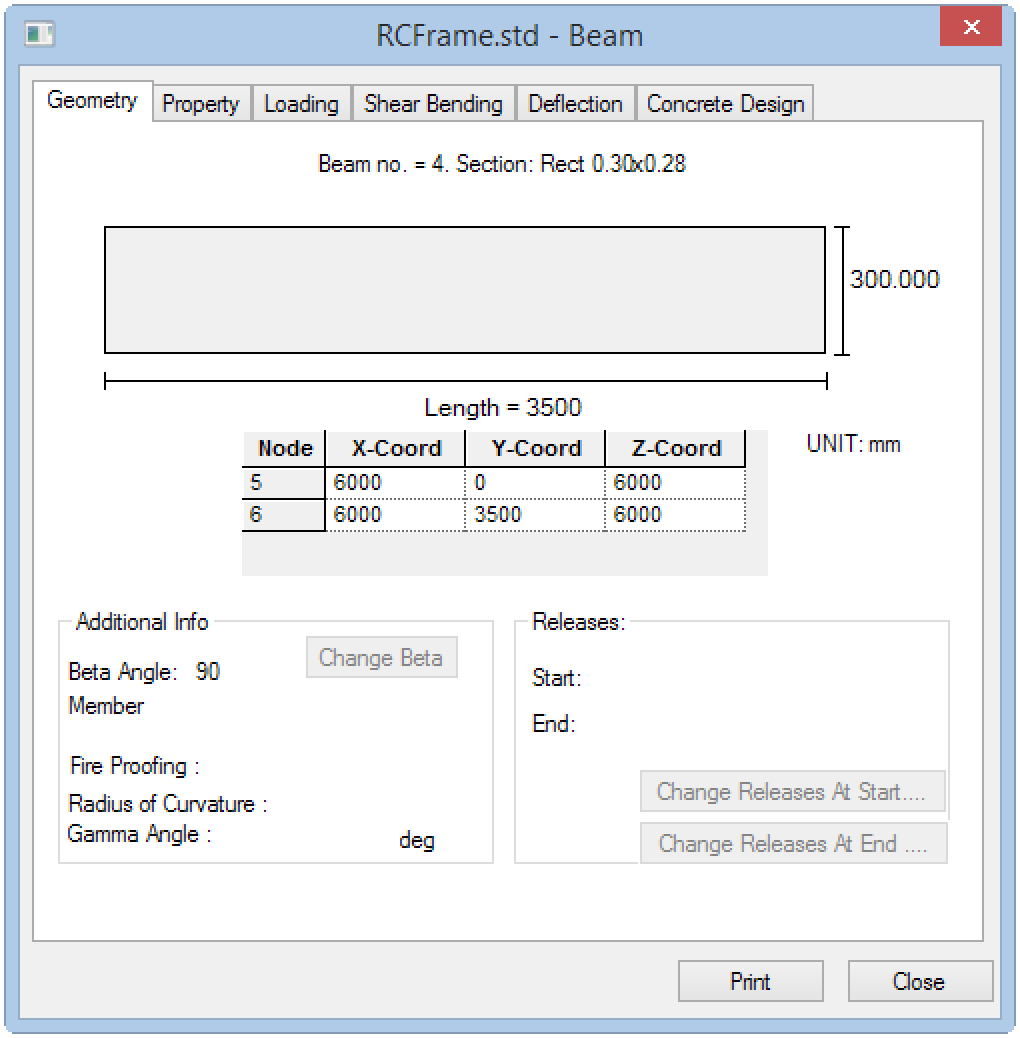
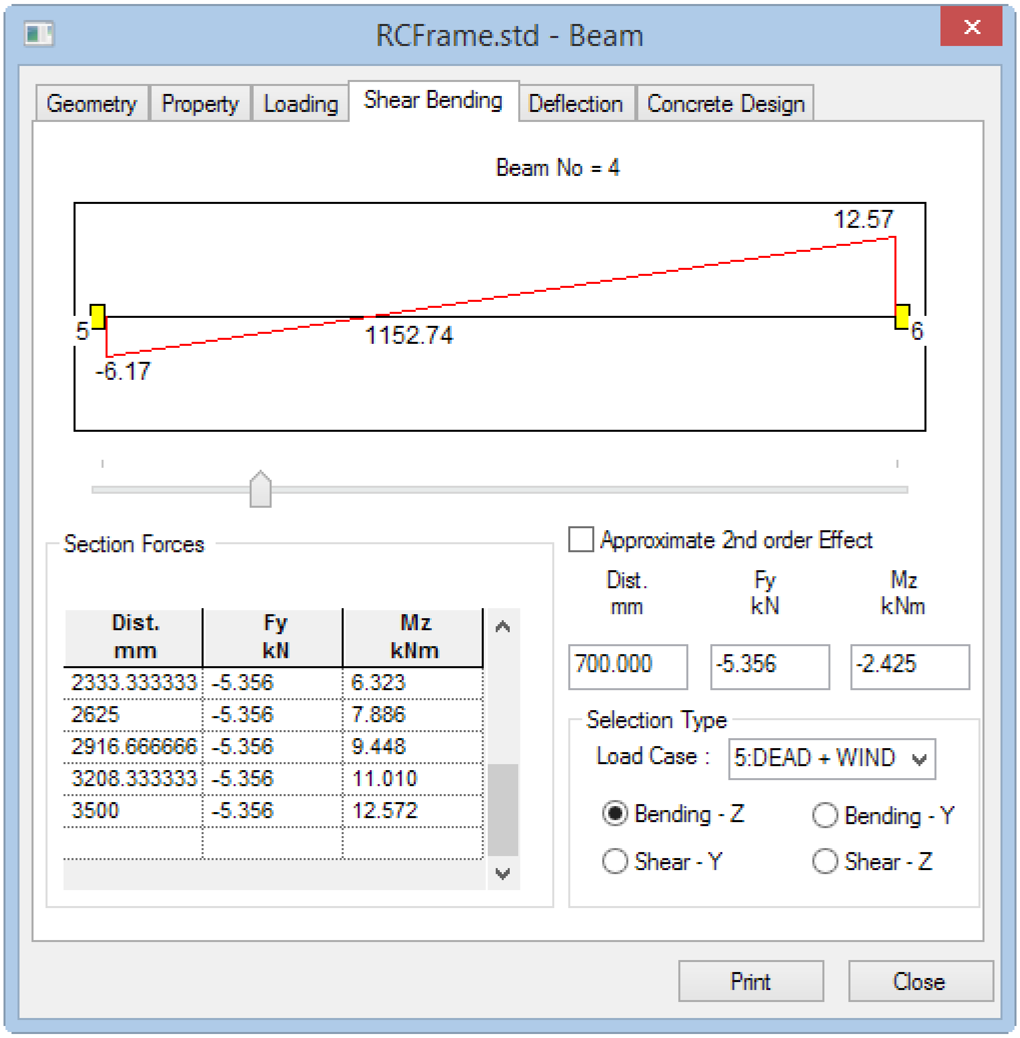


Figure 69:

2. Select the **Shear Bending** tab.

   Here you can review shear and bending diagrams for the member for a selected load case. A table presents the shear and bending a specific locations along the member length. The slider control can be used to review the values at distance.



3. Select the **Deflection** tab.

Here, a similar diagram of the member deflection is shown for the selected load case. A table displays the deflection values at specific points along the member length.

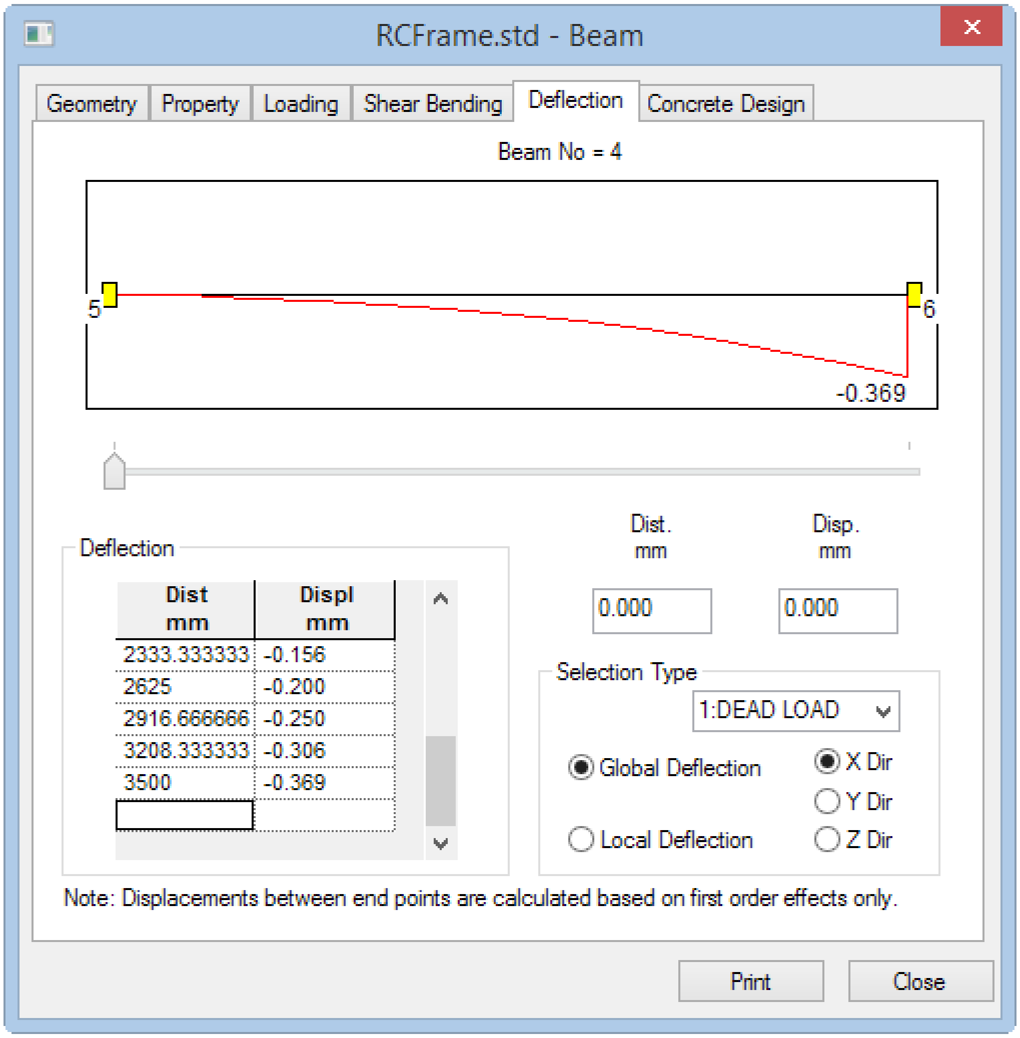
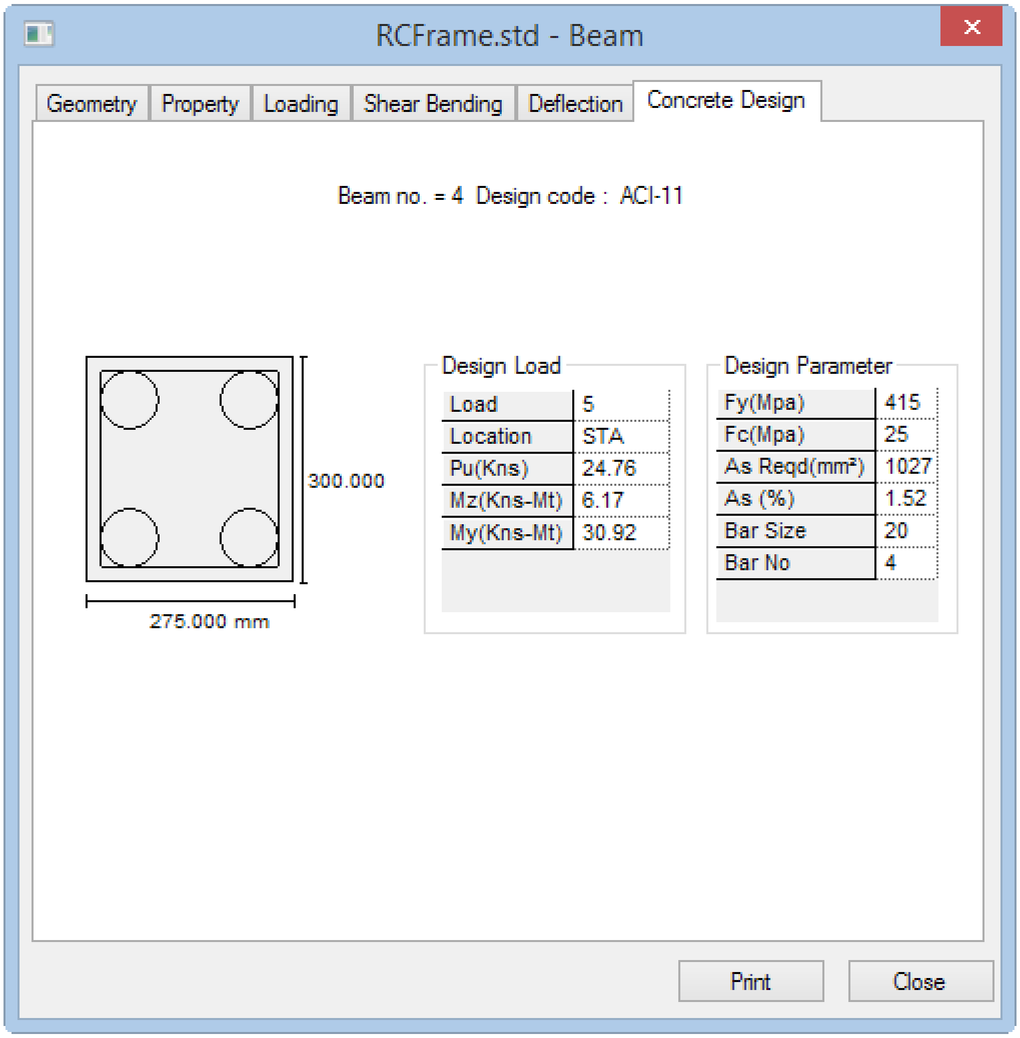


Figure 70:

4. Select the **Concrete Design** tab.

Note: This tab is only available for concrete members which have been designed using the concrete design command and after a successful analysis has been performed.



5. (Optional) Click **Print**

   The query results for the current dialog tab are printed using the report output settings. Refer to T.2 Creating Customized Reports (on page 544) for details on the report settings.
6. Click **Close**.

To review the member query results of another beam, the **Beam** dialog must be closed. Repeat this procedure for another member to query it.

T.2 Producing an on-screen report

Occasionally, we will come across a need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form. The facility which enables us to obtain such customized on-screen results is the Report menu on top of the screen.

Here, you will create a report that includes a table with the member major axis moment (MZ) values sorted in the order High to Low, for members 1 and 4 for all the load cases.

Note: The **Beam Cursor** tool  is active when the **Beam Results** page is selected.

1. Press and hold **<Ctrl>** and click both members 1 and 4 in the View window to select them.
2. On the **Results** ribbon tab, select the **Reports** > **Beam End Forces** tool in the **Reports** group.

   The **Beam End Force** dialog opens.

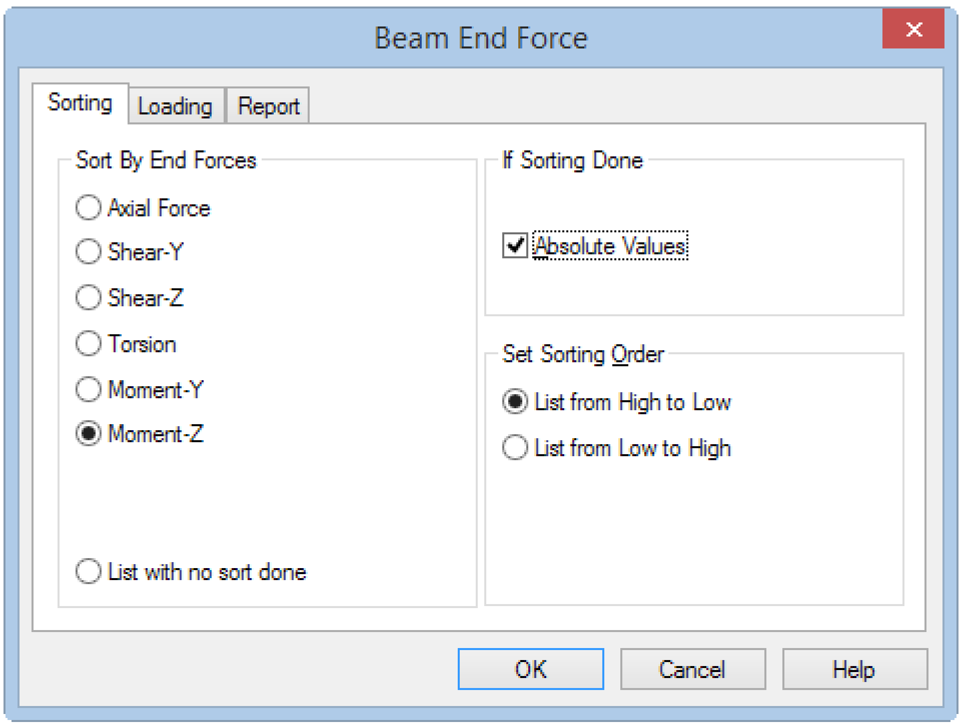
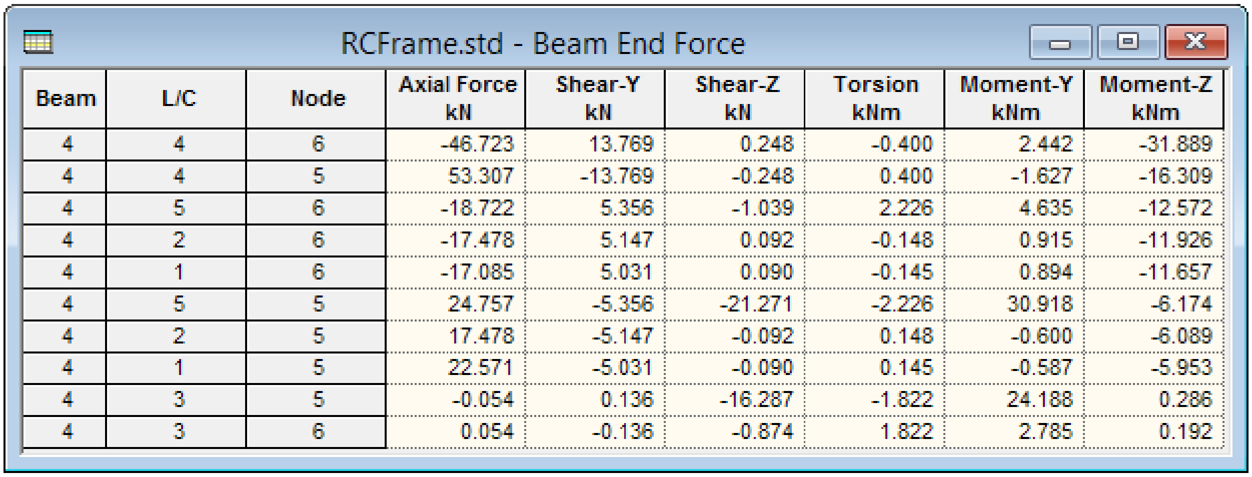


Figure 71:

3. Specify the report contents:
   a. Select the **Sorting** tab.
   b. Select the **Moment-Z** option in the **Sort By End Force** group.
   c. Select **List from High to Low** option in the **Set Sorting Order** group.
   d. Check the **Absolute Values** option in the **If Sorting done** group.
4. Select the **Loading** tab and ensure all the five load cases are added to the **Selected** list.
5. (Optional) You can save this report for future use,
   a. Select the **Report** tab
   b. Type a **Title** for the report.
   c. Set the **Save Report** check box.
6. Click **OK**.

   The member end forces sorted table opens with the MZ values sorted from High to Low based on Absolute numbers.



RCFrame.std - Beam End Force

Beam	L/C	Node	Axial Force kN	Shear-Y kN	Shear-Z kN	Torsion kNm	Moment-Y kNm	Moment-Z kNm
4	4	6	-46.723	13.769	0.248	-0.400	2.442	-31.889
4	4	5	53.307	-13.769	-0.248	0.400	-1.627	-16.309
4	5	6	-18.722	5.356	-1.039	2.226	4.635	-12.572
4	2	6	-17.478	5.147	0.092	-0.148	0.915	-11.926
4	1	6	-17.085	5.031	0.090	-0.145	0.894	-11.657
4	5	5	24.757	-5.356	-21.271	-2.226	30.918	-6.174
4	2	5	17.478	-5.147	-0.092	0.148	-0.600	-6.089
4	1	5	22.571	-5.031	-0.090	0.145	-0.587	-5.953
4	3	5	-0.054	0.136	-16.287	-1.822	24.188	0.286
4	3	6	0.054	-0.136	-0.874	1.822	2.785	0.192

Figure 72:

7. (Optional) To print this table, right-click anywhere within the table and select **Print**

T.2 Taking Pictures

To take a screen capture of the View window contents for use generating external reports, use the following procedure.

1. On the **Utilities** ribbon tab, select the **Take Picture** tool in the **Utilities** group.

   The **Picture #** dialog opens.

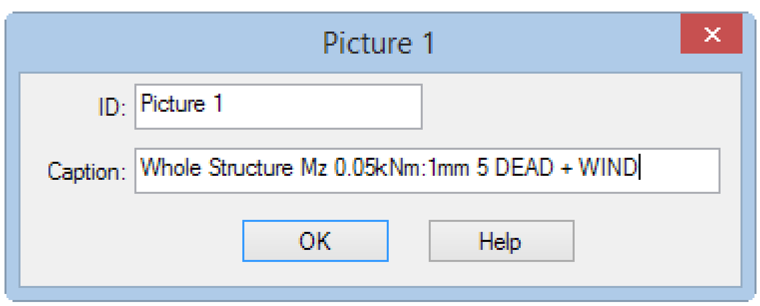


Figure 73: The Picture # dialog with the ID and Capture automatically generated

2. (Optional) Type a Picture **ID**

   The ID is incremented automatically.

3. (Optional) Type a **Caption**

> **Note:** For most View window contents, the caption will be automatically completed with a description of the contents.

4. Click **OK**.
   This picture is saved for use in reports.

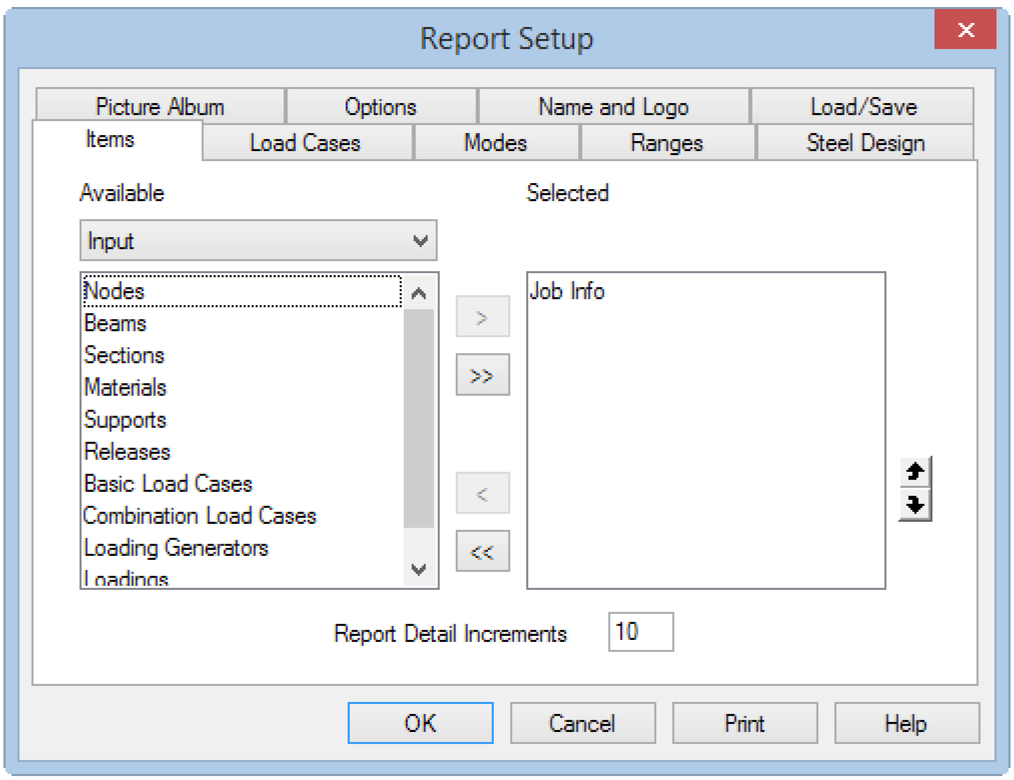
T.2 Creating Customized Reports

STAAD.Pro offers extensive report generation facilities.

Items which can be incorporated into such reports include input information, numerical results, steel design results, etc. You can choose from among a select set of load cases, mode shapes, structural elements, etc. You can also include any picture of the screen taken using the **Take Picture** tool. Other customizable parameters include the font size, title block, headers, footers, etc.

1. Open the Report Setup:
   - **a.** Select the **File** ribbon tab.
     The Backstage view opens.
   - **b.** Select the **Report** tab and then **Setup**.

   The **Report Setup** dialog opens.



Note: The Available items list is filtered by the drop-down selection list above it. This allows you to sort your reports.

2. Add model Output to the report:

   Tip: Job Information from the Input is selected by default.

   - **a.** Select **Output** from the drop-down list.
     The Available list displays output items.
   - **b.** Double-click **Node Displacement Summary** in the Available list.
     This item is added to the Selected list.

   c. Double-click **Beam Max Moments** in the Available list.
      This item is added to the Selected list.
   d. Select **Pictures** from the drop-down list.
      The Available list displays the picture captured earlier in the tutorial.
   e. Double-click **Picture 1**.
      This item is added to the Selected list.

   Leave the **Report Detail Increments** at 10. This is the number of segments into which a member is divided for sectional reports (i.e., displacements, forces, etc.).

3. Select the load cases to include in the report as well as how to order the results:
   a. Select the **Load Cases** tab.
   b. If all the load cases are not already included in the output, click **[>>]**.
   c. Select the **by Load Case** option for the **Grouping for Load Tables**.
   d. Select the **by Node/Beam** option for **Grouping for Result Tables**.

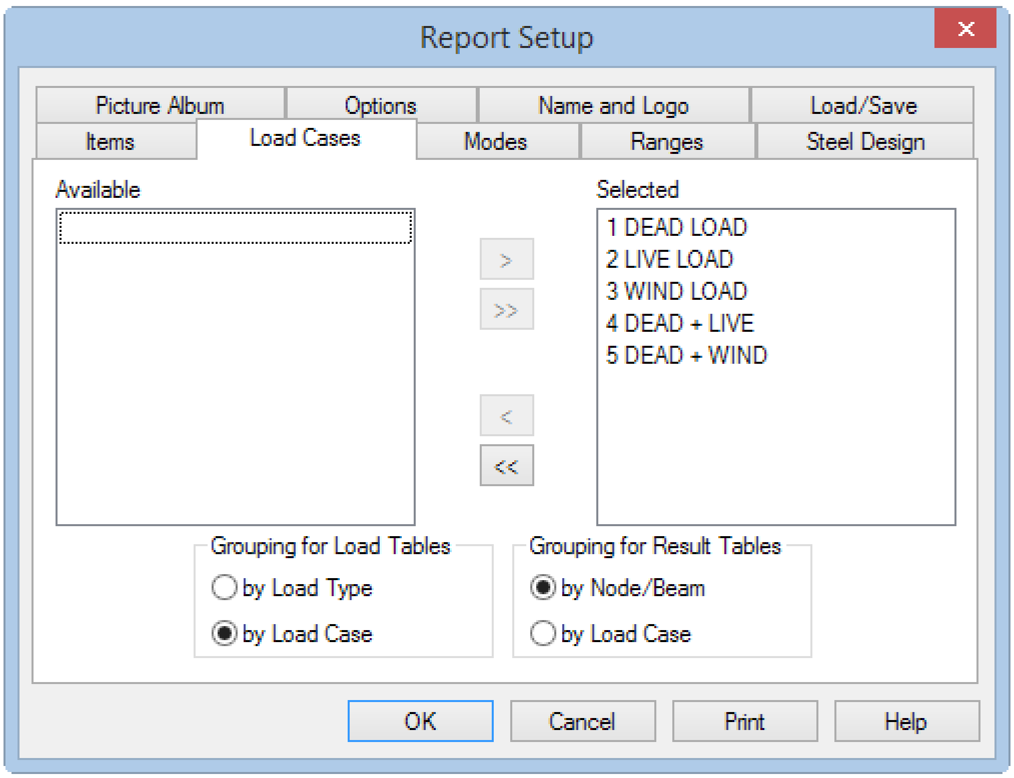


Figure 74:

4. (Optional) Select the **Picture Album**

Tip: You can manage pictures taken using the **Take Picture** tool here.

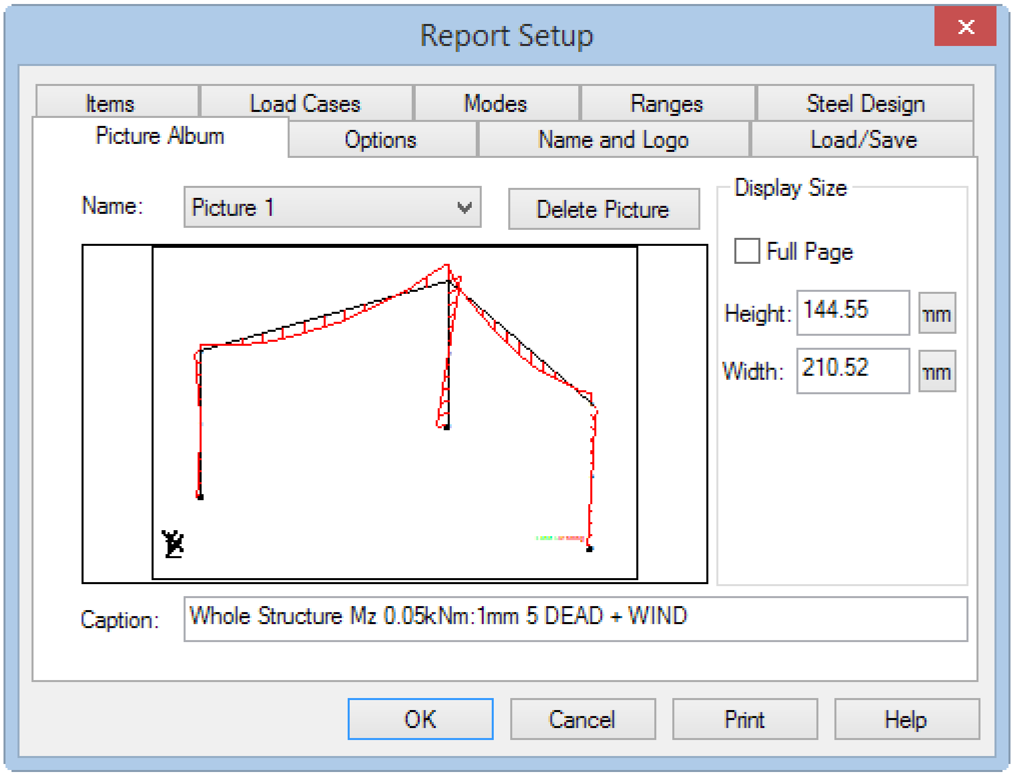
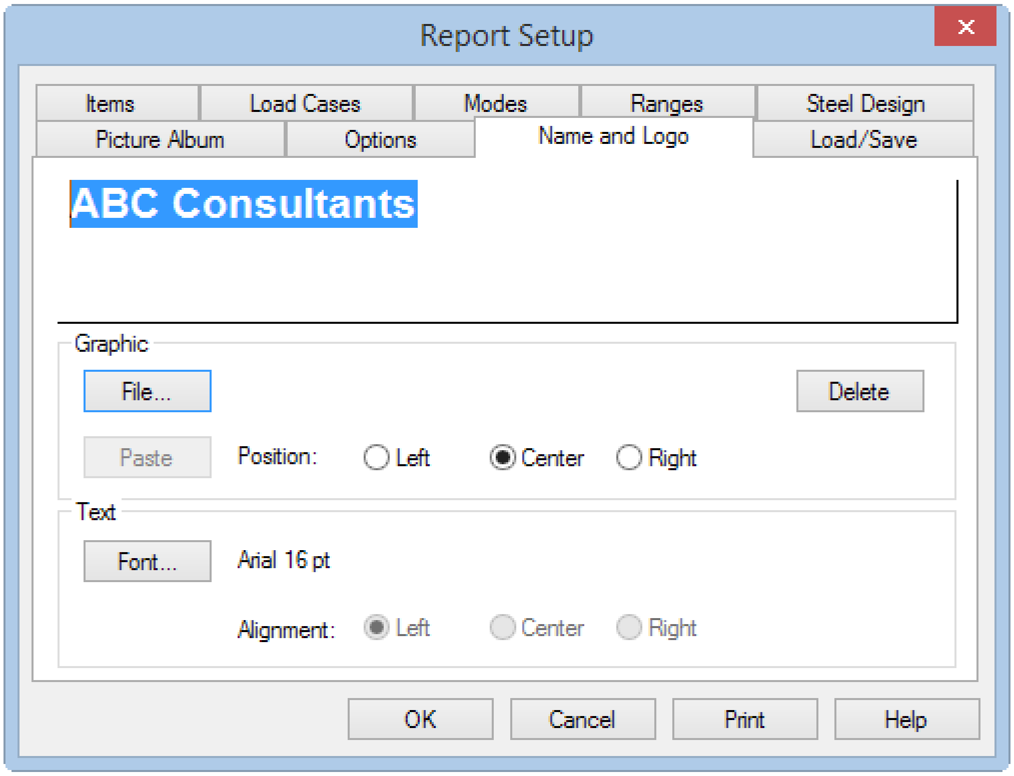


Figure 75:

5. Customize the title block of your report with your organization's information:
   a. Select the **Name and Logo** tab.
   b. Click in the empty text field and type the name and address of your organization.
   c. (Optional) Click **File Graphic**

      By default, the Bentley Systems, Inc. logo is used for STAAD.Pro reports.

      Logo files must be in Bitmap (file extension `.bmp`) format.
   d. Click **Font** in the **Text** group.
      The Font dialog opens. You can select any Windows font, style, and size. Click OK when done.



6. Click **OK**.

Tip: You can click Print to print the report directly from this dialog, but it is recommended to preview the report prior to printing.

7. Select the **File** ribbon tab and then select **Report** > **Print Preview**.

   The Print Preview window opens with the report contents. Here you can review and print the report.

   The first and the last pages of the report are shown in the next two figures.

ABC Consultants

Software licensed to CONNECTED User: Jason Coleman

Job No	Sheet No 1	Rev	
Part			
Job Title	Ref		
By	Date 04-Apr-16	Chd	
Client	File RCFrame.std	Date/Time 06-Apr-2016 09:07	

Job Information

	Engineer	Checked	Approved
Name:			
Date:	04-Apr-16		

Project ID	
Project Name	

Structure Type	SPACE FRAME

Number of Nodes	6	Highest Node	6
Number of Element	5	Highest Beam	5

Number of Basic Load Cases	-2
Number of Combination Load Ca	0

Included in this printout are data for:

All	The Whole Structure

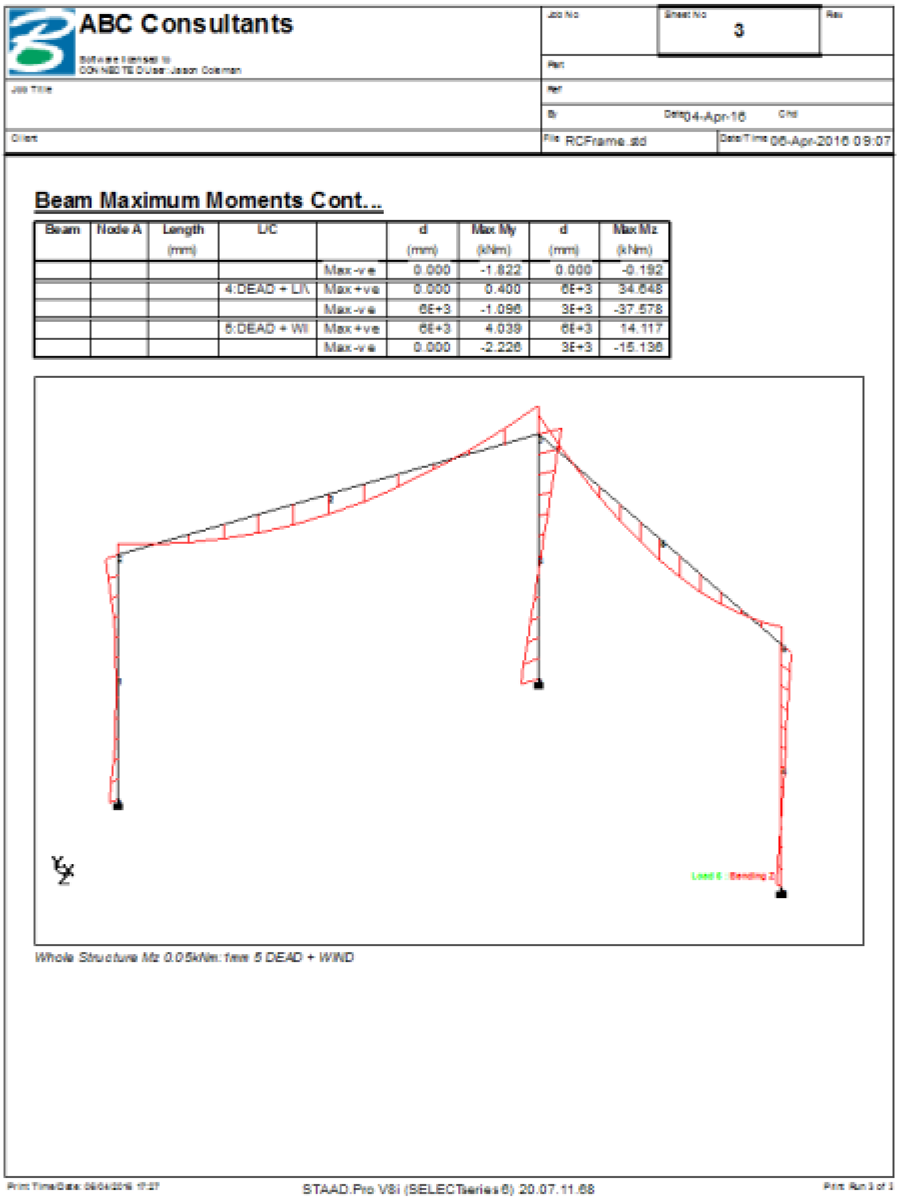
Included in this printout are results for load cases:

Type	L/C	Name
Primary	1	DEAD LOAD
Primary	2	LIVE LOAD
Primary	3	WIND LOAD
Primary	4	DEAD + LIVE
Primary	5	DEAD + WIND

Node Displacement Summary

	Node	L/C	X (mm)	Y (mm)	Z (mm)	Resultant (mm)	rX (rad)	rY (rad)	rZ (rad)
Max X	6	5:DEAD + WII	**6.788**	-0.042	-0.007	6.788	-0.001	0.001	-0.002
Min X	6	4:DEAD + LIV	**-1.020**	-0.096	0.172	1.039	-0.002	-0.000	0.001
Max Y	2	3:WIND LOAI	1.109	**0.003**	0.325	1.155	0.000	-0.000	-0.000
Min Y	3	4:DEAD + LIV	-0.211	**-0.164**	0.211	0.341	0.002	0.000	0.002
Max Z	2	4:DEAD + LIV	-0.172	-0.096	**1.020**	1.039	0.001	0.000	-0.002
Min Z	3	3:WIND LOAI	1.105	-0.002	**-0.068**	1.106	-0.000	0.000	-0.000
Max rX	3	4:DEAD + LIV	-0.211	-0.164	0.211	0.341	**0.002**	0.000	0.002
Min rX	6	4:DEAD + LIV	-1.020	-0.096	0.172	1.039	**-0.002**	-0.000	0.001
Max rY	6	5:DEAD + WII	6.788	-0.042	-0.007	6.788	-0.001	**0.001**	-0.002
Min rY	6	4:DEAD + LIV	-1.020	-0.096	0.172	1.039	-0.002	**-0.000**	0.001
Max rZ	3	4:DEAD + LIV	-0.211	-0.164	0.211	0.341	0.002	0.000	**0.002**
Min rZ	6	5:DEAD + WII	6.788	-0.042	-0.007	6.788	-0.001	0.001	**-0.002**
Max Rst	6	5:DEAD + WII	6.788	-0.042	-0.007	**6.788**	-0.001	0.001	-0.002

Print Time/Date: 06/04/2016 17:27 STAAD.Pro V8i (SELECTseries 6) 20.07.11.68 Print Run 1 of 1



ABC Consultants

Software licensed to CONNECTED User: Jason Coleman

Job No | Sheet No 3 | Rev

Part | Ref

Job Title

By | Date 04-Apr-16 | Chd

Client

File RCFrame.std | Date/Time 06-Apr-2016 09:07

Beam Maximum Moments Cont...

Beam	Node A	Length (mm)	L/C		d (mm)	Max My (kNm)	d (mm)	Max Mz (kNm)
				Max -ve	0.000	-1.822	0.000	-0.192
			4:DEAD + LIV	Max +ve	0.000	0.400	6E+3	34.648
				Max -ve	6E+3	-1.096	3E+3	-37.578
			5:DEAD + WI	Max +ve	6E+3	4.039	6E+3	14.117
				Max -ve	0.000	-2.226	3E+3	-15.136

Whole Structure Mz 0.05kNm:1mm 5 DEAD + WIND

Print Time/Date: 06/04/2016 17:27 | STAAD.Pro V8i (SELECTseries 6) 20.07.11.68 | Print Run 1 of 1

8. Click **Close**.

This brings us to the end of this tutorial.

T.3 - Analysis of a slab

This tutorial provides step-by-step instructions for modeling and analysis of a slab supported along two edges.

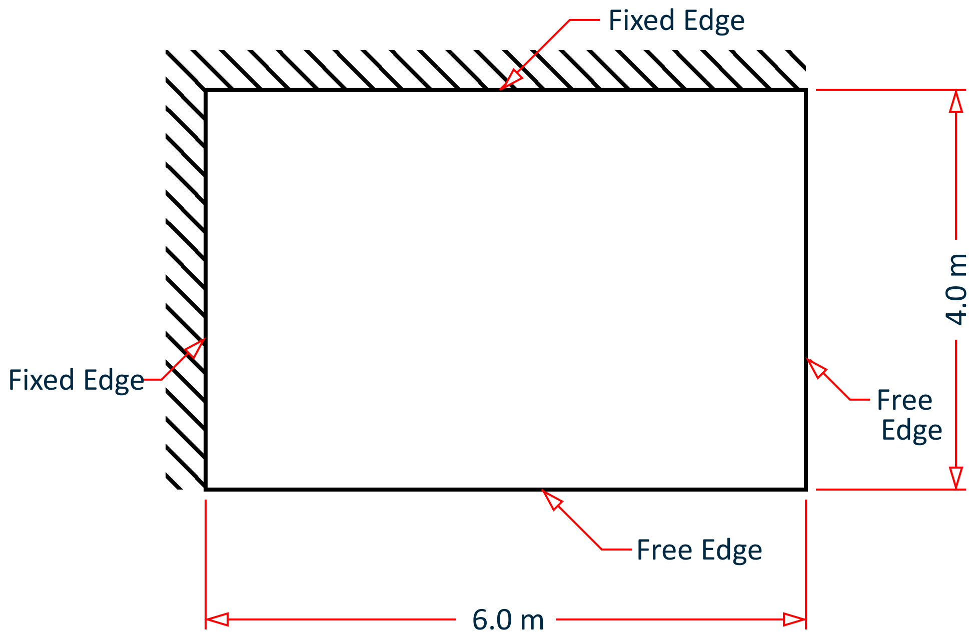


Figure 76: A slab with two supported edges

A copy of the completed input file is typically installed at
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models \Tutorials\Tutorial 3 - Analysis of a Slab.std`.

Related Links

- *EX.Tutorials* (on page 6255)

T.3 Methods of creating the model

As explained in T.1 Methods of creating the model (on page 387) in Tutorial 1, there are three methods of creating the structure data:

1. T.3 Creating the Model using the Physical Modeler (on page 553),
2. T.3 Creating the model using the analytical user interface (on page 561),
3. and T.3 Creating the model using the command file (on page 601).

All three methods of creating the model are explained in this tutorial. The physical modeling method is explained first, beginning with " T.3 Creating the Model using the Physical Modeler (on page 553)," then the analytical model method is explained beginning with " T.3 Description of the tutorial problem (on page 550)." The command file method is explained in " T.3 Creating the model using the command file (on page 601)."

T.3 Description of the tutorial problem

The structure for this project is a slab fixed along two edges. You will model it using 6 quadrilateral (i.e., 4-noded) plate elements. The structure and the mathematical model are shown in the figures below. It is subjected to selfweight, pressure loads and temperature loads. The purpose of this tutorial to create the model, assign all required input, perform the analysis, and review the results.

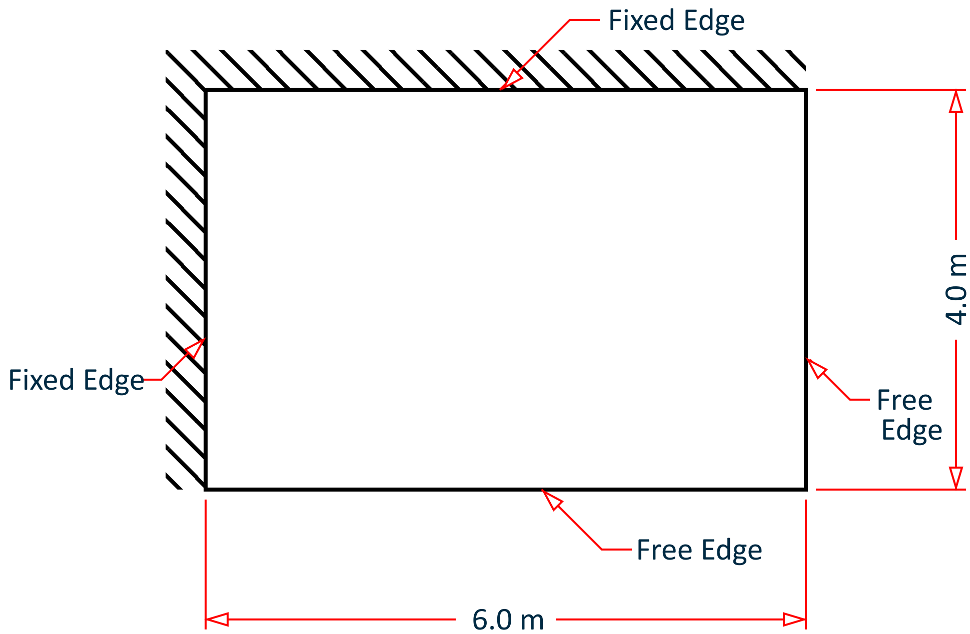


Figure 77: Theoretical model

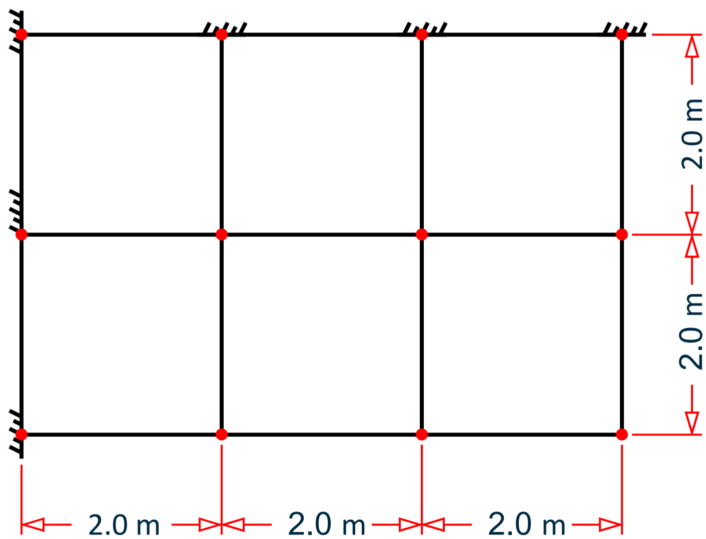


Figure 78: Analytical model

Basic Data for the Structure

Attribute	**Data**
Element properties	Slab is 300 mm thick

Attribute	Data
Material Constants	E, Density, Poisson, Alpha — Default values for concrete
Supports	Nodes along 2 adjacent edges; fixed against rotation and translation
Primary Loads	Load 1: Selfweight Load 2: Pressure Load of 300 kg/m^2 acting vertically downwards Load 3: 75 degree F uniform expansion, plus top surface is 60 degrees hotter than the bottom
Combination Loads	Case 101: Case 1 + Case 2 Case 102: Case 1 + Case 3
Analysis Type	Linear Elastic

T.3 Creating a new structure

On the Start page **New** tab, you will provide some initial data necessary for building the model.

1. On the Start page, select **New**.

   The **New** page opens to the **Model Info** tab.
2. Type as `Plates Tutorial` in the **File Name** field.
3. Specify a **Location** where the STAAD input file will be located on your computer or network.

   You can directly type a file path or click **Browse** to open the **Browse by Folder** dialog, which is used to select a location using a Windows file tree.
4. Select **Physical** for the **Type** of model.

   This option selects the modeling method you want to use:

Analytical	For creating a model using either the STAAD.Pro analytical modeling interface or the command input file editor.
Physical	For creating a model using the STAAD.Pro Physical Modeler interface.
Building	For creating a building model structure using the Building workflow.
5. Select **Metric** as the system of **Units.**

   Tip: The units can be changed later if necessary, at any stage of the model creation.
6. (Optional) Select the Job Info tab to enter related project details, names and dates for quality analysis, and ProjectWise Project information.
7. Click **Create**.

The STAAD.Pro modeling environment opens and your model file is then opened in the STAAD.Pro Physical Modeler.

Alternatively, if you want to use either the analytical modeling workflow or the command input file editor to create the model, choose **Analytical**, click **Create**, and then proceed to either " T.3 Creating the model using the analytical user interface (on page 561) " or " T.3 Creating the model using the analytical user interface (on page 561)," respectively.

T.3 Creating the Model using the Physical Modeler

You are now ready to start building the model geometry. The steps for doing this are described in the following sections.

Tip: Refer to the STAAD.Pro Physical Modeler Application Window Layout for reference on the application window.

T.3 Generate the model geometry

1. On the **Model** ribbon tab, select the **Grid** tool in the **Create** group.

   The **Create Grid** dialog opens.
2. Supply the grid details:
   a. Type `Grid 1` in the **Name** field.
   b. Select **XZ** as the **Plane**.
   c. Leave the **Creation Method** as **By Spacing**.
   d. Type `3` in the **Number of Spaces in X** and `2 m` in the **Grid spacing X** field.
   e. Type `2` in the **Number of Spaces in Z** and `2 m` in the **Grid spacing Z** field.
   f. Click **OK**.
   The grid is created.
3. Orient the view from the top:
   a. (Optional) If the rotation controls are hidden, click the rotation widget in the lower-right corner of the view window.
   b. Select the **Switch to top view** tool.
   c. (Optional) Click the rotation widget again to hide the controls.

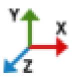
   Tip: Alternatively, you can press and hold the right mouse button and drag the mouse pointer to quickly rotate the view so it is easy to select the column bases.
4. On the **Model** ribbon tab, select the **Surface** tool in the **Create** group.

5. Draw the surface:
   a. Click on the grid point at (0,0,0) (i.e,. top-left corner).
   b. Click on the grid corners in a counterclockwise order. Double-click on the last point to stop drawing surface vertices.

      (6,0,0)
      (6,0,4)
      (0,0,4)

   c. Select the **Surface** tool again to deactivate the tool.
6. On the **View** ribbon tab, select the **Local Axis** tool in the **Model** group.

   The local axes are displayed on the center of the surface. Note that local axis x is aligned with the edge defined by the first two nodes.

T.3 Specifying element and material properties

1. Select the surface element.
2. On the **Surface Tools** ribbon tab, select the **Material** tool in the **Assign Properties** group.

   The **Assign material** dialog opens.
3. Assign a catalog concrete material:
   a. Select the **Source** as **Catalog**.
   b. Select the **Type** as **Standard**.
   c. Select the **Country** as **United States**.
   d. Select the **Material type** as **CONCRETE_ACI318**.
   e. Select **4000 psi** from the **Material name** list.
   f. Click **OK**.

   The dialog closes and the material name is displayed on the surface.
4. On the **Surface Tools** ribbon tab, select the **Thickness** tool in the **Assign Properties** group.

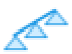
   

   The **Assign Surface Thickness** dialog opens.
5. Type `300 mm` in the **Thickness** field and then click **OK**.
   The dialog closes and the thickness is displayed on the surface.

T.3 Specifying supports

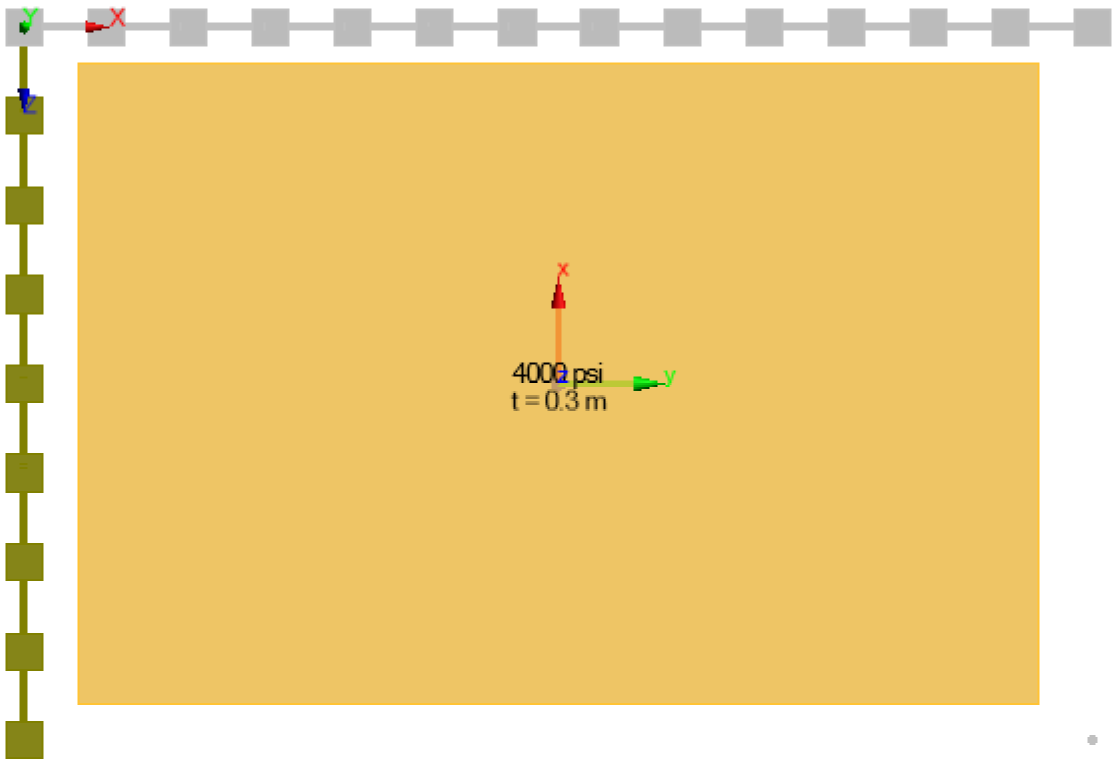
Tip: You may want to turn off the display of Loads to make selecting the surface easier. To do so, select the **Loads** tool in the **Toggle Attributes** group on the **View** ribbon tab.

1. Select the surface and the nodes along the “top” edge:
   a. Click in an empty area of the view window to deselect all objects.
   b. Click on the surface to select it.
   c. While holding **<Ctrl>**, drag a rectangular area surrounding nodes N2 and N3.
2. On the **Surface Tools** ribbon tab, select the **Linear** tool in the **Mat and Edge Supports** group.

   The **Assign Custom Linear Support** dialog opens.
3. Specify the support parameters:
   Leave the **Support type** selection as **Restraints.**
   a. Type 3 in the **Segments** field.
   b. Check all the restrained options for **X**, **Y**, and **Z** in both the **Translation** and **Rotation** groups.
   c. Click **OK**.

   A series of blocks are drawn along the edge to indicate the support condition.
4. Repeat steps 1 through 3 but select the “left” edge nodes (N1 and N2) and use 2 segments.



T.3 Specifying load groups 1 and 2

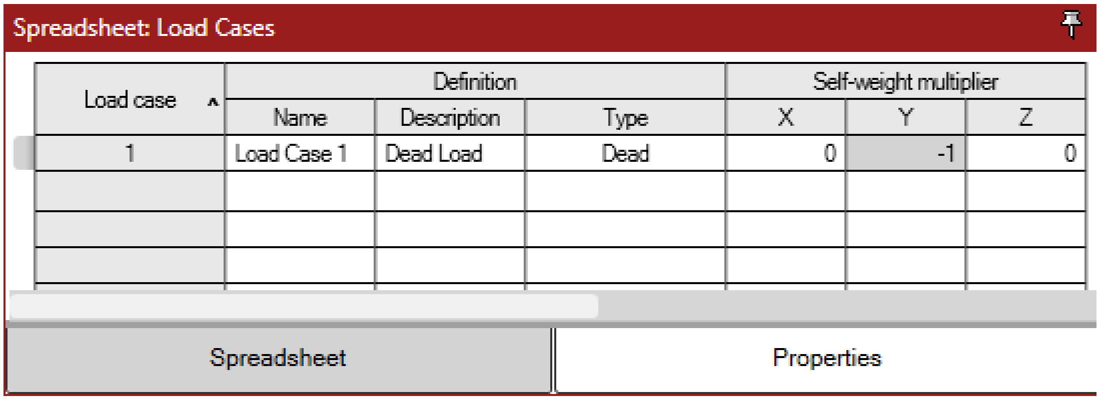
1. On the **Spreadsheet** ribbon tab, select the **Load Cases** tool in the **References** group.

The Load Cases spreadsheet opens.

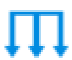
2. For Load 1:
   a. Type `Dead Load` in the description field of Load case 1.
   b. Select **Dead** as the **Type**.
   c. Type `-1` in the **Self-weight multiplier** in field **Y**.

   Note: This value is negative.

   d. Press **<Enter>**.



3. Create a load group for the external pressure load:
   a. On the **Model** ribbon tab, select the **Load Case** tool in the **Load** group.

   The **Add Load Case** dialog opens.
   b. Type `External Pressure Load` in the **Name** field.
   c. Select **Floor Unspecified** as the **Type**.
   d. Click **OK**.

   The new load group is created and selected as the active load group.
4. On the **Surface Tools** ribbon tab, select the **Pressure** tool in the **Loads** group.

   The **Add Surface Uniform Pressure** dialog opens.
5. Specify the pressure load:
   a. Type `-300 kg/m2` in the **Pressure** field.
   b. Select **Global Y** as the **Direction**.

   > **Tip:** Since the surface is parallel to the XZ plane, you could also have selected local 3 as to achieve the same direction. However, due to the counterclockwise selection of nodes, the local 3 axis is oriented in the opposite direction of the Global Y. Therefore, you would use a *positive* magnitude for that case.

   c. Click **OK**.

The temperature load must be added directly to the analytical model in the STAAD.Pro interface. This is done *after* you generate the analytical model.

T.3 Generate the Analysis Model

You have now completed the majority of the modeling portion of this tutorial. You will now send the model data back to STAAD.Pro where you will add the final load case, create load combinations, add output commands, select analysis criteria, and specify post-processing print commands.

1. Change the mesh size:

a. Select the **File** ribbon tab.
   The backstage view opens.
b. Select the **Options** tab.
   The **Options** dialog opens.
c. Select the **Analysis model** tab.
d. Type `2.0 m` in the **Maximum allowed distance between nodes** field.
e. Click **OK**.

2. Either:

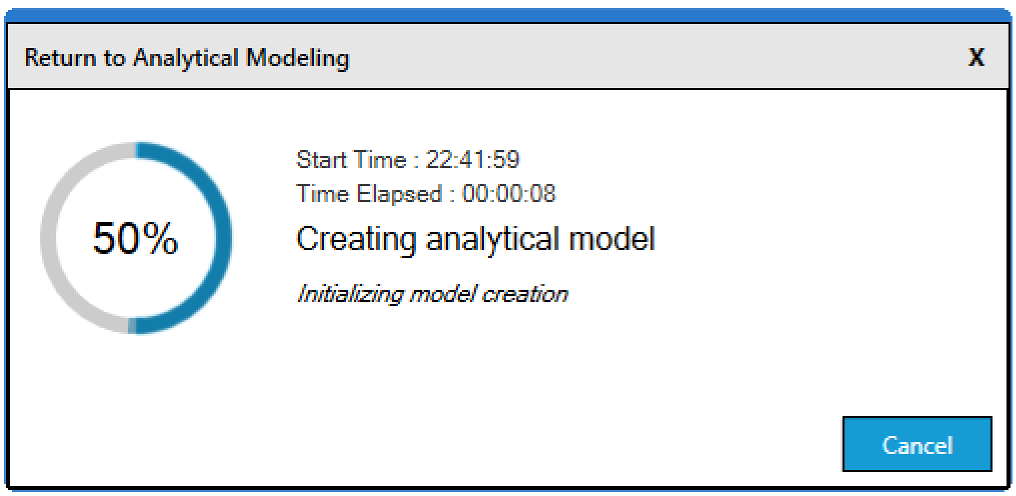
   On the **Model** ribbon tab, select the **Return to Analytical Modeling** tool in the **STAAD.Pro** group

   or

   On the **File** ribbon tab, select **Create analysis model** in the backstage tabs

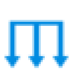
   The Return to Analytical Modeling dialog opens and displays the progress of the analytical model being generated.



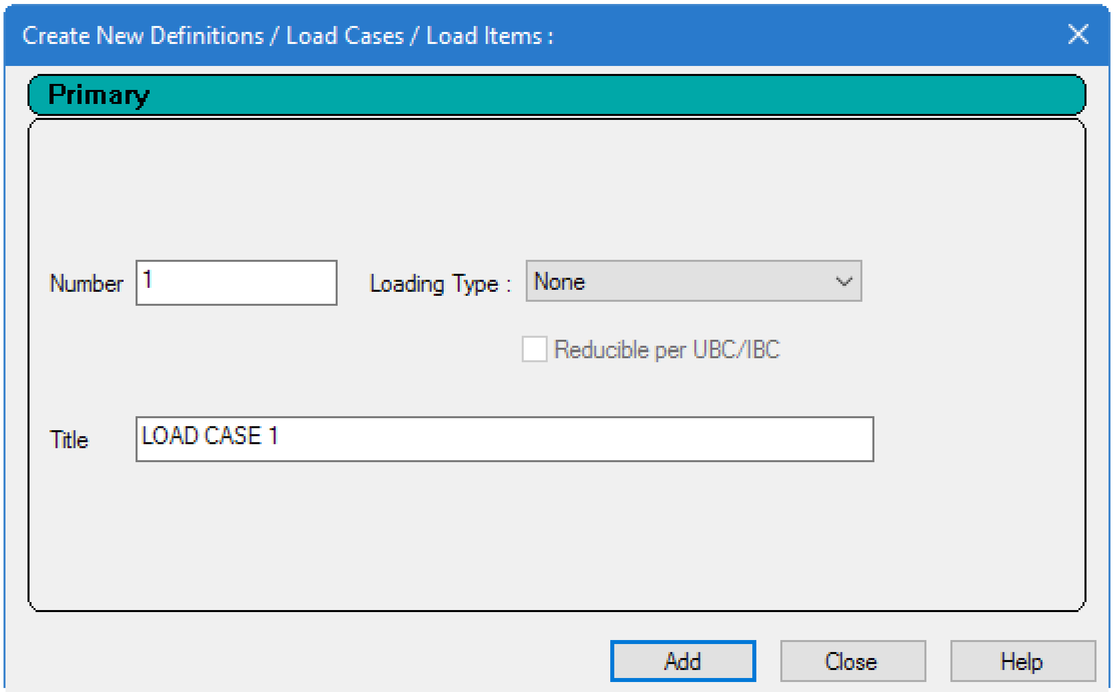
T.3 Create primary load cases

1. In STAAD.Pro, select the **Loading** page in the **Analytical Modeling** page control bar.

   The **Loading** ribbon tab is selected and the **Load & Definition** dialog opens.
2. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.

   The **Add New: Load Cases** dialog opens.



3. Create a pair of primary load cases:
   a. Type `Dead Load` as the **Title** for Load Case 1.

      Leave the **Number** as the default (1) and leave the **Loading Type** as None.
   b. Click **Add**.

      Note that the dialog stays open but the load case number is automatically incremented.
   c. Type `External Pressure Load` as the **Title**.

      Again, it is not necessary to select a **Loading Type**.
   d. Click **Add**.
   e. Click **Close**.
4. Assign the reference loads to the primary load cases:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

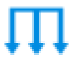
      

      The **Add New Load Items** dialog opens.
   b. Select the **Repeat Load** > **Reference Load** tab on the left.
   c. Select **R1: "Load group 1"** in the **Available Load Cases** list.
   d. Click **>**.
      The reference load case is added to the selected loads list.
   e. Click **Add**.
   f. Click **Close**.

   The reference load case is now added included in the primary load case for analysis.
5. In the window status bar, select **2: External Pressure Load** as the current load case.
6. Repeat Step 4 to add the reference load for Load group 2 to the External Pressure Load primary load case.

T.3 Add temperature load case

1. Create a primary load case titled `Temperature Load` and then select it in the status bar.
   a. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.

   The **Add New Load Cases** dialog opens.
   b. Type `Temperature Load` as the **Title**.
   There are no load combinations used in this model, so you can leave the Loading Type as None.
   c. Click **Add**.
   d. Click **Close**.
2. In the program status bar, select **3: Temperature Load** as the current load case.
3. To generate and assign the third load type:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   The **Add New Load Items** dialog opens.
   b. Select the **Temperature** option under the **Temperature Loads** item.
   c. Type `40` in the **Temperature Change for Axial Elongation**field.
   d. Type `30` in the **Temperature Differential from Top to Bottom** field.
   Leave the **Temperature Differential from Side to Side (Local Z)** field as 0 (default).
   e. Click **Add** and then click **Close**.

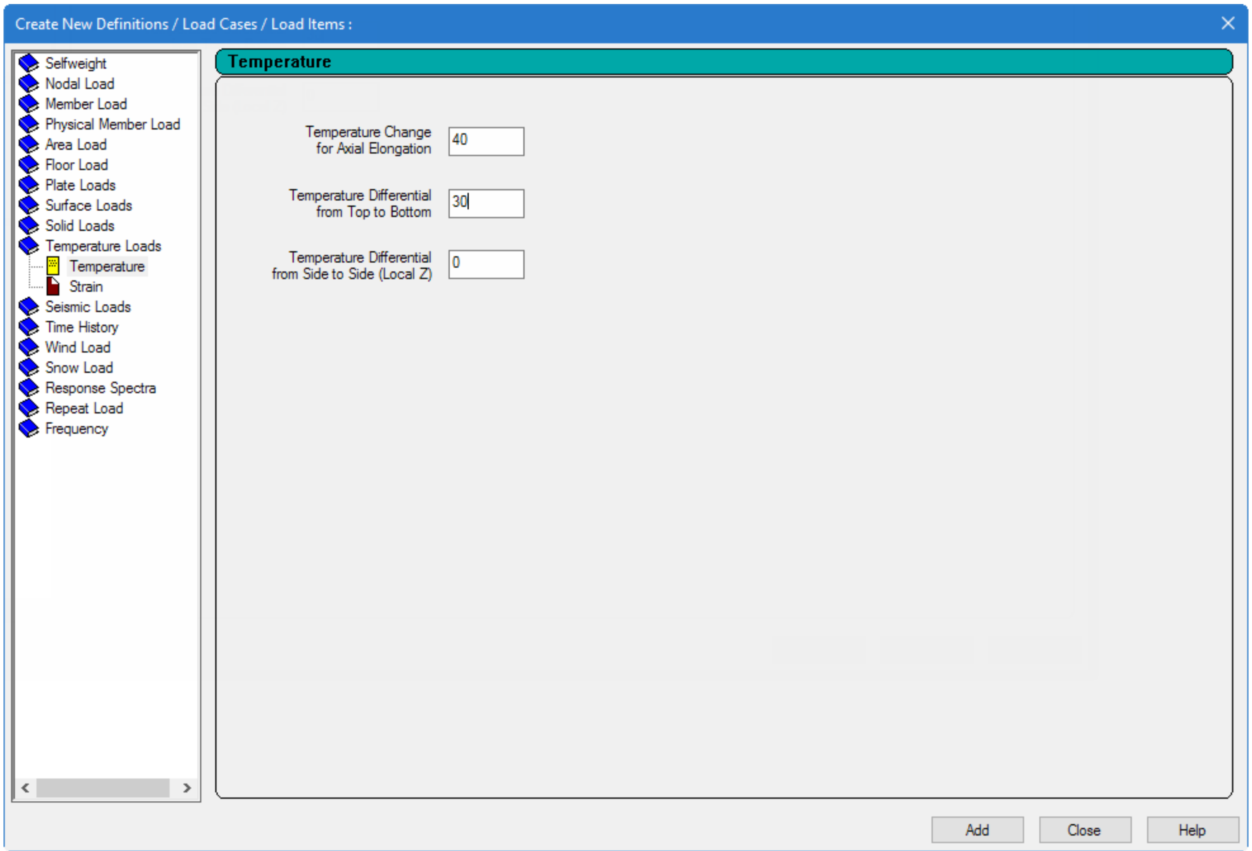


Figure 79:

4. Assign the temperature load case to all of the elements:

a. In the **Load & Definition** dialog, select **TEMP 40 30**.

This is the input command for the temperature load.

b. Select the **Assign to View** option.

c. Click **Assign**.

A message dialog opens confirming you want to make this assignment.

d. Click **Yes**.

T.3 Creating the model using the analytical user interface

The following procedures describe how to use the traditional STAAD.Pro analytical user interface to build the model. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

Tip: Refer to GS. Application Window Layout (on page 54) for reference on the application window.

T.3 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

The STAAD input file commands generated are:

```
JOINT COORDINATES
1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4
ELEMENT INCIDENCES SHELL
1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6
4 4 3 9 10 ; 5 3 6 11 9 ; 6 6 8 12 11
```

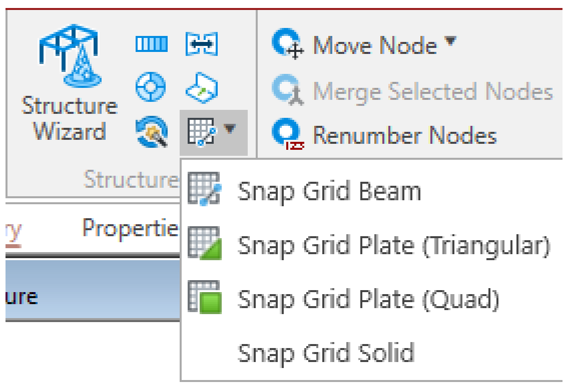
In this tutorial, you will optionally explore four different methods to create the model:

T.3 Creating the Plates - Method 1

To create the plate elements for the slab using a mixture of drawing an element and the Copy/Paste facility, do the following steps.

T.3 Setup the Grid

1. On the **Geometry** ribbon tab, select the **Snap Grid/Plate (Quad)** tool in the **Structure** group.



   The **Snap Node/Beam** dialog closes and the **Snap Node/Plate** dialog opens.
2. On the **Snap Node/Plate** dialog, click **Create**.
   A dialog opens which will enable us to set up a grid.

   Within this dialog, there is a drop-down list from which you can select Linear, Radial, or Irregular form of grid lines.

Linear	used to place the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board
Radial	used to place construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modeled as piece-wise linear straight line segments
Irregular	used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane

3. Select **Linear**, which is the Default Grid.

   In our structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. So, in this dialog, let us keep **X-Y** as the **Plane** of the grid. The size of the model that can be drawn at any time is controlled by the number of **Construction Lines** to the left and right of the origin of axes, and the **Spacing** between adjacent construction lines.
4. Type a **Name** of `Grid 1`.

5. Select **X-Z** as the **Plane** option.
6. Type 6 as the number of lines to the **Right** of the origin along **X** and 4 above the origin along **Z**.

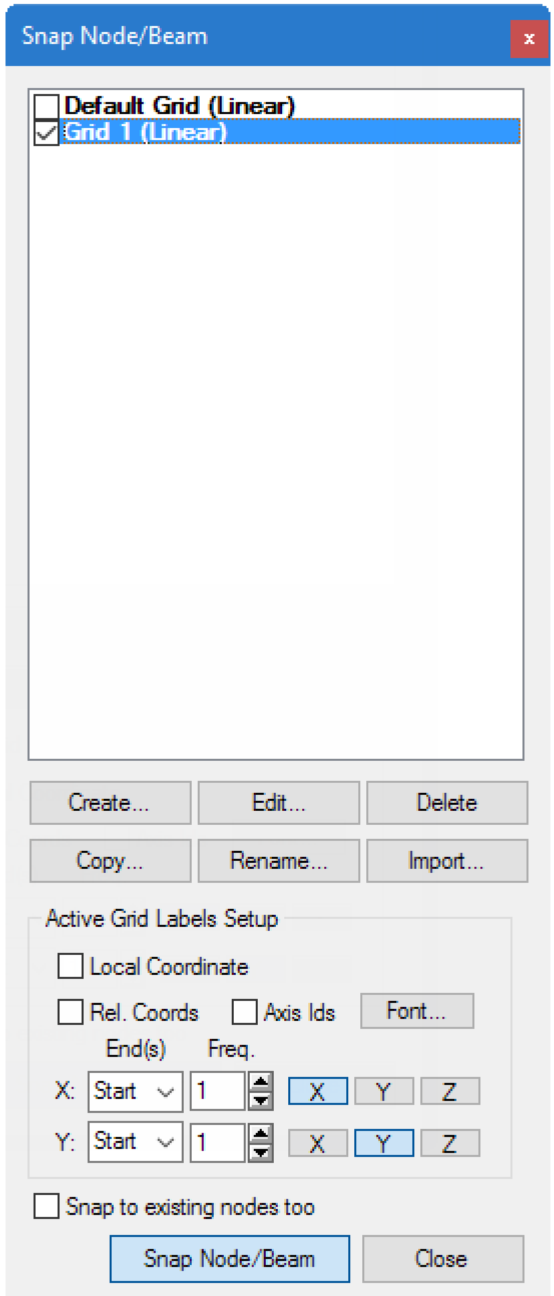
   Leave the spacing as 1 m.
7. Click **OK**.
8. In the **Snap Node/Beam** grids list, check the new **Grid 1** option.

   You can create any number of grids. By providing a name, each new grid can be identified for future reference.

   Tip: Please note that these settings are only used to generated construction lines. These construction lines enable you to easily draw the structure but do not restrict our overall model to those limits.

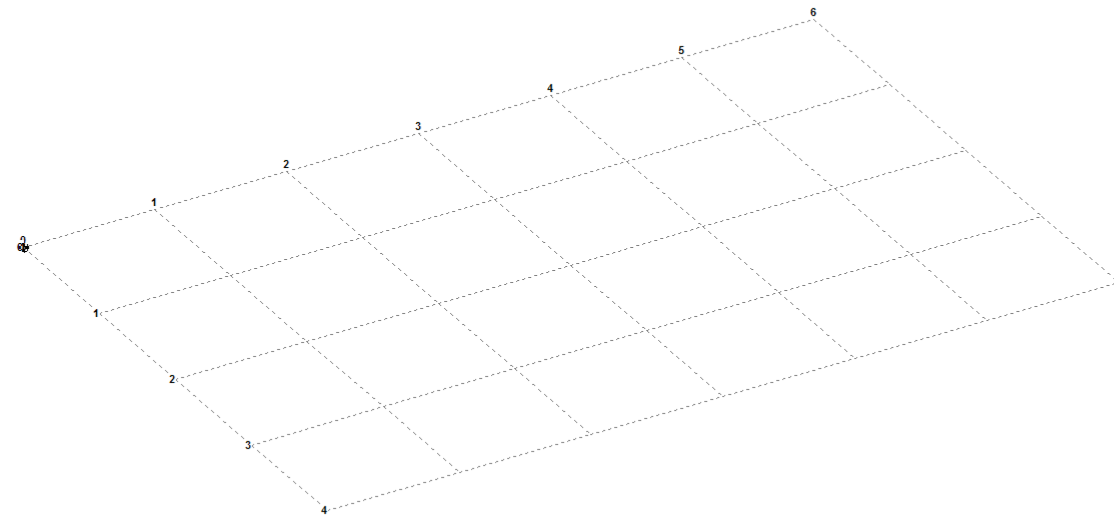
   Tip: To change the settings of this grid, select the name in the **Snap Node/Beam** dialog and then click **Edit**.

   The check by **Default Grid** is automatically cleared. STAAD.Pro will only display a single grid at a time.



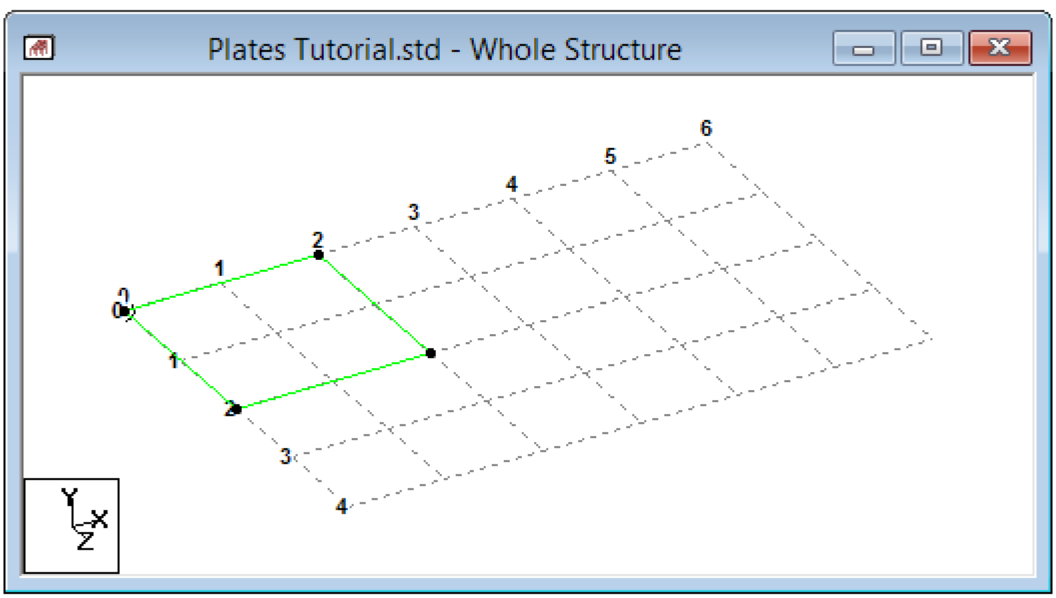
T.3 Create Plates Method 1 Element 1

1. In the View window, click at the origin (0, 0) to create the first node.
   A line is "rubber-banded" between this node and the mouse pointer, which previews the edge of the plate to placed with the next mouse click.



2. Click on the following points to create nodes and automatically join successive nodes by beam members.

   (2, 0, 0)
   (2, 0, 2)
   (0, 0, 2)



3. Click **Close** in the **Snap Node/Plate** dialog.
4. Press **<Shift+P>** to display the plate number.

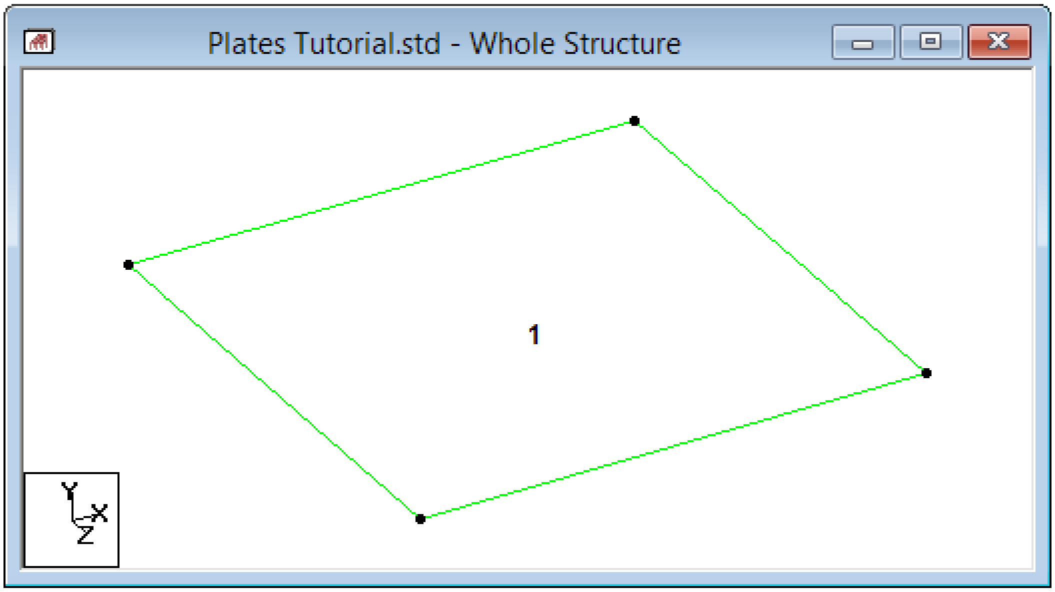


Figure 80: Plate 1 drawn

It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.3 Create Plates Method 1 Element 2

Creating Elements by Coping and Pasting

Examining the structure shown in " T.3 Description of the tutorial problem (on page 550) " of this tutorial, it can be seen that the remaining elements can be easily generated copying the existing plate and then pasting the copied element at specific distances.

1. On the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

2. Select plate 1 in the View window.
3. Either:

   Right-click and select **Copy** from the pop-up menu

   or

   on the **Geometry** ribbon tab, select the **Copy** tool in the **Clipboard** group 

   or

   press **<Ctrl+C>**

4. To paste the copied plate element, either:

   Right-click and select **Paste** from the pop-up menu

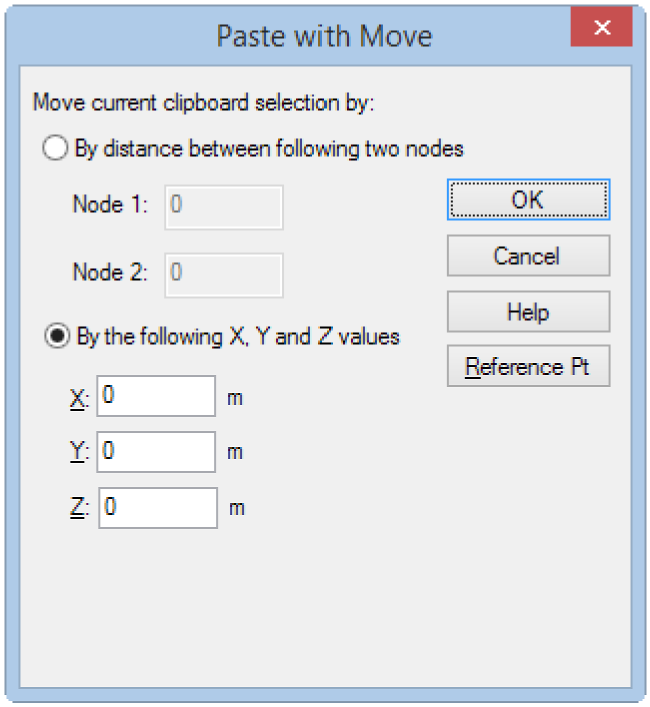
   or

   on the **Geometry** ribbon tab, select the **Paste** tool in the **Clipboard** group 

   or

   press **<Ctrl+V>**

   The **Paste with Move** dialog opens.



5. Select the **By the following X, Y and Z values** option and then type **2** (m) in the X field.

   Leave the Y and Z values as 0.
6. Click **OK**.

   The dialog closes and the plate is copied to Plate 2 at the specified X axis increment.

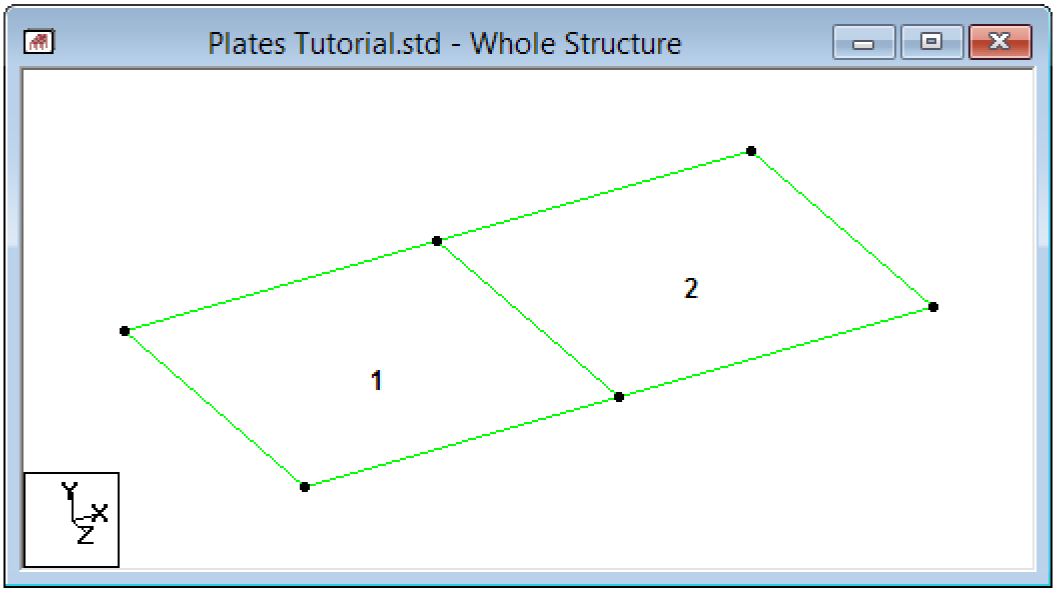


Figure 81: Plate 2 copied from plate 1

7. Repeat steps 4 through 6 except with an **X** of 4 (m) to create plate 3 from plate 1.

   You have now created one half of the symmetric set of plates. You can now copy and paste all three plates to complete the structure.

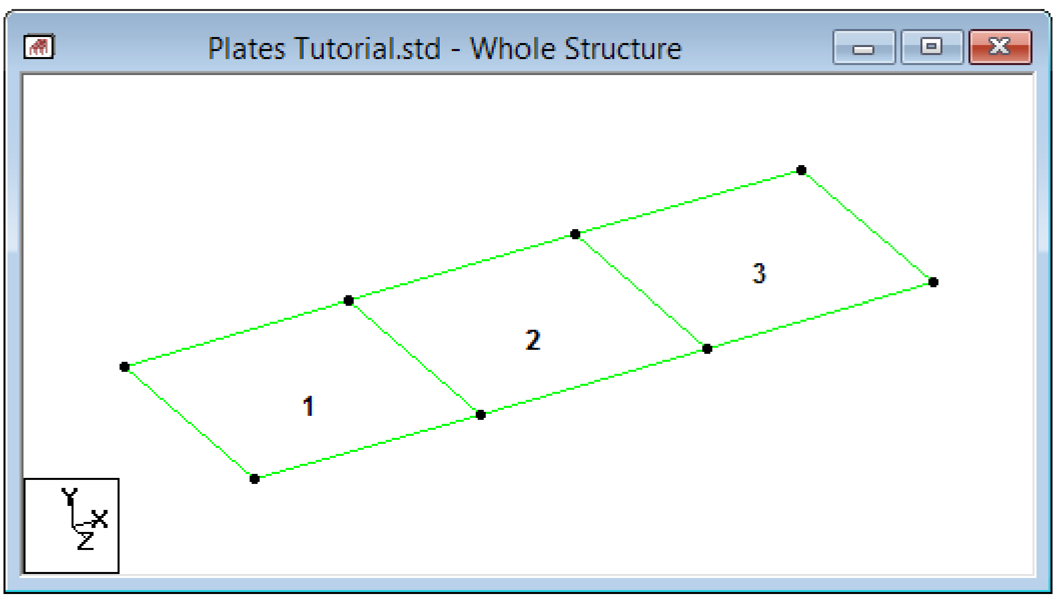


Figure 82: Plate 3 copied from plate 1

8. To select all plates, either:

   click-and-drag a window around all three plates using the **Plates Cursor** tool

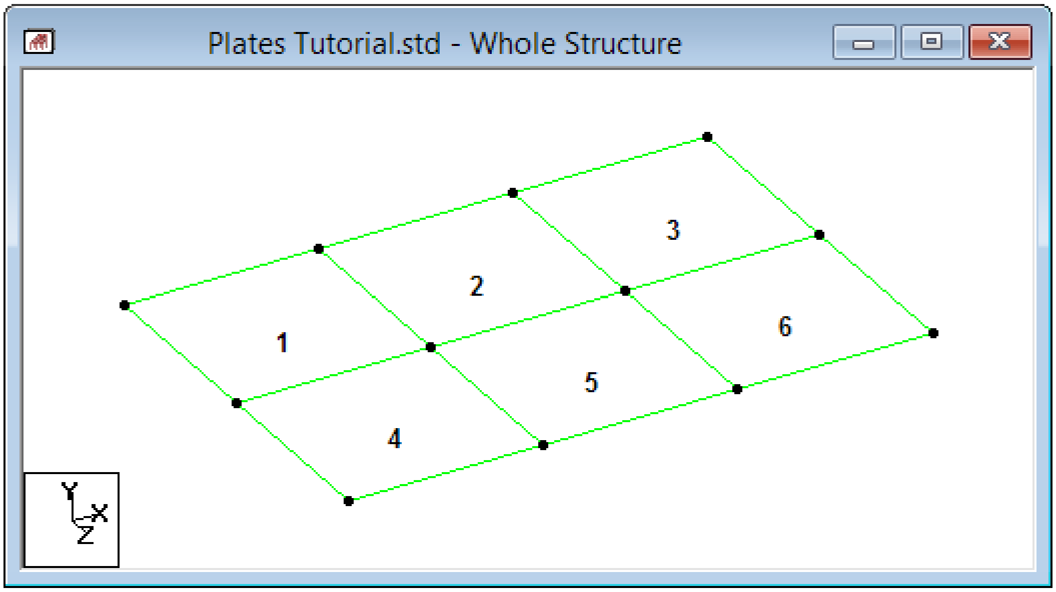
   or

   press **<Ctrl+A>**

   or

on the **Geometry** ribbon tab, select the **Plates Cursor** > **All Plates** tool in the **Selection** group.

9. Repeat steps 3 through 6 to copy and paste all the plates, except with a **Z** of 2 m (leave the X and Y values as 0).



10. Click anywhere in away from the plates to deselect them.

T.3 Create Plates Delete to Try Another Method

If you want to explore the remaining methods of creating this model, the current structure will have to be entirely deleted.

1. To select all plates, either:

   click-and-drag a window around all three plates using the **Plates Cursor** tool

   or

   press **<Ctrl+A>**

   or

   on the **Geometry** ribbon tab, select the **Plates Cursor** > **All Plates** tool in the **Selection** group.
2. Either:

   press **<Delete>**

   or

   on the **Plate Tools** ribbon tab, select the **Delete** tool in the **Clipboard** group

   A message dialog opens to confirm the deletion of the selected plates.
3. Click **OK**
   A message dialog opens indicating that orphan nodes have been created and to confirm their deletion.
4. Click **Yes**.
   The entire structure is now deleted.

T.3 Creating the Plates - Method 2

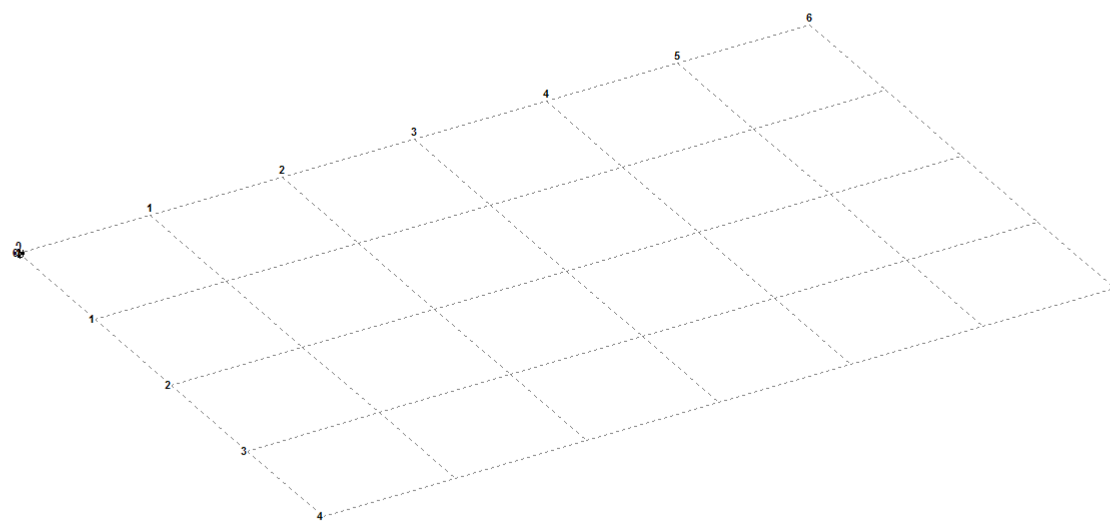
To create the plate elements using a mixture of drawing an element and the Translational Repeat facility, use the following steps.

T.3 Method 2 Creating Element 1

In this method, you will use the Translational Repeat feature in STAAD.Pro to create the model. First, you need at least one existing entity to use as the basis for the translational repeat.

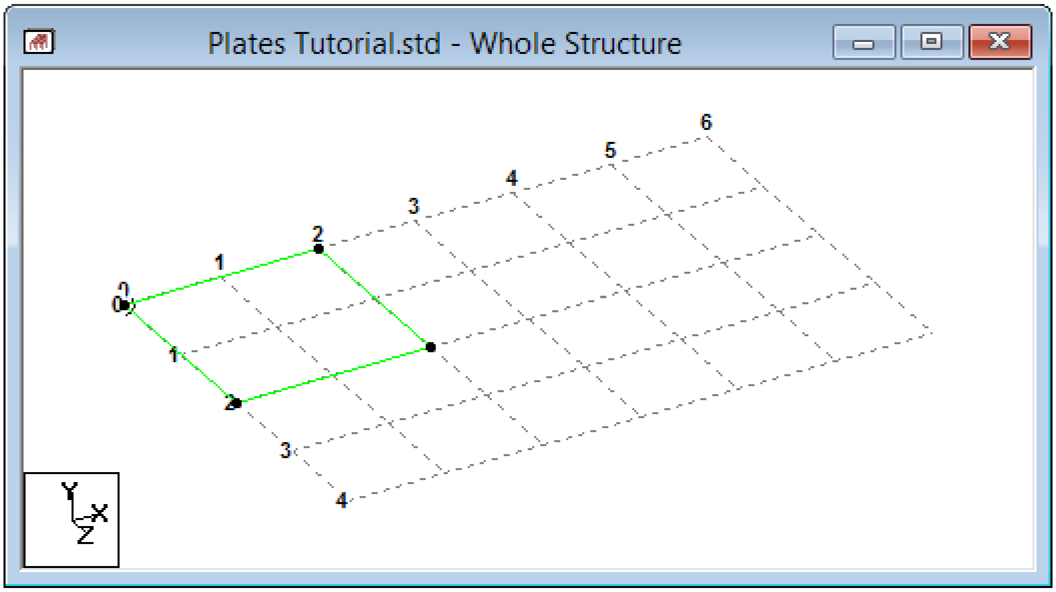
This method uses the same drawing grid from Method 1. Refer to " T.3 Setup the Grid (on page 562)."

1. In the View window, click at the origin (0, 0) to create the first node.
   A line is "rubber-banded" between this node and the mouse pointer, which previews the edge of the plate to placed with the next mouse click.



2. Click on the following points to create nodes and automatically join successive nodes by beam members.

   (2, 0, 0)
   (2, 0, 2)
   (0, 0, 2)



3. Click **Close** in the **Snap Node/Beam** dialog.
   The grid is hidden.
4. Press **<Shift+P>** to display the plate number.

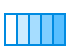
T.3 Method 2 Creating Elements 2 3

In Method 1, it required two separate executions of the Copy/Paste function to create elements 2 and 3. That is because, that feature does not contain a provision for specifying the number of copies you want to create. However, with Translational Repeat you can specify the number of copies and distances to each.

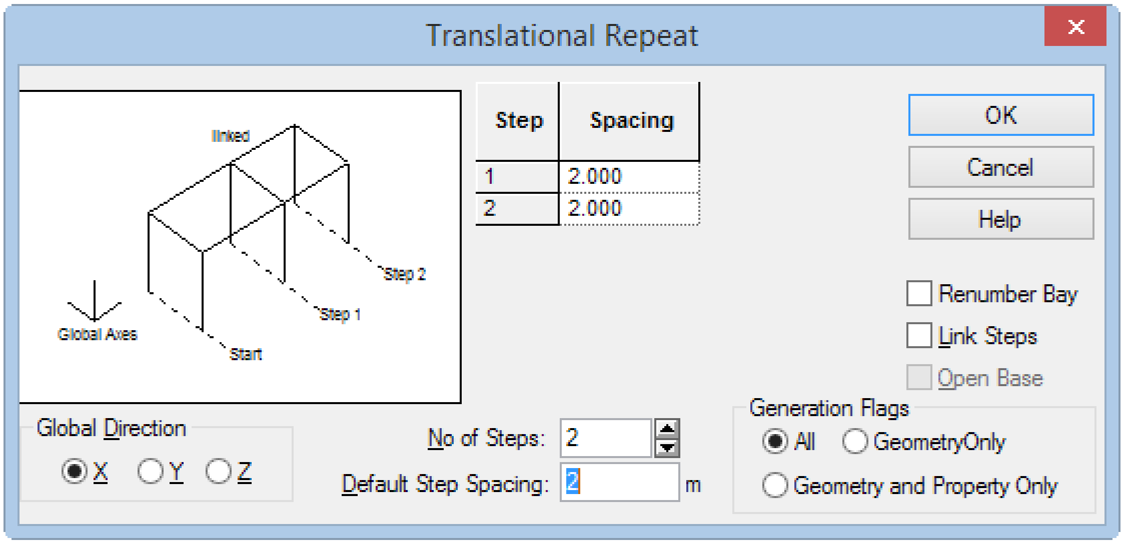
1. On the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

2. Select plate 1 in the View window.
3. On the **Geometry** ribbon tab, select the **Translational Repeat** tool in the **Structure** group.

   The **Translational Repeat** dialog opens.



4. Specify the translation repeat parameters:
   a. Select **X** as the **Global Direction** option.
   b. Type 2 (m) in the **Default Step Spacing** field.
   c. Type **2** in the **No of Steps** field and press the Tab key (or click the up arrow to increment the value to 2). The Step and Spacinng table updates to reflect two steps each at 2m spacing.

   Leave the **Renumber Bay** and **Link Steps** options unchecked and the **Generation Flags** set to **All**.

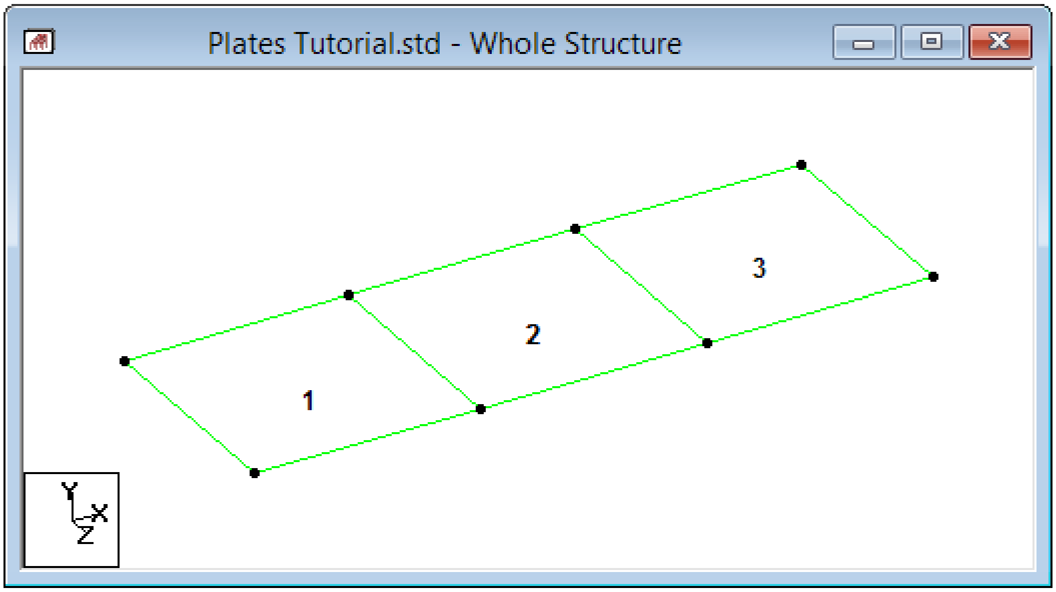
   Note:

   Using the **All** option for Generation Flags (default), all loads, properties, design parameters, member releases, etc. on the selected entities will automatically be copied along with the entities. You can limit this to geometry only or to geometry and properties only. In our example, it does not matter because no other attributes have been assigned yet.

   Renumber Bay is used to specify a custom number scheme for created entities, instead of the sequential number that the program otherwise generates. The **Linked Steps** option is used to generate linking members between new entities that have some distance separating them from the original entities.

5. Click **OK**.

   Elements 2 and 3 are created using the translational repeat parameters.



6. To select all plates, either:

   click-and-drag a window around all three plates using the **Plates Cursor** tool

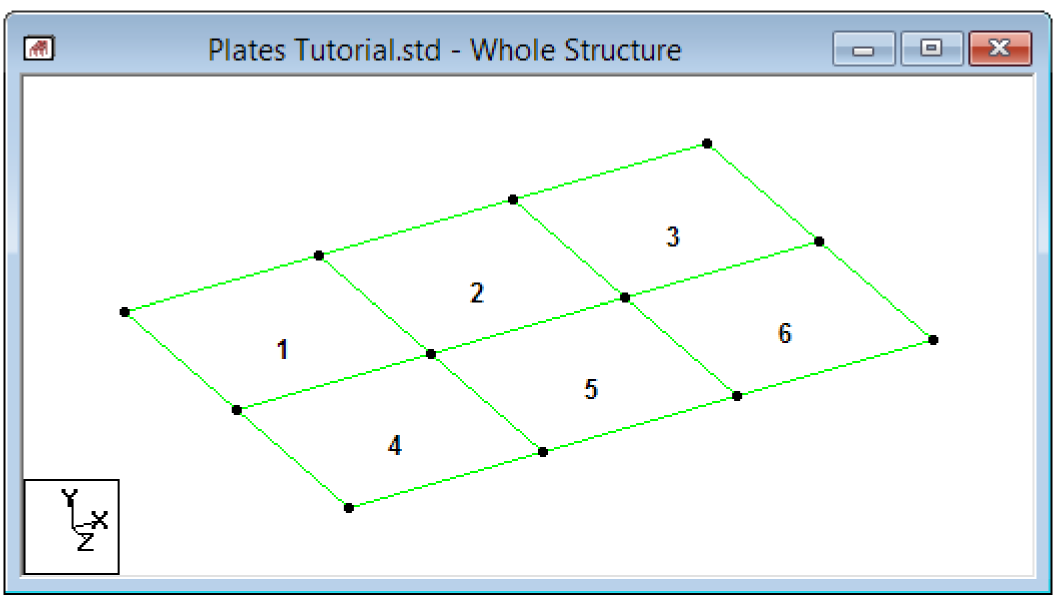
   or

   press **<Ctrl+A>**

   or

   on the **Geometry** ribbon tab, select the **Plates Cursor** > **All Plates** tool in the **Selection** group.
7. Repeat steps 3 and 4 except select **Z** as the **Global Direction** and the **No of Steps** as 1.

> **Tip:** Be sure to type 2 (m) in the **Default Step Spacing** field



8. Click anywhere in away from the plates to deselect them.

T.3 Creating the Plates - Method 3

To create the plate elements using the **Structure Wizard** program, use the following steps.

A program installed with STAAD.Pro called **Structure Wizard** offers a library of pre-defined structure templates (also referred to as “prototypes”), such as Pratt Truss, Northlight Truss, Cylindrical Frame, etc. A surface entity such as a slab or wall, which can be defined using 3-noded or 4-noded plate elements, is one such template. You can also create your own library of structure prototypes. From this wizard, a structural model may parametrically be generated, and can then be incorporated into your main structure.

1. On the **Geometry** ribbon tab, select the **Structure Wizard** tool in the **Structure** group.

   The **Structure Wizard** window opens.

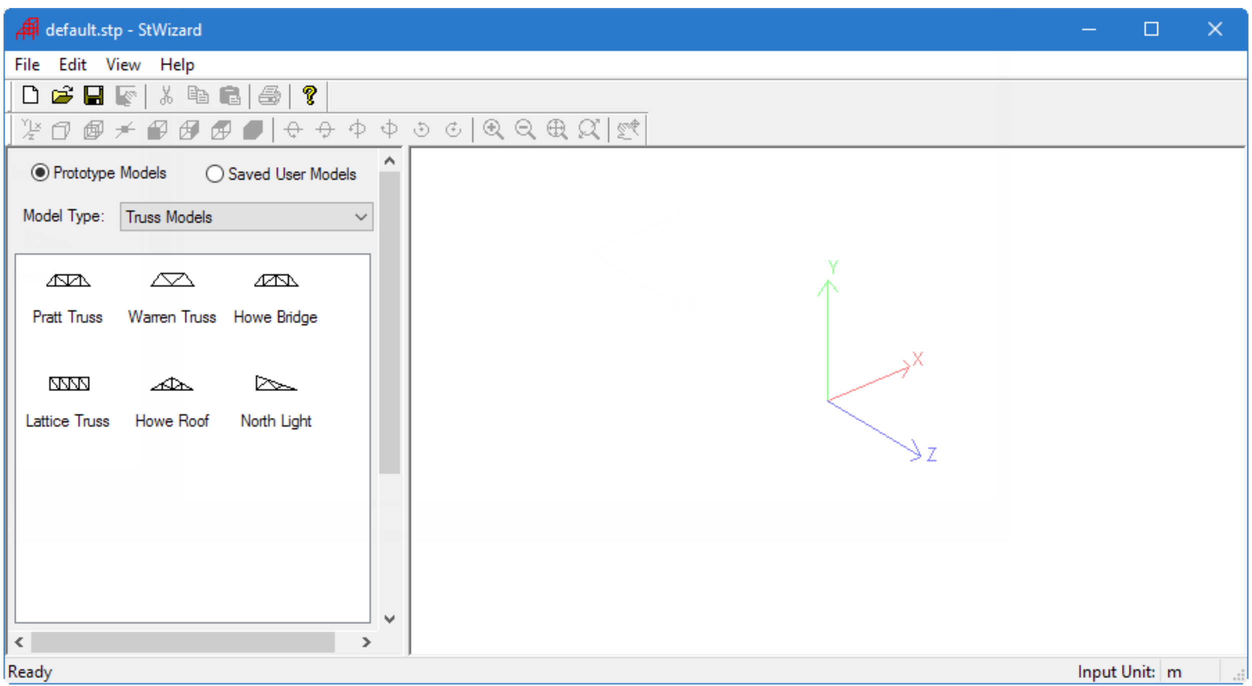
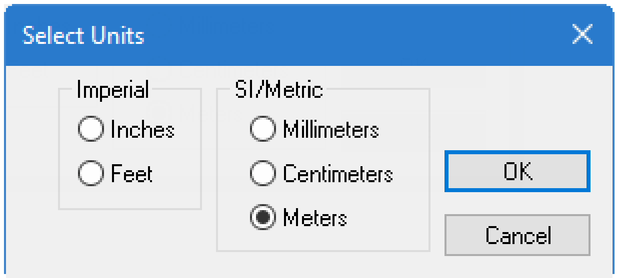
   

   *Figure 84: The **Structure Wizard** window*

> **Tip:** The **Open Structure Wizard** option in the **Where do you want to go?** dialog in the beginning stage of creating a new structure also opens this application.

2. The unit of length should be specified prior to the generation of a model.
   a. Select **File** > **Select Units**in the Structure Wizard window.

      The **Select Units** dialog opens.



b. Select **Meters**.

c. Click **OK**.

3. From the **Model Type** drop-down list, select **Surface/Plate Models**.

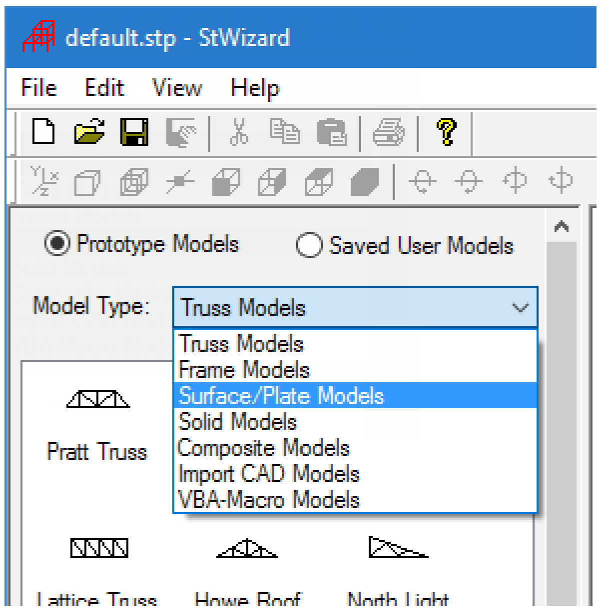


Figure 85:

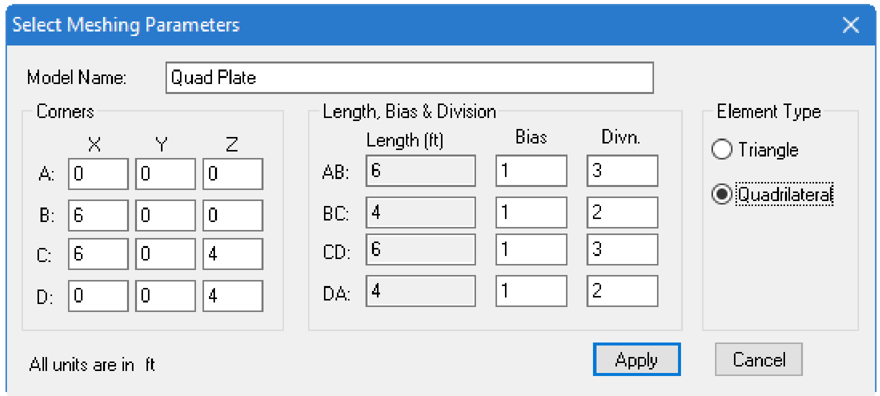
4. Either:

   double-click on the **Quad Plate** option 

   or

   drag the **Quad Plate** option to the right side of the **Structure Wizard** window

   The **Select Meshing Parameters** dialog opens.



5. Specify the meshing boundary corners and the individual element data as follows:
   a. Enter the following **Corners** data:

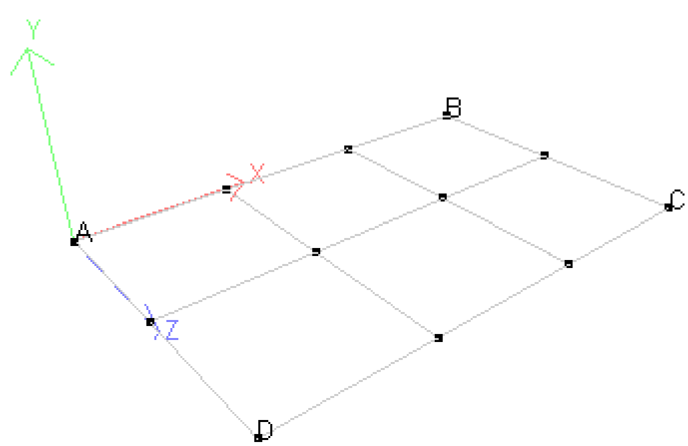
Corner	X	Y	Z
A	0	0	0
B	6	0	0
C	6	0	4
E	0	0	4

   The Length values are automatically calculated.
   b. Type 3 and the **Divisions** along AB and CD sides and 2 for the **Divisions** along BC and DA sides.

   Leave the **Bias** values all as **1**.
   c. Click **Apply**.

The mesh is generated.

Tip: If you made a mistake, right-click in the drawing area and select **Change Property** from the pop-up menu. The dialog opens and you can edit the mesh generations parameters.

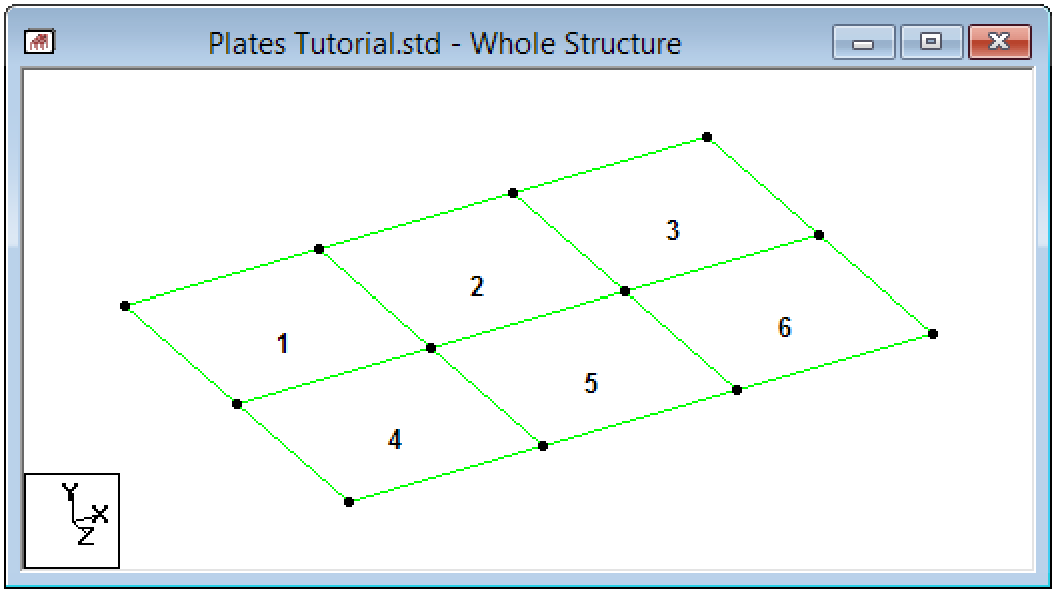


6. Select **File** > **Merge Model with STAAD.Pro Model**.
   You are prompted to confirm that you want to transfer the model data with your STAAD.Pro model.
7. Click **Yes**.

   The **Paste with Move** dialog opens.
8. Click **OK**.

   In this tutorial, there is no existing model nor is there a need to shift the model from the origin.

   The model will now be transferred to the main window.



If you want to proceed with assigning the remainder of the data, go to " T.3 Changing the input units of length (on page 580) ".

T.3 Creating the Plates - Method 4

To create the plates using the Mesh Generation facility, use the following steps.

This method uses the same drawing grid from Method 1. Refer to " T.3 Setup the Grid (on page 562)."

The STAAD.Pro GUI contains a facility for generating a mesh of elements from a boundary (or superelement) defined by a set of corner nodes. The boundary must form a closed surface and must lie within a plane, though that plane can be inclined to any of the global planes.

You define the boundary by selecting the corner nodes. If these nodes do not exist, they must be created before they can be selected.

1. Create the nodes:
   a. On the **Snap Node/Plate** dialog, click **Snap Node/Plate**.
   b. Click at the four corners of the grid at (0, 0, 0), (6, 0, 0), (6, 0, 4), and (0, 0, 4).

   Tip: If the node points are not visible, press **<SHIFT+K>**.
2. In the **Snap Node/Plate** dialog, click **Close**.
3. Use the Plates Cursor and select this plate.
4. To start a quadrilateral mesh of the existing plate element:

a. On the **Plate Tools** dialog, select the **Generate Plate Mesh** tool in the **Model** group.

The **Chose Meshing Type** dialog opens.

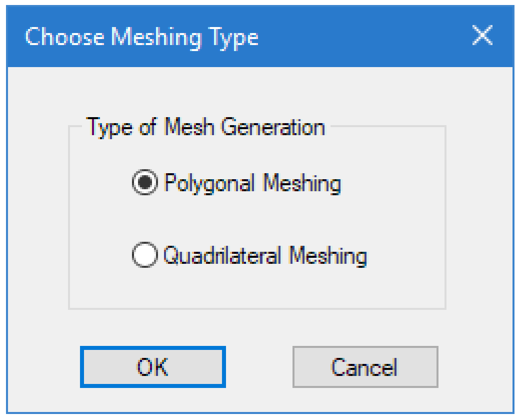


Figure 86:

b. Select the **Quadrilateral Meshing** option.

c. Click **OK**

The **Select Meshing Parameters** dialog opens.

5. Complete the meshing parameters:

   a. Type 3 and the **Divisions** along AB and CD sides and 2 for the **Divisions** along BC and DA sides.

   b. Click **Apply**.

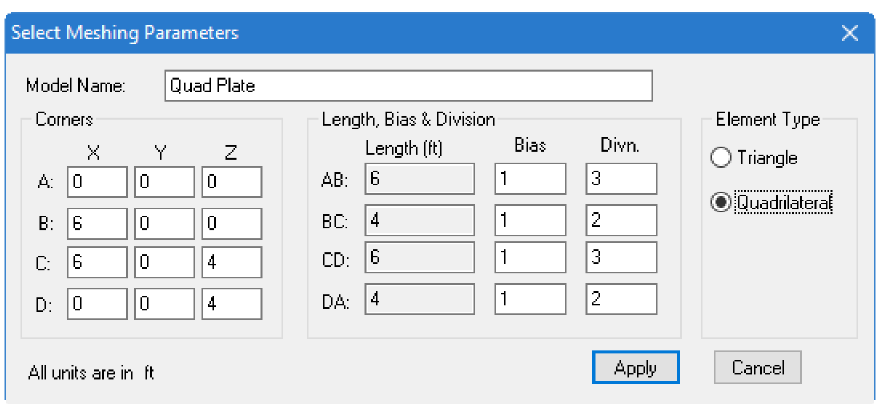


Figure 87:

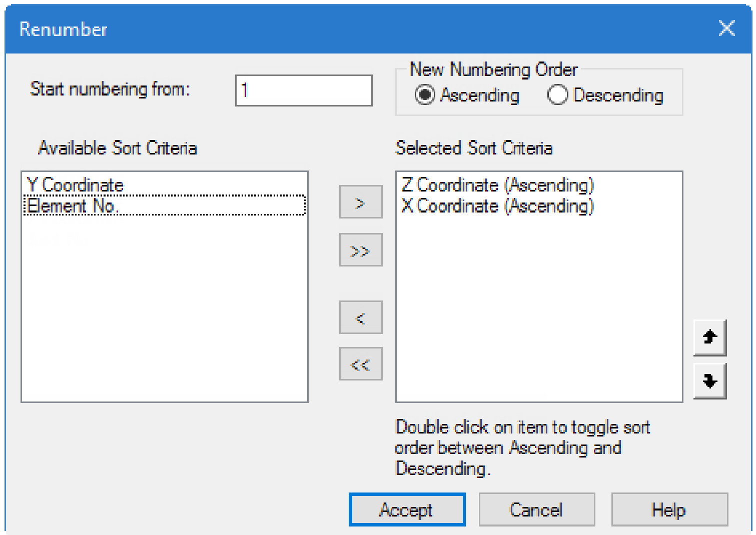
The surface is meshed.

6. Re-number the plate elements:

   a. On the **Plate Tools** ribbon tab, select the **Renumber Plates** tool in the **Model** group.

A warning message opens.

b. Click **Yes**.
The **Renumber** dialog opens.



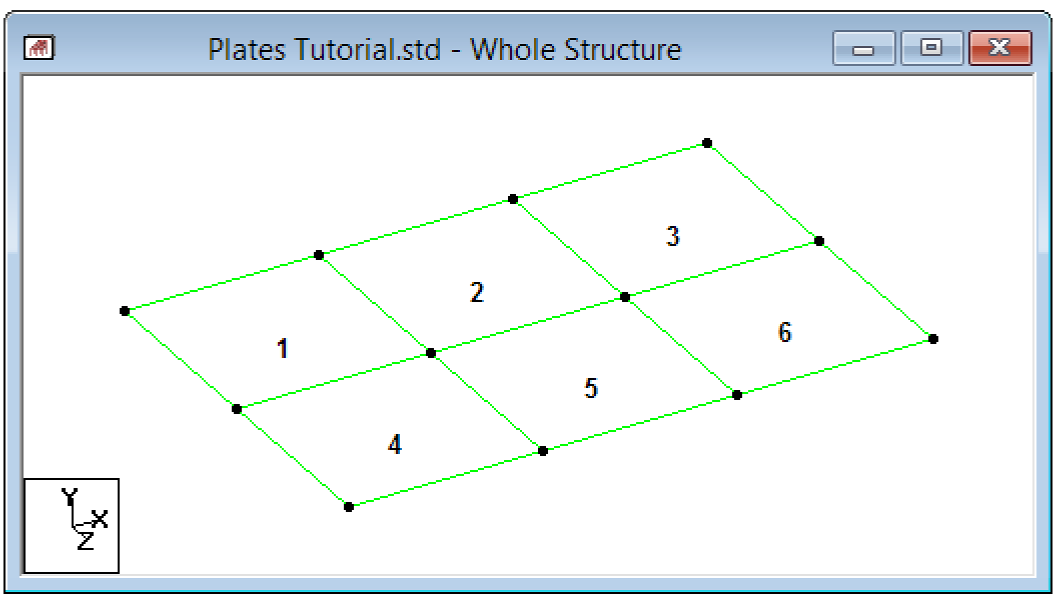
c. Double-click **Z Coordinate** in the **Available Sort Criteria** list.

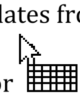
d. Double-click **X Coordinate** in the **Available Sort Criteria** list.

The Z coordinate entry should appear *above* the X coordinate entry in the Selected Sort Criteria list.

e. Click **Accept**.

The six plates are re-numbered as shown below.



As an alternative in step 1, you can press and hold **<Ctrl>** to create only disconnected nodes. Then, when you can use the **Generate Mesh** > **Generate Plate Mesh** tool in the **Plate** group on the **Geometry** ribbon tab to generate the plates from those nodes, rather than re-meshing an existing plate. You then use the plate bound selection cursor  to select these corner nodes to define a meshed boundary.

Tip: Pressing the **<Ctrl>** key while clicking on grid points creates new nodes without connecting those nodes with beams or plates. If the **<Ctrl>** key is not kept pressed, the nodes become connected.

T.3 Changing the input units of length

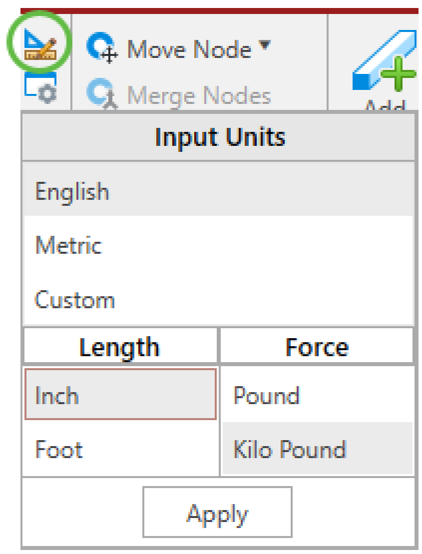
To specify element properties for the structure, it is more convenient to use length units of centimeter instead of meter. This requires changing the current length units of input.

The STAAD input file commands generated are:

```
UNIT CM KN
```

1. On the **Geometry** ribbon tab, select the **Input Units** tool in the **Structure** group.

The **Set Input Units** pop-up opens.



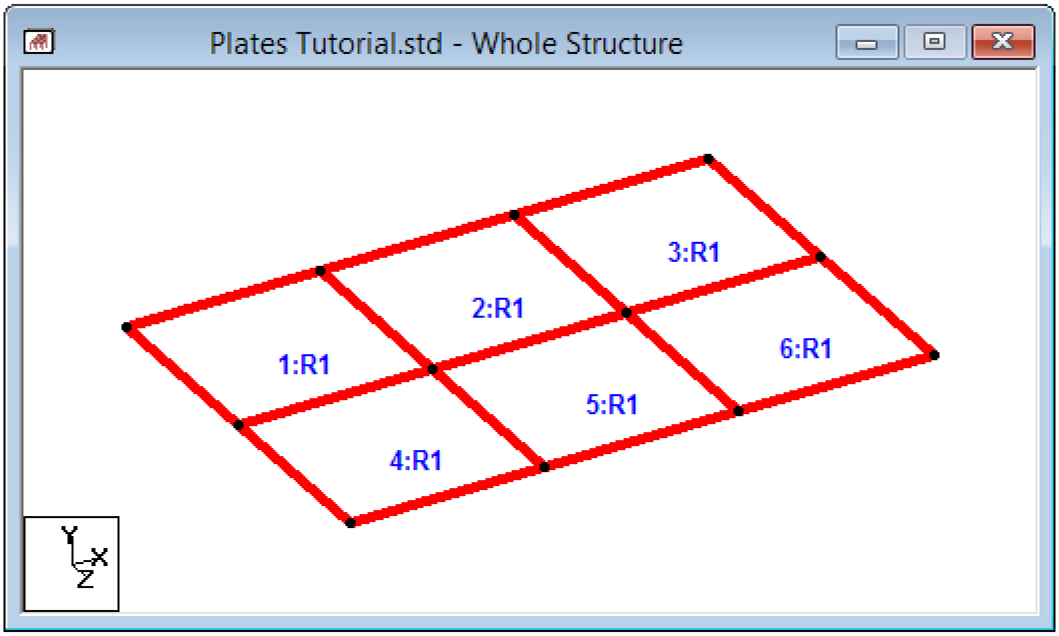
2. Select **Metric** as the **Input Units**.
3. Set the **Length** units to **Centimeter**.
   Leave the Force units as KiloNewton.
4. Click **Apply**.

T.3 Specifying Element Properties

Just as properties are assigned to members, properties must be assigned to plate elements too. The property required for plates is the plate thickness (or the thickness at each node of elements if the slab has a varying thickness).

The STAAD input file commands generated are:

```
ELEMENT PROPERTY
1 TO 6 THICKNESS 30
```



1. To select all plates, either:

   click-and-drag a window around all three plates using the **Plates Cursor** tool

   or

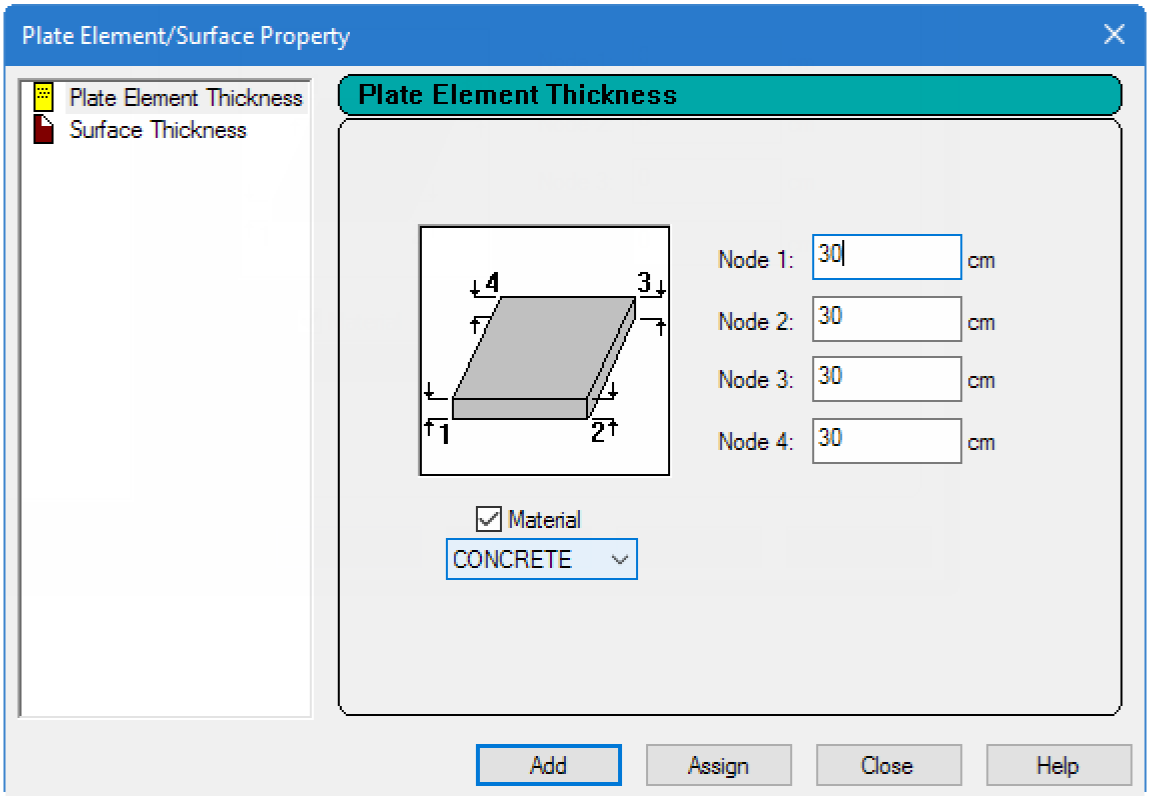
   press **<Ctrl+A>**

   or

   on the **Geometry** ribbon tab, select the **Plates Cursor** > **All Plates** tool in the **Selection** group.
2. On the **Specification** ribbon tab, select the **Plate Thickness** tool in the **Plate Profiles** group.

   The **Properties - Whole Structure** and the **Plate Element/ Surface Property** dialogs open.



3. Specify the material and thickness:
   a. Type `30` cm for the **Plate Element Thickness** at each corner.

   > **Tip:** Typing a value in the first field will automatically populate the remaining fields as a uniform thickness is typical.

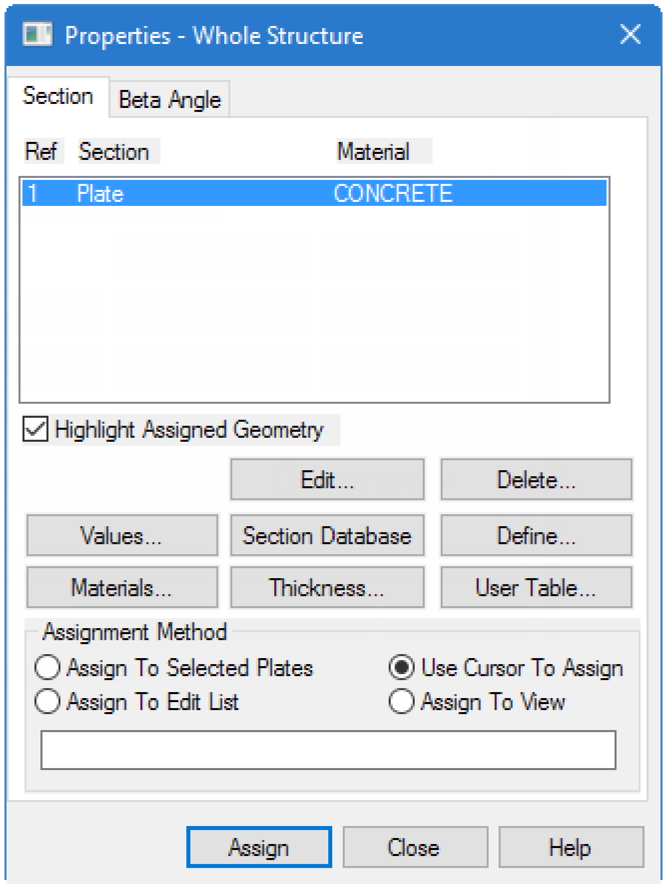
   b. Ensure that **Material** option is checked and that **CONCRETE** is selected.

   This instructs the program to assign the material properties of Concrete (E, Poisson, Density, Alpha, etc.) will be assigned along with the plate thickness.

   > **Tip:** The material property values so assigned will be the program defaults. You can review those values by clicking **Materials** in the **Properties - Whole Structure** dialog.

   c. Click **Assign**.
   d. Click **Close**.

The plate property is added to the **Properties - Whole Structure** dialog list and is assigned to the plate elements..



4. Click anywhere in the drawing area to deselect the selected entities.

> **Tip:** This is a best practice to follow when using the assignment feature. When an entity is highlighted, clicking on any Assign option is liable to cause an undesired attribute to be assigned to that entity.

T.3 Specifying Material Constants

When " T.3 Specifying Element Properties (on page 580) ", you used the **Material** option. Consequently, the material definition (E, Density, Poisson's Ratio, etc.) of concrete were assigned to the plates along with the properties

The STAAD input file commands generated are:

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2171.85
POISSON 0.17
DENSITY 2.35616e-005
ALPHA 1e-005
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 2.7579
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

Tip: However, if you had not assigned the material definition along with the thickness, you could use the **Materials** page in the **Analytical Modeling** worfklow page control to do so.

T.3 Specifying Supports

The slab is fixed-supported along the entire length of two adjacent sides. However, when modeled as plate elements, the supports can be specified only at the nodes along those edges, and not at any point between the nodes.

Tip: The finer the mesh (i.e., the larger the number of elements), then the more supported nodes you would be able to model. For this tutorial, the 2 meter mesh is used as a simple demonstration.

The STAAD input file commands generated are:

```
SUPPORTS
1 2 4 5 7 10 FIXED
```

Note: The node numbers *may be different* depending on the method of plate creation used. Regardless, the nodes should be those along the X and Z axis.

1. Press **<Shift+N>** to turn on the display of the **Node Numbers**.

   This will assist in the identification of the nodes used as supports.
2. On the **Geometry** ribbon tab, select the **Nodes Cursor** tool in the **Selection** group.

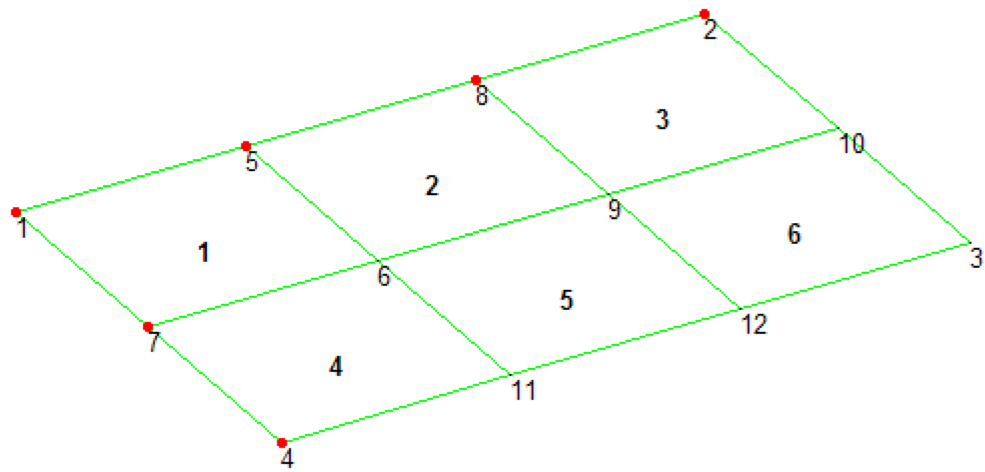
   

3. While pressing the **<Ctrl>** key, select the nodes on two adjacent edges of the plate group:

   These nodes are fixed supports in the model.

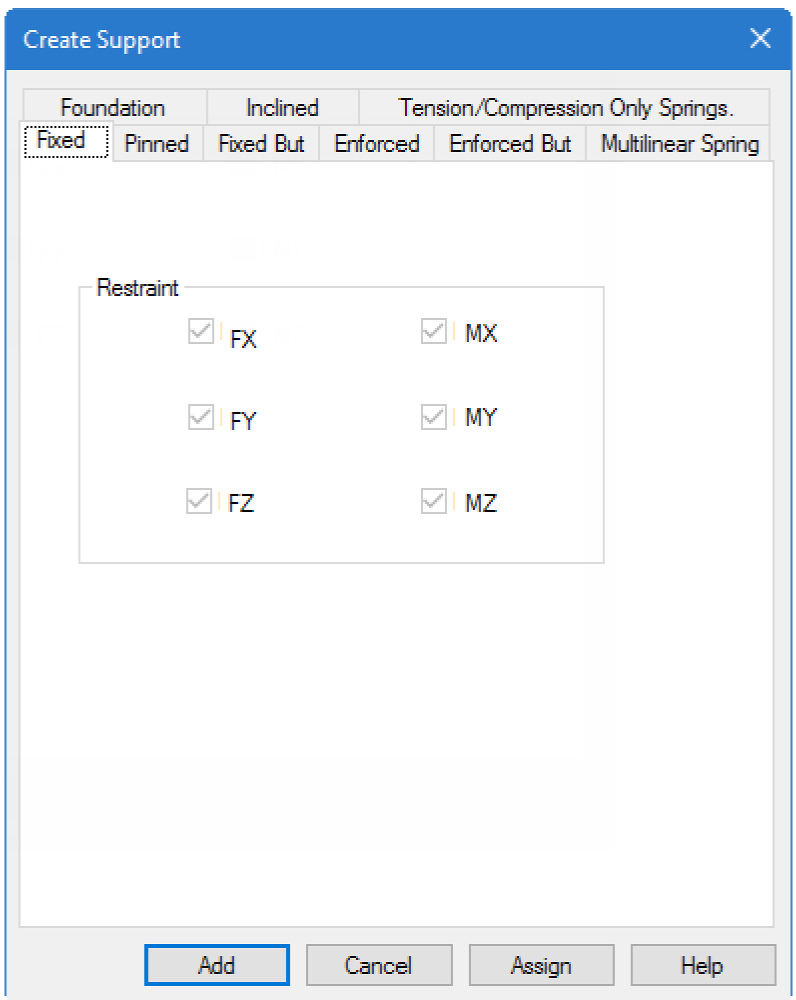
   For example, in the following figure these nodes are:

   1
   7
   4
   5
   8
   2



4. On the **Specification** ribbon tab, select the **Fixed** tool in the **Supports** group.

   The Create Support dialog opens with the **Fixed** tab selected.



5. Click **Assign**.

> **Note:** The **Assign** button is active because you selected the nodes previously.

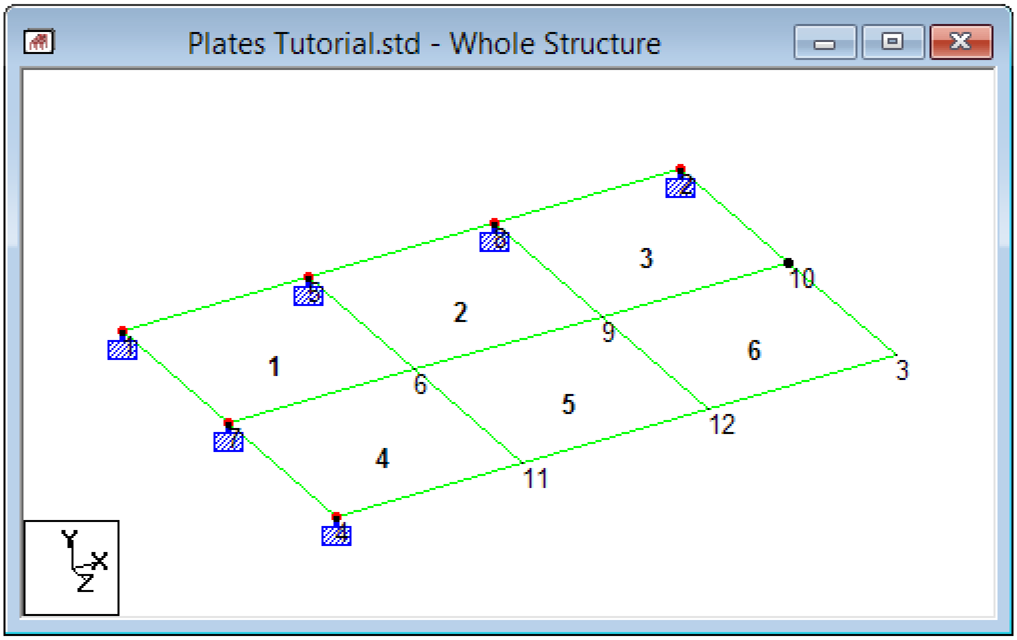


Figure 88: The structure with fixed supports along two adjacent edges

The fixed support type is also added to the **Supports - Whole Structure** dialog as S2.

T.3 Specifying Primary Load Cases

Three primary load cases are required for this structure.

The STAAD input file commands generated are:

```
UNIT METER KG
LOAD 1 DEAD LOAD
SELF Y -1.0
LOAD 2 EXTERNAL PRESSURE LOAD
ELEMENT LOAD
1 TO 6 PR GY -300
LOAD 3
TEMPERATURE LOAD
1 TO 6 TEMP 40 30
```

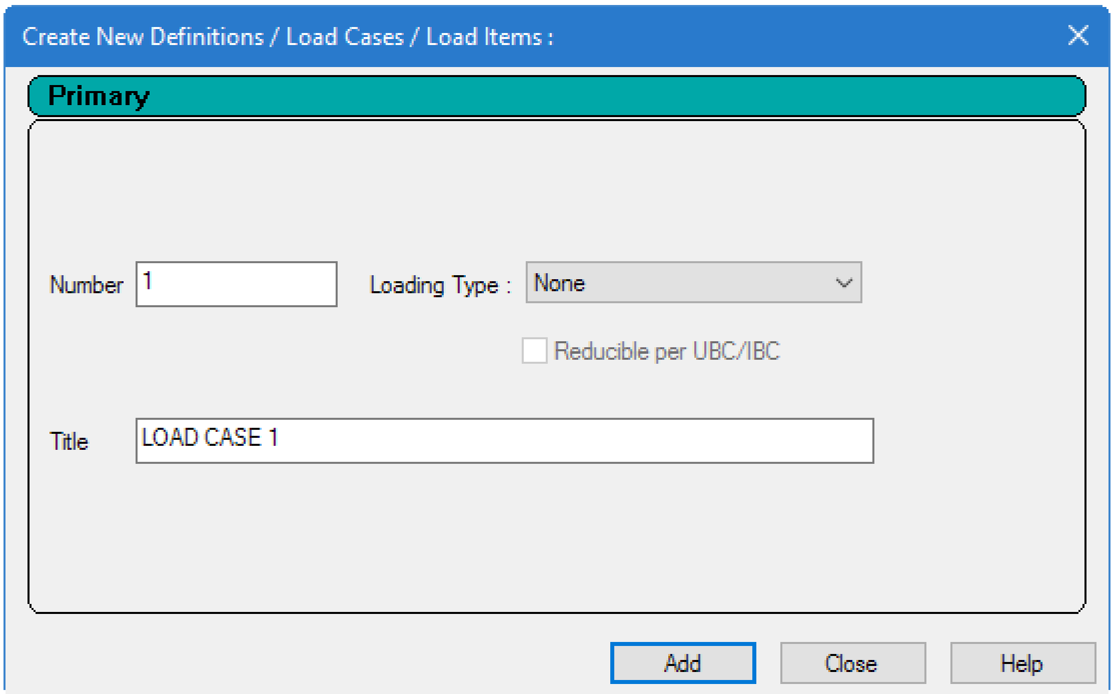
1. Change the units:

   > **Note:** The pressure load value listed in the beginning of this tutorial is in **KN** and **meter** units. Rather than convert that value to the current input units, this tutorial will conform to those units. The current input units, which you last set by specifying the thickness was centimeter. Therefore, you need to change the force unit to Kilogram and the length units to Meter.

   a. Click the drop-down arrow beside the current Input Units in the application window status bar.
      The **Set input Units** dialog opens.
   b. Select the length units as **Meter** and the force units as **Kilogram**.
   c. Click **Apply**.

2. Add the dead load case:
   a. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.

The **Add New Load Cases** dialog opens.



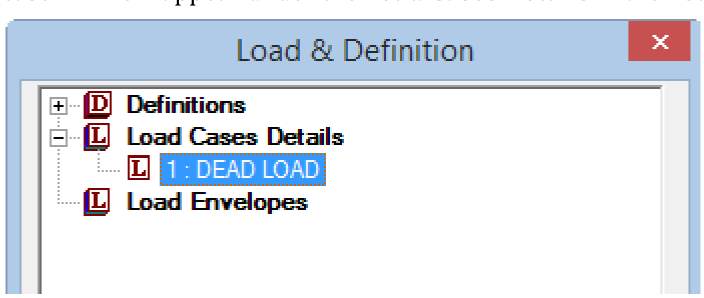
   b. Type `Dead Load` as the **Title** for Load Case 1.

   Leave the **Number** as the default (1) and leave the **Loading Type** as None.

   Note: The **Loading Type** list is used to associate the load case we are creating with any of the ACI, AISC, IBC, or other code-prescribed definitions of Dead, Live, Ice, etc. This type of association needs to be done if you intend to use the program's automatically generating load combinations in accordance with those codes. Note that there is a check box labeled **Reducible per UBC/IBC**. This feature is active only when the load case is assigned a Loading Type called **Live** when you create that load case.

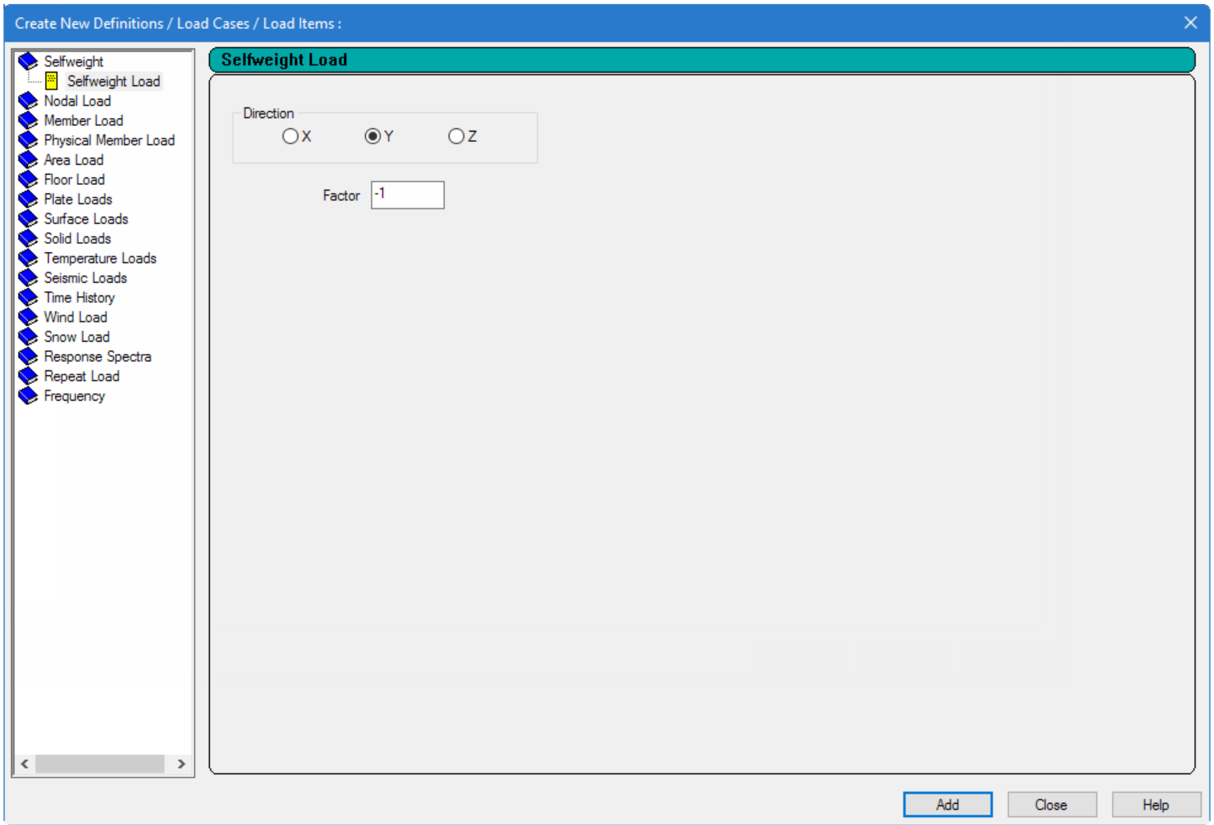
   c. Click **Add**.

The newly created load case will now appear under the **Load Cases Details** in the **Load & Definition** dialog.



3. Create the selfweight load:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

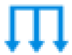
   The **Add New Load Items** dialog opens.



   b. Select the **Selfweight Load** option under the **Selfweight** item.
   c. Select the **Direction** as **Y**
   d. Type the **Factor** as `-1.0`.

   The negative number signifies that the selfweight load acts opposite to the positive direction of the global axis (Y in this case) along which it is applied.
   e. Click **Add**.
4. Assign the selfweight load to all of the elements:
   a. In the **Load & Definition** dialog, select **SELFWEIGHT Y -1**.
   b. Select the **Assign to View** option.
   c. Click **Assign**.

   A message dialog opens confirming you want to make this assignment.
   d. Click **Yes**.
5. Add the pressure load case:
   a. On the **Loading** ribbon tab, select the **Primary Load Case** tool in the **Loading Specifications** group.



The **Add New Load Cases** dialog opens.

b. Type `External Pressure Load` as the **Title**.

   Again, there is no need to associate the load case with any code based Loading Type so leave the selection as **None**.

c. Click **Add**.

6. On the **Loading** ribbon tab **Display** group, select **2: External Pressure Load** from the **Load** drop-down list.
7. Create the pressure load:

   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

   The **Add New Load Items** dialog opens.

   b. Select the **Pressure on Full Plate** option under the **Plate Loads** item.

   This type enables the load to be applied on the full area of the element.

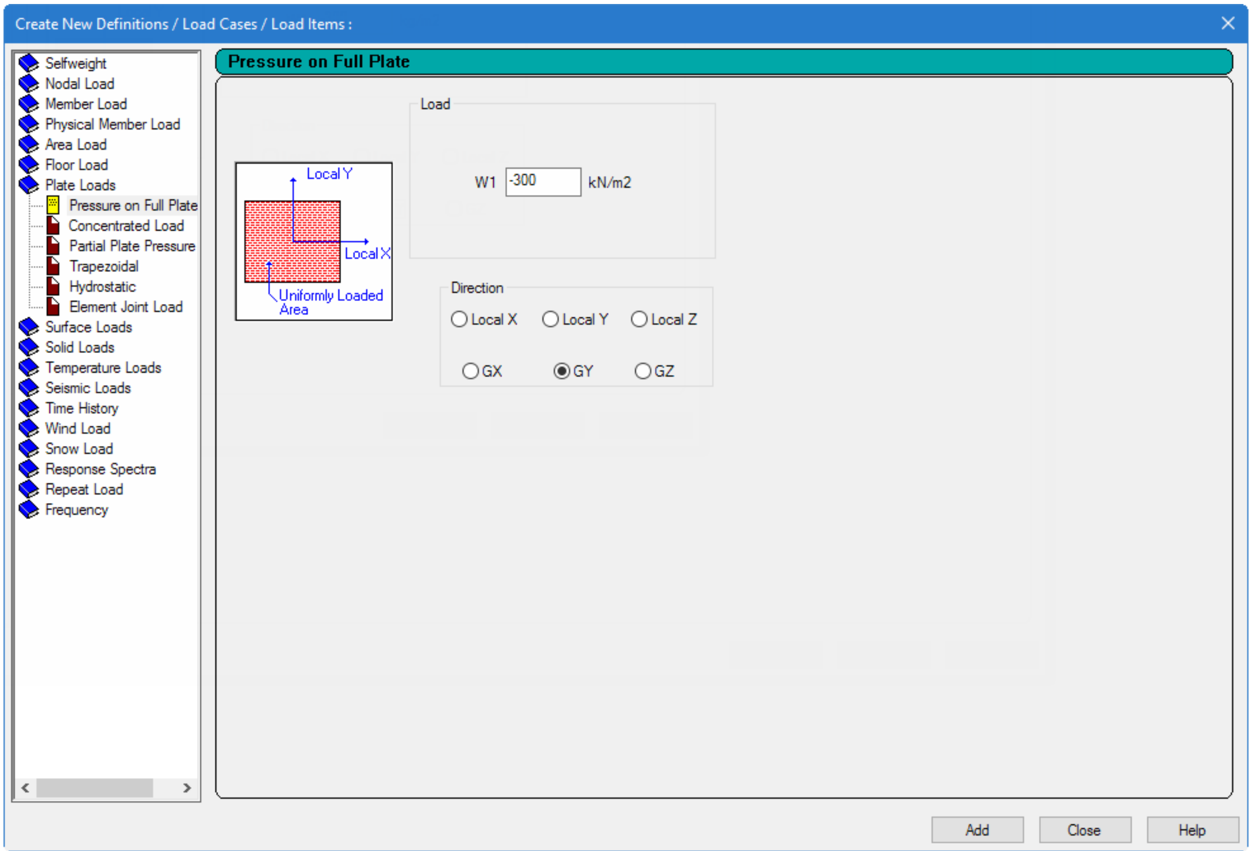
   > **Note:** The **Concentrated Load** is for applying a concentrated force on the element. The **Trapezoidal** and **Hydrostatic** options are for defining pressures with intensities varying from one point to another. The **Partial Plate Pressure Load** is useful if the load is to be applied as a “patch” on a small localized portion of an element.

   c. Type `-300` kg/m^2 in the **W1** field (force)

   d. Select **GY** as the **Direction** (global Y direction).

   e. Click **Add**.

   f. Click **Close**.



8. Repeat steps 5 and 6 to create a third load case titled `Temperature Load` and then select it on the **Loading** ribbon tab **Display** group.
9. To generate and assign the third load type:
   a. On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group.

      The **Add New Load Items** dialog opens.
   b. Select the **Temperature** option under the **Temperature Loads** item.
   c. Type `40` in the **Temperature Change for Axial Elongation**field.
   d. Type `30` in the **Temperature Differential from Top to Bottom** field.

      Leave the **Temperature Differential from Side to Side (Local Z)** field as 0 (default).
   e. Click **Add** and then click **Close**.

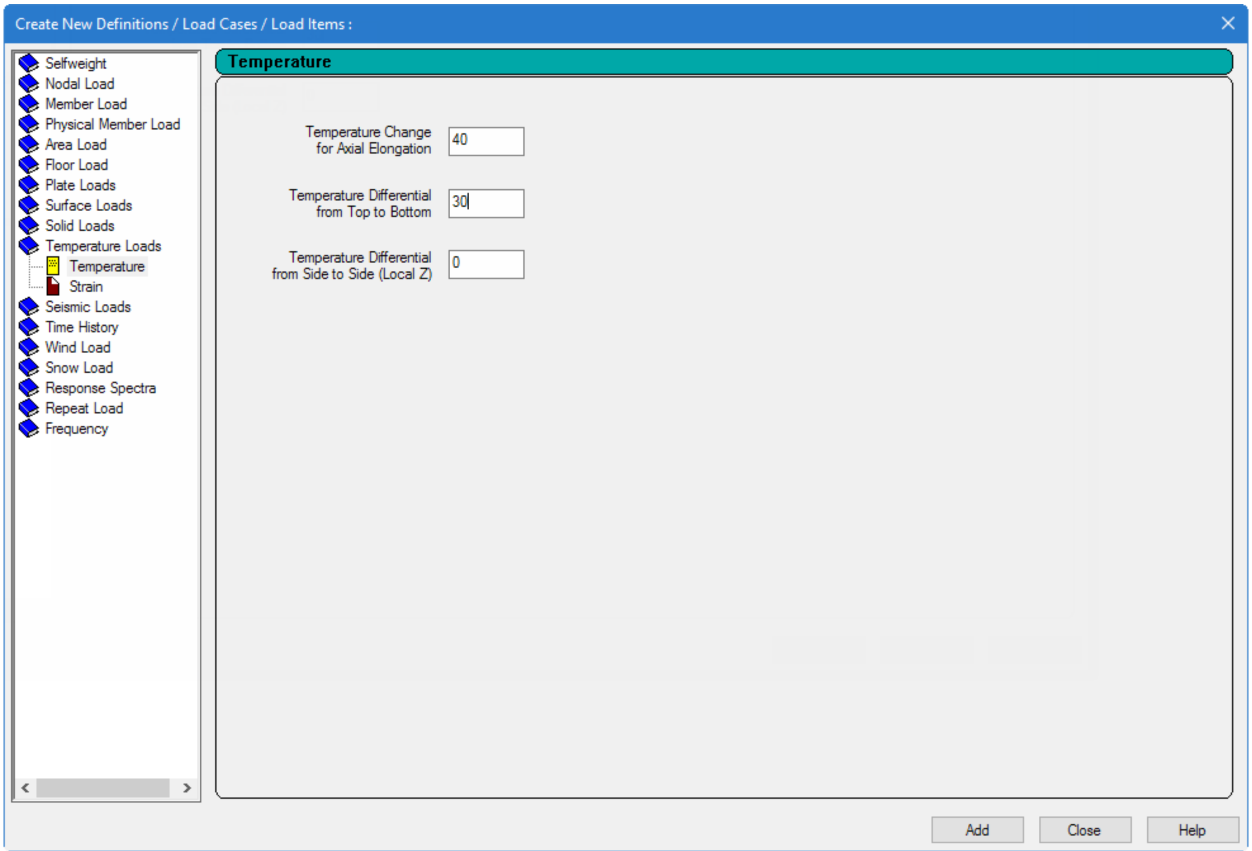


Figure 89:

10. To apply the pressure load and temperature load on all the plates, repeat step 4 for each load item.

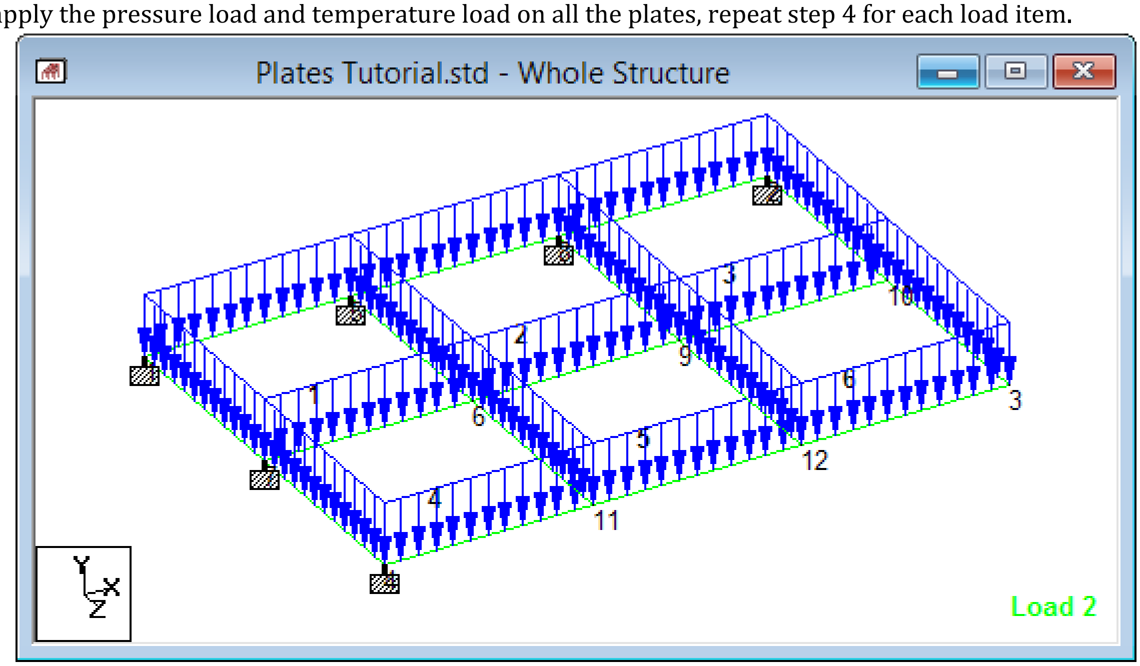


Figure 90: The pressure load applied to all plate elements

T.3 Analysis and Design

The following procedures describe how to add analysis and design commands to the STAAD input file.

Note: The analysis design commands are added using either the analytical user interface or the STAAD.Pro Editor. The STAAD.Pro Physical Modeler interface is not used to assign analysis and design commands.

T.3 Creating load combinations

This tutorial requires creating two combination cases.

The STAAD input file commands generated are:

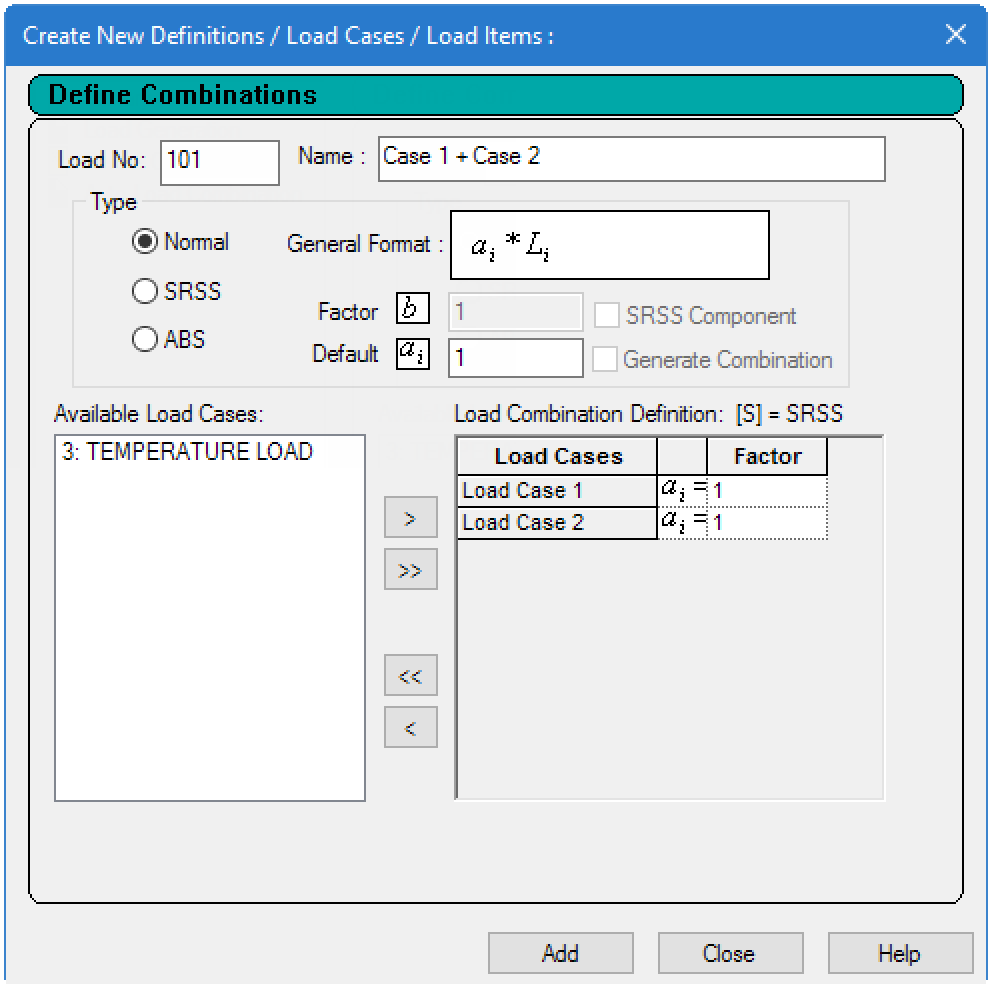
```
LOAD COMBINATION 101 CASE 1 + CASE 2
1 1.0 2 1.0
LOAD COMBINATION 102 CASE 1 + CASE 3
1 1.0 3 1.0
```

1. Define load case 4 as a load combination:
   a. On the **Loading** ribbon tab, select the **Combination Load Case** tool in the **Loading Specifications** group.

      The **Create New Load Combinations** dialog opens.
   b. Select the **Define Combinations** option from the left-hand side.
   c. Type `101` in the **Load No:** field.
   d. Type `Case 1 + Case 2` in the **Title**field.

   Leave the **Type** option as **Normal** and the **Default a1** as `1`. This uses the algebraic sum of the results for the individual load cases in the load combination. The individual results will have a multiplication factor of 1.



2. Select the load cases for the load combination:
   a. Select **1: DEAD LOAD** from Available Load List and then click **[>]**.
   b. Select **2: EXTERNAL PRESSURE LOAD** from Available Load List and then click **[>]**.
   Load cases 1 and 2 will appear in the Load Combination Definition list.
   c. Click **Add**.

   Case 101 is created.
3. To define load case 5 as a load combination, repeat step 1 but type `102` in the **Load No:** field and type `Case 1 + Case 3` in **Title**field.
4. Repeat step 2 except for selecting load cases 1 and 3 instead of cases 1 and 2.

   Tip: The load cases and combination methods used for a load combination can be changed by selecting a load combination and then clicking **Edit.**

   Load 102 is created.
5. Click **Close.**

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.3 Specifying the analysis type

A linear, static analysis is required for this model. You can also instruct STAAD.Pro to provide a static equilibrium report.

The STAAD input file commands generated are:

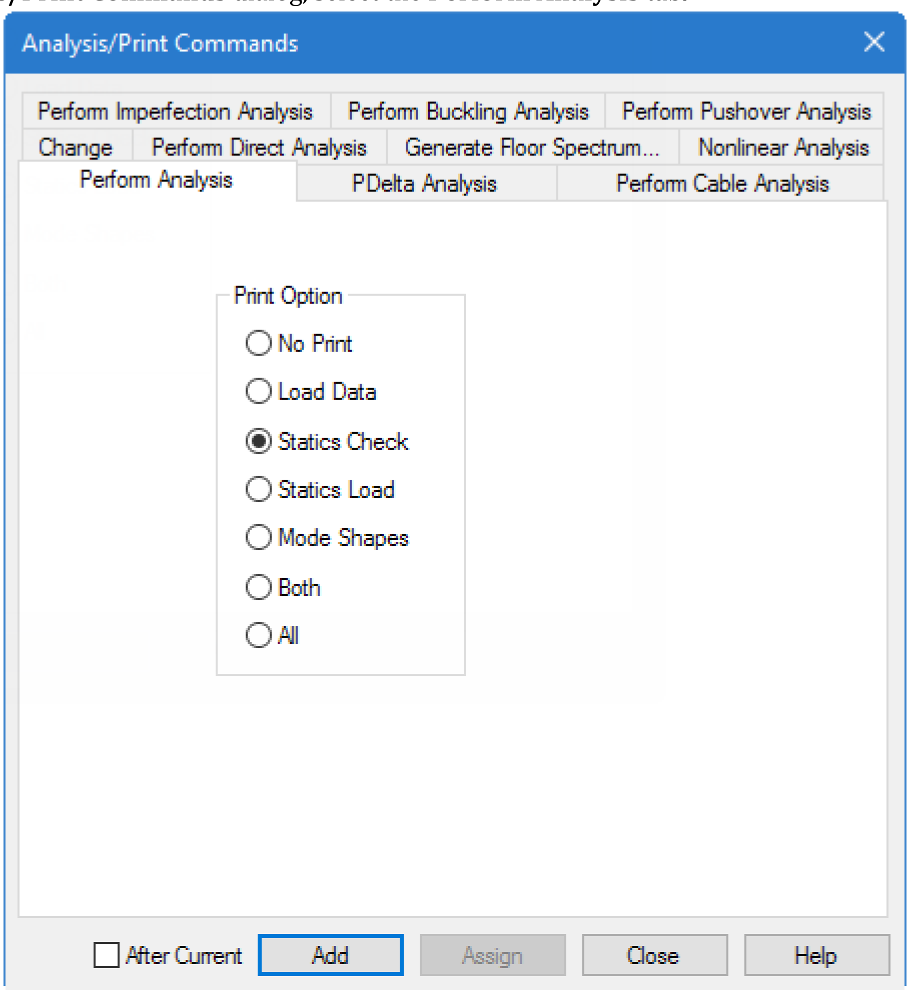
```
PERFORM ANALYSIS PRINT STATICS CHECK
```

1. On the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group.

   The **Analysis/Print Commands** dialog opens.

   Tip: If the **Analysis/Print Commands** dialog does not open automatically, then click **Define Commands** in the **Analysis - Whole Structure** dialog.

2. In the **Analysis/Print Commands** dialog, select the **Perform Analysis** tab.

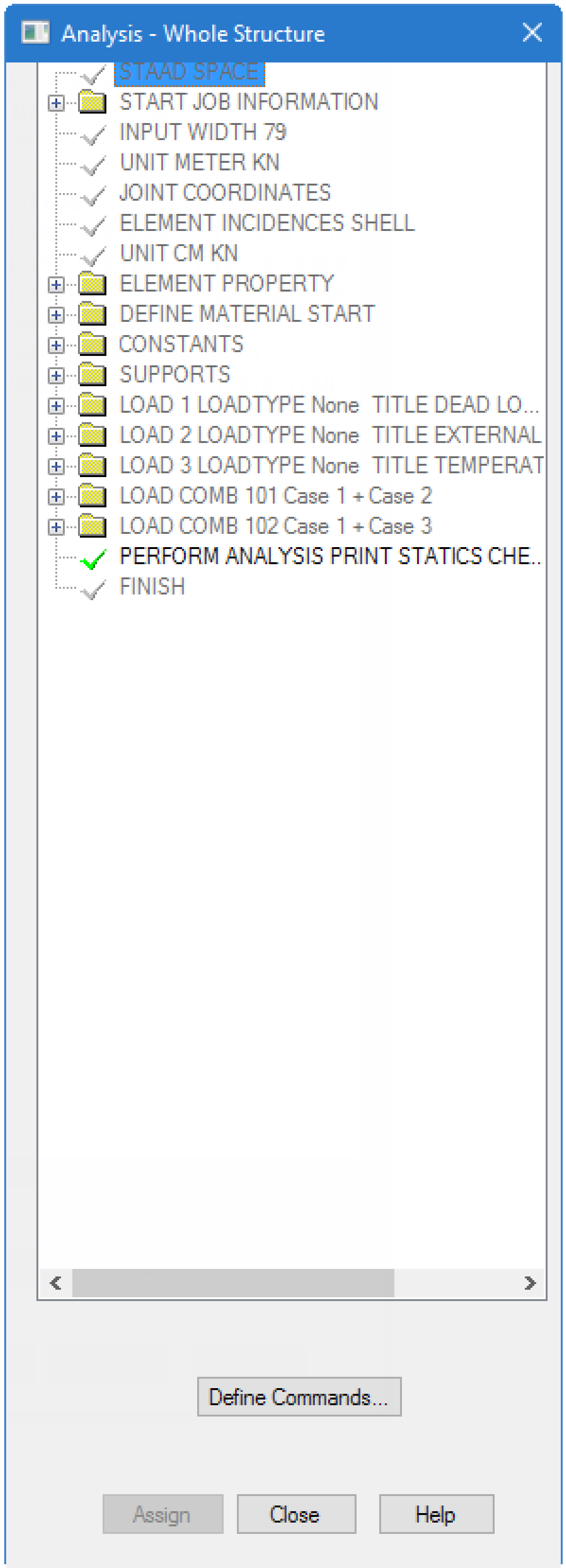


3. Select the **Statics Check** print option.

 Note: In response to this option, a report consisting of the summary of applied loading and summary of support reactions, for each load case, will be produced in the STAAD output file.

4. Click **Add**
5. Click **Close**.

 The **Analysis** command is added to the **Analysis - Whole Structure** dialog.


Analysis - Whole Structure
STAAD SPACE
START JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
ELEMENT INCIDENCES SHELL
UNIT CM KN
ELEMENT PROPERTY
DEFINE MATERIAL START
CONSTANTS
SUPPORTS
LOAD 1 LOADTYPE None TITLE DEAD LO...
LOAD 2 LOADTYPE None TITLE EXTERNAL
LOAD 3 LOADTYPE None TITLE TEMPERAT
LOAD COMB 101 Case 1 + Case 2
LOAD COMB 102 Case 1 + Case 3
PERFORM ANALYSIS PRINT STATICS CHE..
FINISH
Define Commands...
Assign
Close
Help

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.3 Specifying post-analysis print commands

Two types of element results can be requested:

- **a.** element stresses at the centroid or any point on the element surface which consist of stresses and moments per unit width, as explained in G.5.1 Plate and Shell Elements (on page 2099) and TR.42 Print Specifications (on page 2666).
- **b.** the element forces at the nodes which consist of the 3 forces and 3 moments at each node of the elements in the global axis system (refer to TR.42 Print Specifications (on page 2666) for details).

You will add both types of these results for this tutorial.

You will also need to set the units in which these results are printed to KN and Meter for element stresses and Kg and Meter for element forces.

The STAAD input file commands generated are:

```
UNIT METER KN
PRINT ELEMENT STRESSES LIST 3
UNIT KG METER
PRINT ELEMENT FORCE LIST 6
```

These results will be written in the STAAD output file and can be viewed using the procedure explained in " T.3 Viewing the output file (on page 605) ".

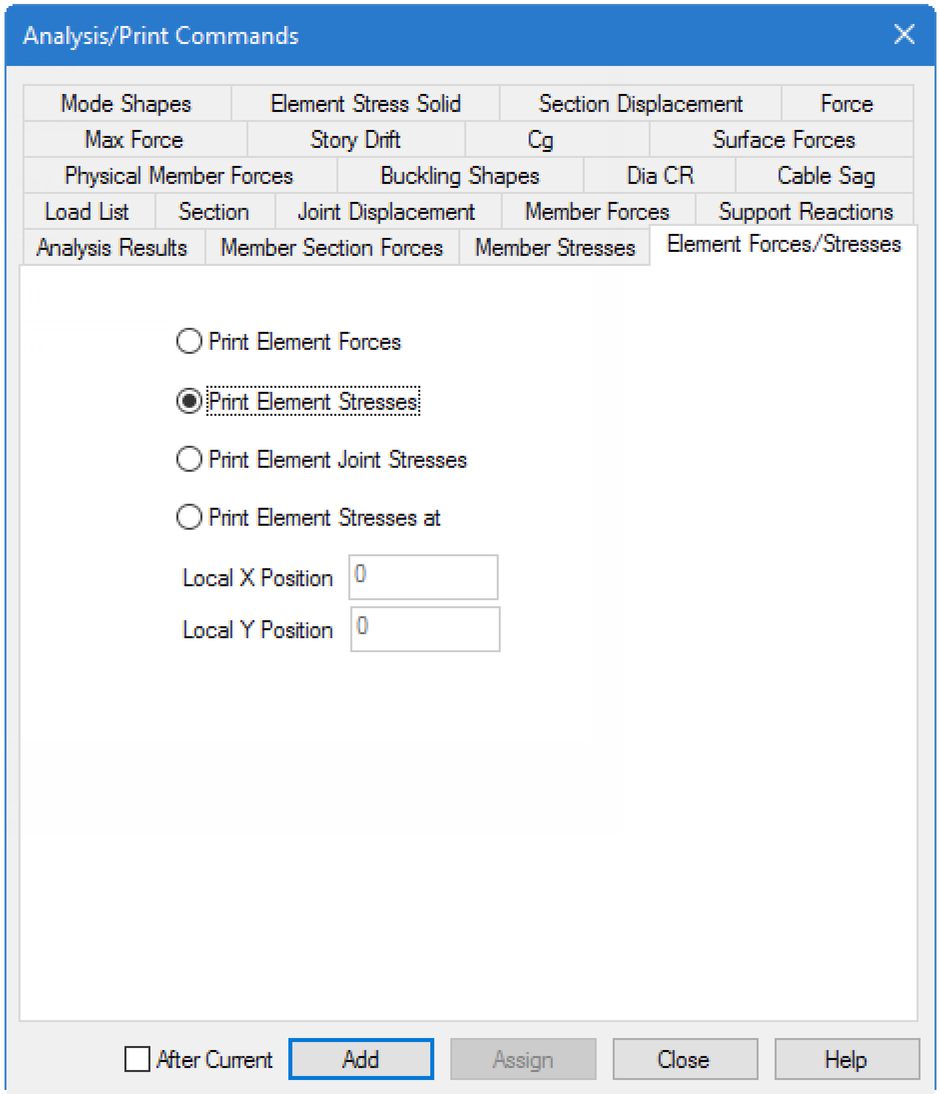
1. Set the length and force units to **Meter** and **Kilonewton** respectively.

   Refer to T.3 Changing the input units of length (on page 580) for additional information on this procedure.
2. On the **Analysis and Design** ribbon tab, select the **Post-Analysis Commands** tool in the **Analysis Data** group.

   The **Post Analysis Print - Whole Structure** dialog opens.
3. On the **Post Analysis Print - Whole Structure** dialog, click **Define Commands**.

   The **Analysis/Print Commands** dialog opens.



4. Select the **Element Forces/Stress** tab.
5. Select the **Print Element Stresses** option.
6. Click **Add** and then click **Close**.
7. Set the length and force units to **Meter**and **Kilogram** respectively.
8. Repeat steps 3 through 6 except select the **Print Element Forces** option in step 5.

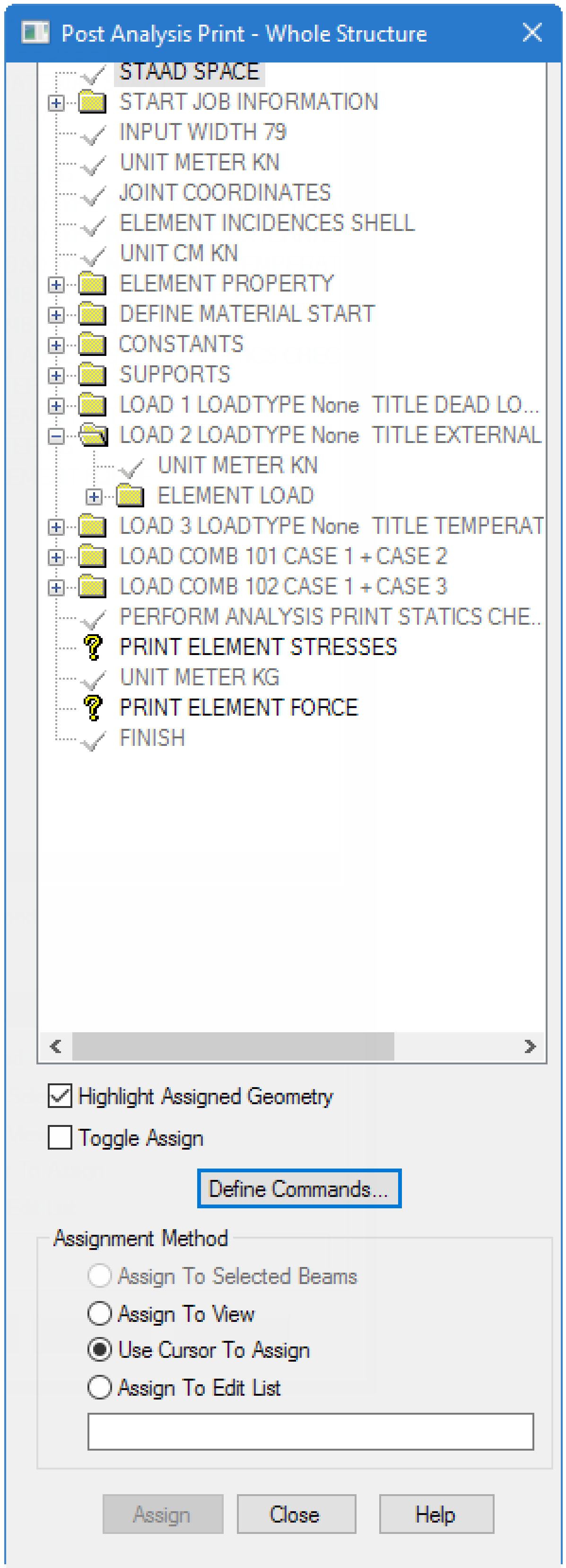


Figure 91: The dialog with unassigned commands

9. Associate the print element stresses with element 3:
   a. Select the **PRINT ELEMENT STRESSES** command in the **Post Analysis Print - Whole Structure** dialog commands list.
   b. On the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

   c. In the view window, select element no. 3.

   When you select the plate, the **Assignment Method** automatically becomes **Assign to Selected Plates**.
   d. Click **Assign**.
10. To associate the **PRINT ELEMENT FORCE** command with element 6, use a similar procedure as step 9 except for selecting element no. 6 in the place of element no. 3.

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

T.3 Viewing the input command file

You can inspect the text data file created during this tutorial.

1. On the **Utilities** ribbon tab, select the **Command File** tool in the **Edit** group.

   The STAAD.Pro Editor opens with the contents of the input file.

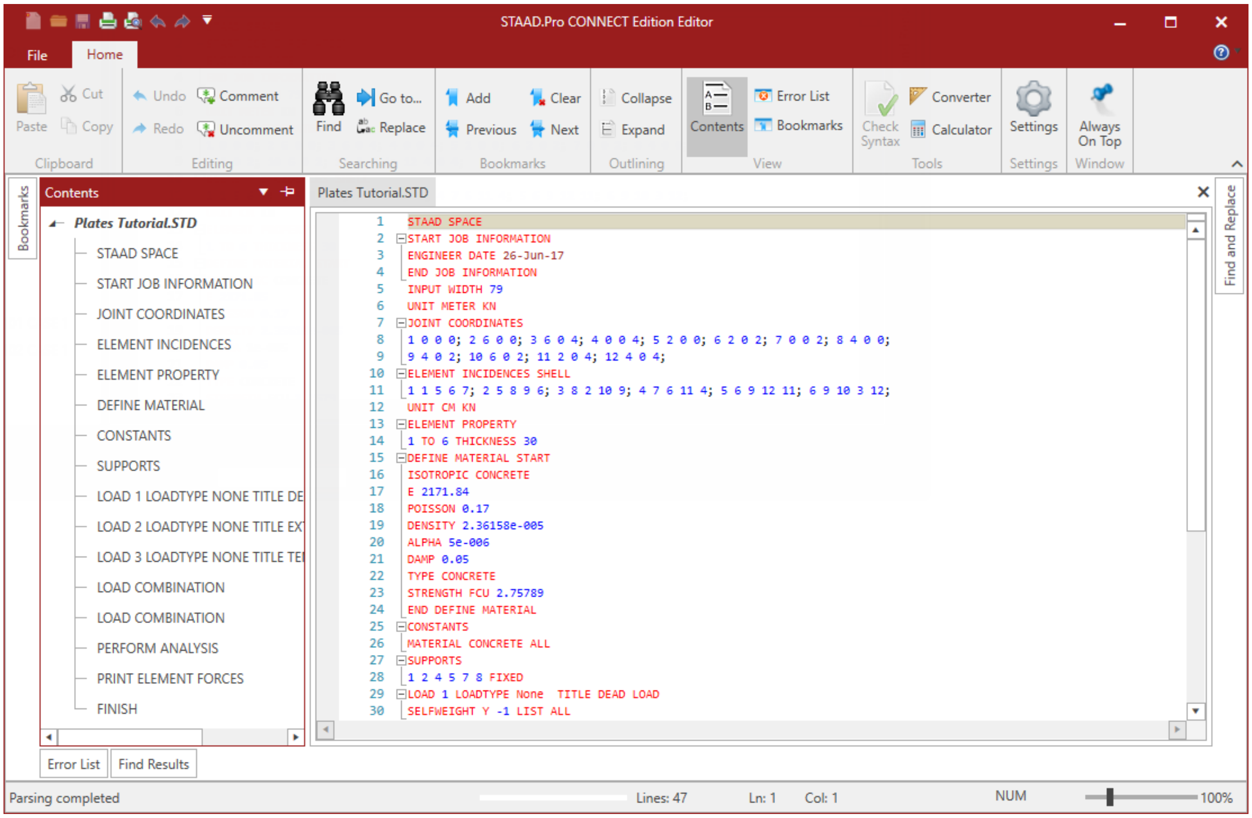


Figure 92: The STAAD.Pro Editor window

2. (Optional) You modify the data of the structure in this editor if necessary.
3. Select **File** > **Exit Editor** in the STAAD.Pro Editor window to close.

As stated in T.3 Methods of creating the model (on page 550), you could also have created the same model by typing the relevant STAAD commands into a text file using the STAAD.Pro Editor. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to T.3 Performing the analysis and design (on page 604) where you will perform the analysis and design on this model.

T.3 Creating the model using the command file

As an alternatively to the procedures described in the proceeding tutorial, you can also create the same STAAD input file using the STAAD.Pro Editor.

To start a new STAAD input file using the STAAD.Pro Editor, follow the procedure described in T.3 Creating a new structure (on page 552). Then select the **Command File** tool in the **Edit** group on the **Utilities** ribbon tab. The STAAD.Pro Editor window opens with the basic commands for your model entered.

Note: A STAAD input file is a plain text file that uses the `.std` file extension. Therefore any standard text editor such as Notepad can also be used to create the command file. However, the STAAD.Pro Editor offers the advantage of syntax checking as you type the commands. The STAAD command syntax are highlighted by command, keyword, value, etc.

For this tutorial, delete all the command lines displayed in the editor window and type the lines shown below. While not necessary, this will allow you to learn more about the required and optional command lines for an input file.

STAAD commands are *not* case sensitive (i.e., they may be typed in upper *or* lower case letters). By convention, this and most input files use all caps, though.

For most all commands and keywords, the first three letters of a keyword are all that are needed. The rest of the letters of the word are not required, but are useful to present a user-friendly command language in mostly plain English for later reference. By convention, the required letters in a command or keyword are underlined here (“PLANE” = “PLA” = “plane” = “pla”).

```
STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES
```

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNIT METER KN
```

Specifies the unit to be used for data to follow.

```
JOINT COORDINATES
1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4
```

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables you to provide multiple sets of data on one line. For example, node 6 has (X, Y, Z) coordinates of (4, 0, 2).

```
ELEMENT INCIDENCES SHELL
1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6 ; 4 4 3 9 10 ;
5 3 6 11 9 ; 6 6 8 12 11
```

The incidences of elements are defined above. For example, element 3 is defined as connected between the nodes 5, 7, 8 and 6.

```
UNIT CM KN
ELEMENT PROPERTY
1 TO 6 THICKNESS 30
```

The length unit is changed from meter to centimeter. Element properties are then provided by specifying that the elements are 30 cm thick.

```
UNIT KN  METER
CONSTANTS
E 2.17185e+007 ALL
POISSON 0.17 ALL
DENSITY 23.5616 ALL
ALPHA 1e-005 ALL
```

Material constants, which are E (modulus of elasticity), Density, Poisson's Ratio and Alpha, are specified following the command CONSTANTS. Prior to this, the input units are changed to Meter and KN.

```
SUPPORTS
1 2 4 5 7 10 FIXED
```

Joints 1, 2, 4, 5, 7 and 10 are defined as fixed supported. This will cause all 6 degrees of freedom at these nodes to be restrained.

```
UNIT KG
LOAD 1 DEAD LOAD
```

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is then initiated along with an accompanying title.

```
SELFWEIGHT Y -1
```

Load case 1 consists of selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```
LOAD 2 EXTERNAL PRESSURE LOAD
```

Load case 2 is initiated along with an accompanying title.

```
ELEMENT LOAD
1 TO 6 PR GY -300
```

Load 2 is a pressure load on the elements. A uniform pressure of 300Kg/m2 is applied on all the elements. GY indicates that the load is in the global Y direction. The negative sign (-300) indicates that the load acts opposite to the positive direction of global Y.

```
LOAD 3 TEMPERATURE LOAD
```

Load case 3 is initiated along with an accompanying title.

```
TEMPERATURE LOAD
1 TO 6 TEMP 40 30
```

Load 3 is a temperature load. All the 6 elements are subjected to a in-plane temperature increase of 40 degrees and a temperature variation across the thickness of 30 degrees. This increase is in the same temperature units as the Alpha value specified earlier under CONSTANTS.

```
LOAD COMB 101 CASE 1 + CASE 2
1 1.0 2 1.0
```

Load combination 101 is initiated along with an accompanying title. Load cases 1 and 2 are individually factored by a value of 1.0, and the factored values are combined algebraically.

```
LOAD COMB 102 CASE 1 + CASE 3
1 1.0 3 1.0
```

Load combination 102 is initiated along with an accompanying title. Load cases 1 and 3 are individually factored by a value of 1.0, and the factored values are combined algebraically.

```
PERFORM ANALYSIS PRINT STATICS CHECK
```

The above command instructs the program to proceed with the analysis. A static equilibrium report is also requested with the help of the words PRINT STATICS CHECK.

```
UNIT METER KN
PRINT ELEMENT STRESS LIST 3
```

The stresses and unit width moments are requested at the centroid of element 3 in KN and Meter units.

```
UNIT KG
METER
PRINT ELEMENT FORCE LIST 6
```

The forces and moments for all 6 d.o.f at the corner nodes of element 6 are requested in KG and Meter units.

```
FINISH
```

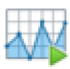
This command terminates the STAAD run.

Save the input file and close the editor. The model is opened in the STAAD.Pro interface.

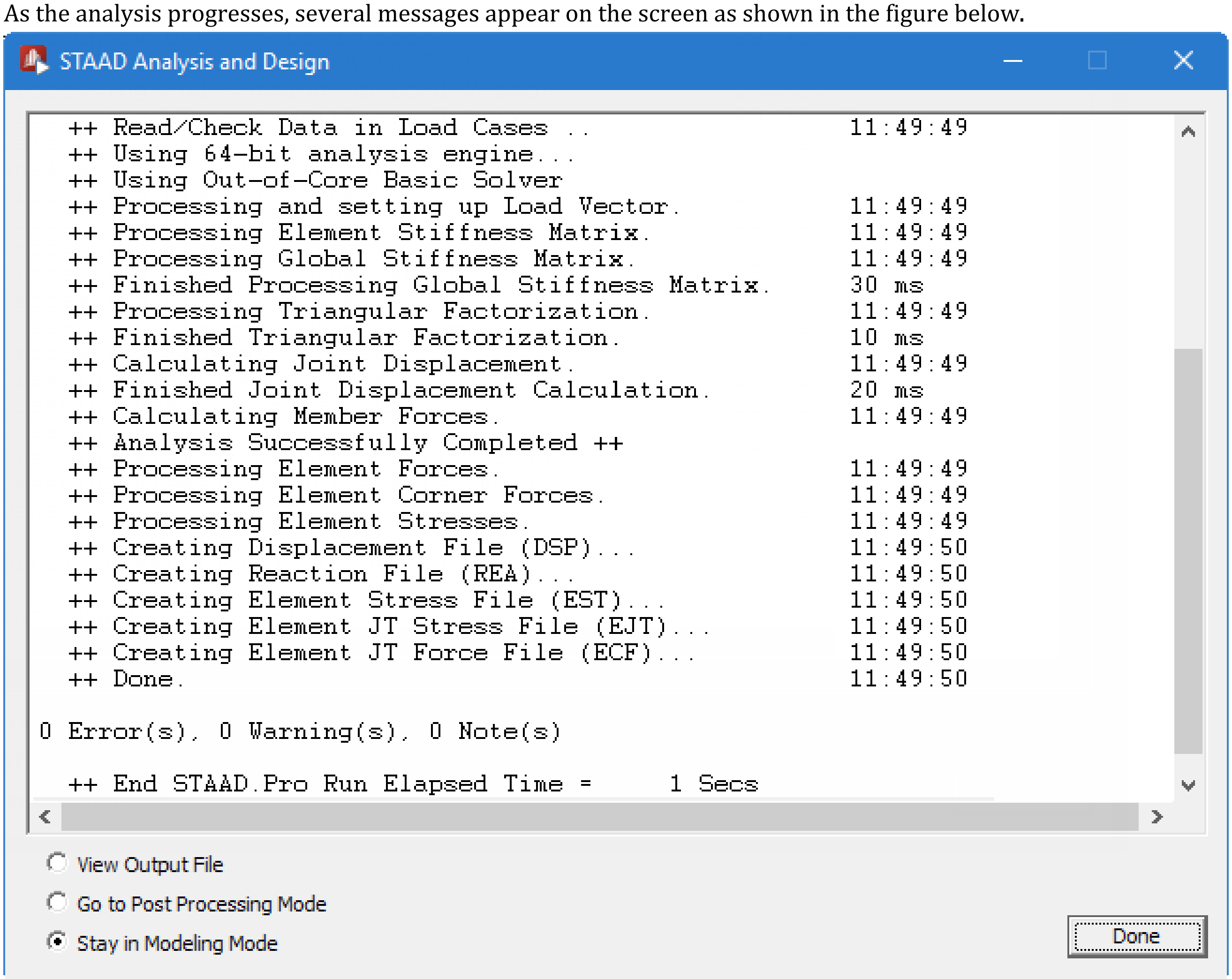
T.3 Performing the analysis and design

Tip: Remember to save your work by either click **Save** on the **File** ribbon tab , the **Save** tool, or pressing **<CTRL +S>**.

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group. .



As the analysis progresses, several messages appear on the screen as shown in the figure below.



2. Select the **View Output File** option once the analysis and design are complete.

   The three options are indicative of what will happen after you click **Done**.

View Output File	This option opens the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands specified during the model generation process. It also provides you with important messages of any errors were encountered, and if so, whether the analysis and design was successfully completed or not. See T.1 Viewing the output file (on page 447) for details on viewing and understanding the contents of the output file.
Go to Post Processing Mode	This option opens the graphical Post-processor mode, which can be used to extensively review and verify the results. This mode allows you to view the results graphically, plot result diagrams, produce reports, etc. See T.1 Post-Processing (on page 455) for details on the Post processing mode.
Stay in Modeling Mode	This option closes the dialog and remains in the Model generation mode of the program, where you initiated the analysis. This is useful if you want to make further changes to the input file.

3. Click **Done**.
   The STAAD.Pro Output Viewer window opens.

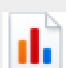
T.3 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly.

For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

Note:

If you did not select to open the output file after running the analysis in the previous procedure, you can open it any time through the user interface. On the **Utilities** ribbon tab, select the **Analysis Output** tool in the **View** group.



Tip: By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the Output file. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Set Echo** tool option from the menu bar and the select the **Echo Off** option in the **Set Echo** dialog.

It is *strongly recommended* that you review the entire output file to ensure that the results are reasonable and that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the post-processing mode. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
                                                        PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
```

```
            *           Proprietary Program of                   *
            *           Bentley Systems, Inc.                    *
            *           Date=    MAR 24, 2022                    *
            *           Time=     9:44: 2                        *
            *                                                    *
            *  Licensed to: Bentley Systems Inc                  *
            ****************************************************
     1. STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES
INPUT FILE: Tutorial 3 - Analysis of a Slab.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 21-AUG-19
     4. END JOB INFORMATION
     5. UNIT METER KN
     6. JOINT COORDINATES
     7. 1 0 0 0; 2 2 0 0; 3 2 0 2; 4 0 0 2; 5 4 0 0; 6 4 0 2; 7 6 0 0; 8 6 0 2
     8. 9 2 0 4; 10 0 0 4; 11 4 0 4; 12 6 0 4
     9. ELEMENT INCIDENCES SHELL
    10. 1 1 2 3 4; 2 2 5 6 3; 3 5 7 8 6; 4 4 3 9 10; 5 3 6 11 9; 6 6 8 12 11
    11. UNIT CM KN
    12. ELEMENT PROPERTY
    13. 1 TO 6 THICKNESS 30
    14. UNIT KN METER
    15. CONSTANTS
    16. E 2.17185E+007 ALL
    17. POISSON 0.17 ALL
    18. DENSITY 23.5616 ALL
    19. ALPHA 1E-005 ALL
    20. SUPPORTS
    21. 1 2 4 5 7 10 FIXED
    22. UNIT METER KG
    23. LOAD 1 DEAD LOAD
    24. SELFWEIGHT Y -1
    25. LOAD 2 EXTERNAL PRESSURE LOAD
    26. ELEMENT LOAD
    27. 1 TO 6 PR GY -300
    28. LOAD 3 TEMPERATURE LOAD
    29. TEMPERATURE LOAD
    30. 1 TO 6 TEMP 40 30
    31. LOAD COMB 101 CASE 1 + CASE 2
    32. 1 1.0 2 1.0
    33. LOAD COMB 102 CASE 1 + CASE 3
    34. 1 1.0 3 1.0
    35. PERFORM ANALYSIS PRINT STATICS CHECK
      SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         12  NUMBER OF MEMBERS       0
     NUMBER OF PLATES          6  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =      36
   TOTAL LOAD COMBINATION  CASES =     2  SO FAR.
      SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    3
   *** NOTE: CAPACITY FOR MAXIMUM #    256 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
          DEAD LOAD
           CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
```

```
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.299999999E+01
                       Y =  0.000000000E+00
                       Z =  0.199999999E+01
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =      -17298.83
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       34597.65  MY=           0.00  MZ=      -51896.48
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       17298.83
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      -34597.65  MY=           0.00  MZ=       51896.48
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      1)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y = -3.20664E-01      12
     Z =  0.00000E+00       0
     RX=  9.80376E-04      12
     RY=  0.00000E+00       0
     RZ= -6.49326E-04       9
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
         EXTERNAL PRESSURE LOAD
          CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.299999999E+01
                       Y =  0.000000000E+00
                       Z =  0.199999999E+01
     SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    4
TOTAL APPLIED LOAD     2
  ***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       -7200.00
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       14400.00  MY=           0.00  MZ=      -21600.00
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =        7200.00
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      -14400.00  MY=           0.00  MZ=       21600.00
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      2)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y = -1.33465E-01      12
     Z =  0.00000E+00       0
     RX=  4.08045E-04      12
     RY=  0.00000E+00       0
     RZ= -2.70258E-04       9
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     3
         TEMPERATURE LOAD
```

```
TOTAL APPLIED LOAD     3
  ***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  1.0313497E-10
      SUMMATION FORCE-Y = -2.2363831E-28
      SUMMATION FORCE-Z =  2.5783741E-10
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX= -7.5974110E-12  MY=  3.3531239E-10  MZ= -1.0471166E-12
TOTAL REACTION LOAD    3
  ***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  6.1880980E-10
      SUMMATION FORCE-Y =  4.0287096E-12
      SUMMATION FORCE-Z = -6.7037728E-10
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX= -1.1002438E-10  MY=  5.1973420E-09  MZ=  2.1611931E-10
     SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    5
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      3)
           MAXIMUMS    AT NODE
     X =  2.01178E-01      12
     Y =  8.97376E-01      12
     Z =  1.66238E-01      11
     RX= -3.51267E-03      12
     RY= -2.41811E-04      11
     RZ=  2.62397E-03      12
  ************ END OF DATA FROM INTERNAL STORAGE ************
   36. UNIT METER KN
   37. PRINT ELEMENT STRESSES LIST 3
 ELEMENT  STRESSES LIST     3
     SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    6
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
  ----------------
         STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
    3      1       -18.13       72.86       -3.96      -20.42       -3.35
                  1308.65     1308.65        0.00        0.00        0.00
                  1404.84     1404.84
      TOP : SMAX=    -220.35 SMIN=   -1404.84 TMAX=     592.24 ANGLE=-11.1
      BOTT: SMAX=    1404.84 SMIN=     220.35 TMAX=     592.24 ANGLE= 78.9
           2        -7.54       30.33       -1.65       -8.50       -1.39
                   544.68      544.68        0.00        0.00        0.00
                   584.71      584.71
      TOP : SMAX=     -91.71 SMIN=    -584.71 TMAX=     246.50 ANGLE=-11.1
      BOTT: SMAX=     584.71 SMIN=      91.71 TMAX=     246.50 ANGLE= 78.9
           3        96.73      -59.42      -30.45      -14.83       18.43
                 10779.75     5300.97    -5044.91    -2309.55     3890.06
                 10912.06     5585.64
      TOP : SMAX=     269.70 SMIN=  -10642.36 TMAX=    5456.03 ANGLE= 55.1
      BOTT: SMAX=     624.69 SMIN=   -4960.95 TMAX=    2792.82 ANGLE= 53.8
        101        -25.67      103.19       -5.61      -28.92       -4.74
                  1853.33     1853.33        0.00        0.00        0.00
                  1989.55     1989.55
      TOP : SMAX=    -312.06 SMIN=   -1989.55 TMAX=     838.74 ANGLE=-11.1
      BOTT: SMAX=    1989.55 SMIN=     312.06 TMAX=     838.74 ANGLE= 78.9
        102         78.61       13.44      -34.41      -35.24       15.08
                 10643.05     5713.20    -5044.91    -2309.55     3890.06
                 11074.60     6408.66
      TOP : SMAX=    -923.25 SMIN=  -11074.60 TMAX=    5075.67 ANGLE= 52.7
```

```
      BOTT: SMAX=    1848.79 SMIN=   -4559.87 TMAX=    3204.33 ANGLE= 57.9
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         1.989549E+03 -1.107460E+04  5.456032E+03  1.077975E+04  1.107460E+04
PLATE NO.       3             3             3             3             3
CASE  NO.     101           102             3             3           102
  ********************END OF ELEMENT FORCES********************
   38. UNIT METER KG
   39. PRINT ELEMENT FORCE LIST 6
 ELEMENT  FORCE    LIST     6
     SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    7
  ELEMENT FORCES    FORCE,LENGTH UNITS= KG   METE
  --------------
  **NOTE- IF A COMBINATION INCLUDES A DYNAMIC CASE OR IS AN SRSS OR ABS COMBINATION
          THEN RESULTS CANNOT BE COMPUTED PROPERLY.
    GLOBAL CORNER FORCES
JOINT     FX          FY          FZ          MX          MY          MZ
              ELE.NO.      6 FOR LOAD CASE     1
    6  0.0000E+00  1.1740E+03  0.0000E+00 -1.1313E+03  0.0000E+00  7.9082E+02
    8  0.0000E+00  1.2269E+03  0.0000E+00 -3.2050E+02  0.0000E+00  2.3979E+02
   12  0.0000E+00  4.4316E-12  0.0000E+00  1.7191E-12  0.0000E+00 -6.5491E-12
   11  0.0000E+00  4.8218E+02  0.0000E+00 -4.6695E+02  0.0000E+00 -6.0134E+02
              ELE.NO.      6 FOR LOAD CASE     2
    6  0.0000E+00  4.8864E+02  0.0000E+00 -4.7087E+02  0.0000E+00  3.2915E+02
    8  0.0000E+00  5.1067E+02  0.0000E+00 -1.3340E+02  0.0000E+00  9.9804E+01
   12  0.0000E+00 -8.5610E-13  0.0000E+00  1.4735E-12  0.0000E+00 -2.1284E-12
   11  0.0000E+00  2.0069E+02  0.0000E+00 -1.9435E+02  0.0000E+00 -2.5029E+02
              ELE.NO.      6 FOR LOAD CASE     3
    6  2.1397E+04  6.6191E+02  1.3024E+04  3.6476E+02  2.7909E+03  4.5923E+02
    8 -1.3861E+04 -9.9012E+02 -7.5358E+03 -1.6985E+03 -3.7448E+03 -1.6516E+03
   12 -5.1567E-11  1.9076E-11  5.1567E-11  6.5491E-12 -3.5365E-11 -9.1687E-12
   11 -7.5358E+03  3.2821E+02 -5.4884E+03  1.9902E+03  9.5385E+02  3.1726E+03
              ELE.NO.      6 FOR LOAD CASE   101
    6  0.0000E+00  1.6627E+03  0.0000E+00 -1.6022E+03  0.0000E+00  1.1200E+03
    8  0.0000E+00  1.7376E+03  0.0000E+00 -4.5390E+02  0.0000E+00  3.3959E+02
   12  0.0000E+00  3.1510E-04  0.0000E+00 -3.8582E-04  0.0000E+00 -5.1630E-04
   11  0.0000E+00  6.8287E+02  0.0000E+00 -6.6131E+02  0.0000E+00 -8.5163E+02
              ELE.NO.      6 FOR LOAD CASE   102
    6  2.1397E+04  1.8359E+03  1.3024E+04 -7.6656E+02  2.7909E+03  1.2500E+03
    8 -1.3861E+04  2.3682E+02 -7.5358E+03 -2.0190E+03 -3.7448E+03 -1.4118E+03
   12 -1.0641E-02  4.2204E-04 -1.7262E-02 -5.7606E-04 -1.3051E-03 -7.7228E-04
   11 -7.5358E+03  8.1039E+02 -5.4884E+03  1.5232E+03  9.5385E+02  2.5712E+03
   40. FINISH
     SLAB SUPPORTED ALONG 2 EDGES                             -- PAGE NO.    8
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44: 2 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
```

```
         *                  http://www.bentley.com                      *
         ****************************************************************
```

T.3 Post-Processing

If there are no errors in the input, the analysis is successfully completed. The extensive facilities of the Post-processing mode can then be used to:

- view the results graphically and numerically
- assess the suitability of the structure from the standpoint of safety, serviceability and efficiency
- create customized reports and plots

The procedure for entering the post processing mode is explained in T.2 Opening the postprocessing workflow (on page 520).

Node results such as displacements and support reactions are available for all models. The methods explained in the first two tutorials may be used to explore these. For this example, you will examine the support reactions.

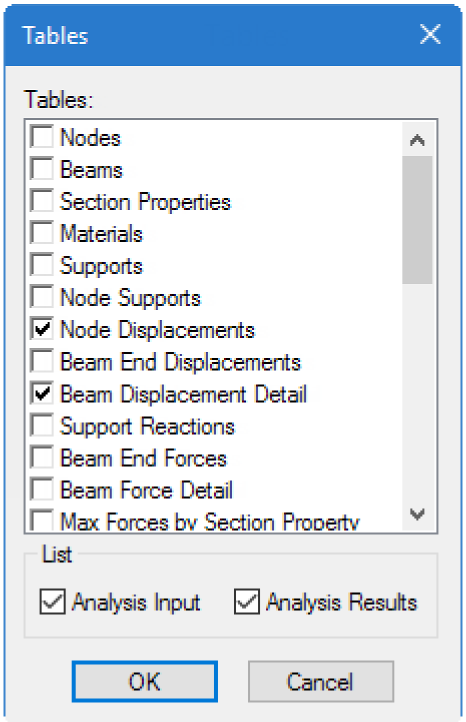
If beams are present in the model, beam results will be available too. As no beams are present in this model, this is not covered for this tutorial.

For plates, the results available are stresses, and “unit width” moments. There are several different methods for viewing these results, as explained in the next few sections.

T.3 Viewing stress values in a tabular form

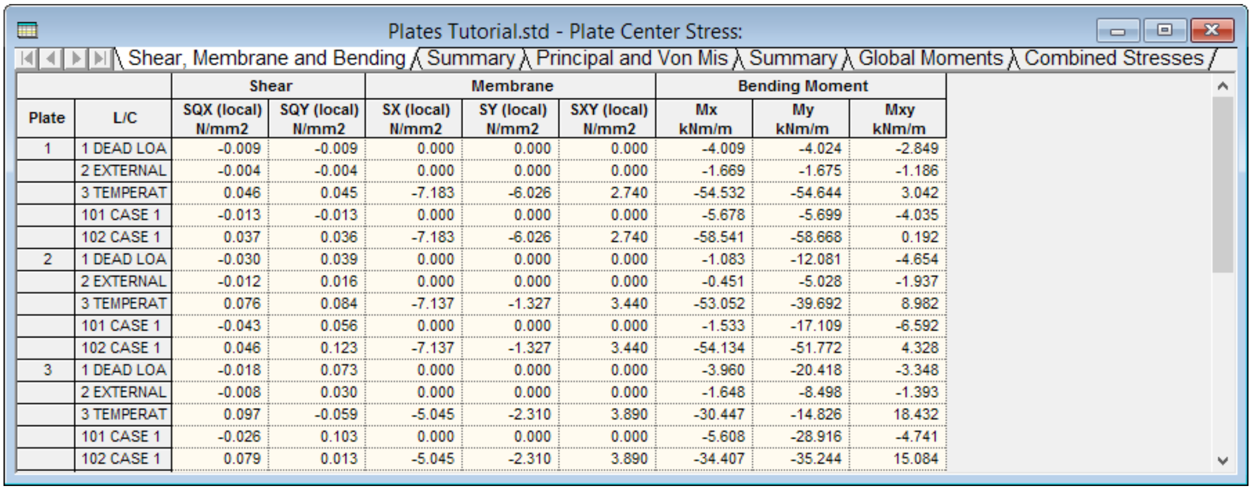
1. On the **View** ribbon tab, select the **Tables** tool in the **Windows** group.

   The **Tables** dialog opens.



2. Select **Plate Center Stress** and click **OK**.

The **Plate Center Stress** table opens.



		Shear		Membrane			Bending Moment		
Plate	L/C	SQX (local) N/mm2	SQY (local) N/mm2	SX (local) N/mm2	SY (local) N/mm2	SXY (local) N/mm2	Mx kNm/m	My kNm/m	Mxy kNm/m
1	1 DEAD LOA	-0.009	-0.009	0.000	0.000	0.000	-4.009	-4.024	-2.849
	2 EXTERNAL	-0.004	-0.004	0.000	0.000	0.000	-1.669	-1.675	-1.186
	3 TEMPERAT	0.046	0.045	-7.183	-6.026	2.740	-54.532	-54.644	3.042
	101 CASE 1	-0.013	-0.013	0.000	0.000	0.000	-5.678	-5.699	-4.035
	102 CASE 1	0.037	0.036	-7.183	-6.026	2.740	-58.541	-58.668	0.192
2	1 DEAD LOA	-0.030	0.039	0.000	0.000	0.000	-1.083	-12.081	-4.654
	2 EXTERNAL	-0.012	0.016	0.000	0.000	0.000	-0.451	-5.028	-1.937
	3 TEMPERAT	0.076	0.084	-7.137	-1.327	3.440	-53.052	-39.692	8.982
	101 CASE 1	-0.043	0.056	0.000	0.000	0.000	-1.533	-17.109	-6.592
	102 CASE 1	0.046	0.123	-7.137	-1.327	3.440	-54.134	-51.772	4.328
3	1 DEAD LOA	-0.018	0.073	0.000	0.000	0.000	-3.960	-20.418	-3.348
	2 EXTERNAL	-0.008	0.030	0.000	0.000	0.000	-1.648	-8.498	-1.393
	3 TEMPERAT	0.097	-0.059	-5.045	-2.310	3.890	-30.447	-14.826	18.432
	101 CASE 1	-0.026	0.103	0.000	0.000	0.000	-5.608	-28.916	-4.741
	102 CASE 1	0.079	0.013	-5.045	-2.310	3.890	-34.407	-35.244	15.084

Figure 93:

The table has the following tabs:

Shear, Membrane and Bending — These terms are explained in G.5.1 Plate and Shell Elements (on page 2099). The individual values for each plate for each selected load case are displayed.

Summary — This tab contains the maximum for each of the 8 values listed in the Shear, Membrane and Bending tab.

Principal and Von Mises — These terms too are explained in G.5.1 Plate and Shell Elements (on page 2099). The individual values for each plate for each selected load case are displayed, for the top and bottom surfaces of the elements.

Summary — This tab contains the maximum for each of the 8 values listed in the Principal and Von Mises tab.

Global Moments — This tab provides the moments about the global X, Y and Z axes at the center of each element.

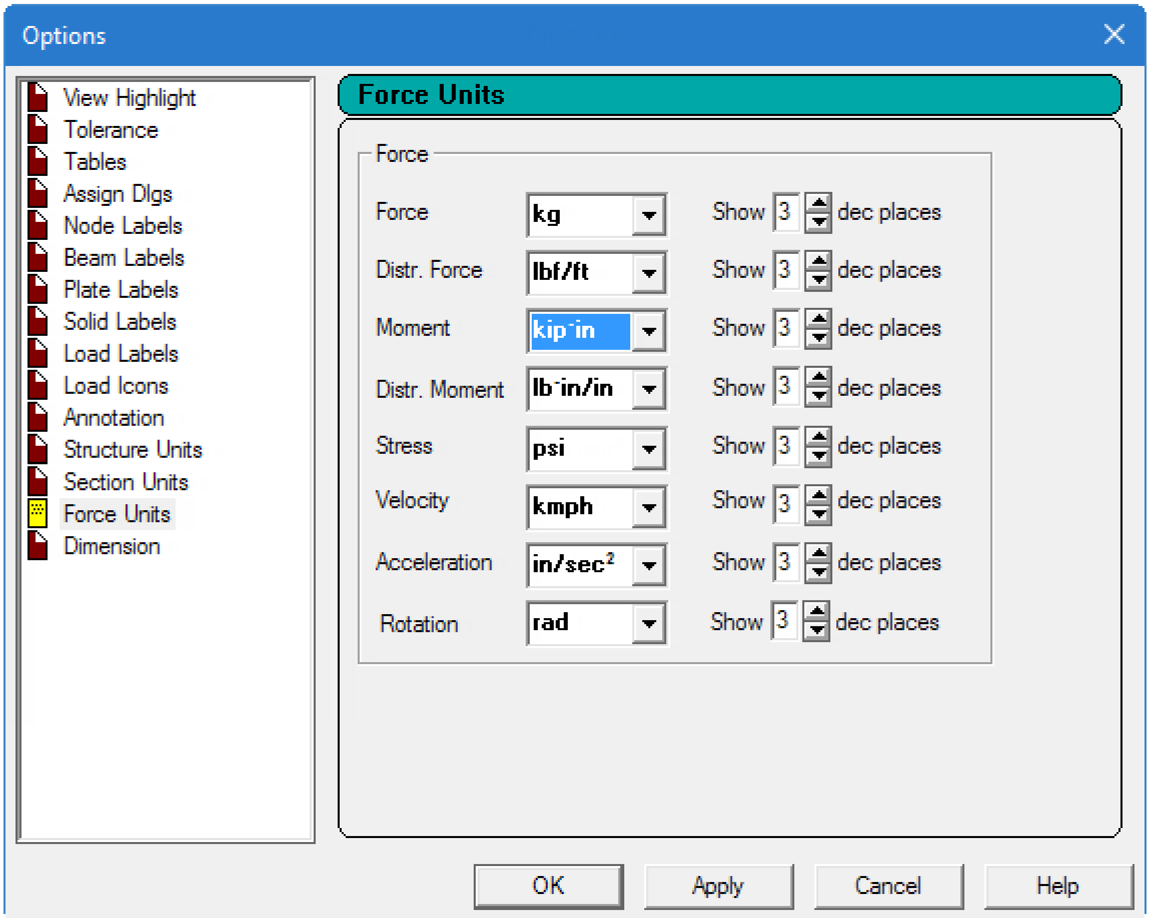
3. (Optional) Right-click in the table area and select **Print** from the pop-up menu.

T.3 Changing the units of values in the output

The length and force units of the stresses and moments are displayed alongside the individual column headings for the terms.

1. On the **File** ribbon tab, select **Display Options** on the **Settings** tab.

   The **Options** dialog opens.

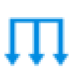


2. Select the **Force Units** tab and specify the required unit from the **Stress** and **Moment** fields.
3. (Optional) Click **Apply**
4. Click **OK**.

T.3 Limiting the load cases for which the results are displayed

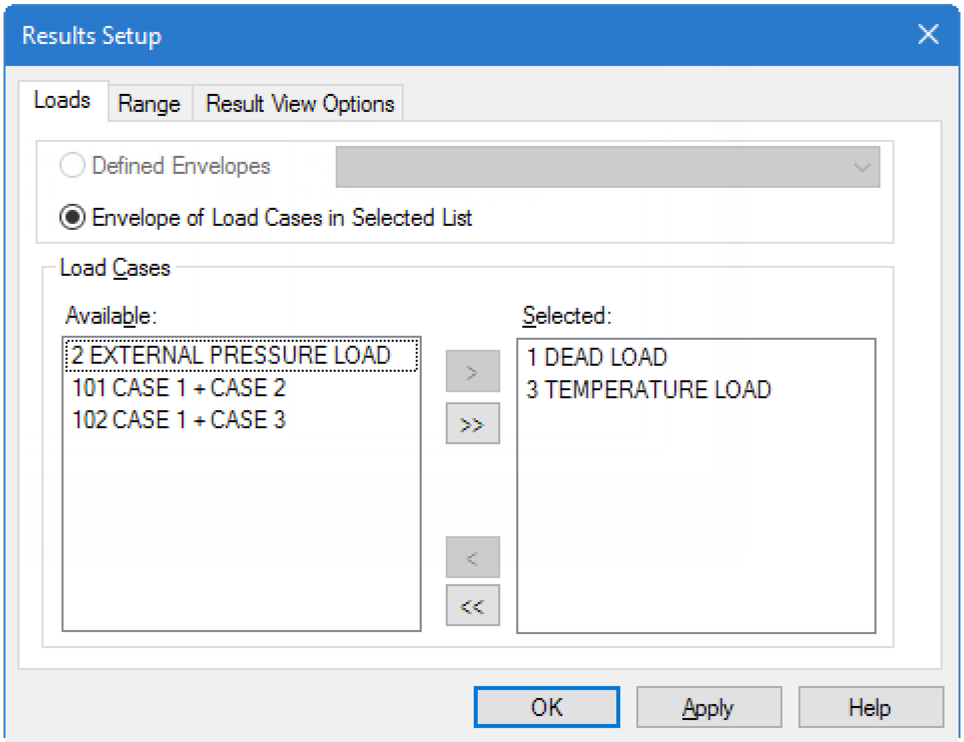
Use the following procedure to change the load list used for present results in the Post Processing mode.

1. On the **Results** ribbon tab, select the **Select Load Case** tool in the **Configuration** group.

   The **Results Setup** dialog opens.
2. Click **[<<]**.

   All load cases are moved from the Selected list to the Available list.
3. Select the load cases you want from the **Available** list and the click **[>]**.
   The selected load cases are transferred from the **Available** list to the **Selected** list



4. Click **OK**.

T.3 Stress Contours

Stress contours are a color-based plot of the variation of stress or moment across the surface of the slab or a selected portion of it.

1. On the **Results** ribbon tab, select the **Plate Stress** tool in the **View Results** group.

   The **Diagrams** dialog opens to the **Plate Stress Contour** tab.
2. Select the load case number from the **Load Case** drop-down list.

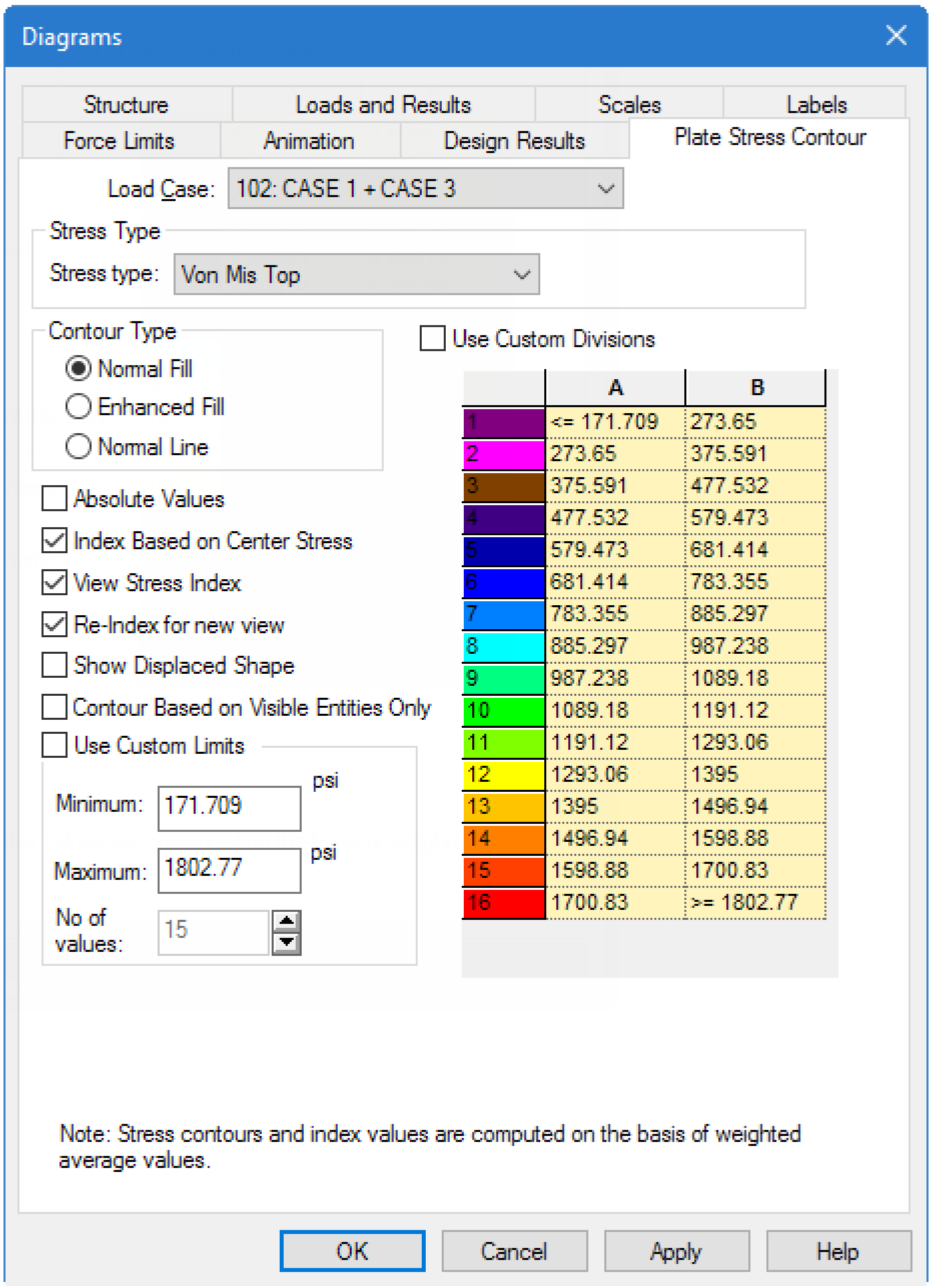
   Stress values are known exactly only at the plate centroid locations. Everywhere else, they are calculated by linear interpolation between the center point stress values of adjacent plates.

   The **Enhanced** type contour chooses a larger number of points compared to the **Normal** type contour in determining the stress variation.
3. Select the specific type of stress for which you want the contour drawn from the **Stress type** drop-down list.
4. Setting the **View Stress Index** option will display a small table consisting of the numerical range of values from smallest to largest which are represented in the plot.

   Set the following:

   Load case — 102: CASE 1 + CASE 3
   Stress Type — Von Mis Top

Contour Type — Normal Fill
Index based on Center Stress checked
View Stress Index checked
Re-Index for new view checked



5. Click **Apply**.

   The following diagram will be displayed.

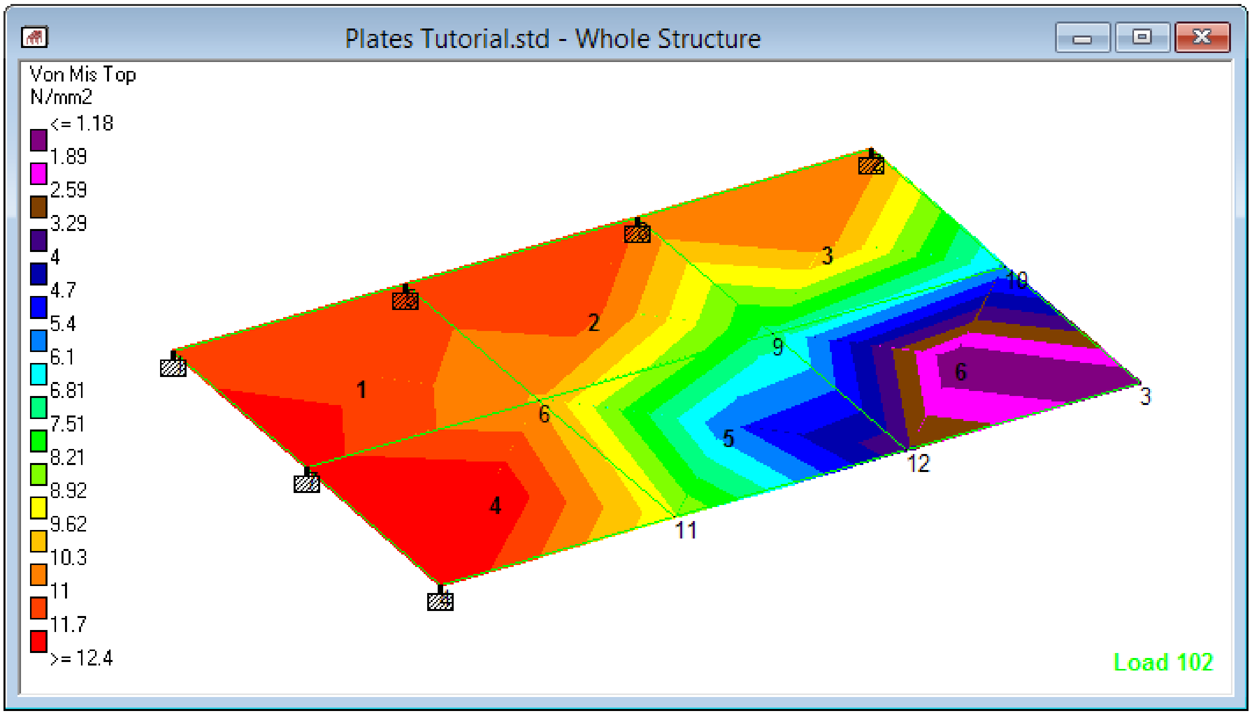


Figure 94: Plate stresses plotted for Load Case 102

6. (Optional) If some portion of the structure appears truncated, use the Zoom and Pan tools located on the View toolbar to adjust the display.

 Tip: If you mouse has a scroll wheel, you can also use that to zoom in or out (scroll up or down) or to pan the image (click-and-drag the wheel button).

Note: Leave the dialog open to explore the Animation feature explained in the next topic.

T.3 Animating stress contours

The Diagrams dialog is used to provide a dynamic animation of stress plots.

In order to animate a diagram, such as the plate stress contour, you must first have a diagram displayed. Refer to T.3 Stress Contours (on page 613) for how to plot those results on the structure.

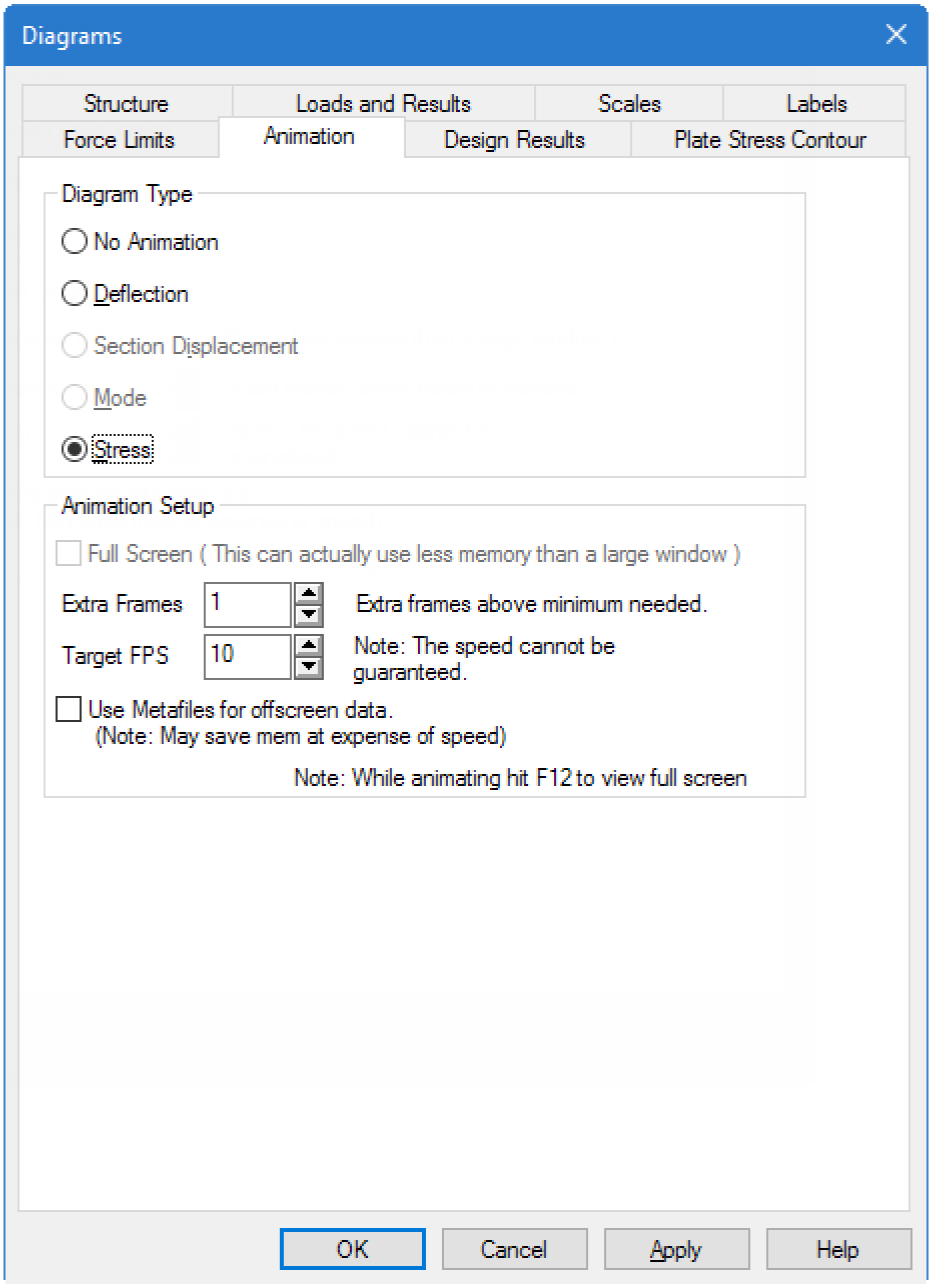
1. Either:

 on the **Results** ribbon tab, select the **Animation** tool in the **Animation** group

 or

 if the **Diagrams** dialog is still open from the previous procedure, select the **Animation** tab
2. Select the **Stress** option.

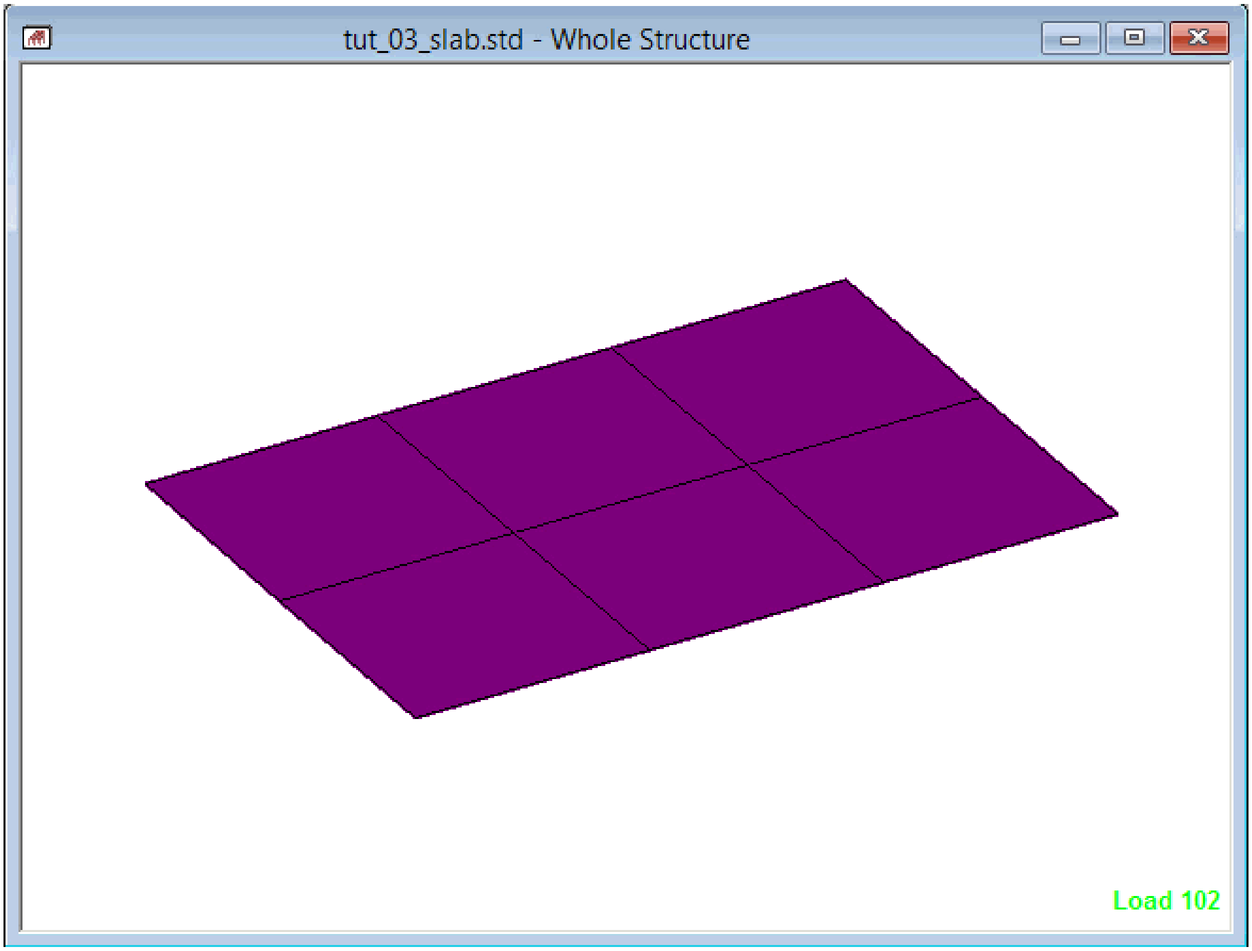


3. (Optional) Set the **Target FPS** 5

Tip: This will slow the animation down to make it easier to distinguish the frames.

4. Click **Apply**.

The plate stress contours are animated loading and unloading.



5. To stop the animation, select the **No Animation** option and click **Apply** again.

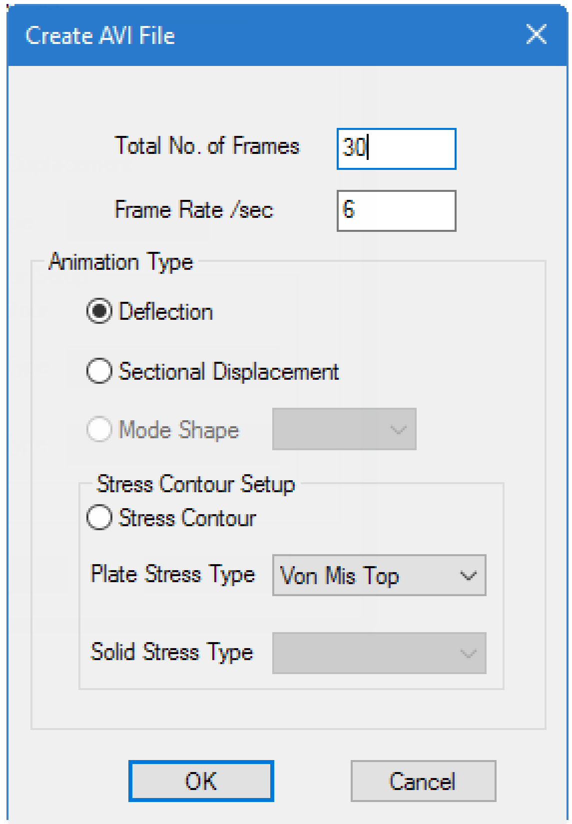
T.3 Creating AVI Files

You can save dynamic result, such as a deflection diagram in animation, to a video file.

This feature is available in STAAD.Pro for node deflection, beam section displacement, mode shape and plate stress contour diagrams. These files can then be viewed using video player programs such as the Windows Media Player.

1. On the **Utilities** ribbon tab, select the **Create AVI File** tool in the **Utilities** group.

The **Create AVI File** dialog opens.



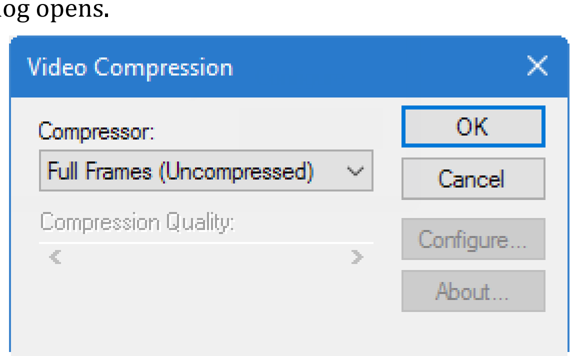
Total No. of Frames Sets the number of frames used to capture the movement. In an animated view, the movement from one extremity to the other is captured as several frames.

Frame Rate /sec sets the speed of the motion

The rest of the options in the above dialog are for the type of diagram from which the video file is to be created. Certain items such as Mode Shape and Plate Stress contour are disabled if the required data of that type are not present in the STAAD file, such as a modal extraction, or finite elements.

2. Select the **Stress Contour** option for Animation Type and then select **Von Mis Top** from the Plate Stress Type drop-down list.
3. Click **OK.**
   A dialog opens to specify the file name.
4. Type a file name and location for the video file and then click **OK**.

   The **Video Compression** dialog opens.



5. (Optional) Select a Compressor option and set the Compression Quality value.

   Note: Video files can be quite large, and compression is a technique used reduce the size of these files, though some video smoothness is lost in this process.
6. Click **OK** to begin creating the video file.

   When the file has been generated, a message opens indicating that the operation was successful.
7. Click **OK** to dismiss.
   The file with the extension `.AVI` is saved in the same folder where the STAAD input file is located.

T.3 Viewing plate results using element query

Element Query is a facility where several results for a specific element can be viewed at the same time from a single dialog.

1. On the **Results** tab, select the **Plate Cursor** tool in the **Selection** group.

   
2. Either:

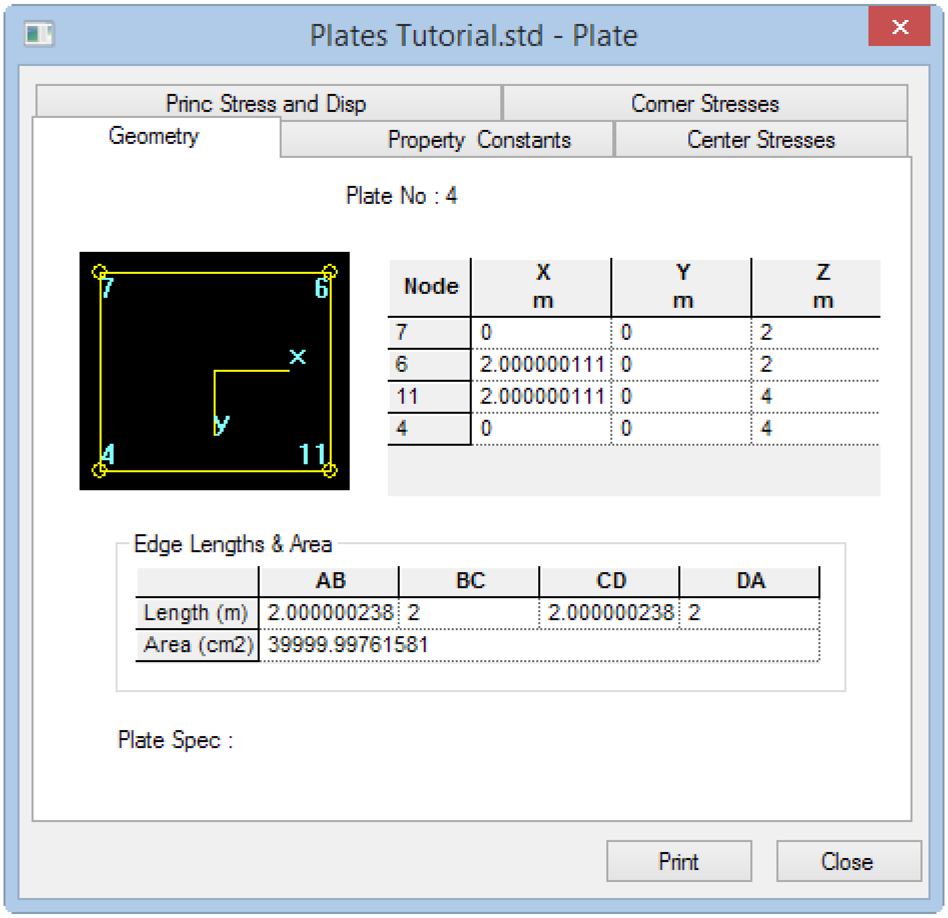
   Double-click on element 4

   or

   select element 4 and then select the **Properties** tool in the **Model** group on the **Plate Tools** ribbon tab.

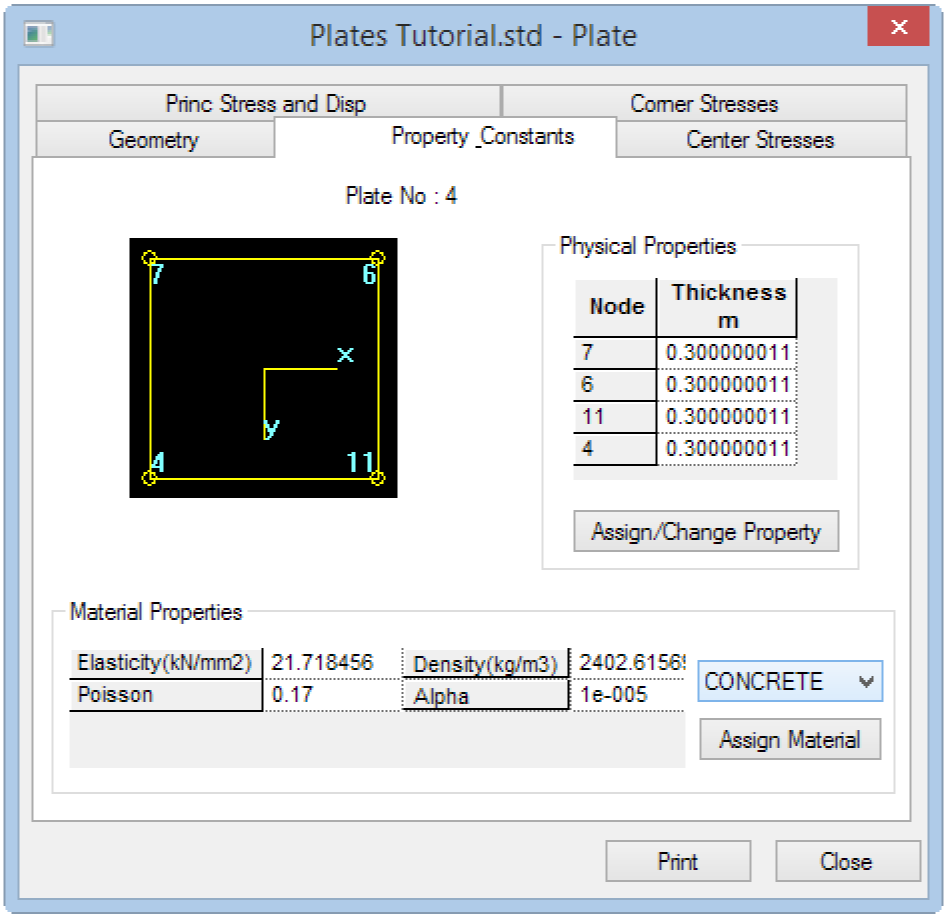
   Tip: Press **<Shift+P>** to display the plate numbers.

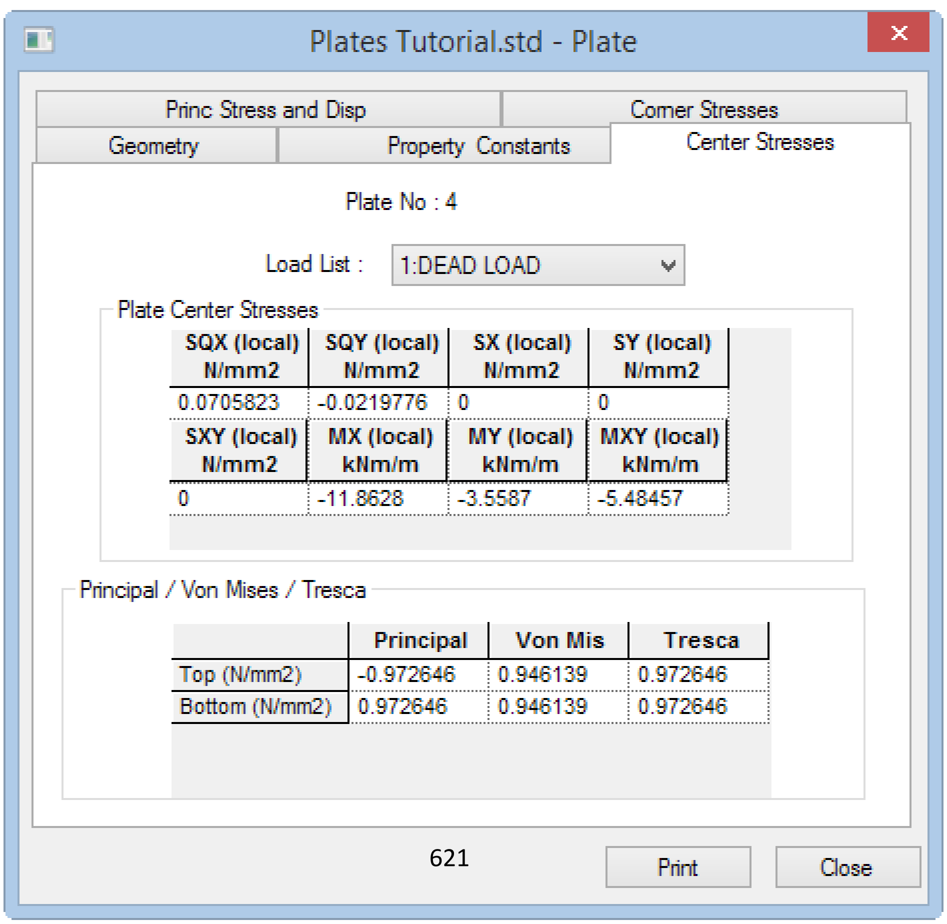
   The **Plate** dialog opens.

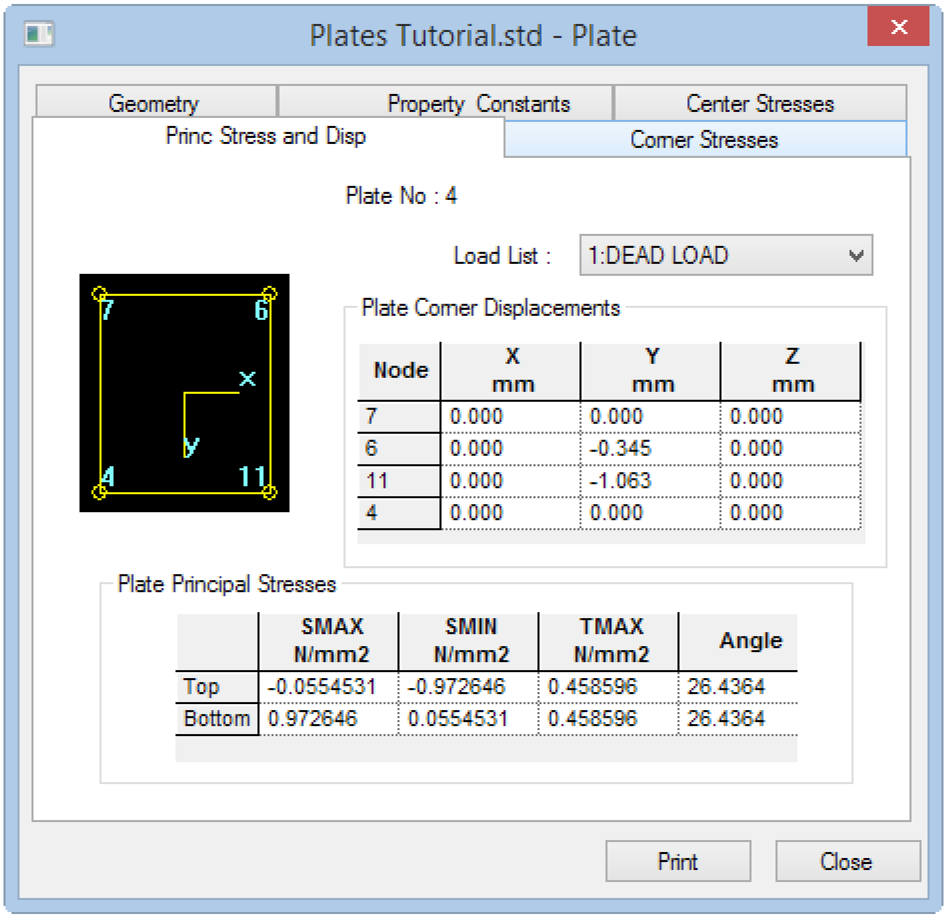


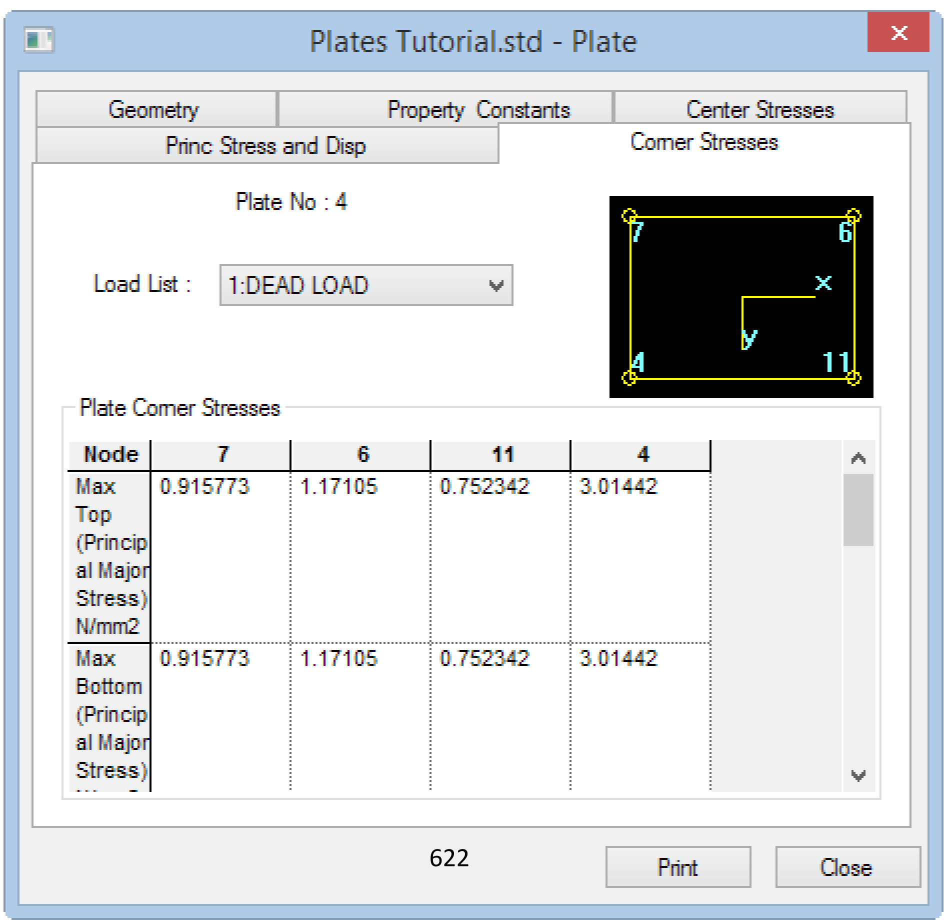
The various tabs of the query box enable one to view various types of information such as the plate geometry, property constants, stresses, etc., for various load cases, as well as print those values.

Some example tabs of this dialog are shown in the following figures.


Plates Tutorial.std - Plate
Princ Stress and Disp
Corner Stresses
Geometry
Property Constants
Center Stresses
Plate No : 4
7
6
x
y
4
11
Physical Properties
Node
Thickness m
7 0.300000011
6 0.300000011
11 0.300000011
4 0.300000011
Assign/Change Property
Material Properties
Elasticity(kN/mm2) 21.718456
Density(kg/m3) 2402.6156
Poisson 0.17
Alpha 1e-005
CONCRETE
Assign Material
Print
Close


Plates Tutorial.std - Plate
Princ Stress and Disp
Corner Stresses
Geometry
Property Constants
Center Stresses
Plate No : 4
Load List :
1:DEAD LOAD
Plate Center Stresses
SQX (local) N/mm2
SQY (local) N/mm2
SX (local) N/mm2
SY (local) N/mm2
0.0705823
-0.0219776
0
0
SXY (local) N/mm2
MX (local) kNm/m
MY (local) kNm/m
MXY (local) kNm/m
0
-11.8628
-3.5587
-5.48457
Principal / Von Mises / Tresca
Principal
Von Mis
Tresca
Top (N/mm2) -0.972646 0.946139 0.972646
Bottom (N/mm2) 0.972646 0.946139 0.972646
Print
Close


Plates Tutorial.std - Plate
Geometry
Property Constants
Center Stresses
Princ Stress and Disp
Corner Stresses
Plate No : 4
Load List :
1:DEAD LOAD
7
6
x
y
4
11
Plate Corner Displacements
Node | X mm | Y mm | Z mm
7 | 0.000 | 0.000 | 0.000
6 | 0.000 | -0.345 | 0.000
11 | 0.000 | -1.063 | 0.000
4 | 0.000 | 0.000 | 0.000
Plate Principal Stresses
| SMAX N/mm2 | SMIN N/mm2 | TMAX N/mm2 | Angle
Top | -0.0554531 | -0.972646 | 0.458596 | 26.4364
Bottom | 0.972646 | 0.0554531 | 0.458596 | 26.4364
Print
Close


Plates Tutorial.std - Plate
Geometry
Property Constants
Center Stresses
Princ Stress and Disp
Corner Stresses
Plate No : 4
Load List :
1:DEAD LOAD
7
6
x
y
4
11
Plate Corner Stresses
Node | 7 | 6 | 11 | 4
Max Top (Principal Major Stress) N/mm2 | 0.915773 | 1.17105 | 0.752342 | 3.01442
Max Bottom (Principal Major Stress) | 0.915773 | 1.17105 | 0.752342 | 3.01442
Print
Close

T.3 Producing a report

Produce a report consisting of the plate principal stresses, for all plates, sorted in the order from Low to High of the Principal Maximum Stress (SMAX) for load cases 101 and 102.

Occasionally, you will need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form.

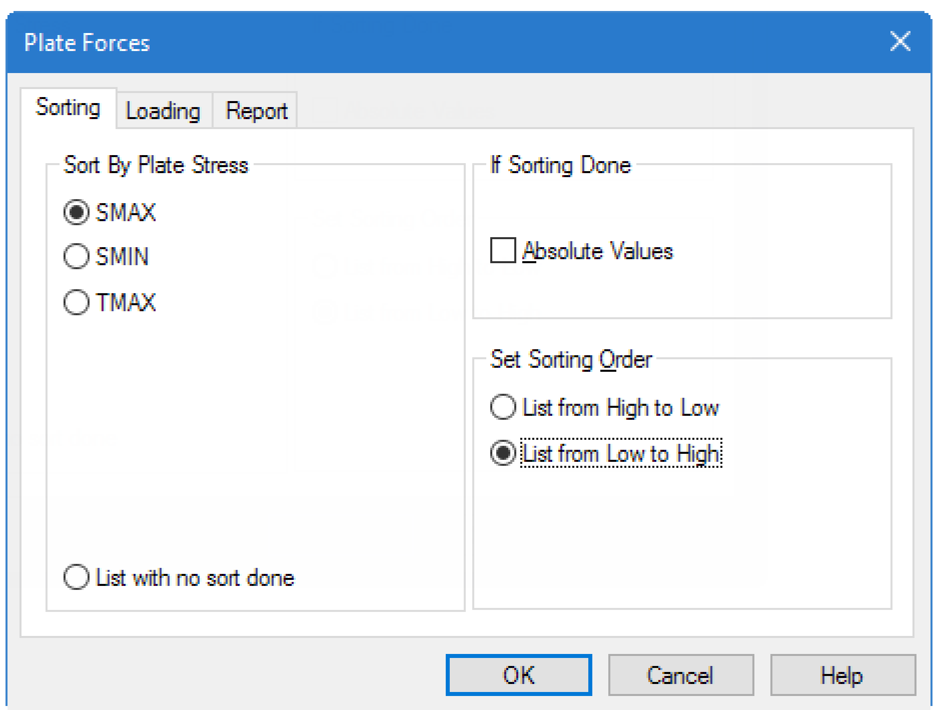
1. On the **Results** tab, select the **Plate Cursor** tool in the **Selection** group.

2. Select the all the plate elements.

Tip: Drag a window around all the elements in the view area or press **<Ctrl+A>**.

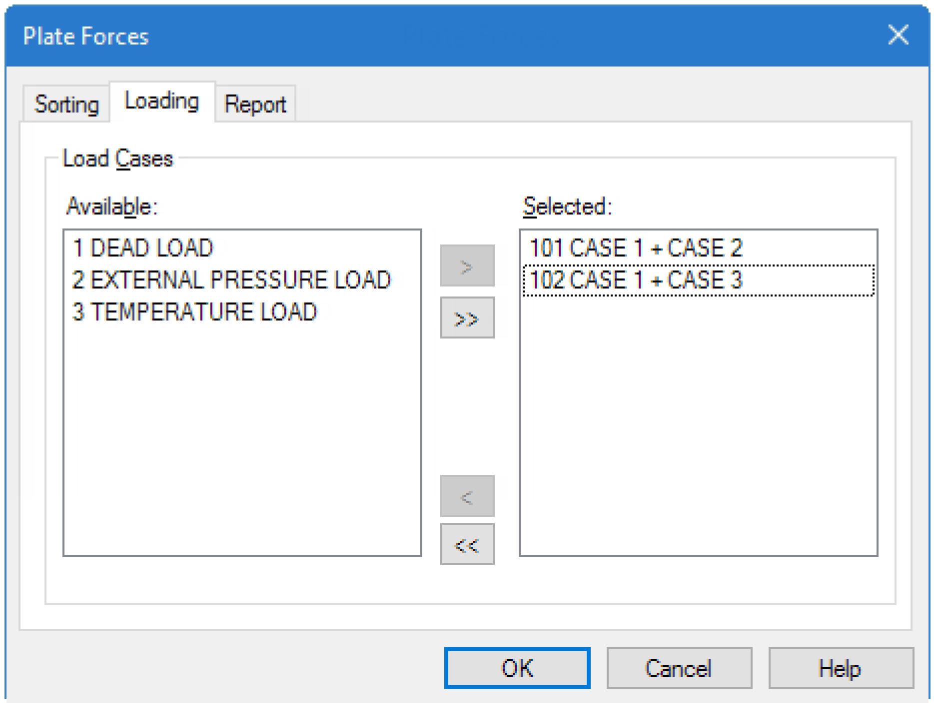
3. On the **Results** ribbon tab, select the **Reports** > **Principal Stresses** tool in the **Reports** group.

The **Plate Forces** dialog opens.



4. Specify the report contents:
   a. Select the **Sorting** tab.
   b. Select the **SMAX** option in the **Sort By Plate Stress** group.

   c. Select **List from Low to High** option in the **Set Sorting Order** group.
   d. Clear the **Absolute Values** check box in the **If Sorting done** group.
5. Select the load cases to use:
   a. Select the **Loading** tab.



   b. Select load cases `101` and `102` in the **Available** list and click **[>]** to add them to the **Selected** list.
6. (Optional) You can save this report for future use,
   a. Select the **Report** tab
   b. Type a **Title** for the report.
   c. Set the **Save Report** check box.
7. Click **OK**.

   The Plate Forces table opens to display the table of maximum principal stress with SMAX values sorted from Low to High.

tut_03_slab.std - Plate Forces

Plate	L/C		SMAX psi	SMIN psi	TMAX psi	Angle
1	102	Top	-1116.735	-1932.408	407.837	-39.107
2	102	Top	-433.175	-1818.515	692.670	-25.667
4	102	Top	-418.965	-1975.360	778.197	-41.610
3	102	Top	-133.907	-1606.236	736.164	-37.347
6	102	Top	-75.538	-196.529	60.496	-34.905
5	102	Top	-65.322	-836.845	385.761	-18.993
3	101	Top	-45.261	-288.561	121.650	78.932
1	101	Top	-15.983	-94.019	39.018	45.073
4	101	Top	-11.390	-199.786	94.198	26.436
2	101	Top	8.528	-188.779	98.654	69.878
1	102	Bottom	13.197	-795.705	404.451	-38.968
6	101	Top	44.692	-76.067	60.380	53.363
6	101	Bottom	76.067	-44.692	60.380	53.363
1	101	Bottom	94.019	15.983	39.018	45.073
5	102	Bottom	107.338	0.745	53.297	38.036
5	101	Top	113.895	-116.089	114.992	54.271
5	101	Bottom	116.089	-113.895	114.992	54.271
4	102	Bottom	138.902	-935.369	537.136	-47.958
6	102	Bottom	182.676	-2.348	92.512	-44.681
2	101	Bottom	188.779	-8.528	98.654	69.878
4	101	Bottom	199.786	11.390	94.198	26.436
3	102	Bottom	268.146	-661.351	464.748	-32.091
3	101	Bottom	288.561	45.261	121.650	78.932
2	102	Bottom	512.208	-715.830	614.019	-24.059

8. (Optional) To print this table, right-click anywhere within the table and select **Print**

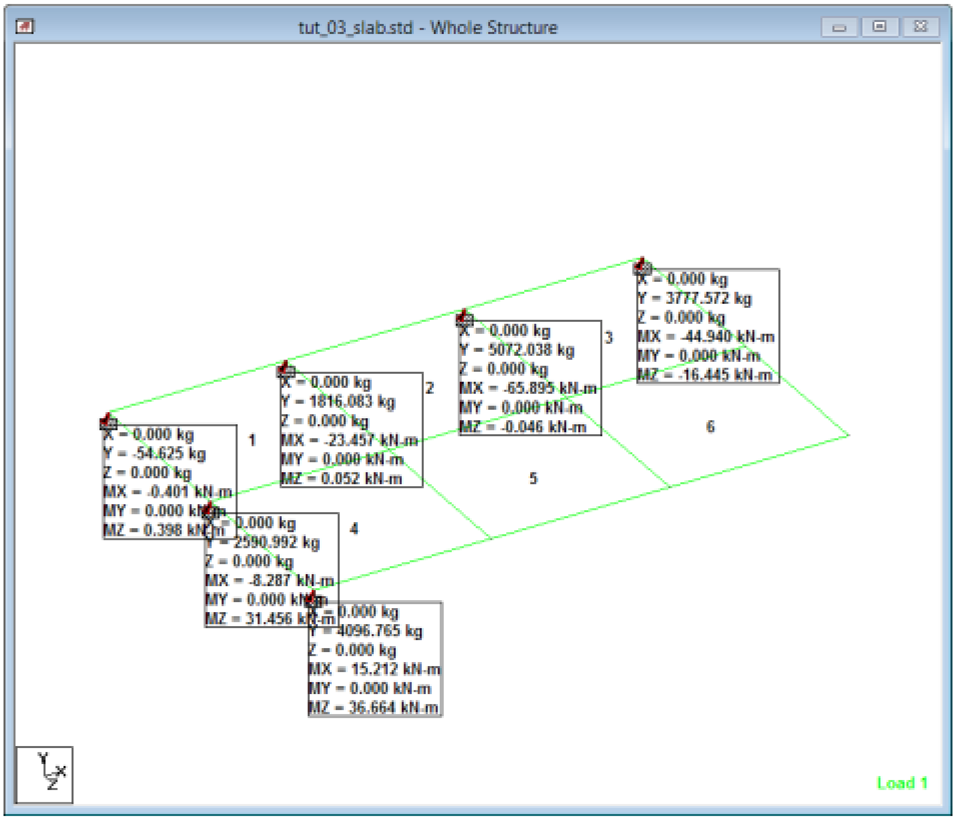
To transfer the contents of this table to a Microsoft Excel file, click the **Plate** label in the top-left corner of the table. The entire table is selected. Right-click and select **Copy** from the pop-up menu. You can now paste the table contents into a Microsoft Office Excel spreadsheet or other spreadsheet file.

T.3 Viewing Support Reactions

Use the nodal results page to obtain support reaction results.

1. Select the **Reactions** page in the **Postprocessing** page control bar.

   The reactions at the supports will be displayed on the drawing and the Support Reactions table is displayed on the right side of the program window.



The six values — namely, the three forces along global X, Y and Z, and the three moments Mx, My and Mz, in the global axis system — are displayed in a box for each support node.

tut_03_slab.std - Support Reactions:

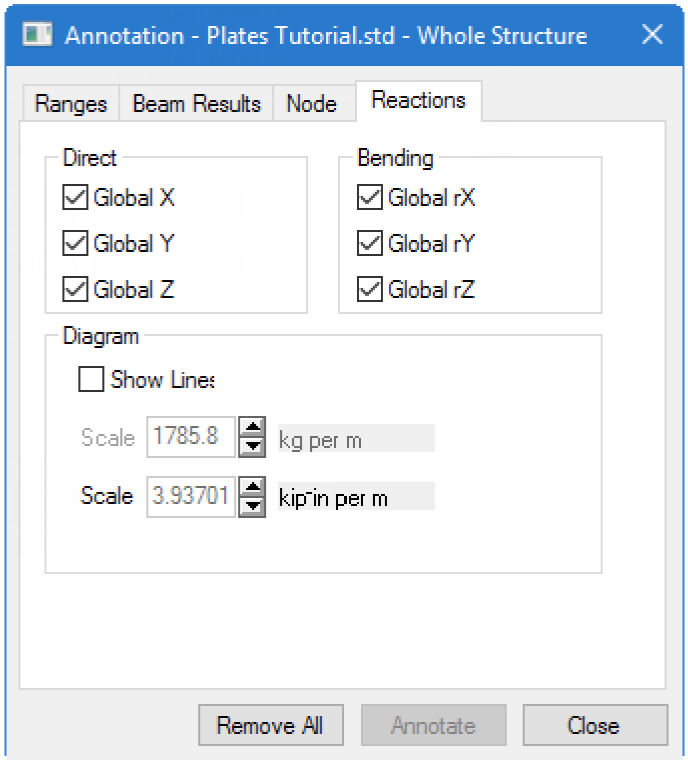
All | Summary | Envelope

		Horizontal	Vertical	Horizontal	Moment		
Node	L/C	Fx kg	Fy kg	Fz kg	Mx kN-m	My kN-m	Mz kN-m
1	1 DEAD LOA	0.000	-54.625	0.000	-0.401	0.000	0.398
	2 EXTERNAL	0.000	-22.736	0.000	-0.167	0.000	0.166
	3 TEMPERAT	167.55337E	2409.264	147.88130E	-71.280	68.375	71.263
	101 CASE 1	0.000	-77.361	0.000	-0.567	0.000	0.564
	102 CASE 1	167.55337E	2354.639	147.88130E	-71.681	68.375	71.661
2	1 DEAD LOA	0.000	1816.083	0.000	-23.457	0.000	0.052
	2 EXTERNAL	0.000	755.878	0.000	-9.763	0.000	0.022
	3 TEMPERAT	-201.83948E	5020.928	171.82707E	-134.655	350.875	25.587
	101 CASE 1	0.000	2571.961	0.000	-33.221	0.000	0.074
	102 CASE 1	-201.83948E	6837.011	171.82707E	-158.112	350.875	25.639
4	1 DEAD LOA	0.000	2590.992	0.000	-8.287	0.000	31.456
	2 EXTERNAL	0.000	1078.405	0.000	-3.449	0.000	13.093
	3 TEMPERAT	287.00058E	2777.288	-234.57780E	16.530	-37.619	97.175
	101 CASE 1	0.000	3669.398	0.000	-11.736	0.000	44.549

2. Display of one or more of the six terms of each support node may be toggled off in the following manner.

a. On the **Results** ribbon tab, select the **Annotate** tool in the **Configuration** group.

The **Annotation** dialog opens.



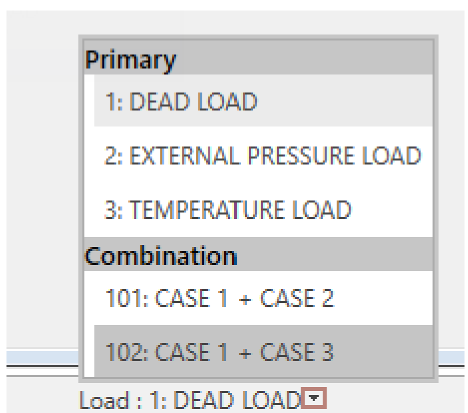
b. Select the **Reactions** tab.

c. Clear the degree of freedom check boxes in the **Direct** or **Bending** groups you want to hide from display.

d. Click **Annotate** and then **Close**.

The drawing will now contain only the selected items.

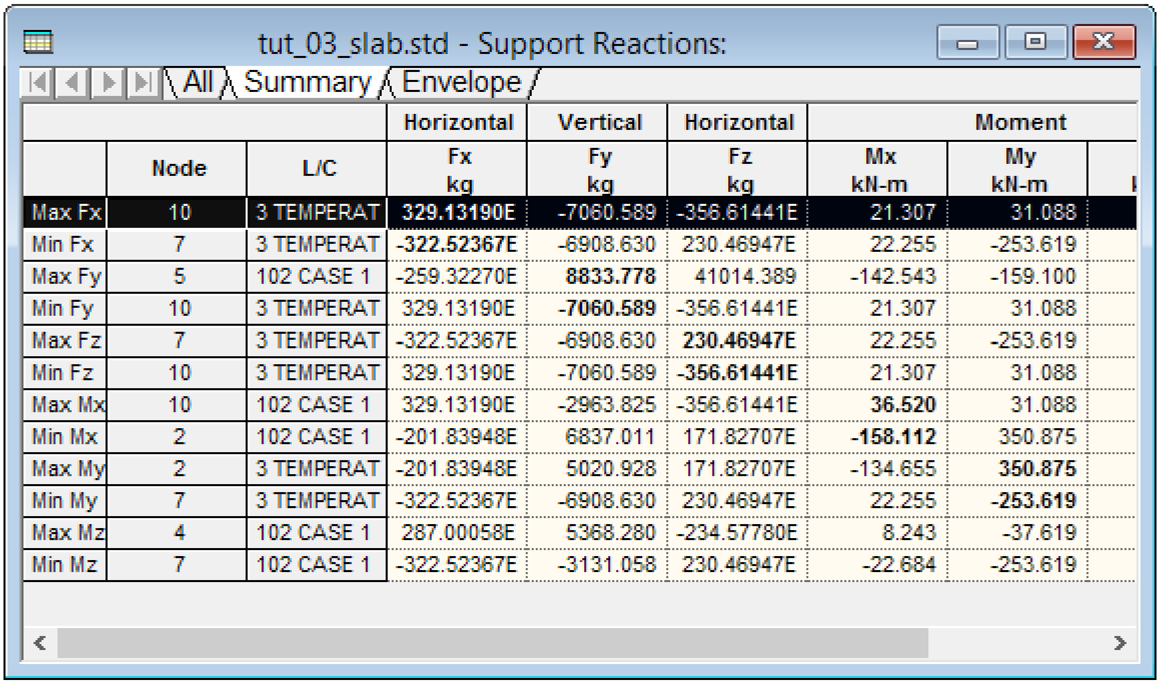
3. (Optional) To change the load case for which the reactions are displayed, select the desired case from the **Load** drop-down in the program status bar.



4. On the **Support Reactions** table, select the **Summary** tab.

Tip: You can also open this table by selecting **Tables** > **Reactions** tool in the **View Results** group on the **Results** ribbon tab.

The summary tab contains the maximum value for each of the 6 degrees of freedom along with the load case number responsible for it.



			Horizontal	Vertical	Horizontal	Moment	
	Node	L/C	Fx kg	Fy kg	Fz kg	Mx kN-m	My kN-m
Max Fx	10	3 TEMPERAT	329.13190E	-7060.589	-356.61441E	21.307	31.088
Min Fx	7	3 TEMPERAT	-322.52367E	-6908.630	230.46947E	22.255	-253.619
Max Fy	5	102 CASE 1	-259.32270E	8833.778	41014.389	-142.543	-159.100
Min Fy	10	3 TEMPERAT	329.13190E	-7060.589	-356.61441E	21.307	31.088
Max Fz	7	3 TEMPERAT	-322.52367E	-6908.630	230.46947E	22.255	-253.619
Min Fz	10	3 TEMPERAT	329.13190E	-7060.589	-356.61441E	21.307	31.088
Max Mx	10	102 CASE 1	329.13190E	-2963.825	-356.61441E	36.520	31.088
Min Mx	2	102 CASE 1	-201.83948E	6837.011	171.82707E	-158.112	350.875
Max My	2	3 TEMPERAT	-201.83948E	5020.928	171.82707E	-134.655	350.875
Min My	7	3 TEMPERAT	-322.52367E	-6908.630	230.46947E	22.255	-253.619
Max Mz	4	102 CASE 1	287.00058E	5368.280	-234.57780E	8.243	-37.619
Min Mz	7	102 CASE 1	-322.52367E	-3131.058	230.46947E	-22.684	-253.619

Refer to T.3 Changing the units of values in the output (on page 611) to change the units in which these values are displayed.

This brings us to the conclusion of this tutorial.

4
Modeling

This section of the help describes methods used to model your structure in STAAD.Pro.

This section begins by detailing how to model analytical elements using the Analytical Modeling workflow. Following that are sections on how to model the structure using other workflows and utilities in STAAD.Pro.

M. Navigating the Graphical View Window

This section describes how to control the graphical display of your model.

M. To select a center of rotation at a node

Used to select an node as the center of rotation. When toggled on, pressing **<Ctrl+Shift>** and clicking a node will set that node as the center of rotation.

1. Either:

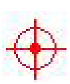
   On the **View** ribbon tab, select the **Toggle View Rotation Mode** tool in the **Rotation** group.

   or

   On the **View** ribbon tab, select the **Always Fit in Current Window** tool in the **Rotation** group.

   > **Note:** The icon for the Toggle View Rotation Mode is depressed when in this mode (and there is no check on the View > Always Fit in Current Window menu item).

   A message dialog appears to remind you how to next select a center of rotation.
2. Hold down the **<Ctrl+Shift>** keys and click on a node.

   A reticle  is placed on this node to indicate it is the center of rotation.
3. Either:

   Use the rotate tools found in the Rotate toolbar or the arrow keys to rotate the structure about this node.

   or

   Hold down the right mouse button while dragging the pointer to free rotate the model about this node.
4. Repeat steps 3 and 4 to re-center and rotate the structure.
5. To return the center of rotation to the center of the structural model:

   select the **Toggle View Rotation Mode** tool

   or

select the **Always Fit in Current Window** tool such that a check mark appears

or

select the **Use Center of the structure as rotational Center** option in the **Orientation** dialog.

Tip: You may specify the exact center of rotation coordinates at any point using the **Orientation** dialog.

M. To view a 3D rendering of your model

To display the model with full 3D sections and lighting, use the following procedure.

Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Note: The 3D rendering is used to display the model only. You cannot select model objects or otherwise manipulate the model in this view.

1. On the **View** ribbon tab, select the **3D Rendering** tool in the **Windows** group.

   A new view window opens with the model displayed as a 3D rendering.
2. Navigate the rendering view as follows:

To...	Do this...
rotate the structure	left-click in the view and drag the mouse in any direction
pan the structure	click-and-drag the middle mouse button (typically the scroll wheel)
zoom in/out	scroll the mouse wheel up or down

3. (Optional) To generate a picture for inclusion in reports:
   a. Right-click and select **Take Picture** from the pop-up menu.
      The **Picture #** dialog opens.
   b. Type a picture **ID** and **Caption**.
   c. Click **OK**.
4. Click the **[X]** in the top, right-hand corner of the window to close the view.

M. 3D Rendering View Right-click View Menu

Menu item	Description
View Axes	Toggles the display of the global axes.
Perspective	Toggles the perspective lens for the rendered window.
Wireframe View	When selected, solid rendering is turned off. Only the outline of shapes are shown.

Menu item	Description
Enable Lighting	Toggles the light source. Turning this feature off removes any reflection.
Lights (list)	Select the light color from the list.
Change Light Color	The lighting is by default gray to resemble the color of steel. This can be changed to any other color.
Enable Texture Mapping	The structure will be displayed with a unique texture.
Change Entity Color…	Opens the Change Entity Color dialog, which is used change the color of structure entities by class.
Change Background Color…	Opens a color selector dialog, so the default background color can be changed.
Change Font…	Opens a font selection dialog, so the text display can be changed.
Model View Details	Opens the Structure Diagram Info dialog, which contains a summary of the entity counts in the structure.
Take Picture	Takes a STAAD image of the rendered view for inclusion in reports. A picture ID dialog opens to name the image.

M. Labels

STAAD.Pro has numerous options for displaying labels in the view window.

M. To switch on labels for nodes, beams, plates, etc.

Labels are a way of identifying the entities we have drawn on the screen.

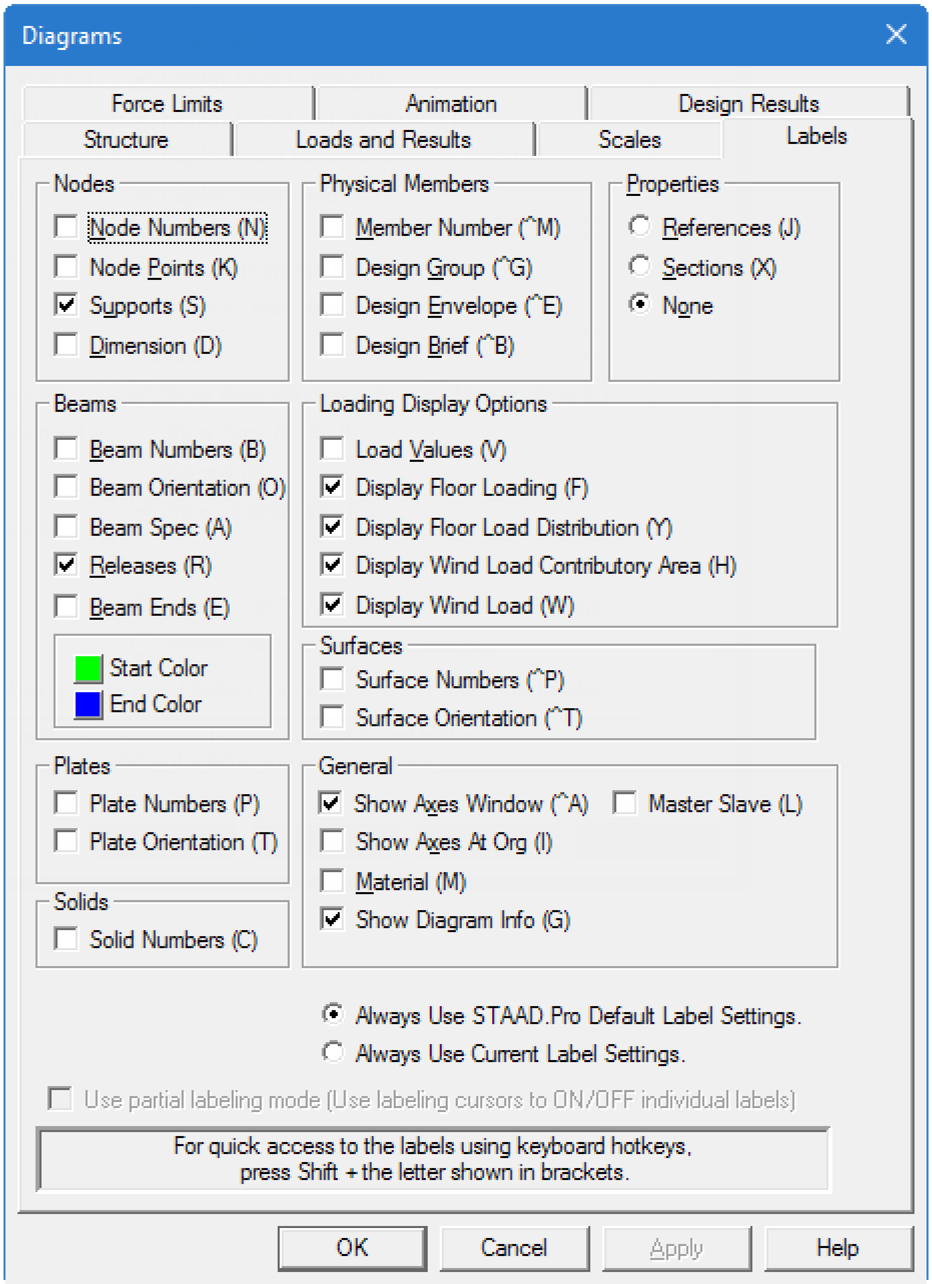
1. Either:

   Select the **Labels** tool 

   or

   Right-click anywhere in the View area and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens.
2. Select the **Labels** tab and then select the options for the appropriate labels (examples shown in the following figure).


Diagrams
Force Limits
Animation
Design Results
Structure
Loads and Results
Scales
Labels
Nodes
Node Numbers (N)
Node Points (K)
Supports (S)
Dimension (D)
Physical Members
Member Number (^M)
Design Group (^G)
Design Envelope (^E)
Design Brief (^B)
Properties
References (J)
Sections (X)
None
Beams
Beam Numbers (B)
Beam Orientation (O)
Beam Spec (A)
Releases (R)
Beam Ends (E)
Start Color
End Color
Loading Display Options
Load Values (V)
Display Floor Loading (F)
Display Floor Load Distribution (Y)
Display Wind Load Contributory Area (H)
Display Wind Load (W)
Surfaces
Surface Numbers (^P)
Surface Orientation (^T)
Plates
Plate Numbers (P)
Plate Orientation (T)
Solids
Solid Numbers (C)
General
Show Axes Window (^A)
Master Slave (L)
Show Axes At Org (I)
Material (M)
Show Diagram Info (G)
Always Use STAAD.Pro Default Label Settings.
Always Use Current Label Settings.
Use partial labeling mode (Use labeling cursors to ON/OFF individual labels)
For quick access to the labels using keyboard hotkeys,
press Shift + the letter shown in brackets.
OK
Cancel
Apply
Help

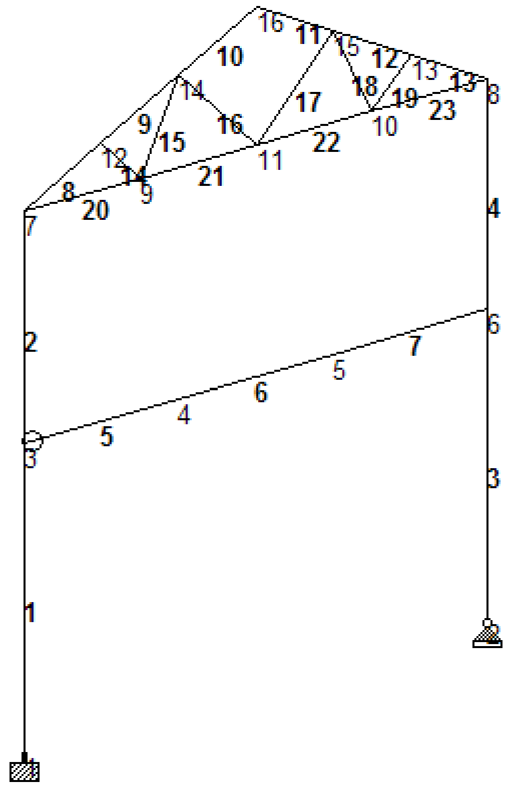
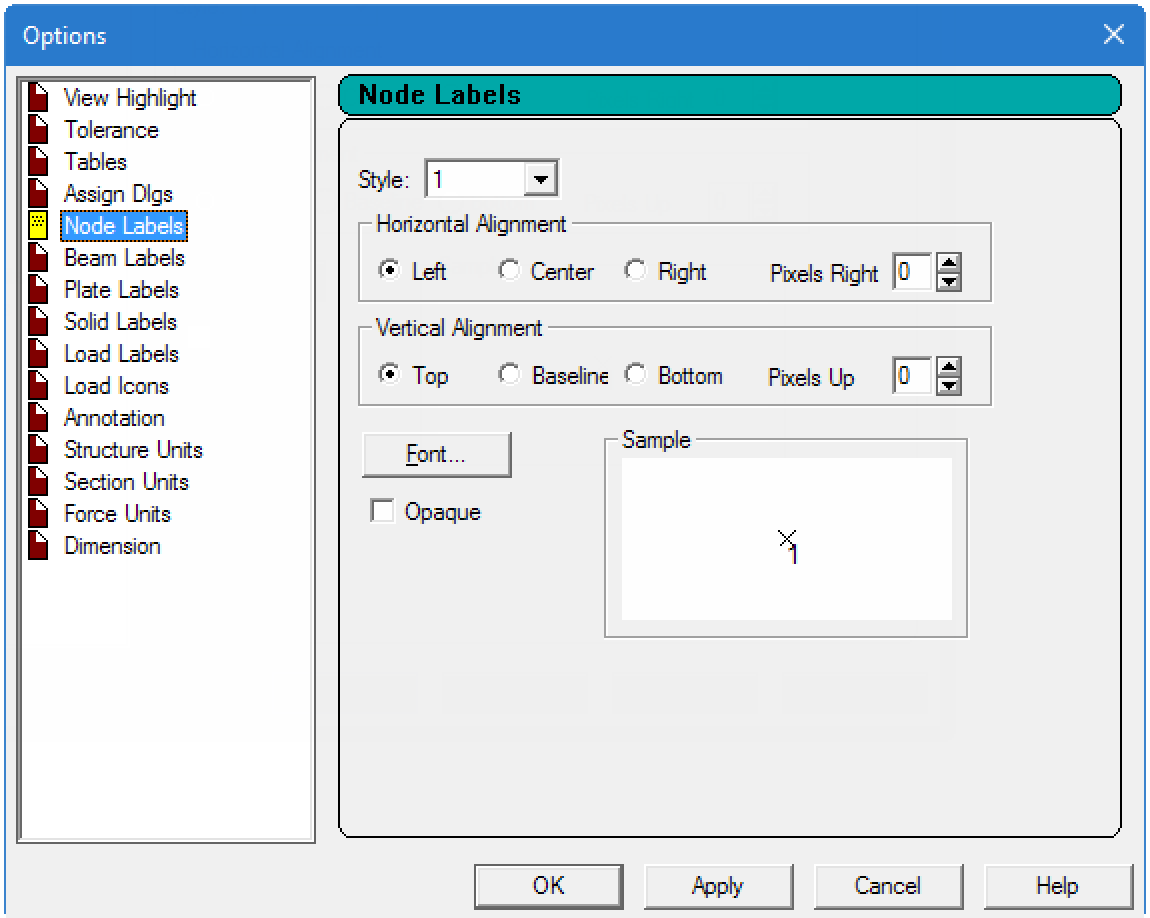


Figure 95: `US.1. Plane Frame with Steel Design.std` *with the beam and node labels displayed*

3. Click **OK** to update the View window and close the dialog.
4. To change the font of the node/beam labels, select **View** > **Options**.
    a. Select the **File** ribbon tab.
    The STAAD.Pro Backstage view opens.
    b. On the **Settings** tab, select **Display Options**.

    The **Options** dialog opens.
    c. Select the appropriate tab (Node Labels / Beam labels) and then click **Font** to make the desired changes.



d. Click **OK** to close the **Font** dialog.

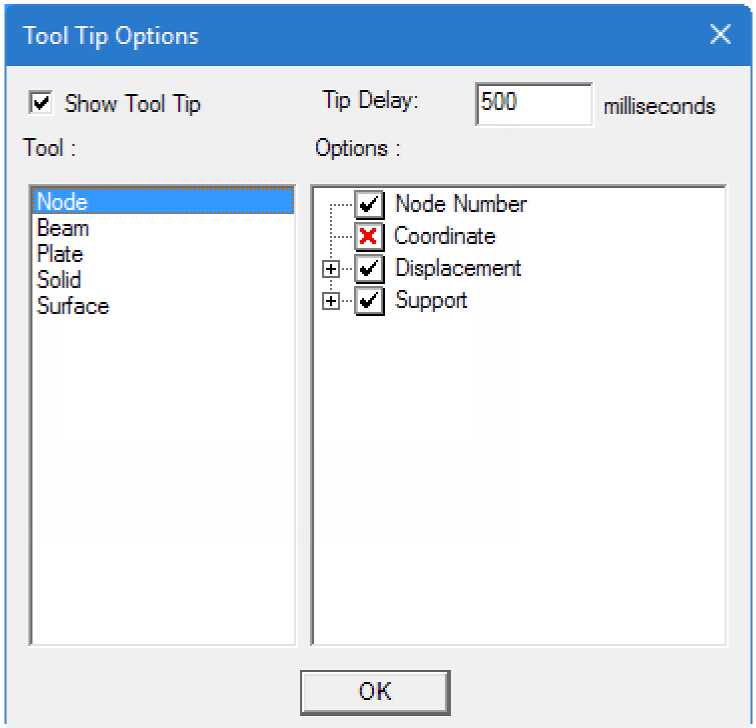
e. Click **OK** to close the **Options** dialog.

M. To change the structural tool tip options

Structural tool tips offer a facility for displaying any customized input or output information about a node, beam, plate or solid element when the mouse cursor is placed over the structural entity. The tool tips are similar to the ones displayed when the mouse cursor hovers over a toolbar icon.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. On the **Settings** tab, select **Structural Tooltip Options**.

   The **Tool Tip Options** dialog opens.



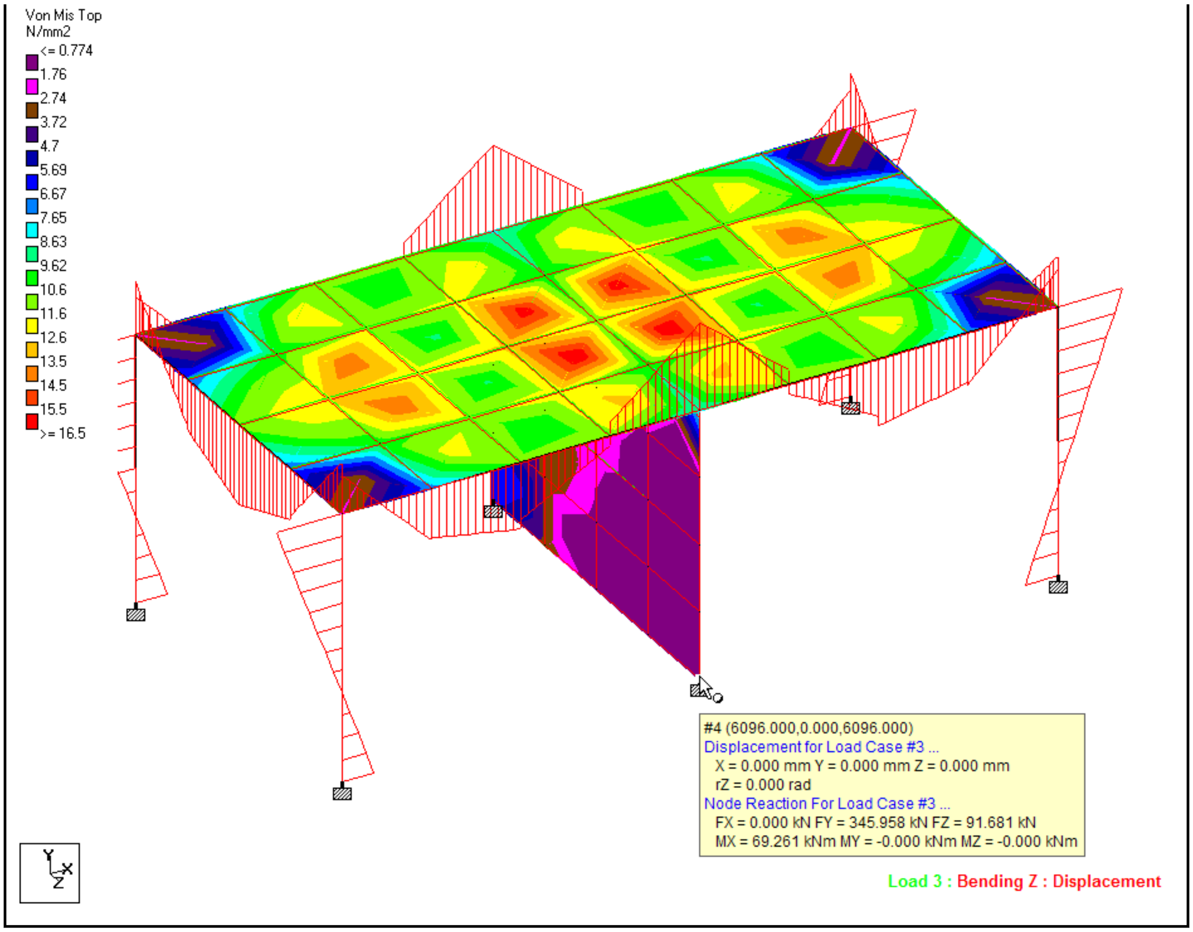
3. Check the **Show Tool Tip** option.
4. (Optional) Change the **Tip Delay**

   This number is expressed in milliseconds (i.e., 1,000 = 1 second).
5. For a selected model entity (Node, Beam, etc. in the left list), set the information and results you want to display in the tool tip in the right list.

   The options (items that can be displayed) for each entity are shown under the **Options** list. A check mark signifies that the particular data item will be displayed in the tool tip. An option with a "+" next to it signifies that further options can be enabled or disabled. A red "X" indicates the data will not be shown in the tool tip. Simply click on the check box to turn an option on or off.
6. Click **OK**.

Example

Structural tool tips can be configured to display a wide variety of information by hovering over a model entity, as shown in the following figure.



Note: The tool tips automatically display the results for the active load case. All values are reported in the current display units.

M. To switch individual object label display

To control the display of individual node, beam, plate, or solid labels, use the following procedure.

These tools activate the Partial Labeling Mode, which allows you to control the display of individual object labels in STAAD.Pro.

1. On the **View** ribbon tab, select one of the following tools in the **Labels** group:

   Individual Nodel Label

   or

   Individual Beam Label

or

Individual Plate Label

or

Individual Solid Label

2. Select the model object that you want to switch the label state.

   The model object must be of the same type as the selected tool.

   If this is the first time you are switching individual labels, you will be prompted that the program will turn on Partial Labeling Mode. Click **Yes** to proceed.

M. Views

You can navitage the view, control the display of model objects, and save named views in STAAD.Pro.

M. Displaying a Portion of the Model

Sometimes, the large number of entities that are drawn on the screen may make it difficult to clearly see the details at any particular region of the structure. In such cases, one is confronted with the task of decluttering the screen or looking at specific regions or entities while removing the rest of the structure from the view.

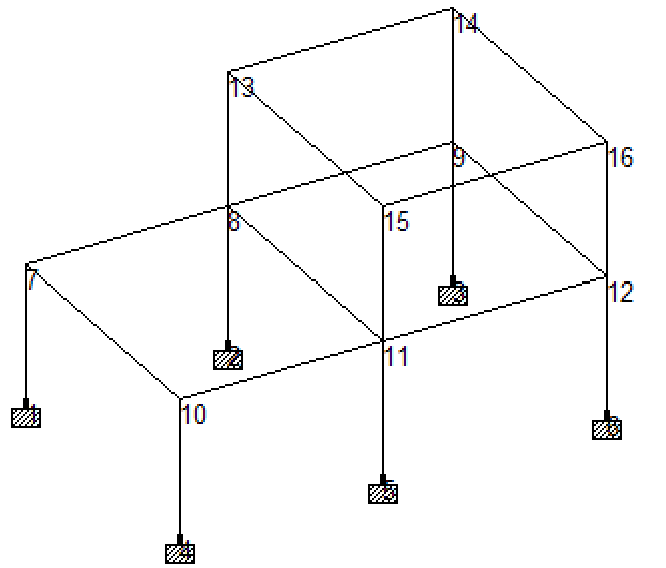
There are different methods in STAAD.Pro by which you can view a portion of the structure.

M. To cut a section of a model

The file `US.8. Concrete Design for a Space Frame.STD` is used to demonstrate this feature.

1. (Optional) Press **<Shift+N>** to display the node numbers.

   This will aide in specifying a node number in the following step.



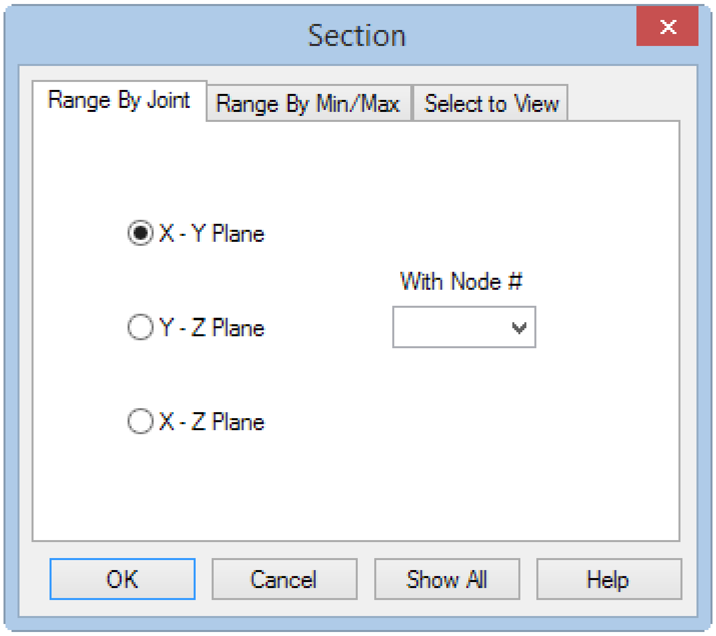
2. Use the cut section tool:
   a. On the **Utilities** ribbon tab, select the **Structure Tools** tool in the **Geometry Tools** group.

   b. Select the **Cut Section** tool from the drop-down list.

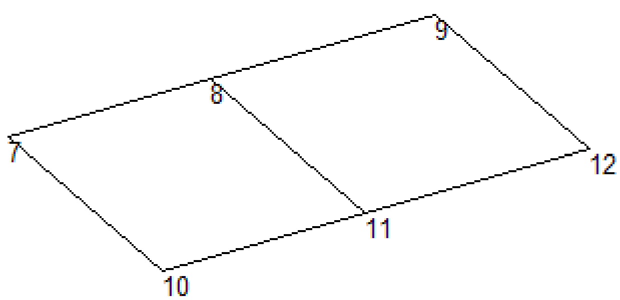
   The **Section** dialog opens.



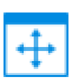
3. Specify a cut plane by joint and plane orientation (range by joint method):
   a. Select the **Range By Joint** tab.
   b. Select the **X - Z Plane** option.
   c. Select **10** from the **With Node #** drop-down list (which contains all node numbers in the model).

   Tip: You can select any node with in this same plane to achieve the same result.

   d. Click **OK**.



4. To restore the original view, select the **Whole Structure** tool in the **Tools** group on the **View** ribbon tab.



Tip: Alternatively, you can click **Show All** on the **Section** dialog.

5. Specify a cut plane by minimum and maximum distances and plane orientation (the range by min/max method):
   a. Repeat step 2 to open the Section dialog.
   b. Select the **Range By Min/Max** tab.

c. Select the **X-Z Plane** option.
d. Type `10` in the **Minimum** field and type `14` in the **Maximum** field.

The **Minimum** and **Maximum** values are the boundary distances along the axis perpendicular to the sectional plane. Every object lying entirely between these two distances will be displayed.

Note: When using this method, make sure that the current input units of length are in the intended units.

6. Repeat step 4 to restore original view.
7. Display only the nodes for quick selection (the Select to View method):

a. Select the **Select To View** tab.

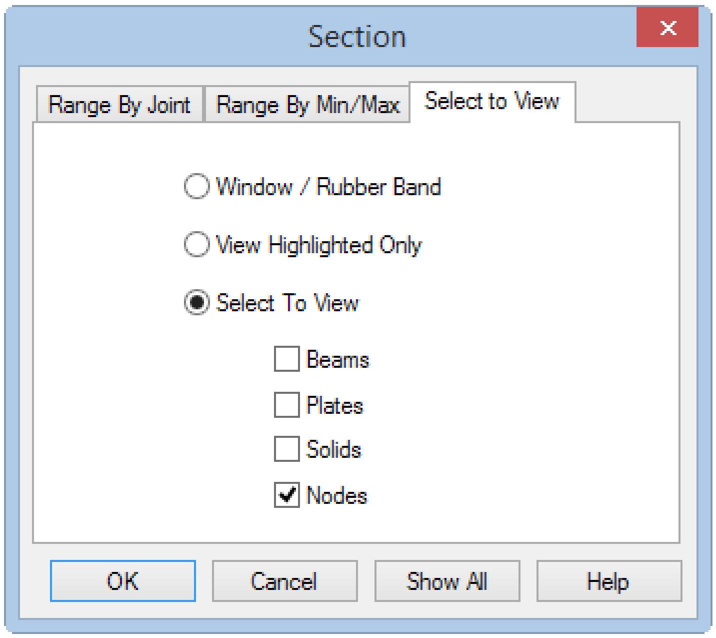


Figure 96:

b. Select the **Select To View** option.
c. Check the **Node** option.
d. Click **OK**.

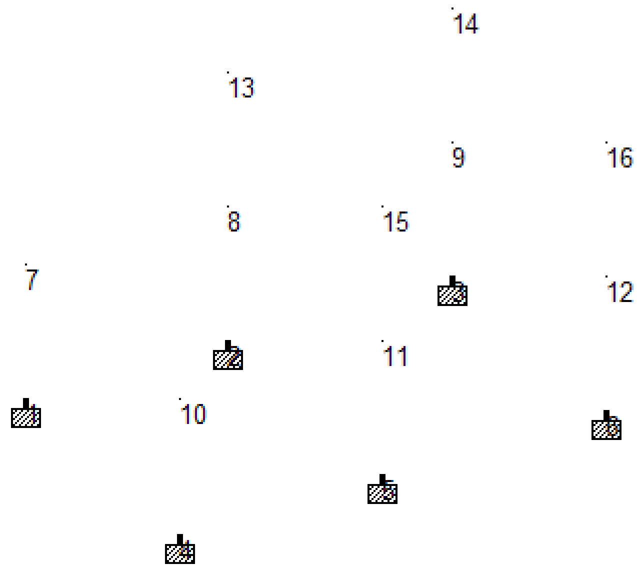


Figure 97: The nodes only displayed for

8. Select the **Whole Structure** tool to restore original view.

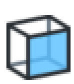
Note: You can save these views by selecting the **View Management** > **Save View** tool in the **Views** group on the **View** ribbon tab. Provide a title for the new view. These saved views may later be opened by selecting the **Open View** tool in the **Views** group on the **View** ribbon tab.

M. To create a new view

The file `US.8. Concrete Design for a Space Frame.STD` is used to demonstrate this feature.

New view windows are helpful for performing such operations as adding and deleting members, assigning properties, loads, supports, and more. A new view of a selected portion offers the advantage of de-cluttering the screen and limiting the displayed objects to just a few chosen entities.

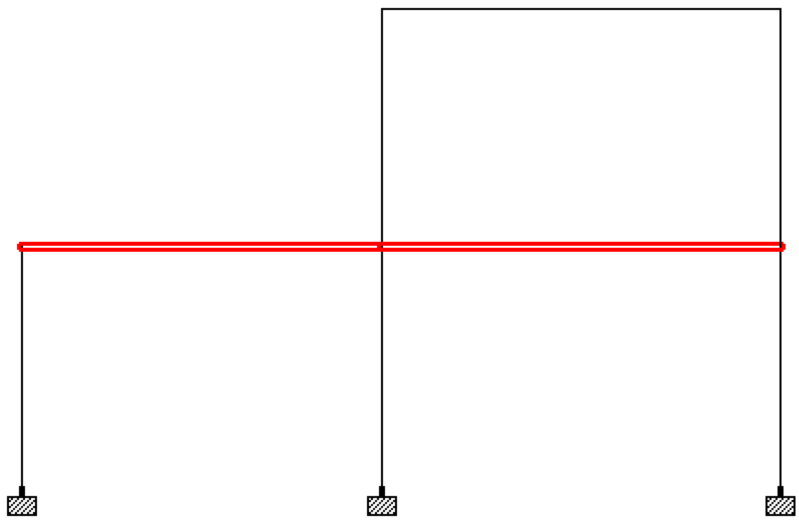
1. On the **View** ribbon tab, select the **Front View** tool in the **Tools** group.



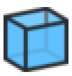
2. In the view window, drag a window area around the middle level of beams to select them.

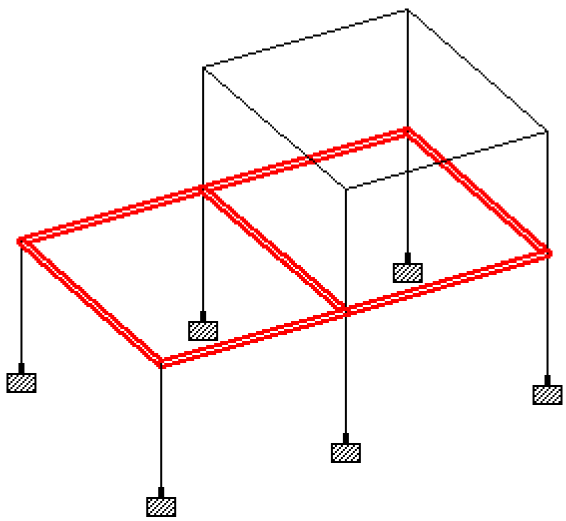
Tip: Make sure that the **Beam Cursor** tool  is the active selection tool on the **Select** ribbon tab in the **Cursors** group.

The selected members are highlighted.



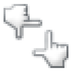
3. (Optional) On the **View** ribbon tab, select the **Isometric View** in the **Tools** group.





4. Either:

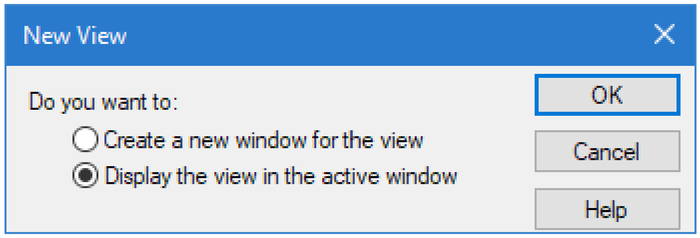
   On the **View** ribbon tab, select the **New View** tool in the **Views** group.

   or

   Right-click anywhere in the current view window and select **New View** from the pop-up menu.

The **New View** dialog opens.



The dialog includes options open the new view in a new (child) window or to replace the current (parent) view window.

5. Select the **Create a new window for the view** option and then click **OK.**

   The portion of the structure that we selected is displayed in a new window.

Tip: Multiple child view windows may be created in this way.

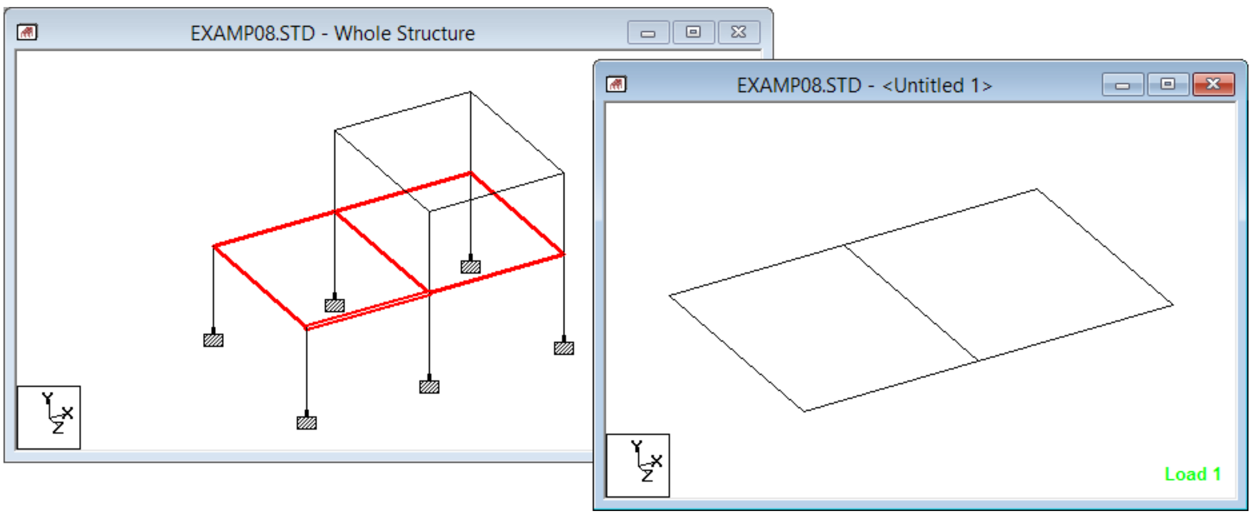


Figure 98:

6. Save the view for later use:
   a. On the **Views** ribbon tab, select the **View Management Save View** tool in the **Views** group.

   The **Save View As** dialog opens.
   b. Type a name for the view.
   c. Click **OK.**

   The view window is titled with the view name. You can
7. Close the new view window by clicking close **[X]** in the top-right corner.
8. Repeat steps 2 through 4 to start another New View.
9. Select the **Display the view in the active window** option and click **OK.**
   The full structure is hidden and the selected portion is displayed in the View window.
10. To restore the original view, select the **Whole Structure** tool in the **Tools** group on the **View** ribbon tab.

Note: You can save these views by selecting the **View Management** > **Save View** tool in the **Views** group on the **View** ribbon tab. Provide a title for the new view. These saved views may later be opened by selecting the **Open View** tool in the **Views** group on the **View** ribbon tab.

M. To restore the window layout

To restore the window arrangement for the current workflow page, use the following procedure.

If you have closed some of the layout tables, dialogs, or view windows you can quickly restore the default layout.

1. On the right-hand side of the page control bar, select the **Restore View** tool.
   The default window layout is opened.

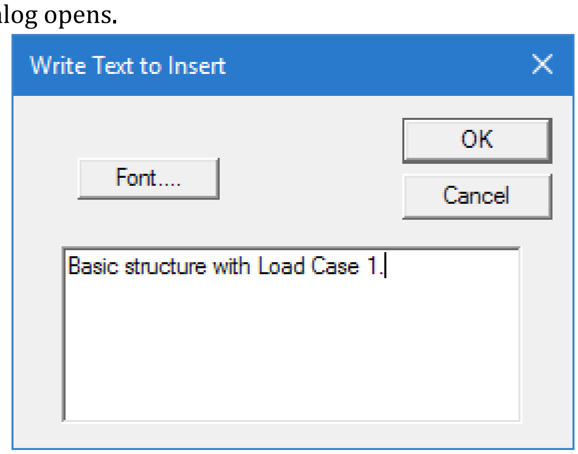
M. To insert custom text in a view

To add custom text to the view window and, subsequently, pictures taken for reports, use the following procedure.

The text can serve as comments or titles to pictures and result diagrams. The added text can be plotted too.

1. Select the **Insert Text** tool in the **Display** group on the **Utilities** ribbon tab.

   The mouse pointer changes to the text cursor ( ).
2. Click in the view window where you want to insert the text.
   The **Write Text to Insert** dialog opens.



3. (Optional) Click **Font** to set the type display.
4. Type the text into the text field and then click **OK**.
5. Either:

   Repeat steps 2 through 4 to insert additional text items.

   or

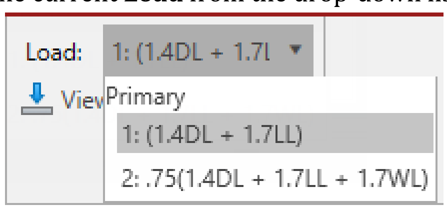
   Repeat step 1 to turn off the insert text tool.

6. (Optional) To modify previously inserted text:
   a. Select the **Text Cursor** tool in the Selection group on the **Geometry**, **Results**, or **Member Design** ribbon tabs.
   b. Double-click on the text you want to modify.
      An edit dialog opens with the existing text.
   c. Make changes and then click **OK**.
   d. Click-and-hold the left mouse button on the text to move it to another location in the view window.
7. (Optional) To delete previously inserted text:
   a. Select the **Text Cursor** tool in the Selection group on the **Geometry**, **Results**, or **Member Design** ribbon tabs.
   b. Click on the text you want to delete.
   c. Press **<Delete>**.

M. To display loads graphically

You can display the load diagrams and load values on the model.

1. On the **Loading** ribbon tab, select the current **Load** from the drop-down list in the **Display** group.



> **Tip:** The current load case number is displayed in the lower, right-hand side of the view window as well as in the application status bar.

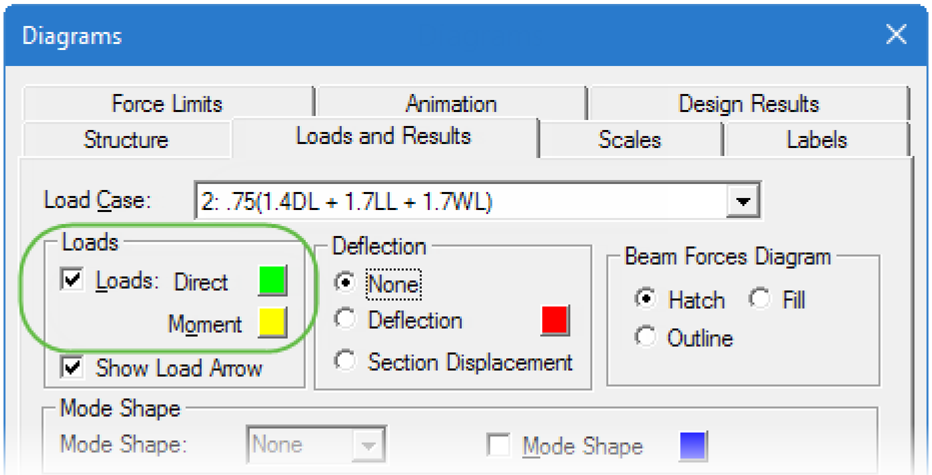
2. Either:

   Select the **Labels** tool 

   or

   Right-click anywhere in the View area and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens.
3. Either:
   a. select the **Loads and Results** tab.
   b. Check the **Loads** option.



c. (Optional) Click the color squares for Direct or Moment to change the load pattern colors.

d. Click **Apply**.
The loads are graphically displayed on the model.

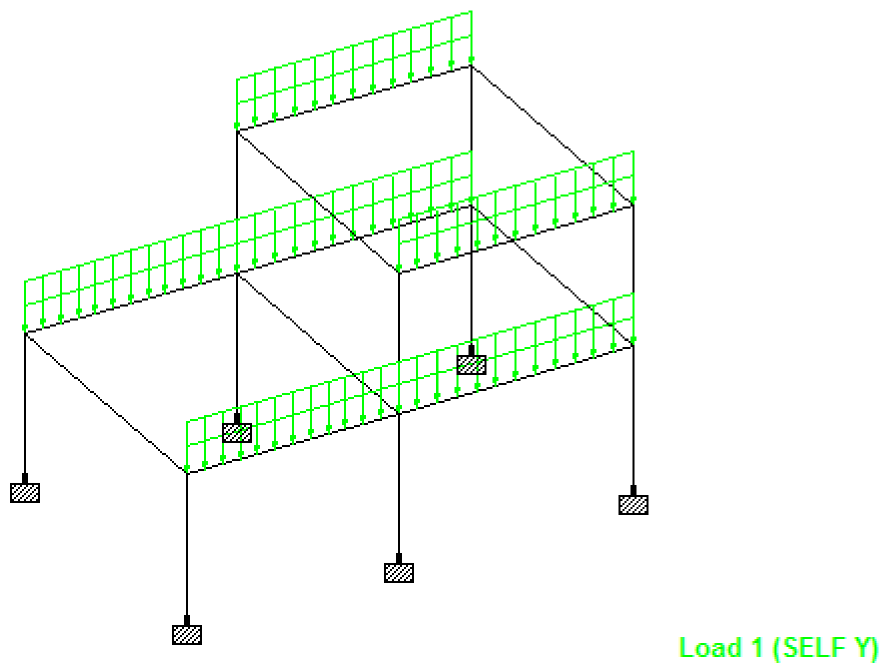


Figure 99: Load case 1 displayed on `US.8. Concrete Design for a Space Frame.std`

4. a. Select the **Labels** tab.
   b. Set the **Load Values** check box in the **Loading Display Options** group.
   c. Click **OK**.

The load values are displayed for the current load case, as seen here for the file `US.8. Concrete Design for a Space Frame.STD`.

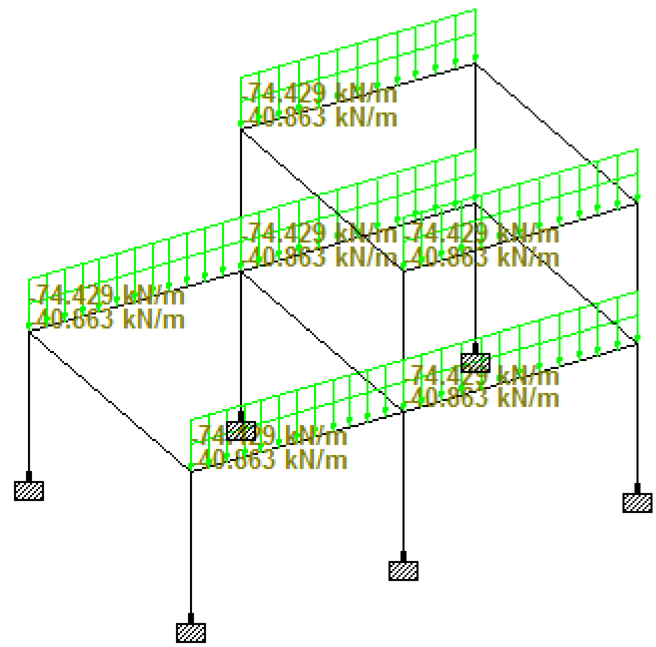
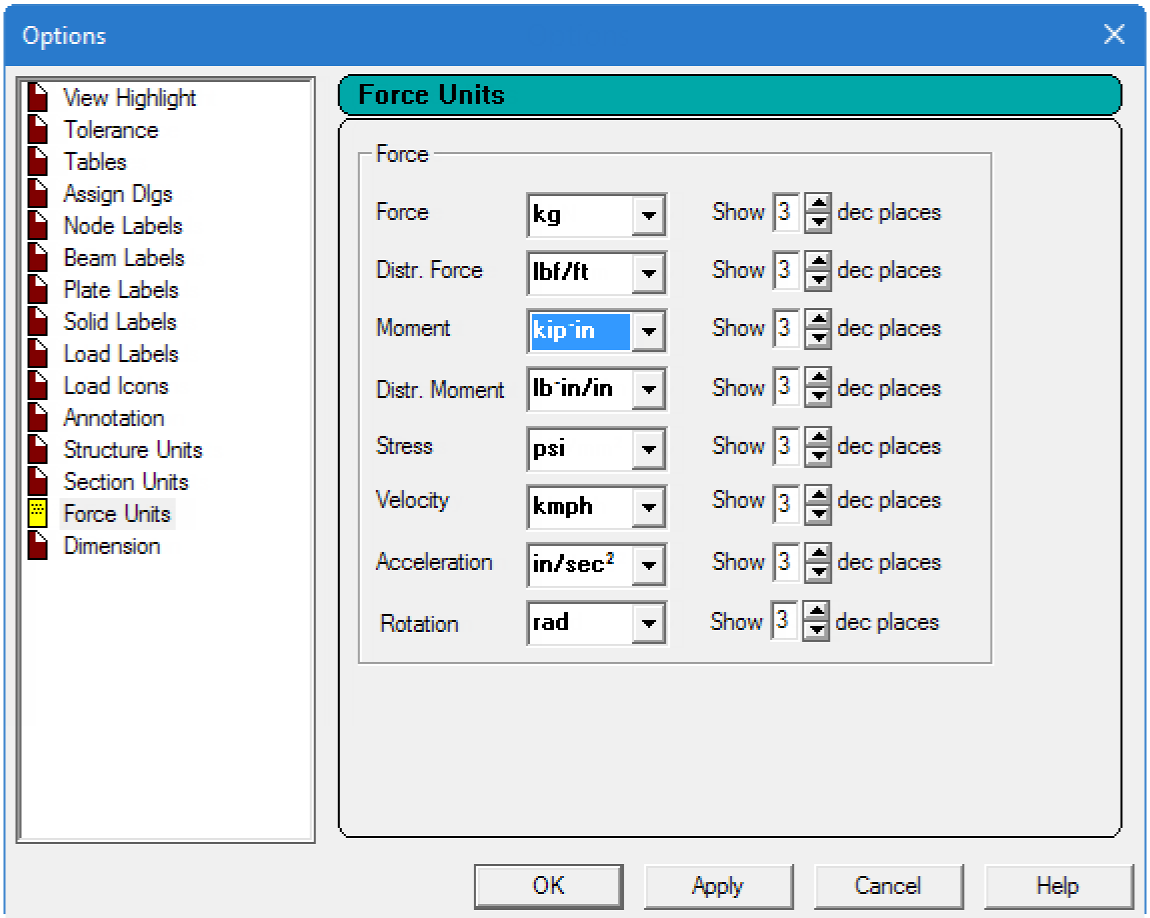


Figure 100:

5. (Optional) To change the units displayed:
   a. Select the **File** ribbon tab.
      The STAAD.Pro Backstage view opens.
   b. On the **Settings** tab, select **Display Options**.

      The **Options** dialog opens.
   c. Select the **Force Units** tab.

      The **Options** dialog opens.



d. Select the **Force** and **Distributed Force** values as needed.

e. Click **OK**.

The values are updated with the new units.

M. To identifying beam start and end

Beam ends can be colored to easily identify the start and end joints of a beam. By default, the start (i) joint is colored green and the end (j) joint is colored blue.

1. Either:

   Select the **Labels** tool 

   or

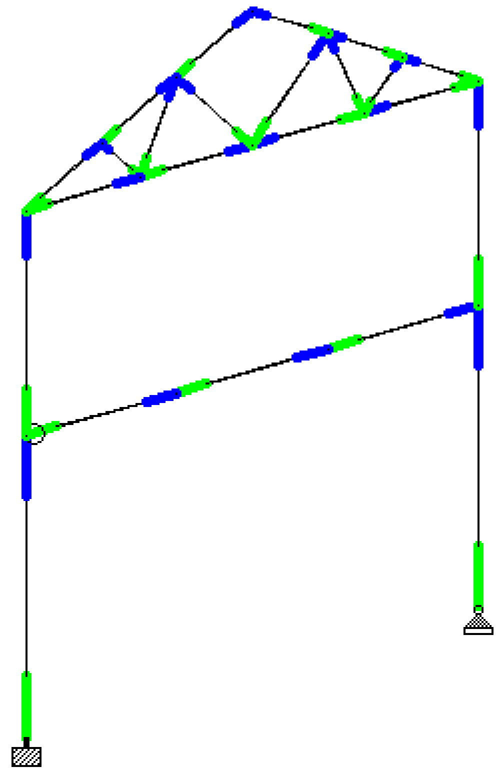
   Right-click anywhere in the View area and select **Labels** from the pop-up menu.

   The **Diagrams** dialog opens.
2. Check the **Beam Ends** option in the **Beams** group.
3. Click **OK**.

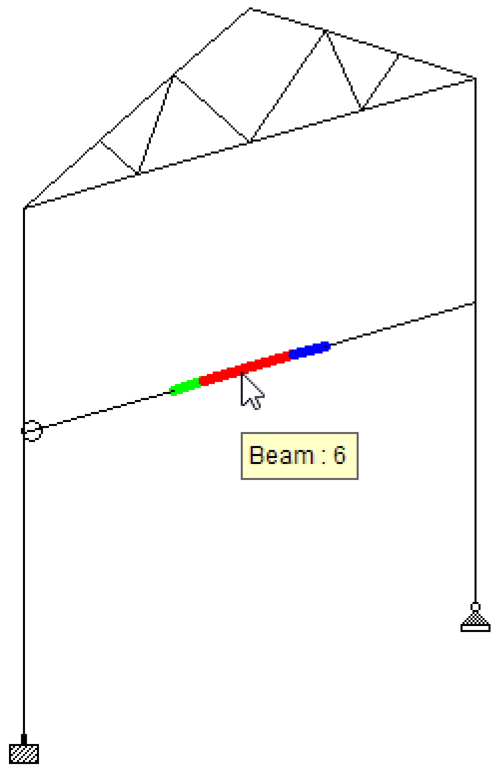
Tip: Alternatively, you can press **<Shift+E>** to toggle the beam end colors.

The beam ends are colored for the entire structure.

The file `US.1. Plane Frame with Steel Design.STD` is used to demonstrate this feature.

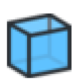
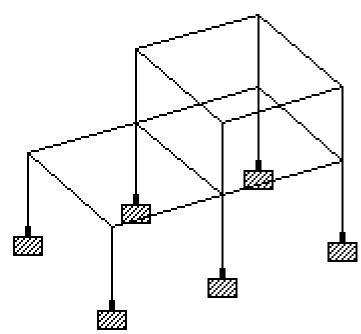


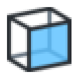
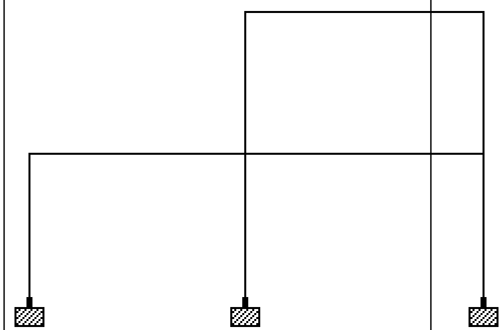
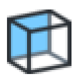
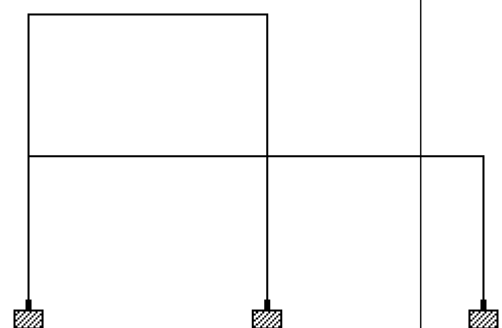
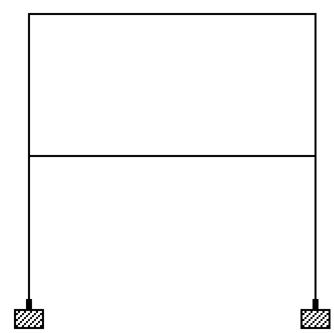
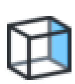
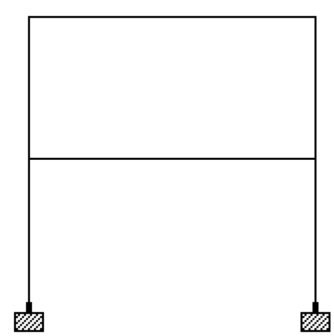
Alternatively, you can hover your mouse over any beam momentarily and the beam ends for that member are displayed using the same colors.

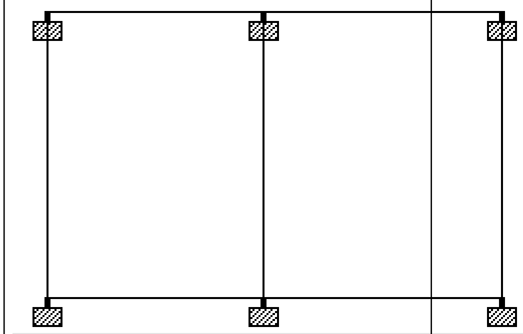
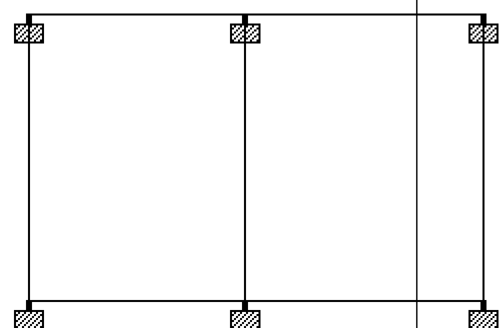


M. Rotation tools

These tools found on the **View** ribbon tab can rotate your model to a predefined orientation.

Tool name	Description	Example
 **Isometric View**	Displays the structure in the isometric view. The angle which defines isometric view is generally X = 30, Y = 30, Z = 0	

Tool name	Description	Example
 **Front View**	Displays the structure as seen from the positive Z axis. When the global Y axis is vertical, this is the elevation view, as looking towards the negative direction of the Z-axis.	
 **Back View**	Displays the structure as seen from the negative Z axis. When the global Y axis is vertical, this is the elevation view, as seen looking towards the positive direction of the Z-axis.	
 **Left View**	Displays the structure as seen from the negative X axis. When the global Y axis is vertical, this is the side elevation, as seen looking towards the positive direction of the X-axis.	
 **Right View**	Displays the structure as seen from the positive X axis. When the global Y axis is vertical, this is the side elevation, as seen looking towards the negative direction of the X-axis.	

Tool name	Description	Example
 **Top View**	Displays the structure as seen from the top looking down (positive Y axis). When the global Y axis is vertical, this is the plan view, as seen from the sky looking down.	
 **Bottom View**	Displays the structure as seen from the bottom looking up (negative Y axis). When the global Y axis is vertical, this is as if one is looking up skywards.	

M. To display control nodes

To highlight control nodes in a model that contains control/dependent joints, use the following procedure.

Tip: Open example 26 to demonstrate this procedure for yourself.

1. Either:

   on the **View** ribbon tab, select the **Label Settings** tool in the **Labels** group

   or

   in the view window, right-click and select **Labels** from the pop-up menu

   The **Diagrams** dialog opens to the Labels tab.
2. Check the option from **Control/Dependent** in the General group.
3. Click **Apply**.

   Tip: Press **<Shift+L>** to highlight control nodes quickly.

   The control nodes are highlighted with green cubes.

4. Click **OK**.

Related Links

- *Diagrams dialog* (on page 2751)
- *EX. US-26 Modeling a Rigid Diaphragm Using Control-Dependent* (on page 6517)
- *EX. UK-26 Modeling a Rigid Diaphragm Using Control-Dependent* (on page 6804)

M. Creating Model Objects

This section describes the tools used in STAAD.Pro to model your structure.

M. Drawing Aids

M. To add a grid for drawing objects

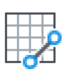
To add a grid for drawing beams or plates, use the following procedure.

Only one grid may be displayed at a time. However, you may have multiple grids associated with your model.

Note: Grids added to the model will be available to use in both the **Snap Node/Beam**, **Snap Node/Plate**, and **Snap Node/Solid** dialogs.

1. Either:

   On the **Geometry** ribbon tab, select the **Grids** > **Beam Grid** tool in the **Structure** group.

   or

   On the **Geometry** ribbon tab, select the **Grids** > **Triangular Plate Grid** tool in the **Structure** group.

   or

   On the **Geometry** ribbon tab, select the **Grids** > **Quad Plate Grid** tool in the **Structure** group.

   or

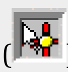
   On the **Geometry** ribbon tab, select the **Grids** > **Solid Grid** tool in the **Structure** group.

The corresponding **Snap Node/Beam**, **Snap Node/Plate**, or **Snap Node/Solid** dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.

2. Click **Create**.
   A pop-up dialog opens for defining grid details.
3. Select the type of grid from the drop-down list.

Grid type	Description
Linear	a Cartesian grid with even spaces along both of the axis.
Radial	a polar grid with even spaces between rays and radial lines.
Irregular	a Cartesian grid with irregular spaces along both of the axis.

   The pop-up dialog fields update with the grid type selection.
4. Type a grid **Name** to identify the grid in the Snap Node/Beam dialog.
5. Specify the location of the grid with respect to the global axes:
   a. Select the parallel global **Plane** for alignment (or reference when an angle is specified).
   b. (Optional) Select a global axis and then type an **Angle of Plane** to incline the grid with respect to that axis.
   c. (Optional) (For Irregular grids) Set the option to Use Arbitrary Plane and then type the vectors of the grid X and Y axis.
      This method can be used instead of specifying the Plane and Angle of Plane.
   d. Type global coordinates from the global origin to the **Grid Origin**.

> **Tip:** Click the node selection tool () and then click on an existing node in the model to use as the grid origin.

6. Specify the **Construction Lines** for the grid:

Grid type	Description
Linear	For both grid X and Y, type the number of grids spaces to the **Left** and **Right** of the origin, the **Spacing** (in current units) between each grid line, and (optionally) a **Skew** angle.
Radial	Type the **Start Angle** and **Sweep** to specify the beginning and end of the radial lines along with the number of bays (spaces between radial lines). Type the inner-most and outer-most radius in the **Radius 1** and **Radius 2**, respectively, along with the number of **Bays** (spaces between arcs).
Irregular	Type the distance between each consecutive grid line in the current units along each axis.

7. Click **OK**.
   The pop-up dialog closes and the new grid name is added to the list.
8. Check the name of the grid in the list that you want displayed in the view window.

Click **Edit** to make changes to the currently selected grid in the list. You can change the grid orientation and construction lines but the type of grid cannot be changed.

Related Links

- *Snap Node-Beam dialog* (on page 2719)
- *Linear* (on page 2721)
- *Radial grid dialog* (on page 2721)
- *Irregular grid dialog* (on page 2722)

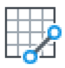
M. To import a STAAD.Pro grid file

To import a STAAD.Pro grid file which was contains grids used in another model, use the following procedure.

Any time you save a model with custom grids, the grid data is saved into a STAAD.Pro grid file (file extension `.grd`) with the same file name and location as the STAAD input file.

1. Either:

   On the **Geometry** ribbon tab, select the **Grids** > **Beam Grid** tool in the **Structure** group.

   or

   On the **Geometry** ribbon tab, select the **Grids** > **Triangular Plate Grid** tool in the **Structure** group.

   or

   On the **Geometry** ribbon tab, select the **Grids** > **Quad Plate Grid** tool in the **Structure** group.

   or

   On the **Geometry** ribbon tab, select the **Grids** > **Solid Grid** tool in the **Structure** group.

   The corresponding **Snap Node/Beam**, **Snap Node/Plate**, or **Snap Node/Solid** dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.
2. Click **Import**.
   The **Import Options** dialog opens.
3. Select **STAAD.Pro Grid (.grd)** and then click **OK**.
   An Windows **Open** dialog opens.
4. Select the STAAD.Pro grid file and then click **Open**.
   The grids in the file are added to the Snap Node/Beam dialog.

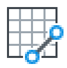
M. To import a DXF file as a grid

To import an AutoCAD® DXF file to use as a drawing grid, use the following procedure.

The straight lines within a DXF file can be imported as a grid. Complex shapes and curved lines will not be imported.

1. Either:

   On the **Geometry** ribbon tab, select the **Grids** > **Beam Grid** tool in the **Structure** group.



or

On the **Geometry** ribbon tab, select the **Grids** > **Triangular Plate Grid** tool in the **Structure** group.

or

On the **Geometry** ribbon tab, select the **Grids** > **Quad Plate Grid** tool in the **Structure** group.

or

On the **Geometry** ribbon tab, select the **Grids** > **Solid Grid** tool in the **Structure** group.

The corresponding **Snap Node/Beam**, **Snap Node/Plate**, or **Snap Node/Solid** dialog opens. If an existing grid is selected in this dialog, this grid is displayed in the view window.

2. Click **Import**.
   The **Import Options** dialog opens.
3. Select the **DXF** option and click **OK**.
   The **DXF Import** dialog opens.
4. In the File Name field, either:

   type the file path and file name of the DXF file

   or

   click **[...]** locate the file using the **Open DXF File** dialog

   The layers and line data is read and displayed graphically.
5. Select the layer number containing the line data you want to use from the **Layers** drop-down list.
   The lines are displayed in the list view and graphically.
6. (Optional) Uncheck any lines you do not want to include in the imported grid.
   The lines are hidden in the view window.
7. (Optional) Type a **Name** for the imported grid.

   By default, the name of the DXF file is used.
8. Select the **Unit** to use for the grid.

   Tip: The drop-down list contains most common length units. Use the up or down arrows to navigate through the options.

9. Click **OK**.
   The Imported Grid pop-up dialog opens.
10. (Optional) Specify the **Angle of Plane** and **Grid Origin** to orient the grid.
11. Click **OK**.

When you save your model, the imported DXF grid will be saved to the corresponding STAAD.Pro grid file (file extension `.grd`).

Related Links

- *Imported Grid dialog* (on page 2724)
- *DXF Import dialog* (on page 2723)

M. Beams

M. To set attributes for new beams

To specify a set of attributes to automatically assign to new beams, use the following procedure.

For member profiles and member end releases, you will need to add the corresponding specification to your model before you can select it for most attribute sets.

Tip: The predefined attribute sets have some pre-populated values that will be added to your model automatically, though.

In STAAD.Pro, you will often assign attributes (beta angle, end offsets, material, profile, etc.) after you have placed a member. However, you can also assign attribute sets when creating new members by defining these ahead of time. STAAD.Pro allows you to create and edit named attribute sets so you can easily change between attribute sets.

Note: This feature is not active by default so new members will not have any attributes assigned initially.

1. On the **Geometry** ribbon tab, select **Add Beam** > **Set New Beam Attributes** in the **Beam** group.
   The **Define Member Attributes** dialog opens.
2. Select an attribute set in the list.

   Tip: There are several named sets which can be used or you can simply use the default “(None)” set.
3. Check the box for each attribute you want assigned.

   You can check multiple items. Each item has one or more options to select once you have checked to assign it.

Attribute	Description
Member Property	Check to **Assign Profile with Material** and then select the profile and associated material from the drop-down list.
Beta Angle	Check to **Assign Member Rotation** and then select the option for rotation from the list. Type in the **Angle in Degrees** if you select that option.
Member Release	Check to **Assign to Member Start** and **Assign to Member End** as needed. Select a end release specification from the drop-down list for each end you have added a member release.
4. Click **Save**.
   The attribute set is updated with the new specification selections.
5. Check the option to **Assign these attributes when creating members**.

   Note: This option can only be checked for one attribute set.
6. Click **OK**.

Related Links

- *Define Member Attributes dialog* (on page 2729)

M. To add beams by drawing on a grid

1. On the **Geometry** ribbon tab, select the **Grids** > **Snap Grid Beam** tool in the **Structure** group.

   The Snap Node-Beam dialog (on page 2719) opens and the current grid is displayed in the active View window.

   Tip: The dialog and grid are automatically opened if you select Start adding beams in the New model wizard.
2. (Optional) Either:

   click **Edit** to edit the currently selected grid

   or

   click **Create** to create a new custom grid (radial and irregular grids must be custom)
3. (Optional) Set the **Snap to existing nodes too**

   Note: Existing nodes have precedence over grid intersections, in order to prevent creating M. To check for and remove duplicate entities (on page 914).
4. If it is not already active, click **Snap Node/Beam** in the Snap Node/Beam dialog.
5. In the Active view window, click any grid intersection or existing node (if the option was set) to select the start point of a new beam member.

   A cross icon "snaps" to the nearest grid line intersection or existing node to the mouse pointer. This indicates the tentative point for placing a beam end. Once the start end is selected, a red circle indicates the start point and a line is rubber-banded to the snap point nearest the mouse pointer.
6. Click the end point of the beam.
7. Repeat steps 5 and 6 until you are finished drawing beams.
8. To stop drawing beams, either:

   press **<Esc>**

   or

   click **Snap Node/Beam** in the Snap Node/Beam dialog

M. To add beams with new nodes

To add beams between existing nodes or along beams by generating new nodes, use the following procedure.

1. On the **Geometry** ribbon tab, **Add Beam** tool in the **Beam** group.

   The mouse pointer changes to the Add Beams cursor ().
2. Click on any point along the first beam where the starting node of the new beam will lie.

   If the program detects you are clicking on an existing node, skip to Step 6.

   If an existing node is not present at that point, a dialog box will prompt for a new node to be created.

3. Click **Yes**.
The **Insert Nodes into Beam #** dialog opens. The value in the **Distance** edit box is the exact location clicked on from the start of the beam.
4. (Optional) Edit the **Distance** value as necessary.

> **Tip:** Other nodes can also be generated along the beam as well. For more information on how to use the **Insert Node** dialog box, please refer to M. To insert a node in a single member (on page 682).

5. Click **OK**

   A new node is created on the selected beam and a the new beam member is “rubber banded” to this point.

> **Tip:** If the new node input is not within a close proximity of the point clicked on the screen (that which resulted in the dialog opening), no “rubber band” line will be shown. In this case, simply click on the new node to start the creation of the beam.

6. Click a point along another beam or an existing node to specify the end point of the new beam.

   If this is not at an existing node, refer to steps 2 through 5 for inserting a new node into a beam.
7. To stop adding beams:

   Press **<Esc>**

   or

   select any tool

Related Links
- *TR.12 Member Incidences Specification* (on page 2221)

M. To add beams from mid-points

1. On the **Geometry** ribbon tab, select the **Add Beam** > **Add Beam Between Mid-Points** tool in the Beams group.

   The mouse pointer changes to the Add Beams cursor ().
2. Click on any point along the first beam where the starting node of the new beam will lie.
A new beam member is "rubber banded" to this point.
3. Click on any point along the second beam where the end node of the new beam will lie.

   New nodes are created on the selected beams at their mid-points and a the new beam member connects these two new points.
4. Repeat steps 1 through 3 to continue adding beams.
5. To stop adding beams:

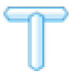
   Press **<Esc>**

   or

   select any tool

M. To add beams perpendicular to existing beams

1. On the **Geometry** ribbon tab, select the **Add Beam by Perpendicular Intersection** tool in the **Beam** group.

   The mouse pointer changes to the Add Beams cursor ().
2. Click on any point along the first beam where the starting node of the new beam will lie.
   A new beam member is "rubber banded" to this point.
3. Click on any point along the second beam where the end node of the new beam will lie.
   A new node is created on the selected beam at the at the calculated perpendicular point and a the new beam member is added.
4. Repeat steps 2 and 3 to continue adding beams.
5. To stop adding beams:

   Press **<Esc>**

   or

   select any tool

M. To add a curved beam

To add a curved beam between two existing nodes, use the following procedure.

Curved members may only be created between two existing nodes. Any non-tapered cross-section is permitted for curved members. The internal angle subtended by the arc must be less than 180 degrees.

Note: The design of curved members is *not* supported.

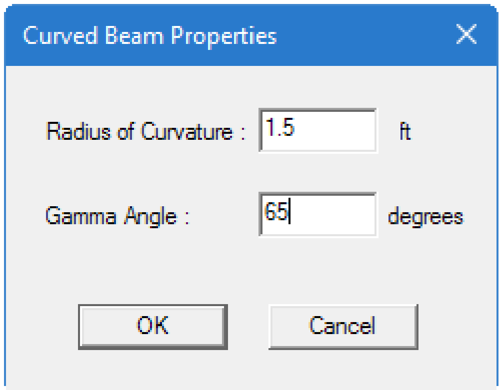
1. On the **Geometry** ribbon tab, select the **Add Beam** > **Add Curved Beam** tool in the **Beam** group.

   Tip: This tool is contained on a drop-down list below the **Add Beam** tool.

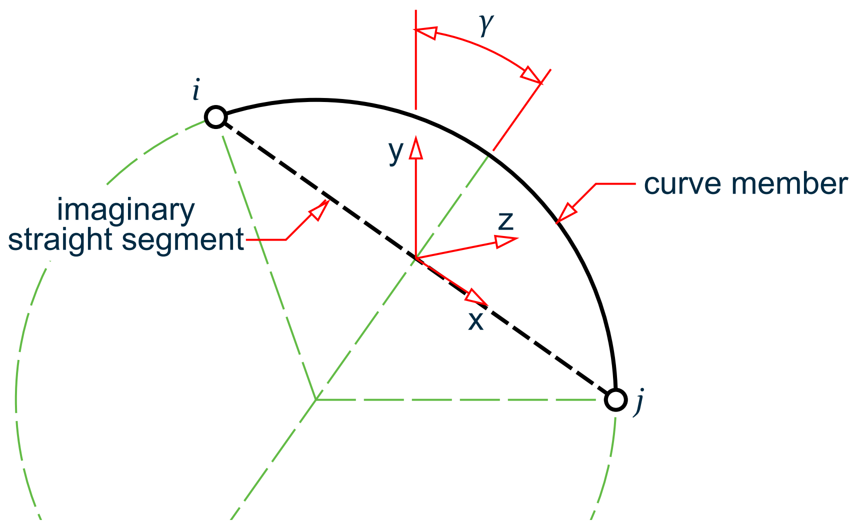
   The pointer changes to the Add Curved Beam cursor ().
2. Click on the start and end nodes, respectively.

   The **Curve Beam Properties** dialog opens.



3. Type the **Radius of Curvature** (in current units of length) and the **Gamma Angle** (in degrees) values.

   The gamma angle, γ , is the angle between the member local Y axis and the plane of the circular arc of the member.



4. Click **OK**.
   The curved beam will is added to the model.
5. Repeat steps 2 through 4 to add more curved beams.
6. To stop adding beams:

   Press **<Esc>**

   or

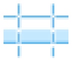
   select any tool

Related Links

- *G.6.8 Curved Members* (on page 2120)
- *TR.20.8 Curved Member Specification* (on page 2273)

M. To split a beam at selected node(s)

1. Select one or more nodes located along a beam.
2. On the **Geometry** ribbon tab, select the **Break Beams at Selected Nodes** tool in the **Beam** group.



A message box appears which displays the number of new beams created.

3. Click **OK**
   The dialog closes.

M. To stretch a member

1. Select one or more members which are to be extended.
2. On the **Geometry** ribbon tab, select the **Geometry** > **Stretch Selected Member(s)** tool in the **Beam** group.

the Stretch Member(s) dialog (on page 2732) opens.

3. (Optional) Click the drop-down list in the **Select member(s)** and then uncheck any members you do not wish included in the stretch operation.
4. Specify the method of stretch and required parameters:

   Select **To a point** and then type the coordinates of an arbitrary point

   or

   Select **Through a distance** and then select the member end and distance to extend to

   or

   Select **To an existing node** and then select the node number

   Tip: Click the tool to graphically select a node.

   or

   Select **To an existing member** and then select the member number

   Tip: Click the tool to graphically select a member.
5. Click **OK**.

Related Links

- *Stretch Member(s) dialog* (on page 2732)

M. To merge two or more members

To merge two or more members into a single member, user the following procedure.

1. Select the beams to be merged.
2. On the **Geometry** ribbon tab, select the **Merge Selected Beams** tool in the **Beam** group.

The **Merge Selected Beams** dialog opens.
3. Specify the member properties the merged member will have.
4. Click **Merge**.

Related Links
- *Merge Selected Beams dialog* (on page 2733)

M. To renumber selected beams

To renumber the currently selected members, use the following procedure.

1. Select the beams to be renumbered.
2. On the **Geometry** ribbon tab, select the **Renumber Beams** tool in the **Beam** group.

A message dialog opens to warn you that renumbering may not be undone.
3. Click **Yes** to confirm you wish to proceed.
The **Renumber** dialog opens.
4. (Optional) Specify a **Start number from**
5. (Optional) Select the **New Numbering Order**
6. (Optional) Select the sort criteria to be used and add them to the **Selected Sort Criteria**
7. (Optional) Reorder the items in the **Selected Sort Criteria**
For example, select the Z Coordinate and click **Move Up** twice so the order from top to bottom is: Z Coordinate, X Coordinate, Y Coordinate. This instructs the program to renumber beams parallel to the Y axis until no more beams are in that line. It then skips to the next selected entity along the X axis and renumbers those, again along the Y axis. Once it has reached the end of selected beams parallel to the X axis, it then skips to the next selected entity parallel to the Z axis and repeats the previous procedures.
8. Click **Accept**.
A message dialog opens with the status of the renumbering operation.
9. Click **OK** to dismiss the message dialog.
The selected beams are renumbered following the indicated pattern.

Related Links
- *Renumber dialog* (on page 2728)

M. Physical Members

Physical members are composites of one or more analytical members which represent the true physical geometry of a member in a structure.

Note: Physical members use the nomenclature `PMEMBER` in the STAAD input file.

About Physical Members

Beam objects connecting two nodes in a structural model are referred to as analytical beams as they are the fundamental, one-dimensional object processed by the STAAD engine during analysis. While necessary for this purpose, they are often difficult elements to use for design as they do not possess the same geometry of a physical structural element. For example, a girder is not physically broken where a beam frames, but a node must be placed to represent the connection in the model. The design of this girder is then complicated as the bracing conditions and other parameters of design can not be directly interpreted by the program.

In this case, STAAD.Pro has the facility to use group one or more analytical beam objects into physical members for the purpose of design. This concept is used in both the Steel Design and Concrete Design workflows. Additionally, physical members can be created in the modeling mode for the purpose of integrated steel design.

Related Links

- *TR.16.2 Physical Members* (on page 2238)
- *M. To assign catalog section to physical members* (on page 718)

M. To manually form physical members

To form a physical member for a selected set of analytical members, use the following procedure.

The following are required to form a physical member:

- analytical members must be interconnected
- analytical members are collinear
- the local axes of the component members should be identical (i.e., the x, y, and z are respectively parallel and in the same sense)
- a single analytical member may not be in more than one physical member

1. Select one or more beams which represent a single physical member.

   Note: Component analytical members must meet the criteria for physical members.

2. Either

   on the **Beam** ribbon tab, select the **Form Member** tool in the **Model** group

   or

   on the **Geometry** ribbon tab, select the **Form Member** tool in the **Physical Member** group (Analytical Modeling workflow)

   or

   on the **Member Design** ribbon tab, select the **Form Member** tool in the **Physical Member** group (Steel Design workflow)

   or

   right-click and select **Form Member** from the pop-up menu.

   The physical member (i.e., M#) number is displayed on each component analytical member and the physical member is added to the **Physical Member** table.

M. To automatically form physical members

To have the program automatically detect and form physical members within a selected set of members, use the following procedure.

1. Select a set of beams that you want formed into physical members.
2. Either:

   on the **Geometry** ribbon tab, select the **AutoForm Member** tool in the **Physical Members** group (Analytical Modeling workflow)

   or

   on the **Member Design** ribbon tab, select the **AutoForm Member** tool in the **Physical Members** group (Steel Design workflow)

The program applies the rules of forming physical members to detect which members should be grouped to a single physical member.

M. To automatically generate physical member restraints

Used the following procedure to automatically generate top and bottom flange restraint conditions for the selected physical members.

Note: The PBRACE command generated by this menu item is valid only for D2.B.12 Physical Member Design (on page 1372) .

1. Create a STAAD.Pro model with steel members.
2. Either:

   on the **Analysis and Design** ribbon tab, select the **Design** > **Steel Design** tool in the **Design** group

   or

   select the **Design** page

   The **Steel Design - Whole Structure** dialog opens.
3. Form one or more physical members in the model.
4. Select **AS4100** as the **Current Code**.
5. Select one or more of the physical members.
6. On the Utilities ribbon tab, select the **Physical Member Restraints** tool in the **Geometry Tools** group.

   The PBRACE commands are added for the current design parameter set with the tag (Physical).

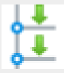
   Note: Only top or bottom flange restraints can be described using a single PBRACE command. Therefore, this command generates to lines of PBRACE commands for each physical member.

M. To manually add physical member restraints

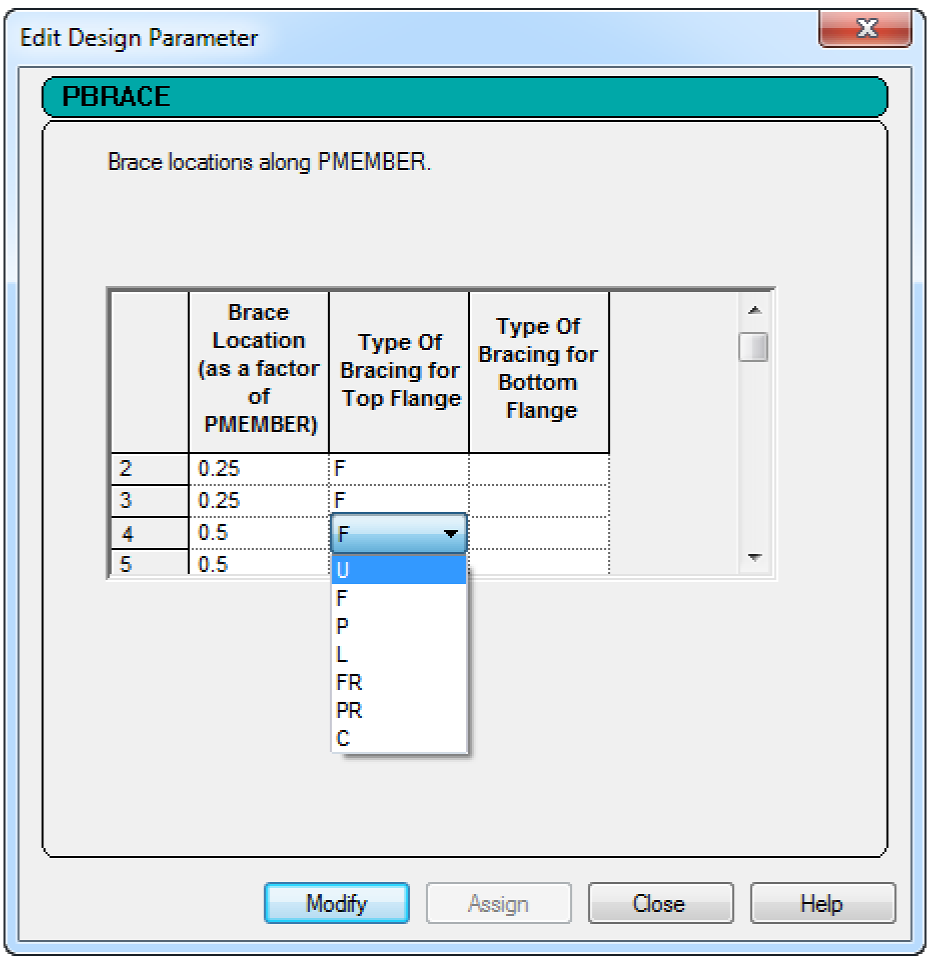
To manually add physical member restraints for assignment to physical members, use the following procedure.

Manual restraints apply to steel physical members which will be designed per AS 4100.

Tip: The restraint details can be automatically generated using a new **Member Restraints** item found in the **Utilities** ribbon tab in the **Geometry Tools** group.



1. In the Analytical Modeling workflow page controls, select the Design page.
2. Select **AS4100** as the **Current Code** in the **Steel Design - Whole Structure** dialog.
3. Click **Define Parameters.**
   The **Design Parameters** dialog opens.
4. Select the **PBRACE** parameter.
5. For each brace point, specify a fraction of the total physical member length in the **Brace Location** cell.



6. Select the type of restraint present for the top or bottom flange.

> **Note:** Only top or bottom flange restraints can be described using a single PBRACE command. If both top and bottom bracing is added in the Design Parameters dialog, this will generate two new command lines in the input file.

7. Click **Add**.
   The new command appears in the tree as a child element of the current design parameter set.
8. Click **Close**.
   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
9. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

The PBRACE specification is tagged as (Physical), and therefore can only be assigned to physical members (PMEMBER groups).

Related Links

- *D2.B.12.6 Physical Member Restraints Specification* (on page 1376)

M. To delete a physical member

1. Select the **Member Number ID** or any component Beam Number ID in the **Physical Member Table**.
2. Right click to display the pop-up menu.
3. Select **Delete**.
   A confirmation dialog opens.
4. Select **Yes**.

> **Note:** Only the physical member group assignment is removed. The component analytical beams are not deleted.

> **Caution:** This cannot be undone.

M. Plates

M. To set new plate attributes

To specify a set of attributes to automatically assign to new beams, use the following procedure.

1. On the **Geometry** ribbon tab, select the **Add Plate** > **Set New Plate Attributes** tool in the **Plate** group.
   The **Define Plate Object Property** dialog opens.
2. Select the check box for each of the plate object properties you want to use and then select the property value you want to use from the drop-down list:

Option	Description
Property	Check the **Use Property** option and then select a defined plate or surface thickness property from the drop-down list.
Material	Check the **Use Material** option and then select a material definition name from the drop-down list.
Release	Check the **Use Release** option and then select a plate release specification from the drop-down list.

3. (Optional) To create a new specification to use, click the associated Create button with the use option checked.
   The corresponding dialog opens to add a property, material, or release. Once you add this specification, the new specification is available from that drop-down list for use.
4. Check the option to **Assign these attribute when creating new plates**.
5. Click **OK**.

Related Links

- *Define Plate Object Property dialog* (on page 2740)

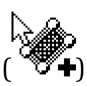
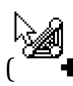
M. To draw plates connecting existing nodes

To draw triangular or quadrilateral plate elements connecting existing nodes, use the following procedure.

This procedure is used to manually add plate elements to your model. STAAD.Pro also has tools to automatically generate a finite element mesh with a surface area.

1. On the **Geometry** ribbon tab, select one of the following tools in the **Plates** group:

Use this tool...	To...
 **Add Quad Plate**	create four-node plate elements
 **Add Triangle Plate**	create three-node plate elements

   The mouse pointer changes to either the Add Quad Plate cursor () or Add Triangle Plate cursor (), respectively.
2. Click on any node that will form the first corner of the plate element.
   A line is "rubber banded" to the cursor from this node. This represents the first edge of the surface.
3. Click on the subsequent nodes to form the plate vertices in either a clockwise or counterclockwise order.

   The plate shape is "rubber banded" to the cursor as you move the mouse pointer to give you a preview of the plate shape.

   Note: Plate vertices should lie within a flat plane. Refer to " M. To check for warped plates (on page 912) " to verify that plates are planar.

   The plate is added to the model.
4. Repeat steps 2 and 3 to add more plates.
5. To stop adding plates:

   press **<Esc>**

   or

select any tool

Related Links

- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2224)

M. To add a plate bounded by beams

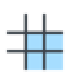
To create an “infill” plate by selecting exiting members as plate edges, use the following procedure.

Note: The beams to be used as bounding edges of an “infill” plate must meet the following requirements:

- three or four beams that form a closed polygon
- the beams cannot extend past the vertex nodes of the closed polygon
- the beams must lie within a plane

1. Select the beams you want to use as bounding edges of the plate.
   You must select three or more beams.

   Note: Use the **Beam Cursor** tool in the **Selection** group on the **Geometry** ribbon tab.
2. On the Geometry ribbon tab, select the **Add Plate** > **Create Infill Plates** tool in the **Plates** group.

   A message dialog indicates that the plate was successfully generated.

M. To generate plate mesh from corner nodes

To generate a finite element mesh by selecting corner nodes, use the following procedure.

You must have three or more existing nodes that lie in a plane in your analytical model.

This procedure is used to immediately generate a mesh within a selection of vertices. If you would like to investigate changing parameters prior to generating a mesh or to add openings with the mesh, then you may want to use the procedure a M. Parametric Models (on page 670) instead.

1. On the **Geometry** ribbon tab, select the **Generate Mesh** > **Create Mesh** tool in the **Plate** group.

   The mouse pointer changes to a mesh cursor.
2. Click the nodes that will form the vertices of the mesh area, in either a clockwise or counter clockwise order.

   Notes:

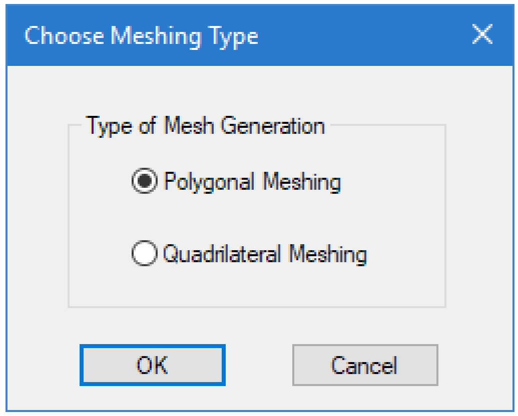
   - The order of the nodes should be in either a clockwise or counterclockwise direction.
   - The nodes must for a polygonal area (i.e., they edges cannot cross).
   - The nodes must lie in the same plane.
3. When you have select the corner nodes, either:

select the first node again

or

press **<Esc>**

The **Choose Meshing Type** dialog opens.



4. Select the type of meshing to use and then click **OK**.

Polygonal Meshing	Only triangular elements will be created
Quadrilateral Meshing	This will create principally four sided elements, but where the geometry dictates, some places may require triangular elements.

   The **Define Mesh Region** dialog opens.
5. Review the mesh vertices:
   a. Click the Add New Row tool to add a new vertex.
   b. Click the Delete Row tool to remove the currently selected row.
   c. Select the Holes tree entry to add openings to the mesh.
6. Click **OK**.
   The specified area within the vertices is meshed.

M. Parametric Models

STAAD.Pro allows you to model surfaces by automatically generating a mesh from a set of parameters. You can preview the final meshed state of the wall, slab, or panel before deploying the meshed entity into the remainder of the model. This way you can visually examine several trials or prototypes to find the most suitable alternative.

Tip: In past versions of STAAD.Pro, facilities for mesh generation have been available through the Geometry menu of the main program window and through the Structure Wizard utility which too is accessible through the Geometry menu. While these facilities continue to be available, an enhanced mesh generation tool has been added to the Geometry page.

This method offers some advantages over other methods of mesh generation in STAAD.Pro:

a. It enables you to preview the final meshed state of the wall, slab or panel before deploying the meshed entity into the remainder of the model. Several trials or prototypes can visually examined before committing to the most suitable one. This greatly minimizes the inconvenience of ending up with an undesirable meshed panel.
b. During the process of meshing, the program will automatically take into consideration existing nodes that lie within the boundary of the panel, but are not chosen as corner nodes of the panel. You simply have to define the panel boundary. The remainder of the nodes will be automatically considered by the program as “control

points" which is to say that elements will be created in such a manner that some of those elements will have these nodes in their incidence list. Control points might be panel points of a truss which supports a multi-span slab, or a foundation slab that supports several columns.

c. It allows for the definition of lines on the surface along which nodes should be created. Thus, if for a continuous slab over a wall, a line only needs to be drawn to define the intersection. Nodes will then be created along that line, and they will automatically become control points for the wall when it is meshed later. The problem of correctly modeling the monolithic connection between the 2 panels is now easier to handle as a result of this feature.

M. To create a parametric surface model

To create a parametric surface model for generating a finite element mesh, use the following procedure.

1. Select the **Parametric Models** tool in the Plate group on the **Geometry** ribbon tab.

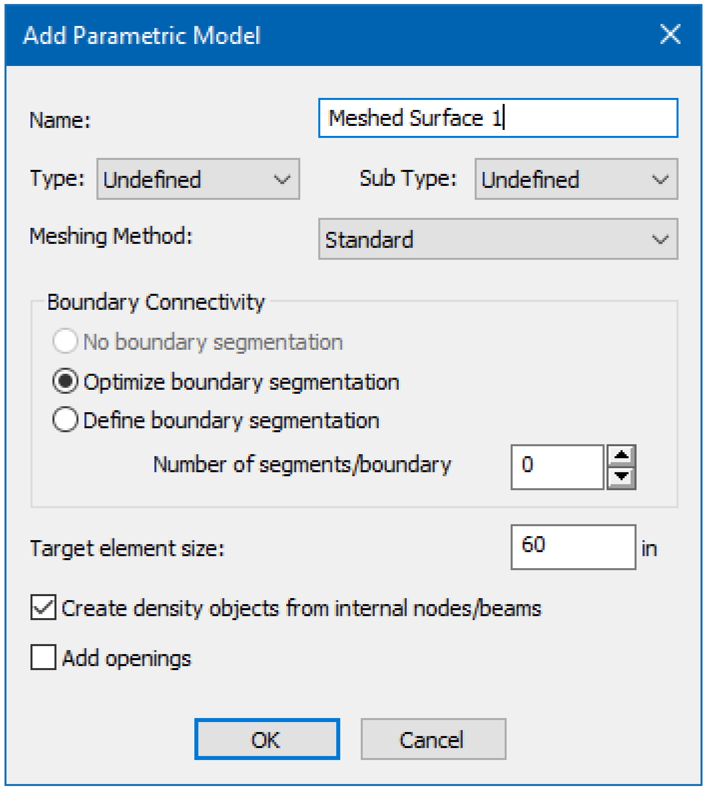
   The **Parametric Models** dialog opens. The structure is displayed as a series of dashed lines to signify this mode can be used to experiment with various settings.
2. Click **Add** in the **Parametric Models** dialog.

   The mouse pointer changes to the add surface cursor .
3. Click on the points that form the corners of the panel boundary.

   Note: Click the points in either a clockwise or counter-clockwise order.

   Tip: If you choose the option for including nodes and beams in the boundary area in the subsequent steps, then you can skip over intermediate points lying and a straight line segment, as the program will detect these and add them as mesh nodes automatically.

   When the first point is clicked a second time to close the loop, the **Add Parametric Model** dialog opens.



4. Specify the desired meshing parameters:
   a. Type a **Name** for the meshed surface.

      The default name is incremented.
   b. (Optional) Select a **Type** and **Sub Type** to identify the surface use.
   c. Select the **Meshing Method**.

      Selecting the **Standard** method will use quadrilateral elements where as the **Basic** method will use triangular elements.
   d. Select the **Boundary Connectivity** to identify the number of divisions along each surface boundary.
   e. Type a **Target element size** in the specified units of length.
   f. Check the **Create density objects from internal nodes/beams** to generate additional nodes along any members or defined nodes that lie in the surface plane.
5. (Optional) If the surface will have openings, check the **Add openings** option to specify those in the next dialog.

> **Tip:** Openings can be added later from the **Parametric Models** dialog if needed.

6. Specify the desired meshing parameters and click **OK**.
   If you selected to add openings, the dialog to define those opens.
7. (Optional) Select an opening shape and then click **OK** to draw the opening on the meshed surface.

   Refer to the following tasks for additional information on adding openings.

   The new mesh model is displayed as an overlay on the structure with control points highlighted. The details of the mesh parameters are available in the Parametric Models dialog.

Note: You may edit and refine your mesh by varying the parameters as necessary until the resulting finite element is satisfactory for your model requirements.

8. Once you are satisfied with the surface mesh, click **Merge Mesh**. in the Parametric Models dialog to commit it to the STAAD.Pro input file.

Tip: To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

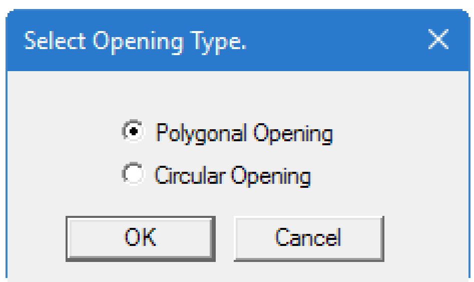
Related Links

- *Add Parametric Model dialog* (on page 2737)
- *Parametric Models dialog* (on page 2734)
- *Add Parametric Model dialog* (on page 2737)
- *New Mesh Model dialog* (on page 2736)

M. To create a polygonal opening in a mesh model

1. In the **Parametric Models** dialog Mesh Models list, select the leaf called **Openings** under the mesh model to which you wish to add an opening.
2. Click **Add**.

   The **Select Opening Type** dialog opens.

3. Select the **Polygonal Opening** option and then click **OK**.

   The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.

   Tip: The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.

4. (Optional) Edit the grid positions using the **Snap Node/Panel**
5. Click the points inside the panel to define the corners of the polygonal hole.

   Note: The cursor displays a blinking cross (+) at grid intersections. This blinking cross marks the point on the grid at which nodes of the hole boundary may be set. Click points around the mesh boundary in either a clockwise or counter-clockwise order, clicking on the first point a second time to close the loop.

   The mesh is regenerated taking into consideration the hole.

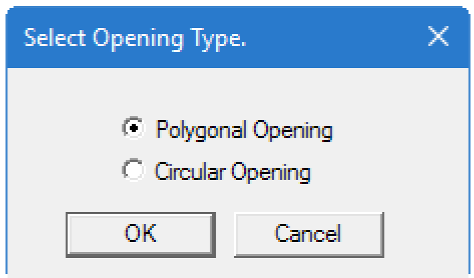
Tip: To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

M. To create a circular opening in a mesh model

Note: In case an opening is required on a part of the structure that is already meshed with plate elements, the existing plates have to be deleted first, and then, the plates with the opening can be modeled as a whole using this feature.

1. In the **Parametric Models** dialog Mesh Models list, select the leaf called **Openings** under the mesh model to which you wish to add an opening.
2. Click **Add**.

   The **Select Opening Type** dialog opens.

3. Select the **Circular Opening** option and then click **OK**.

   The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.

   Tip: The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.

4. (Optional) Edit the grid positions using the **Snap Node/Panel**
5. Click the point which will be the center of the circular opening.
   The cursor displays a blinking cross (+) at grid intersections. This blinking cross marks the point on the grid at which nodes of the hole boundary may be set.
6. Click a second point to define the radius of the circle.

   A dialog opens for defining the number of divisions around the circular opening.
7. (Optional) Change the number of divisions used around the circumference of the circular opening.
8. Click the **OK** button.
   The mesh is regenerated taking into consideration the hole.

Tip: To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

M. To add a density line or point

1. In the Parametric Models dialog Mesh Models list, click on the leaf called **Density Lines** or **Density Points** under the mesh model to which you wish to add an opening.
2. Click **Add**.

   The drawing will change to display just the mesh region. The right hand side of the screen will display a grid with the grid line settings and spacing.

> **Tip:** The various viewing options and zoom options on top of the screen can be used to obtain a convenient view of the panel.

3. (Optional) Edit the grid positions using the **Snap Node/Panel**
4. Either:

   Click the start and points of a density line.

   or

   Click a density point.

   The mesh is regenerated to account for the new density line or point
5. Repeat step 4 as many times as needed.
6. To stop adding density points or density lines, press the **<Esc>** key.

> **Tip:** To delete a parametric mesh, opening, density line, or density point, select that entry within the **Parametric Models** dialog and click **Delete**.

M. To define a slab/wall connection

Use the following procedure to generate a mesh for a wall super-element when compatibility with a connected slab super-element is required.

A slab mesh must first be created. The slab mesh should include a density line representing the connecting edge of the wall.

> **Tip:** Using beam elements is fast way to help generate meshed surfaces and ensure that the density line edge for the slab/wall connection is included when the slab is generated. These "construction" beams can be deleted once the surface elements have been successfully generated.

1. On the **Geometry** ribbon tab, select the **Generate Slab/Wall Connection** tool in the **Structure** group.

   The mouse pointer changes to the add surface cursor .
2. In the Active view window, click the corner nodes of the wall super-element in either a clockwise or counterclockwise order.

   > **Note:** The first and second nodes must be the corner nodes which form the slab/wall connection. Failure to do so will open a warning dialog.
3. Continue clicking the remaining corner nodes to for the wall super-element. Close the polygon by clicking on the first node again.

   The **Division Along Wall** dialog opens.

   > **Note:** The divisions along the horizontal side of the wall have already been decided as the number of points created along the wall base in the previously mesh slab.
4. (Optional) Specify a **No. of divisions in extrusion direction**

> **Note:** The divisions along the horizontal side of the wall have already been decided as the number of points created along the wall base in the previously mesh slab.

5. Click **OK**.
   The wall is meshed.

> **Tip:** To verify the compatibility between plates, select the **Plate Tools** > **Check Improperly Connected Plates** tool in the **Geometry Tools** on the **Utilities** ribbon tab.

Related Links

- *EX. Meshed Wall-Slab Connection* (on page 6861)

M. Composite Decks

STAAD.Pro can model composite steel deck systems for design.

An important feature to note is that when the structure is modified, the deck automatically gets modified with it. Note that when the dimension of the structure is changed along the length from 30 ft to 40 ft, the deck gets modified with the structure and there is no need to redefine the deck all over again. The deck loading is also revised automatically.

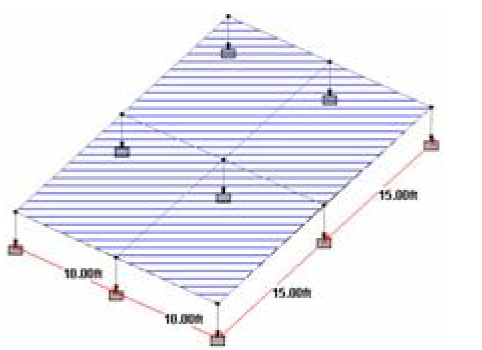
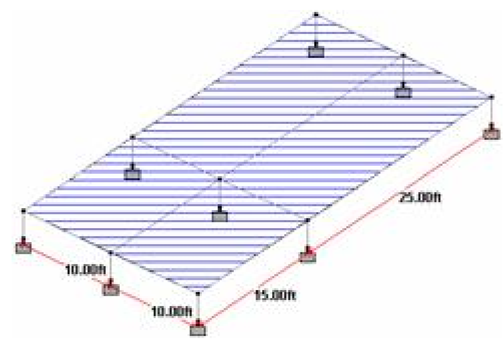
Original structure	Modified structure
	

Figure 101: Composite deck and load update with structure changes

The design parameters for the composite deck are already defined through the interface. Few additional design parameters like `TRACK` can be assigned, if so desired, and finally, design commands like `CHECK CODE` has to be assigned to the composite deck.

Related Links

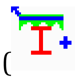
- *Composite Deck dialog* (on page 2741)
- *TR.20.7 Composite Decks* (on page 2270)
- *M. To add a floor load or one-way load* (on page 840)
- *P. To generate a floor vibration report* (on page 1996)
- *P. Floor Vibrations Engineering Theory* (on page 1998)

M. To create a new composite deck from perimeter beams

1. Select the **Composite Deck Layout** tool in the **Composite Deck** group on the **Geometry** ribbon tab.

The **Composite Deck** dialog opens.

2. Select the perimeter beams:
   a. Select the **Clicking on Nodes** option.
   b. Click **Create New Deck**.

      The mouse pointer changes to the Add Composite Deck cursor ().
   c. Click on the end nodes defining the corners of the perimeter beam area.
   d. Click on the first node a second time to close the decking area.

   Alternatively, you can select the perimeter beams in the view window first. Then select the **Use Selected Beam** option and click **Create New Deck**.

   The **New Composite Deck** dialog opens.
3. Type a **New Composite Deck Name** and then click **OK**.

The deck name is added to the list of composite decks with all the perimeter and interior beams listed.

Note: You can delete a composite deck definition by selecting in within the list in the **Composite Deck** dialog and then pressing **<Delete>**. Confirm you want to delete the deck in the dialog. Only the deck definition is deleted. Constituent members will remain in the model.

Related Links

- *Composite Deck dialog* (on page 2741)

M. To specify a direction for the composite deck ribs

To specify the orientation of the deck ribs, use the following procedure.

The program will automatically assign a rib direction parallel to the detected shortest span in the deck area. However, you will often need to edit the orientation.

1. Select a composite deck name in the **Composite Deck** dialog list.
2. Select a pair of parallel beams.

   The deck ribs will run perpendicular to these two beams.
3. Click **Create Direction.**
   If you have previously assigned a deck rib direction, you will be prompted to confirm you want to update this direction.

The deck span direction arrow is updated to reflect the change in direction.
Related Links

- *Composite Deck dialog* (on page 2741)

M. To assign composite deck properties

To define the concrete, rib, and connectivity details of a composite deck, use the following procedure.

1. Select a composite deck name in the **Composite Deck** dialog list.
2. Type values to use for the concrete in the current units:

Concrete thickness above flutes
Unit Wt. of Concrete
Concrete Grade

3. Specify the steel deck rib properties:

To...	Do the following...
use a catalog value	select the deck name from the **Use Database** drop-down list
specify a custom value	type values for **Rib Width** and **Rib Height**.

> **Note:** The program includes three standard catalogs: ASC™, Vulcraft™, and VERCO™.

4. Specify the **Stud Diameter** by either:

   selecting a predefined diameter from the drop-down list

   or

   selecting **Custom** in the drop-down list and then typing the diameter in the current units
5. Type the **Stud Length** in the current units.
6. Select if the composite deck is **Shored** (i.e., propped) or **Unshored** during construction.
7. Click **Update Deck Property**.

Related Links

- *Composite Deck dialog* (on page 2741)

M. To modify composite steel beam properties

To change the steel beam profile or effective flange width, use the following procedure.

The program will use the assigned steel sections for beams and also automatically calculate the effective flange width of the composite section. However, you can update steel sections from the **Composite Decks** dialog as well as specify a value for the effective flange width.

1. Select a beam in the **Composite Decks** dialog within a composite deck name.
   The beam options are displayed in the dialog.
2. (Optional) Click **Add/Change Property** to open the **Section Profile Tables** dialog.

   From there, you can select a different beam or beam properties.

> **Note:** You should *not* specify a composite section type specification for these beams. The use of the composite deck definition allows the program to calculate all the necessary composite section parameters automatically. Refer to G.6.7 Composite Beams and Composite Decks (on page 2119) for details.

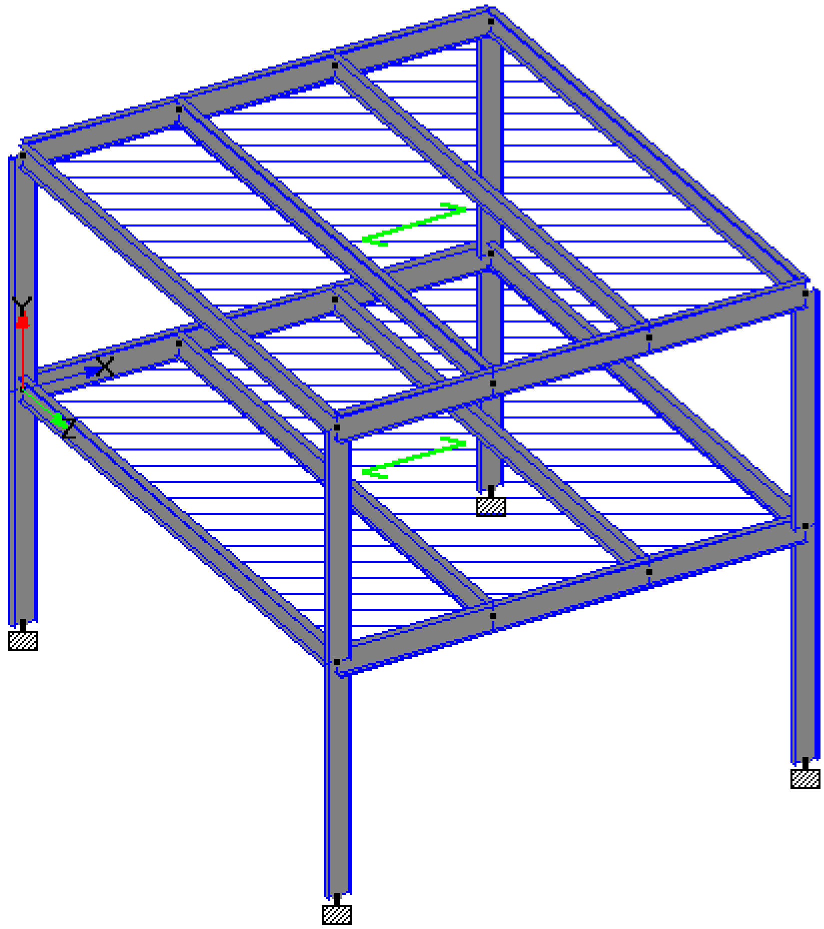
3. (Optional) Type an **Effective Width** in the current units and then click **Update** to change the effective width of the composite deck for the beam.

Related Links

- *Composite Deck dialog* (on page 2741)

M. Example Composite Deck

The following STAAD.Pro input file contains a simple composite deck.




```
STAAD SPACE
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 20 0 0; 4 30 0 0; 5 0 0 35; 6 10 0 35; 7 20 0 35;
8 30 0 35; 9 0 -15 0; 10 30 -15 0; 11 0 -15 35; 12 30 -15 35; 13 0 15 0;
14 30 15 0; 15 0 15 35; 16 30 15 35; 17 10 15 0; 18 20 15 0; 19 20 15 35;
20 10 15 35;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 8; 5 8 7; 6 7 6; 7 6 5; 8 5 1; 9 2 6; 10 3 7; 11 1 9;
12 4 10; 13 5 11; 14 8 12; 15 1 13; 16 4 14; 17 5 15; 18 8 16; 19 13 17;
20 17 18; 21 18 14; 22 14 16; 23 16 19; 24 19 20; 25 20 15; 26 15 13; 27 17
20;
28 18 19;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
```

```
4 8 TO 18 22 26 TO 28 TABLE ST W18X35
1 TO 3 5 TO 7 19 TO 21 23 TO 25 TABLE ST W21X50
START DECK DEFINITION
_DECK C2
PERIPHERY 1 TO 8
DIRECTION -1.000000 0.000000 0.000000
COMPOSITE 10 9 4 8
OUTER 1 4 8 5
DIA 0.000000
HGT 0.000
CT 0.271
FC 576.000
RBW 2.000
RBH 0.167
SHR 0
VENDOR NONE
CD 0.110
CMP 2.0
CW 10.000000 MEMB 10
CW 10.000000 MEMB 9
CW 5.000000 MEMB 4
CW 5.000000 MEMB 8
_DECK C3
PERIPHERY 19 TO 26
DIRECTION -1.000000 0.000000 0.000000
COMPOSITE 28 27 22 26
OUTER 13 14 16 15
DIA 0.000000
HGT 0.000
CT 0.800
FC 476.000
RBW 0.500
RBH 0.500
SHR 0
VENDOR NONE
CD 0.150
CMP 2.0
CW 10.000000 MEMB 28
CW 10.000000 MEMB 27
CW 5.000000 MEMB 22
CW 5.000000 MEMB 26
END DECK DEFINITION
CONSTANTS
MATERIAL STEEL MEMB 1 TO 28
SUPPORTS
9 TO 12 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1
UNIT FEET POUND
ONEWAY LOAD
YRANGE 0 0 ONE -57 GY
UNIT FEET KIP
PERFORM ANALYSIS PRINT STATICS CHECK
FINISH
```

M. Solids

M. To draw a solid connecting existing nodes

To draw a solid with 4 to 8 nodes connecting existing nodes, use the following procedure.

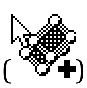
You model must have at least four nodes which do *not* all lie in the same plane.

The basic solid element consists of 8 nodes. However, by collapsing nodes together progressively, different three-dimensional solid shapes can be formed. The minimal number of nodes is a four-noded, pyramidal shape.

1. On the **Geometry** ribbon tab, select one of the tools in the **Add Solid** menu in the **Solid** group.

Use this tool...	To...
	create eight-noded solid elements (i.e., a cuboid)
	create seven-noded solid elements
	create six-noded solid elements (i.e., a triangular prism)
	create five-noded solid elements (i.e., a rectangular pyramid)
	create four-noded solid elements (i.e., a triangular pyramid, or tetrahedron)

   The mouse pointer changes to either the Add Solid cursor ().
2. Click on any node that will form the first corner of the solid element.
   A line is “rubber banded” to the cursor from this node. This represents the first edge of the solid.
3. Continue clicking on nodes up to the number corresponding to the tool selected in step 1.

   It is important that you click the nodes in an order which does not result in a negative volume.

   a. Click on the node counter-clockwise to the first node on the same face.
   b. Click the subsequent nodes on the same face in a counter-clockwise order.
   c. Click on a node on the opposing face.
   d. Click on the node counter-clockwise to the first opposing face node.
   e. Click the subsequent nodes on the opposing face in a counter-clockwise order.

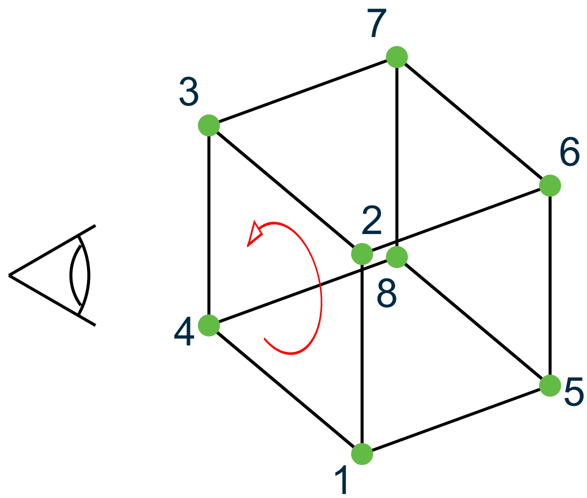


Figure 102: Select nodes in a counter-clockwise order from an outside view perspective on one face and then repeat on the opposing face.

The solid element is added to the model.

Tip: It is recommended to use the **Solid Tools** > **Negative Volume** tool in the **Geometry Tools** group on the **Utilities** ribbon tab to verify the solid incidences are correctly specified. The 3D rendering is also a useful tool in identifying negative volumes.

Related Links

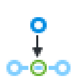
- *G.5.2 Solid Elements* (on page 2110)
- *TR.13.2 Solid Element Incidences Specification* (on page 2225)

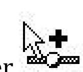
M. Nodes

M. To insert a node in a single member

Used to insert nodes in an existing member(s). The member is split into the corresponding number of segments with automatic generation of node numbers, member numbers, member properties, and loads.

1. On the **Geometry** ribbon tab, select the **Insert Node** tool in the **Beam** group.

   The mouse pointer changes to the **Insert Node** pointer .
2. Select a beam member.
   The Insert Nodes into Beam # dialog (on page 2730) opens.
3. Either:

Specify a **Distance** or **Proportion** value and click **Add New Point**.

or

Click **Add Mid Point** to add a new node halfway along the length of the beam.

or

Specify an integer number of new nodes in the **n =** box and click **Add n Points** to divide the beam into n+1 number of equal divisions.

The new node point is listed in the **Insertion Points** list.

4. Repeat Step 3 as many times as needed.

Tip: Click **Remove** to remove the selected node point from the Insertion Points list if you make an error.

5. Click **OK**.
The dialog closes and the specified nodes are inserted into the beam.

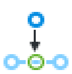
Related Links

- *Insert Nodes into Beam # dialog* (on page 2730)

M. To insert a node in multiple members

Used to insert nodes in an existing member(s). The member is split into the corresponding number of segments with automatic generation of node numbers, member numbers, member properties, and loads.

1. Select two or more beam elements.
2. On the **Geometry** ribbon tab, select the **Insert Node** tool in the **Beam** group.



The Insert Node / Nodes dialog (on page 2731) opens.

3. Either:

Select the **New point by distance** option and then type a **Distance** value.

or

Select the **New point by proportion** option and then type a **Proportion** value.

or

Select the **Add Mid Point** option.

or

Select the **Add 'n' Points** option and then type an integer number of new nodes in the **n =** box to divide the beam into n+1 number of equal divisions.

4. Click **OK**.
The dialog closes and the specified nodes are inserted into the beams.

Related Links

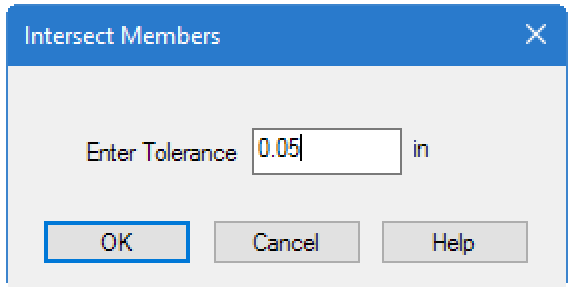
- *Insert Node / Nodes dialog* (on page 2731)

M. To add a node at overlapping beams

To connect to overlapping beams with a node at that location, use the following procedure.

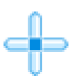
You can manually select beams that are overlapping or the program can automatically detect these for you using the optional first step below.

1. (Optional) Automatically select overlapping beams:
   a. On the **Geometry** ribbon tab, select the **Intersecting Beams** > **Highlight Intersecting Beams** tool in the **Beam** group.

   The **Intersect Members** dialog opens.
   b. In the **Enter Tolerance** field, type a tolerance used for how close members should pass before the program considers them to be overlapping.
   c. Click **OK**.

   The program automatically selects all overlapping beam sets.
2. On the **Beam Tools** ribbon tab, select the **Intersect Beams** tool in the **Model** group.

3. Type a tolerance to use in the **Intersect Members** dialog and then click **OK**.
   A message dialog opens to indicate that new beams have been created.

M. To merge two nodes

1. Select the nodes to be merged using the Nodes Cursor.
2. On the **Geometry** ribbon tab, select the **Merge Selected Nodes** tool in the **Node** group.

   The **Select Node** dialog opens.
3. Select the **Node To Keep** from the list of selected nodes.
   The remaining nodes will be removed. All members and elements which were connected to those nodes will be reconnected to the selected node to keep.
4. Click **OK**.

M. Modify Your Model

The following procedures are used to manipulate existing model geometry.

M. To move selected objects

You must have one or more objects selected.

Refer to GS. Selecting Objects in STAAD.Pro (on page 40) for details.

1. On the **Geometry** ribbon tab, select one of the following tools:

   the **Move Node** tool in the **Node** group

   or

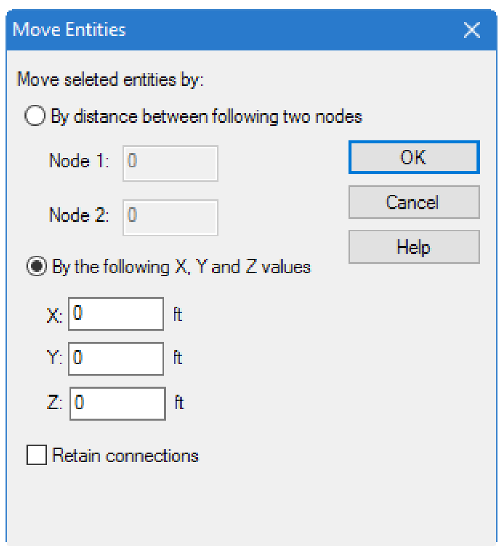
   the **Move Beam** tool in the **Beam** group

   or

   the **Move Plate** tool in the **Plate** group

   or

   the **Move Solid** tool in the **Solid** group
   The **Move Entities** dialog opens.



> **Note:** This dialog has a different title depending on the object type being moved but the functionality is similar.

2. Select the method by which the objects are to be moved:

   By distance between following nodes uses two node numbers and determines the vector between them. The selected entities are moved along a parallel vector of the same distance.

   or

   By the following X, Y and Z values uses the Cartesian coordinates specified to create a vector.

3. Specify node numbers or vector component values.
4. (Optional) Check the **Retain connections** option to retain connections between entities.

   Note: The model geometry will be adjusted to some degree, even if this is not selected, when nodes are the entities being moved as they member, plate, surface, and solid geometry is defined by node location. However, if you are moving a member, plate, surface, or solid, then selecting this option will move the associated nodes and thus retain the connection with adjacent entities (though their geometry will stretch, warp, or shift as a result of the move).
5. Click **OK**.
   A message dialog opens asking if you would like new nodes created where any moved entities cross other existing entities.
6. Click **Yes** to create new nodes or **No** to ignore intersections.
   The model is updated with the moved geometry.

Related Links

- *Move Entities dialog* (on page 2726)

M. To move the model origin

To shift the translate the entire model by shifting the origin location along a vector, use the following procedure.

Tip: You can use the **Generate Rotate** tool to rotate the entire model about the origin as well.

1. On the Geometry ribbon tab, select the **Move Node** > **Move Origin** tool in the **Node** group.

   The **Move Origin** dialog opens.
2. Specify the origin offset vector method:

Vector method	Do the following...
two nodes	select the **By distance between following two nodes** and then type the node numbers in the corresponding fields.
global axes components	select the **By the following X, Y and Z values** and then type the global axes component distances in the corresponding fields.

   Either method generates a vector by which the origin is shifted.

   Note: A positive value in the coordinates (or resulting positive value resulting the from the selected nodes) will move the origin in a positive direction along the corresponding global axis. Thus, the resulting coordinate of each node is therefore reduced by that same magnitude.

   For example, providing a origin shift along the X axis of 5 units, means that a node previously located at the origin now has an X coordinate of -5 units.
3. Click **OK**.

The coordinates of each node are modified in the STAAD input file to reflect the change in distance to the new origin.

Related Links

- *Move Origin dialog* (on page 2727)

M. To rotate selected entities

You must have one or more objects selected.

Refer to GS. Selecting Objects in STAAD.Pro (on page 40) for details.

1. On the **Geometry** ribbon tab, select the **Rotate** tool in the **Structure** group.

   The **Rotate** dialog opens.
2. Specify an **Angle** of rotation.
3. Provide to points used to define the axis of rotation by either:

   selecting two **Nodes**

   or

   typing coordinates of two arbitrary **Points**
4. Select if the selected geometry is to be copied of moved and, if copied, if linking members are to be created.
5. Click **OK**.

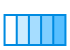
Related Links
- *Rotate dialog* (on page 2719)

M. To generate copies of geometry along a line

To generate copies of selected geometry along a path parallel to a global axis, use the following procedure.

1. Select the structure geometry (beams, plates, solids, etc.) that you want to copy.

   Tip: You do not need to select the nodes, as they will be copied along with the other model entities.
2. On the **Geometry** ribbon tab, select the **Translational Repeat** tool in the **Structure** group.

   The **Translational Repeat** dialog opens.
3. Select the **Global Direction** along which the copies will be generated.
4. Specify the **No. of Steps** to generate.

   Tip: By setting the **Default Spacing** value first, the entries will update with the new value for each step.
5. (Optional) Edit the **Spacing** values for each step (copy) if the steps are irregular or if the desired **Default Spacing** was not first specified.
6. (Optional) Set the option to **Renumber Bay** and specify a **Number from** value (start number of that copy) if needed.
7. (Optional) Set the **Link Steps** option to generate additional members between copies, parallel to the **Global Direction**.

You may also set the **Open Base** option, which will prevent links from being generated between nodes with the lowest Y coordinates (when Y is vertical/up).

8. (Optional) Set the **Generation Flags** to the level of geometry, properties, or specifications to be copied to the new, generated members.
9. Click **OK**.

   The new members are generated using the specified parameters.

Related Links

- *Translational Repeat dialog* (on page 2716)

M. To generate copies of geometry along an arc

To create copies of selected geometry along a arc about a global axis, use the following procedure.

1. Select the structure geometry (beams, plates, solids, etc.) that you want to copy.

   Tip: You do not need to select the nodes, as they will be copied along with the other model entities.
2. On the **Geometry** ribbon tab, select the **Circular Repeat** tool in the **Structure** group.

   The **3D Circular** dialog opens.
3. Select the global **Axis of Rotation** about which the copies will be generated.
4. Specify the **Total Angle** of the arc.

   This is measured in a positive (right-hand rule) direction about the selected **Axis of Rotation** between the original selection and the last copy.
5. Specify the **No. of Steps** to generate.

   The copies will be equally distributed about the specified arc.
6. Specify a center of rotation by either:

   typing a Node number. You may also click the Node Selection tool to graphically select a node.

   or

   planar coordinates for an arbitrary point (i.e., not at a node)

   Note: The axis of rotation for the circular copy will pass through this selected node or point, parallel to the selected global axis of rotation.
7. (Optional) Check the **Use this as Reference Point for Beta angle generation** option to orient the copies towards the central axis of rotation.
8. (Optional) Set the **Link Steps** option to generate additional members between copies.

   You may also set the **Open Base** option, which will prevent links from being generated between nodes with the lowest Y coordinates (when Y is vertical/up).
9. (Optional) Set the **Geometry Only** option to generate only nodes, members, elements, etc. with no properties assigned.
10. Click **OK**.

    The new members are generated using the specified parameters.

Related Links

- *3D Circular dialog* (on page 2717)

M. To generate mirror copies of model entities

To generate symmetric copies of the model selection about a plane, use the following procedure.

1. Select the geometry to copy or move.

   Note: Refer to the discussion on the Select menu for details on the selection procedure.
2. On the **Geometry** ribbon tab, select the **Mirror** tool in the **Structure** group.

   The **Mirror** dialog opens.
3. Specify an mirror plane.
4. Specify a Plane position.
5. Select if the selected geometry is to be copied of moved and, if copied, if linking members are to be created.
6. (Optional) Set the option to mirror members if their orientation is also to be mirrored.
7. Click **OK**.

Related Links

- *Mirror dialog* (on page 2725)

M. Groups

A group is a way to collect a series of objects together for selection, loading, or design purposes.

M. To create a group from a selection

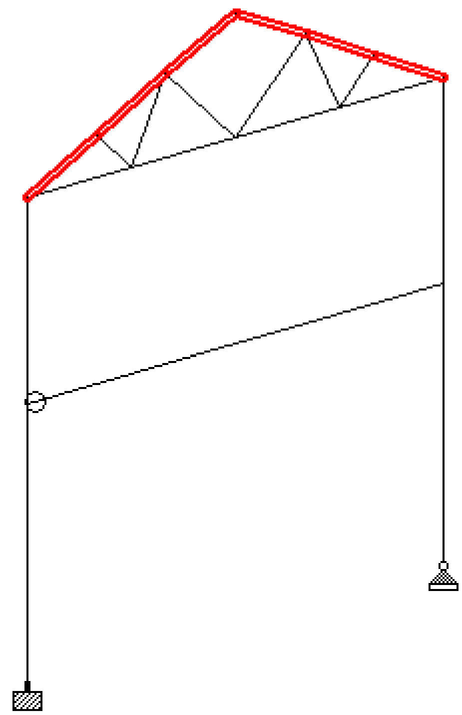
To create a new group using a selection of model objects, use the following procedure.

Group names are a means for easily identifying a collection of entities like Beams, Plates or Solids using a single moniker. By grouping these entities, we need to assign attributes such as member properties and material constants just to the group, a simple process, compared to the task of assigning them to the individual members.

The file EX. US-1 Plane Frame with Steel Design (on page 6256) is used to demonstrate this feature.

Consider the members which form the truss as being in one of three groups: Top Chords, Bottom Chords, and Web Members.

1. Select the top chord members:
   With the model open, use the **Beam Cursor** tool to select the members forming the top chords (i.e., members 8 through 13).



Tip: Press and hold the **<Ctrl>** key and then click the members in the View window.

2. Either:

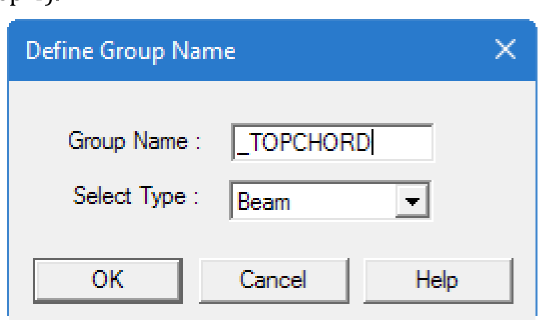
   On the **Utilities** ribbon tab, select the **Groups** tool in the **Geometry Tools** group

   or

   press **<Ctrl+G>**

   If you have not previously defined any groups, the **Define Group Name** dialog opens. Otherwise, the **Create Group** dialog opens (skip to step 4).



3. Specify the group details:
   a. Type a name in the **Group Name** field.

Note: Group Names must begin with the underscore "_" character.

A group name such as _TOPCHORD can be used for this example.

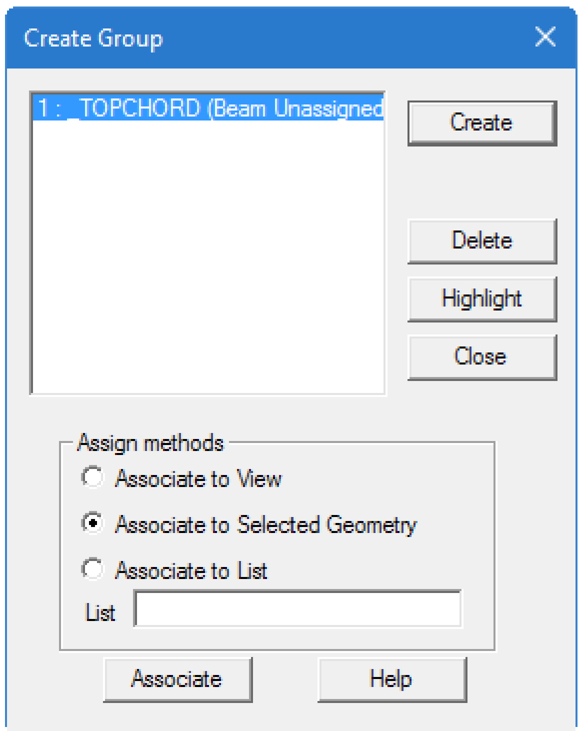
b. Select the object type from the **Select Type** drop-down list.

Groups can consist of nodes, beams, plates, solids, and a general category called geometry. Geometry can be used to associated objects of mixed types.

Select **Beam** for the top chords example.

c. Click **OK**.

The **Create Group** dialog opens. At this point, you have created a group name but it does not yet contain any objects.

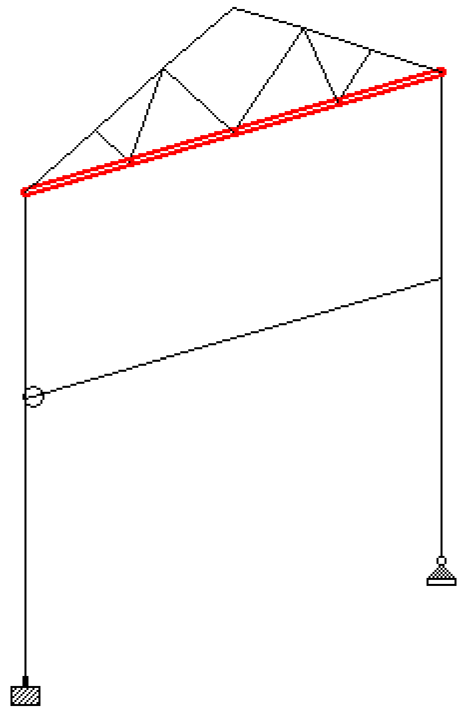


4. Select **Associate to Selected Geometry** and then click **Associate**.

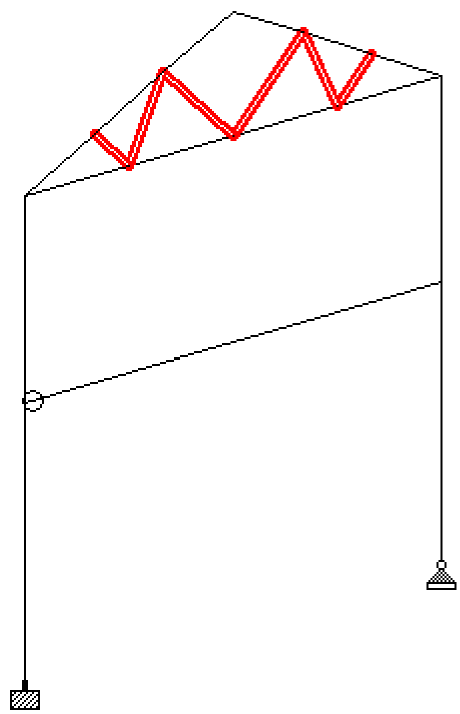
   The selected members are added to the group.
5. Select other model objects.

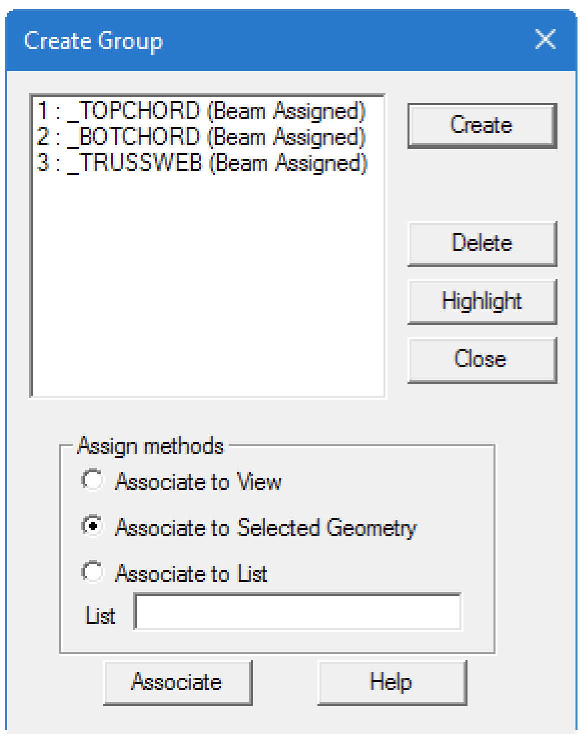
Tip: The **Create Group** dialog can remain open while you change selection tools and select objects in the View window.

For the example file, the members forming the bottom chord (members 20, 21, 22, and 23).



6. **Associated these members with a new group:**
   - **a.** Click **Create** in the **Create Group** dialog.
   - **b.** Follow steps 4 and 5 and assign the **Group Name** and objects type as the **Select Type**.
     For the example file `US-1 Plane Frame with Steel Design.std`, use the name `_BOTCHORD` for the bottom chord members.
7. Repeat steps 5 and 6 to create additional groups as needed.
   For the example file `US-1 Plane Frame with Steel Design.std`, you can select the truss web members (members 14, 15, 16, 17, 18, and 19) and add them to a group named `_TRUSSWEB`





8. Click **Close** in the **Create Group** dialog.

Select the **Group Selection** tool in the **Selection** group on the **Selection** ribbon tab to open the **Select Groups** dialog, which can be used to select a named group.

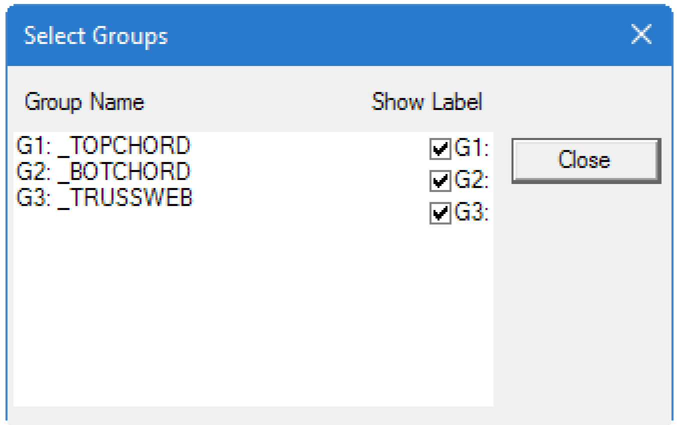


Figure 103:

Note: If the option for **Show Label** are unchecked, labels such as beam numbers will not be displayed for these specific members, even if the beam numbering icon is switched on for the entire structure.

Related Links

- *Create Group dialog* (on page 2950)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2235)

M. To add objects to an existing group

To add more objects to an existing group, use the following procedure.

It is important to select the current objects in a group along with the additional objects so that the current objects are not removed. When associating objects to a group, *only* the current selection (or other association method) will be included in the group.

1. On the Utilities ribbon tab, select the Groups tool in the Geometry Tools group.

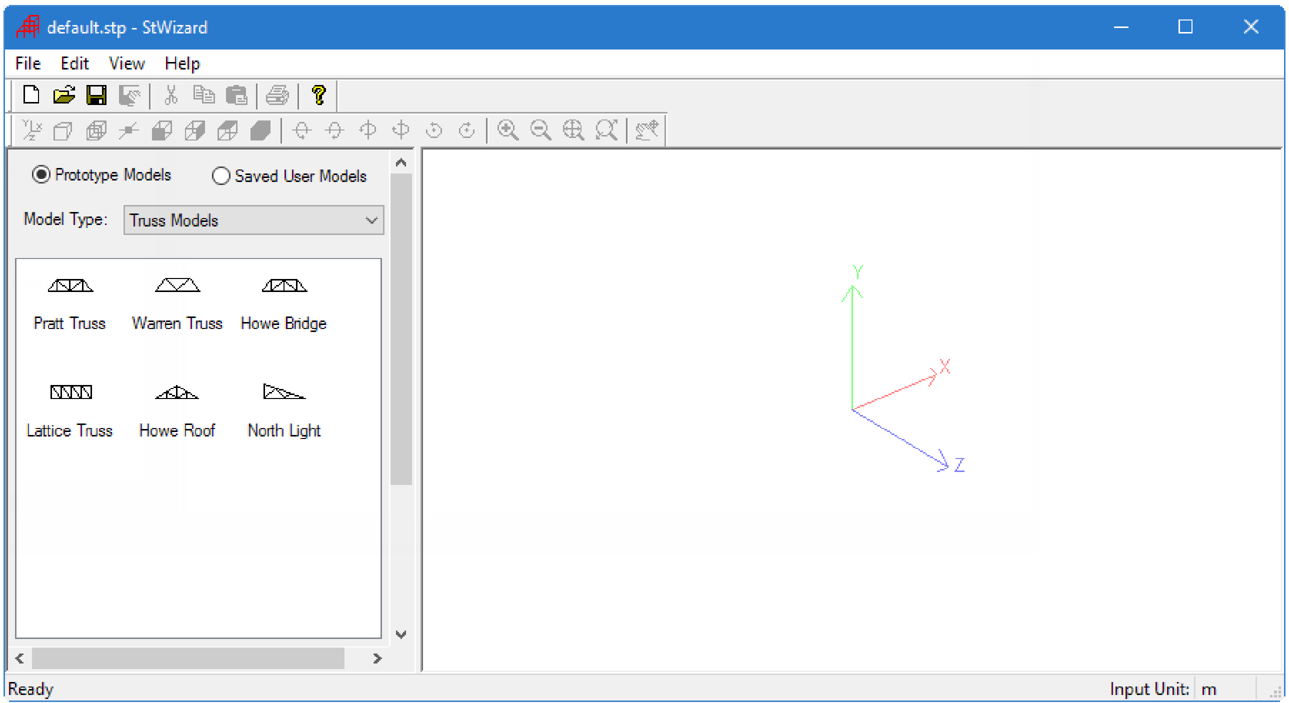
   The **Create Groups** dialog opens.
2. Select the group name to which you want to add objects.
3. Click **Highlight**.
   The current objects in the group are selected in the view window.
4. In the view window, use the selection cursors to select the additional geometry.

   Note: Hold the **<Ctrl>** key in order to select additional objects without unselecting the current group contents.
5. Select the **Associated to Selected Geometry** option.
6. Click **Associate**.

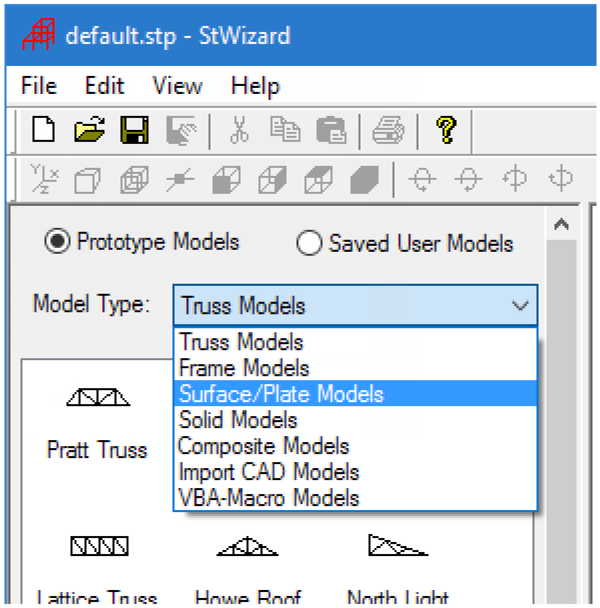
M. Structure Wizard

Used to parametrically generate a structural model and then transfer and superimpose it on the current structure in STAAD.Pro.

Opens when **Geometry** > **Run Structure Wizard** is selected in the STAAD.Pro window.



The **Prototype Models** and **Saved User Models** options appear on the top of the left side of the screen. If the **Prototype Models** option is selected, the **Model Type** will list the types of prototype structures available (such as Trusses, Frames, Plates, Solids, etc.) as shown below. If the **Saved User Models** option is selected, the **Model Type** will display the list previously done and saved models by the user.



M. To set units

To set the units of length used in the parametric model types, use the following procedure.

The current units of length are displayed in the Structure Wizard status bar.

1. Select **File** > **Select Units**.
   The **Select Units** dialog opens
2. Select one of the units of length.

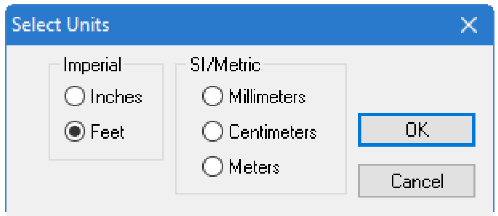
   Inches
   Feet
   Millimeters
   Centimeters
   Meters
3. Click **OK**.

M. Generation of Structure from Models

In this section, the process of generating a structural model and combining it with the existing STAAD.Pro structure will be explained using a Howe Roof truss. Follow these steps to create the other truss types also.

Selection of Units

The unit of length should be specified prior to the generation of a model. Select **File** > **Select Unit** and the **Select Unit** dialog opens. You can select any unit of length from Imperial (inch, feet) or SI/Metric (millimeter, centimeter, meter) system of units.



M. To create a truss model

To create a parametric truss model, use the following procedure.

1. Select **Truss Models** from the **Model Type** drop-down list.
2. Select one of the parametric truss types by either:

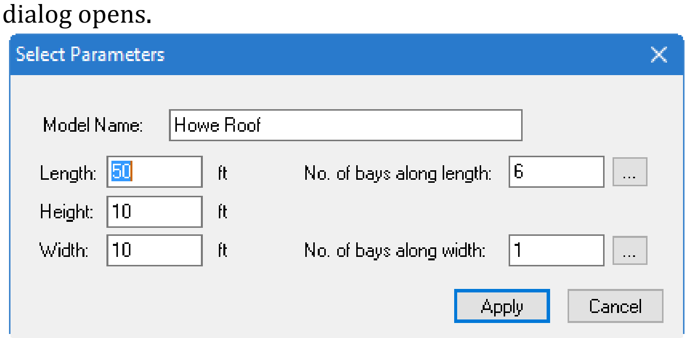
   double-click on the type of model

   or

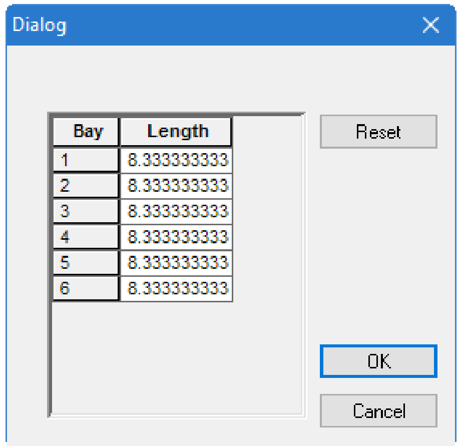
   click-and-drag the model icon into the view window

   Pratt Truss
   Warren Truss
   Howe Bridge
   Lattice Truss
   Howe Roof
   North Light

   The **Select Parameters** dialog opens.



3. (Optional) Type a **Model Name**.
4. Type the overall truss **Length**, **Height**, and **Width** distances.
5. Type the **No. of bays along length** to define the number of equally spaced bay along the length of the truss.
6. (Optional) To use unequaly length bays:
   a. Click the **[...]** adjacent to the **No. of bays along length** field.
      A dialog opens with the length of each bay.



b. Type the length of the different bays in the corresponding **Length** cells.

> **Note:** The total length of the bays must add to the overall length specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

c. Click **OK**.

7. Type the **No. of bays along width** to define the number of transverse bays.
8. (Optional) To use unequaly length transverse bays:

   a. Click the **[...]** adjacent to the **No. of bays along width** field.
   A dialog opens with the length of each bay.

   b. Type the length of the different bays in the corresponding **Length** cells.

   > **Note:** The total length of the bays must add to the overall width specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

   c. Click **OK**.
9. Click **Apply**.

M. Frame Models

The process of generating a Frame structural model and combining it with the existing STAAD.Pro structure is the same as for Trusses (see the above description).

M. To create a frame or continuous beam model

To create a continuous beam, bay frame, grid frame, or floor grid model, use the following procedure.

1. Select **Frame Models** from the **Model Type** drop-down list.
2. Select one of the rectangular frame or beam types by either:

   double-clicking the icon

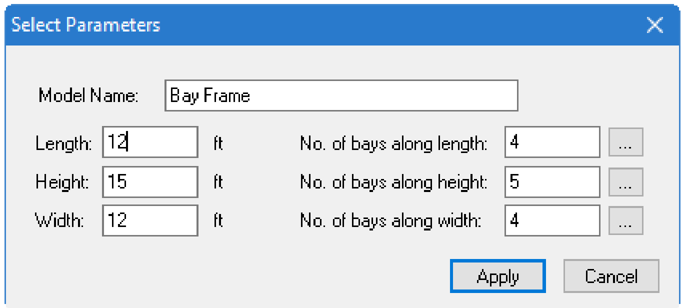
   or

   click-and-drag the icon to the view window

   Bay Frame – a 3D frame with no horizontal connecting members at the base
   Grid Frame – a 3D frame that has horizontal connecting members at all levels
   Floor Grid – a 2D frame in the XZ plane

Continuous Beam – a linear beam continuous over internal supports (i.e., the bay length)

The **Select Parameters** dialog opens.



3. (Optional) Type a **Model Name**.
4. Type the overall dimensions of the frame or beam:
   - **a.** Type the **Length**.
   - **b.** For bay frames and grid frames, type the **Height**.
   - **c.** For frames, type the **Width.**
5. Type the number of bays in each dimension:
   - **a.** Type the **No of bays along the length**.
   - **b.** For bay frames and grid frames, type the **No of bays along the height**.
   - **c.** For frames, type the **No of bays along the width**.
6. (Optional) To use unequal length division in any direction:
   - **a.** Click the **[...]** button adjacent to that field.
     A dialog opens.
   - **b.** Type the length of each bay.
   - **c.** Click **OK**.

> **Note:** The total length of the bays must add to the overall length specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

7. Click **Apply**.

M. To create a cylindrical frame or beam

To create a cylindrical frame or circular beam, use the following procedure.

1. Select **Frame Models** from the **Model Type** drop-down list.
2. Select one of the cylindrical frames or beam types by either:

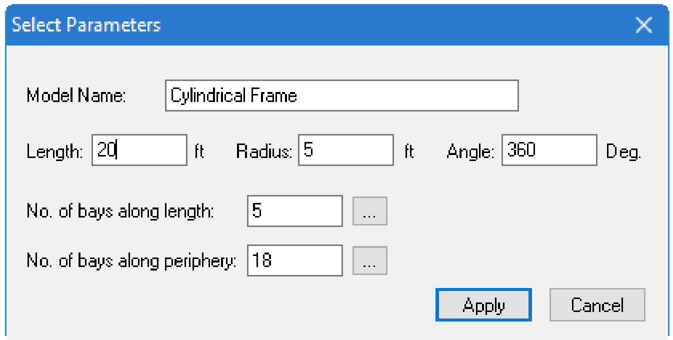
   double-clicking the icon

   or

   click-and-drag the icon to the view window

   Cylindrical Frame — a cylindrical frame with the cylinder height along the Z axis
   Reverse Cylindrical Frame — a cylindrical frame with the cylinder height along the Y axis
   Circular Beam – a single layer of coordinates which form a beam along a circular arc (i.e., polar coordinates)

   The **Select Parameters** dialog opens.



3. (Optional) Type a **Model Name**.
4. For frames, type the total **Length** along the central axis of the cylinder.
5. Type the **Radius** and the sweep **Angle** of the frame.

Tip: For a full circular frame or beam, leave the Angle 360.

6. For frames, type the **No. of bays along length**.
7. (Optional) For frames with irregular spaced bays along the length:
   a. Click the **[...]** button adjacent to the **No. of bays along length** field.
      A dialog opens.
   b. Type the length of each bay.
   c. Click **OK**.

Note: The total length of the bays must add to the overall length specified in the **Select Parameters** dialog. The precision should be to about 3 decimal places (e.g., 11.333).

8. Type the **No of bays along periphery** to specify the number of spaces along the circumference of the circle.
9. (Optional) For frames with irregular spaced bays along the length:
   a. Click the **[...]** button adjacent to the **No. of bays along periphery** field.
      A dialog opens.
   b. Type the angle subtended by each bay.
   c. Click **OK**.

Note: The total of the angles of all the bays must sum to equal the total Angle given in the **Select Parameters** dialog.

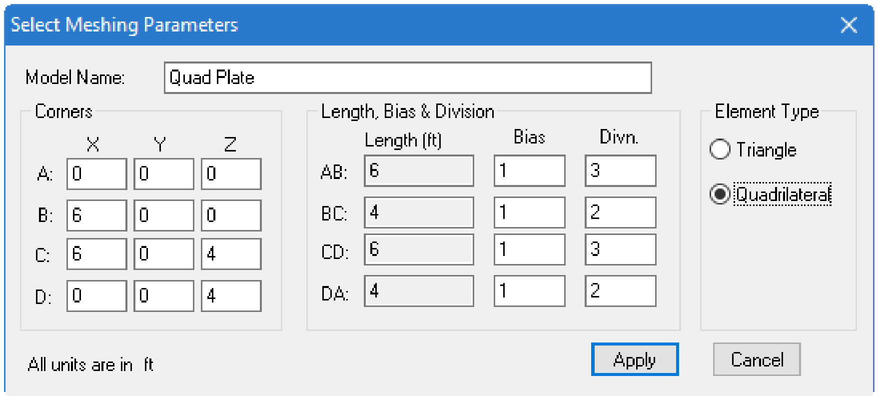
10. Click **Apply**.

M. Surface or Plate Models

The surfaces like Quadrilateral Plate, Cylindrical Surface, Hyperbolic Paraboloid Shell, Polygonal plate with Holes and Circular plate with Holes can be created and meshed with the given parametric values.

M. Quadrilateral Plate

The Quadrilateral prototype can be used to mesh a quadrilateral surface into small plate elements. Select the **Quadrilateral** prototype under the model type **Surfaces**. Drag the item into the right-side window and release the button. The **Select Meshing Parameters** dialog box will appear to specify the parameters.



Corners Provide the relative coordinates of the corners of the Quadrilateral Surface or region you want to mesh.

Bias & Divisions Specify the number of divisions you want along all the edges and the respective biasing. The minimum and maximum limits of number of divisions of each side are 1 and 100 respectively. Two opposite sides may have different numbers of divisions. When the number of divisions for two opposite sides are different, the sum of all divisions must be an even number for quadrilateral elements.

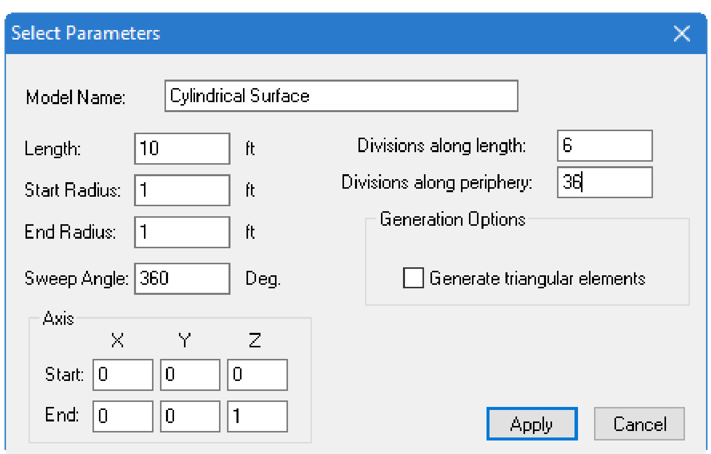
Bias If you want equal divisions along the length of a side, keep the Bias as 1. If your intention is to divide a side in such a way that the length of the last division is five times the length of the first division, specify Bias for that side as 5 along with the number of divisions.

Biasing may be negative. When negative Biasing has been specified, the side is divided so that the first division length is the value of the Biasing times the last division length.

Element Type Select the appropriate radio button depending on whether you want to mesh the region in smaller **Triangular Elements** or **Quadrilateral Elements**.

M. Cylindrical Surface

Select the **Cylindrical Surface** prototype under the model type **Surfaces.** Drag the item into the right-side window and release the button. The **Select Parameters** dialog box will appear to specify the parameters as shown in the next figure.



The Cylindrical Surface can be generated by providing the basic geometrical parameters **Length**, **Start Radius**, **End Radius** and **Sweep Angle**. You can generate a tapered cylinder by providing unequal values at **Start** and **End radius**. Entering a **Sweep Angle** less than 360 degree can also generate a longitudinally cut cylinder.

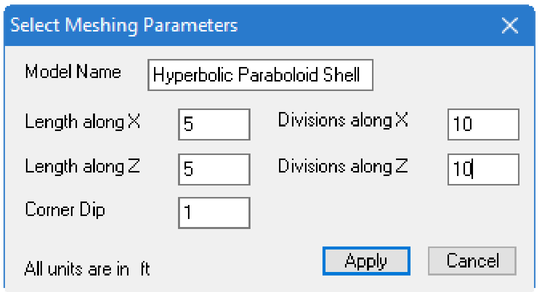
The **Axis** of the cylinder can also be controlled by entering the values of **Start -** X, Y, Z values and **End -** X, Y, Z values. The direction of the line joining the points gives the axis of the cylinder.

Providing the number of **Divisions along Length** and **Divisions along periphery**, the mesh can be generated. The option, **Generate open cylinder** will keep the ends open and **Generate triangular element** will generate triangles instead of quadrilaterals.

M. Hyperbolic Paraboloid Shell

Select the **Hyperbolic Paraboloid Shell** prototype under the model type **Surfaces.** Drag the item into the right-side window and release the button. The **Select Meshing Parameters** dialog box will appear to specify the parameters.

The basic geometrical parameters for generating a Hyperbolic Paraboloid Surface are the length of the two adjacent sides and the **Corner Dips**. The meshing parameters are the divisions along the adjacent edges. Click **Apply** after specifying all the parameters to generate the model.



M. Spherical Surface

A Spherical Surface can be generated and meshed with this feature. Three types of Spherical Surfaces may be generated, viz., **Spherical Cap**, **Spherical Surface** and **Spherical Region**. The **Diameter of Sphere** is required for all those cases. **Base Diameter** is required for Cap and Region only. **Top Diameter** is required for Spherical Region only. For Spherical Surface you have to define the **Start Angle** and **End Angle** in degrees for both the directions **- Latitude** and **Longitude.**

For the purpose of mesh generation, the number of **Divisions** in both the directions is to be specified. Click **Apply** after specifying all these parameters to generate the model.

The **Select Meshing Parameters** dialog box is shown below in the case of a Spherical Surface.

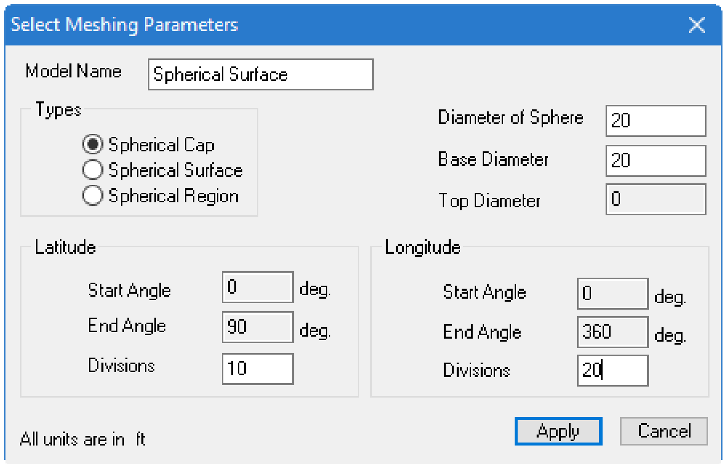
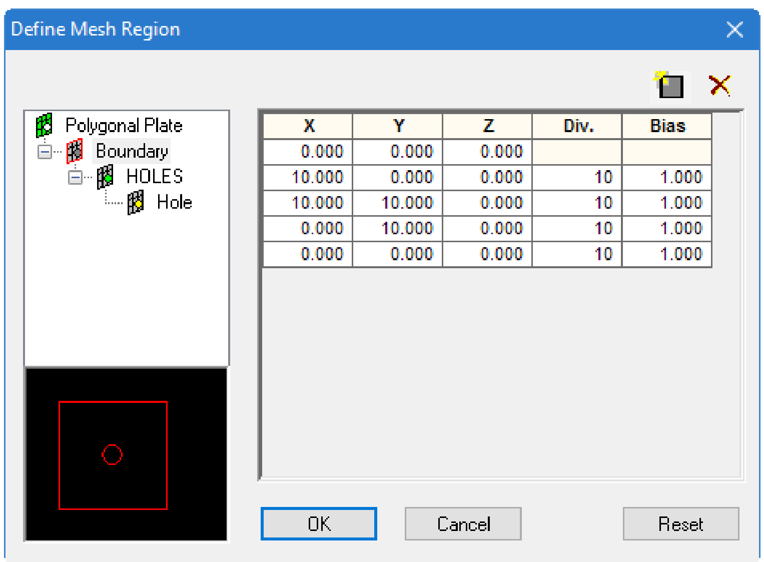


Figure 104:

M. Polygonal Plate with Holes

Select the **Polygonal Plate with Hole** prototype under the model type **Surfaces/Plates**. Drag the item into the right-side window and release the button. The **Define Meshing Region** dialog box will appear as shown below.

The Polygonal prototype allows us to mesh a polygonal surface, with or without different kinds of holes inside the boundary, into small triangular plate elements.



Boundary

The Boundary tab as shown above in the **Define Mesh Region** dialog box provides us an option to specify the **location of Corners, Number of Divisions** of each side and Bias of each side's division for the Boundary to mesh the surface.

X, Y, Z are the coordinates of the various corners of the surfaces. The sides will be defined as joining the various corners. **Div.** option is to define the divisions of various sides of the polygonal surface to be considered for generating the surface meshing. The **Bias** option helps the user to create divisions having unequal spacings among them.

On the top of the right side of this dialog box there are two icons namely, **Add New Row** and **Delete Row**. Clicking on the first icon will add a new row enabling us to specify the coordinates of the corner of the polygonal surface. Clicking on the second icon will enable us to delete an existing row.

Holes

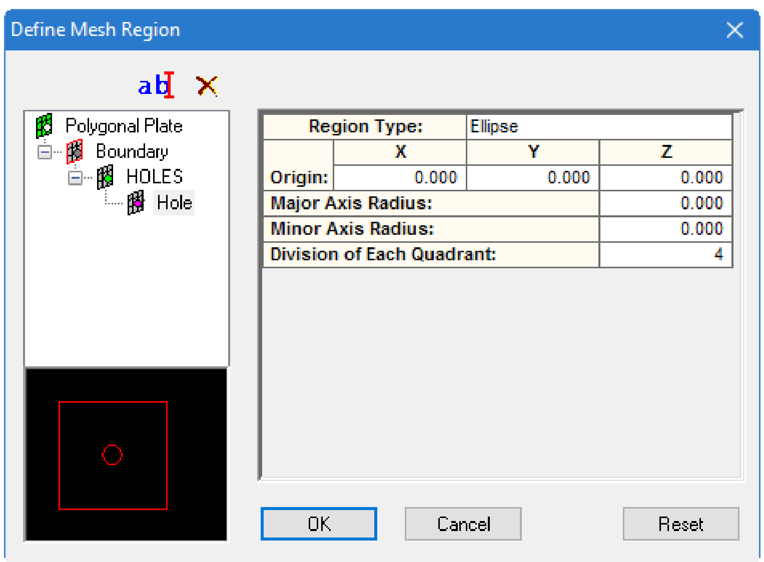
After creating the boundary, the user can start introducing the holes in the plate. In STAAD, we may create circular, elliptical as well as polygonal holes.

When we click on the **HOLES** option, two icons appear as shown below namely, **Add New Hole** and **Delete All Holes**.

Add New Hole	This icon allows us to add new holes to the surface.
Delete All Holes	This icon allows us to remove all the existing holes from the surface. Once a new hole is added, the name of the that hole appears under **HOLES** in a tree view. A new icon titled **Rename** Hole will now appear which will enable us to rename the newly created hole.

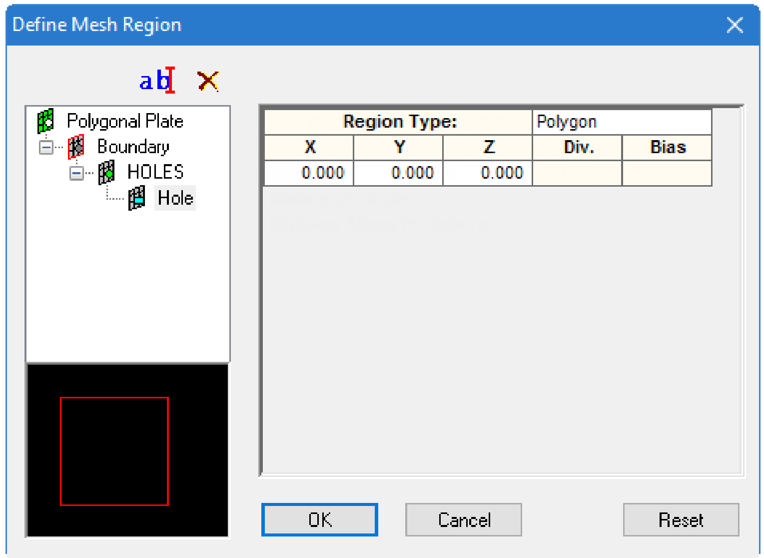
Hole

To create a hole within the boundary, click this option. This is the default name of the hole. When we click on Hole, a table will be displayed on the right side of the dialog box, where we can define the geometry of the hole. Specify what kind of hole we want to create from among the three available options namely Polygon, Circle and Ellipse, in the **Region Type**, as shown in the next figure. For each hole, one tab will be created automatically in the **Define Meshing Region** dialog box to allow us to specify different parameters for that hole.

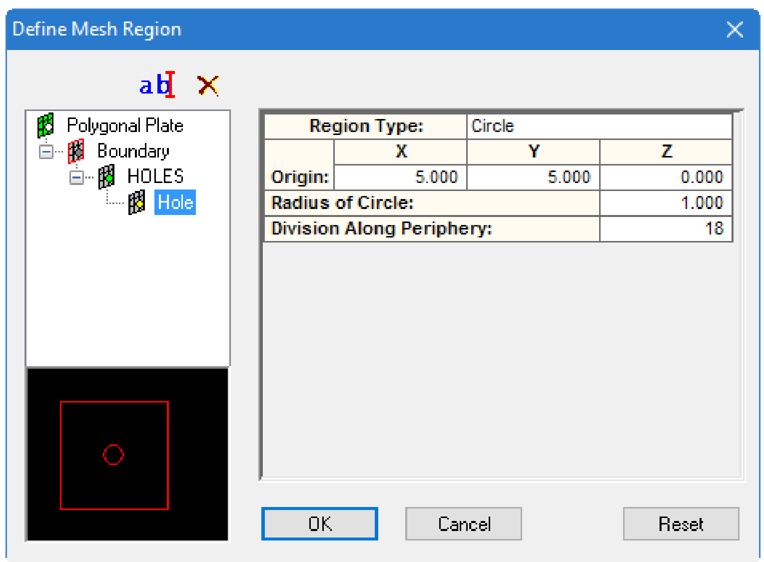


The creation and editing of the region types are explained below.

Polygon	This option is used to define polygonal holes. The geometry of the polygonal holes is generated in the same way as the geometry of the polygonal. Please refer to the description under polygonal surfaces for details on defining the geometry of polygonal holes.



Circle This option is used to define circular holes. The parameters for defining the geometry of circular holes are described below:

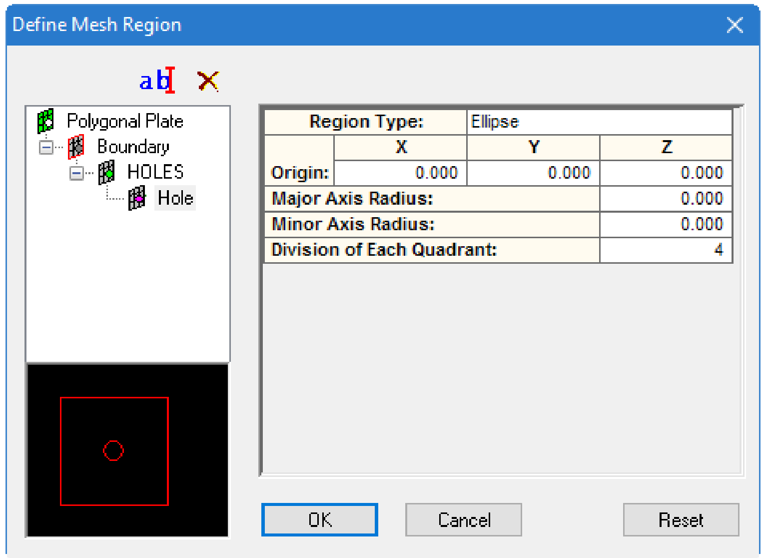


Origin We have to specify the X, Y, and Z coordinates in its proper places to define the position of the center of the circular hole.

Radius of circle The value in this box will define the radius of the circular hole.

Division Along Periphery The value in this box defines the number of divisions to be done along the circumference of the circular hole. This information will be considered in mesh generation of the surface

Ellipse This option is used to generate ellipsoidal holes. The parameters to define the geometry of ellipsoidal holes are as follows:



Origin	We have to specify the X, Y, and Z coordinates in its proper places to define the position of the center of the ellipsoidal holes.
Major Axis Radius, Minor Axis Radius	The values in these boxes will define the major axis and the minor axis of the ellipsoidal holes.
Division of each Quadrant	The values specified in this box specifies the number of divisions of each quadrant. This information will be used for the mesh generation of the surface.

Once all the parameters have been specified, clicking on **OK** to carry out the mesh generation operation of the surface. Clicking on **Cancel** will cancel the mesh generation operation. Clicking on Reset will reset the parameters provided at the beginning.

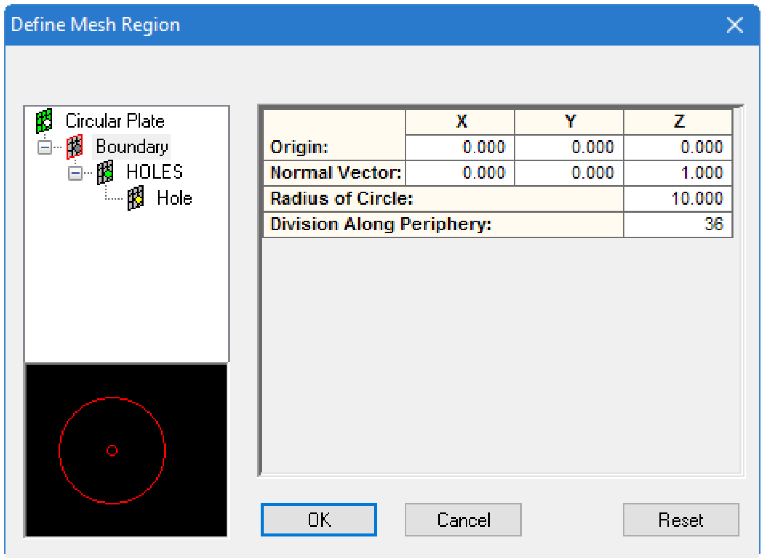
Note: The one limitation of the Polygonal Surface Meshing is that the total number of nodes must not exceed 15000.

M. Circular Plate with Hole

Circular Plate with hole is a special case of Polygonal Plate with Hole, where the Boundary is circular, keeping the other parameters unaltered.

Select the **Circular Plate with Holes** prototype under the model type **Surfaces/Plates**. Drag the item into the right-side window and release the button. The **Define Meshing Region** dialog box will appear as shown in the next figure.

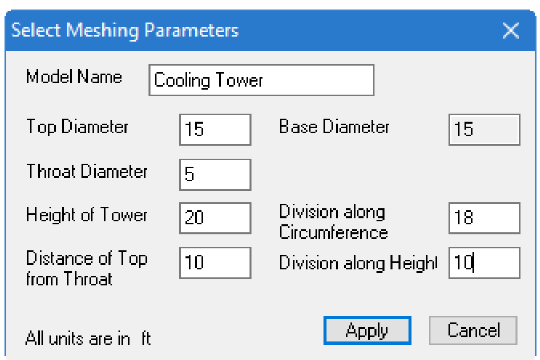
The basic parameters required to define the circular plate geometrically are the location of the center (**Origin**) and **Radius**. To specify the plane of the circular plate the **Normal Vector** can also be defined. As the meshing parameter, **Number of Division along Periphery** is required.



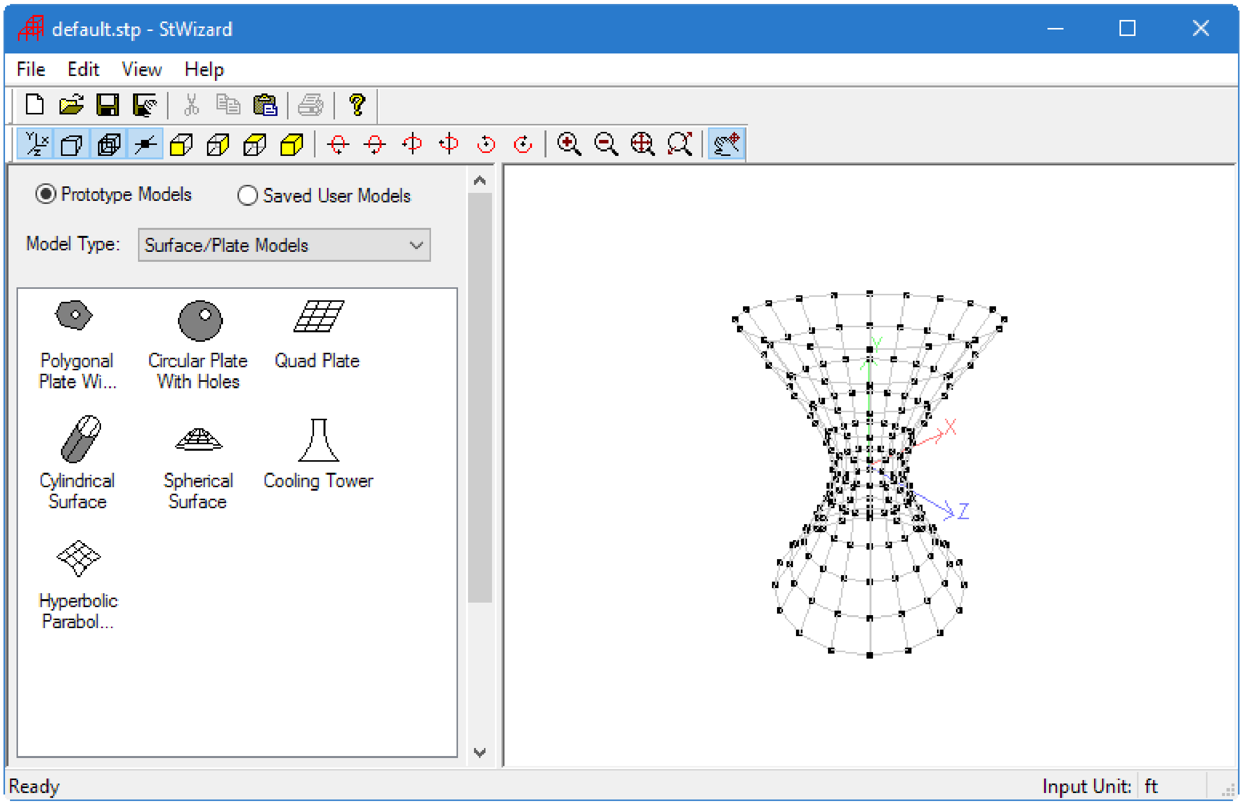
M. Cooling Tower

A hyperbolic paraboloid shaped Cooling Tower may be generated and meshed with the parameters displayed in the **Select Meshing Parameters** dialog box as shown in the next figure.

The basic geometrical parameters required to model such a structure are Top Diameter, Throat Diameter, Height of Tower and Distance of Top from Throat. The Base Diameter is internally calculated and displayed to the user. For the purpose of mesh generation, the number of Division along Circumference and Division along Height is to be specified.



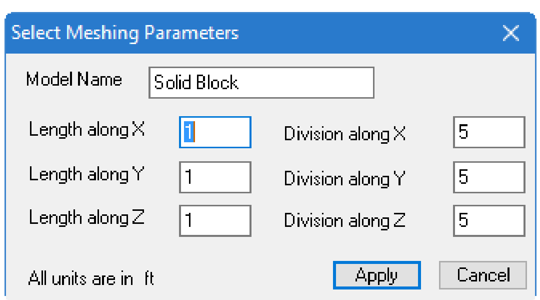
A complete model of a Cooling Tower is shown below.



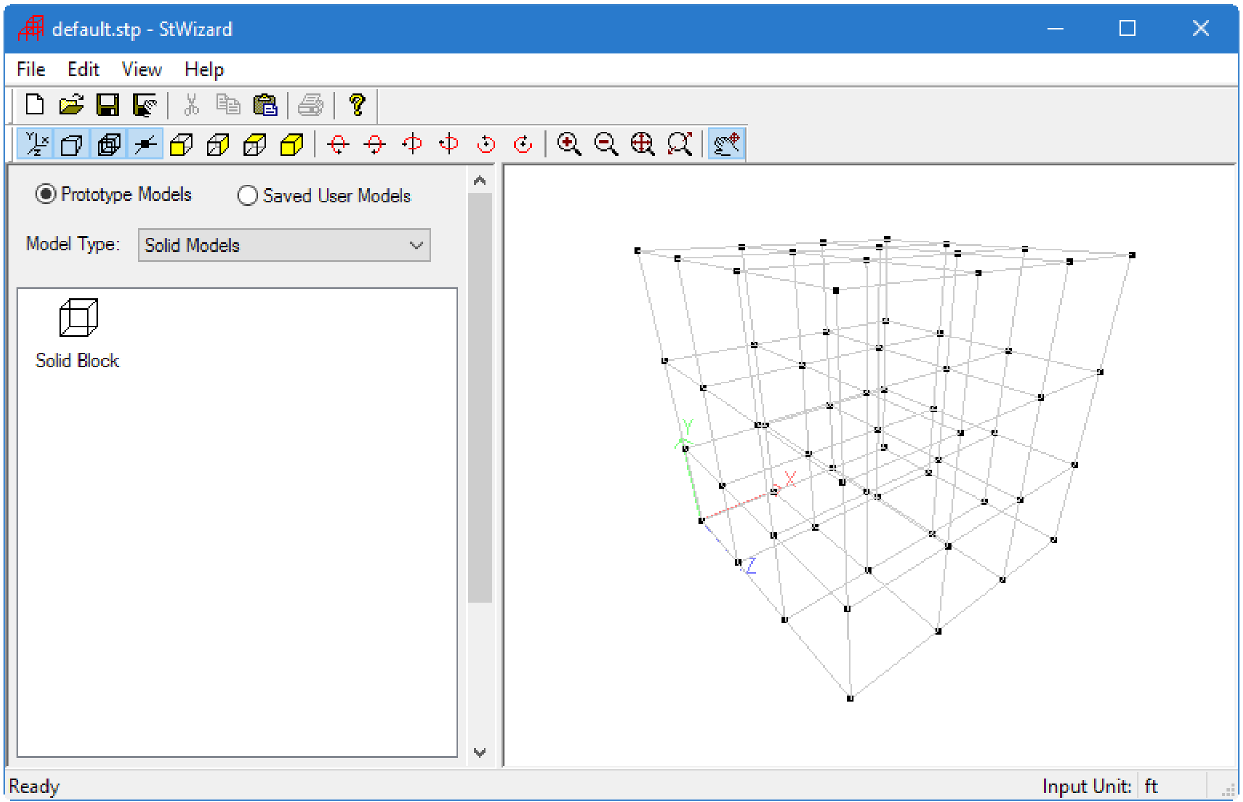
M. Solid

A Solid rectangular block can be generated and meshed with the parameters displayed in the **Select Meshing Parameters** dialog box as shown below.

The required parameters are the **Lengths along X**, **Y** and **Z** directions and the corresponding **Number of Divisions** along those edges.



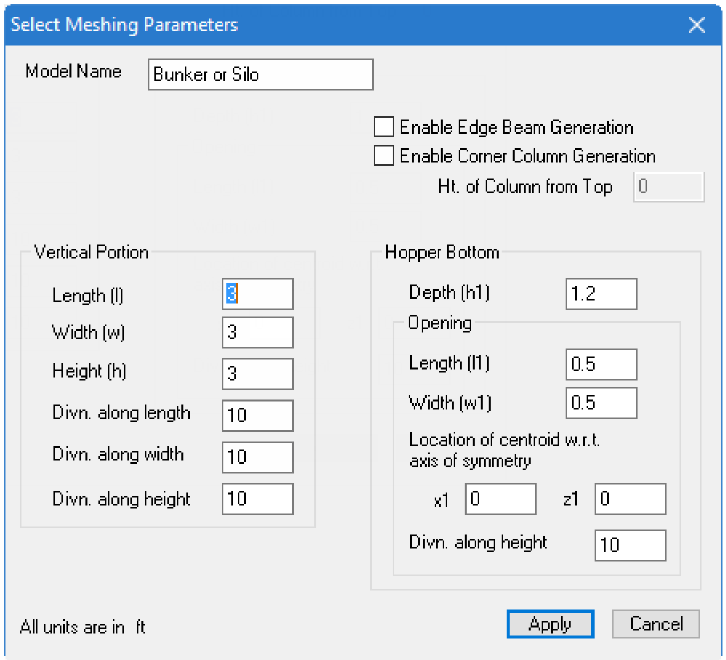
A Solid rectangular block is shown below.



M. Composite Model: Bunker or Silo

A Bunker or Silo can be modeled parametrically and meshed with this feature.

Select the **Bunker or Silo** prototype under the model type **Composite Models**. Drag the item into the right-side window and release the button. The **Select Meshing Parameters** dialog box will appear as shown below.



To define the geometry of the Vertical Portion, you have to define the **Length**, **Width** and **Height** of that portion along-with the number of **Divisions** along them for the purpose of mesh generation.

The **Depth of the Hopper Bottom** is required with the number of Divisions along depth. The **Length** and **Width** of the opening at the bottom is required and to locate this opening the **shift of Centroid** of the opening w.r.t. the axis of symmetry of the Silo is required.

Checking the box **Enable Edge Beam Generation** can generate edge beams and also checking the box **Enable Corner Column Generation** can generate Corner Columns, for which the **Height of Column from Top** is to be provided.

M. Import CAD Model

This feature, Import CAD Model, has two separate utilities, Scan DXF and STAAD Models.

M. To generate geometry from a DXF file

This is a utility which allows the geometry (nodes, members, plates, etc.) of a previously created model to be imported and altered.

This feature supports a limited number of CAD entities like Line, 3D-Polyline, and 3D-Face.

If the geometry of a model is created using a drawing program like AutoCAD and saved in a DXF file, it can be imported into STAAD using this utility.

1. Select **Import CAD Models** from the **Model Type** drop-down list.
2. Either:

   Drag-and-drop the **Scan DXF** icon into the view window. 

   or

Double-click the **Scan DXF** icon.

The **Open** dialog opens.

3. Select the DXF file you want to use and click **Open**.
The drawing geometry is opened in Structure Wizard.

M. To generate geometry from a STAAD model

This feature can be used to import an existing model which can then be quickly scaled up or down in the Structure Wizard.

1. Select **Import CAD Models** from the **Model Type** drop-down list.
2. Either:

   Drag-and-drop the **STAAD Model** icon into the view window. 

   or

   Double-click the **STAAD Model** icon.

   The **Open** dialog opens.
3. Select the STAAD input file (`.std`) you want to use and click **Open**.
The model geometry is opened in Structure Wizard.

M. VBA Macro Models

This feature allows you to parametrically define models using a VBA Macro.

Two sample files are supplied with the program.

Tip: Select **File** > **Set as Top Most** to turn *off* that feature before launching a macro. Otherwise, the macro form may be hidden beneath the **Structure Wizard** application window.

- Dragging a macro icon from the left side of the window to the right to launch the macro.
- Double-click an icon on the left-hand side to open the code in the VBA editor. You can then make changes and save the file to suit your needs.

Related Links

- *EX. Structure Wizard Macro Example Files* (on page 6253)

M. To add a new plugin

1. Either:

   Select **File** > **Add Plugin**

   or

   Right-click in the model types list and select **Add Plugin** from the pop-up menu.

   The **Select an existing Macro** dialog opens.
2. Select the Visual Basic Macro file (file extension `.vbs`) you want to add and click **Open**.
The new macro is added to the list of plugins in the model types list.

M. To edit model parameters

1. Either:

   Right-click in the view window and select **Change Property** from the pop-up menu.

   or

   Select **Edit** > **Change Property**.

   or

   Double-click on the model view window.
   The **Select Parameters** dialog opens.
2. Change one or more parameters as needed..
3. Click **Apply**.

M. To rescale a model

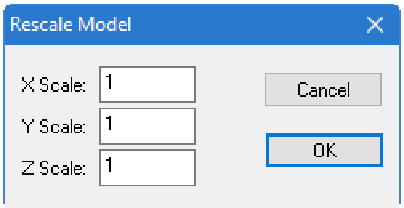
1. Either:

   Right-click in the view window and select **Scale** from the pop-up menu.

   or

   Select **Edit** > **Scale**.

   The **Rescale Model** dialog opens.



2. Type in corresponding Z, Y, and Z scale factors in the fields.
3. Click **OK**.

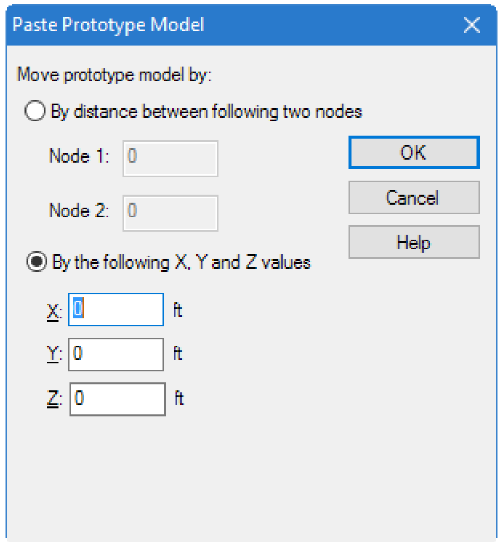
M. To delete an entire model

Right-click in the view window and select **Delete** from the pop-up menu

M. To transfer the generated model to STAAD.Pro

1. Select **File** > **Merge Model with STAAD.Pro Model**.
   A dialog prompts you to confirm you want to merge this prototype with your current STAAD.Pro model.
2. Click **Yes**.

   The **Paste Prototype Model** dialog opens.



3. (Optional) Specify the method of offset:

Tip: This is useful, for example, to place a prototype truss model at the top of already modeled columns.

Move option	Do the following
By distance between following two nodes	Type the node numbers
By the following X, Y and Z values	Type the coordinates (in the indicated units) where to place the prototype origin in the global coordinates of the STAAD model.

4. Click **OK**.
   The model is placed.

In the event that nodes, members, or elements overlap existing similar objects in the STAAD model, a message dialog opens indicating that the duplicate objects have been ignored. Click **OK** to proceed.

M. To add items to the library

You can customize the list of model types using the following procedure.

1. Select the **Model Type** from the drop-down list.
2. Either:

   Select **File** > **Add Plugin**

   or

   right-click in the model type list and select **Add Plugin** from the pop-up menu
   The **Open** dialog opens.
3. Select the library file (file extension `.dll`) for the plugin and click **Open**.
4. (Optional) To rename a selected model generator, do the following:
   a. Select a model generator and either:

      Select **Edit** > **Rename Model Generator**

or

right-click on the model generator and select **Rename Model Generator** from the pop-up menu

The **Select Name** dialog opens.

b. Type a new name.

c. Click **OK**.

5. (Optional) Save the library of prototypes to a new file:
   a. Select **File** > **Save As**.
      The **Save As** dialog opens.
   b. Type a new file name for the StWizard Model Files (file extension `.stp`) and click **Save**.
   c. You can now open this file again by selecting **File** > **Open**.

M. Pages in the Analytical Modeling Workflow

- Geometry – used to layout your structure, including nodes, beams, plates, and solids

> **Note:** If you are using the Physical Modeling workflow, the majority of the tools on this page will be disabled. Model geometry for physical models must be updated in the STAAD.Pro Physical Modeler interface.

- Properties
- Specifications
- Supports
- Loading
- Analysis
- Design

Nodes table

Used to display a table of nodes with coordinates for adding and editing nodes. The table may also be used to highlight the node on the graphical view of the structure.

The *Nodes* table lists the node numbers and their coordinates. This table can be used to define new nodes and edit existing node-coordinates. If we click on a node number in this table, the node gets highlighted in the table as well as in the structure view..

Opens when the **Geometry** page is selected in the **Analytical Modeling** workflow.

Node	Lists the node numbers in numerical order.
X	The X coordinates of the nodes in the currently selected unit. Changing this immediately changes the X-coordinate of the node in the graphical display also.
Y	The Y coordinates of the nodes in the currently selected unit. Changing this immediately changes the Y-coordinate of the node in the graphical display also.
Z	The Z coordinates of the nodes in the currently selected unit. Changing this immediately changes the Z-coordinate of the node in the graphical display also.

Goto <object> dialog

Used to go to a specific node within the **Nodes**, **Beams**, etc. table.

Opens when **Go To** is selected from the right-click pop-up menu on many tables.

Type the number of a node, beam, etc. and then click **OK**. The table will highlight that entry.

Node Supports

Lists all nodes for which supports have been defined along with the type of support.

M. Properties and Specifications

M. Section Profiles

Beam members must have a section profile assigned for analysis.

STAAD.Pro supports a wide variety of section profiles, including catalog sections, prismatic sections, tapered member sections, and so on.

M. Section Database Profiles

STAAD.Pro comes with an extensive, built-in database of common catalog sections for many countries.
Related Links

- *TR.20.1 Assigning Properties from Steel Tables* (on page 2257)

M. To add a new table section property

To add a table section property from a standard catalogs for steel, cold-formed steel, timber, or aluminum, use the following procedure.

These are also referred to as the “built-in” tables of sections in STAAD.Pro.

1. Either:

   select the **Database** tool in the **Specifications** group on the **Specifications** ribbon tab

   or

   select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog

   Tip: Selecting the country from the **Database** tool drop-down list first results in a faster dialog load time.

   The **Section Profile Tables** dialog opens.
2. Select the appropriate table:
   a. Select the material tab at the top of the dialog:

      Steel
      Coldformed Steel

Timber
Aluminium

b. Select the country catalog and shape profile on the left-hand side of the dialog.

> **Tip:** If you selected the material and country using the **Database** tool drop-down, then only that material and country tables are shown in the dialog.

3. Select the shape profile on the left-hand side of the dialog.
4. Either:

   select the specific shape (i.e., beam, profile, shape, section, etc.) in the list

   or

   select to define a shape for some (i.e., pipes)

> **Note:** Refer to related codes in the Design (on page 969) for information on section nomenclature.

5. Select the appropriate **Type Specification** for some shapes.
   Some type specifications require additional input (such as cover plate dimensions, spacing, composite slab data, etc.).
6. (Optional) Select an associated **Material**.
   This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.
7. Either:

To...	Do the following...
add the section to the model for later assignment	click **Add**.
add the section to the model and assign to the current member selection	click **Asign**.

8. (Optional) Repeat steps 2 through 7 as needed to add more sections to the model.
9. Click **Close**.

The section and, if selected, associated material, is added to the **Properties - Whole Structure** dialog.

If you did not assign the section to a selection set, you must select the section label in the **Properties - Whole Structure** dialog and assign it using one of the assignment methods.

Related Links

- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *Properties - Whole Strucutre dialog* (on page 2793)
- *TR.20 Member Property Specification* (on page 2255)
- *Section Profile Tables dialog* (on page 2793)
- *TR.20.1 Assigning Properties from Steel Tables* (on page 2257)

M. To add an American steel joist section

To add an American steel joist section and, optionally, have it sized by span and load, use the following procedure.

STAAD.Pro includes the facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called “Fortieth edition standard specifications, load tables and weight tables for steel joist and joist girders.” Member properties can be assigned by specifying a joist designation contained in tables supplied with the program, including the following joists and joist girder types:

Open web steel joists – K series and KCS joists
Longspan steel joists – LH series
Deep Longspan steel joists – DLH series
Joist Girders – G series

Tip: It is recommend that use the default selection of **Steel** material definition with joists.

1. Either:

   select the **Database** tool in the **Specifications** group on the **Specifications** ribbon tab

   or

   select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog

   Tip: Selecting the country from the **Database** tool drop-down list first results in a faster dialog load time.

   The **Section Profile Tables** dialog opens.
2. Select the **Steel** tab and then the **American Steel Joist** entry on the left-hand side.
3. Select the series of joists you want to use.
4. (Optional) To have the program select a least-weight option of joist for you, do the following:
   a. Type the search criteria (differs by joist series) in the **Find suitable joist section** fields.
   b. Click **Find Section.**

   For example, if you select the LH series of joists and then specify a Clear Span of 35 ft, a Depth Limit of 24 in (2ft), a Total Load of 750 lbs/ft (0.75k/ft), a Live Load of 500 lbs/ft (0.5k/ft), and a Deflection Limit of L/360, the program will select a **24LH09** when you click **Find Section.**
   The least weight suitable joist in the joist series will be selected. If not suitable joist can be found in the series, a message dialog opens.
5. Either:

To...	Do the following...
add the section to the model for later assignment	click **Add.**
add the section to the model and assign to the current member selection	click **Asign.**

6. Click **Close.**

Example Input

An example of a structure with joist:

```
STAAD SPACE EXAMPLE FOR JOIST GIRDER
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0
3 30 10 0; 4 30 0 0
MEMBER INCIDENCES
```

```
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W21x50
MEMBER PROPERTY SJIJOIST
2 TABLE ST 22K6
CONSTANTS
E STEEL ALL
DENSITY STEEL ALL
POISSON STEEL ALL
SUPPORTS
1 4 FIXED
UNIT POUND FEET
LOAD 1
SELFWEIGHT Y -1
LOAD 2
MEMBER LOAD
2 UNI GY -250
LOAD COMB 3
1 1 2 1
PERF ANALY PRINT STAT CHECK
PRINT SUPP REAC
FINISH
```

Related Links

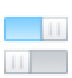
- *Section Profile Tables dialog* (on page 2793)
- *G.6.6 Steel Joist and Joist Girders* (on page 2118)

M. To assign catalog section to physical members

To select and assign a catalog section to physical members, use the following procedures.

Note: For a physical member whose physical member property is already assigned, the individual analytical members in that Physical member will adopt the same member property that of the physical member. However, in case, where the analytical member property is assigned to any member in that physical member, then this analytical member property will supersede physical member property.

1. Select the Properties page.
   The **Properties - Whole Structure** dialog opens.
2. On the **Geometry** ribbon tab, select the **Toggle Physical Member Mode** tool in the **Physical Members** group.

   The tool is highlighted to indicate the mode is active.
3. Either:

   select the **Database** tool in the **Specifications** group on the **Specifications** ribbon tab

or

select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog

Tip: Selecting the country from the **Database** tool drop-down list first results in a faster dialog load time.

The **Section Profile Tables** dialog opens.

4. Select the appropriate table:
   a. Select the material tab at the top of the dialog:

      Steel
      Coldformed Steel
      Timber
      Aluminium
   b. Select the country catalog and shape profile on the left-hand side of the dialog.

Tip: If you selected the material and country using the **Database** tool drop-down, then only that material and country tables are shown in the dialog.

5. Select the shape profile on the left-hand side of the dialog.
6. Either:

   select the specific shape (i.e., beam, profile, shape, section, etc.) in the list

   or

   select to define a shape for some (i.e., pipes)

Note: Refer to related codes in the Design (on page 969) for information on section nomenclature.

7. Select the appropriate **Type Specification** for some shapes.
   Some type specifications require additional input (such as cover plate dimensions, spacing, composite slab data, etc.).
8. (Optional) Select an associated **Material**.
   This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.
9. Click **Add**.
   The new member property is added to the list as a Physical member property in the **Properties - Whole Structure** dialog. Such properties are designated with the word "Physical".

Note: Only the Physical Member properties can be assigned to a physical member, and no analytical member property can be assigned to a physical member.

10. Click **Close**.

You must assign the physical member properties to physical members using one of the standard assignment methods from the **Properties - Whole Structure** dialog.

Related Links

- *M. Physical Members* (on page 663)

M. Section Database Manager

This utility application is used to manage the section catalog databases used in STAAD.Pro.

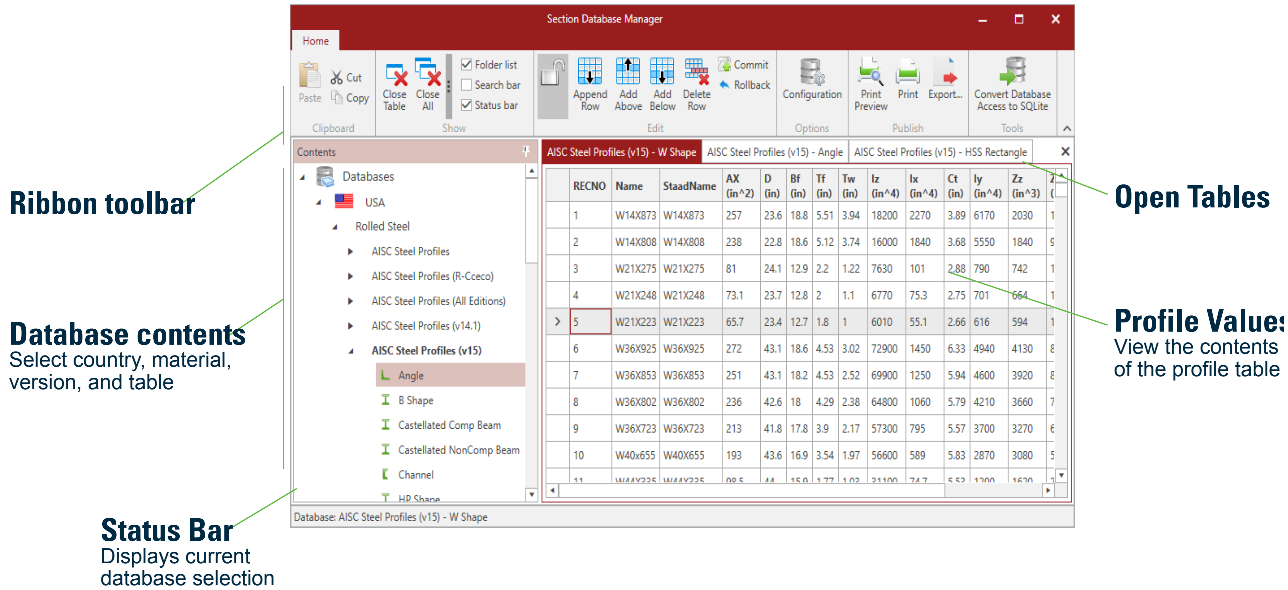


Figure 105: Section Database Manager Window Overview

M. To change a default section database

To change the section database used the default for a particular country and material combination, use the following procedure.

The latest edition of each country's database is selected as the default value. However, to use older editions of catalogs (e.g., when historical sections in the AISC database are needed), you may want to change the default catalog used. Please note that some catalogs are significantly larger, such as the “AISC Steel Profiles (All Editions)” and this can slow dialog loading times.

This procedure is also used if your organization has developed a custom section database.

1. Select the **Section Database** tool in the **Tools** group on the **Specifications** ribbon tab.

   The **Section Database Manager** application opens.
2. On the **Home** ribbon tab, select the **Configuration** tool in the **Options** group.

   The **Default Database Configuration** dialog opens.
3. (Optional) Select the **Default Country** from the drop-down list.

   This will be the first country expanded when the **Section Profiles Tables** dialog open is opened.
4. Select the tab corresponding to the material type.
5. For a particular **Country**, select a default database from the drop-down list in the **Selected Database** field.
6. Repeat steps 4 and 5 to change additional default databases.
7. Click **Save**.

8. Click **Close**.

Tip: Alternatively, you can navigate to the database you want to use as the default for a particular country and material combination in the Contents panel. Then, right-click and select **Set as Default Database** from the pop-up menu.

M. To add a custom table section property

To edit the section database tables to include custom sections, use the following procedure.

Warning: It is strongly recommended that you do not modify the database files that are installed with STAAD.Pro. These files are typically overwritten when the program is updated. Instead, you can create your own custom copy and modify that database file.

Adding a section to a database table is useful if you need to commonly use that custom section in many models or projects.

This procedure is not the same as adding a user-defined (UPT) section or catalog. This should only be performed if you need to make edits to the standard shape catalogs that are included with STAAD.Pro. Using a UPT section allows you to enter dimensions and, for many sections, have the program calculate section values.

1. Select the **Section Database** tool in the **Tools** group on the **Specifications** ribbon tab.

   The **Section Database Manager** application opens.
2. In the Databases list, double-click on the country, material, catalog, and table you want to edit.
   The selected table opens in read-only mode.
3. Select the **Lock** tool.
   The table is unlocked for editing.
4. Either:

To...	Do the following...
add a row to the end of the table	select the **Append Row** tool
add a row above the current table row	select the **Add Above** tool
add a row below the current table row	select the **Add Below** tool

   The order of the table here is how it will also be presented in the **Section Profile Tables** dialog in STAAD.Pro.

   An empty row is added to the table.
5. Type a record number (RECNO), name, STAAD.Pro nomenclature name (StaadName), and then the section values for each column in the table.

   You can press **<Tab>** to move to the next column within a row when entering values.
6. (Optional) Repeat steps 4 and 5 as many times as needed to add additional shapes.
7. Click **Commit** to save the changes to the database.
8. Select the **Unlock** tool.
   The database is now locked against further editing.
9. Close the SectionDBManager program.

You can now use this section just as you would any default catalog section in STAAD.Pro.

Note: You can use the **Delete Row** tool to remove the current selected table row. Exercise caution when using this tool!

M. To add a custom section database

To create a custom section database for use in STAAD.Pro, use the following procedure.

Note: This procedure involves editing SQLite files and requires third-party software and a basic understanding of updating SQL queries.

Warning: It is strongly recommended that you do not modify the database files that are installed with STAAD.Pro. These files are typically overwritten when the program is updated. Instead, you can create your own custom copy and modify that database file.

The database files used in STAAD.Pro have a specific structure. Therefore, you should start by copying an existing section database file. Attempting to create a new section database file from scratch is strongly discouraged.

The best practice is to identify the section database that includes tables *most similar* to your custom table requirements. You can change the country of the section database, but it is recommended to start with a source database from the same country you intend to use as your custom database.

1. Copy an existing section database file:
   - a. In Windows Explorer, navigate to `C:\ProgramData\Bentley\Engineering\STAAD.Pro CONNECT Edition\Sections` (typical installation location).
   - b. Copy the appropriate section database file (extension `.db3`) to a convenient location.
   - c. Give this copied file a new file name.
2. Use a SQLite database editor program to update the database values:

   You must use a third-party SQLite database editor such as SQLiteStudio or SQLite Administrator. Details on the use of these products is beyond the scope of this documentation.
   - a. **Required:** Update the sections database name to something unique.

     This will be used to identify the sections database in STAAD.Pro so it must be different from existing databases, both those installed with the program and other custom databases. The section database files contain a table named “DBInfo” that contains the name used in STAAD.Pro.

     Typically, this can be done by running the SQL query: `UPDATE 'DBInfo' SET name='`*`custom database name`*`'`, where *`custom database name`* is the unique name you want to use.
   - b. (Optional) Update the field units if necessary.

     The units for each section value are maintained outside of the section tables in a table named “Field Units”. If these need to be changed (e.g., you are creating a metric version of an imperial table), then these must also be updated using a SQLite editor application.
   - c. If you need to add additional field (columns) to a table, then this must be done in an SQLite editor.
   - d. Save the database if necessary.

     Typically, SQL queries such as UPDATE write the changes to the database and do not require saving.

     You can use the SQL query `SELECT * FROM 'DBInfo'` to view all the field values in that table to verify the name has been updated.
3. Select the **Section Database** tool in the **Tools** group on the **Specifications** ribbon tab.

The **Section Database Manager** application opens.

4. On the **Home** ribbon tab, select the **Configuration** tool in the **Options** group.

   The **Default Database Configuration** dialog opens.
5. Select and add your custom database:
   a. Click **Browse** and then navigate to where you saved your custom database file.
   b. Select the file and click **OK**.

   > **Note:** Alternately, you can copy and paste the full file path to the database file.

   c. Click **Add**.
   An information dialog opens to confirm the database was added.
   d. Click **OK**.
   e. Click **Save**.
   An information dialog opens to confirm the Database INI file was updated.

The database is added and can now be edited.

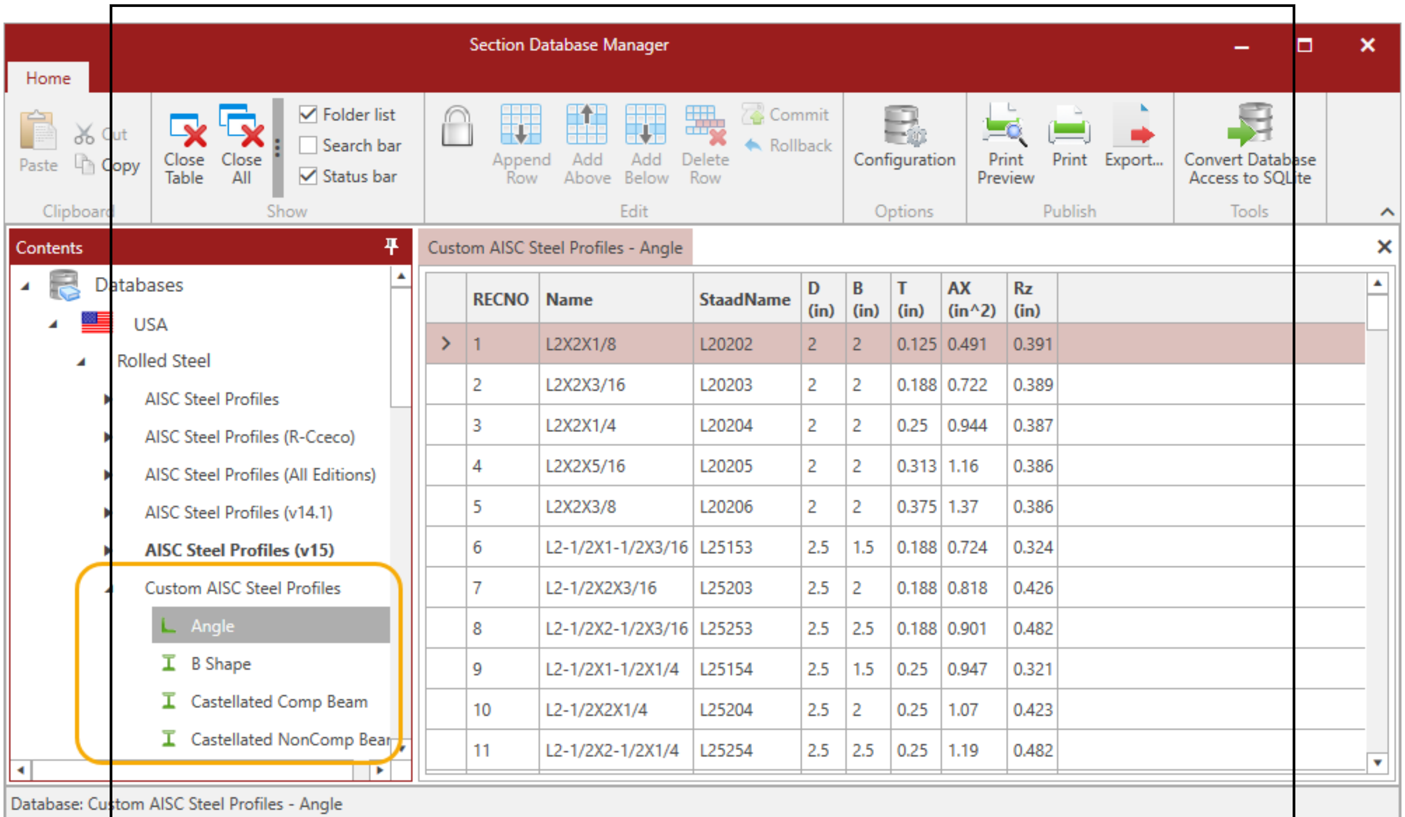


Figure 106: A custom database has been added to the Section Database Manager

M. To convert a legacy database

To migrate custom databases of shapes to the SQLite format, use the following procedure.

You can convert legacy Microsoft Access database files (file extension `.mdb` or `.accdb`) to SQLite database files.

1. Select the **Section Database** tool in the **Tools** group on the **Specifications** ribbon tab.

   The **Section Database Manager** application opens.
2. On the **Home** ribbon tab, select the **Convert Database Access to SQLite** tool in the **Tools** group.

   The **Convert Access Database to SQLite Database** dialog opens.
3. Either:

   type the complete path to your existing Access database file in the **Source** field

   or

click the **Browse** button to navigate to and select the existing Access database file

4. Either:

   type the complete path to and file name of the new SQLite database file in the **Destination** field

   or

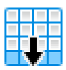
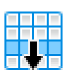
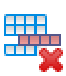
   click the **Browse** button to navigate to and type a file name for the new SQLite database file
5. Click **Convert**.

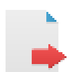
You now have an SQLite database you can utilize add as a custom database for use in STAAD.Pro CONNECT Edition.

Section Database Manager Ribbon toolbar

Table 24: Section Database Manager Home ribbon tab

Tool Name	Description	Shortcut
 **Paste**	Paste clipboard contents into the table.	**<Ctrl+V>**
 **Cut**	Copy the selection to the clipboard and delete the original.	**<Ctrl+X>**
 **Copy**	Copy the selection to the clipboard.	**<Ctrl+C>**
 **Close Table**	Closes the currently selected table.	
 **Close All**	Closes all open tables.	

Tool Name	Description	Shortcut
 **Unlock** / Lock	Unlocks the current table for editing. Once unlocked, the same tool re-locks the table to prevent further changes.	
 **Append Row**	Adds a new table row at the end of the table.	
 **Add Above**	Adds a new table row before the current selection.	
 **Add Below**	Adds a new table row after the current selection.	
 **Delete Row**	Deletes the currently selected row from the table.	**<Delete>**
 **Commit**	Saves all changes to the current table.	
 **Rollback**	Undoes all uncommitted changes made to the current table.	

Tool Name	Description	Shortcut
 **Configuration**	Opens the **Default Database Configuration** dialog, which is used to select the default country and section databases to use in the **Section Profiles Table** dialog.	
 **Print Preview**	Opens a print preview of the currently selected table.	
 **Print**	Prints the currently selected table contents.	**<Ctrl+P>**
 **Export**	Opens a Save As dialog, where you can choose a location and file format for exporting the current table contents to an external file.	
 **Convert Database Access to SQLite**	Opens the **Convert Access Database to SQLite Database** dialog, which is used to convert legacy section database files in Microsoft Access database format to SQLite database files.	

Default Database Configuration dialog
Used to select the default country and section databases to use in the **Section Profiles Table** dialog.

Opens when the **Configuration** tool is selected on the **Home** tab in the **Section Database Manager** application.

When an existing STAAD.Pro model is opened, the program reads the contents of the file, and checks the validity of data in that file. One of those data items validated is names of sections assigned from steel tables. Since steel sections are country-specific, such as British, German, etc., the program needs to know the country or organization whose steel table is the underlying database for validating the sections being read in from the file. Normally, the input file contains the name of the database as part of the member property command. In the absence of an explicit name, STAAD.Pro uses a default. That default is set using this facility.

Section Profile Table tab

Default Country	Select the default country to display when the **Section Profiles Table** dialog opens.
Tables Selection	The materials are organized as tabs along the top of the table. For each material, the countries are listed along with the default catalog to use for each. Select the catalog from the drop-down list for each as needed.
Custom Profile Table	Used to associate a user-defined section table with STAAD.Pro. STAAD.Pro profile table files are Microsoft Office Access® database files (file extension `.mdb`). **Category** Select the profile category (typically material and/or country) with which the profile database is to be associated. **Table** Type the name of the STAAD.Pro profile table file (file extension `.mdb`) to add or click **[...]** to browse for a file. **Add** Adds the selected custom profile table to the program.

M. Prismatic and Tapered Sections

You can define parametric sections for use your model.

M. To assign a prismatic section

To define and assign a prismatic section, use the following procedure.

Prismatic sections such as circle, rectangle, tee, and trapezoidal can be used for concrete members used in design.

Circle prismatic sections can also be used to represent steel rods for analysis.

> **Tip:** If you need to perform design for a steel rod, you can closely approximate a solid steel rod by creating a circular hollow section with a very small inside diameter. Alternately, you can select a steel library section from a catalog that contains solid rods or create your own.

1. Either:

   select the **Database** tool in the **Specifications** group on the **Specifications** ribbon tab

   or

   select the **Properties** page and then click **Section Database** on the **Properties - Whole Structure** dialog

   > **Tip:** Selecting the country from the **Database** tool drop-down list first results in a faster dialog load time.

   The **Section Profile Tables** dialog opens.
2. Either:

select the Properties page and then click **Define** on the **Properties - Whole Structure** dialog

or

on the **Specifications** ribbon tab, select the **Prismatic** tool in the **Beam Profiles** group

The **Property** dialog opens.

3. Select the tab corresponding to the profile shape you want to use.

Notes:

The geometric shapes circle, rectangle, tee, trapezoidal, and general are typically used for concrete members.
Tapered I and tapered tube are typically used for steel members.
The Assign Profile tab is used to assign a profile shape for steel design for later member selection.

4. Type the profile dimensions for the selected shape.
5. (Optional) Select an associated **Material**.

   This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.
6. Either:

To…	Do the following…
add the section to the model for later assignment	click **Add**.
add the section to the model and assign to the current member selection	click **Asign**.

7. Click **Close**.

Related Links

- *G.6.1 Prismatic Properties* (on page 2114)
- *Property dialog* (on page 2794)
- *TR.20.2 Prismatic Property Specification* (on page 2263)
- *TR.20.2.1 Prismatic Tapered Tube Property Specification* (on page 2264)

M. To assign a tapered I section

To define and assign a web-tapered I section, use the following procedure.

Note: *f1* (Depth of section at start node) should always be greater than *f3* (Depth of section at end node). You must provide the member incidences accordingly.

If the tapered section requires a composite slab, then M.To create an I shape user table section (on page 739) instead.

Tapered I-sections have constant flange dimensions and a linearly varying web depth along the length of the member.

Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the `SET SHEAR` command has no effect on the deformation computed for members with these cross sections.

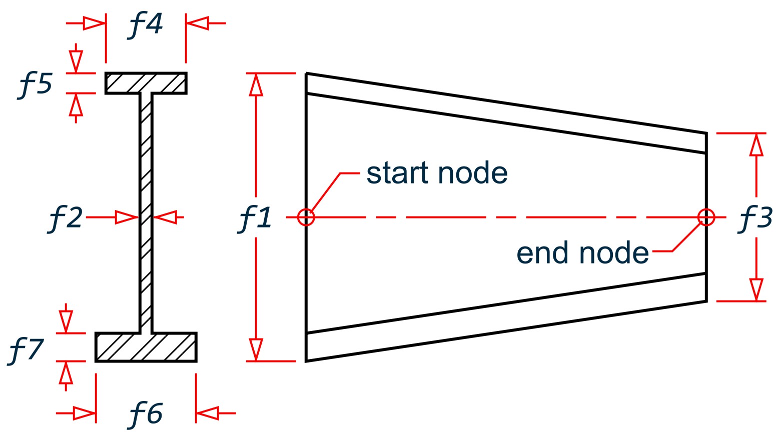
1. On the **Specification** ribbon tab, select the **Tapered** tool in the **Beam Profiles** group.

   The **Tapered I** dialog opens.
2. Select an associated **Material**.

   This feature allows you to use material definitions for steel, stainless steel, concrete, aluminum, or any previously specified, custom material definitions.

   Note: Tapered I sections are typically steel or aluminum members.
3. Specify the profile dimensions for the shape.

4. Either:

To...	Do the following...
add the section to the model for later assignment	click **Add**.
add the section to the model and assign to the current member selection	click **Asign**.

5. Repeat steps 2 through 4 as needed to add more tapered I sections to the model.
6. Click **Close**.

The section and, if selected, associated material, is added to the **Properties - Whole Structure** dialog.

If you did not assign the section to a selection set, you must select the section label in the **Properties - Whole Structure** dialog and assign it using one of the assignment methods.
Related Links

- *G.6.4 Tapered Sections* (on page 2117)
- *TR.20.3 Tapered Member Specification* (on page 2266)
- *Property: Tapered I dialog* (on page 2797)

M. User Table Sections

User-provided tables (UPT) give you a means to specify many different types of parametric sections, including general properties, polygonal sections, and complex sections generated in Section Wizard.

M.To create a wide-flange user table section

To create a user-provided table section with a wide flange profile, including those with composite flanges or additional bottom plates, use the following procedure.

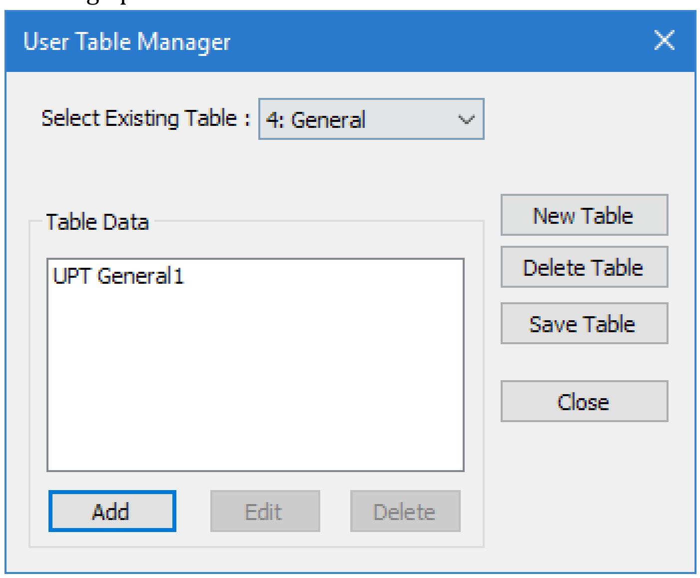
You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.

2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **WIDE FALNGE** and then click **OK**.
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.
   The **Wide Flange** dialog opens.
5. Type a **Section Name**.
6. **Required:** Enter the following wide flange section parameters:

D	Depth of the section
TF	Thickness of top flange (or both flanges when `WF1` is not specified)
WF	Width of the top flange (or both flanges when `WF1` is not specified)
TW	Thickness of web

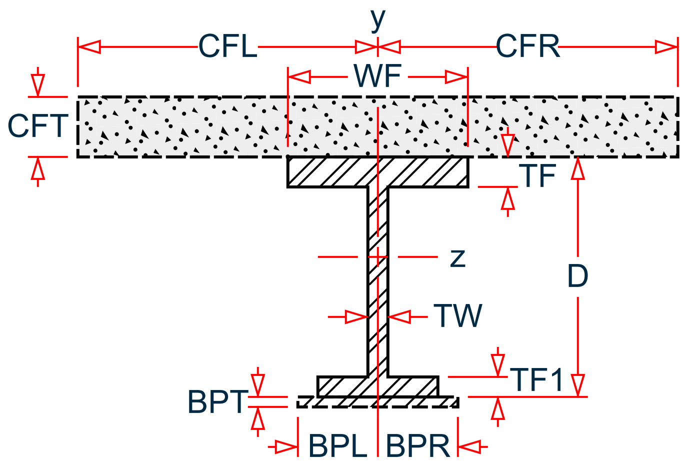


Figure 107: Wide flange section

7. (Optional) If the bottom flange has different dimensions, also specify:

WF1	(Optional) Width of the bottom flange. The width of the top flange will be used if this value is not specified.
TF1	(Optional) Thickness of bottom flange. The thickness of the top flange will be used if this value is not specified.

8. (Optional) To specify a composite top flange:
   a. Check the **Additional Composite Flange** option.
   The Additional Composite Flange Specifications become active.
   b. Specify the following values:

B(left)	Width of the composite slab to the left of the web center line
B(right)	Thickness of the composite slab
Thickness	Width of the composite slab to the right of the web center line
Modular Ratio	Modular ratio of the concrete in the composite slab with respect to the steel section material

9. (Optional) To specify an additional bottom flange plate:
   a. Check the **Additional Bottom Steel Plate** option.
   The Additional Bottom Steel Plate Specifications become active.
   b. Specify the following values:

B(left)	Width of the additional bottom flange plate to the left of the web center line
B(right)	Width of the additional bottom flange plate to the right of the web center line
Thickness	Thickness of the additional bottom flange plate

10. Specify the section properties by either:

Options	Description
To automatically have the program calculate the section properties...	Click **Calculate**

Options	Description
To manually specify the section properties...	Type values for **Cross Section Area (Ax)**, **Inertia about local z (Iz)**, **Inertia about local y (Iy)**, **Torsional Constant (Ix)**, **Shear Area in Y (Ay)**, and **Shear Area in Z (Az)**

11. Click **OK**.
12. (Optional) Repeat steps 4 through 11 to add more wide flange sections.

> **Tip:** The wide flange sections in the same user table can have different options for composite flanges or additional bottom steel plates.

13. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

Refer to EX. US-17 User-Provided Tables (on page 6443) and EX. UK-17 User-Provided Tables (on page 6730) for examples of wide flange user-provided tables.

Related Links

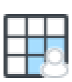
- *Wide Flange dialog* (on page 2801)
- *TR.19.1 Wide Flange* (on page 2244)
- *User Provided Table dialog* (on page 2798)
- *User Table Manager dialog* (on page 2798)

M.To create a channel user table section

To create a user-provided table section with a channel profile, use the following procedure.

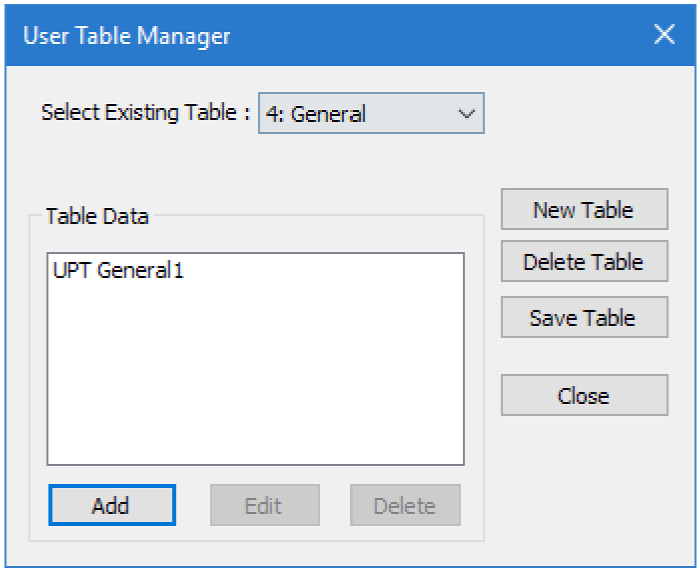
You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.



2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **CHANNEL** and then click **OK**.
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.
   The **Channel** dialog opens.
5. Type a **Section Name**.
6. **Required:** Enter the following channel section parameters:

D	Depth of the section
TW	Thickness of web
WF	Width of flange
TF	Thickness of flange

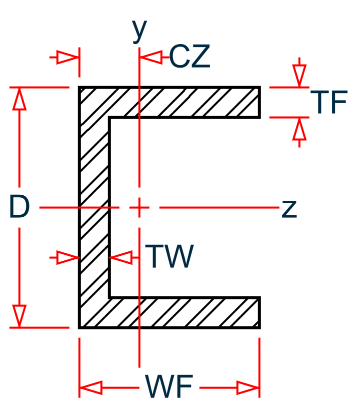


Figure 108: Channel section

7. Specify the section properties by either:

Options	Description
To automatically have the program calculate the section properties...	Click **Calculate**
To manually specify the section properties...	Type values for **Cross Section Area (Ax)**, **Inertia about local z (Iz)**, **Inertia about local y (Iy)**, **Torsional Constant (Ix)**, **C.G. Along local Z (Cz)**, **Shear Area in Y (Ay)**, and **Shear Area in Z (Az)**

8. Click **OK**.
9. (Optional) Repeat steps 4 through 8 to add more channel sections.
10. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

Related Links

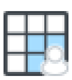
- *TR.19.2 Channel* (on page 2246)
- *Channel dialog* (on page 2802)

M.To create an angle user table section

To create a user-provided table section with a single angle profile, use the following procedure.

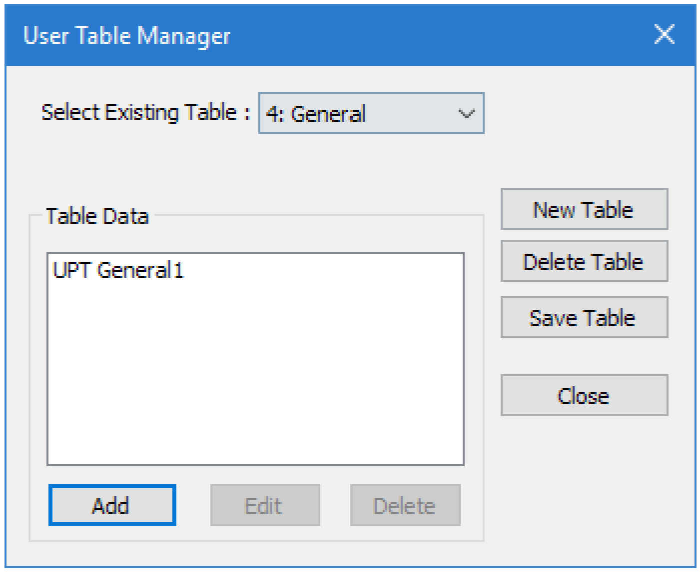
You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.



2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **ANGLE** and then click **OK**.
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.
   The **Angle** dialog opens.
5. Type a **Section Name**.
6. **Required:** Enter the following angle section parameters:

D	Depth of angle (i.e., length of the leg along the local y-axis)
WF	Width of angle (i.e., length of the leg along the local z-axis)
TF	Thickness of angle leg

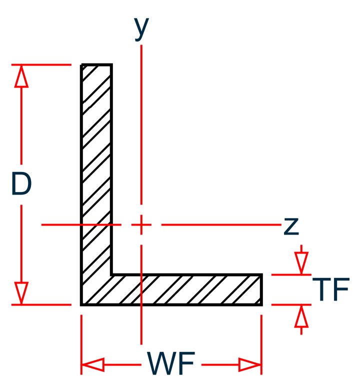


Figure 109: Angle section

7. Specify the section properties by either:

Options	Description
To automatically have the program calculate the section properties...	Click **Calculate**

Options	Description
To manually specify the section properties...	Type values for **Rad. of Gyration about Principal Axis (R)**, **Shear Area in Y (Ay)**, and **Shear Area in Z (Az)**

8. Click **OK**.
9. (Optional) Repeat steps 4 through 8 to add more angle sections.
10. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

> Refer to EX. US-17 User-Provided Tables (on page 6443) and EX. UK-17 User-Provided Tables (on page 6730) for examples of angle user-provided tables.

Related Links

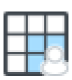
- *TR.19.3 Angle* (on page 2247)
- *Angle dialog* (on page 2803)

M.To create a double angle user table section

To create a user-provided table section with a double angle profile, use the following procedure.

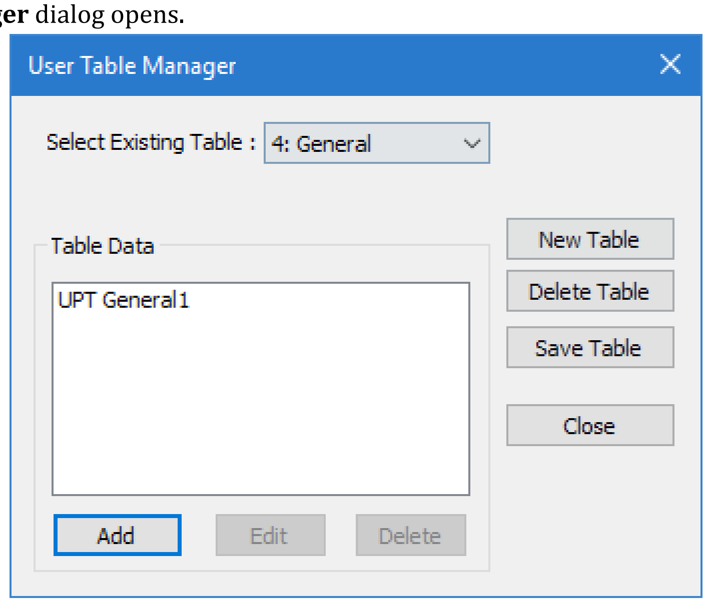
You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.



If no User Defined Table exists, you will be prompted to create one.

The **User Table Manager** dialog opens.



2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **DOUBLE ANGLE** and then click **OK**.
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.
   The **Double Angle** dialog opens.
5. Type a **Section Name**.
6. **Required:** Enter the following angle section parameters:

D	Depth of angle (i.e., length of the leg along the local y-axis)
WF	Width of angle (i.e., length of the leg along the local z-axis)
TF	Thickness of angle leg
SP	Space between angles.

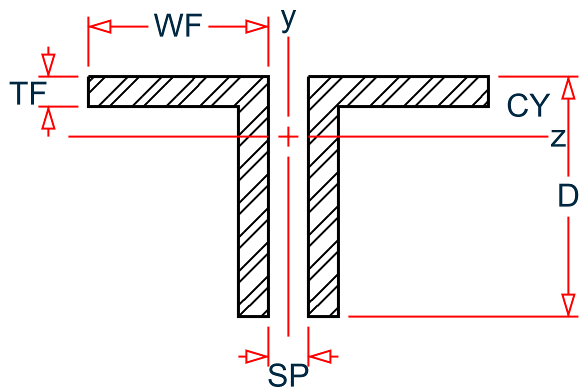


Figure 110: Double angle section

7. Specify the section properties by either:

Options	**Description**
To automatically have the program calculate the section properties...	Click **Calculate**
To manually specify the section properties...	Type values for **Inertia about local z (Iz)**, **Inertia about local y (Iy)**, **Torsional Constant (Ix)**, **C.G. Along local Y (Cy)**, **Shear Area in Y (Ay)**, and **Shear Area in Z (Az)**

8. Specify the Single Angle (rvv) value.
   This is the radius of gyration about the minor principal axis for one of the angles taken as a single angle.
9. Click **OK**.
10. (Optional) Repeat steps 4 through 9 to add more angle sections.
11. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

> Refer to EX. US-17 User-Provided Tables (on page 6443) and EX. UK-17 User-Provided Tables (on page 6730) for examples of angle user-provided tables.

Related Links

- *Double Angle dialog* (on page 2804)
- *TR.19.4 Double Angle* (on page 2248)

M.To create an I shape user table section

To create a user-provided table section with an I section profile, including those with composite flanges or tapered web depth, use the following procedure.

You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

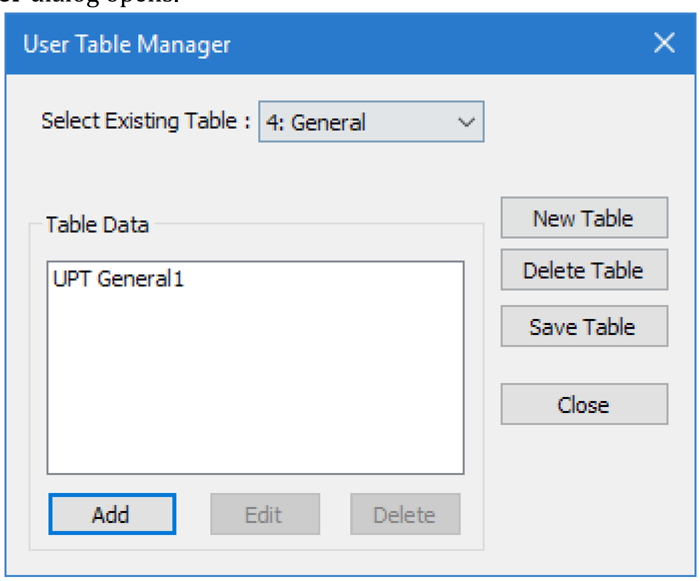
This user-provided section allows you to specify a tapered I-Section with an optional composite top flange. For I sections with a bottom steel plates, you must M.To create a wide-flange user table section (on page 731). Alternatively, you may M. To assign a tapered I section (on page 729) when a non-composite section is required.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.

2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **ISECTION** and then click **OK.**
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add.**
   The **ISection** dialog opens.
5. Type a **Section Name.**
6. **Required:** Enter the following wide flange section parameters:

TWW	Thickness of web

TFF	Thickness of top flange
BFF	Width of top flange
TFF1	Thickness of bottom flange
BFF1	Width of bottom flange
Depth at Start Node	The overall depth of the section at the starting node.
Depth at End Node	The overall depth of the section at the ending node. **Note:** This must be less than or equal to the depth at the start node (i.e., the section depth must taper *down* from start to end nodes or remain constant depth).

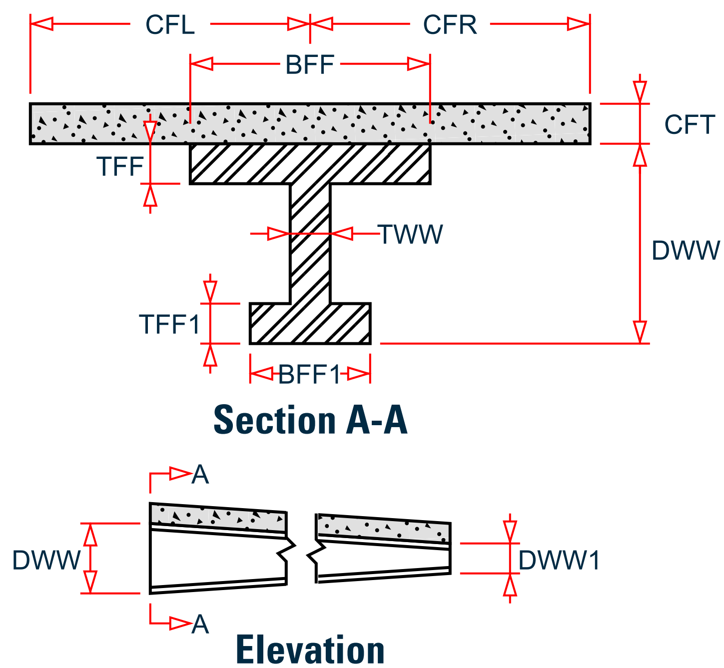


Figure 111: I section and tapered web

7. (Optional) To specify a composite top flange:
   a. Check the **Additional Composite Flange** option.
   The Additional Composite Flange Specifications become active.
   b. Specify the following values:

B(left)	Width of the composite slab to the left of the web center line
B(right)	Thickness of the composite slab
Thickness	Width of the composite slab to the right of the web center line
Modular Ratio	Modular ratio of the concrete in the composite slab with respect to the steel section material

8. (Optional) Specify the section properties **Torsional Constant (Ix)**, **Shear Area in Y (Ay)**, and **Shear Area in Z (Az)**.

If positive values are entered, then those values are used for each of these section properties. If a zero is entered, then the program will calculate these values. If a negative value is used, then the absolute value of each is applied as a factor for that section property.

The remaining section properties are calculated at finite points along the section for use in the analysis.

9. Click **OK**.
10. (Optional) Repeat steps 4 through 9 to add more I sections.

    Tip: The I sections in the same user table can have different options for composite flanges or section depths.

11. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

Related Links

- *TR.19.9 I Section* (on page 2252)
- *I Section dialog* (on page 2808)

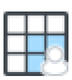
M.To create an prismatic user table section

To create a generic user-provided table section providing only cross section properties, use the following procedure.

You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

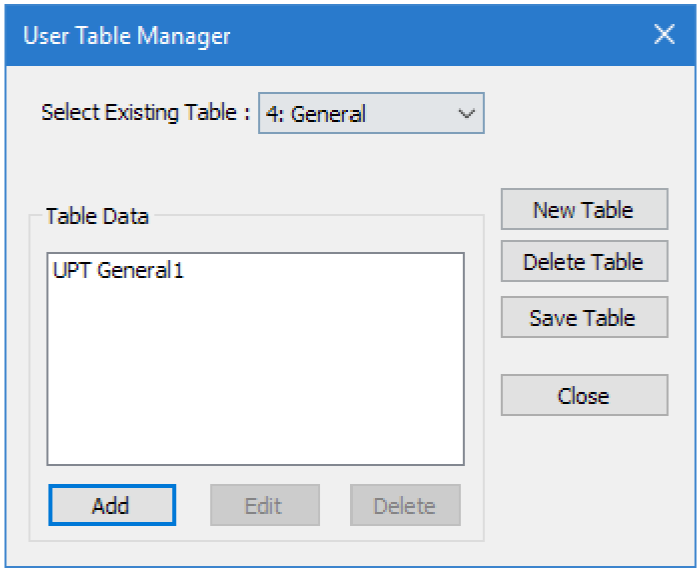
This section type is useful when you only have cross-section property values or if the necessary section shape does not classify as any of the other user-provided section types.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.



2. Click **New Table**.
   The **New User Table** dialog opens.
3. the **Select Section Type** list, select **PRISMATIC** and then click **OK**.
   The dialog closes and the **Table Data** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.
   The **Prismatic** dialog opens.
5. Type a **Section Name**.
6. **Required:** Enter the following angle section parameters:

YD	Depth of the section in the direction of the local y-axis.
ZD	Depth of the section in the direction of the local z-axis

7. Specify the section properties:

Cros Section Area (Ax)	Cross-section area
Inertia about local z (Iz)	Moment of inertia about the local z-axis
Inertia about local y (Iy)	Moment of inertia about the local y-axis
Torstional Constant (Ix)	Torsional constant
Shear Area in Y (Ay)	Shear area for shear parallel to local y-axis.
Shear Area in Z (Az)	Shear area for shear parallel to local z-axis.

8. Click **OK**.
9. (Optional) Repeat steps 4 through 8 to add more prismatic sections.
10. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

> Refer to EX. US-17 User-Provided Tables (on page 6443) and EX. UK-17 User-Provided Tables (on page 6730) for examples of angle user-provided tables.

Related Links

- *TR.19.10 Prismatic* (on page 2253)
- *Prismatic dialog* (on page 2809)

M. To create a general section

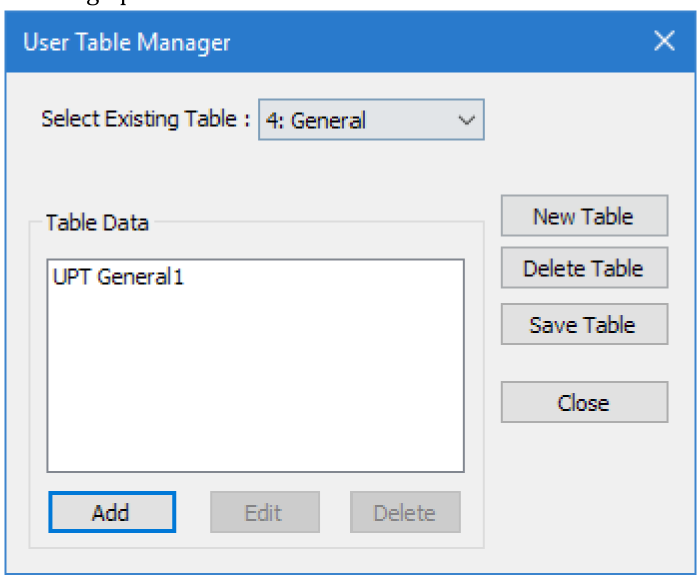
To create a user provided table section with a general profile, use the following steps.

You may want to change the input dimensions prior to creating user provided table sections. Select **Tools** > **Set Current Input Unit** to do so.

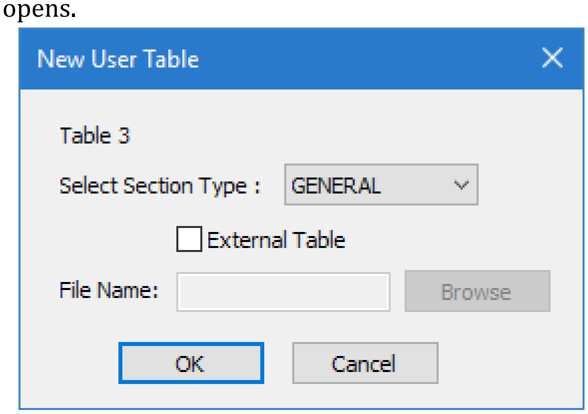
1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

   If no User Defined Table exists, you will be prompted to create one.
   The **User Table Manager** dialog opens.

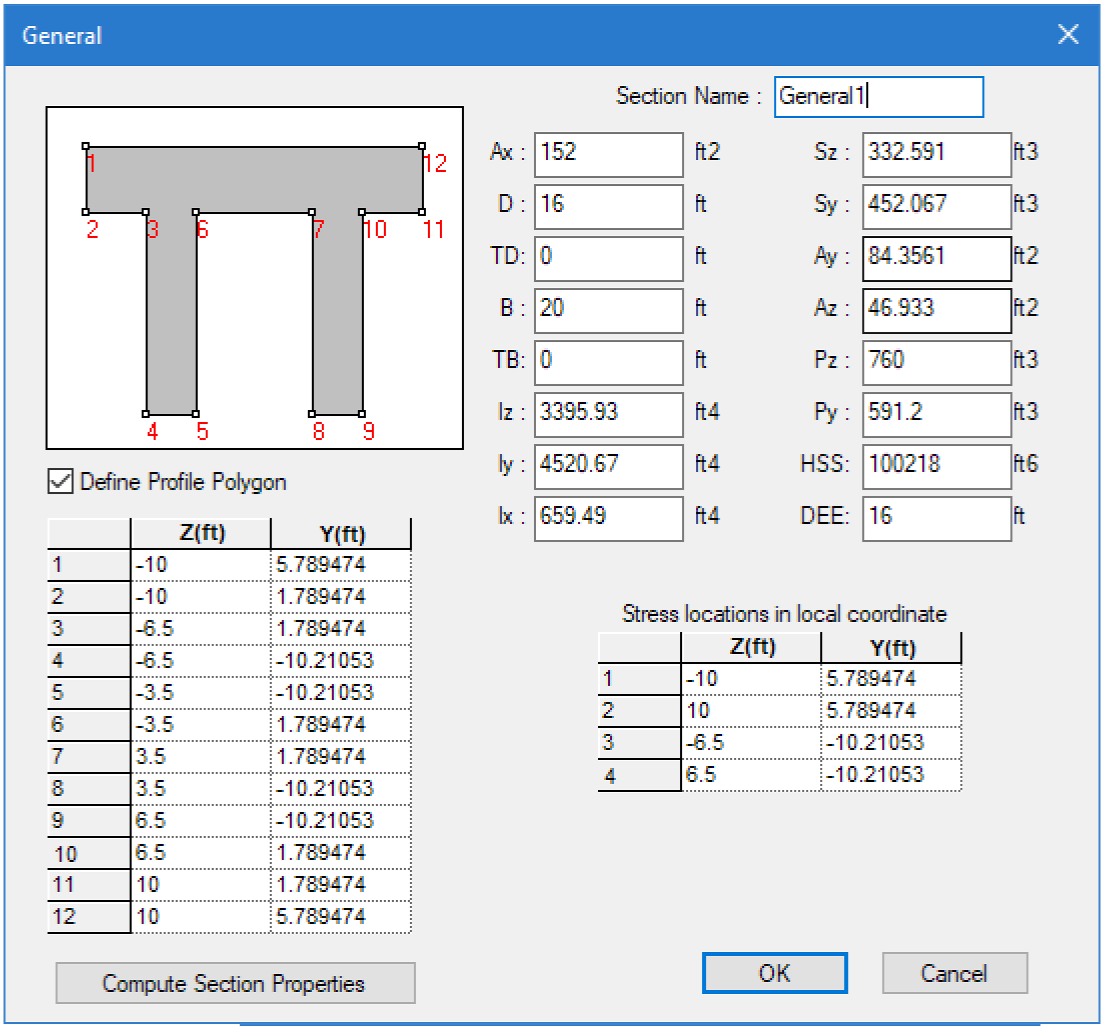
   

2. Click **New Table**.
   The **New User Table** dialog opens.

3. In the **Select Section Type** list, select **GENERAL** and then click **OK**.
   The dialog closes and the **Select Existing Table** list in the **User Table Manager** dialog now has at least one entry.
4. Click **Add**.

   The **General** dialog opens.



5. Type a **Section Name**.
6. Either:

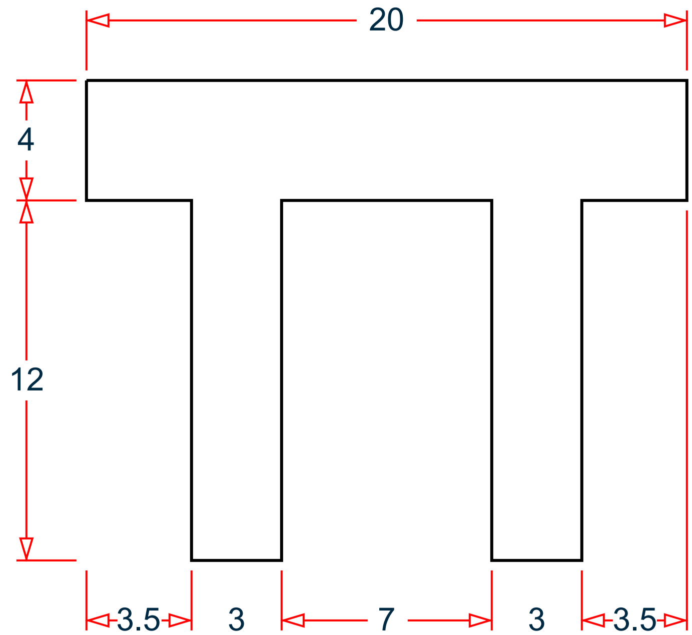
   Type the section properties if these are available in the corresponding fields

   or

   Set the **Define Profile Polygon** and enter in Z,Y coordinates of the section apexes.

Tip: These values may be copied and pasted from a spreadsheet.

The following is an example cross section, with corresponding Z,Y coordinates which can be copy/pasted directly into the table.



Coordinate Number	Z (units)	Y (units)
1	0	0
2	0	-4
3	3.5	-4
4	3.5	-16
5	6.5	-16
6	6.5	-4
7	13.5	-4
8	13.5	-16
9	16.5	-16
10	16.5	-4
11	20	-4
12	20	0

7. (Optional) If you entered in section corner coordinates, click **Compute Section Properties**

   The section properties are calculated and the fields are updated. The coordinates are also updated from an arbitrary origin to the an origin at the center of gravity of the cross section.

Tip: You can manually overwrite any of these values if necessary.

8. (Optional) If you want to have section stress calculated, enter Y,Z coordinates in the **Stress locations in local coordinates**

   Tip: The coordinate locations for stresses should be based on the center of gravity of the section, rather than an arbitrary data point. These may be copied/pasted from the re-centered apex coordinates table.

9. Click **OK**.
   The section is added to the list in the **Create User Property Table** dialog.
10. (Optional) Repeat steps 4 through 9 to add more General sections.
11. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

Example

The example given in 6 above creates the following lines in a STAAD input file:

```
…
TABLE 1
UNIT INCHES KIP
GENERAL
General1
152 16 0 20 0 3395.93 4520.67 638.828 332.591 452.067 83.3308 -
46.3944 760 625.529 95637.5 16
PROFILE_POINTS
-10 5.78947 -10 1.78947 -6.5 1.78947 -6.5 -10.2105 -3.5 -10.2105 -3.5 1.78947
3.5 1.78947 -
 3.5 -10.2105 6.5 -10.2105 6.5 1.78947 10 1.78947 10 5.78947
STRESS_LOCATIONS
-10 5.78947 10 5.78947 -6.5 -10.2105 6.5 -10.2105
END
…
```

Related Links

- *G.6.3 User-Provided Steel Table* (on page 2117)
- *TR.19 User Steel Table Specification* (on page 2241)
- *TR.20.4 Property Specification from User Provided Table* (on page 2267)
- *User Table Manager dialog* (on page 2798)
- *User Provided Table dialog* (on page 2798)

M. To use a general shape created in Section Wizard

To use a general shape created in Section Wizard as a user provided shape table in STAAD.Pro, doe the following.

You must create a profile outline in Section Wizard and export it as a General Shape (file extension `.upt`) for use in STAAD.Pro. Refer to TR.20.10 Member Property Reduction Factors (on page 2282) for details.

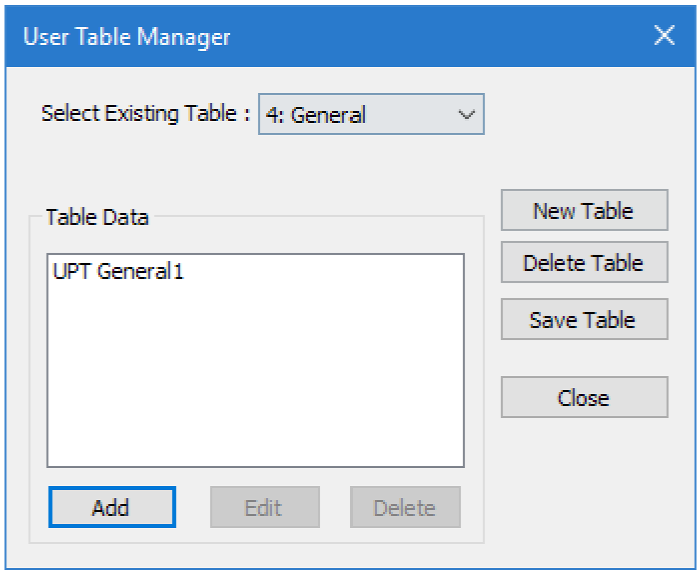
Note: This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.

1. On the **Specification** ribbon tab, select the **User Table** > **User Table Manager** tool in the **Beam Profiles** group.

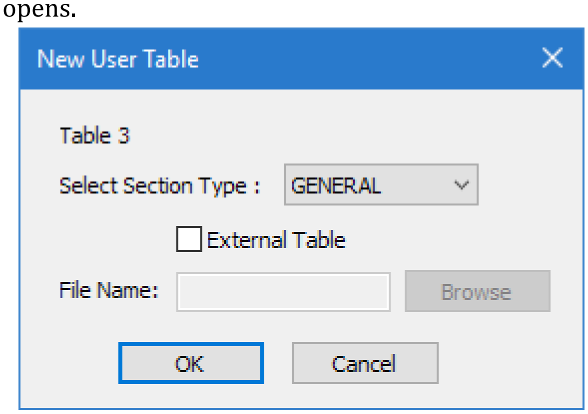
   

   If no User Defined Table exists, you will be prompted to create one.

   The **User Table Manager** dialog opens.

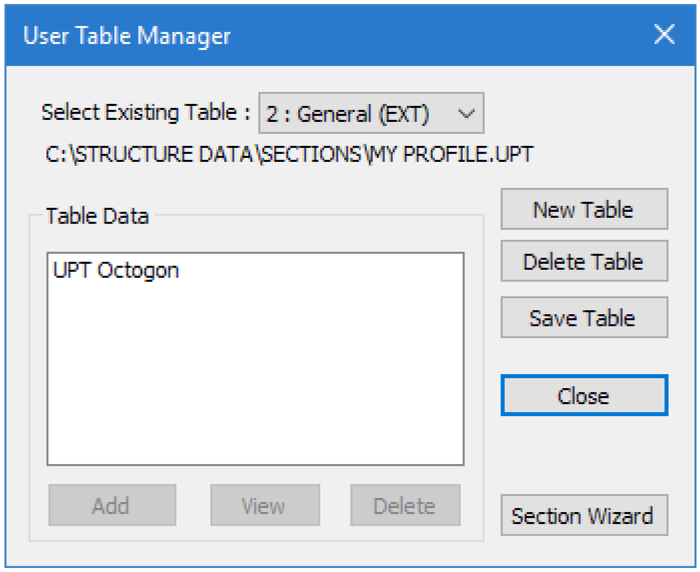
   

2. Click **New Table**.
   The **New User Table** dialog opens.

3. Specify the new external table:
   a. In the **Select Section Type** list, select **GENERAL**.
   b. Check the **External Table** option.
   c. Click **Browse**.
   d. Navigate to and select the `.upt` file exported from Section Wizard and the click **OK**.

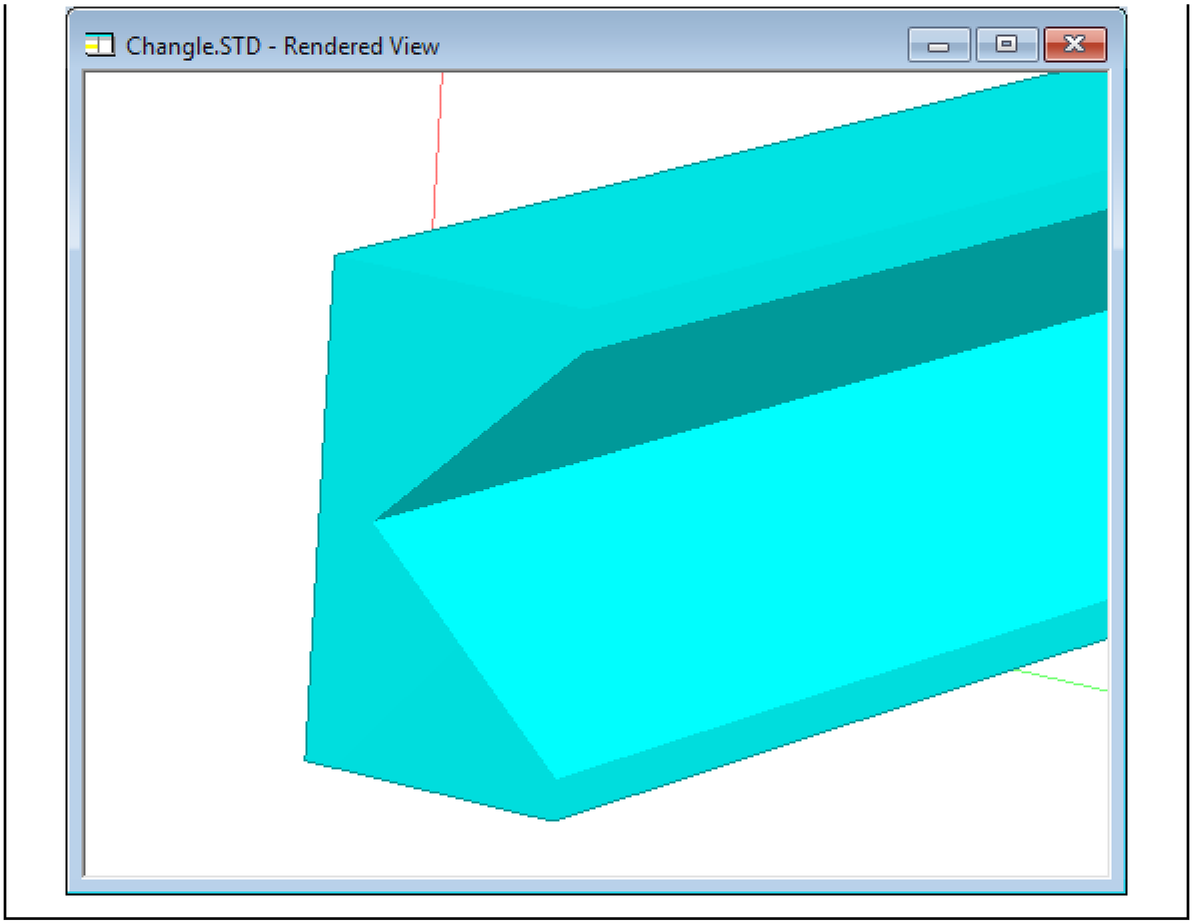
   The dialog closes and the **Table Data** in the **User Table Manager** dialog shows the name of the profile read from the `.upt` file.



4. (Optional) To view the calculated section properties, select the profile entry in the **User Table Manager** dialog and then click **View**.
   The **General** dialog opens.
5. Click **Close**.

The section can now be added for use in the Properties - Whole Strucutre dialog (on page 2793) by selecting it in the **User Property Table** dialog.

> **Tip:** Changing the structure diagram to display 3D rendering types for Full Sections or Sections Outline will display the shape profile. Similarly, 3D renderings of the structure will also display the shape profile.



M. Section Wizard Help

M. To start Section Wizard

Section Wizard consists a set of utility programs used to generate custom sections for use in STAAD.Pro.

1. Open a model in the STAAD.Pro Analytical Modeling workflow.
2. On the **Specifications** ribbon tab, select **Section Wizard** tool in the **Tools** group.

   The **Section Wizard** program opens.

Alternatively, you can locate the file `\SectionWizard\Section.exe` in the folder where STAAD.Pro was installed on your computer.

M. Section Builder

The Section Builder module is used to calculate the properties of sections built up from an arrangement of cross sections taken from a number of standard tables. This module is composed of two windows. The main **Section Builder** window, which displays the composite section along with a number of tools to manipulate the sections and the **Section Element** dialog box which is used to select the sections to be used and method of locating them onto the current overall section.

M. Application Window Layout

The main Section Builder window displays the current composite section. All of the primary commands are activated through the menus along the top of the window. There is also a status bar along the bottom of the window which displays the overall dimensions of the composite section and the location of the cursor when it lies over the main workspace. Additionally a dimension value will be displayed in the status bar by clicking and dragging the mouse across the window.

Sections created in the Section Dialog box can be located onto a composite section relative to the nodes of the 'current section.' A section is made into the current section simply by clicking on it so that it is shown in yellow. Note how the nodes of the current section are identified as potential locating points for the addition of new sections.

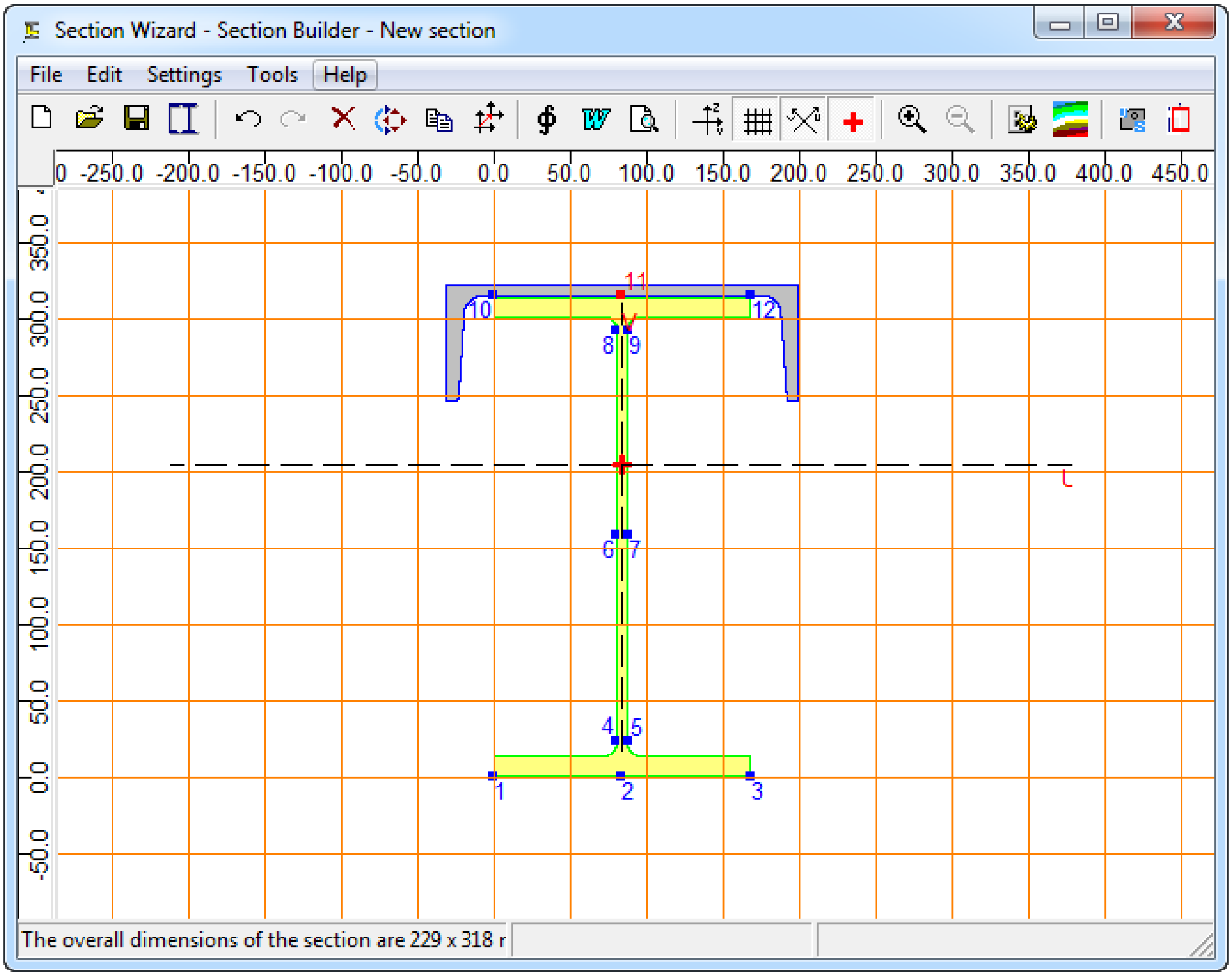


Figure 112: Section Builder module window

Window controls

Menu	
Toolbar	A series of commonly used tools are arranged at the top of the module window.

M. Toolbar
Contains several of the most commonly used tools found in the Section Builder module.

Tool	Description	Shortcut
New		

Tool	Description	Shortcut
Open	Opens the Open Data File dialog, which is used to select either Section Builder (file extension `.SEC`) files or Free Sketch (file extension `.CNS`) files. Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the **Preview** option is selected, then a graphical preview of the section in a selected file is displayed prior to opening the file.	
Save	Opens the Save dialog box if the file has not previously been saved and allows the Equivalent Section, section to be saved. If the file has previously been saved, then that file is updated with the new data. Files are saved in the Section Builder *.SEC format.	
Create Standard Section	Opens the M. Section dialog (on page 768) which is used to quickly generate a composite section based on one of seven options. This is the same option that is available in the Section Builder module.	
Undo	Negates the last drawing operation.	
Redo	Negates the last undo operation.	
Delete		
Shift/Rotate		
Copy		
Shift Coordinate Center		
Calculate Basic Geometry		

Tool	Description	Shortcut
Report	Once an equivalent section has been 'found', the data displayed in the table and a dimensioned drawing will be sent to a report formatted to the Report Type option defined in the Misc sheet of the Program Preferences dialog. See Preferences below.	
Preview		
Show Coordinate Axis	Toggles the display of the coordinate axis in the Section Builder view.	
Show Grid		
Show Principle Axis		
Show Current Center of Gravity		
Zoom In		
Zoom Out		
Preferences	Opens the Preferences dialog.	
Display Normal Stress Field		
Launch Free Sketch module	Launches the M. Free Sketch (on page 769) of Section Wizard and creates section based on the section that was 'found'.	
Launch Equivalent Section module		

M. Menus
Menus in the Section Builder module window.
M. File menu
Contains items for creating, opening, and closing section data files, printing, and exiting the program.

Table 25: File menu items

Menu item	Description	Same effect as selecting…
New	Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to comencing with the new design.	CTRL+N
Open	Opens the Open Data file dialog, which is used to select an existing Section Data file (file extenstion `.SEC`). If the **Show Drawing** option is checked, then a preview of the section can be displayed prior to opening the file.	CTRL+O
Create Standard Section	Opens the M. Section dialog (on page 768) which is used to quickly generate a composite section based on one of seven options.	
Save	Saves any changes made to the current data file. **Note:** If the current file has not been previously saved, this has the same effect as selecting **Save As...**	CTRL+S
Save As...	Opens the Save data file dialog which is used to specify a file name and location for the section data file.	
Report	Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.	
Preview	Displays the section as will be drawn in the report. Toggles the display of the Section Element dialog.	
Send...	Opens an e-mail (using your default e-mail client) and attaches the current data file.	

Menu item	Description	Same effect as selecting…
Export to STAAD.Pro >	Select either to export the section as a General or Prismatic section. A Save If an existing User Provide Table file is selected, the current section will be appended to the end of this table. Refer to the STAAD.Pro Technical Reference for additional information on using section types.	
Sections > <list>	A list of the most recent four section data files. Select one to open that data file.	
Exit	Closes the program. If any changes were made since the last time the data file was saved, a message dialog opens confirming you wish to save those changes.	ALT+F4

M. Edit menu

Menu item	Description	Same effect as selecting…
Undo	Negates the last modification performed in the window (for example, shift/rotate a section). You can undo a series of modifications by repeatedly selecting Undo.	
Redo	Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.	
Delete	Removes the selected section element from the composite section.	Delete key

Menu item	Description	Same effect as selecting…
Shift, Rotate Element	Opens the Element Shift/Rotation dialog which is used to displace the currently selected section by a relative dimension or rotation about it's basic node. See section Orientation of Elements for the basic node of each section type used by Section Builder.	
Shift Coordinate Center	Opens the M. Shift coordinate center dialog (on page 756) which is used to relocate the origin of the geometric axes relative to the composite section.	
Copy Element	Opens the Copy element dialog which is used to create one or more copies by specified offset dimensions in the Y and Z axes.	

M. Shift coordinate center dialog
Used to relocate the origin of the geometric axes relative to the composite section.

Dialog controls

1. Set it to the current center of gravity
2. Set it to a given node on the current section.
3. Set it to a given node on the current section.

M. Settings menu

Table 26: Settings menu items

Menu item	Description	Same effect as selecting…
Undo		
Redo		
Delete		
Shift, Rotate Element		
Shift Coordinate Center		
Copy Element		

M. Tools menu

Table 27: Tools menu items

Menu item	Description	Same effect as selecting…
Calculate	Calculates the cross-sectional properties of the composite section and the opens the Basic Geometry dialog to display the values.	
Stress Contour	Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the composite section.	
Free Sketch	Opens the M. Free Sketch (on page 769).	
Equivalent Section	Opens the M. Equivalent Section module (on page 763).	
Windows Calculator	Opens the Windows system Calculator window.	
Formula Calculator	Opens the M. Formula Calculation window (on page 790) which is used to evaluate a formula, from the numerical result can then be copied back into the Section Builder program.	
Unit Converter	Opens the Unit Converter Window which is used to as a value converter between different preset units of measurement. Select the tab pertaining to the type of measurement and then enter the known value in the appropriate field. Upon pressing **<Enter>**, all the other fields are updated to equivalent values.	

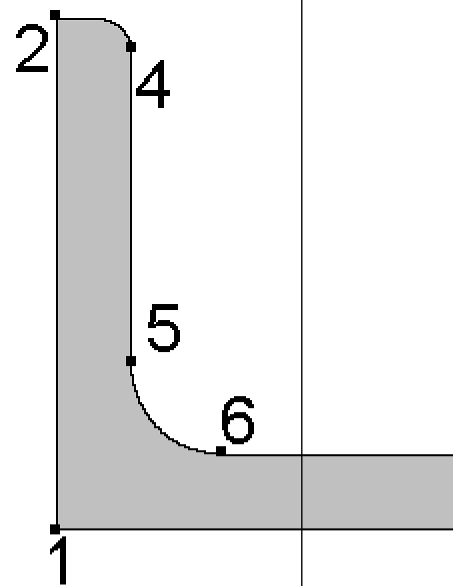
M. Help menu
The Help menu contains items for using online help.

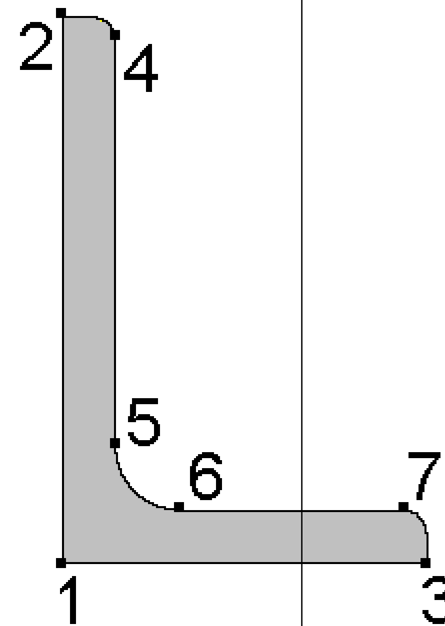
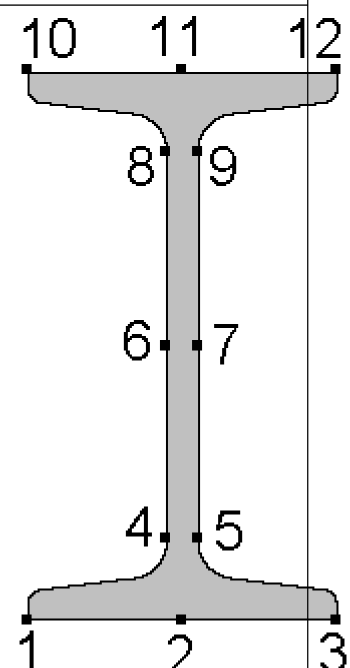
Table 28: Help menu items

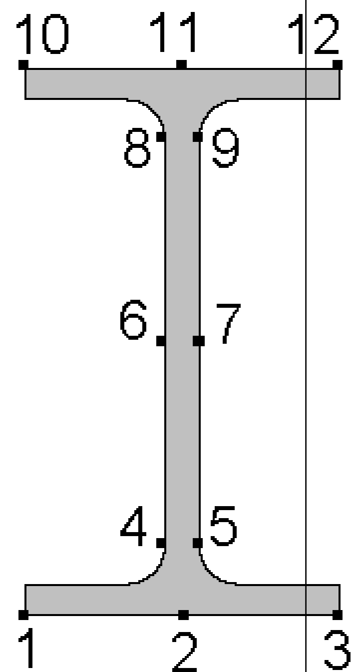
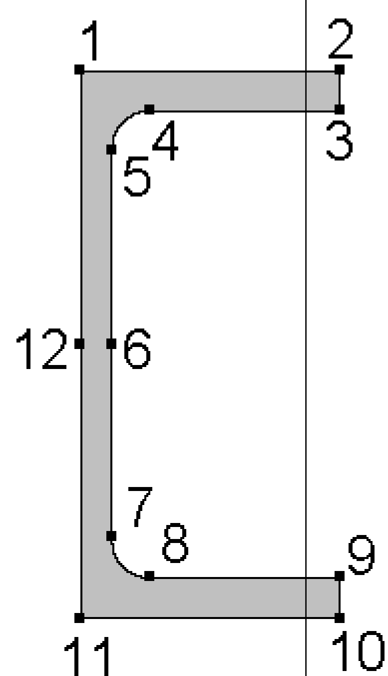
Menu item	Description	Same effect as selecting…
Help Topics	Opens the Section Builder help window.	F1
About	Opens the About Section Builder window, which displays version and copyright information.	

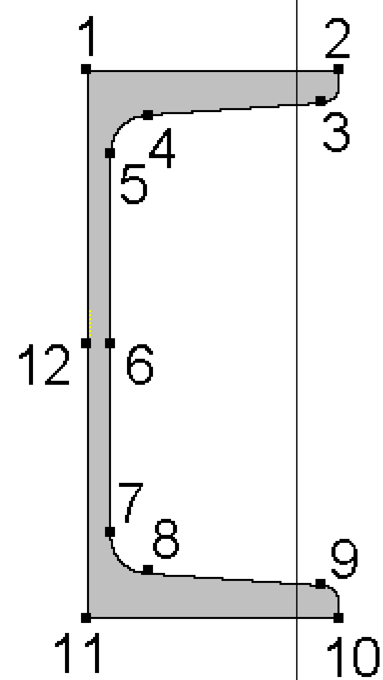
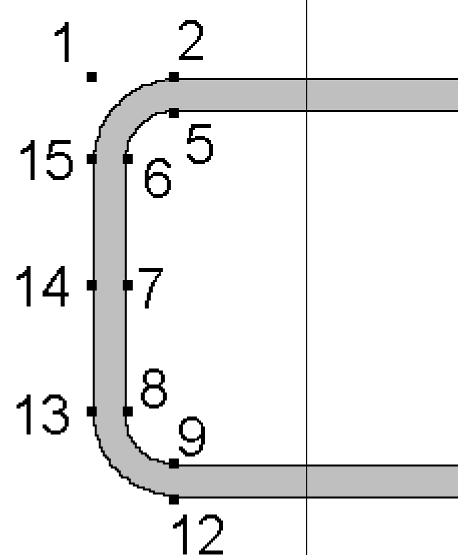
M. Orientation of Elements

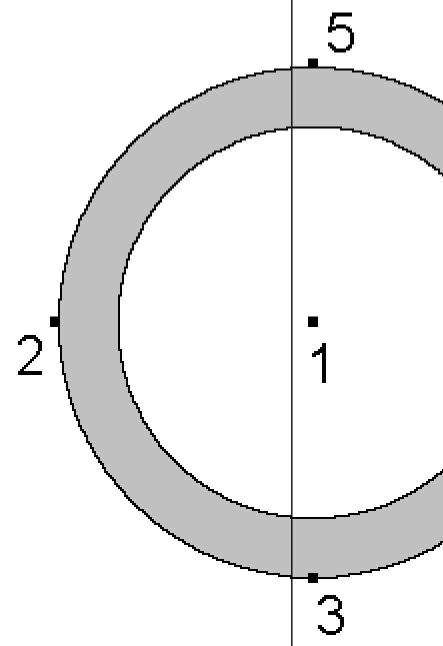
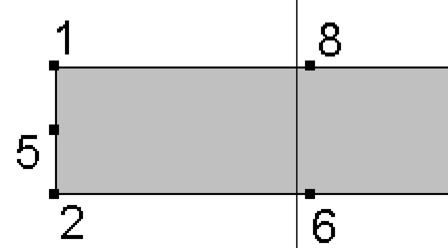
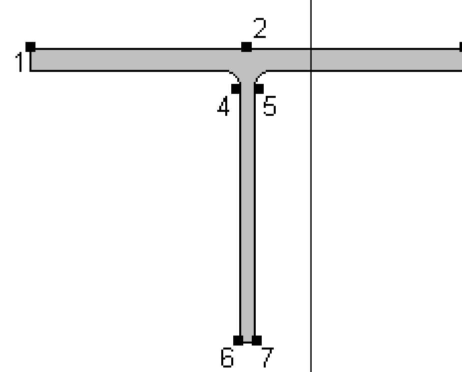
Table 29: Standard element shapes and their default orientation and base nodes

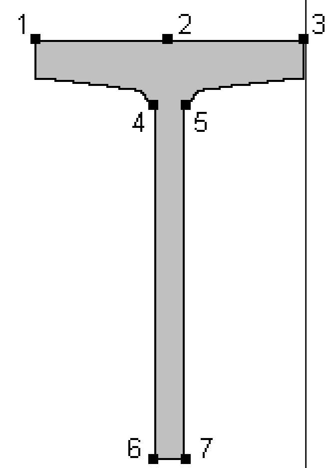
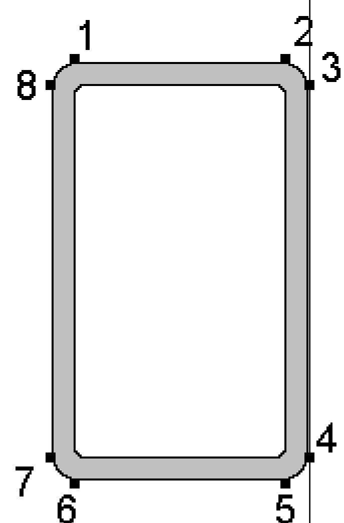
Profile Type	Section	Base Node #	Nodes
Angles	Equal Leg Angle	1	

Profile Type	Section	Base Node #	Nodes
	Unequal Leg Angle	1	
I Sections	I Sections with Tapered Flanges	10	

Profile Type	Section	Base Node #	Nodes
	I Sections with Parallel Flanges	10	
Channels	Parallel Flanged Channels	1	

Profile Type	Section	Base Node #	Nodes
	Tapered Flange Channels	1	
	Cold Rolled Channels	1	

Profile Type	Section	Base Node #	Nodes
CHS	Circular Hollow Sections	1	
Plate	Plate Sections	1	
Tee Section	T Sections with Parallel Flanges		

Profile Type	Section	Base Node #	Nodes
	T Sections with Tapered Flanges	1	
RHS	Rectangular Hollow Section	1	

M. Equivalent Section module

This module is used to find a best fit solution from the table of values of area, inertia, and elastic moduli. The data can either be entered directly into the table, or generated from the Section Builder or Free Sketch modules, or from the Standard Section toolbar command.

Section Types

The Equivalent Section module then finds the best fit on one of the following section types:

1. I Section, with independent dimensions for the top and bottom flanges
2. Rectangular Hollow Section, with separate dimensions for the side walls and the top and bottom flanges
3. I Section, with the same sizes for the top and bottom flanges
4. Rectangular Hollow Section, with the same thickness for the flanges and the side walls

5. Channel Section

Each of the values can be weighted for importance by setting a bias for that value. The larger the bias, the more the program will attempt to match that value rather than any of the others.

M. Application Window Layout

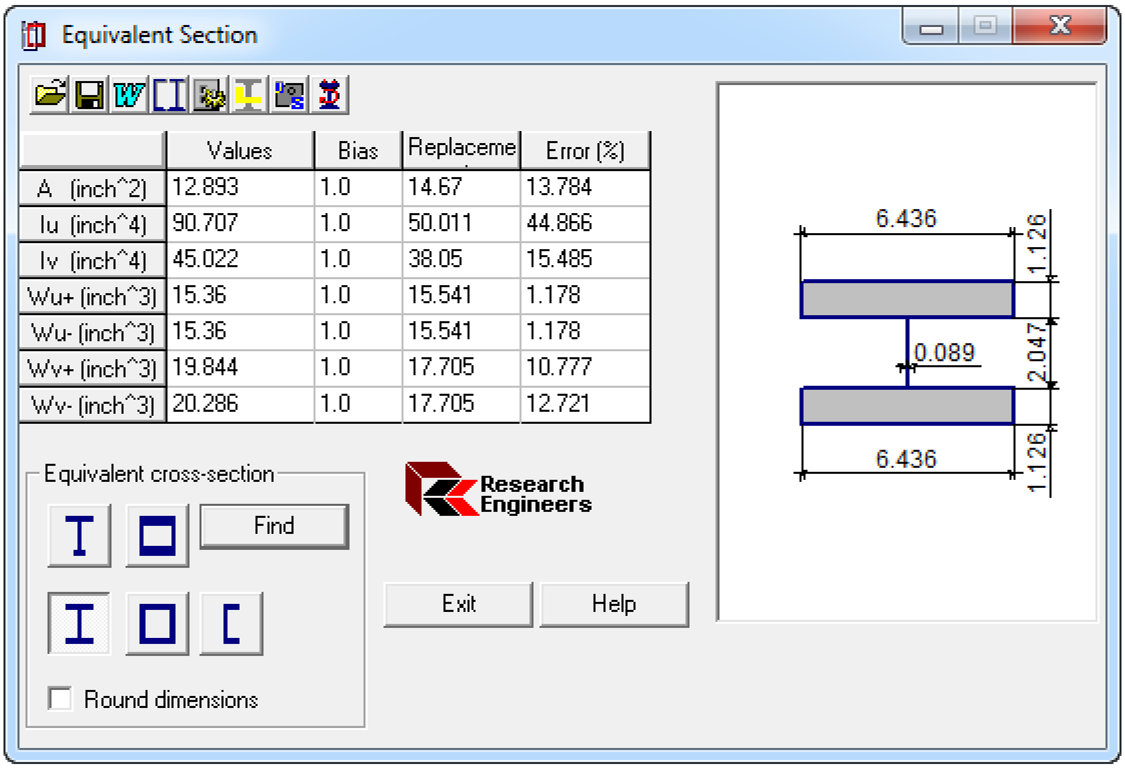


Figure 113: Equivalent Section module window

Window controls

Toolbar	A series of commonly used tools are arranged at the top of the module window.
Properties table	A list of the reference property values and bias values are entered here to provide data for the cross-section search. Once the Find button is clicked, a set of associate Replacement values and the percent Error will also be displayed, along with a graphical representation of the found equivalent section dimensions.
Equivalent cross-setion	A group of controls are arranged here depicting general cross-section types. The currently selected type is shown as depressed.
Find	Click to perform a search for an equivalent cross-section based on the specified section Values, weighted per the specified Bias values.
Round dimensions	Select this option to have the search use rounded dimensional values. If a search has already been performed, then the current dimension results are rounded.
Exit	Closes the module. Any unsaved data will be lost.
Help	Opens the Section Wizard help window to display the relevant help topic.

Views panels A graphical representation of a section as well as dimensions of the recommended section from a Find action are displayed in the right-hand side of the window.

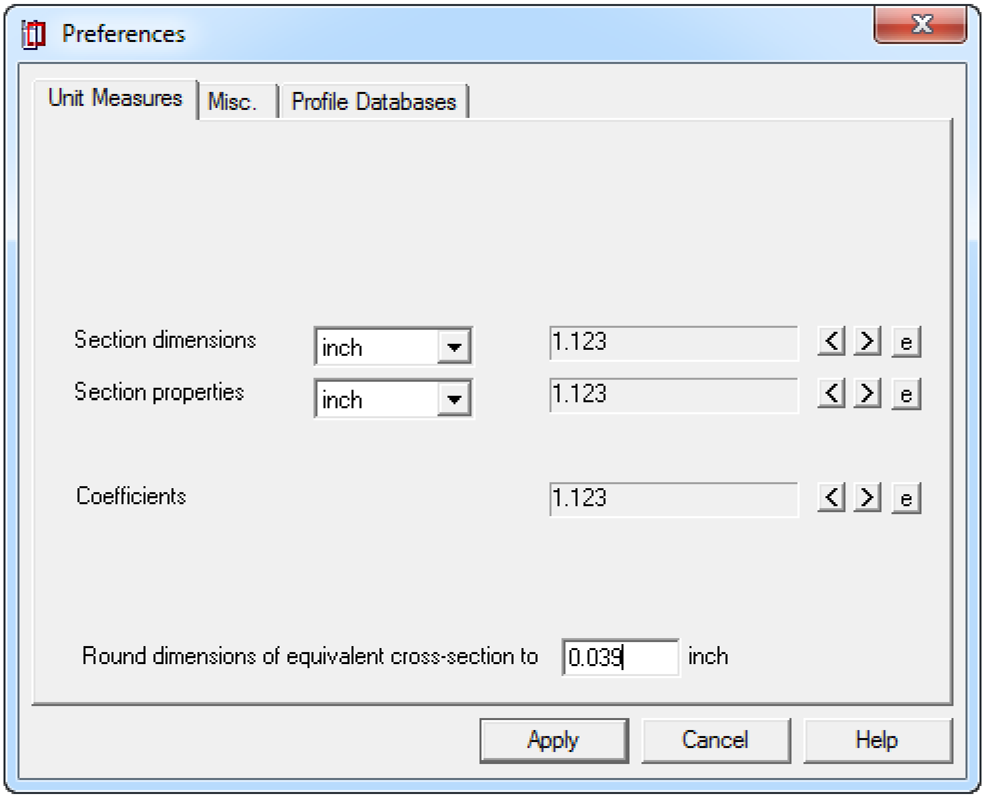
M. Toolbar
A list of tools found in Equivalent Section.

Tool	Description	Shortcut
Open	Opens the Open Data File dialog, which is used to select either Section Builder (file extension `.SEC`) files or Free Sketch (file extension `.CNS`) files. Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the **Preview** option is selected, then a graphical preview of the section in a selected file is displayed prior to opening the file.	
Save	Opens the Save dialog box if the file has not previously been saved and allows the Equivalent Section, section to be saved. If the file has previously been saved, then that file is updated with the new data. Files are saved in the Section Builder *.SEC format.	
Report	Once an equivalent section has been 'found', the data displayed in the table and a dimensioned drawing will be sent to a report formatted to the Report Type option defined in the Misc sheet of the Program Preferences dialog. See Preferences below.	
Create Standard Section	Opens the M. Section dialog (on page 768) which is used to quickly generate a composite section based on one of seven options. This is the same option that is available in the Section Builder module.	
Preferences	Opens the Preferences dialog.	

Tool	Description	Shortcut
Launch Section Builder module	Launches the Section Builder module of Section Wizard and creates section based on the section that was 'found'	
Launch Free Sketch module	Launches the M. Free Sketch (on page 769) of Section Wizard and creates section based on the section that was 'found'.	
Export STAAD User Database	This allows the current section data to be used to create STAAD External User tables. There are two types of database that can be created , either a Prismatic Section or a General Section. Selecting either command will ask for a name of file to be used. If the filename chosen exists already, then the current section will be added to the bottom of the file.	

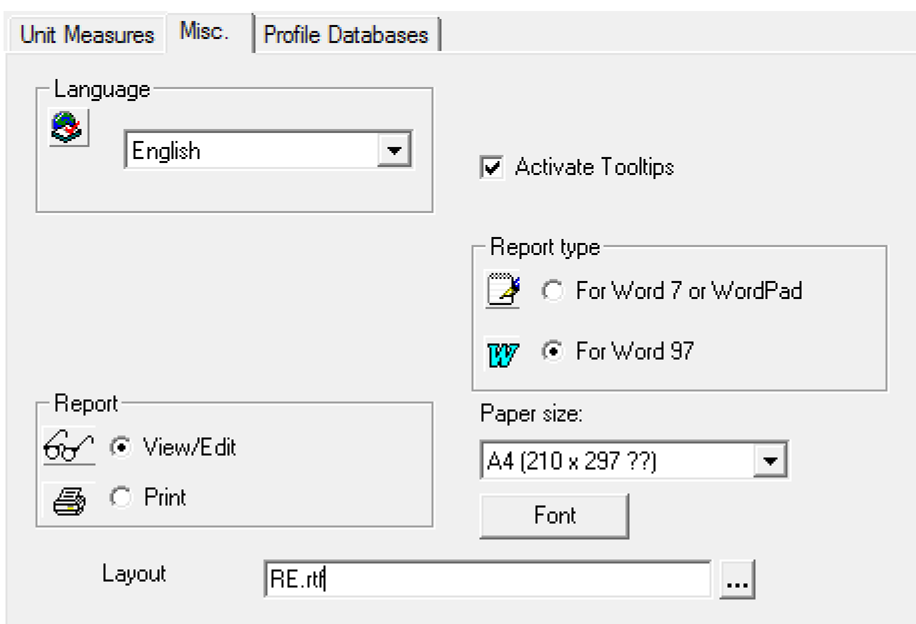
M. Preferences dialog
Used to set preferences for the Equivalent Section module.

Units Measures tab



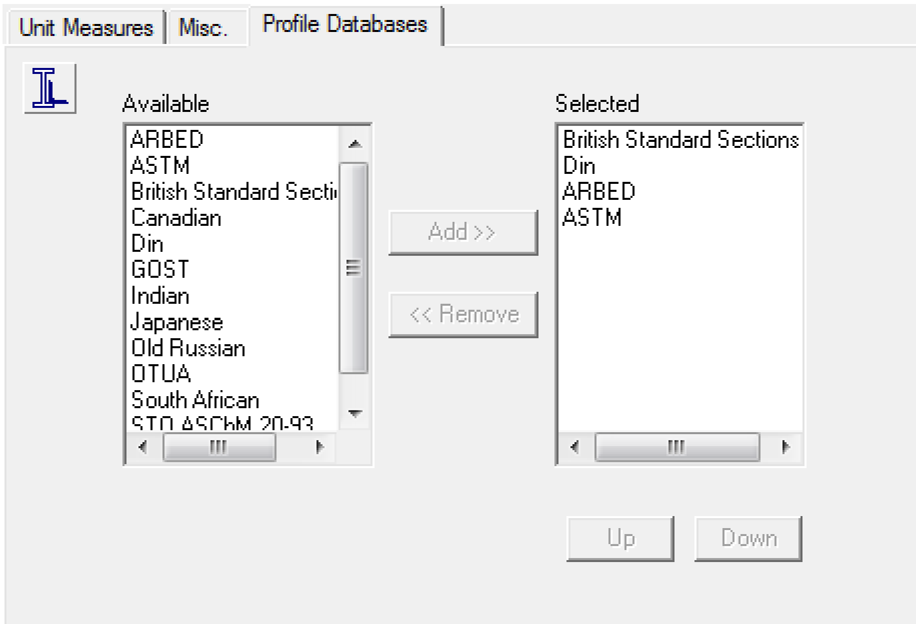
Select the units for angles, section dimensions, section properties, axial forces and moments, as well as the number of decimal places and whether or not to use an exponential form.

Misc. tab



Select the language, Report settings such as the template document, whether the documents should use Word 7 or Word 97 file formats, the paper size and font. Set whether documents should be sent direct to the printer or into Word for editing. Additional options determine the number of vertices to created by a circle, option to snap to grid and whether the ends of vertices are indicated by a circle or not.

Profile Databases tab



Select the range of databases that are referred to by the program. See Appendix 2: Section Databases for a current list of available database sections.

Other Controls

These controls are displayed at the bottom of the dialog for all tabs.

Apply	Saves any changes made in the dialog.
Cancel	Closes the dialog without saving any changes.
Help	Opens the Section Wizard help window to the relevant topic.

M. Section dialog

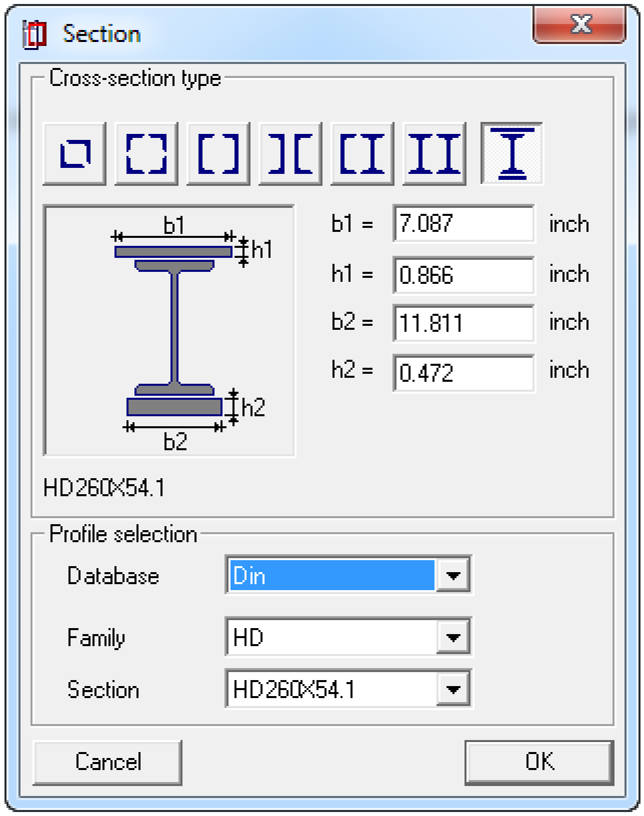


Figure 114: the Section dialog

Dialog controls

Section type selectors

Cover Plate dimensions (For I-sections with cover plates only) These fields become active when the I-section with cover plate type is selected. Specify values for **b1** and **h1** (width and thickness of top cover plate, respectively) as well as **b2** and **h2** (width and thickness of bottom cover plate, respectively).

Database

Family

Section

Cancel

OK

M. Free Sketch

Used to create sections with an outer boundary of any shape with a number of internal openings if required.

Free Sketch data files can saved in either one of two formats. `<filename>.CNS` is a binary file format which is the most compact and preferred way to store files. Alternatively, files can be stored as `<filename>.CON`, stores the file in an ASCII format which can be opened and edited by any text editor, such as Windows Notepad.

M. To set up the drawing environment

The scope of the section is determined by first setting the overall dimensions of the workspace to extend to (at least) the maximum dimensions of the section that is to be considered. A grid is then placed within this workspace, which if selected in the options, can be snapped to in order to accurately locate the coordinates of the boundary nodes.

1. Select either:

   Edit > **Overall dimensions**

   or

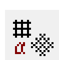
   the **Overall dimensions** tool 

   The **Overall dimensions** dialog opens
2. Type the maximum width and height of your section.

   Tip: This can be edited later by repeating this procedure.
3. Click **OK**.
   The section axis is displayed.
4. To set up a grid, do the following:
   a. Select either:

      Settings > **Grid options**

      or

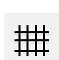
      the **Grid step** tool 

      The **Grid parameters** dialog opens.
   b. Type the **Y** and **Z** direction grid increments.
   c. (Optional) Type an **Angle** used to orient the grid.

      The angle is measured positive counter-clockwise from the Y axis.
   d. Click **OK**.
5. To display the grid, select either:

   Settings > **Grid**

   or

   the **Grid** tool 

   The grid is displayed and the tool is depressed on the toolbar and in the menu.
6. To snap points to the grid, select **Settings** > **Snap to grid**.

   Note: This tool is on by default when the grid is displayed.

M. To draw an external contour

Used to define the outside of the section shape.

Only one closed shape may be used to define a cross section.

1. Select either

   Edit > **Draw External Contour**

   or

   the **External Contour** tool 
2. Click any point in the View window as the first vertex of the exterior of the shape.
   A line is rubber-banded between this point and the mouse pointer.
3. Click subsequent points to form additional vertices.

   Tip: You may right-click to undo a previous vertex.
4. Double click the final point once you have formed a closed geometric shape.

   The final point does not need to be a previously entered data point. The rubber band to the first point will give you a preview of what the shape will look like for any point in the View window.

   The shape is shaded solid.
5. Exit the external contour drawing mode by either

   repeating step one

   or

   select some other drawing mode

M. To draw an internal contour

Used to define the openings or negative spaces within the cross section shape.

Inner contours must be closed shapes. Inner contour areas or vertices may not overlap.

1. Select either

   Edit > **Draw Internal Contour**

   or

   the **Internal Contour** tool 
2. Click any point in the View window as the first vertex of the inner contour.
   A line is rubber-banded between this point and the mouse pointer.
3. Click subsequent points to form additional vertices.

   Tip: You may right-click to undo a previous vertex.
4. Double click the final point once you have formed a closed geometric shape.

   The final point does not need to be a previously entered data point. The rubber band to the first point will give you a preview of what the shape will look like for any point in the View window.

   The shading inside of the shape is removed.
5. Exit the internal contour drawing mode by either

   repeating step one

   or

   select some other drawing mode

M. To round a corner

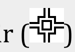
Uses this procedure to round –also referred to as fillet– interior or exterior contour corners.

1. Select either

   Edit > **Radius Corner…**

   or

   the **Smooth** tool 
2. Select any interior or exterior contour vertex.

   Tip: When hovering over a vertex, the mouse pointer changes to a double-line cross-hair ().

   The Smooth Radius dialog opens.
3. Type a radius value, **R**, and click **OK**.
   The previously specified radius is the default value.
   The corner is redrawn as a series of points forming an arc with the specified radius.
4. Exit the radius drawing mode by either

   repeating step one

   or

   selecting some other drawing mode

M. To insert a round opening

Use the following procedure to insert a round hole by one of two means.

A hole is an interior contour which is parametrically defined with a center and radius. You may specify a radius by an exact value or by drawing in the View window, depending on your tool selection.

1. Select either

   the **Circle hole** tool 

   or

   the **Circle hole with specified radius** tool 
   If you selected to specify a radius, the Radius of the circle hole dialog opens.
2. If you selected to specify a radius, enter the value and click OK in the Radius of the circle hole dialog.
3. Click a point to define the center of the circular hole.
   If you are defining the radius graphically, a circle is rubber-banded to the mouse pointer. Otherwise, the hole is drawn with the previously specified radius.
4. Double click a point to define a radius.
   The hole is drawn.
5. Exit the hole drawing mode by either

   repeating your selection in step one

   or

selecting some other drawing mode

M. To copy an internal contour

Use this procedure to make a copy of any internal contour or hole.

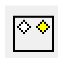
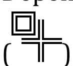
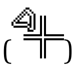
Inner contours must be previously specified for the tools used to be active.

Inner contours can be placed anywhere within the external contour.

1. Select either

   Edit > **Copy inner contour**

   or

   the **Copy inner contour(s)** tool 
2. From there, select either to use a **Rectangle** or **Polygon** window method for performing a selection.
   Depending on the window method selected, the mouse pointer either changes to a rectangular selection icon () or a polygonal selection icon ().
3. Click an initial point outside of the inner contour(s) you wish to copy.
   Depending on the window method selected, either a rectangle (rectangular) or a single line (polygon) is rubber banded to your mouse pointer.
4. Select the next point based on the following:

   For a rectangle, select the opposite corner of the rectangular window enclosing all of the inner contours you wish to copy.

   or

   For a polygon, click the next vertex along the polygon defining the outer edge of the window are. Repeat this step until the contours you wish to copy are enclosed, double-clicking the last point to close the polygon.
5. Specify a location of the copy by either

   Clicking anywhere within the windowed area and dragging the copy to where you wish to place it. Your mouse pointer becomes a multi-arrow icon () to indicate this option.

   or

   Right-click to open the **Copy of contour(s)** dialog which is used to specify **Y** and **Z** offsets. Click **OK** to place the copy at this location.
   Copies inner contours or their vertices may not overlap with other inner contours.

M. To delete vertices

Use this procedure to remove the vertices of external or internal contours.

1. Select either

   Edit > **Delete vertices**

   or

   the **Delete vertices** tool 

2. From there, select one of the following methods for vertex selection:

   Single

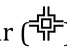
   or

   Rectangle

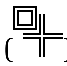
   or

   Polygon

   The mouse pointer updates to reflect the selection method.
3. Select either:

   a single vertex to delete; the mouse pointer changes to a double-line cross-hair ()

   or

   one corner of a rectangle window; the mouse pointer changes to a rectangular selection icon ()

   or

   the first vertex of a polygon window; the mouse pointer changes to a polygonal selection icon ()

   A single point selection is deleted and the shape is redrawn connecting the adjacent points. Once one or more vertices are enclosed in a completed rectangle or polygon window area, then those vertices are deleted and the adjacent vertices to those are connected.

M. To delete an opening

Use this procedure to delete an interior contour or hole from the shape.

Inner contours must be previously specified for the tools used to be active.

1. Select either

   Edit > **Delete internal contour**

   or

   the **Delete** tool 
2. Click anywhere inside the contour or hole you want to remove.
   The internal contour is removed from the shape.
3. Exit the delete contour mode by either:

   repeating step one

   or

   selecting some other drawing mode

M. To shift the coordinate center

Use this procedure to shift the coordinate center to any arbitrary point or to the center of gravity.

1. Select **Edit** > **Coordinate Center**
   The **Coordinate Center** dialog opens.

2. Specify a new coordinate center by either

   specifying both **Y** and **Z** shift values and click **Apply**

   or

   click **Shift** to specify the current shape center of gravity.

   Note: The origin must remain within the overall dimensions of the workspace.

   The coordinate origin shifts to the specified location.

M. To import a CAD drawing

This procedure is used to import a 2D CAD, closed area as a section profile. This allows you to use the CAD program of your choice to generate complex cross sections.

The following AutoCAD® entity types are supported in the import facility:

3DFACE
SOLID
TRACE
LINE
POLYLINE
LWPOLYLINE
ELLIPSE
CIRCLE
ARC

1. Select **File** > **Import DXF**
   The Import dialog opens.
2. Navigate to and select the `.dxf` file which you want to import.

   Note: The files that are to be imported are such that all the vertices are in a single plane and form closed areas.

3. Click **Open**.

To export shape for use in STAAD.Pro

To export the shape profile for use in STAAD.Pro as a user-provided table, do the following.

Note: This profile shape should consist of only an outer contour. Any internal voids such as internal contours or openings will be ignored, though section properties calculated by Section Wizard based on sections with voids will be used.

1. Select **File** > **Export to STAAD.Pro** > **General Section**.
   A **Save As** dialog opens.
2. Navigate to where you want to save the file, type a file name, and then click **Save**.
   The **User Table Units** dialog opens.
3. Select the unit of length for use with this shape and then click **OK**.
   The **General** shape dialog opens.
4. Type the **Section Name** for use in the STAAD.Pro user provided table.

5. (Optional) Type in any override values for section properties you want to use instead of those calculated by Section Wizard.

The shape vertices and calculated section properties are saved to the `.upt` file.

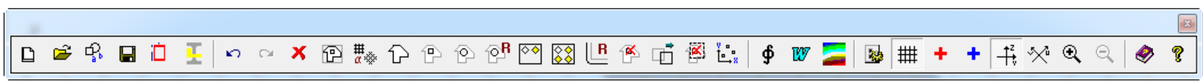
You can use this file as an external file for a user-provided section. Refer to M. To use a general shape created in Section Wizard (on page 746).

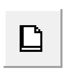
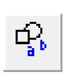
M. Application Window Layout

Drawing Window

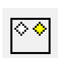
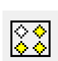
Left click selects a data point. Double click to accept. Right click to void last data point or current action (same as selecting the **Cancel operation** tool).

M. Toolbar
Contains several of the most commonly used tools found in the Free Sketch module.



Tool	Description	Same effect as selecting ...
 **New**	Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.	**File** > **New**
 **Open**	Opens the Open Data file dialog, which is used to select an existing Section Sketch Data file (file extension `.cns`). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the **Preview** option is checked, then a preview of the section can be displayed prior to opening the file.	**File** > **Open ...**
 **Parametric Section**	Opens the Parametric Section dialog which is used to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.	**File** > **Parametric Sections ...**

Tool	Description	Same effect as selecting ...
Save	Saves any changes made to the current data file. **Note:** If the current file has not been previously saved, this has the same effect as selecting **Save As...**	**File** > **Save**
Find Equivalent Section	Opens the M. Equivalent Section module (on page 763) with the current section properties as input values, which can be used to search for an Equivalent section to the current drawing.	**File** > **Equivalent Section ...**
Section Builder	Launches the M. Section Builder (on page 750).	
Undo	Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting Undo.	**Edit** > **Undo**
Redo	Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.	**Edit** > **Redo**
Cancel Operation	Cancels the current drawing operation.	ESC
Overall Dimensions	Opens the Overall Dimensions dialog, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.	**Edit** > **Overall Dimensions ...**
Grid Setup	Opens the Grid Parameters dialog, which is used to specify the grid spacing along the **Y** and **Z** directions and the Angle of rotation of the grid axes.	**Settings** > **Grid Options ...**

Tool	Description	Same effect as selecting ...
 **External Contour**	Used to set or clear the external contour drawing mode, which defines the outside of the section.	**Edit** > **Draw External Contour**
 **Internal Contour**	Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.	**Edit** > **Draw Internal Contour**
 **Circle hole**	Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.	**Edit** > **Insert circular hole, center and radius**
 **Circle hole with specified radius**	Insert circular hole with defined radius	**Edit** > **Insert circular hole with defined radius**
 **Copy inner contour(s)**	Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.	**Edit** > **Copy inner contour**
 **Multiple copies of inner contour(s)**	Used to window either a rectangle or polygon area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.	**Edit** > **Multiple copy of inner contour**
 **Smooth**	Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.	**Edit** > **Radius Corner...**
 **Delete**	Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.	**Edit** > **Delete internal contour**

Tool	Description	Same effect as selecting ...
Move Vertices	Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.	**Edit** > **Move vertices**
Delete Vertices	Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.	**Edit** > **Delete vertices**
Edit coordinates of Vertices	Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.	**Edit** > **Edit vertices**
Calculation	First opens the Poisson's Ratio dialog which is used to specify a value for **Nu**. Calculates the cross-sectional properties of the composite section and the opens the Basic Geometry dialog to display the values.	**File** > **Calculation**
Report	Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.	**File** > **Report**
Stress Field	Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the section drawing.	**File** > **Stress field**
Options	Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.	**Settings** > **Preferences ...**
Grid	Used to set or clear the display of the drawing grid in the View window.	**Settings** > **Grid**

Tool	Description	Same effect as selecting …
Center of Gravity	Used to set or clear the icon that shows the location of the center of gravity.	**Settings** > **Center of Gravity**
Shear Center	Used to set or clear the icon that shows the location of the shear center.	**Settings** > **Shear Center**
Show Coordinate Axis	Used to set or clear the display of the coordinate axis.	**Settings** > **Coordinate Axis**
Principle Axes	Used to set or clear the display of the principle axis of the current cross section.	**Settings** > **Principle Axis**
Zoom In	Used to increase the magnification in the View window.	**Settings** > **Zoom In**
Zoom Out	Used to decrease the magnification in the View window.	**Settings** > **Zoom Out**
Help	Opens the Section Builder help window.	**Help** > **Help Topics**
About	Opens the About Section Builder window, which displays version and copyright information.	**Help** > **About ...**

M. Menus
Menus in the Section Builder module window.
M. File menu
Contains items for creating, opening, and closing section data files, printing, and exiting the program.

Table 30: File menu items

Menu item	Description	Same effect as selecting…
New	Opens an empty section design file. If the currently loaded section has been modified, then the option is given to save it prior to commencing with the new design.	CTRL+N
Open	Opens the Open Data file dialog, which is used to select an existing Section Sketch Data file (file extension `.cns`). Controls in this dialog are analogous to a common Windows open dialog, except as noted in the following. If the **Preview** option is checked, then a preview of the section can be displayed prior to opening the file.	**<CTRL+O>**
Save	Saves any changes made to the current data file. **Note:** If the current file has not been previously saved, this has the same effect as selecting **Save As...**	**<CTRL+S>**
Save As…	Opens the Save data file dialog which is used to specify a file name and location for the section data file. Files are saved in the Free Sketch (`.CNS`) format.	
Close	This closes the current section and leaves Free Sketch ready to start a new file or open an existing file.	
Report	Formats and sends the data to the Report Type option defined in the Misc sheet of the Program Options dialog. See Options below.	

Menu item	Description	Same effect as selecting…
Calculation	First opens the Poisson's Ratio dialog which is used to specify a value for **Nu**. Calculates the cross-sectional properties of the composite section and the opens the Basic Geometry dialog to display the values.	
Stress Field	Opens the Section forces dialog, which is used to specify major axis moments and axial force on the section. Click OK to close the dialog and display the stress distribution on the section drawing.	
Parametric Sections…	Opens the Parametric Section dialog which is used to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.	
Rolled Section…	Opens the Rolled Section dialog, which is used to select a catalog section to insert into an new drawing file. The section can then be modified, just as you would with any other sketch.	
Equivalent Section…	Opens the M. Equivalent Section module (on page 763) with the current section properties as input values, which can be used to search for an Equivalent section to the current drawing.	
Export to STAAD.Pro >	Select either to export the section as a General or Prismatic section. A Save If an existing User Provide Table file is selected, the current section will be appended to the end of this table. Refer to the STAAD.Pro Technical Reference for additional information on using section types.	

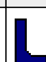
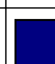
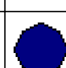
Menu item	Description	Same effect as selecting…
Import DXF	Opens a File Open dialog which is used to select CAD files to import in the *.DXF file format	
Send…	Opens an e-mail (using your default e-mail client) and attaches the current data file.	
Recent File list	A list of the most recent four section data files. Select one to open that data file.	
Exit	Closes the program. If any changes were made since the last time the data file was saved, a message dialog opens confirming you wish to save those changes.	**<ALT+F4>**

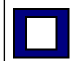
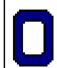
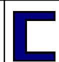
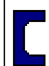
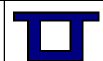
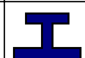
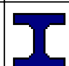
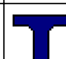
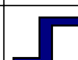
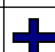
M. Parametric Sections dialog
Used to to quickly create sections based on a section type and a number of key dimensions and insert that section as an external contour.

Opens when **File** > **Parametric Sections ...** is selected.

Dialog controls

section type selector Select the general section type that you wish to use from the graphical list. The list contains 22 different generic section types. The parameters and diagram update according to the selection.

	Symbol	Section Type
1		Angle
2		Rectangle
3		Rounded Rectangle
4		Triangle
5		Six Side Polygon (Hexagon)
6		Eight Side Polygon (Octagon)

	Symbol	Section Type
7		Rectangular Hollow Section
8		Rounded Rectangular Hollow Section
9		Channel
10		Rounded Channel
11		Flanged Rectangular Hollow Section
12		Unsymmetric I Section
13		Rounded Symmetric I Section
14		Rounded Tee Section
15		Double Tee Section
16		Wedge
17		Z Section
18		Cruciform
19		Circle
20		Hollow Pipe
21		Half Pipe
22		Ellipse

section parameters	Fields for each of the section dimenions for the selected section type are displayed. Enter a dimension into each field (each section has at least one required dimension). Refer to the diagram for the selected section type for an explination of each field.
OK	Closes the dialog and adds an external contour with defined parametric section.
Cancel	Closes the dialog and returns to the previous sketch file.

M. Rolled Section dialog
Used to select a catalog section to insert into an new drawing file. The section can then be modified, just as you would with any other sketch.

Opens when **File** > **Rolled Sections ...** is selected.

Dialog controls

catalog lists	Catalog sections are listed in a tree format by Country/Standard, shape type, and section size. Expand the tree to locate the section you wish to add and select its entry.
OK	Closes the dialog and adds an external contour with selected catalog section.
Cancel	Closes the dialog and returns to the previous sketch file.

M. Edit menu
Contains items for

Table 31: Edit menu items

Menu item	Description	Same effect as selecting…
Undo	Negates the last drawing operation. You can undo a series of modifications by repeatedly selecting Undo.	CTRL+Z
Redo	Negates the last undo modification. You can redo a series of negated modifications by repeatedly selecting Redo.	CTRL+Y
Overall Dimensions ...	Opens the Overall Dimensions dialog, which is used to specify the size of the workspace that will be required to create the section. The minimum it should be set to is the overall dimensions of the section but can be set larger.	
Draw external contour	Used to set or clear the external contour drawing mode, which defines the outside of the section.	

Menu item	Description	Same effect as selecting…
Locate circular external contour	Opens the Radius of the external contour dialog, which is used to specify a radius value for a parametrically defined, circular external contour. **Note:** This menu item is only active if no other external contour has been defined.	
Draw internal contour	Used to set or clear the internal contour mode, which is used for creating internal openings within an external boundary.	
Copy inner contour > <list>	Used to window either a rectangle or polygonal area to select one or more internal contours for copying. Copies can be placed manually or by specifying offset distances.	
Multiple copy of inner contour > <list>	Used to window either a rectangle or polygon area to select one or more internal contours making multiple copying. Copies can be placed manually or by specifying offset distances and a number of copies.	
Delete internal contour	Used to set or clear the delete internal contour drawing mode, which is used to remove internal contours by clicking anywhere in their boundary.	
Insert circular hole, center, and radius	Used to set or clear the circular hole drawing mode, which is used to define the center of a hole (e.g., a parametrically defined, circular inner contour) and radius graphically.	
Insert circular hole with defined radius	Used to set or clear the circular hole with defined radius drawing mode, which is used to define the center of a hole graphically and then specify an exact radius.	

Menu item	Description	Same effect as selecting…
Parametric hole …	Opens the Parametric holes dialog, which is used to define the location and dimensions of either a circular or rectangular inner contour.	
Radius corner …	Used to set or clear the smooth corner drawing mode, which is used to select a corner to fillet and then opens the Smooth radius dialog to specify a radius value for the rounded corner.	
Move vertices	Used to window either a rectangular or polygonal area to select one or more vertices to move. Vertices can be moved either manually or by specifying offset distances.	
Edit vertices	Displays all vertex numbers in the View window and opens the Coordinates dialog which is used to specify precise coordinates of each vertex for a selected contour.	
Delete vertices	Used to select a single vertex or window either a rectangular or polygonal area to select one or more vertices for deletion.	
Coordinate center …	Opens the Coordinate Center dialog, which is used to specify shift the global coordinates to either an arbitrary location or to the center of gravity of the shape.	

M. Settings menu
Contains items for controlling non-element specific settings as well as the behavior and display of the drawing environment.

Table 32: Settings menu items

Menu item	Description	Shortcut
Preferences …	Opens the Preferences dialog, which is used to set the units, colors, output types, and databases for standard sections.	

Menu item	Description	Shortcut
Grid Options ...	Opens the Grid Parameters dialog, which is used to specify the grid spacing along the **Y** and **Z** directions and the Angle of rotation of the grid axes.	
Grid	Used to set or clear the display of the drawing grid in the View window.	
Snap to grid	Used to set or clear the snap to grid behavior, which provides drawing assistance by selecting the nearest grid point to a selected point in the View window.	
Snap to vertices	Used to set or clear the snap to vertices behavior, which provides drawing assistance by selecting the nearest vertex point to selected in the View window.	
Center of Gravity	Used to set or clear the icon that shows the location of the center of gravity.	
Shear Center	Used to set or clear the icon that shows the location of the shear center.	
Coordinate Axis	Used to set or clear the display of the coordinate axis.	
Principle Axis	Used to set or clear the display of the principle axis of the current cross section.	
Zoom In	Used to increase the magnification in the View window.	
Zoom Out	Used to decrease the magnification in the View window.	

M. Preferences dialog
Used to set the units, colors, output types, and databases for standard sections.

Opens when **Settings** > **Preferences ...** is selected.

Units of measurement tab

Select the units for angles, section dimensions, section properties, axial forces and moments, as well as the number of decimal places and whether or not to use an exponential form.

Misc. tab

Select the language, Report settings such as the template document, whether the documents should use Word 7 or Word 97 file formats, the paper size and font. Set whether documents should be sent direct to the printer or into Word for editing. Additional options determine the number of vertices to created by a circle, option to snap to grid and whether the ends of vertices are indicated by a circle or not.

Stress scale tab

Used to set the colors of the maximum compression and tension stresses and the colour these grade to for zero stress, along with the number of bands to be displayed

Profile databases tab

M. Grid Parameters dialog
Used to specify the grid spacing along the **Y** and **Z** directions and the Angle of rotation of the grid axes.

Opens when **Settings** > **Grid Options ...** is selected.

Dialog controls

Y	Grid spacing along the section local y axis (horizontal on screen).
Z	Grid spacing along the section local z axis (vertical on screen).
Angle	Angle from the coordinate axis to the grid lines, taken counterclockwise. Accepted values range between zero (inclusive) and 90 degrees. Negative values are not accepted. Only the drawing grid is rotated; no section data is changed.
OK	Saves changes and closes the dialog.
Cancel	Closes the dialog without saving any changes.

M. Service menu
Contains items for opening tools and utilities related to the Free Sketch module.

Table 33: Service menu items

Menu item	Description
Windows Calculator ...	Opens the Windows system Calculator window.
Formula Calculator ...	Opens the M. Formula Calculation window (on page 790) which is used to evaluate a formula, from the numerical result can then be copied back into the Free Sketch module.
Unit Converter ...	Opens the **Unit Converter** Window which is used to as a value converter between different preset units of measurement. Select the tab pertaining to the type of measurement and then enter the known value in the appropriate field. Upon pressing **<Enter>**, all the other fields are updated to equivalent values.

Menu item	Description
Section Builder ...	Launches the M. Section Builder (on page 750).

M. Help menu
The Help menu contains items for using online help.

Table 34: Help menu items

Menu item	Description	Shortcut
Help Topics	Opens the Section Builder help window.	F1
About	Opens the About Section Builder window, which displays version and copyright information.	

M. Formula Calculation window

Used for holding evaluations under the formulas, which are set by the user in the window of lead

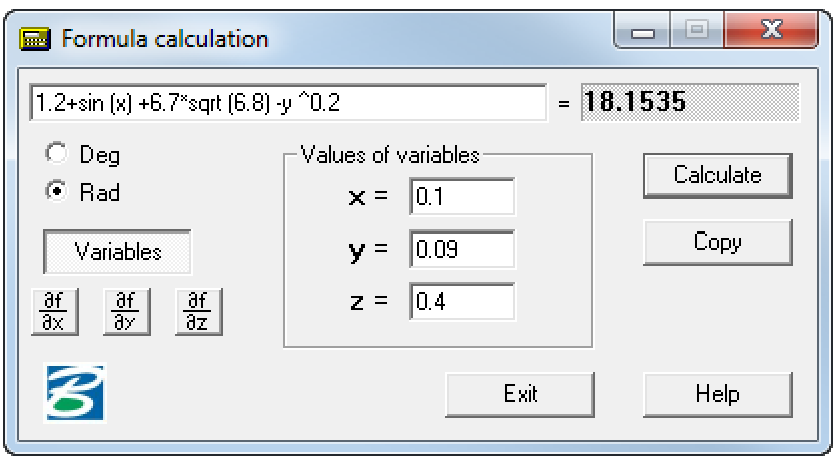


Figure 115: Formula calculation window

Formula Syntax

At lead of the formulas it is necessary to observe the following rules:

- the names of functions are entered by lower case letters of the Latin alphabet;
- a separator of fractional and whole parts of number is the point;
- the arithmetical operations are set by characters +, -, *, /, exponentiation ^ (for example, `2.5*2.5*2.5` is typed as `2.5 ^ 3`).

Supported Functions

At record of the formulas it is possible to use the following functions:

floor - the greatest integer not exceeding preset
tan - a tangent
sin - sine
cos - cosine
asin - an arcsine
acos - an arccosine
atan - an arctangent
exp - an exponential curve
ceil - the least integer exceeding preset
tanh - a tangent hyperbolic
sinh - sine hyperbolic
cosh - cosine hyperbolic
log - a Napierian logarithm
log10 - a Brigg's logarithm
abs - an absolute value
sqrt - the radical square

Depending on a state of the switch Degrees / radians, arguments trigonometrically functions (sin, cos, tan) and the outcomes return trigonometrically functions (asin, acos, atan) are reduced in degrees or radians accordingly.

Usage only of parenthesizes is admitted at arbitrary depth of an enclosure.

Examples

The formula:

$$1.2 + \sin(0.43) + 6.7\sqrt{6.8} - \sqrt[5]{0.003}$$

is typed as follows:

```
1.2+sin (0.43) +6.7*sqrt (6.8) -0.003 ^ 0.2
```

Click **Variable** and then the **x**, **y**, and **z** fields become active. Thus values of variables are set in appropriate windows of lead. It allows to carry out a series of one-type evaluations at different values of parameters. For example, in this condition the following formula

$$1.2 + \sin(x) + 6.7\sqrt{6.8} - \sqrt[5]{y}$$

is typed as follows:

```
1.2+sin (x) +6.7*sqrt (6.8) -y ^ 0.2
```

Then click **Calculate** to have the program resolve the variables into the formula.

Click **Copy** to copy the result to the Windows clipboard.

Moreover, the program allows to input some symbolic expression (depending on the variables x,y,z); click $\frac{\partial \mathbf{f}}{\partial \mathbf{x}}$, $\frac{\partial \mathbf{f}}{\partial \mathbf{y}}$, or $\frac{\partial \mathbf{f}}{\partial \mathbf{z}}$ and retrieve symbolic expression for the corresponding partial derivative.

M. Unit Converter

This is a utility program used to convert measurements from a known units of measurement into several different units.

The dialog includes several tabs, each corresponding to a type of measurement.

Length
Area
Volume
Force
Angle
Pressure
Moment of Couple

To convert units

1. Select the tab with the type of measurement you want to convert.
2. Type the known value in the field with the known units.
3. Press **<Enter>**. The value is converted in each field with different units on this tab.

Tip: You can double-click in a field to quickly select the contents. Right-click and then select **Copy** from the pop-up menu to save this value to the Windows clipboard.

Click **Exit** when you are finished converting values.

M. Materials and Constants

You can use material definitions or assign individual material constant values to beams, plates, and solids.

Note: Material definitions are recommended for most models.

M. To create a material definition

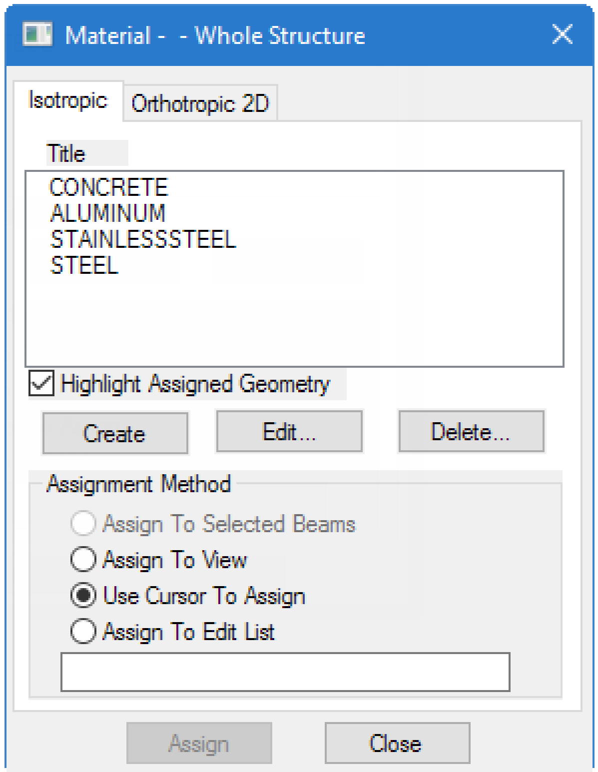
To create a new isotropic material definition for use with your model, use the following procedure.

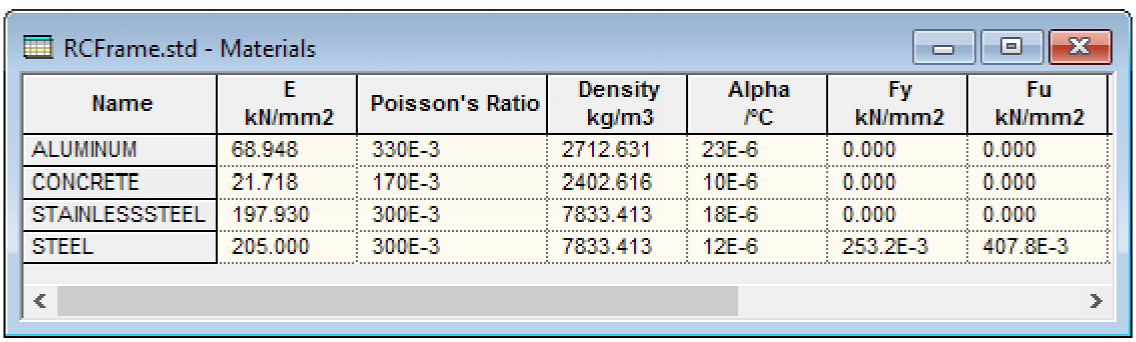
You may want to set the input units to a familiar set of units for defining materials before creating a new material definition.

STAAD.Pro includes a set of predefined materials for concrete, aluminum, steel, and stainless steel.

1. Select the **Materials** page in the **Analytical Modeling** page control bar.

   The **Material - Whole Structure** dialog and the **Materials** table open.

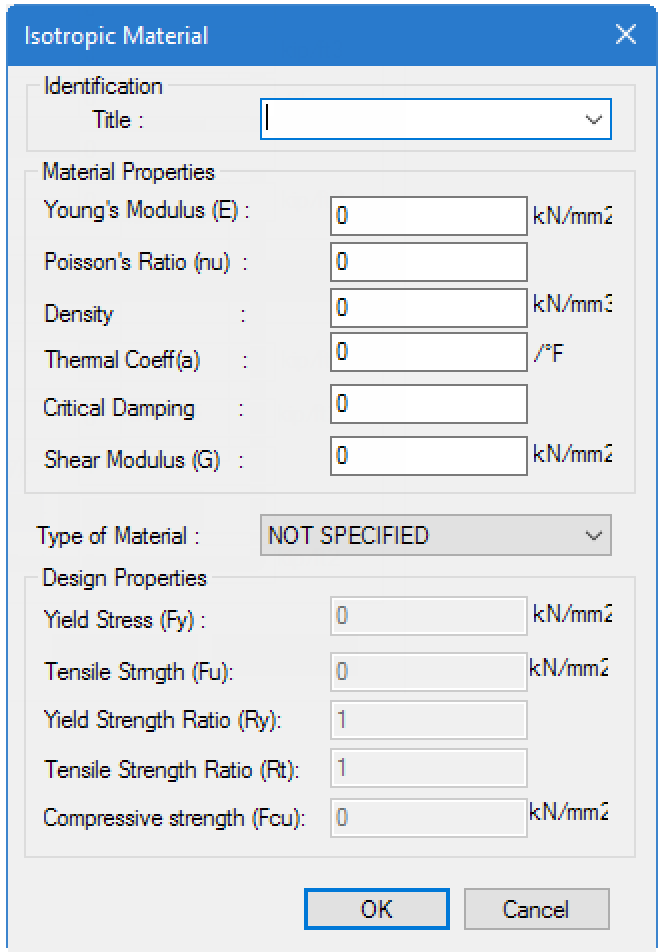



RCFrame.std - Materials

Name	E kN/mm2	Poisson's Ratio	Density kg/m3	Alpha /°C	Fy kN/mm2	Fu kN/mm2
ALUMINUM	68.948	330E-3	2712.631	23E-6	0.000	0.000
CONCRETE	21.718	170E-3	2402.616	10E-6	0.000	0.000
STAINLESSSTEEL	197.930	300E-3	7833.413	18E-6	0.000	0.000
STEEL	205.000	300E-3	7833.413	12E-6	253.2E-3	407.8E-3

2. In the **Material - Whole Structure** dialog **Isotropic** tab, click **Create**.

   The **Isotropic Material** dialog opens.



3. Type a **Title** to identify the material.

   This title must be different than the predefined material names or any other existing user-defined material name.
4. Type the values of the Material Properties used for analysis:

   Young's Modulus, E
   Poisson's Ratio, ν
   Density, γ
   Thermal Coefficient, α
   Critical Damping
   Shear Modulus, G
5. Select the **Type of Material** from the drop-down list.
6. Type the strength values in the **Design Properties** section.

   The fields corresponding to the selected material type are active.
7. Click **OK**.

The material definition is now added to the **Materials** table and is available for selecting when assigning shapes and in **Material - Whole Structure** dialog.

Related Links

- *TR.26.1 Define Material* (on page 2303)

- *Isotropic Material dialog* (on page 2821)

M. To add a predefined material

To add a predefined material definition, use the following procedure.

The program ships with a macro which reads predefined material data from a spreadsheet file and can be used to quickly add standard AISC and CSA steel material grades to your model.

Note: You can edit the data file `MaterialSpreadsheet.csv` (typically installed in `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\PlugIns\VBS\`) listed in the dialog to add more material values.

1. On the **Utilities** ribbon tab, select the **User Tools** > **Add Material** tool in the **Developer** group.

   The **Add Materials** dialog opens.
2. Select a material name and grade combination in the list.
3. Click **Add Material**.
   The material name is added to the **Material - Whole Structure** dialog.
4. (Optional) Repeat steps 2 and 3 to add additional materials.
5. Click **Cancel** to close the dialog.

M. To create an orthotropic material

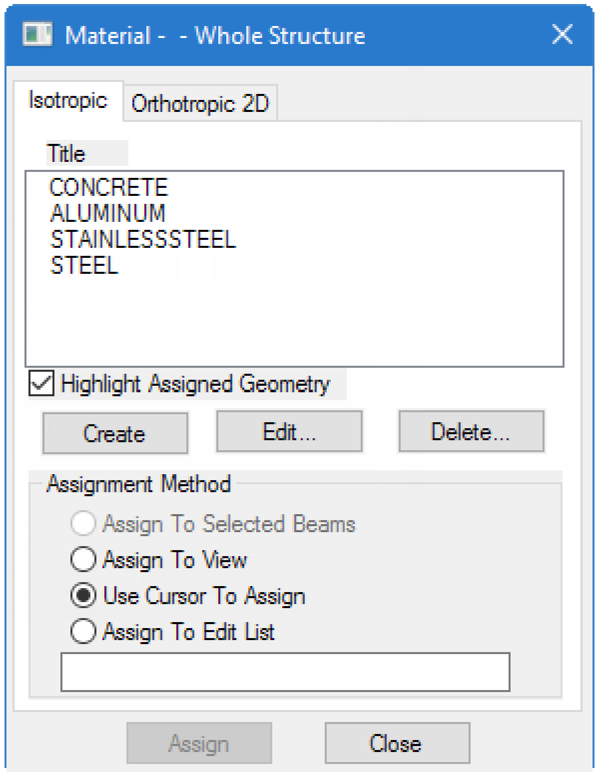
To create a 2D orthotropic material definition for use with orthotropic plate elements, use the following procedure.

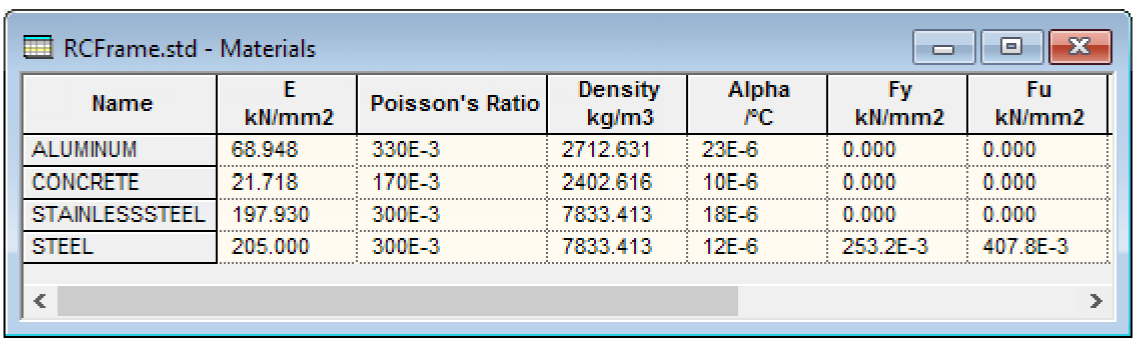
You may want to set the input units to a familiar set of units for defining materials before creating a new material definition.

Note: 2D othrotropic material definitions are for use with plate elements only.

1. Select the **Materials** page in the **Analytical Modeling** page control bar.

   The **Material - Whole Structure** dialog and the **Materials** table open.

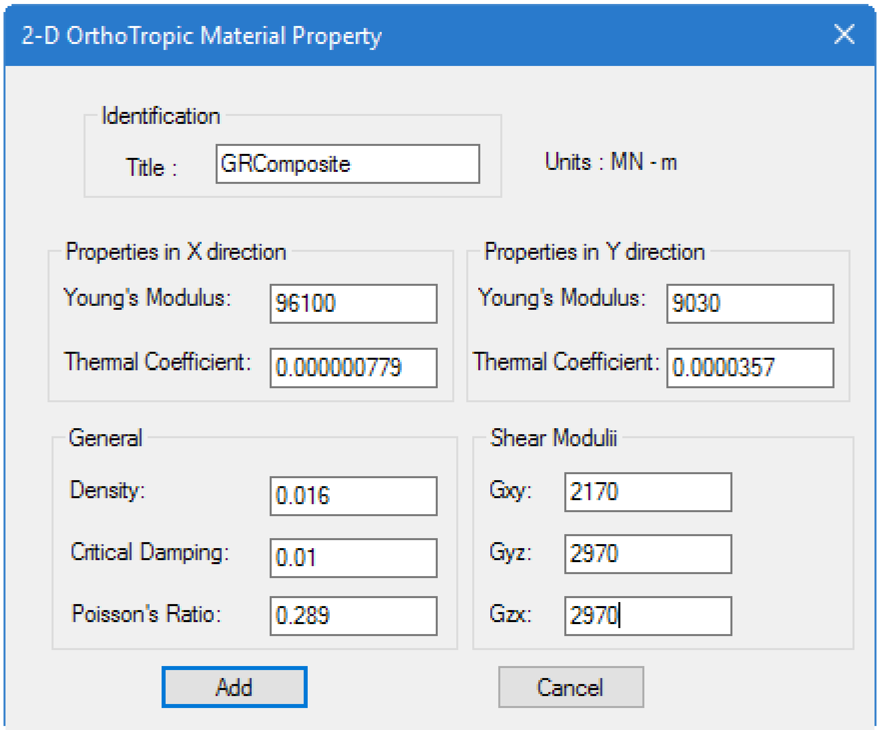




RCFrame.std - Materials

Name	E kN/mm2	Poisson's Ratio	Density kg/m3	Alpha /°C	Fy kN/mm2	Fu kN/mm2
ALUMINUM	68.948	330E-3	2712.631	23E-6	0.000	0.000
CONCRETE	21.718	170E-3	2402.616	10E-6	0.000	0.000
STAINLESSSTEEL	197.930	300E-3	7833.413	18E-6	0.000	0.000
STEEL	205.000	300E-3	7833.413	12E-6	253.2E-3	407.8E-3

2. In the **Material - Whole Structure** dialog, select the **Orthotropic 2D** tab.
3. Click **Create**.
   The **2-D OrthoTropic Material Property** dialog opens.



4. Type a **Title** for the material.
5. Type the material values:
   a. Type the **Young's Modulus** and **Thermal Coefficients** in the X direction.
   b. Type the **Young's Modulus** and **Thermal Coefficients** in the Y direction.
   c. Type the **Density**, **Critical Damping**, and **Poisson's Ratio** values.

      The same values are used in both X and Y directions.
   d. Type the Shear Modulus values for in-plane shear (**Gxy**), shear transverse to the local Y-Z direction (**Gyz**), and shear transverse to the local Z-X direction (**Gzx**).
6. Click **Add**.

The material definition is now added to the **Materials** table and is available for selecting when assigning shapes and in **Material - Whole Structure** dialog.

Related Links

- *TR.26.1 Define Material* (on page 2303)
- *2D Orthotropic Material Property dialog* (on page 2822)

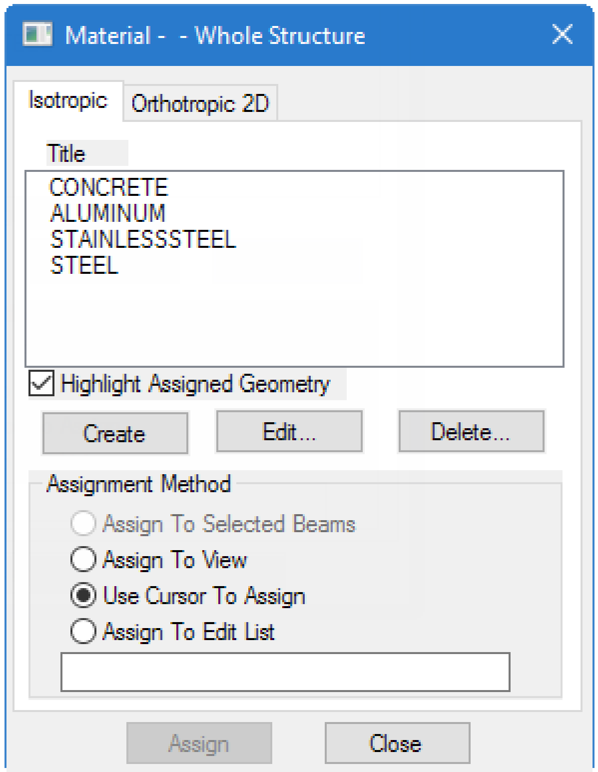
M. To assign material definitions

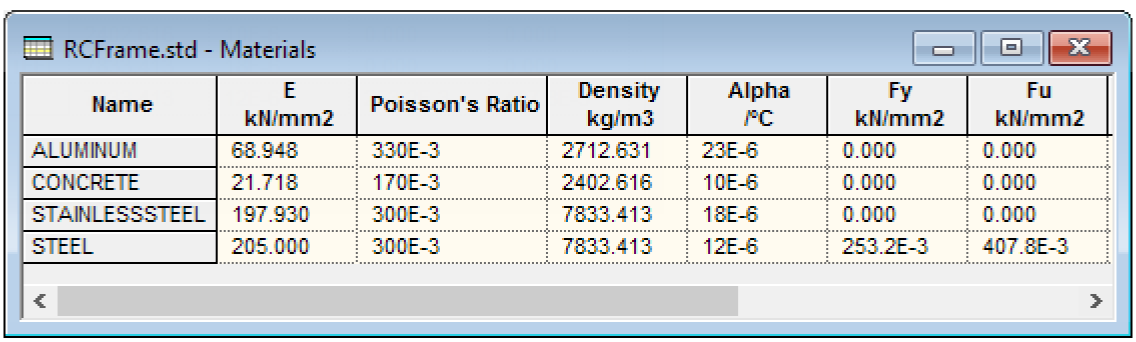
To assign material definitions to model objects (beams, plates, or solids), use the following procedure.

Often, you will assign a material definition to a beam along with the section assignment or to a plate with the thickness definition, as the material and profile or thickness are typically intrinsically associated. However, if you need to change materials or make assignments separately, this is done using the **Material - Whole Structures** dialog.

1. Select the **Materials** page in the **Analytical Modeling** page control bar.

   The **Material - Whole Structure** dialog and the **Materials** table open.




RCFrame.std - Materials

Name	E kN/mm2	Poisson's Ratio	Density kg/m3	Alpha /°C	Fy kN/mm2	Fu kN/mm2
ALUMINUM	68.948	330E-3	2712.631	23E-6	0.000	0.000
CONCRETE	21.718	170E-3	2402.616	10E-6	0.000	0.000
STAINLESSSTEEL	197.930	300E-3	7833.413	18E-6	0.000	0.000
STEEL	205.000	300E-3	7833.413	12E-6	253.2E-3	407.8E-3

2. Select the **Materials** page on the **Analytical Modeling** workflow page control bar.
   The
3. Assign the materials using one of the standard assign methods in the **Material - Whole Structures** dialog.

M. To assign material constants

To assign material constants to model objects (beams, plates, or solids), use the following procedure.

You may want to set the input units to a familiar set of units for a particular material value before assigning a material constant.

Tip: It is recommended to use material definitions in place of material constants. Material definitions also allow you to assign design strength properties.

1. Select the model object which have the same material constant.
2. On the **Specifications** ribbon tab, select the **Constants** tool in the **Materials** group.

A drop-down list of material constants opens.

3. Select the constant you want to assign:

    Young's Modulus
    Poisson's Ratio
    Shear Modulus
    Density
    Thermal Coefficient
    Damping Ratio

    The corresponding dialog for the material constant opens.

4. Either:

    select a predefined material constant value for a built-in material name: Aluminum, Concrete, or Steel

    or

    select the Enter Value option and type a value for the material constant
5. Select the **To Selection** option to limit the assignment to the selection set.
6. Click **OK**.

Related Links
- *Material Constant dialog* (on page 2809)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

M. Member Orientation

Related Links
- *M. To change a beam incidence* (on page 917)

M. To assign a member rotation angle

To assign an arbitrary member rotation angle about its longitudinal axis, use the following procedure.

The rotation of a member about it's longitudinal axis is referred to as the “beta angle” in STAAD.Pro.

1. Select one or more members in the view window.
2. On the **Specification** ribbon tab, select the **Beam** > **Beta Angle** tool in the **Specifications** group.

    The **Beta Angle** dialog opens.
3. Select the **Angle in Degrees** option and then type the angle of rotation.
4. Select the **To Selection** option.

> **Tip:** The **To View** option assigns the specification to all members in the view window.

5. Click **OK**.
   The rotation angle is assigned. A new beta angle specification is added to the Beta Angle tab on the **Properties - Whole Structure** dialog. This angle can be assigned to additional beams from this dialog.

Related Links

- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *Beta Angle dialog* (on page 2811)

M. To align a single angle to its flanges

To orient a single angle member aligned to its flanges, use the following procedure.

1. Select one or more members with single angle profiles in the view window.
2. On the **Specification** ribbon tab, select the **Beam** > **Beta Angle** tool in the **Specifications** group.

   The **Beta Angle** dialog opens.
3. Select the option to align the flanges (i.e., geometric axis):

To align…	Description
the long leg with the local y axis (ST angles)	select the **Angle** option This will rotate the angle section by *90°-a*.
the short leg with the local y axis (ST angles)	select the **RAngle** option This will rotate the angle section by *180°-a*.

where

a = the angle between the principle axis system and the geometric axis (i.e., parallel to the flange faces) system of the single-angle profile

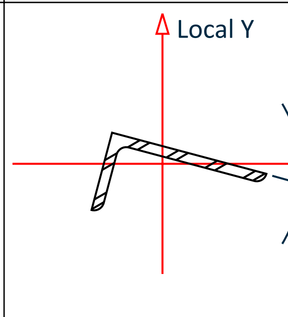
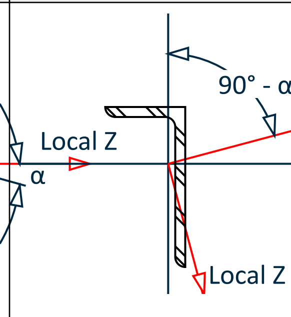
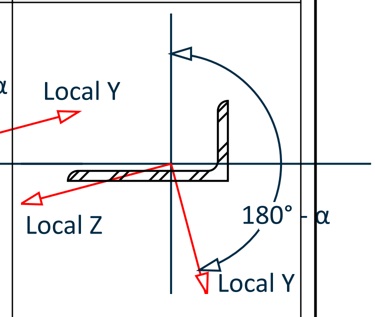
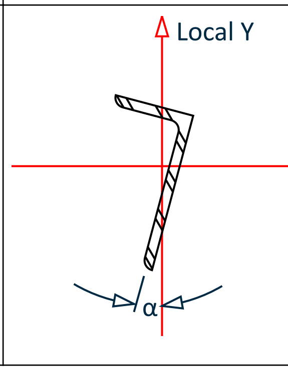
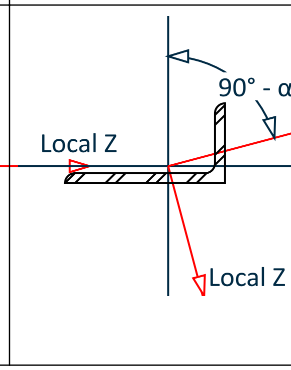
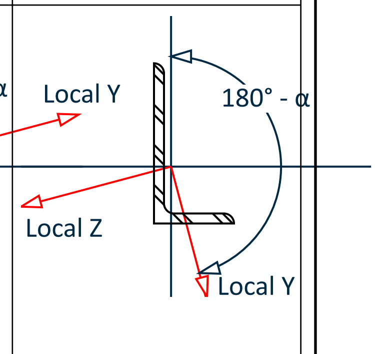
> **Note:** The orientation with the local y axis is for ST angles. RA angles are defined as being rotated 90° and thus the Angle and RAngle options similarly rotate from that initial orientation.

4. Select the **To Selection** option.

> **Tip:** The **To View** option assigns the specification to all members in the view window.

5. Click **OK**.

Table 35: Effect of BETA ANGLE and BETA RANGLE commands

BETA value =	Zero (0)	ANGLE	RANGLE
ST Angle			
RA Angle			

Related Links

- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *Beta Angle dialog* (on page 2811)

M. To align a member to a reference point

To orient a member to an existing node or arbitrary point, use the following procedure.

1. Select one or more members in the view window.
2. On the **Specification** ribbon tab, select the **Beam** > **Beam Reference Point** tool in the **Specifications** group.

   The **Referene Point** dialog opens.
3. Select the type of point to use as a reference for orientation:

To orient by...	Select the following...
an arbitrary point in space	select the **Point** option and define the coordinates with respect to the model origin

To orient by...	Select the following...
a vector from the member start node	select the **Vector** option and then define the second point of the vector coordinates with respect to the member start node
a node in the model	select the **Node** option and then select the node number from the drop-down list

4. Select the **To Selection** option.

   Tip: The **To View** option assigns the specification to all members in the view window.
5. Click **OK**.

Related Links

- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *Reference Point dialog* (on page 2812)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

M. Member Specifications

M. To add a member specification

To add a specification for beam members in your model, use the following procedure.

Tip: As with most assignment options, you can select a set of objects prior to starting this procedure and make the assignments to this selection set by using the dialog's **Assign** button.

Member specifications are used to assign beam members as cables, truss, tension- or compression-only. Similarly, additional specifications for member end offsets, fireproofing, and inactive members can be assigned.

1. Select the **Specifications** page in the Analytical Modeling page control bar.

   The **Specifications - Whole Structure** dialog opens.
2. In the **Specifications - Whole Structure** dialog, click **Beam**.
   The **Member Specification** dialog opens.
3. Select the dialog tab corresponding to the member specification you want to add:

   Release
   Offset
   Property Reduction Factor
   Cable
   Truss
   Compression
   Tension
   Inactive
   Fire Proofing
   Imperfection

> **Note:** Truss, Compression, Tension, and Inactive specifications have no additional parameters required. All other specifications require some additional input on the respective tab.

4. Either:

To...	Do the following...
add the specification to the model for later assignment	click **Add.**
add the specification to the model and assign to the current member selection	click **Assign.**

   The dialog closes.

The member specification is added to the **Specifications - Whole Structure** dialog and the **Specification Values** table.

You can repeat this procedure as many times as necessary to add additional member specifications to your model.

If you did not assign the specification to a selection set, you must select the specification in the **Specifications - Whole Structure** dialog and assign it using one of the assignment methods.

Related Links

- *Specifications - Whole Structure dialog* (on page 2783)
- *Member Specification dialog* (on page 2786)

M. To assign axial action members

To assign a member as compression-only, tension-only, or a truss (axial-only) member, use the following procedure.

STAAD.Pro allows you to specify the axial actions of members.

> **Note:** This assigns the analytical specification for truss members. For the purpose of Connection Design, you must additionally D. To assign member type attributes (on page 1031).

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** tool in the **Specification** group.

   A list of member specification types opens.
3. Select the specification type you want to assign:

   Compression Only – members are capable of carrying compression forces only

   or

   Tension Only – members are capable of carrying tensile forces only

   or

   Truss – members are capable of carrying axial forces only

   > **Note:** These member specifications have no additional parameters.

The **Member Specification** dialog opens to the corresponding tab.

4. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

The member specification is added to the **Specifications - Whole Structure** dialog and the **Specification Values** table.

Related Links

- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)
- *TR.23.1 Member Truss Specification* (on page 2289)
- *TR.23.3 Member Tension/Compression Specification* (on page 2292)
- *Member Specification dialog* (on page 2786)

M. To assign member end release

To assign a member end release or partial member end release, use the following procedure.

Release specifications are created for each of a member separately. To release both ends, you must create two specifications.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specifications** ribbon tab, select the **Beam** > **Release** tool in the **Specifications** group.

The **Member Specification** dialog opens to the **Release** tab.

3. Select which location on the member to which the release applies: **Start** or **End**.
4. Select the **Release Type**:

Partial Moment Release

or

Release

5. For **Partial Moment Release**, select the directions which are partially released and the release ratio:
   a. Check the boxes corresponding to the directions of release.

      Check MP to assign the same partial moment release in all three local directions. Check a combination of MPX, MPY, and MPZ to apply different partial moment releases in each direction.
   b. For each direction checked, type a release ratio.

      This is a ratio between 0 and 1, where 0 indicated full moment restraint and 1 indicates full release.
6. For full **Release**, check the release directions which apply.

   These are FX, FY, FZ, MX, MY, and MZ. Any directions not checked will be fully restrained in that degree of freedom.
7. (Optional) To specify a spring in any degree of freedom, check the corresponding Release spring direction and type a spring constant in the adjacent field (in the indicated units).

Springs can be applied to partial moment release or release types.

8. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links
- *G.7 Member and Element Release* (on page 2120)
- *TR.22.1 Member Release Specification* (on page 2286)
- *Member Specification dialog* (on page 2786)

M. To assign member end offsets

To assign a rigid end offset to one end of a member, use the following procedure.

Member end offsets can be used to model any situation where the end of the actual member does not coincide with the analytical node used for the member end incidence. For example,

working points that do not align (e.g., a gusset plate connection)
rigid end zones (e.g., a beam that connects to the flange of a stiff column)
modeling top of steel (e.g., the center line of a beam that supports a slab)

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** > **Offset** tool in the **Specification** group.

The **Member Specification** dialog opens to the **Offset** tab.
3. Select which **Location** on the member to which the offset applies: **Start** or **End**.
4. Select which **Direction** reference to use for the offset values: **Global** or **Local**.
5. Type the **Offsets** to use along each of the **X**, **Y**, and **Z** axes.
6. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links
- *G.11 Member and Plate Offsets* (on page 2125)
- *TR.25.1 Member Offset Specification* (on page 2296)
- *Member Specification dialog* (on page 2786)

M. To assign member imperfection for members

To assign member drift or camber for use with an imperfection analysis, use the following procedure.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** > **Imperfection** tool in the **Specification** group.

   The **Member Specification** dialog opens to the **Imperfection** tab.
3. Do either of the following:

   select **Camber** to define the maximum offset along a beam from a line connecting the end points (typically for beams)

   or

   select **Drift** to define the offset at the end of a member (typically for columns)
4. Select the **Local Direction** of the imperfection and then type the **Value**, which is taken as a ratio of the member length to offset (i.e., L/d).
5. (Optional) For Camber, type a **Respect** value which is used to determine when to skip camber imperfection calculations.

   This ratio results in the camber imperfection calculations being skipped when:

   - the compressive load is small,
   - the member stiffness (EI) is large, or
   - the member is short
6. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

   The dialog closes.

You must assign the member imperfection specification to one or more members. An imperfection analysis is required in order to perform these calculations.

Related Links

- *TR.26.6 Member Imperfection Information* (on page 2313)
- *Member Specification dialog* (on page 2786)
- *Perform Imperfection Analysis tab* (on page 2924)
- *TR.26.6 Member Imperfection Information* (on page 2313)

M. To assign nonlinear cable members

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** > **Cable** tool in the **Specification** group.

The **Member Specification** dialog opens to the **Cable** tab.

3. (Optional) Do either of the following:

   select the **Initial TENSION** option and then type a value for the tension force in the cable

   or

   select the **Unstressed TENSION** option and then type a value for the initial length of the cable

   If no value is given for either option, then a minimal cable tension is assumed.
4. Type a ratio value for one or more of the **Factor in global X : Fwx**, **Factor in global Y : Fwy**, or **Factor in global Z : Fwz**.

   Note: These loads are used for Advanced Cable Analysis only.

   The values are a multiplier of the self weight of the cable, applied in the selected global direction.
5. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *TR.23.2 Member Cable Specification* (on page 2290)
- *Member Specification dialog* (on page 2786)

M. To assign cracked section properties to a member

To assign reduced section property factors to a member, use the following procedure.

Global reduction factors will be applied to any member, regardless of material. Code-specific reduction factors are only applied to concrete members (non-concrete members included in the assignment are ignored).

Note: Automated stiffness reduction requires a STAAD.Pro Advanced license.

Note: For reducing the stiffness of a steel member for direct analysis per AISC 360, it is recommended use the `REDUCEDEI` parameter in the steel design parameters instead.

Notes:

a. Reduction factors are considered for analysis only but not for design.
b. Results using the reduced section properties are not available when using the member query feature.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** > **Cracked Property** tool in the **Specification** group.

The **Member Specification** dialog opens to the **Property Reduction Factors** tab.

3. Select either:

   Global – specify the **Reduction Factors** to be used below, which will apply to any member regardless of material.

   or

   Code Specific – select the building code to use reduction factors specific to load cases per that code which are only applied to concrete members:

   IS1893 2016
4. Type the reduction factor values to assign to the current member selection:
   - Reduction Factor for Cross sectional Area (RAX)

     Note: For IS1893 2016 reduction factors, the **RAX** input field is inactive as this property reduction is not mandated by that code for concrete members.
   - Reduction Factor for Torsion Constant (RIX)
   - Reduction Factor for Moment of Inertia, major axis (RIY)
   - Reduction Factor for Moment of Inertia, minor axis (RIZ)

   Note: Reduction factor values should be between 0 and 1 (inclusive).
5. Either:

To…	Do the following…
add the specification to the model and assign to the current member selection	click **Assign.**
add the specification to the model for later assignment	click **Add.**

The dialog closes.

Notes:

a. Reduction factors are considered for analysis only but not for design.
b. Results using the reduced section properties are not available when using the member query feature.

Related Links

- *Member Specification dialog* (on page 2786)
- *TR.20.10 Member Property Reduction Factors* (on page 2282)
- *TR.20.10 Member Property Reduction Factors* (on page 2282)
- *Member Specification dialog* (on page 2786)

M. To assign member fire proofing

To assign fire proofing to a member, use the following procedure.

Tip: Fire proofing thickness is specified in the current units of distance. You may want to change the units to a convenient value before beginning.

The fire proofing specification allows the program to automatically calculate the weight of the fire proofing material applied to members.

1. (Optional) Select the members which will all have the same member specification assigned.
2. On the **Specification** ribbon tab, select the **Beam** > **Fire Proofing** tool in the **Specification** group.

   The **Member Specification** dialog opens to the **Fire Proofing** tab.
3. Select the **Fire Proofing Type**:
   - **BFP (Block Fireproofing):**

     The fire-protection material forms a rectangular block around the steel section. The thickness specified is the minimum thickness which defines the outer block dimensions.

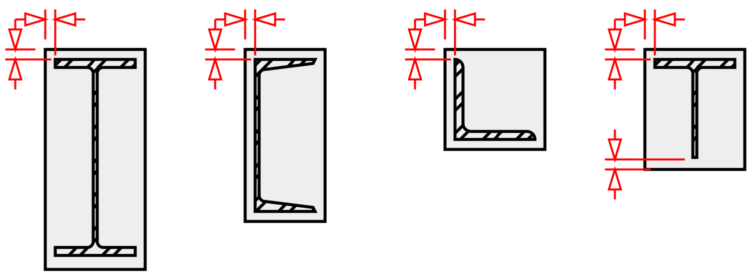
     

     Figure 116: BFP: Block Fireproofing
   - **CFP (Contour Fireproofing):**

     The fire-protection material forms a coating around the steel section. The thickness specified is a constant thickness around the section profile.

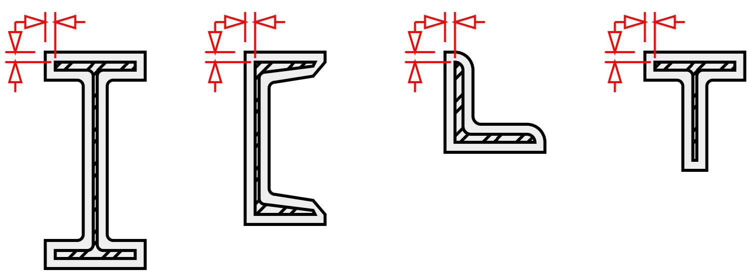
     

     Figure 117: CFP Contour Fireproofing
4. Type the **Thickness** of the fire proofing in the units indicated.
5. Type the **Density** of the fire proofing material.
6. Either:

To...	Do the following...
add the specification to the model and assign to the current member selection	click **Assign.**
add the specification to the model for later assignment	click **Add.**

The dialog closes.

Related Links

- *TR.20.9 Applying Fireproofing on members* (on page 2279)
- *Member Specification dialog* (on page 2786)

M. To assign specifications to physical members

To assign member end or axial specifications to physical members, use the following procedure.

Physical members can have member end releases, member end offsets, and truss specifications. Other analytical member specifications do not apply to physical members.

Note: For a physical member whose physical member specification is already assigned, the individual analytical members in that Physical member will adopt the same member specification that of the physical member. However, in case, where the analytical member specification is assigned to any member in that physical member, then this analytical member specification will supersede physical member specification.

1. Select the **Specifications** page in the Analytical Modeling page control bar.
   The **Specifications - Whole Structure** dialog opens.
2. On the **Geometry** ribbon tab, select the **Toggle Physical Member Mode** tool in the **Physical Members** group.

   The tool is highlighted to indicate the mode is active.
3. Click **Beam** in the **Specifications - Whole Structure** dialog.
   The **Member Specification** dialog opens.

   Note: The dialog only shows three tabs in the physical modeling mode.
4. Select the specification type you want to assign:

   Release – one or more degrees of freedom are free or partially restrained at one end of the member

   or

   Offset – a rigid offset is present between one end of the member and the joint

   or

   Truss – members are capable of carrying axial forces only
5. Add Release or Offset attributes as you would for analytical members.
   a. Select the **Location** on the member: **Start** or **End**.
   b. For a **Release**, select if either partial or full release and the degrees of freedom of the release.
   c. For an **Offset**, select the directions of the offset.
6. Click **Add**.

Related Links

- *TR.23.1 Member Truss Specification* (on page 2289)
- *TR.25.1 Member Offset Specification* (on page 2296)
- *TR.22.1 Member Release Specification* (on page 2286)

M. Plate Specifications

M. To align a plate to a reference point

To orient the local axis of a plate element towards or away from an arbitrary point, use the following procedure.

The local z axis of the plates will remain perpendicular to the plane of the elements. This tool will simply orient the local axis such that this z axis points generally toward or away from the specified reference point. Refer to G. 5.1 Plate and Shell Elements (on page 2099) for additional details.

1. Select one or more plates in the view window.
2. On the **Specification** ribbon tab, select the **Plate** > **Plate Reference Point** tool in the **Specifications** group.

   The **Plate Reference Point** dialog opens.
3. Specify the **X**, **Y**, and **Z** coordinates of an arbitrary reference point.
4. Select if the **Local Z Axis** of the elements should point towards or away from the reference point.

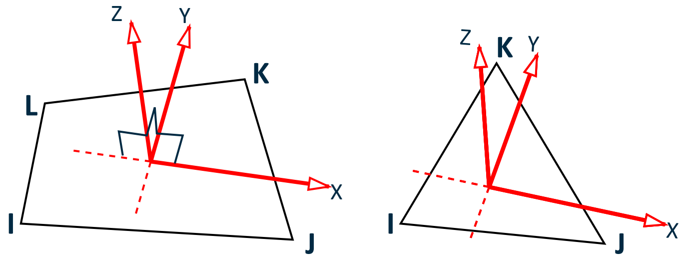


Figure 118: The local z axis of the plate is perpendicular to the plane of the element

5. Select the **To Selection** option.

   Note: The **To View** option assigns the specification to all plates in the view window.
6. Click **OK**.

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *Plate Reference Point dialog* (on page 2813)

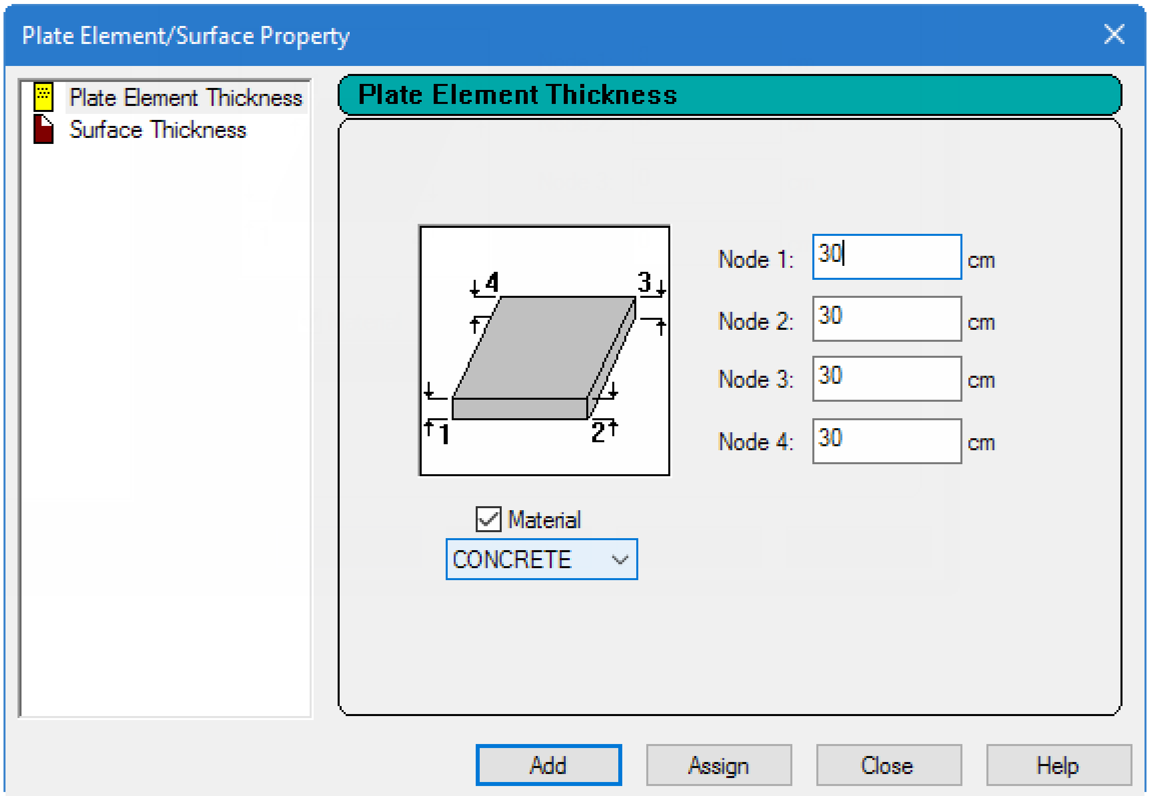
M. To specify plate thickness

To specify the thickness of a plate element, use the following procedure.

1. Select the plates which will have a similar thickness.
2. On the **Specification** ribbon tab, select the **Plate Thickness** tool in the **Plate Profiles** group.

The **Properties - Whole Structure** and the **Plate Element/ Surface Property** dialogs open.



3. Select the Plate Element Thickness tab.
4. Type the thickness values for the plate:

For a...	Do this...
uniform plate thickness	type the thickness in the Node 1 field. The other node fields will use this value by default
linearly varying thickness	type the thickness of the plate at each corner node in the corresponding field

5. (Optional) Check the **Material** option and select the material definition from the drop-down list.
6. Either:

   click **Assign** to assign the thickness (and optional material) to the selection set

   or

   click **Add** to add the thickness property to the model for assignment later

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *TR.21.1 Element Property Specification* (on page 2284)
- *Plate ElementProperty dialog* (on page 2813)

To assign plate offsets

To add rigid link offsets to one or more corners of a plate, use the following procedure.

Plate corner offsets can be used to model any situation where the corners of a plate element do not coincide with the analytical node used for the plate incidences. For example,

walls that meet at a corner (e.g., so that walls do not overlap)
wall and slab intersections (e.g., a wall that bears on top of a slab)

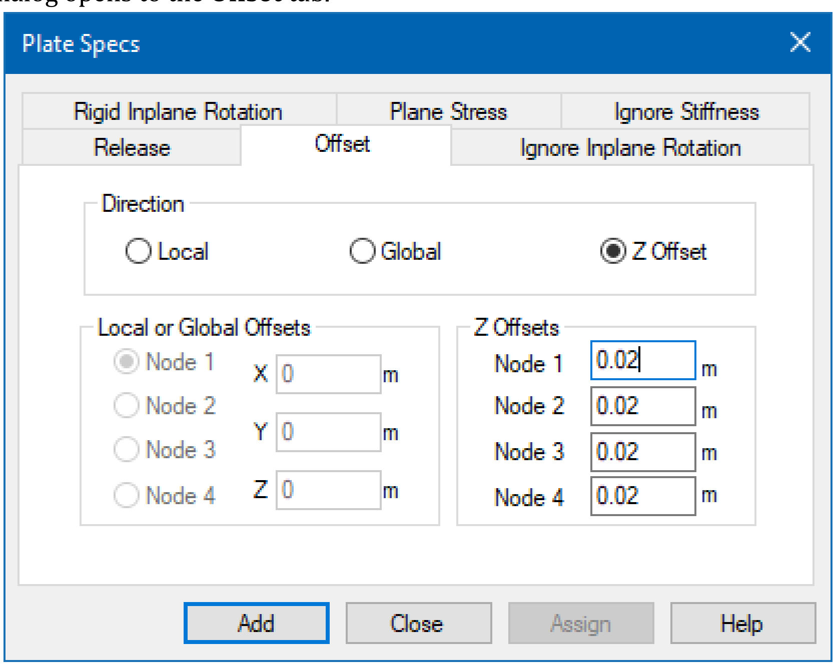
1. (Optional) Select the plates which will have the same plate offset specifications assigned.
2. Either:

   on the **Specification** ribbon tab, select the **Plate** > **Offsets** tool in the **Specification** group

   or

   on the **Specifications - Whole Structure** dialog, click **Plate**
   The **Plate Specs** dialog opens to the **Offset** tab.



3. Select the **Direction** in which you want to specify offsets:
   - **Local -** the nodal offsets at a specified joint are given in the element local axes.
   - **Global -** the nodal offsets at a specified joint are given in the global axes.
   - **Z Offset -** the offset is specified along the local z axis of the element (i.e., parallel offset to the plane of the element)
4. Specify the offset values:

For...	Do...
Local or global offsets	Select the local incidence node number and then specify the offsets in the local or global directions.
Z offsets	Type the offset value to use. By default, the same value will populate in all the node fields (i.e., a parallel offset to the plane of the plate). You can override these values by typing different values at different nodes.

Tip: To quickly identify the order of element corners, use the **Plate Cursor** tool and double click a plate. The Plate query dialog displays the nodes in order on the Geometry tab.

Note: The joint offsets must result in offset corners which are co-planar to prevent a warped surface.

5. Either:

To...	Do the following...
add the specification to the model and assign to the current plate selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.11 Member and Plate Offsets* (on page 2125)
- *TR.25.2 Element Offset Specification* (on page 2298)
- *Plate Specs dialog* (on page 2791)

M. To assign plate corner release

To release degrees of freedom of a plate element corner node, use the following procedure.

Releases are created for each node (by reference order) separately. To release multiple corners, you must create multiple specifications.

1. (Optional) Select the plates which will have the same plate release specifications assigned.
2. On the **Specification** ribbon tab, select the **Plate** > **Release** tool in the **Specifications** group.

The **Plate Specs** dialog opens to the **Release** tab.
3. Select which **Node** to which the release applies.

These are based on the local coordinate system for the element.

Tip: To quickly identify the order of element corners, use the **Plate Cursor** tool and double click a plate. The Plate query dialog displays the nodes in order on the Geometry tab.

4. Check the **Release** directions.

These are FX, FY, FZ, MX, MY, and MZ. Any directions not checked will be fully restrained in that degree of freedom.
5. Either:

To...	Do the following...
add the specification to the model and assign to the current plate selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.7 Member and Element Release* (on page 2120)
- *Plate Specs dialog* (on page 2791)
- *TR.22.2 Element Release Specification* (on page 2287)

M. To assign plates as plane stress

To assign plate elements as plane stress elements, use the following procedure.

This specification results in plates that only resist stress within the plane of the plate, but resist no out-of plane bending. These plate elements are analogous to truss members.

Tip: You can also assign this plate specification from the **Plate Specs** dialog, which opens when you click **Plate** on the **Specifications - Whole Structure** dialog.

1. Select the plates which will be plane stress elements.
2. On the **Specification** ribbon tab, select the **Plate** > **Plane Stress** tool in the **Specifications** group.

   The **Plate Specs** dialog opens to the **Plane Stress** tab.

   Note: There are no parameters to provide for this plate specification.
3. Either:

To...	Do the following...
add the specification to the model and assign to the current plate selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *Plate Specs dialog* (on page 2791)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)

M. To assign inplane rotation behavior to plates

To specify either no inplane rotational stiffness or completely rigid inplane rotational stiffness, use the following procedure.

Typically, the plate element used in STAAD.Pro has a very "soft" in-plane rotational, Mz, stiffness. There may be circumstances where this rotation should be ignored entirely (that is, with zero in-plane rotational stiffness). Alternatively, there may be circumstances where a rigid body rotation is required in plate elements.

1. Select the plates which will have the same inplane rigidity.
2. On the **Specification** ribbon tab, in the **Specifications** group, select either:

To...	Select this tool...
ignore the in-plane rotation actions of the plates	**Plate** > **Ignore Inplane Rotatoin**
treat the plates as rigid bodies for in-plane rotation	**Plate** > **Rigid Inplane Rotatoin**

   The **Plate Specs** dialog opens to the corresponding tab.

   Note: There are no parameters to provide for this plate specification.
3. Either:

To...	Do the following...
add the specification to the model and assign to the current plate selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

   The dialog closes.

Related Links
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *Plate Specs dialog* (on page 2791)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)

M. To ignore plate stiffness

To ignore the stiffness of a selection of plates, use the following procedure.

A plate element whose stiffness is ignored by the program acts will distribute any element loads but the stiffness of the plate does not contribute to the stiffness of the structure. In this way, plate elements can be used to model non-structural components, such as glass cladding.

Tip: You can also assign this plate specification from the **Plate Specs** dialog, which opens when you click **Plate** on the **Specifications - Whole Structure** dialog.

1. Select the plates which will have their stiffness ignored by the program.
2. On the **Specification** ribbon tab, select the **Plate** > **Ignore Stiffness** tool in the **Specifications** group.

   The **Plate Specs** dialog opens to the **Ignore Stiffness** tab.

Note: There are no parameters to provide for this plate specification.

3. Either:

To...	Do the following...
add the specification to the model and assign to the current plate selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *Plate Specs dialog* (on page 2791)
- *TR.22.3 Element Ignore Stiffness* (on page 2288)

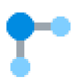
M. Node Specifications

M. To assign a rigid link between nodes

To create a rigid link between a control node any number of dependent nodes, use the following procedure.

A control/dependent specification establishes a primary node and a set of one or more subordinate nodes. Any displacements or rotations along the specified direction at the control node will be directly translated to the dependent nodes.

1. Select the nodes that will be the dependent nodes.
2. On the **Specifications** ribbon tab, select the **Nodes** > **Add Control/Dependent Specification** tool in the **Specifications** group.



The **Node Specification** dialog opens to the **Control/Dependent** tab.
3. Select the **Control Node** number from the drop-down list of nodes.
4. Select a rigid link specification in the **Dependent Directions**:

Rigid link type	Select...
in all directions	the **Rigid** option
within a plane	either the **XY**, **YZ**, or **XZ** option, corresponding to the plane
in specified degrees of freedom	the specific degrees of freedom which are linked between the primary and subordinate nodes: **FX**, **FY**, **FZ**, **MX**, **MY**, and **MZ**

5. Either:

To...	Do the following...
add the specification to the model and assign to the current node selection	click **Assign**.
add the specification to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.14 Rigid Diaphragms* (on page 2128)
- *TR.28.1 Control/Dependent Specification* (on page 2327)
- *Node Specification dialog* (on page 2784)

M. To assign nodes to a floor diaphragm

To define nodes that make up a floor diaphragm, use the following procedure.

Tip: Diaphragm heights and optional ranges are given in the current units of length. You may want to change the units of length to a convenient value before defining the diaphragm.

In order to generate the center of gravity for a diaphragm, you must create a mass load case which includes the weights to be considered for the diaphragm.

Note: A single diaphragm may be defined at a given level in STAAD.Pro.

1. (Optional) Select a set of nodes which will be used to define the diaphragm.
2. Select the **Specifications** page in the **Analytical Modeling** page control bar.
   The **Specifications - Whole Structure** dialog opens.
3. On the **Specifications - Whole Structure** dialog, click **Node**.
   The **Node Specification** dialog opens.
4. Select the **Floor Diaphragm** tab.
5. Specify the floor level of the diaphragm by either:

   selecting **Height** and then typing the height of the floor (Y coordinate value)

   or

   selecting **YRange** and then typing the **Minimum** and **Maximum** values of the diaphragm (Y coordinates)
6. (Optional) Select the **Define Floor Range** option to specify a boundary in the XZ plane of the diaphragm to define its edges:

   select the **Select Nodes** option to either assign the diaphragm to the current node selection or to assign it later

   or

   select the **Floor Range** option to type maximum and minimum values in the global X and Z directions to bound the diaphragm

   Nodes lying outside of this selection or range will not be considered to be part of the diaphragm even if they are at the diaphragm height.
7. (Optional) You can specify a control node to use as the center of gravity by selecting the **Select Control Node** option and then selecting the node number to use as the center of gravity.

   Otherwise, the program will calculate the center of gravity location for the diaphragm and add an analytical node at this location.

   Tip: To report calculated CGs, add the Cg (Center of Gravity) option in the Post-Analysis Print Commands.
8. Either:

To...	Do the following...
add the specification to the model and assign to the diaphragm to the current nodes	click **Assign**.
add the specification to the model for later assignment or to use all nodes at the specified Height	click **Add**.

The dialog closes.

Note: Soft story checks can be made for structures with rigid floor diaphragms per the IS 1893 2002, IS 1893 2016, and ASCE 7 specifications.

Related Links

- *TR.28.2 Floor Diaphragm* (on page 2328)
- *Node Specification dialog* (on page 2784)

M. Supports

This section describes how to create boundary conditions for your model.

Select the **Supports** page in the in the **Analytical Modeling** workflow to open the **Supports - Whole Structure** dialog.

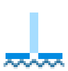
In STAAD.Pro, you will create support types that can be assigned to multiple nodes.

M. To assign a fixed or pinned support

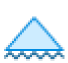
To specify a node as either a fixed or pinned support, use the following procedure.

A fixed support is restrained against movement (translation and rotation) in all degrees of freedom. A pinned support is restrained against translation only, but is otherwise free to rotate. Refer to Description of Pinned and Fixed (on page 2316) for additional details.

1. Select the nodes that will have the same support condition.
2. On the **Specifications** ribbon tab, select one of the following tools in the **Supports** group:

   Fixed 

   or

   Pinned 
3. The **Create Support** dialog opens to the corresponding tab.
4. Either:

To...	Do the following...
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.13 Supports* (on page 2127)
- *Create Support dialog* (on page 2815)
- *Supports - Whole Structure dialog* (on page 2814)
- *TR.27.1 Global Support Specification* (on page 2316)

M. To assign an enforced support

To assign a support which can have a support displacement imposed as a load, use the following procedure.

In STAAD.Pro, a special support type is used for supports that will have an imposed displacement as part of a load case. These are called "enforced" supports.

Note: If no support displacements are defined for an enforced support, then this support behaves as a fixed support (i.e., no translation or rotation). Similarly, an enforced but support behaves like a corresponding fixed but support if no support displacements are defined.

1. Select the nodes that will have the same support condition.
2. On the **Specifications** ribbon tab, select the **Other Supports** tool in the **Supports** group.
3. Select one of the drop-down support types as follows:

Support condition...	Select the following other support...
fixed in all degrees of freedom	**Enforced**
released in one or more degrees of freedom	**Custom Enforced**

   To impose a displacement on a joint with releases (e.g., a "pinned" support), you must use a custom enforced (i.e., `ENFORCED BUT`) support type and define the released degrees of freedom.

   The **Create Supports** dialog opens to the corresponding tab.
4. (Optional) For an enforced but support type, check the **Releases** for this support type.

   For example, to create a "pinned" type support, you would check the **MX**, **MY**, and **MZ** release options.
5. Either:

To...	Do the following...
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

   The dialog closes.

You must create a load case with support displacements defined at nodes with enforced or enforced but support types assigned in order for this support type to have an effect on your model beyond behaving as a corresponding fixed or fixed but support.

Related Links

- *G.13 Supports* (on page 2127)
- *EX. US-24 Analysis of a Concrete Block Using Solid Elements* (on page 6495)
- *EX. UK-24 Analysis of a Concrete Block Using Solid Elements* (on page 6782)
- *TR.27.1 Global Support Specification* (on page 2316)

- *Create Support dialog* (on page 2815)

M. To assign custom release supports

To assign a set of custom releases to a supported node, use the following procedure.

1. Select the nodes that will have the same support condition.
2. On the **Specifications** ribbon tab, select the **Custom** tool in the **Supports** group.

   The **Create Supports** dialog opens to the **Fixed But** tab.
3. In the **Releases** group, check the degrees of freedom to be released for this support type.

   Note: If a spring constant is specified in a direction, then the corresponding degree of freedom cannot be released. A given support type can have a mix of releases and springs, but only in different degrees of freedom.
4. Either:

To...	Do the following...
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

   The dialog closes.

M. To assign a spring support

To assign a linear spring in one or more degrees of freedom to a support, use the following procedure.

You may want to change the system of input units to convenient values for your spring constants before creating a spring support type.

In STAAD.Pro, linear spring constants are specified as part of “fixed but” support types.

1. Select the nodes that will have the same support condition.
2. On the **Specifications** ribbon tab, select the **Custom** tool in the **Supports** group.

   The **Create Supports** dialog opens to the **Fixed But** tab.
3. In the **Define Springs** group, type the linear spring constant to use for each translational and rotational spring direction.

   KFX - translation in the global x direction
   KFY - translation in the global y direction
   KFZ - translation in the global z direction
   KMX - rotation about the global x axis

KMY - rotation about the global y axis
KMZ - rotation about the global z axis

Note: If a spring constant is specified in a direction, then the corresponding degree of freedom cannot be released. A given support type can have a mix of releases and springs, but only in different degrees of freedom.

For example, a fixed but support could have MX, MY, and MZ released as in a pinned support and also have spring constants defined in FX, FY, and FZ.

4. Either:

To…	Do the following…
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

The dialog closes.

You can now assign tension-only, compression-only, or multi-linear springs to these same supported nodes if necessary.

Related Links

- *G.13 Supports* (on page 2127)
- *EX. US-3 Soil Springs for Portal Frame* (on page 6327)
- *EX. UK-3 Soil Springs for Portal Frame* (on page 6611)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)

M. To assign multilinear springs to spring supports

To model varying resistance to external loads at a spring support, use the following procedure.

You must assign spring supports to nodes before assigning multilinear spring definitions. You may not use multilinear springs in a model that contains tension-only or compression-only springs.

You may want to change the system of input units to convenient values for your spring constants before creating a spring support type.

Note: Multilinear springs can only be used with a select set of analysis types. Each load case should be separated by a Change command as well. See TR.27.4 Multilinear Spring Support Specification (on page 2322) for details.

1. Select one or more nodes assigned with spring support types.
2. On the **Specifications** ribbon tab, select the **Other Supports** > **Multi-Linear Spring** tool in the **Supports** group.
   The **Create Support** dialog opens to the **Multilinear Spring** tab.
3. In the table, type the Spring Stiffness constant which corresponds to a Displacement.
   These should be in order starting with the minimum displacement (e.g., this may also be the largest negative displacement) to the maximum positive displacement. All spring constants should be zero or positive values.
4. Click **Assign**.

M. To assign support springs as a one-way

To specify sprint supported nodes as capable of carrying only tension or compression, use the following procedure.

You must assign spring supports to nodes before assigning tension-only or compression-only spring definitions. You may not use tension-only or compression-only springs in a model that contains multilinear springs.

STAAD.Pro assumes that negative displacement is compression and positive displacement is tension.

1. Select one or more nodes assigned with spring support types.
2. On the **Specifications** ribbon tab, select the **One-way Spring** tool in the **Supports** group.

   The **Create Support** dialog opens to the **Tension/Compression Only Springs** tab.
3. Select the load direction the support is capable of resisting:

   Tension Only

   or

   Compression Only
4. Select the Spring Direction options for this support type:

   KFX
   KFY
   KFZ

   Note: One-way action is for translational springs only.
5. Click **Assign**.

Related Links

- *TR.27.5 Spring Tension/Compression Specification* (on page 2324)
- *Create Support dialog* (on page 2815)

M. To assign an inclined support

To assign a support that acts inclined with respect to the global axis, use the following procedure.

You may want to change the length unit to a more convenient value if you are going to define an incline direction by coordinate or vector.

See Inclined Support Axis System (on page 2318) for details on how the reaction directions are determined from the reference coordinate.

1. Select the nodes that will have the same support condition.
2. On the **Specification** ribbon tab, select the **Other Supports** > **Inclined** tool in the **Supports** group.
   The **Create Support** dialog opens to the **Inclined** tab.
3. Select the method for specifying the **Incline Reference Point**:

To use this reference...	Do...
an arbitrary point with respect to the supported node	select the **Coordinate** option and then type the relative global X, Y, and Z distances to the reference point
an arbitrary point with respect to the model origin	select the **Ref** option and then type the global X, Y, and Z coordinates
an existing node in the model	select the **RefJt** option and then select the node number from the drop-down list

Note: If you select the **Ref** or **RefJt** options, then the orientation of the inclined support axis system may vary from node to node.

4. Select the **Support Type** to use.

   Pinned - restrained against translation but free to rotate
   Fixed - restrained in all degrees of freedom
   Fixed But - released in specified degrees of freedom and can have spring constants assigned in restrained degrees of freedom
   Enforced - restrained in all degrees of freedom and can have imposed support displacements assigned as loads
   Enforced But - released in specified degrees of freedom and can have imposed support displacements assigned as loads
5. (Optional) For **Fixed But** support type, type linear spring constants to use as necessary.
6. For **Fixed But** or **Enforced But** support types, select the released degrees of freedom.
7. Either:

To...	Do the following...
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

The dialog closes.

Related Links

- *G.13 Supports* (on page 2127)
- *EX. US-19 Inclined Supports* (on page 6458)
- *EX. UK-19 Inclined Supports* (on page 6744)
- *TR.27.2 Inclined Support Specification* (on page 2318)
- *Create Support dialog* (on page 2815)

M. To assign a foundation support

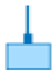
To automatically model a support using foundation parameters, use the following procedure.

Isolated footing dimensions and subgrade modulus values are specified in the current input units. You may want to change those to a convenient value before specifying a foundation support.

STAAD.Pro can take foundation parameters for footings or mat foundations and automatically model spring supports.

1. Select the nodes that will have the same support condition.

2. On the **Specification** ribbon tab, select the **Foundation** tool in the **Supports** group.

   The **Create Support** dialog opens to the **Foundation** tab.
3. Select the **Foundation** type to model:

   Footing - a rectangular, isolated footing (i.e., spread footing) area is used to model spring stiffness using the subgrade modulus

   or

   Elastic Mat - the program calculates the influence area of the load and then calculates a spring stiffness based on this area using the subgrade modulus

   or

   Plate Mat - the program calculates the influence area of the load within a specified plate and then calculates a spring stiffness based on this area using the subgrade modulus

   For additional details on the foundation types and their applicability, refer to TR.27.3 Automatic Spring Support Generator for Foundations (on page 2319).
4. Select the **Direction** of the resistance of the spring supports.

   The directions X, Y, or Z generate a spring in that global direction. The other two global directions are fixed against translation and moment about the selected direction is also fixed. The directions X Only, Y Only, or Z Only generate a spring in that global direction only.
5. Type the **Subgrade Modulus** of the soil.
6. (Optional) For **Elastic Mat** or **Plate Mat** foundation types, check the option **Print influence area of each joint** to include this information in the output.
7. (Optional) For **Elastic Mat** or **Plate Mat** foundation types, select a nonlinear spring option if necessary:

Option	Description
None	linear springs are generated
Compression Only	generated springs are compression only (made inactive if in tension)
Multi-Linear	generated springs can have a multilinear displacement-spring constant curve associated with the mat foundation

Tip: Multilinear springs must be specified using the STAAD Editor. The dialog box cannot accept displacement-spring constant curve data.

8. Either:

To...	Do the following...
add the support type to the model and assign to the current node selection	click **Assign**.
add the support to the model for later assignment	click **Add**.

   The dialog closes.

Related Links

- *G.13 Supports* (on page 2127)
- *EX. US-23 Spring Support Generation for a Slab on Grade* (on page 6484)

- *EX. UK-23 Spring Support Generation for a Slab on Grade* (on page 6771)
- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2319)
- *Create Support dialog* (on page 2815)

M. Loading Your Model

Load items in STAAD.Pro follow the general assigning workflow: you will define the load items within a primary load case. Then you will assign these load items to structural objects.

M. Available Structural Load Specifications in STAAD.Pro

The program contains the following load specifications:

Response Spectrum

Table 36: Codes available in STAAD.Pro with Response Spectrum loads

Country	Code	Title
Canada	TR.32.10.1.2 Response Spectrum Specification per NRC 2005 (on page 2506)	National Building Code(NRC/CNRC) of Canada
	TR.32.10.1.3 Response Spectrum Specification per NRC 2010 (on page 2512)	National Building Code(NRC/CNRC) of Canada
China	TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010 (on page 2531)	*Code for seismic design of buildings* GB50011-2010 (2016 Edition)
India	TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534)	Criteria for Earthquake Resistant Design of Structures - Part 1: General Provisions and Buildings and Part 4: Industrial structures including Stack-like structures
	TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016 (on page 2544)	Criteria for Earthquake Resistant Design of Structures - Part 1: General Provisions
	TR.32.10.1.9 Response Spectrum Specification per IS: 1893 (Part 4)-2015 (on page 2555)	Criteria for Earthquake Resistant Design of Structures Part 4 Industrial Structures Including Stack-Like Structures

Country	Code	Title
Europe	TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2520)	Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings (published 1996)
	TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2525)	Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings (published 2004)
Russia	TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81 (on page 2584)	Строительство в сейсмических районах (Construction in Seismic Regions)
	TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011 (on page 2589)	Строительство в сейсмических районах (Construction in Seismic Regions)
US	TR.32.10.1.10 Response Spectrum Specification per IBC 2006 (on page 2561)	International Building Code, 2006 edition
	TR.32.10.1.11 Response Spectrum Specification per IBC 2012 (on page 2566)	International Building Code, 2012 edition
	TR.32.10.1.12 Response Spectrum Specification per IBC 2015 (on page 2573)	International Building Code, 2015 edition
	TR.32.10.1.13 Response Spectrum Specification per IBC 2018 (on page 2578)	International Building Code, 2018 edition

Seismic

Table 37: Codes available in STAAD.Pro with Seismic loads

Country	Code	Title
Algeria	TR.31.2.1 RPA (Algerian) Seismic Load (on page 2353)	*Règles Parasismiques Algériennes*
Canada	TR.31.2.2 Canadian Seismic Code (NRC) - 1995 (on page 2355)	National Building Code (NRC/ CNRC) of Canada

Country	Code	Title
	TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1 (on page 2358)	National Building Code (NRC/ CNRC) of Canada
	TR.31.2.4 Canadian Seismic Code (NRC) - 2010 (on page 2362)	National Building Code (NRC/ CNRC) of Canada
China	TR.31.2.5 Chinese Static Seismic per GB50011-2001 (on page 2369)	*Code for Seismic Design of Buildings* GB50011-2001
	TR.31.2.6 Chinese Static Seismic per GB50011-2010 (on page 2375)	*Code for Seismic Design of Buildings* GB50011-2010 (2016 Edition)
Colombia	TR.31.2.7 Colombian NSR-98 Seismic Load (on page 2382)	*Reglamento Colombiano de Construcción Sismo Resistente* (NSR-98), *Normas Colombianas de Diseño y Construcción*, 1998, Asociación Colombiana de Ingeniería Sísmica
	TR.31.2.8 Colombian NSR-10 Seismic Load (on page 2384)	NSR-10 *Reglamento Colombiano Sismo Resistente*
India	TR.31.2.9 IS:1893 - 1984 Code - Lateral Seismic Load (on page 2386)	Criteria for Earthquake Resistant Design of Structures
	TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)	Criteria for Earthquake Resistant Design of Structures - Part 1 : General Provisions and Buildings
	TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)	Criteria for Earthquake Resistant Design of Structures - Part 4 : Industrial Structures Including Stack-Like Structures
	TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393)	Criteria for Earthquake Resistant Design of Structures - Part 1 : General Provisions and Buildings
	TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load (on page 2398)	Criteria for Earthquake Resistant Design of Structures Part 4 Industrial Structures Including Stack-Like Structures
Japan	TR.31.2.18 Japanese Seismic Load (on page 2418)	Building Codes Enforcement Ordinance 2006

Country	Code	Title
Mexico	TR.31.2.19 CFE (Comisión Federal De Electricidad) Seismic Load (on page 2420)	*Manual de Diseño por Sismo* - Comisión Federal de Electricidad (*Seismic Design Handbook* - Electric Power Federal Commission)
	TR.31.2.20 NTC (Normas Técnicas Complementarias) Seismic Load (on page 2423)	Reglamento de Construcciones del Distrito Federal de México (Mexico Federal District)
Turkey	TR.31.2.21 Turkish Seismic Code (on page 2426)	"Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention" Amended on 2.7.1998, Official Gazette No. 23390
US	TR.31.2.22 UBC 1994 or 1985 Load Definition (on page 2429)	Uniform Building Code, 1985 edition
	TR.31.2.22 UBC 1994 or 1985 Load Definition (on page 2429)	Uniform Building Code, 1994 edition
	TR.31.2.23 UBC 1997 Load Definition (on page 2432)	Uniform Building Code, 1997 edition
	TR.31.2.13 IBC 2000/2003 Load Definition (on page 2401)	International Building Code, 2000 & 2003 editions
	TR.31.2.14 IBC 2006/2009 Seismic Load Definition (on page 2405)	International Building Code, 2006 & 2009 editions
	TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409)	International Building Code, 2012 edition
	TR.31.2.16 IBC 2015 Seismic Load Definition (on page 2413)	International Building Code, 2015 edition
	TR.31.2.17 IBC 2018 Seismic Load Definition (on page 2416)	International Building Code, 2018 edition

Wind

Table 38: Codes available in STAAD.Pro with Wind loads

Country	Code	Title
China	Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog (on page 2887)	Load Code for the Design of Building Structures

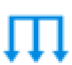
Country	Code	Title
India	IS-875 (Part 3): Wind Load dialog (on page 2890)	Design Loads (Other than Earthquake) for Buildings and Structures - Code of Practice: Part 3 Wind Loads (2015)
Russia	Russian Wind Loads (on page 2438)	Loads and Actions (1985)
	Russian Wind Loads (on page 2438)	Loads and Actions (2016)
US	ASCE 7 Wind Load dialog box (on page 2882)	Minimum Design Loads for Buildings and Other Structures
	ASCE 7 Wind Load dialog box (on page 2882)	Minimum Design Loads for Buildings and Other Structures
	ASCE 7 Wind Load dialog box (on page 2882)	Minimum Design Loads for Buildings and Other Structures
	ASCE 7 Wind Load dialog box (on page 2882)	Minimum Design Loads for Buildings and Other Structures

M. To create a new primary load case

This procedure applies to the **Loading** page in the **Analytical Modeling** workflow.

1. Either:

   select the **Primary Load Cases** tool in the **Loading** group on the **Loading** ribbon tab

   or

   select the **Load Cases Details** section on the **Load & Definitions** dialog and then click **Add**

   The Add New Load Cases dialog opens.
2. Select the **Primary** tab.
3. (Optional) Type a Number for the load case.

   The program will increment the load case number based on existing load case numbers.
4. (Optional) Select the **Loading Type**.

   This selection is used when automatically generating load combinations. If you select a live load type, you may also indicated if the load is **Reducible per UBC/IBC**.

Dead	Soil	Ice

Live	Rain Water/ Ice	Wind on Ice
Roof Live	Ponding	Crane Hook
Wind	Dust	Mass (Notes (on page 2461))
Seismic-H (horizontal, Notes (on page 2461))	Traffic	Gravity
Seismic-V (vertical, Notes (on page 2461))	Temperature	Push
Snow	Accidental	None
Fluids	Flood	

5. Type a **Title** used to easily identify this load case.
6. Click **Add**.
   The empty load case is added to the **Load Cases Details** list.

You must now add load items to this load case and assign them to objects in the structure.

You can edit the load case title and loading type by selecting the entry in the **Load Cases Details** list and then clicking **Edit** in the **Load & Definitions** dialog.

You can remove the load case (along with any associated load items) by selecting the entry in the **Load Cases Details** list and then clicking **Delete** in the **Load & Definitions** dialog.

Related Links

- *TR.32 Loading Specifications* (on page 2461)
- *Create Primary Load Case dialog* (on page 2835)
- *Create Primary Load Case dialog* (on page 2835)
- *TR.32 Loading Specifications* (on page 2461)

M. Load Items

Individual load items are added to primary load cases.

M. To add selfweight load

To add the calculated weight of the structure objects as a load item, use the following procedure.

You must create a primary load case first.

Note: The material definition or density constant is used for each member, element, or solid to determine the self weight load.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Select the **Selfweight** > **Selfweight Load** tab.
3. Select a global **Direction** for the selfweight.

   This is typically the "vertical" direction but can be any global direction.
4. Type a **Factor**.

   This is multiplier on the calculated weight of the structure objects. You do not need to use a load factor for ultimate strength design here, as that is typically applied to load combinations.

   Note: A positive value will act in the positive global direction, so you will typically use a negative value here for gravity loads (`-1` being the most commonly used value).
5. Click **Add**.
   The load item is added to the selected load case.

You must assign the load items to structure objects.

Tip: As selfweight typically will act on all members, plates, etc., you can use the **Assign to View** option and click **Assign** to quickly assign this load item to all objects in the structure (assuming that you have all objects currently displayed in the view window).

Related Links
- *TR.32.9.1 Selfweight Loads* (on page 2496)
- *Selfweight tab* (on page 2841)

M. To add a nodal load

To add forces or moments that act a nodes, use the following procedure.

Tip: Nodal loads are also referred to as "Joint Loads" in the STAAD command reference.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Nodal Load** > **Node** tab.
3. Type the force and moment magnitudes acting in the global directions (with sign) applied at the same nodes.

   You can use any combination of forces and moments.
4. (Optional) To apply the load along an inclined coordinate system:
   a. Select the **Inclined Load?** option.

The force and moment labels are updated with (') to indicate that the loads and moments will act in the inclined axes.

b. Select the method of defining the reference point.

- **Reference Node -** select an existing node number from the model. The global coordinates of that node are displayed as read-only.
- **Absolute -** type the global coordinates of an arbitrary point to use as the reference.
- **Relative -** type relative distances from the loaded joint to the reference point. These distances are measured along the global axes.

Tip: To easily select a reference node with respect to an existing node, first use the **Reference Node** option to populate the fields with that node's coordinates. Then select the **Absolute** option to modify the values. For example, you can update the Y coordinate of an existing node to select a point directly above or below that node.

c. Type the global coordinates (Absolute) or distances (Relative) to the reference point.

5. Click **Add**.
The load item is added to the selected load case.

Related Links

- *Nodal Load tab* (on page 2841)
- *G.15.1 Joint Loads* (on page 2128)
- *TR.32.1 Joint Load Specification* (on page 2462)
- *Nodal Load tab* (on page 2841)

M. To add a support displacement

To add a support displacement to a supported node, use the following procedure.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Select the **Nodal Load** > **Support Displacement** tab.

3. Select the displacement **Direction**.

4. Either

type the **Displacement** distance for translations (i.e., Fx, Fy, or Fz directions).

or

type the **Rotation** angle for rotations (i.e., Mx, My, or Mz directions).

5. Click **Add**.
The load item is added to the selected load case.

Related Links

- *G.15.7 Support Displacement Loads* (on page 2133)
- *TR.32.8 Support Joint Displacement Specification* (on page 2494)
- *Nodal Load tab* (on page 2841)

M. Member Load Items

M. To add a concentrated force or moment on members

To add either concentrated forces or members at a point along the length of a member, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   > **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select either:

   the **Member Load** > **Concentrated Force** tab

   or

   the **Member Load** > **Concentrated Moment** tab
3. Type the location along the length of the member to the concentrated load, **d1**.

   This distance is measured from the start node of the member.
4. Type the lateral offset from the geometric centerline to the load, **d2**.
5. Select the **Direction** option in which the load acts.
   Concentrated loads can be applied in the global or local coordinates.
6. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.2 Member Load* (on page 2129)
- *TR.32.2 Member Load Specification* (on page 2464)
- *Member Load tab* (on page 2842)

M. To add a uniform load to members

To add a uniform force or moment along the full or partial member length, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

> **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select either:

   the **Member Loads** > **Uniform Force** tab

   or

   the **Member Loads** > **Uniform Moment** tab
3. Type the magnitude of the distributed load (with sign), **W1**.
4. Type the location along the length of the member to the start of the uniform load, **d1** and to the end of the uniform load, **d2**.

   Both distances are measured from the start node of the member. If **d2** is zero, then the load will act to the end node of the member.
5. Type the lateral offset from the geometric centerline to the load, **d3**.
6. Select the Direction option in which the load acts.

   Uniform loads can be applied in the global, local, or projected coordinates.
7. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.2 Member Load* (on page 2129)
- *TR.32.2 Member Load Specification* (on page 2464)
- *Member Load tab* (on page 2842)

M. To add a linear varying load to members

To add linear varying force along the length of a member, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   > **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.
2. Select .

   the **Member Loads** > **Linear Varying** tab

   or

   the **Member Loads** > **Trapezoidal** tab
3. Select the load shape to apply:

Load increases...	**Select...**
linearly along entire length of beam	the **W1**, **W2** option on the **Linear Varying** tab
load increases from zero to peak at midspan and back to zero	the **W3** option on the **Linear Varying** tab
linearly along a partial section of the beam	the **Trapezoidal** tab

4. Type the load values at the ends (**W1**, **W2** option) or at midspan (**W3** option) (with sign).
5. Select the **Direction**.
6. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.2 Member Load* (on page 2129)
- *TR.32.2 Member Load Specification* (on page 2464)
- *Member Load tab* (on page 2842)

M. To add a prestress or post-tension load to members

To add a load due to prestressing or post-tensioning to members, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the loading **Type**:

   Prestress – prestress force is considered at the time of stressing and thus the force is transferred to the adjacent members or supports

   or

   Poststress - the prestress force is considered after the time of stress and thus the force is applied to the member itself (not transferred to the adjacent members or supports)
3. Type the prestress **Force** (in current units).
4. Type the **Eccentricity Distances** used to control the tendon profile at the **Start**, **Middle**, and **End** of the member.

   Eccentricities are measured in the local y axis (i.e., positive is "up" with respect to the local y axis). The cable profile is assumed to be parabolic (i.e, a "draped" tendon profile). Refer to G.15.5 Prestress and Poststress Member Load (on page 2131) for further details on how the tendon profile shape is calculated.
5. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.5 Prestress and Poststress Member Load* (on page 2131)

- *EX. US-6 Prestress and Poststress Loading* (on page 6351)
- *EX. UK-6 Prestress and Poststress Loading* (on page 6635)
- *TR.32.5 Prestress Load Specification* (on page 2489)
- *Member Load tab* (on page 2842)

M. To add fixed end member loads

To apply fixed end loads to ends of a beam, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Member Loads** > **Fixed End** tab.
3. Type the load values to apply at the **Start Node** and **End Node**.

   These reference the ends of the member but the loads are applied directly to the member ends, not the joints. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member.
4. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.4 Fixed End Member Load* (on page 2131)
- *TR.32.7 Fixed-End Load Specification* (on page 2494)
- *Member Load tab* (on page 2842)

M. Plate, Surface, Area, and Solid Load Items

M. To add pressure load on a plate

To add a uniform pressure load over an entire plate or a rectangular portion of a plate, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Select either:

   the **Plate Loads** > **Pressure on Full Plate** tab

   or

   the **Plate Loads** > **Partial Plate Pressure** tab
3. Type the magnitude of the uniform pressure (with sign), **W1**.
4. For partial plate pressure loads, type the coordinates (with respect to the center of the element) to the first and second corners of the loaded rectangular area: **X1**, **Y1**, **X2**, and **Y2**

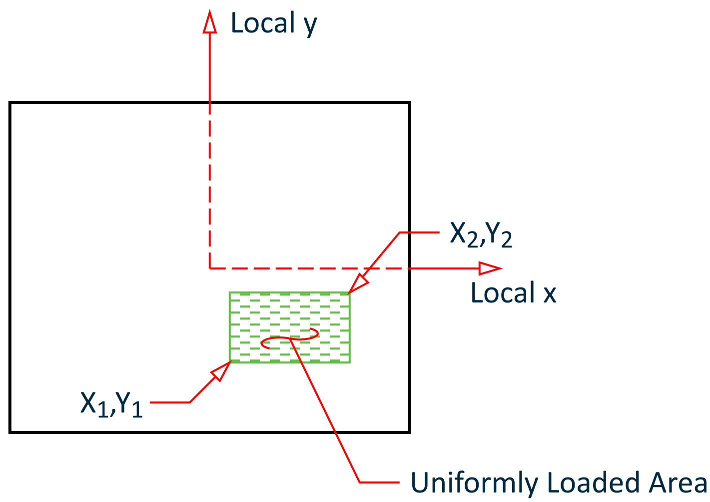


Figure 119: Coordinate values, x1,y1 & x2,y2, in the local coordinate system

5. Select the **Direction** option in which the load acts.

   Pressure loads can act in any of the global directions: **GX**, **GY**, or **GZ** as well as in the **Local Z** direction of the plate.

   Additionally, full pressure loads can act in the **Local Y** or **Local X** directions of the plate as well along projected directions, **PX**, **PY**, or **PZ.**
6. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *Plate Loads tab* (on page 2847)
- *G.15.8 Loading on Elements* (on page 2133)
- *Plate Loads tab* (on page 2847)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)

M. To add a concentrated load on a plate

To add a point load anywhere on the face of a plate, use the following procedure.

Note: If a load acts at a node point of an element, it is advisable to apply it using the Nodal Load option instead.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Select the **Plate Loads** > **Concentrated Load** tab.
3. Type the magnitude of the concentrated **Force** (with sign).
4. Type the local **X** and **Y** coordinates of the force on the plate face.

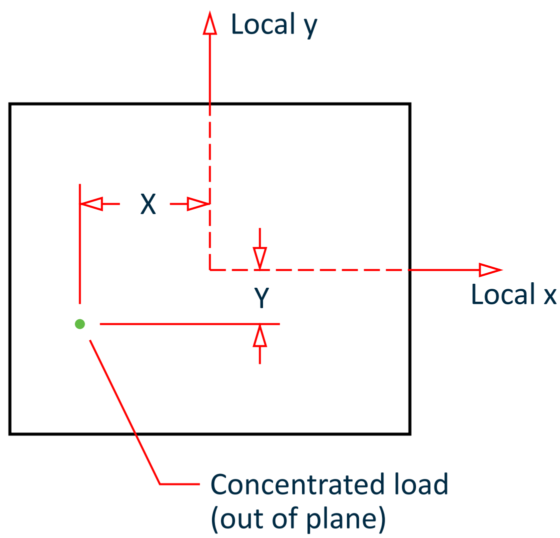


Figure 120: Coordinate values, X & Y, in the local element coordinate system

5. Select the **Direction** option in which the load acts.

   Concentrated loads can act in any of the global directions: **GX**, **GY**, or **GZ** as well as in the **Local Z** direction of the plate.
6. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.8 Loading on Elements* (on page 2133)
- *Plate Loads tab* (on page 2847)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)

M. To add an area load

To add an loaded area which distributes pressure to beams forming a closed loop, use the following procedure.

Note: The `AREA LOAD` command has been deprecated in favor of the `ONEWAY LOAD` or `FLOOR LOAD` commands.

1. Either:

On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

Tip: This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Type the magnitude of the **Pressure** in the current units.
3. Select the **Direction** in which the pressure acts.
   - **Local Z -** parallel to the member local Z axis
   - **GX** – parallel to the global X axis
   - **GY** – parallel to the global Y axis
   - **GZ** – parallel to the global Z axis
4. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *TR.32.4.1 Area Load Specification* (on page 2476)
- *Area Load tab* (on page 2845)

M. To add a floor load or one-way load

To generate a pressure load over a floor area using either two-way or one-way load distribution, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Floor Load** tab.
3. Either:

   select a range option –XRANGE, YRANGE, or ZRANGE– and then type the upper and lower bound distances (i.e., range) to define the members which define this loaded floor

   or

   select the **Group** option to apply the load to a named group and then select the Member Group name from the list

   Tip: This can be used to select a Composite Deck to assign a floor load to an entire deck. When decks are formed, this automatically creates a new group with the same name.

4. Type the **Pressure** in the current units.
5. Select the **Direction** in which the load acts.
6. (Optional) Select the **One Way Distribution** option to distribute the load towards the longer supporting members (i.e., along the shorter load path).
7. (Optional) When applying the load to a Group, you can check the **Inclined Floor** option to instruct the program that the load is applied to a set of members that form panels inclined to one of the global planes.
8. (Optional) You can report the panel and load data the program calculates:
   a. Check the **Print to output file** to have data included in the STAAD Output and then select the data to include there.
   b. Check the **Print to external text file** to have the data included in an external file and then select the data to include there.
9. Click **Add**.
   The load item is added to the selected load case.

Related Links

- *M. Composite Decks* (on page 676)
- *G.6.7 Composite Beams and Composite Decks* (on page 2119)
- *Composite Deck dialog* (on page 2741)
- *TR.20.7 Composite Decks* (on page 2270)
- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *Floor Load tab* (on page 2845)
- *TR.32.4.2 One-way Load Specification* (on page 2476)
- *TR.32.4.3 Floor Load Specification* (on page 2483)

M. To add a surface selfweight load

To add a selfweight load for surface elements so the weight of the surface elements is included in the analysis of the structure, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Surface Loads** > **Selfweight Load** tab.
3. Select a global **Direction** for the selfweight.

   This is typically the “vertical” direction but can be any global direction.
4. Type a **Factor**.

   This is multiplier on the calculated weight of the structure objects. You do not need to use a load factor for ultimate strength design here, as that is typically applied to load combinations.

   Note: A positive value will act in the positive global direction, so you will typically use a negative value here for gravity loads (`-1` being the most commonly used value).

5. Click **Add**.
   The load item is added to the selected load case.

The `SSELFWT` command, along with direction and factor, are added to the load case.

You must assign the load items to structure objects.
Related Links

- *TR.32.9.2 Surface Selfweight Load* (on page 2497)

M. To add a hydrostatic load to objects

To assign a hydrostatic load to members or plate elements, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   > **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

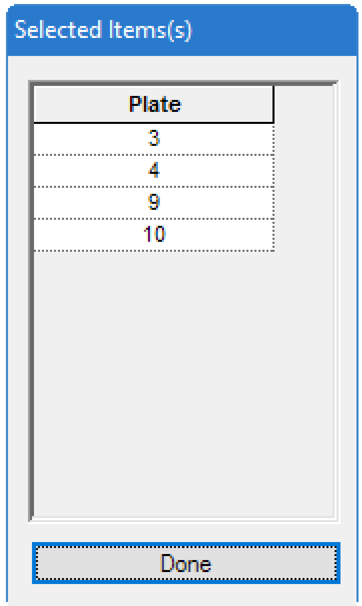
   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the appropriate tab:

Object type	Select this tab
Members	**Member Load** > **Hydrostatic**
Plates	**Plate Loads** > **Hydrostatic**

   > **Note:** A message on the dialog indicates this option is disabled because no objects are selected.
3. Click **Select *`objects`*** .
   The **Selected Item(s)** dialog opens and the **Add New Load Items** dialog closes temporarily.



4. Use the appropriate selection tool to select all the objects to which you want to apply the hydrostatic load in the view window.

   All objects must be of the correct type to populate the list (e.g., only members can be added to the Member list, only plates can be added to the Plate list).

   Note: Hold the **<Ctrl>** to select multiple objects individually.

   As objects are selected, their corresponding object id number is added to the object list.
5. Click **Done**.
   The **Selected Item(s)** dialog closes and the **Add New Load Items** dialog re-opens. The selected objects are now displayed in the objects list.
6. Type values for the minimum and maximum global forces.
7. Select the direction in which the force acts and the global axis along which the program should interpolate loads.
8. Click **Add**.
   The dialog closes and the hydrostatic load items are added to the selected load case.

Unlike most other loads, you do not need to assign these load items. The selection of the objects is used for assigning the load. The program will calculate the exact trapezoidal load pattern based on the other parameters.
Related Links

- *G.15.2 Member Load* (on page 2129)
- *TR.32.2 Member Load Specification* (on page 2464)
- *Member Load tab* (on page 2842)

M. To calculate the structure frequency

To calculate the frequency of the structure for a load case, use the following procedure.

Frequency calculation is made for a specific load case, so you must have at least one primary load case added to the input file.

This can be done by either the Rayleigh method or the eigenvalue extraction method. The latter can consider missing mass mode.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select either:

   the **Frequency** > **Rayleigh Frequency** tab to use the approximation method

   or

   the **Frequency** > **Modal Calculation** tab to use the more exact eigenvalue extraction method
3. (Optional) For Modal Calculation, check the **Consider Missing Mass** option to include the missing mass procedure for steady state or harmonic analysis.
4. Click **Add**.
   The frequency calculation is added to the load case.

Related Links

- *TR.34.1 Rayleigh Frequency Calculation* (on page 2614)
- *TR.34.2 Modal Calculation Command* (on page 2615)
- *Frequency tab* (on page 2865)
- *EX. US-28 Calculation of Modes and Frequencies of a Bridge* (on page 6536)
- *EX. UK-28 Calculation of Modes and Frequencies of a Bridge* (on page 6823)
- *TR.34.2 Modal Calculation Command* (on page 2615)

M. Wind Loads

This section describes how to add wind loads to your STAAD.Pro model.

M. To add a wind load definition

To add a custom wind profile definition to your model, use the following procedure

Wind load definitions describe the vertical wind profile that acts in a specific direction.

1. In the **Load & Definition** dialog, select the **Definitions** > **Wind Load** entry and then click **Add**.
   The **Add new: Wind Definition** dialog opens.
2. Select **Custom** from the **Code** drop-down list.

   Tip: You can also type optional **Comments** to further describe the wind definition (e.g., “Wind in X direction”).
3. Click **Add**.

The definition type is added to the **Definitions** > **Wind Load** entry in the **Load & Definition** dialog.

4. Either:

   Repeat steps 2-3 to add more wind definitions

   or

   click **Close**
5. In the **Load & Definition** dialog, select the new wind definition type and click the **Add**.
   The **Add New: Wind Definitions** dialog opens. The dialog now displays the **Intensity** and **Exposures** tabs.

   Note: The previously selected **Code** displays as a read-only selection.
6. Enter the wind profile data in a specific direction in pressure and height above ground pairs.

   Tip: You can paste data copied from a spreadsheet.
7. On the **Exposure** tab, type the **Factor** value.

   A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction.
8. Click **Add**.
9. (Optional) Repeat Steps 5, 7, and 8 to add additional exposure factors to this wind definition.

   You may add up to 99 exposure factors to a wind definition.

You must now apply this wind profile in a load case and specify the direction of the wind.
Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *Create Wind Type Definition dialog* (on page 2880)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *Create Wind Type Definition dialog* (on page 2880)

M. To add an ASCE 7 wind load definition

To populate the wind intensity table with values calculated per ASCE 7, use the following procedure.

These steps are performed when M. To add a wind load definition (on page 844) in the **Add New: Wind Definitions** dialog.

1. In the **Load & Definition** dialog, select the **Definitions** > **Wind Load** entry and then click **Add**.
   The **Add new: Wind Definition** dialog opens.
2. Select one of the following options from the **Code** drop-down list:

   - ASCE 7: 1995
   - ASCE 7: 2002
   - ASCE 7: 2010
   - ASCE 7: 2016

   Tip: You can also type optional **Comments** to further describe the wind definition (e.g., “Wind in X direction”).
3. Click **Add**.
   The definition type is added to the **Definitions** > **Wind Load** entry in the **Load & Definition** dialog.

4. In the **Load & Definition** dialog, select the new wind definition type and click the **Add**.
   The **Add New: Wind Definitions** dialog opens. The dialog now displays the **Intensity** and **Exposures** tabs.

   Note: The previously selected **Code** displays as a read-only selection.

5. Click **Generate**.
   The **ASCE-7: Wind Load** dialog opens.
6. On the Common tab, enter the general code and site information:
   a. Select the year (edition) of ASCE-7 code.
   b. Select the **Building Classification Category**.
   c. Type the **Basic Wind Speed** in the selected units.
   d. Select the **Exposure Category**.
   e. Select the **Structure Type** that best describes your structure.
   f. For ASCE 7-16, type the **Height Above Sea Level** of the base of the structure and select the appropriate units of length for this input.
   g. (Optional) If you need to **Consider wind speed-up over Hills or Enscarpment**, select the **Yes** option and then provide the rise type and dimensions
7. Click **Apply**.

   Note: The second tab of the dialog updates to reflect the **Structure Type** selection.

8. Select the second tab and enter the structure data:
   a. Type the structure dimensions.
   b. Select the structure cross-section type (where applicable).
   c. Select the orientation of members exposed to wind (where applicable).
   d. Select the shape of exposed structural members (where applicable).
   e. Type the **Structure Natural Frequency**.

      When the natural frequency is less than 1 Hz, the structure is considered “flexible” when calculating the gust effect factor, G. When the natural frequency is greater than or equal to 1 Hz, then the structure is considered “rigid” when calculating the gust effect factor.
   f. Type the **Structure Damping Ratio**.
   g. Select the Enclosure Classification (where applicable).
   h. (Optional) Select any combination of the **Use Kzt**, **Use Kd**, **Use Kz**, or **Use Ke** options to manually type values for these coefficients.

      The calculated coefficients are otherwise used.
9. Select the **Design Pressure** tab.
   The height vs. intensity data calculated for the current input is displayed here.
10. (Optional) You can check the options for Use G and Use Cf to manually type values for these coefficients.

    The calculated coefficients are otherwise used.
11. Click **OK**.
    The dialog closes and the wind intensity at height data is added to the table on the Intensity tab.

Note: The input fill will contain a section on ASCE 7 wind load generation data. Though this is not read directly by the STAAD analysis engine, these values are stored to allow you to make changes to the wind load parameters.

You can now proceed with the wind load definitions using the calculated values per ASCE-7.
Related Links

- *ASCE 7 Wind Load dialog box* (on page 2882)

- *TR.31.3 Definition of Wind Load* (on page 2435)
- *ASCE 7 Wind Load dialog box* (on page 2882)
- *Wind Load tab* (on page 2850)
- *Create Wind Type Definition dialog* (on page 2880)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Wind Load tab* (on page 2850)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *ASCE 7 Wind Load dialog box* (on page 2882)

M. To add a SNiP wind load definition

To add a parametric wind load definition per the SP 2.07.7 or SP 20.13330.2016 Loads and Actions building codes, use the following procedure.

Tip: Change units for convenience.

1. In the **Load & Definition** dialog, select the **Definitions** > **Wind Load** entry and then click **Add**.
   The **Add new: Wind Definition** dialog opens.
2. Select one of the following options from the **Code** drop-down list:
   - SNiP 2.07.7-85
   - SNiP 20.13330.2016

   Tip: You can also type optional **Comments** to further describe the wind definition (e.g., “Wind in X direction”).
3. Click **Add**.
   The definition type is added to the **Definitions** > **Wind Load** entry in the **Load & Definition** dialog.
4. In the **Load & Definition** dialog, select the new wind definition type and click the **Add**.
   The **Add New: Wind Definitions** dialog opens. The dialog now displays the **Intensity** and **Exposures** tabs.

   Note: The previously selected **Code** displays as a read-only selection.
5. Type the **Wind Pressure**.
6. Select the **Terrain** type from the drop-down list.
7. Select the **Region** from the drop-down list.
8. Type the **Delta** value.
9. Click **Add**.

Related Links

- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Wind Load tab* (on page 2850)
- *TR.30.1 Cut-Off Frequency, Mode Shapes, or Time* (on page 2344)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Create Wind Type Definition dialog* (on page 2880)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *Wind Load tab* (on page 2850)

M. To add a GB50009 wind load definition

To add a wind load definition and load cases for the Chinese GB50009-2012 code, use the following procedure.

These steps are performed when M. To add a wind load definition (on page 844) in the **Add New: Wind Definitions** dialog.

This procedure is used to generate a series of wind intensity vs. height values based on the procedure of the GB 50009-2012 code.

1. In the **Load & Definition** dialog, select the **Definitions** > **Wind Load** entry and then click **Add**.
   The **Add new: Wind Definition** dialog opens.
2. Select **GB 50009-2012** from the **Code** drop-down list.
3. Click **Add**.
   The definition type is added to the **Definitions** > **Wind Load** entry in the **Load & Definition** dialog.
4. In the **Load & Definition** dialog, select the new wind definition type and click the **Add**.
   The **Add New: Wind Definitions** dialog opens. The dialog now displays the **Intensity** and **Exposures** tabs.

   Note: The previously selected **Code** displays as a read-only selection.
5. Click **Generate**.
   The **Generate Wind Definition and Wind Load Case for Chinese GB 50009** dialog opens.
6. Enter the parameters to calculate the height variation factor for the wind pressure, μ_z:
   a. Type the **Building Height** and **Bottom Elevation**.

      Note: STAAD.Pro supports negative values for the Y coordinate used for wind load generation for the GB 50009-2012 code only.
   b. Select the method to sub-divide the height above ground into discreet points of the wind intensity curve.
      - **Segment Count to Divide Height (H) Equally -** This is the segment count to divide equally the total height (H) of the current building. This is used to calculate the individual height (z) where the wind pressure needs to be calculated.
      - **Equal Segment Length Along Height (H) -** This is the segment length, and the length is equal. The length is along the total height (H) of the current building. This is used to calculate the individual height (z) where the wind pressure needs to be calculated.
      - **Provide Special Height (z) -** This is the special height (z) list. This is the individual height (z) where the wind pressure needs to be calculated. The delimiter for the individual height (z) can be comma (,) or space, such as `3,6,9`.
   c. Select the **Roughness Type**.
   d. (Optional) Type a **Modification Factor**.
7. Select the options used to calculate the wind shape factor, μ_s:
   a. Select the **Shape Item ID** from the drop-down list:
      - 30 : Closed Polygon Building
      - 37 : Circular Section Structure (Chimney)
   b. For closed polygon buildings, select the **Secondary Shape Type** from the drop-down list.
   c. Click **Set Shape Factor** to specify shape factor parameters in the **Set Shape Factor** dialog.

      For closed polygon buildings, refer to the dialog diagrams for definitions of the faces. For circular structures, select the change in diameter and specify the diameter values accordingly.
   d. Type shape factor values for each structure face and wind direction.

   e. Click **OK**.
   f. (Optional) Type the **Interference Factor from Other Building** value if appropriate.
8. Specify the values for the reference wind pressure, $w0$:
   a. Type the **Reference Wind Pressure** value.
   b. (Optional) Type a **Modification Factor** value to use if appropriate.
9. (Optional) If the wind-induced vibration factor, β_z, along-wind is to be considered, do the following:
   a. Check the **Consider Along-wind Vibration Factor** option.
   b. Type the **Damping Ratio** to use.

      This should be greater than zero.
   c. Type the **Basic Natural Vibration Period (T1)** of the building.
   d. Select the appropriate **Structure Type**:
      - **High-rise Building -** The structure type of the current building is high-rise building. The high-rise building is the modern building which is very tall and has many levels or floors. The width of the windward face of the high-rise building is larger than that of the high-tower structure.
      - **High-tower Structure -** The structure type of the current structure is high-tower structure. The width of the windward face of the high-tower structure is far less than its height, and is also less than that of the high-rise building.
   e. Click **Provide Width of Windward Face** to specify the windward face dimensions based on the Structure Type selected.
10. Generate wind load cases for the load definition automatically:
    a. Select the **Wind Load Case** tab.
    b. Check the **Generate Wind Load Case** option.
    c. Enter the range values in each of the global axis directions to specify the range of the wind load.
    d. If the structure is open, then check the **Open** option.
11. Click **OK**.

    The Intensity vs. Height table is populated with the calculated values. If a negative value is used for the ground elevation, then the first row indicates this elevation only (i.e., the intensity value in this case is given as “0.000”).
12. Click **Add**.

    The wind definition and generated load cases are added to the model.

A series of four load cases are generated (one for each structure face), each with four load items for each of the four wind directions. The wind load generation data is stored within the STAAD input file.

```
DEFINE WIND LOAD
TYPE 1 WIND 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
GB50009-2012:PARAMS Version 3 Whole Wind_Group_ID 20210621162429414
Shape_Item "30 : -
CLOSED POLYGON BULDING" Second_Shape_UI "Rectangle" Interference_Factor 1 -
Shape_Factor Shape_Factor_Count 16 "LEFT Wind" "LEFT Face" 0.8 "LEFT Wind" -
"BACK Face" -0.7 "LEFT Wind" "RIGHT Face" -0.5 "LEFT Wind" "FRONT Face" -0.7 -
"RIGHT Wind" "LEFT Face" -0.5 "RIGHT Wind" "BACK Face" -0.7 "RIGHT Wind" -
"RIGHT Face" 0.8 "RIGHT Wind" "FRONT Face" -0.7 "FRONT Wind" "LEFT Face" -
-0.7 "FRONT Wind" "BACK Face" -0.5 "FRONT Wind" "RIGHT Face" -0.7 "FRONT -
Wind" "FRONT Face" 0.8 "BACK Wind" "LEFT Face" -0.7 "BACK Wind" "BACK Face" -
0.8 "BACK Wind" "RIGHT Face" -0.7 "BACK Wind" "FRONT Face" -0.5 -
Building_Height_H 9 Bottom_Elevation 0 z_Provide_Mothod 2 Segment_Count 5 -
Segment_Length 3 z_Special_List "4.5,9" Rough A -
Modify_Factor_of_Height_Factor 1 Province "" City_UI "" Refer_Wind_Press -
```

```
0.45 Modify_Factor_of_Press 1 Is_Calc_Vibration_Factor 0 Damp_Ratio 0.01 -
Basic_Period 1 Structure_Type 1 Windward_Width Width_Count 4 "LEFT Wind" -
Bottom_Width 0 Top_Width 0 "RIGHT Wind" Bottom_Width 0 Top_Width 0 "FRONT -
Wind" Bottom_Width 0 Top_Width 0 "BACK Wind" Bottom_Width 0 Top_Width 0 -
Generate_Load_Case 1 Select_Method 0 Face_Info Face_Count 4 "LEFT Face" -
Group "" Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 -
Is_Open 0 "BACK Face" Group "" Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max -
0 Z_Min 0 Z_Max 0 Is_Open 0 "RIGHT Face" Group "" Member_List "" X_Min 0 -
X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 Is_Open 0 "FRONT Face" Group "" -
Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 Is_Open 0 -
Build_Rotation_In_Model 0 Each_Type "LEFT Face" "LEFT Wind"
!> END GENERATED DATA BLOCK
INT 0.49 0.559 HEIG  4.5  9
LOAD 1 LOADTYPE Wind  TITLE WIND FROM LEFT (+X) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "LEFT Wind"
WIND LOAD X 0.8 TYPE 1
WIND LOAD -Z -0.7 TYPE 1
WIND LOAD -X 0.5 TYPE 1
WIND LOAD -Z 0.7 TYPE 1
LOAD 2 LOADTYPE Wind  TITLE WIND FROM RIGHT (-X) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "RIGHT Wind"
WIND LOAD -X -0.5 TYPE 1
WIND LOAD -Z -0.7 TYPE 1
WIND LOAD X -0.8 TYPE 1
WIND LOAD -Z 0.7 TYPE 1
LOAD 3 LOADTYPE Wind  TITLE WIND FROM FRONT (-Z) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "FRONT Wind"
WIND LOAD -X -0.7 TYPE 1
WIND LOAD -Z -0.5 TYPE 1
WIND LOAD -X 0.7 TYPE 1
WIND LOAD Z -0.8 TYPE 1
LOAD 4 LOADTYPE Wind  TITLE WIND FROM BACK (+Z) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "BACK Wind"
WIND LOAD -X -0.7 TYPE 1
WIND LOAD Z 0.8 TYPE 1
WIND LOAD -X 0.7 TYPE 1
WIND LOAD -Z 0.5 TYPE 1
```

You can use the same procedure to modify the stored wind load generation to update the wind definition and load cases.

Related Links

- *Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog* (on page 2887)
- *Create Wind Type Definition dialog* (on page 2880)
- *Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog* (on page 2887)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *Add New Wind Definitions (data) dialog* (on page 2880)

M. To add an IS-875 (Part 3): 2015 wind load

To add a wind load definition and load cases for the Indian IS-875 (Part 3): 2015 code, use the following procedure.

This procedure is used to generate a series of wind intensity vs. height values based on the procedure of the IS-875 (Part 3): 2015 code. The wind intensity will either be the net pressure coefficient or the force coefficient multiplied by the design wind pressure, based on the user selection.

Note: This procedure may be used for Rectangular Clad Buildings and Unclad Frame Buildings. The Load & Definition dialog can be used to also create definitions for Signs or Lattice Tower structures.

Notes: The following limitations pertain to generation of IS 875 (Part 3) : 2015 wind loads:

- The wind profile for the loading is calculated based on Y ordinate set to 0 rather than lowest point of the building. Thus, the topographical factor (k3) may not be relevant and should be applied with caution.
- Only Y-up models may be used (that is, the `SET Z UP` command may not be used).
- The resulting wind load intensity vs. height values generated for IS 875 (Part 3) : 2015 are in metric, regardless of the current input units.
- The dynamic effect of wind load is not considered.
- Cl. 6.3.2.4 and Annex B for changes in terrain categories (Table 3 Fetch and Developed Height Relationships) are not considered.
- For rectangular buildings, the frictional drag per Cl. 7.4.1 is not considered.
- For free standing walls and hoarding type structures, the check for oblique wind loads per Cl. 7.4.2.3 is not considered.
- For lattice tower structures, the corner wind effect on square lattice towers with flat-sided members per Cl. 7.4.3.5(b) is not considered.

1. On the **Loading** ribbon tab, select **Wind Load Generator** > **IS 875 (Part 3) : 2015** in the **Load Generation** group.

   The **Wind Load Generation - IS 875 (Part 3) : 2015** dialog opens.
2. Specify the **General Information** for the wind load:
   a. Select the **Type of Structure** from the drop-down list.
      - Rectangular Clad Building
      - Unclad Frame Building

      The parameters on the **Wind Load Data** tab update based on this selection.
   b. (Optional) Type the **Structure Height** and **Ground Level Elevation** values.
      The default values are taken as the maximum Y coordinate of the structure and zero, respectively.
   c. Type the **Height Interval for Intensity** to use for developing the height vs. intensity values for the wind load definition.
      The default value is 5 m.
   d. (Optional) Check the **Custom** option for the **Starting Loadcase No** and then type a load case number for the first generated wind load case.
      The default is the next load cases number after the highest load cased in the model.
3. Specify the **Wind Parameters** for the wind load:
   a. Specify the **Basic Wind Speed (Vb)** by either:

      selecting a city from the drop-down list

or

checking the **Custom** option and typing the basic wind speed in m/s

b. Specify the **Risk Factor Coefficient (k1)** by either:

   selecting the class of structure the drop-down list

   or

   selecting **Custom** from the drop-down list and typing the value for k1

c. Type the **Topography Factor (k3)** to use.

d. Specify the **Importance Factor (k4)** by either:

   selecting the importance category from the drop-down list

   or

   selecting **Custom** from the drop-down list and typing the value for k4

e. Specify the **Aerodynamic Roughness Height (z0i)** by either:

   selecting the **Terrain Category** from the drop-down list

   or

   selecting **Custom** from the drop-down list and typing the value for z0i. The values of the Height Factor (k2) may also be entered for each height interval in the wind intensity data table for the custom option.

f. Type the **Wind Directionality Factor (Kd)**.

g. Type the **Area Averaging Factor (Ka)**.

h. Type the **Combination Facotr (Kc)**.

The wind intensity table and graph update with the resulting values.

4. Select the **Wind Load Data** tab.
5. Enter the wind load data based on the selected structure type

   Rectangular Clad Buildings

Wind Direction	Select one of the global directions for the wind: +X, -X, +Z, or -Z.
Building Dimensions	By default, the dimensions are calculated based on the minimum and maximum global coordinates of the structure along the X, Z, and Y global axes, respectively. You may also type the **Length Along X (Lx)**, **Length Along Z (Lz)**, and **Height (H)** dimension values.
Use Coefficient Factor	Select the option for evaluating the coefficient factor: • **None -** no coefficient factor is multiplied with wind intensity value. • **Pressure Coefficient -** select this option to specify parameters to calculate the pressure coefficients on building face per Cl. 7.3.2 and 7.3.3. • **Force Coefficient -** select this option to directly specify the **Force Coefficient Factor (Cf)** on the building per Cl. 7.4.2.
Pressure Coefficient on Face	Use these options when the **Use Coefficient Factor** option selected is **Pressure Coefficient**: **Wind on Face**: Select the face of the structure (as labeled in the diagram in the dialog) to which this wind definition is applicable. **External Pressure Coefficient (Cpe)**: Select to use either the **Code Calculated** or **Custom** (manually typed) value of Cpe. If the maximum range of l/w from Tabe 5 of

	IS 875 (Part 3): 2015 is exceeded, then the upper values of Cpe for $3/2 < l/w < 4$ are still used. You may also enter custom Cpe values.
Internal Pressure Coefficient (Cpi)	Check this option to include the Internal Pressure Coefficient (Cpi).
Building Permeability	Select the range of openings to overall building exterior from the drop-down list. This is used to select the Cpi and Cpnet values per Clause 7.3.2.1 of IS-875. Alternatively, select **Custom** from the drop-down list to type a value to use for Cpi
Use (-) Cpi	Check this option to use negative Internal Pressure Coefficient (Cpi) value to calculate Net Pressure Coefficient (Cpnet)

Force Coefficient on Building	Type the Force Coefficient (Cf) value as per clause 7.4.2 of IS-875 when the **Use Coefficient Factor** option selected is **Force Coefficient**.

Unclad Frame Building

Frame Type	Select if the structure contains a **Single** frame or **Multiple** frames spaced along the wind direction. For either frame type, you must specify the **Solidity Ratio** (Φ) at each height interval of the structure in the frame information table. For multiple frames, you must also specify the **Effective Solidity Ratio** (Φ_e) at each height interval. Providing the **Solidity Ratio** (Φ) for a single frame at the given height interval, the force coefficient (Cf) will be selected according to the user provided solidity ratio values from Table 31 of IS 875 (Part 3) : 2015 by interpolation. Providing the **Effective Solidity Ratio** (Φ_e) for multiple frames at the given height interval, the shielding factor (H) will selected according to the user provided effective solidity ratio values from Table 32 of IS 875 (Part 3) : 2015 by interpolation.
Member Profile Cross Section	Select if the cross section profile of the frame(s) is **Flat Sided** or **Rounded**. For rounded members, you must also specify the **Diameter** of the members used at each height interval in the frame information table.
Frame Spacing Ratio	For **Multiple** frames, type the frame spacing ratio as indicated in Table 32 (Clause 7.4.3.4) of IS-875.
Force Coefficient	The intermediate values along with the calculated values are displayed in the table. If the **Custom** option is selected, then you must specify the Force Coefficient (**Cf**) values at each height interval. For multiple frames in the Custom option, you must also specify the **Shielding Factor** at each height interval in the table.

6. Select the method of assigning the wind loads to the model:

Type of Structure	Assignment method options
Rectangular Clad Building	There are four method of assignments for the generated load cases: • **Automatic -** The program will select faces A, B, C, and D. The list of members in this selection are displayed in the read-only **Member List** cell. • **Group -** An existing group name is used to define the members which make up the building wind faces. • **Member List -** Each face is defined by typing a list of members. • **Range -** Type the minimum and maximum values for the global coordinates (in the units shown) for each face. Check the **Exclude** option for an row to exclude assigning wind loads to this building face.
Unclad Frame Building	There are three method of assignments for the generated load cases: • **Group -** An existing group name is used to define the members which make up the frames. • **Member List -** Each frame is defined by typing a list of members. • **Range -** Type the minimum and maximum values for the global coordinates (in the units shown) for each frame. Check the **Exclude** option for an row to exclude assigning wind loads to this frame.

7. Click **Generate**.

For a rectangular clad building type, a series of four load definitions and eight load cases are generated, each with four load items for each of the four wind directions. The definitions contain the load vs design wind pressure values.

For an unclad frame type, the definitions include height vs intensity. For this case, the intensity generated is the force coefficient (Cf or Cf') multiplied by the design wind pressure (Pd).

The wind load generation data is stored within the STAAD input file.

You may edit the height vs design wind pressure values generated by clicking **Edit** in the **Load & Definition** dialog when one of the wind definition intensities is selected. After editing and regenerating the wind loads, all previous wind definitions and associated loads are deleted.

Related Links

- *Create Wind Type Definition dialog* (on page 2880)
- *IS-875 (Part 3): Wind Load dialog* (on page 2890)
- *Wind Load Generation - IS 875 (Part 3): 2015 dialog* (on page 2867)

M. To assign exposure for joints

To assign wind load definition exposure to joints, use the following procedure.

If your wind load definition has multiple exposure factors, you can specify the exact nodes for each factor this way.

This method assumes that you select the nodes first and then assign the exposure value to them. However, you can select the exposure method and then assign that to nodes using a different assignment method if more convenient.

Note: You can alternatively use a coordinate value range to assign exposure. These are applied in the load case.

1. Select the nodes which will all use a specific exposure ratio.
2. In the **Load & Definition** dialog, select that **Exposure** entry within a wind load definition.
3. Select the **Assign to Selected Nodes** method for assignment.
4. Click **Assign**.
   A message prompts you to confirm this assignment.
5. Click **Yes**.

The exposure ratio is assigned to these joints. When an exposure is selected, the assigned nodes are highlighted red in the view window.

You must now apply the wind load to the structure within a load case.

M. To apply a wind load

To apply a defined wind load to your structure, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Wind Load** > **Wind Load** tab.

   Note: If you have not previously added a wind load definition to the model, this tab is inactive. Refer to M. To add a wind load definition (on page 844) for details.
3. In the **Select Type** drop-down list, select the wind load definition for this load case.
4. Select the **Exposed Surface and Direction** of the load and then type the load **Factor** value.
5. (Optional) If the wind load definition is for a SNiP code, specify the additional SNiP Parameters:
   a. Select the configuration from the drop-down list.
   b. Type the wind pressure coefficient.
6. Specify the range of values over which the load applies. Type the along the global axis of how the wind load acts.

   Depending on the arrangement of walls, you may need to sub-divide the wind load application into multiple entries with overlapping ranges. The program may not automatically recognize all open-ended wind panels. Refer to Controlling Open-Ended Panel Identification (on page 2606) for additional details.

   Tip: Alternately, you may specify member lists or group definitions using the command input.
7. (Optional) Check the **Open Structure** option for truss, lattice, or other open structure types.

   This will apply the load only to the projected area of the members and joints, not to the bounded area of members or surfaces.
8. Click **Add**.

The program will automatically calculate the joint loads for structure that lies within the specified range. Selecting the wind load item in the **Load & Definition** dialog will highlight the loading members. The calculated surface exposure and the tributary areas will also be displayed for closed structures.

Review the wind panels for each wind load case to ensure that the intended areas are loaded. The panels are color coded to identify unique tributary areas.

M. To apply a dynamic wind load per SP 20.13330.2016

To apply a SP 20.13330.2016 wind definition as a dynamic load to your structure, use the following procedure.

A modal analysis and at least one static load cases are required in order to use a dynamic wind load case.

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2157) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the `WIND LOAD` command.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Wind Load** > **Wind Load - Dyanmic** tab.

   Note: If you have not previously added a SP 20.13330.2016 wind load definition to the model, this tab is inactive. Refer to M. To add a SNiP wind load definition (on page 847) for details.
3. In the **Select Type** drop-down list, select the wind load definition for this load case.

   Only SP 20.13330.2016 wind definitions are present in this list.
4. Type the **Width of building along wind dir.** and **Width of building across wind dir.** values.

   These are the width parallel and perpendicular to the wind direction, respectively.
5. Type the load **Factor** value.
6. (Optional) Check the **Allow All Mode Shapes** to include all mode shapes from the modal analysis.

   Otherwise, only the first mode shape will be used.
7. Select the **Static Wind Case** from the drop-down list of primary load cases that precede this load case in the model.
8. Select the direction in which the wind is applied from the graphical plan view.
9. Click **Add**.

Related Links

- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *Wind Load tab* (on page 2850)

M. Seismic Loads

This section describes how to add seismic loads to your STAAD.Pro model.

M. To add a seismic load definition

To add seismic load parameters per a building code, use the following procedure.

Static equivalent seismic loads are applied based on a set of parameters used to define the load. The parameters vary for each building code.

1. On the **Load & Definition** dialog, select **Definitions** > **Seismic Definitions** and the click **Add**.
   The **Add New: Seismic Definitions** dialog opens.
2. On the **Seismic Parameters** tab, select the building code from the **Type** drop-down list.
3. (Optional) Check the **Include Accidental Load** option to consider accidental torsion.

   Note: Not applicable for the Canadian NRC 1995 code.
4. (Optional) For the Indian IS 1893 - 2002/2005/2016 codes:
   a. (Optional) For the IS 1893 - 2002/2005 codes, check the **Include IS 1893 Part 4** option for industrial or stack-like structures.
   b. Click **Generate** to open the IS 1893 Seismic Parameters dialog, which provides a simpler method for entering the parameters with additional context for IS 1893 Part 1.
   c. Click **Generate**.
5. Type the seismic parameters in the table as required for the selected building code.
   A contextual note is displayed below the table for each parameter. Refer to TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2349) for detailed descriptions of the parameters for each code.
6. Click **Add**.
   If you have not provided all the required parameters, a message opens to alert you of missing items. Once the minimum required parameters have been provided, the seismic definition is added to the **Definitions** > **Seismic Definitions** entry in the **Load & Definition** dialog. The **Seismic Parameters** tab is removed from the **Add New: Seismic Parameters** dialog.
7. Click **Close**.

You will need to add weight items to the load definition. You can do this through a series of loads, a reference load case, or a combination of both.

For IS 1893 2016 seismic loads on buildings with either RC structural walls, you will also need to specify the first story wall area data required to calcualte the natural period of the structure.

Related Links

- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *TR.31.2.5 Chinese Static Seismic per GB50011-2001* (on page 2369)
- *Add New Seismic Definitions dialog* (on page 2895)

M. To add wall data area to an IS1893 2016 seismic definition

To add the first story wall area data for an IS 1892 2016 seismic load definition, use the following procedure.

You must create a reference load case containing the load items to be used as weights in the seismic load. Refer to M. To create a reference load (on page 893) for details.

The wall data area is only required for an IS 1893 2016 seismic load applied to a buiding of structure type 4 (i.e., reinforced concrete structural walls). This data is ignored for any other code or building type.

1. In the **Loads & Defintions** dialog, select the seismic defintion to which you want to add weights and then click **Add**.
   The **Add New: Seismic Definitions** dialog opens.
2. Select the **Wall Area** tab in the **Add New: Seismic Definitions** dialog.
3. For each wall in the global X direction in the first story above ground, type the Width and Length of the wall in plan.
4. For each wall in the global Z direction in the first story above ground, type the Width and Length of the wall in plan.
5. Click **Add**.
6. Either:

   To close the dialog, click **Close**.

   or

   To proceed to manually add weight data, select one of the weight types. See M. To add weight items to a seismic load definition (on page 858) for details.

   or

   To proceed to use a reference load for weight data, select the Reference Load tab. See M. To add weight by a reference load to a seismic load definition (on page 859) for details.

Related Links

- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load* (on page 2393)
- *Add New Seismic Definitions dialog* (on page 2895)
- *IS:1893 Seismic Parameters dialog box* (on page 2898)

M. To add weight items to a seismic load definition

To create structural weights for a seismic load definition, use the following procedure.

If you want to assign floor weights to a floor group, you must have at least one group of the type **Floor** previously defined to use this option.

Tip: If you want to use the same loads for both seismic weights and another load (e.g., the dead load case), you can M. To add weight by a reference load to a seismic load definition (on page 859) instead of this procedure.

1. In the **Loads & Defintions** dialog, select the seismic defintion to which you want to add weights and then click **Add**.
   The **Add New: Seismic Definitions** dialog opens.
2. Select the tab in the **Add New: Seismic Definitions** dialog corresponding to the weight you want to add.

3. Specify the weight parameters:
   For the following weight types, enter the data as indicated:

Weight type	Data to enter
Self	Type a **Self Weight Factor**
Joint	Type a **Joint Weight**
Member	Select the **Loading Type.** Type the **Weight** to use, the **Starting Distance**, and for uniform weights, the **Ending Distance**.
Element	Type a uniform **Pressure**

   For Floor Weights, do the following:
   a. Type a uniform floor **Pressure.**
   b. Select if the load is applied over a **Range** or a floor **Group**.
   c. (Optional) Check the **One Way Distribution** option to use that load distribution to supporting members.
      You can select the default behavior or select a specific member towards which the load is directed.
   d. For the **Range** option, specify Y range limits and optionally X and Z range limits.
   e. For the **Group** option, select a named group of members.
   f. (Optional) For the **Group** option, select the **Inclined Floor** option if the group includes members forming a panel that is inclined from the global XZ plane.

   Refer to TR.32.4.3 Floor Load Specification (on page 2483) for additional details on specifying floor loads.
4. Click **Add.**
5. Repeat steps 1 through 3 to add additional weights to your seismic load definition.
6. Click **Close.**

You must assign the weight items to the appropriate objects.

M. To add weight by a reference load to a seismic load definition

To use a previously defined reference load case for the weights in your seismic load definition, use the following procedure.

You must create a reference load case containing the load items to be used as weights in the seismic load. Refer to M. To create a reference load (on page 893) for details.

This approach is preferable if you need to use the same weight tables for multiple loads or analysis.

1. In the **Loads & Defintions** dialog, select the seismic defintion to which you want to add weights and then click **Add.**
   The **Add New: Seismic Definitions** dialog opens.
2. Select the **Reference Load** tab in the **Add New: Seismic Definitions** dialog.
3. Either:

   select a reference load definition and then click **>** to add it to the Reference Load item

   or

   click **>>** to add all the available reference load definition to the Reference Load item.
4. (Optional) Type a Factor to apply to each reference load definition in the Reference Load item as needed.
5. Select the global direction along which the load is applied from the **Along** drop-down list.
6. Click **Add.**

7. Click **Close**.

Related Links

- *TR.31.6 Defining Reference Load Types* (on page 2453)

M. To add a seismic load

To add a seismic load to a primary load case, use the following procedure.

Note: You must have at least one M. To add a seismic load definition (on page 857) in your model before you can add seismic loads.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Seismic Loads** tab.

   Note: If you do not have a seismic load definition created, this tab is disabled.
3. Select the global **Direction** in which this seismic load acts.
4. (Optional) Type the scale **Factor** to apply to the load.
5. (Optional) Check the option to use **Multiplying factor for Accidental Torsion Moment** and type a **Factor**.

   This value is the ratio of the total building width and is taken from the center of mass for each level.
6. (Optional) Check the option to use **Multiplying factor for Natural Torsion Moment** and type a **Factor**.

   Note: This is only applicable for structures modeled with a rigid diaphragm.

   This torsion -referred to as the natural torsion- accounts for the static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm
7. Click **Add**.

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)

M. Response Spectra

This section describes how to add response spectra to your STAAD.Pro model.

Tip: In STAAD.Pro, a response spectrum analysis is performed on the model when it contains response spectra load case. It is possible to use a linear, elastic analysis with a response spectrum load.

M. To add a generic response spectrum

To add a custom response spectrum (referred to as the “generic” method) to your model, use the following procedure.

Note: You must enter in spectra values using the dialog. To reference an external file containing spectra data, you must use the STAAD.Pro Editor.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **Custom** from the **Code** drop-down list.
4. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   Note: For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.
5. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.
6. Specify the spectrum data:
   a. Select if the **Spectrum Type** uses **Acceleration** or **Displacement** values.
      The spectrum data table updates accordingly.
   b. Type in the period and spectra data pairs in the table.
      As you add pairs of data, the response spectrum plot is generated to help review the input.
7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.
8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
10. (Optional) You can apply each of the following optional steps as needed:
    a. Select one of the **Signed Response Spectrum Results Options** if needed:

       check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

   If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

c. Type a **Scale** factor to apply to the spectra data.

d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

   If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.

Related Links

- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *Response Spectra tab* (on page 2854)

M. To add an IBC 2000 response spectrum

To specify a response spectrum for seismic loading per the IBC 20005 code, use the following procedure.

The period and acceleration or displacement values for an IBC 2000 response spectrum can be generated by the program to use with a “custom” response spectrum load.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **Custom** in the **Code** list.
4. Generate the IBC 2000 values for the table:

   Note: In order to use the generated spectrum values, you must leave the **Spectrum Type** selection as **Acceleration.**

   a. Click **Generate IBC Spectrum - 2000**.
      The **Spectrum Parameters** dialog opens.
   b. Specify the mapped acceleration values by either:

      select the **Zip** code from the drop-down list, click **Find Lat / Long**, and then click **Calculate S1 / SS**

      or

enter the **Latitude** and **Longitude** and then click **Calculate S1 / SS**

or

enter the **S1** and **SS** values directly

c. Select a **Site Class** for the location.
   The Fa and Fv values for the site location are displayed.
d. Type the **Start**, **End**, and **Interval** values for the period.
e. Click **Generate Spectrum**.
   The Generated Spectrum: IBC 2000 dialog opens. The values are tabulated and the spectrum is plotted. You can use the settings on this dialog to review the data.
f. Click **Close**.

The response spectrum values and plot is generated.

> **Note:** If you select

5. Select the **Combination Method** to use to combine the responses from each mode into a total response.

> **Note:** For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.

6. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.
7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.
8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
10. (Optional) You can apply each of the following optional steps as needed:
   a. Select one of the **Signed Response Spectrum Results Options** if needed:

      check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

      or

      check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.
   b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

      If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.
   c. Type a **Scale** factor to apply to the spectra data.
   d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.
   e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.
The new response spectrum is added to the currently selected load case.

Related Links

- *G.17.3.4 Response Spectrum* (on page 2163)
- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *Spectrum Parameters dialog* (on page 2862)
- *Generated Spectrum dialog* (on page 2862)

M. To add an IS 1893 response spectrum

To specify a response spectrum for seismic loading per the IS 1893 code, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **IS-1893** in the **Code** list.
4. Specify the IS 1893 specific parameters:
   a. Select a **Subsoil class** for the location.

Subsoil option	Description
Custom	You must enter Period and Acceleration or Displacement value pairs in the table, based on the **Interpolation Type** selected. You can also select the **Interpolation Type** for values between data points.
Hard, Medium, or Soft Soil	Average response acceleration coefficient is calculated. You will not need to select **Spectrum Type** or **Interpolation Type** for these options.

   b. (Optional) Select the **Ignore mode(s) with mass participation (IGN)** option and then enter a percentage value.

   Local and torsional modes whose mass participation is less than this percent value are then considered negligible and there fore will be excluded.

   The response spectrum plot is generated as these parameters are specified.
5. (Optional) Select the **Use Torsion** option and then enter the **Dynamic Eccentricity (DEC)** and **Accidental Eccentricity** (ECC) factors.

   The dynamic eccentricity factor is multiplied by the static eccentricity (i.e., the distance between the center of mass and center of rigidity) to get the dynamic eccentricity. This value should typically be 1.0 or higher.

The accidental eccentricity is used to calculate the value of accidental eccentricity. This value is typically 0.05 but may be increased to 0.10 for highly irregular buildings. This value can be positive or negative to accommodate clockwise or counter-clockwise rotation.

6. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   Note: For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.

7. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.

8. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.

9. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
10. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
11. (Optional) You can apply each of the following optional steps as needed:
    a. Select one of the **Signed Response Spectrum Results Options** if needed:

       check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

       check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

    b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

       If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

    c. Type a **Scale** factor to apply to the spectra data.
    d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.
    e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

       If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

12. Click **Add**.
    The new response spectrum is added to the currently selected load case.

Related Links

- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)

M. To add an IBC 2006 response spectrum

To specify a response spectrum for seismic loading per the IBC 2006 / ASCE 7-05 codes, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   > **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **IBC 2006/ASCE 7-05** in the **Code** list.
4. Specify the IBC 2006 / ASCE 7-05 specific parameters:
   a. Specify the mapped acceleration values by either:

      typing the **Zip** code (location and values are populated)

      or

      delete the Zip code value and type **Latitude** and **Longitude** (values are populated)

      or

      delete the Latitude value and type the **Ss** and **S1** values directly
   b. Type the **Long Period (TL)** value for the structure.
   c. Type the **Fa** and **Fv** values for the soil conditions at the location.
   d. Select a **Site class (SCL)** for the location.

   The response spectrum plot is generated as these parameters are specified.
5. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   > **Note:** For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.
6. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.
7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.
8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
10. (Optional) You can apply each of the following optional steps as needed:

a. Select one of the **Signed Response Spectrum Results Options** if needed:

   check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

   or

   check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.

b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

   If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.

c. Type a **Scale** factor to apply to the spectra data.

d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.

e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

   If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.

11. Click **Add**.
    The new response spectrum is added to the currently selected load case.

Related Links

- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)

M. To add an EC8 response spectrum

Floor height is determined by the program as a joint where one or more beams frame in a column.

Eurocode 8 requires the use of cracked section stiffness when considering concrete buildings, which however is lacking in the current analysis engine. This can be overcome by using a section reduction factor as suggested in the code. STAAD.Pro has a “section reduction factor” that can be used for this purpose (i.e., the `MEMBER CRACKED` command). Refer to TR.20.10 Member Property Reduction Factors (on page 2282) for details on this command.

Note: Refer to TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2525) for additional information on using the response spectrum specifications per Eurocode 8.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **EURO (EC8) - 2004** in the **Code** list.

4. Specify the EC8-2004 specific parameters:
   a. In the Spectrum table group, type the **Design Ground Acc**(eleration) factor.
   b. Type the **Behaviour Factor**.
   c. Select the **Subsoil Class** from the drop-down list.
   d. In the **Load Type** group, select if the load is in the **Elastic** or **Design** range and then select if the load is **Type I** or **Type II**.

   The response spectrum plot is generated as these parameters are specified.
5. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   Note: For the **ASCE** method, there are two optional parameters, **FF1** and **FF2**, which can be checked and specified.
6. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.
7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.
8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
10. (Optional) You can apply each of the following optional steps as needed:
    a. Select one of the **Signed Response Spectrum Results Options** if needed:

       check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

       check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.
    b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

       If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.
    c. Type a **Scale** factor to apply to the spectra data.
    d. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.
    e. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

       If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.
11. Click **Add**.

    The new response spectrum is added to the currently selected load case.

This response spectrum may be used with a linear, elastic analysis, geometric nonlinear analysis, or a pushover analysis for review of Eurocode 8 seismic requirements in P. Earthquake workflow (on page 2032).

Related Links

- *TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004* (on page 2525)
- *P. Using the Earthquake Workflow* (on page 2032)

M. To add a GB 50011-2010 response spectrum

To specify a response spectrum for seismic loading per the Chinese GB 50011 2010 (2016 edition) code, use the following procedure.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Response Spectrum** tab.
3. Select **Chinese GB 50011 - 2010** in the **Code** list.
4. Select the **Combination Method** to use to combine the responses from each mode into a total response.

   The **CQC** and **SRSS** methods are available for the GB 50011-2010 response spectra.
5. (Optional) Check the **Save** option to create a file contain the joint accelerations in ratios of gravity and in radians/sec^2.

   The file name is saved in the same location as the STAAD input file, will have the same root filename as the STAAD input file but with an `.acc` file extension.
6. Specify the GB 50011-2010 specific parameters:
   a. Select a **Fortification Intensity**, **Seismic Frequency**, and **Site Class (SCL)** for the location.
   b. Select the **Seismic Group** appropriate for the structure.

   The resulting **Max. Horizontal Influence Factor** and building **Period (Tg)** are displayed for the selected spectrum table input.

   The response spectrum plot is generated as these parameters are specified.
7. Select if the **Interpolation Type** is **Linear** or **Logarithmic**.

   The logarithmic option is recommended when only a few spectra data points are entered, as spectra vs period curves are typically only linear on a log-log scale.
8. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
9. Select the directions which spectrum is applied and for each, type a factor (0-1.0).
10. (Optional) Select the **Use Torsion** option and then enter the **Dynamic Eccentricity (DEC)** and **Accidental Eccentricity** (ECC) factors.

    The dynamic eccentricity factor is multiplied by the static eccentricity (i.e., the distance between the center of mass and center of rigidity) to get the dynamic eccentricity. This value should typically be 1.0 or higher.

The accidental eccentricity is used to calculate the value of accidental eccentricity. This value is typically 0.05 but may be increased to 0.10 for highly irregular buildings. This value can be positive or negative to accommodate clockwise or counter-clockwise rotation.

11. You can apply each of the following optional steps as needed:
    a. Select one of the **Signed Response Spectrum Results Options** if needed:

       check **Dominant** and type a **Mode No** to use for determine the signs of all modes. You can alternatively type 0 for the Mode No to have the program select the mode with the greatest percent participation in the excitation direction for the dominant mode.

       or

       check **Sign** to create signed values for all results by comparing the sum of the squares of positive versus negative results.
    b. To generate individual modal response load cases, check the **Generate load case(s) for first** option and then type the number of load cases to generate. You can also type the Load Case number to use for the first of the generated load cases.

       If the number of load cases requested is larger than the number of modes extracted, then only the number of modes extracted is used.
    c. Check the **Missing Mass** option to include the static effect of masses not represented in the modes. Type an optional spectral acceleration for this missing mass mode.
    d. If the Missing Mass option is used and you want to specify an acceleration corresponding to a frequency instead, check the **ZPA** option and type a frequency.

       If neither the missing mass acceleration value or ZPA frequency is specified, then the spectral acceleration at 33Hz is used to calculate the missing mass mode.
12. Click **Add**.
    The new response spectrum is added to the currently selected load case.

Related Links

- *TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010* (on page 2531)
- *Response Spectra tab* (on page 2854)
- *Generated Spectrum dialog* (on page 2862)
- *Spectrum Parameters dialog* (on page 2862)

- *TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010* (on page 2531)
- *Response Spectra tab* (on page 2854)
- *Generated Spectrum dialog* (on page 2862)
- *Spectrum Parameters dialog* (on page 2862)

M. Snow Loads

This section describes how to add snow loads to your STAAD.Pro model.

M. To add an ASCE 7-02 snow load

Loads and definitions are added from the **Load & Definition** dialog in the **Loading** page.

Snow load definitions are applied to floor groups. You must have at least group of the type **Floor** previously defined.

Tip: You may want to change your input units to make entering snow loads convenient.

1. In the **Load & Definition** dialog, select the **Definitions** > **Snow Definition** entry and click then **Add**.
   The **Add New: Snow Definition** dialog opens.
2. Enter the general snow parameters and click the **Add** button.

   The newly created snow load definition will appear in the **Definitions** > **Snow Definition** section in the **Load & Definition** dialog.
3. Either:

   Repeat step 2 to add a second snow load definition.

   or

   Click **Close** to stop adding snow load definitions.
4. (Optional) Create a new primary load case for use with the snow load.
5. In the **Load & Definition** dialog, select the primary load case which will include the snow load.
6. Click **Add**.
   The **Add New: Load Items** dialog opens.
7. Select the **Snow Load** tab.

   Note: If no floor groups have been defined, this page will be inactive and a warning message is displayed in the dialog.
8. Select the **Floor Group** and select the Snow Load parameters.
9. Click **Add**.
   The snow load is added to the primary load case.

Related Links

- *Add New Snow Definition dialog* (on page 2894)
- *TR.31.5 Definition of Snow Load* (on page 2452)
- *G.16.4 Snow Load* (on page 2138)
- *Snow Load tab* (on page 2854)
- *TR.32.12.4 Generation of Snow Loads* (on page 2609)

M. Notional Loads

Notional loads are nominal lateral loads used in a direct analysis, such as specified in AISC 360.

M. To define direct analysis parameters

To define the member parameters used in an AISC 360 direct analysis, use the following procedure.

Note: The yield strength of the steel is taken from the TR.26.1 Define Material (on page 2303). If the material definition is not provided for a member, then a default value of F_y = *36 ksi* is used.

1. On the **Loading** ribbon tab, select the **Direct Analysis** tool in the **Define Load Systems** group.
   The **Create New Load Items** dialog open.
2. Specify the initial τ_b value:

Members whose flexural stiffness are considered to contribute to the lateral stiffness of the structure will have their flexural stiffness (EI) reduced by $0.8 \times \tau_b$.

a. Select the **FLEX Parameter** tab.
b. Type a **FLEX** value.
c. Click **Add**.

3. (Optional) Identify any axial members which contribute to the lateral stiffness of the structure:
   a. Select the **AXIAL Parameter** tab.
   b. Click **Add**.
4. Click **Close**.
5. In the **Load & Definition** dialog, expand the **Definitions** > **Direct Analysis Definition** group.
6. Assign each of the `FLEX` and `AXIAL` parameters listed to the corresponding members in the model.

You must add one or more load cases with notional loads to your model for a direct analysis.
Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *TR.31.7 Definition of Direct Analysis Members* (on page 2454)
- *Add New Direct Analysis Definition dialog* (on page 2899)

M. To add a notional load case

To add notional loads to a load case for use with direct analysis, use the following procedure.

Notional loads are lateral loads which are defined as a small percentage of gravity loads. Therefore, you must define your gravity loads in a separate reference load case (recommended) or primary load case before adding notional loads.

The notional loads are calculated as lateral loads applied to the joints in the selected global direct.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
2. Select the **Repeat Load** > **Notional Load** tab.

   Two load lists are displayed. The first is a list of primary load cases which occur *before* the current primary load case and the second is a list of reference load definitions.
3. Either:

   select a load case or reference load case on the right and click **>** to add it to the notional load

   or

   click either **>>** to add either all of the load cases or reference loads to the notional load
4. Type a **Factor** to use for the ratio of lateral to gravity loads for each load case added to the notional load.

   Typically, codes recommend 0.2 to 0.3 % (0.002 to 0.003). The default value will be the Notional Load Factor specified in the Direct Analysis Definition.

5. Select the global **Direction** for each load case added to the notional load.
6. Click **Add**.

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *TR.32.13 Notional Loads* (on page 2610)
- *Repeat Load tab* (on page 2863)

M. Moving Loads

This section describes how to generate basic moving loads to your STAAD.Pro model.

Moving loads generated by the moving load generator in STAAD.Pro are only applied to beams. Plates and surfaces cannot be loaded using these loads.

For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the M. Bridge Deck workflow (on page 908) is recommended is recommended. The Bridge Deck workflow works on the influence line/influence surface method, and is considerably superior to the moving load generator. It also has the advantage of being able to calculate the critical load positions on decks modeled using plate elements.

Related Links

- *(Moving) Load Generation Type dialog* (on page 2832)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)
- *M. To add vehicles to the load generation* (on page 874)
- *TR.31.1 Definition of Moving Load System* (on page 2346)
- *V. Moving Load Generator* (on page 3597)

M. To define a vehicle for loading

In order to generate a set of static loads, due to the movement of the vehicle or load on the structure, there are multiple steps involved.

Note: This procedure is used to generate a set of static loads at intervals on a structure. The M. Bridge Deck workflow (on page 908) can also be used to automatically place loads at locations to achieve maximum or minimum actions on a bridge deck.

1. Either:

   on the **Loading** ribbon tab, select the **Moving Loads** tool in the **Loading** group

   or

   Select **Vehicle Definitions** under **Definitions** in the **Load & Definition** dialog and then click **Add**.

   The **Add New Vehicle Definition** dialog opens.
2. Specify a **Vehicle Type Ref** number.

   This identification number is used to refer to this vehicle load definition in the moving load generator.

3. Either:

   define a custom vehicle using the **Define load** tab

   or

   select a standard AASHTO vehicle definition using the **AASHTO Spec** tab

   or

   select an external file containing vehicle data using the **File Input** tab
4. Click **Add**.

Related Links

- *G.16.1 Moving Load Generator* (on page 2134)
- *Add New Vehicle Definitions dialog* (on page 2899)
- *TR.31.1 Definition of Moving Load System* (on page 2346)

M. To generate moving load cases

To create a generation of load cases, use the following procedure.

1. In the **Load & Definition** dialog, select the **Load Case Details** entry and then click **Add**.
   The **Add New: Load Cases** dialog opens.
2. Select the **Load Generation** tab.
3. Type the **No. of Loads** to be generated.

   This is the total number of load positions that will be generated. You will later define the starting location and the increment distance between each position.
4. (Optional) Select the **Predefined Loads to be Added** from the drop-down list.
5. Click **Add**.

The moving load generation case is added to the **Load Case Details** entry in the **Load & Definition** dialog. These cases are marked with a **[G]** to indicate that they are generations of load cases. The load case number range is given in the title.

Related Links

- *G.16.1 Moving Load Generator* (on page 2134)
- *Load Generation dialog* (on page 2866)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)

M. To add vehicles to the load generation

To add a vehicle definition to the moving load generation, use the following procedure.

1. Select a moving load generation case in the **Load Case Details** entry in the **Load & Definition** dialog and then click **Add**.
   The **Add New: Load Cases** dialog opens.
2. Select the vehicle definition in the **Type** drop-down list.
3. Type the coordinates for the **Initial Position of the Load**.

   This is where the reference load (i.e., the first specified concentrated load in the moving load system) is placed on the structure for the first load case in the generation.
4. Type the **Load Increment** values in the planar directions to move the load for each subsequent load case.

5. Click **Add**.
6. (Optional) Repeat steps 2 through 5 to add additional moving loads to the load generation case.
7. Click **Close**.

Related Links

- *M. Moving Loads* (on page 873)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 6398)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 6684)
- *(Moving) Load Generation Type dialog* (on page 2832)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)

M. Time History Loads

This section describes how to add time history loads to your STAAD.Pro model.

M. To define a time history type from tabular data

To use a set of time value pairs to define a time history load, use the following procedure.

1. On the **Loading** ribbon tab, select the **Time History** > **Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.
2. Type an **Integration Time Step** value in seconds.
3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.
4. (Optional) Select the **Save** option to generate a file with time history data.
5. Select the **Loading Type** for your data:

   Acceleration

   or

   Force

   or

   Moment
6. Type the **Define Time vs** load data values in the table.
   As you add data pairs, the chart below is populated.
7. Click **Add**.

Note: You must define time history parameters to complete the time history load definition.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *TR.31.4 Definition of Time History Load* (on page 2441)

- *Add New Time History Definitions dialog* (on page 2907)

M. To define a time history type from a function

To use a parametric function to define a time history load, use the following procedure.

1. On the **Loading** ribbon tab, select the **Time History** > **Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.
2. Type an **Integration Time Step** value in seconds.
3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.
4. (Optional) Select the **Save** option to generate a file with time history data.
5. Select the **Loading Type** for your data:

   Acceleration

   or

   Force

   or

   Moment
6. Select the **Harmonic Function** option in the Function Options group.
7. Select the harmonic function to use: **SINE** or **COSINE.**
8. Select one of the options for specifying wave length:

   select **Frequency** and enter a cyclic frequency in cycles/sec.

   or

   select **RPM** and enter the revolutions per minute
9. Specify the harmonic wave parameters:
   a. Enter the **Amplitude** in the units indicated.
      These correspond to the current input units and the **Loading Type** selection.
   b. (Optional) Enter the **Phase** angle in degrees.
   c. Enter the number of **Cycles** of loading.
10. Specify the number of time steps per cycle by either:

    select the **Step** option and enter the time step of the loading.

    or

    select the **SubDiv** option and enter a number of subdivisions per quarter cycle (i.e., from zero to peak amplitude along the sine or cosine wave).
11. Click **Add.**

Note: You must define time history parameters to complete the time history load definition.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *Add New Time History Definitions dialog* (on page 2907)

M. To define a time history type by spectrum

To define a time history type using spectral parameters, use the following procedure.

1. On the **Loading** ribbon tab, select the **Time History** > **Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.
2. Type an **Integration Time Step** value in seconds.
3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.
4. (Optional) Select the **Save** option to generate a file with time history data.
5. Select the **Loading Type** for your data:

   Acceleration

   or

   Force

   or

   Moment
6. Select the **Spectrum** option in the Function Options group.
7. Specify the time data:
   a. Enter the maximum time (in second) in the generated time history in the **Tmax** field.
   b. Enter the time step (in seconds) in the **DeltaT** field.
   c. Enter the time at the end of rising acceleration in the **T1** field.
   d. Enter the time at the end of the steady acceleration in the **T2** field.
   e. Enter the time at the end of the acceleration decay in the **T3** field.

   Note: The value of **T1** must be greater than zero. Further, **T3** > **T2** > **T1** and **Tmax** must be greater than **T3**.
8. Enter the damping ratio in the **Damp** field.
9. Specify the options:
   a. (Optional) Type a **Random Seed** value.

      Enter a positive integer (in the range of 1 to 2,147,483,647) to be used as a unique random number generation "seed." A unique time history will be produced for each seed value. Change this value when you want to produce a "different (from the time history generated with the prior seed value)" but statistically equivalent time history.
   b. Enter the **No of digitized Freq**.

      This is the number of equally spaced frequencies at which the input shock spectrum is re-digitized (by interpolation).
   c. Enter the **No. of Iteration** which will be used to perfect the computed time history.
10. Click **Add**.

For response spectra time history, there are some additional output parameters that must be input via the STAAD.Pro Editor if used.

Note: You must define time history parameters to complete the time history load definition.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *Add New Time History Definitions dialog* (on page 2907)

M. To generate output for time history spectrum

To generate some additional output data for spectrum input for a time history definition, use the following procedure.

Create a structure with a time history definition using the Spectrum option.

These optional commands cannot be entered via the graphical user interface. They are required to be added using the STAAD.Pro Editor.

1. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

   The STAAD.Pro Editor window opens with the current STAAD input file.
2. (Optional) To output time history data: after the`DEFINE TIME HISTORY` command, in the`SPECTRUM` options, add the command `THPRINT` *`f18`* .

   Where *f18* directs the program to output either the beginning and last 54 data points (*f18* = 1) or the entire curve (*f18* = 2) or a select number of beginning and last data points (*f18* ≥ 10).

   The spectrum input parameters will be included in the STAAD Output (`.anl`) file along with the Time History Output (limited to the *f18* value specified).
3. (Optional) To generate spectrum output for a time history: after the`DEFINE TIME HISTORY` command, in the`SPECTRUM` options, add the command `SPRINT` *`f19`* .

   Where *f19* represents an integer value after the `SPRINT` command to instruct the program to only output the beginning and last number of values equal to this integer.

   A summary of the Spectrum input and the curve points will be included in the STAAD Output (`.anl`) file.
4. Save the command input file and then exit the STAAD.Pro Editor.

Related Links

- *TR.31.4 Definition of Time History Load* (on page 2441)

M. To use frequency-spectra pairs in a time history load

To instruct the program to use frequency-spectra pairs in lieu of period-spectra pairs for the time history spectrum input, use the following procedure.

Create a structure with a time history definition using the Spectrum option.

These optional commands cannot be entered via the graphical user interface. They are required to be added using the STAAD.Pro Editor.

1. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

   The STAAD.Pro Editor window opens with the current STAAD input file.
2. After the `DEFINE TIME HISOTRY` command, in the `SPECTRUM` options, add the command `FREQ`.
3. Save the command input file and then exit the STAAD.Pro Editor.

Related Links

- *TR.31.4 Definition of Time History Load* (on page 2441)

M. To define a time history type by external file

To define a time history type using data in an external text file, use the following procedure.

You must have a separate text file containing pairs of times and values (acceleration, force, or moment). The file extension is not important, but the values must be in plain text.

1. On the **Loading** ribbon tab, select the **Time History** > **Forcing Function** tool in the **Dynamic Specifications** group.

   The **Add New: Time History Definitions** dialog opens.
2. Type an **Integration Time Step** value in seconds.
3. (Optional) Select the **Consider Missing Mass Mode** option if necessary.
4. (Optional) Select the **Save** option to generate a file with time history data.
5. Select the **Loading Type** for your data:

   Acceleration

   or

   Force

   or

   Moment
6. Select the **From External File** option in the Function Options group.
7. Type the **File Name** (with or without file extension).
8. Click **Add**.

An example of an external data file:

```
0.0 1.0 1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6
```

If you need to specify a delta time spacing used for the external file data, this must be done using the STAAD.Pro Editor.

Note: You must define time history parameters to complete the time history load definition.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *Add New Time History Definitions dialog* (on page 2907)

M. To define time history parameters

To define arrival times and damping used for time history loads, use the following procedure.

You must first define a time history type by one of the previously described methods.

One set of time history parameters can defined for your model for all time history load types.

1. On the Loading ribbon tab, select the **Time History** > **Parameters** tool in the Dynamic Specifications group.

   The **Define Param** dialog opens.
2. (Optional) Check the **Time Step** option and type the time step value.

   This is the solution time step used in step-by-step integration of the uncoupled equations. If not selected, the default value of 0.0013888 seconds is used.
3. Select the **Damping Type** to use and, if explicit **Damping** is selected, type the damping ratio to be used for all modes.
4. Type in one or more **Arrival Times** for the dynamic loads.

   Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion).
5. Click **Add**.

The time history load parameters are added to the **Time History Definitions** in the **Load & Definition** dialog.

In order to use a time history load definition, you must add a time history load item to a primary load case.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *Define (Time History) Parameters dialog* (on page 2909)
- *TR.31.4 Definition of Time History Load* (on page 2441)

M. To add a time history load

To add a time history load to the current primary load case, use the following procedure.

Select the current load case in the program status bar.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

> **Tip:** This will add the load item to the currently selected load group selected in the program status bar.

or

In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**. The **Add New Load Items** dialog opens.

2. Select the **Time History** tab.
3. Select the **Load Type**:

   Time Load for a forcing function that applies forces or moments to joints

   or

   Ground Motion is applied global to all supports as an acceleration type
4. Select Direction in which the load is applied.

   Time Load types are applied in FX, FY, or FZ for forces and MX, MY, or MZ for moments. Ground motion types are applies in one of the three global directions.
5. Select the arrival time for this load from the arrival times in the time history definition parameters.
6. Select the load definition type from the Defined Types drop-down list.

   The list includes the definition number and function type.
7. (Optional) Select the results type from the Response Types drop-down list.
8. (Optional) Type a Force Amplitude Factor to multiply for the forces, moments, or accelerations at the joints.
9. Click **Add**.

You must assign the time history load to the joints to which it applies.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *Time History tab* (on page 2850)
- *TR.32.10.2 Time Varying Load for Response History Analysis* (on page 2593)

M. Pushover Loads

This section details create a pushover loads for use with a pushover analysis in STAAD.Pro.

> **Note:** An advanced analysis module license is required to use a pushover analysis.

Related Links

- *Perform Pushover Analysis tab* (on page 2926)
- *A. To specify a pushover analysis* (on page 957)

M. To define general pushover data

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. On the Define Input tab (on page 2900), select the **General Input Parameters** option.
3. (Optional) Select the **Type of Frame**
4. (Optional) Select **Include Effect Geometric Non-Linearity Effect**

Related Links

- *Define Input tab* (on page 2900)

- *G.17.4.2.1.1 Define Steel Moment and Braced Frames* (on page 2178)
- *TR.37.7.2.1 Type of Frame* (on page 2643)
- *Define Input tab* (on page 2900)
- *TR.37.7.2.4 Consideration of Geometric Nonlinearity Effect* (on page 2645)
- *Define Input tab* (on page 2900)
- *TR.37.7.2.5 KG Matrix Iteration* (on page 2645)
- *Define Input tab* (on page 2900)
- *TR.37.7.2.6 Maximum number of Analysis cycle* (on page 2646)
- *Define Input tab* (on page 2900)
- *TR.37.7.2.8 Save Output Results for Multiple Steps* (on page 2647)
- *Define Input tab* (on page 2900)
- *TR.37.7.2.7 Print Output Result* (on page 2646)
- *Define Input tab* (on page 2900)
- *TR.37.7.7 End Pushover Data* (on page 2653)

M. To define member specific pushover data

Use the following optional procedure to specify parameters for individual members.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. On the **Define Input** tab, select the **Member Specific Parameters** option.
3. (Optional) Set the **Expected Yield Stress**
4. (Optional) Set the **Effective Length Factor for Member (Y/Z)**
5. Click **Add.**
6. Once any other parameters have been added, click **Close**.
   Entries for the member parameters are added in the **Loads & Definitions** dialog in the **Definitions** > **Pushover Definitions** section.
7. Select one of the member parameters and click **Assign.**

> **Note:** Refer to Graphical Environment help for using the various assignment methods in the Loads & Definitions dialog.

Related Links

- *Define Input tab* (on page 2900)
- *TR.37.7.2.2 Expected Yield Stress* (on page 2644)
- *TR.37.7.2.3 Effective Length Factor of Member* (on page 2644)
- *Define Input tab* (on page 2900)

M. To manually define and assign hinges

Use the following optional procedure to define parameters and assign hinges manually.

> **Note:** If the pushover analysis will make use of the built-in FEMA hinge properties for all members,

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.

2. Select the **Define Hinge Property** tab.
3. Select the **Hinge Type**.

   Types **FEMA** or **Ignore** do not require any additional parameters.
4. (Optional) For user-defined hinge types only:
   a. Select an existing **Type ID** or select **New Type** to define a new hinge type.
   b. Specify a unique integer for the **Type Identifier**.
   c. Specify coordinate values for points C, D, and E in the **Load Deformation Curve Points** and IO, LS, and CP values in the **Acceptance Criteria**.

      These values are can be determined using Tables 5-6 and 5-7 in FEMA 356.
   d. Specify values for **Yield Moment (YR)** and **Yield Rotation (YR)**.
      These values are calculated per Section 5.5.2.2.2 of FEMA 365 (G.17.4.1.6 Frame element hinge properties (on page 2174)).
5. Click **Add**.
6. Once any other parameters have been added, click **Close**.
   Entries for the hinge types are added in the **Loads & Definitions** dialog in the **Definitions** > **Pushover Definitions** section.
7. Select one of the hinge types and click **Assign**.

> **Note:** Refer to Graphical Environment help for using the various assignment methods in the Loads & Definitions dialog.

Related Links

- *Define Hinge Property tab* (on page 2904)
- *G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria* (on page 2180)
- *G.17.4.1.4 Types of Nonlinearity* (on page 2173)
- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *TR.37.7.5.1 User-Defined Hinge Property* (on page 2650)
- *TR.37.7.5.2 Assignment of Hinge Property to the Members* (on page 2651)
- *Define Hinge Property tab* (on page 2904)

M. To define pushover spectral data

Use the following procedure to define the seismic hazard per FEMA 356.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Spectral Details** tab.
3. Specify up to four **Critical Damping** values for the 1st through 4th spectra.

   At least one is required, which has a default value of 5.0%.
4. (Optional) Select the **Site Category**

   Site Class D is used by default.
5. Specify the Mapped Spectral Acceleration **At Short Period**, S_S, to be used per Table 1-4 of FEMA 356.
6. Specify the Mapped Spectral Acceleration **At One-Second Period**, S_1, to be used per Table 1-5 of FEMA 356.
7. Click **Add**.

Related Links

- *G.17.4.2.1.7 Define Input for Demand Spectrum* (on page 2182)

- *TR.37.7.6 Define Spectral Parameters* (on page 2652)
- *Define Spectrum Details tab* (on page 2904)

M. To add a pushover loading

Use the following procedure to define the pushover loading to be used in the pushover analysis.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Loading Pattern** tab.
3. (Optional) Select **User Defined**
   If the default Auto load pattern is used, the program will internally compute the gravity loads.
4. (Optional) Set the **Method for Lateral Load Calculation**
   TR.37.7.3 Define Loading Pattern (on page 2647) for additional information on each method.
5. (Optional) Set the **Total Base Shear to be Distributed Direction Total Base Shear**
6. (Optional) Specify a **Number of Push Load Steps**
7. Click **Add**.

> **Tip:** Only one pushover loading definition may be used for a model. If a different pushover loading definition is required, either change the individual parameters or delete them and create a new definition.

Related Links

- *TR.37.7.3.2 Total Base Shear to be Distributed* (on page 2649)
- *Define Loading Pattern tab* (on page 2902)
- *TR.37.7.3.3 Number of Push Load Steps* (on page 2649)
- *Define Loading Pattern tab* (on page 2902)
- *TR.37.7.3.1 Program Defined Push Load Distribution Pattern* (on page 2648)
- *Define Loading Pattern tab* (on page 2902)
- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2179)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2177)
- *TR.37.7.3 Define Loading Pattern* (on page 2647)
- *TR.37.7.8 Pushover Loading Input* (on page 2653)
- *Define Loading Pattern tab* (on page 2902)

M. To define solution control

Use the following procedure to define at least one solution control method for the pushover analysis.

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. Select the **Define Solution Control** tab.
3. (Optional) Set the **Push Up to Defined Base Shear Direction Defined Base Shear**
4. (Optional) Set the **Push Up to Defined Displacement at Control Joint Direction Joint Displacement Value Joint Number**
5. Click **Add**.

Related Links

- *G.17.4.2.1.6 Define Pushover Analysis Solution Control* (on page 2182)
- *TR.37.7.4 Define Solution Control* (on page 2649)
- *Define Solution Control tab* (on page 2906)

M. To use starting vectors with load-dependent Ritz vectors

You must set the eignen method to use load-dependent Ritz (LDR) vectors in order to specify the starting vectors for that method. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Set Eigen Method** tool in the **Analysis Data** group. Then select the **Load Dependent Ritz Vectors (LDR)** option and click **OK**.

Note: This method is used to specify starting mass loads only. In order to specify starting load from reference load cases, you must directly edit the STAAD input file.

1. Select the **Loading** page in the **Analytical Model** workflow.
2. Select **Starting Load Definition** under **Definitions** in the **Load & Definition** dialog and then click **Add**. The **Add New Define Starting Mass Load** dialog opens.
3. Select the directions you want to use for the starting mass load vectors.
4. (Optional) For each direction selected, enter the number of vectors to be extracted corresponding to that load.
5. Click **OK**.

Related Links

- *TR.31.9 Defining Starting Load* (on page 2459)
- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *Add New Define Starting Mass Load dialog* (on page 2833)
- *TR.31.9 Defining Starting Load* (on page 2459)

M. Load Combinations

This section describes how to add load combinations to your STAAD.Pro results.

Note: Load combinations of primary load cases are algebraic combination of analysis results. Therefore, they are not applicable to nonlinear analyses.

Load Combinations with Loads from STAAD.Pro Physical Modeler

When you are creating load combinations using loads from STAAD.Pro Physical Modeler, there are two approaches:

1. Creating primary loads for each reference load generated from STAAD.Pro Physical Modeler. This will combine the analysis results of primary load cases and allow you to include the STAAD.Pro Physical Modeler loads in those results.
2. Using repeat load combinations. These load combinations will use the reference load cases generated by STAAD.Pro Physical Modeler (with any necessary load factors).

> **Note:** Do not use primary load cases with STAAD.Pro Physical Modeler reference loads *and* repeat style load combinations, or the results of those load cases will be included twice.

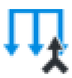
M. To define a new load combination

To manually define a load combination for post-processing, use the following procedure.

One or more primary load cases or load combinations must be defined in order to create a combination.

> **Note:** Load combinations are *not* directly processed by the analysis engine. Instead, their results are combined as specified by the combination method. In order to process combinations of load cases use either a Repeat Load or a Reference Load in a new Primary Load case.

1. On the **Loading** ribbon tab, select the **Combinations** tool in the **Loading** group.

   The **Define Load Combinations** dialog opens.
2. Type the **Load No** and load combination **Name**.
3. Select the combination Type to use:
   - **Normal -** a linear combination of a factor multiplied by a load result value
   - **SRSS -** a linear combination of either a Normal type or a square root of the sums squared (SRSS) combination. The factors for each control how each load component is added to the resulting combination
   - **ABS -** a linear combination of a factor multiplied by the absolute value of a load result
4. Select a Primary load case or existing load combination from the **Available Load Cases** list and then click **>** to add the load combination definition.

   The selected load case is added with the default type factors defined above. You can change the factors for an individual load combination component as needed.

   > **Moving Loads:** In order to include the individual load cases from a moving load generator, you must add these using the STAAD.Pro Editor. They cannot be entered using the dialog as they are not generated until the STAAD engine processes the input file.
5. Repeat step 4 for all load components to be added to this load combination.

   > **Tip:** Click **>>** to include all **Available Load Cases**, which will be use the default factors.
6. (Optional) To remove a load component from the **Load Combination Definition**, select the component in and then click **<**. To remove all load components, click **<<**.
7. Click **Add**.

Related Links

- *TR.35 Load Combination Specification* (on page 2616)
- *Define Load Combinations dialog* (on page 2836)

M. To define primary load type

To define or change the load type specified for a primary load case, use the following procedure.

You can define or change the load type for any previously added primary load case.

The load type is used for automatic load combinations.

1. On the **Loading** ribbon tab, select the **Primary Load Type** tool in the **Load Generation** group.

   The **Define Load Type** dialog opens.
2. For each primary load case, select the **Type** from the drop-down list in the cell.
3. (Optional) For Life or Roof Life load types, check the **Reducible-IBC 2003** if this particular load is considered a reducible live load per IBC.
4. Repeat steps 2 and 3 for all primary load cases.
5. Click **OK**.

Related Links

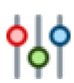
- *Define Load Type dialog* (on page 2871)
- *TR.32 Loading Specifications* (on page 2461)

M. To define automatic load combination rules

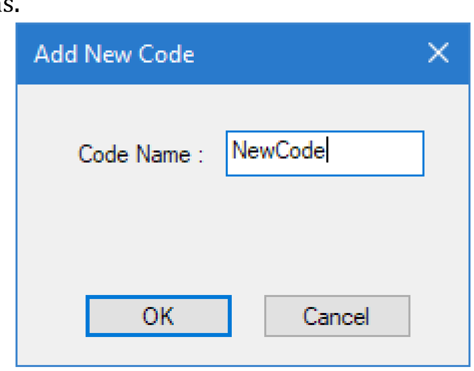
To define new or to edit existing rules for generating automatic load combinations, use the following procedure.

Note: In order to properly generate load combinations for Eurocode (unofficially referred to as EC0), the program contains a macro which allows for gamma inputs. For that code, refer to “ M. To generate load combinations per Eurocode (on page 890).”

1. On the **Loading** ribbon tab, select the **Automatic Combinations** > **Edit Auto Combination Rules** tool in the **Load Generation** group .

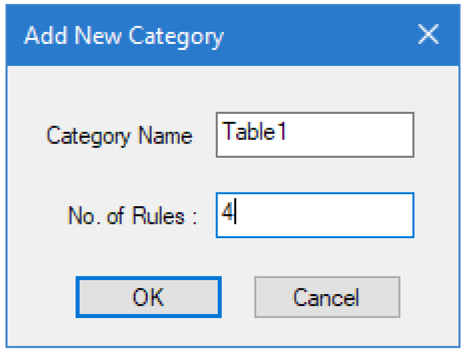
   

   The **Edit Loading Rules for Auto Load Combination Generation** dialog opens.
2. (Optional) To create a new Code and Category of load combinations:
   a. Click **New Code**.
      The **Add New Code** dialog opens.



   b. Type a **Code Name** and then click **OK**.
      A new, empty table is loaded.

c. Click **New Category**.
The **Add New Category** dialog opens.



d. Type the **Category Name** and the **No. of Rules** to initially create in the table.

You can add or remove table rows after you create the table.

e. Click **OK**.
The specified rows are added for the new table.

3. For each load type (i.e., column):
   a. Set the **Include Notional Loads?** check box to include notional loads in this type for direct analysis.

   Notional loads are included by default only in dead, live, roof life, snow, and ice load types.

   b. Select the appropriate **Combination Rule** option for this load type:

   - **Aggregate -** Combine all cases together: For each rule, a single combination will be created which will include all the load cases of that load category multiplied by the factor in the table.
   - **Separate -** Separate combination for each case: For each rule, multiple combinations will be created, each will include one of the load cases of that load category multiplied by the factor in the table.
   - **Matrix -** All possible combinations: For each rule, multiple combinations will be created which will include each of the load cases of that load category on their own and with each and every other load case of that category multiplied by the factor in the table.
4. For each combination (i.e., row) in the table, specify the load multiplier for the load types present in that load combination.

Tip: To add a new road, click **Add Row**. To remove a row, click the row number to highlight the entire row and then press **<Delete>**.

5. To save the changes to a table, click **Update Table**.
6. Repeat steps 3, 4, and 5 individually as needed to specify the load combinations for the code category.
7. Click **Close**.

You can now generate load combinations using this rule set.

Related Links

- *Edit Load Rules for Auto Load Combination Generator dialog* (on page 2875)
- *Edit Load Rules for Auto Load Combination Generator dialog* (on page 2875)

M. To automatically generate load combinations

To automatically generate load combinations based on predefined rules for load types, use the following procedure.

In order to use this feature, one or more primary load cases must be created with a load type defined. A load type can be assigned to a primary load case either at the time when the load is being created or later.

If you want to customize the load combination rules or create your own for your organization, you must do this before creating the automatic load combinations.

Note: In order to properly generate load combinations for Eurocode (unofficially referred to as EC0), the program contains a macro which allows for gamma inputs. For that code, refer to " M. To generate load combinations per Eurocode (on page 890)."

1. On the **Loading** ribbon tab, select the **Automatic Combinations** > **Automatic Load Combinations** tool in the **Load Generation** group .

   The **Auto Load Combination** dialog opens.

   Note: If no load cases have yet been defined for the model, a warning message is displayed over the dialog and the parameters are inactive. Primary load cases are defined using the **Create Primary Load Case** dialog.
2. Choose the code to use from the **Select Load Combination Code** drop-down list.

   ACI:318-2002
   AISC 9th Ed
   ASCE 7-10
   BS:5950
   BS:8110
   GB:50009-2012
   GB:55002-2021
   IBC-2012
   IBC-1997
   NBCC-2005
   NBCC-1995
   IS:456 / IS:800
   SNiP 2.01.07-85
   UBC-1997
   ASCE 7-16
3. Chose the code-specific load combination table to use from the **Select Load Combination Category** drop-down list.
4. (Optional) Type the **Select Starting Combination No** to use for the first generated load combination.
5. (Optional) Check the **Create Repeat Load Cases** option to generate repeat load cases in place of results load combinations.

   Standard load combinations are algebraic combinations of results and therefore are not appropriate for second order effects. A repeat load case is a new primary load case which will re-use the loads from previously defined primary load cases in the analysis engine.
6. (Optional) Check the **Include Notional Load?** option to include notional loads (when notional loads have been specified) in the Repeat Load Cases.
7. Click **Generate Loads**.
   The load combinations based on the selected code and category are displayed in the **Selected Load Combinations** list.

8. Select any loads you do *not* want included in the load combinations and then click **<** to place them in the **Discarded Load Combinations** list.
9. Click **Add**.

Related Links

- *Auto Load Combination dialog* (on page 2872)
- *Auto Load Combination dialog* (on page 2872)
- *TR.35 Load Combination Specification* (on page 2616)

M. To generate load combinations per Eurocode

To generate load combinations for the Strength limit state per *Eurocode - Basis of structural design, BS EN 1990:2002+A1:2005* (sometimes referred to as "Eurocode 0").

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

The load combination generator is capable of creating load combinations per equations 6.10, 6.10a, or 6.10b found in Cl. 6.4.3.2.

These equations specify the following combinations of loads:

$$\sum_{j\geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} Q_{k,1} + \sum_{i>1} \gamma_{Q,1} \psi_{0,1} Q_{k,1}$$

Alternatively, for the strength limit state, the less favorable of equations 6.10a and 6.10b may be used:

$$\sum_{j\geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i>1} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \qquad (6.10a)$$

$$\sum_{j\geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_p P + \gamma_{Q,1} Q_{k,1} + \sum_{i>1} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \qquad (6.10b)$$

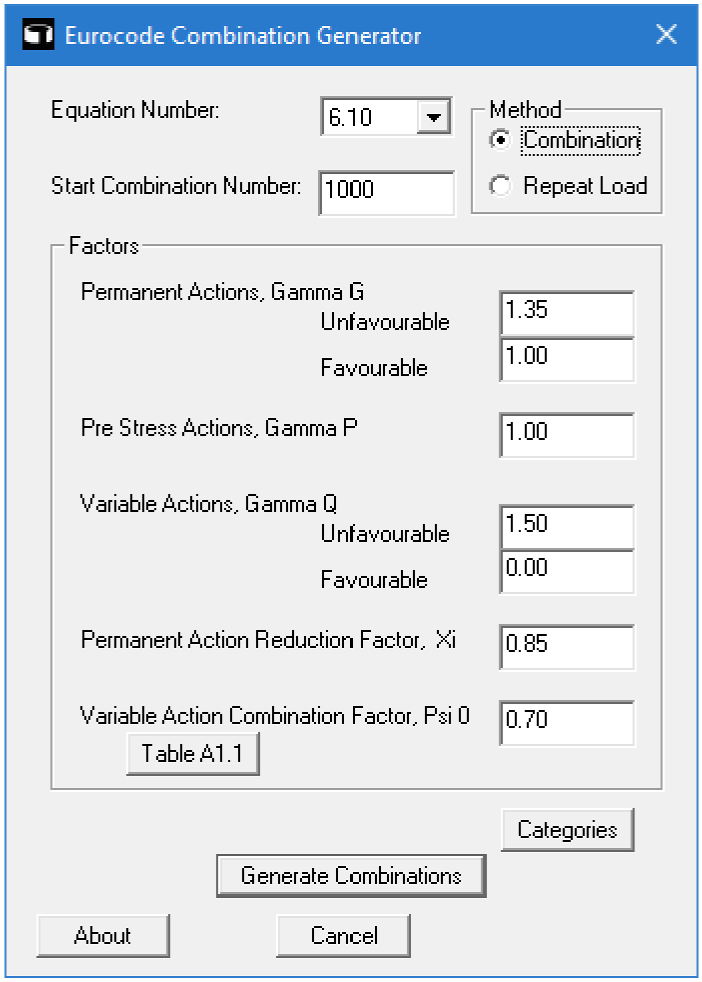
where

Gk	=	Permanent actions
P	=	Prestress actions
Qk	=	Variable actions

Note: The effects in each of the above equations are always additive. If any effect is negative (that is, would reduce the final sum), its effect is taken as zero.

1. On the **Utilities** ribbon tab, select the **User Tools** > **Euro Code Load Combination Generator** tool in the **User Tools** group.

   The **Eurocode Combination Generator** dialog opens.



2. Select the **Equation Number** to use for the generation of load combinations.
3. (Optional) Specify a **Start Combination Number**.
4. Select the **Method** as either:

   Combination to use post-analysis combination of results, which appropriate for linear elastic analysis

   or

   Repeat Load to generate repeat load cases from the primary load cases, which is appropriate for second order effects and dynamic analysis
5. (Optional) Specify load Factors for use in the combination equations.

Factor title	Equation notation
Permanent Actions, Gamma G	γ_G
Pre Stress Actions, Gamma P	γ_P
Variable Actions, Gamma Q	γ_Q
Permanent Action Reduction Factor, Xi	ξ

Factor title	Equation notation
Variable Action Combination Factor, Psi 0	ψ_0

Note: The default values are taken from those provided in EC0.

6. (Optional) Click **Table A1.1** to select one of the recommended values for the Permanent Action Reduction Factor, Psi (ψ_0).
7. Click **Categories** to specify in which load action classification each STAAD.Pro load category is to be assigned.

Action	Included Loads
(None)	Loads of this type will *not* be included in generated load combinations.
Permanent	Gk (permanent) By default, Dead loads are included. **Tip:** You may also want to include Mass or Gravity load.
Variable	Q (variable) By default, Live, Roof Live, Wind, and Snow are included. **Tip:** You may also want to include loads such as Seismic, Temperature, etc.
Pre Stress	P (prestress) No loads are included by default in this category. **Tip:** As STAAD.Pro does not use a load category for prestressing forces, a less-common load category such as **Imperfection** can be used for this action.

8. Click **Generate Combinations**.
A confirmation dialog box opens with the status of the export. Load cases are generated with the selection equation and load case number in the title.

M. To add a repeat load case

To create a primary load case using combinations of previously defined primary load cases, use the following procedure.

A repeat load case is treated as a new primary load. That is, it is processed by the analysis engine rather than being a combination of results. It is therefore applicable for P-Delta analysis.

A repeat load is added similar to a load item within a primary load case.

1. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

The **Add New Load Items** dialog opens.

2. Select the **Repeat Load** tab.
   A list of the primary load cases which occur *before* the current primary load case is displayed.
3. Either:

   select a load case on the left and then click **>** to add it to the repeat load definition

   or

   click **>>** to add all the primary load cases to the repeat load definition
4. (Optional) Type a load Factor to use for each primary load case included in the repeat load definition.

   Note: The primary load case data is factored rather than results.
5. Click **Add**.
6. Click **Close**.

Related Links

- *TR.32.11 Repeat Load Specification* (on page 2595)
- *Repeat Load tab* (on page 2863)

M. To create a reference load

To define a reference load and add it to a reference load item, use the following procedure.

A reference load case allows you to construct a load case with load items but it will not be directly solved by the analysis engine. It is only solved when it is called by another load case. This allows you to limit the number of load cases analyzed.

Tip: A reference load of type Mass is especially useful to define the structure masses used for dynamic analyses. One reference load case can be used by each of those analysis commands.

1. In the **Load & Definition** dialog, select the **Definitions** > **Reference Load Definitions** entry and then click **Add**.
   The **Add New Reference Load Definitions** dialog opens.
2. (Optional) Type a Number for the load case.

   The program will increment the load case number based on existing load case numbers.
3. (Optional) Select the **Loading Type**.

   This selection is used when automatically generating load combinations. If you select a live load type, you may also indicated if the load is **Reducible per UBC/IBC**.

Dead	Soil	Ice
Live	Rain Water/ Ice	Wind on Ice
Roof Live	Ponding	Crane Hook
Wind	Dust	Mass (Notes (on page 2461))
Seismic-H (horizontal, Notes (on page 2461))	Traffic	Gravity

Seismic-V (vertical, Notes (on page 2461))	Temperature	Push
Snow	Accidental	None
Fluids	Flood	

4. Type a **Title** used to easily identify this load case.
5. Click **Add**.
   The reference load definition is added to the **Definitions** > **Reference Load Definitions** entry in the **Load & Definition** dialog.
6. Add load items to the reference load definition.

   This is done similar to how you would add load items to a primary load case. Reference load definitions can contain:

   selfweight loads
   nodal loads
   member loads
   physical member loads
   area loads
   floor loads
   plate loads
   surface loads
   solid loads
   temperature loads
7. Either:

   On the **Loading** ribbon tab, select the **Load Items** tool in the **Loading Specifications** group 

   Tip: This will add the load item to the currently selected load group selected in the program status bar.

   or

   In the **Load & Definition** dialog, select a primary load case in the **Load Cases Details** list and then click **Add**.

   The **Add New Load Items** dialog opens.
8. Select the **Repeat Load** > **Reference Load** tab.
   A list of the reference load definitions is in the Available Load Cases list.
9. Either:

   select a reference load definition and then click **>** to add it to the Reference Load item

   or

   click **>>** to add all the available reference load definition to the Reference Load item.
10. Repeat Step 9 to add additional reference load definitions to the Reference Load item.
11. (Optional) Type a Factor to apply to each reference load definition in the Reference Load item as needed.
12. Click **Add**.

Related Links
- *Repeat Load tab* (on page 2863)

- *TR.31.6 Defining Reference Load Types* (on page 2453)

M. Damping Modeling

In STAAD.Pro, you can specify damping for the entire structure, composite damping for each material in the structure, or damping for each mode. Modal damping can be explicitly defined for each mode or calculated by the program based on the first two frequencies.

M. To assign a composite damping ratio

To assign a damping ratio to a model selection, use the following procedure.

This method assigns a composite damping ratio by a material constant.

Note: If you M. To assign material definitions (on page 797) to members, elements, or solids, then the critical damping ratio for that material will be used.

1. On the **Specifications** ribbon tab, select the **Constants** > **Damping Ratio** tool in the **Materials** group.

   The **Material Constant - Damping Ratio** dialog opens.
2. Select either:

   the **Enter Value** option and then type the composite damping ratio value (must be between 0.001 and 0.99)

   or

   one of the three predefined materials: **Aluminum**, **Concrete**, or **Steel**

Material name	Composite damping ratio value
Aluminum	0.03
Concrete	0.05
Steel	0.03

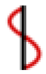
3. Select the **To Selection** option to limit the assignment to the selection set.
4. Click **OK**.

Related Links
- *G.17.3.3.1 Composite Damping* (on page 2160)
- *Material Constant dialog* (on page 2809)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

M. To explicitly define damping values for modes

To define damping values for modes for use with dynamic analysis, use the following procedure.

1. On the **Loading** ribbon tab, select the **Modal Damping** tool in the **Dynamic Specifications** group.

   The **Modal Damping** dialog opens.
2. Select the **Explicit** option.
3. Type a **Damping** ratio to use for each mode number.
4. Click **OK**.

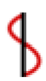
Related Links
- *G.17.3.3.2 Modal Damping* (on page 2160)
- *Modal Damping dialog* (on page 2910)
- *TR.26.4 Modal Damping Information* (on page 2312)

M. To evaluate damping for modes

To instruct the program to evaluate the modal damping values from the first two

STAAD.Pro also has a similar method available for calculating the modal damping when provided the mass-proportional damping coefficient, α and the stiffness-proportional damping coefficient, β. This method must be input using the STAAD.Pro Editor.

1. On the **Loading** ribbon tab, select the **Modal Damping** tool in the **Dynamic Specifications** group.

   The **Modal Damping** dialog opens.
2. Select the **Evalute** option.
3. Type **Min** and **Max** values to evaluate damping values for each mode.

   The minimum damping value is used for the first two modes. This is also used to evaluate the damping coefficients for subsequent modes based on the first two modal frequencies. If any evaluated damping is greater than the specified maximum value, then the maximum damping is used instead. Refer to Evaluate Damping (on page 2162) for additional details.
4. Click **OK**.

Related Links
- *G.17.3.3.2 Modal Damping* (on page 2160)
- *Modal Damping dialog* (on page 2910)
- *TR.26.4 Modal Damping Information* (on page 2312)

M. Mass Modeling

Rules Used for Generation of Mass Models

The following rules are applied by the program when the **Mass Model Generator** tool is used to create equivelant

1. Only vertical load components are considered out of the selected loads.

   Typically, these are loads parallel to the global Y direction. In the case of Z Up, these are loads parallel to the global Z direction.
2. Only the user selected loads are considered, regardless of load item types and directions included in all load cases.
3. Only load items in global or projected directions are considered. Any load item in the local direction (typically those labeled X, Y, or Z; e.g., uniform member loads in local coordinate directions) are ignored.

   Note: Selfweight loads are by default applied in the global X, Y, and Z directions. So the vertical selfweight loads are considered.
4. Moment component of loads are ignored for generating equivalent gravity loads.
5. Repeat and reference loads are expanded into a set of internal loads and the rules are then recursively applied to these.
6. Additionally, some load types are ignored as they are not applicable to mass models. Please see the following table for additional details by load item category and load item type.

Table 39: Load Item Types Considered

Category	Type of LoadItem	Consider
Self-weight		Yes
Nodal Load	Node	Yes
	Support Displacement	No
Member Load	Uniform Force	Yes
	Uniform Moment	No
	Concentrated Force	Yes
	Concentrated Moment	No
	Linear Varying Load	No
	Trapezoidal Load	Yes
	Hydrostatic Load	Yes
	Pre/Post Stress Load	No
	Fixed End Load	No
Physical Member Load	Uniform Force	Yes

Category	Type of LoadItem	Consider
	Uniform Moment	No
	Concentrated Force	Yes
	Concentrated Moment	No
	Trapezoidal Load	Yes
Area Load	Area	No
Floor Load	Floor	Yes
Plate Load	Pressure on Full Plate	Yes
	Concentrated Load	Yes
	Partial Plate Pressure	Yes
	Trapezoidal	Yes
	Hydrostatic	Yes
	Element Joint Load	Yes
Solid Load	Solid Pressure Load	Yes
Temperature Load	Temperature	No
	Strain	No
	Strain Rate	No
Seismic Load	Seismic	No
Time History	Time History	No
Wind Load	Wind Load	No
	Wind Load Dynamic	No
Snow Load	Snow	No
Response Spectra	Response Spectrum	No
Repeat Load	Repeat Load	Yes
	Reference Load	Yes
	Notional Load	No
Frequency	Rayleigh Frequency	No

Category	Type of LoadItem	Consider
	Modal Calculation	No

Related Links

- *Create Mass Model dialog* (on page 2870)
- *M. To generate a mass model* (on page 899)

M. To generate a mass model

To automatically generate from existing primary load cases or reference load definitions, use the following procedure.

Your model must contain one or more primary load cases or reference load definitions as load sources. The load case types must be defined as: dead, live, snow, ice, crane hook, or dust.

Any load case or reference load of load type mass will automatically be included in the generated mass model. The generated mass model contains all load items from the included load sources. These load items are factored using the specified load factors and act in the global X, Y, and Z directions.

Tip: Use the **Primary Load Type** tool to quickly set the load types for multiple primary load cases.

1. On the **Loading** ribbon tab, select the **Mass Model Generator** tool in the **Load Generation** group.

   The **Create Mass Model** dialog opens.
2. Select the load case where the mass model will be generated as either:

   New Reference Load Case

   or

   Add into 1st Dynamic Load Case

   Note: Your model must contain at least one dynamic load case in order to use the second option.
3. Add the load cases to use in order to define the mass model.

   All primary load cases of the appropriate load types are automatically added to the **Selected Load Cases** list. Reference load cases of the appropriate load types are may be selected from the **Available Reference Load Cases** list as needed.

   Tip: You can remove any load cases from the **Selected Load Cases** list by selecting that item and then clicking the **<** button.
4. Click **Generate**.

Related Links

- *Rules Used for Generation of Mass Models* (on page 897)
- *Create Mass Model dialog* (on page 2870)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *TR.31.8.2 Reference Load Mass Tables* (on page 2456)

- *TR.31.8.3 Mass Model Using Reference Load* (on page 2457)
- *Create Mass Model dialog* (on page 2870)

M. To add a mass model reference load

A mass model reference load is added in the same way as any other reference load, but with the Loading Type set to Mass.

1. Either:

   on the **Loading** ribbon tab, select the **Reference Load Case** tool in the **Loading Specifications** group

   or

   select the **Definitions** > **Reference Load Definitions** section of the **Load & Definition** dialog box and then click **Add**.

   or

   s

   The **Add New: Reference Load Definition** dialog box opens.
2. (Optional) Type a reference load identification number in the Number field.

   Tip: This number is incremented by one from any previously defined reference loads and typically does not need to be changed.
3. Select `Mass` as the **Loading Type**.
4. (Optional) Type a label in the **Title**
   For example, you may want to label the reference by typing `Mass Model`.
5. Click **Add**.
   The dialog box closes and a new reference load definition is added to the input file.

Related Links

- *TR.31.6 Defining Reference Load Types* (on page 2453)

M. To add mass loads to the mass model reference load

1. Select the mass Reference Load Case in the **Load & Definition** dialog box.
2. Click **Add**.
   The **Add New: Reference Load Items** dialog box opens.
3. Select the type of load you want to add from the tree.
4. Specify load parameters (e.g., magnitude, direction, etc.).
5. Click **Add**.
6. (Optional) Repeat steps 3 through 5 to add additional loads.
7. Click **Close**.

Related Links

- *TR.31.6 Defining Reference Load Types* (on page 2453)

M. To create a load envelope

To create a load envelope for grouping loads together in the analytical workflow, use the following procedure.

1. Select the **Load Envelopes** list on the **Load & Definition** dialog and then click **Add**.
   The **Add New Load Envelopes** dialog opens.
2. (Optional) Select an envelope **Type** from the drop-down list.

   Note: In the case of designing steel members for strength and serviceability, it is important to select the applicable load type.
3. Either:

   select a load case or load combination in the **Available** list and then click **>** to add that item to the envelope

   or

   click **>>** to add all the Available load cases and load combinations to the envelope
4. Repeat step 3 to add additional load cases or combinations to the envelope.
5. Click **Add**.
   The load envelope is added to the input file.
6. Repeat steps 2 through 5 to create additional load envelopes.
7. Click **Close**.

Related Links

- *TR.40 Load Envelope* (on page 2663)
- *TR.40 Load Envelope* (on page 2663)
- *Add New Load Envelopes dialog* (on page 2833)

M. To edit a previously assigned load

To modify the load magnitudes, direction, or assignments for existing load items, use the following procedure.

Loads can be changed within the Analytical Modeling workflow.

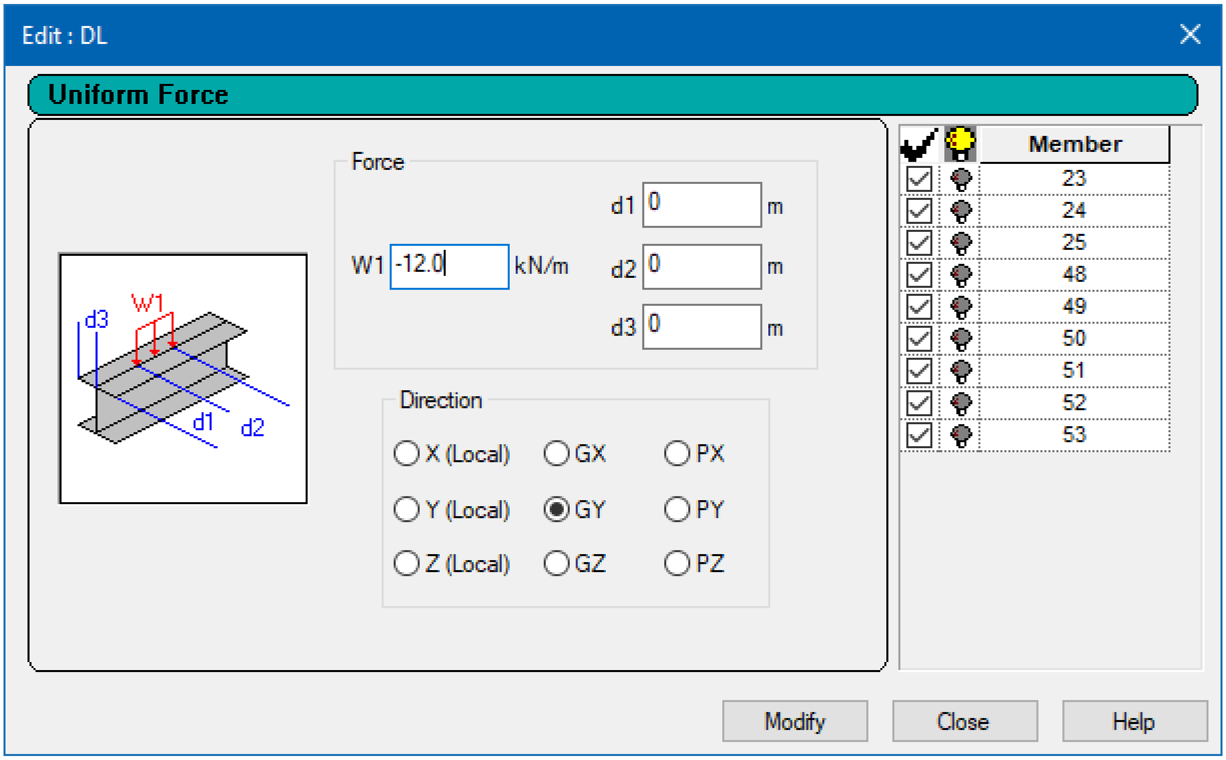
1. Select either:

   the **Loading** ribbon tab

   or

   the **Loading** page

   The **Load & Definition** dialog opens.
2. Expand the Loads section and load case containing the load item you wish to edit.
3. Select the load item you wish to edit.
4. Click **Edit**.
   A **Edit:** load item dialog specific to the defined load type opens.



This dialog is analogous to the **Add New Load Items** dialog, with additional controls included for editing the structural objects to which the load is assigned.

5. Make any changes necessary to magnitude, directions, or other load parameters entered when the load was created.

Tip: To aid in the load editing process, you can select the highlight option ( ) to select the corresponding structural object in the active view window.

6. Un-check individual rows for each structural object in the right-hand table to remove the load from this object.
7. Click **Modify** to save the changes.
8. Repeat steps 5 through 7 to make additional changes as needed.
9. Click **Close**.

Related Links

- *Load & Definition dialog* (on page 2827)
- *Add New Load Items dialog* (on page 2839)

M. Piping workflow

The STAAD.Pro Piping workflow is used to import piping support geometry and reactions from Bentley AutoPIPE.

The piping system in a plant model is supported by a number of structural components – such as pipe racks, cradles or shoes – constructed exclusively for that purpose or from the existing beams, columns and slabs of the structure. To facilitate the design of structures to sustain the loading imparted by the piping system, STAAD.Pro includes a seamless transfer of support reactions calculated from an AutoPIPE analysis to the structure modeled in STAAD.Pro.

M. Using the Piping Workflow

General Workflow

1. Import a piping model via the PipeLink utility.
2. Use the Support Connection Wizard to automatically assign connections between pipe nodes and structural entities.
3. Modify or add connections as necessary either graphically or via the Pipe Support table.
4. Transfer loads from the pipe stress model to the STAAD.Pro model.
5. Analyze the STAAD.Pro model for the new loads.
6. Export any changed or updated model data back to AutoPIPEvia the PipeLink utility or the Export Macro.

M. To import a piping model

Tip: The scale, orientation, and origin of both the piping model and structural model should be the same.

1. From within Bentley AutoPIPE, export the pipe stress model using the PipeLink utility.

   Note: Refer to the AutoPIPE documentation for additional assistance on using this product.
2. Open a corresponding model in STAAD.Pro.
3. Select the **Piping** workflow.
   The Piping ribbon tab and pages open.
4. Either:

   on the **Piping** ribbon tab, select **Interop** > **Import** in the **Models** group

   or

   click **Import** on the Pipe Model dialog (on page 2962).

   An **Open** dialog opens with the **Files of type** filter set for PipeLink files.
5. Select the **PipeLink file created from AutoPIPE** in the Files list box.
6. Click **Open**.

   A message dialog displays the status of the file import process.

   Note: If clashes are detected when importing, then the Merging Support Connection dialog (on page 2971) opens. This is used to specify if either the imported or local data should be used.
7. Click **OK** to dismiss the dialog.
   The **Support Connection Wizard** opens.

Tip: Multiple models can be imported to a single STAAD.Pro project. Any actions are performed on the current active model as selected in the **Pipe Model** dialog.

Tip: Imported data is saved with the structure upon exiting the Piping mode for later use.

Related Links

- *Pipe Model dialog* (on page 2962)
- *Merging Support Connection dialog* (on page 2971)
- *Support Connection Wizard* (on page 2964)

M. To use the Support Connection Wizard

1. Either:

   on the **Piping** ribbon tab, select the **Support Connection Wizard** tool in the **Connection** group

   or

   M. To import a piping model (on page 903) as described above.

   The **Move Pipe Model** dialog opens.
2. Enter the offset distance between pipe and structural models and the click the **Next >** button.

   The **Pipe Nodes** dialog opens.

   The filtering options are All, Connected, Unconnected, V-Stop, or Anchor (relating to whether a connection to the STAAD.Pro model has been defined). When support type information is available the filter will be expanded to include these as well.
3. Select which nodes will be used by adding them to the Selected list and click the **Next >** button.

   The **Structure Beams** dialog opens.

   The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group, View and Property.
4. Select which structure elements will be used by adding them to the Selected list and click the **Next >** button.

   The **Structure Node** dialog opens.

   The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.
5. Select which structure nodes will be used by adding them to the Selected list and click the **Next >** button. The **Parameters** dialog opens.
6. Here you will set some parameters so the wizard can establish pipe connections and click the **Next >** button.

   The results of the Wizard will be displayed in the final dialog.

   See "Parameter Page / Connection Finder" for additional details on these settings.
7. Review these results and if changes are necessary, you may click the **< Back** button. Otherwise, click the **Finish** button to accept.

The results will be presented in a table with columns for pipe node id and for the structural item it is to be connected to. This second column will provide a drop list allowing the user to choose either “No connection” or one of up to 5 closest items. The items will be listed with the closest at the top. The initial state will be the closest item or ‘Ground’, if no good matches were found.

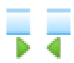
Tip: Manual connections can be generated using the following procedures.

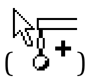
Related Links

- *Support Connection Wizard* (on page 2964)

M. To draw connections between piping supports and the structure

1. On the **Piping** ribbon tab, select the **Connection Support** tool in the **Connection** group.

   The mouse pointer changes appearance to the Support Connection cursor ().
2. Click a piping model support.
   A dynamic line is now attaches this support to the cursor.
3. Click either a node or a beam element in the structural model.

   The support association is shown as a dashed, red line in the view window. The Pipe Supports table (on page 2968) is updated to reflect the **Connect To** structural entity.

   Note: You may wish to edit the **Dist. Along Beam** value in the Pipe Supports table to control the exact connection location.
4. Repeat steps 3 and 4 to connect as many supports as needed.
5. Press the **<ESC>** key to exit the Support Connection cursor.

Related Links

- *Pipe Supports table* (on page 2968)

M. To manually specify connections between piping supports and the structure

1. Select the **Supports** page in the **Piping** workflow.
   The Pipe Supports table (on page 2968) opens.
2. Select the row of the Pipe Support Node you wish to connect to the structure.
3. Select if the support is **Connected To** either a **Node** or **Beam**.

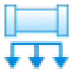
   Note: Ground supports are the default and are assumed to *not* be supported by the structural model.
4. Press **<Tab>**.
5. Type the connecting Beam or Node number in the **Structure Entity** cell for the Pipe Node.
   The connection association is shown as a dashed, red line in the view window.
6. Press **<Tab>** to update the model.
7. (Optional) (For Beam supports) Specify a distance along the beam, from the start node, to the connection point and then press **<Tab>**.
8. Repeat Steps 2 through 7 to connect as many supports as needed.

Related Links

- *Pipe Supports table* (on page 2968)

M. To transfer load data for structural analysis

1. On the **Piping** ribbon tab, select the **Transfer Loadings** tool in the **Loading** group.

   The Transfer Pipe Reactions to Structure Model dialog (on page 2970) opens.
2. Make any changes to the load data as needed.
3. Click **OK**.
   A Transfer Loads summary dialog opens to inform you of the status of the load data transfer.
4. Click the **OK** button to dismiss the dialog.

   The loads from the pipe stress supports are now added to the **Load & Definition** dialog in the model.

> **Note:** The structure is now ready to be analyzed for the imported pipe support loads.

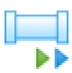
Related Links

- *Transfer Pipe Reactions to Structure Model dialog* (on page 2970)

M. To export model data for use in AutoPIPE

Before you export the model data, you should analyze the structural model. Select the **Analytical Modeling** workflow and then on the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group. Return to the **Piping** workflow once the analysis is succesfully completed.

1. On the **Piping** ribbon tab, select **Interop** > **Export** in the **Models** group.

   The Export Revised Model dialog (on page 2961) opens.
2. (Optional) Edit **User Name**
3. Click **OK**.
   The **PipeLink for STAAD.Pro V8i** utility program opens.
4. Either:

   Select an existing data exchanger file (file extension `.pipelink`.) by clicking **[...]**.

   or

   Create a new file by clicking **Use Blank**.
5. Either:

   Select **Export** > **Start Run**.

   or

   Select the **Run** tool. 

   The progress of the export process is displayed in the Output window. A message dialog opens to provide you with the status of the export.
6. Click **OK**.

7. Review any error or warning messages as needed.
8. Select **File** > **Exit** to close the utility program.

Note: Refer to the documentation included with the PipeLink plug-in for additional help in using this utility.

Related Links

- *Export Revised Model dialog* (on page 2961)

M. To export STAAD.Pro model data into AutoPIPE

Tip: The macro `ToAutoPipePub.vbs` is included with STAAD.Pro for the purpose of creating structural support data in AutoPIPE from a STAAD.Pro model. The resulting file contains only the support frame data for the current STAAD input file.

1. On the **Utilities** ribbon tab, select **User Tools** > **Export Model to AutoPIPE** in the **User Tools** group.

   The Export STAAD Model to AutoPIPE (on page 2959) opens.
2. Select the AutoPIPE Neutral file (file extension `.NTL`) you wish to create or to which you wish to add data.

   Tip: Click **[...]** to select an existing AutoPIPE Neutral file.
3. Specify **Job Information** and **Header** data as necessary.
4. Select the appropriate system of **Units.**
5. Select an option for how the **Member Properties** are displayed in AutoPIPE.
6. Click **OK.**

   Note: If any invalid structural properties or specifications are found, a warning dialog opens to display these issues. Otherwise, a dialog opens to confirm the AutoPIPE Neutral file has been created.

Related Links

- *Export STAAD Model to AutoPIPE* (on page 2959)

M. Pages in the Piping Workflow

The Pages in the Piping workflow are described below in brief.

Table 40: Piping Page Controls in STAAD.Pro

Page	Purpose
Piping	Used to display piping models and select the active model when more than one piping model has been imported. When the **Piping** page is selected, the Pipe Model dialog (on page 2962) opens.
Supports	Used to review and edit support connections manually. When the **Supports** page is selected, the Pipe Supports table (on page 2968) opens.

M. Bridge Deck workflow

STAAD.beava (Bridge Engineering Automated Vehicle Application) mode is integrated into the Bridge Deck workflow. This facility is used for generating loads for use in the analysis of bridge structures.

Overview

The general philosophy governing the design of bridges is that, subject to a set of loading rules and constraints, the worst effects due to load application should be established and designed against. The process of load application can be complex as governing rules can impose interdependent parameters such as loaded length on a lane, lane factors, and load intensity. To obtain the maximum design effects, engineers have to try many loading situations on a trial and error basis.

This leads to the generation of many live load application instances (and a large volume of output data) that then must be combined with dead load and other effects, as well. Bridge Deck is used to minimize the load application process while complying with national code requirements.

The program is based on the use of influence surfaces, which are generated by STAAD.Pro as part of the loading process. An influence surface for a given effect on a bridge deck relates its value to movement of a unit load over the point of interest. The influence surface is a three-dimensional form of an influence line for a single member (or, in other words, it is a 2D influence function).

STAAD.Pro will automatically generate influence surfaces for effects such as bending moments for elements, deflection in all the degrees of freedom of nodes, and support reactions. You then instruct the program to utilize the relevant influence surfaces and, with due regards to code requirements, optimize load positions to obtain the maximum desired effects.

Once the influence surfaces have been generated, they are saved and can be reused for any further investigation that may be required. These remain valid as long as the you do not altered the structural model. Changes to the structural model can alter the pattern of the influence surfaces and you must ensure that a further analysis takes place before any further processing.

Your engineering knowledge and judgement is critical in deciding which effects are required and at what position to obtain them. This is where you can save a lot of processing time and also can ensure critical positions are not missed.

Bridge Standards

The Bridge Deck mode supports the following standards:

- UK: BS 5400 Specific Parameters (on page 2983)
- UK: BD21/01 BD21/01 Specific Parameters (on page 2984) and BD21/01 Annex D Specific Parameters (on page 2984)
- American: AASHTO Specific Parameters (on page 2985)
- American: AASHTO LRFD Specific Parameters (on page 2986)
- Indian: IRC Specific Parameters (on page 2986)

All the relevant code instructions for loading definitions and traffic lane calculations are incorporated in BEAVA and, in cases where vehicle axle arrangements are not standard, it is possible to define a vehicle and save it in the library for use it in the analysis.

Note: It is not uncommon for local authorities to have superseding documents or amendments to national documents. It is the responsibility of the engineer to be familiar with these codes and input the appropriate parameters for loads.

BEAVA is fully integrated in STAAD.Pro and utilizes the same GUI for all input and output data.

Roadway and Load Modeling

You must define the width of the Roadway as straight or curved parallel lines. BEAVA then automatically calculates the following in accordance with the selected code:

- Number of Notional Lanes (Traffic Lanes)
- Influence lines along the center line of notional lanes
- Loaded length along the Lanes
- Critical location of uniformly distributed load
- Critical location of knife edge load
- Critical location of vehicle load
- Maximum effect value
- Associates effects values

Once the program has completed calculating the above, a text file containing the results is displayed on the screen.You can also then examine the results graphically.

Loading arrangements for the effects requested can be displayed on the model and, for every loading arrangement produced, you can instruct the program to generate a STAAD.Pro load case. The added live load cases can be combined with dead loads using the normal STAAD.Pro load combination generation. The final model can then be analyzed in STAAD.Pro and then post-processed.

M. Using BEAVA

There are a number of distinct stages in the use of the program. It is strongly advised to follow this order when using the program.

Tip: It is recommended that you work through the EX. Bridge Deck Loading Example (on page 6988) to familiarize yourself with this process.

M. To open a model in the bridge deck workflow

1. Create a structural model –including member properties and support conditions– in STAAD.Pro.
2. Run an analysis on the structure.
3. Select **Bridge Deck** in the **Workflows** panel.

   Note: If you do not have a license for this module, you will not be able to proceed.

   The **Bridge Deck** ribbon tab opens.

M. To define a bridge deck

1. (Optional) If the mouse pointer is not already displayed as the Plate Cursor, in the **Analytical Modeling** workflow on the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

2. Select the elements and/or members that will be used to define the bridge deck area of the models.
3. In the **Bridge Deck** workflow on the **Bridge Deck** ribbon tab, select the **Create Deck** tool in the **Deck** group.

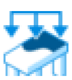
   

   The **Save Deck as** dialog opens.
4. (Optional) Type a **Name** for the deck.
5. Click **OK**.

M. To generate influence surfaces

To generate influence surfaces for the currently selected deck, do the following.

1. On the **Bridge Deck** ribbon tab, select the **Loading > Influence Surface Generator** tool in the **Loading** group.

   A temporary STAAD input file (`filename_deck.std`) is sent to the STAAD analysis engine for processing. The **STAAD Analysis and Design** dialog opens to display the progress. When the analysis is complete, the dialog closes automatically.

M. To define a roadway

To define a roadway (i.e., carriageway in some regions), do the following.

1. Either:

   Select a deck from the View window.

   or

   On the **Bridge Deck** ribbon tab, select the **Select Deck** tool in the **Deck** group and then select one of the defined decks from this list.

2. On the **Bridge Deck** ribbon tab, select the **Define Roadway** tool in the **Deck** group.

The Roadways dialog (on page 2974) opens.

3. Click **New**.
   The Define Roadway dialog (on page 2976) opens.
4. Specify the distances from the origin to each curb edge.
   The preview updates to display the roadway on the currently selected deck.
5. Click **OK**.
   A new roadway is now added to the Roadways list in the **Roadways** dialog and is outlined in blue on the deck in the active view window.
6. Click **Close**.

M. To generate loads on the roadway

1. On the **Bridge Deck** ribbon tab, select the **Loading** > **Run Load Generator** tool in the **Loading** group.
   The Load Generator Parameters dialog (on page 2981) opens.
2. On the **General** tab, select the appropriate **Design Code**.
   The ***<code>*** tab updates to reflect this selection.
3. Select the appropriate **Limit State**.
4. Select the ***<code>*** tab.
5. Make the code-specific selections for vehicles, road conditions, multiple presence factors, and impact factors.
6. Select the Node Displacements, Support Reactions, Beam End Forces, or Plate Center Stress tab.

   > **Note:** An action must be defined for which the maximum results determine the placement of the moving loads.

7. Specify a structural object number, direction of action, and sign of effect.
8. Repeat steps 6 and 7 as many times as needed.
9. Click **OK**.
   The program analyzes the deck to obtain the critical load positions for the specified action.

A summary of the analysis is opened in your default text editor (e.g., Notepad). The file (named `filename_deckx.out`) is saved in the same location as the input file. Close the text editor once you have completed reviewing this file.

The placement of the loads for each effect requested may be reviewed.

M. To transfer the loads to STAAD.Pro

To transfer the generated load cases to STAAD.Pro for analysis and design, do the following.

1. On the **Bridge Deck** ribbon tab, select the **Loading > Create Loading in STAAD Model** tool in the **Loading** group.
   A message dialog opens to confirm the load was successfully added in the STAAD input file.
2. Select the **Analytical Modeling** workflow.
3. On the **Loading** page, review the loads added to the input file in the **Load & Definitions** dialog.

   > **Note:** One load was added for each action requested in the **Load Generators Parameters** dialog.

4. (Optional) Specify additional analysis commands or design parameters as needed.

5. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

M. Checking Your Model

STAAD.Pro comes with a number of tools used to check your structure and model objects.

M. To check for multiple structures

To check for multiple, independently defined set of entities in your model, use the following procedure.

The STAAD.Pro analysis engine can accommodate multiple structures in a single model. However, this check allows you to confirm the number of detected structures to prevent an unintentionally defined model.

1. On the Utilities ribbon tab, select the **Structure Tools** > **Multiple Structures** tool in the **Geometry Tools** group.

   The **List of Structures** dialog opens.
2. (Optional) Select a structure name in the list.
   The entities detected in this structure are selected in the view window.

M. To check for warped plates

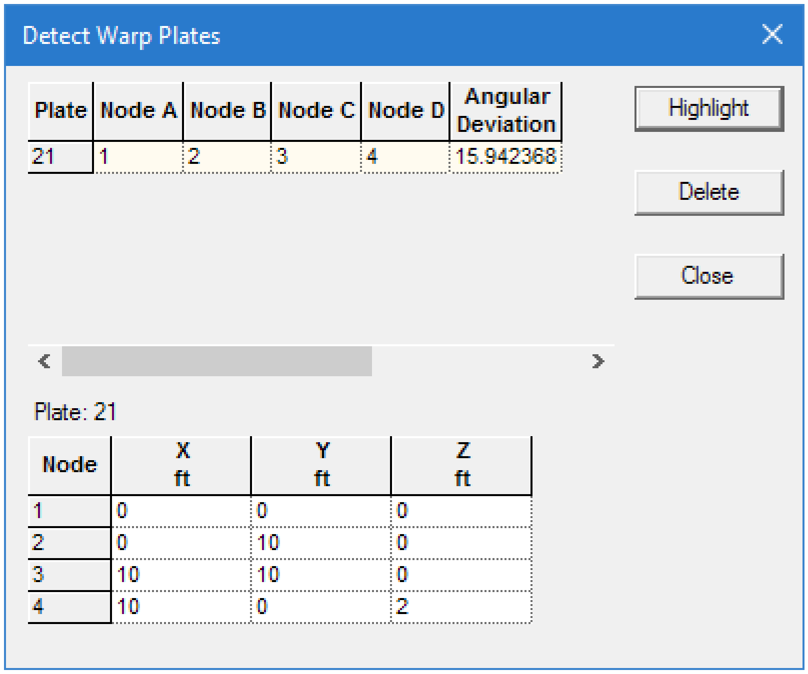
A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.

You can set the tolerance for warped plates by first setting the **Tolerance for Warped Plate Element Detection** value on the **Options** dialog Tolerance tab. Only plates whose angular deviation exceeds this tolerance value will be displayed.

For this check, warping is defined as excessive angular deviation between all vertex normals.

1. Select one or more plates in the model.
2. Select the **Plate Tools** > **Check for Warped Plates** tool in the **Geometry Tools** group on the **Utilities** ribbon tab.

   If any warped plates are detected, the **Detect Warp Plates** dialog opens. A list of all selected plates and their maximum angular deviation is displayed. The global coordinates for a selected plate are displayed below.



3. (Optional) Click **Highlight**
4. (Optional) Click **Delete**

Tip: You can undo this action through the Undo tool in the main interface.

5. Click **Close** when you have completed reviewing warped plates.

An example of such an element is demonstrated in the following STAAD input.

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 2
ELEMENT INCIDENCES SHELL
21 1 2 3 4
FINISH
```

Nodes 1, 2, and 3 lie in the XY plane at Z = 0, but node 4 has a Z coordinate of 2, thereby causing the plate to become non-planar.

The warped plate check sub-divides the quad element into two sets of triangles: one with a diagonal from point 1 to point 3 and another diagonal from point 2 to point 4. These triangles do not lie in the same plane so an angle is formed between them at their shared edge. The largest angle between adjacent triangles in the warped quad element is compared against the tolerance threshold. If this tolerance angle is exceeded, then the plate is reported as excessively warped.

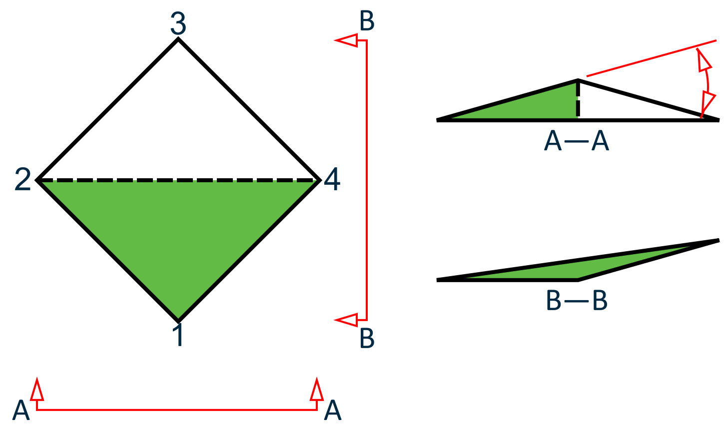


Figure 121: A) Checking triangles formed by the diagonal 2-4.

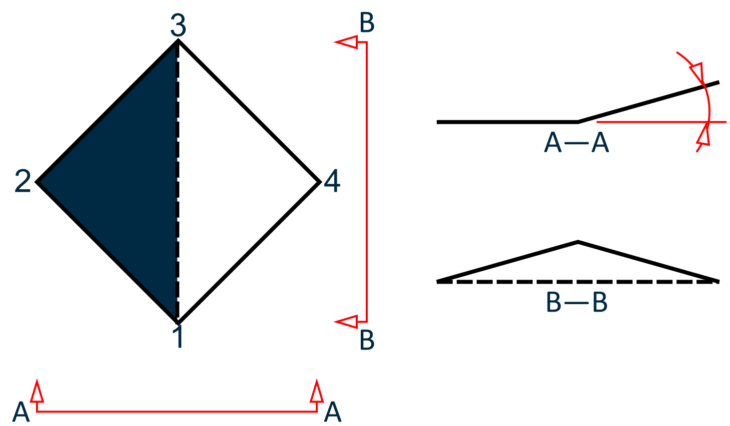


Figure 122: B) Checking triangles formed by the diagonal 1-3

M. To check for and remove duplicate entities

To detect the presence of and remove instances of nodes, beams, plates, etc. which are duplicates, use the following procedure.

One definition of duplicate nodes is that they are 2 or more nodes, having distinct node numbers, but the same X, Y, Z coordinates. For example, if node number 5 has coordinates of (7, 10, 0), and node 83 also has coordinates of (7, 10, 0), node 5 and 83 are considered duplicate. However, STAAD.Pro allows you to expand the definition of this term to include nodes separated by any distance. When this menu is selected, the following message box pops up, asking the user to specify a tolerance.

Duplicate members are 2 or more members, having distinct member numbers, but connected between the same 2 nodes. For example, if there are 3 members, say, 17, 46 and 75, all connected between the nodes 105 and 117, those 3 members are considered duplicate. The order of the node incidence is disregarded during this check. For example, if member 17 has 105 as its first node, 117 as its second, and member 46 has 117 as its first node, 105 as its second, they are still treated as duplicate.

A similar definition is applicable for duplicate triangular or quadrilateral plates. Plates are considered duplicates if they share the same vertex nodes, regardless of order.

1. Depending on the entity type you are checking, select one of the following tools from the **Utilities** ribbon tab in the **Geometry Tools** group:

To check for duplicate...	Select the following tool...
Nodes	**Node Tools** > **Duplicate Nodes**
Beams	**Beam Tools** > **Duplicate Beams**
Plates	**Plate Tools** > **Duplicate Plates**

   (Nodes only) The **Remove Duplicate Nodes** dialog opens.
2. (Nodes only) Type a distance in the **Enter Tolerance** field and click the **OK**.

   The tolerance is the value used to define the distanced at which nodes are to be considered separate or a duplicate. In some cases, the process of creating the model can result in nodes that a minuscule distance apart. Specifying a tolerance greater than zero allows the program to consider those nodes as duplicates and allows you to process them as needed.

   The program reviews the model for any duplicates of the selected model element type (using the tolerance for Nodes).

   If no duplicates are found, a message is displayed.

   If duplicates are detected, the **List of Duplicate Nodes / Members / Plates** dialog opens. Each entry in the list contains two or more entities that the program has detected to be duplicates.
3. Either:

   click **Remove All Duplicates** to remove the second and additional duplicate for item in the list and then click **OK** to confirm the deletion

   or

   select individual duplicate items to retain as follows:
   a. Select a duplicate set in the list and then click **Retain >>** to remove the duplicate from the model. The **Select Node / Member / Plate** dialog opens.
   b. Select which item to remove:

Options	Description
For nodes...	Either choose the node for deletion, either by clicking it, the first in the list, or the last in the list.
For beams and plates...	Select the **Member / Plate To Keep** drop-down list.

   c. Click **OK**.
      The remaining duplicates are removed and other model entities which refer to those duplicates now reference the selected (merged) entity.
   d. Repeat steps 3a through 3c as necessary to review all duplicates of the entity type selected.
4. Click **Close**.

Related Links

- *List of Duplicate Nodes / Beams / Plates dialog* (on page 2948)

M. To detect and remove zero length members

A zero length member is one of the undesirable consequences of having duplicate nodes in the structure. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero (or nearly zero).

1. On the **Utilities** ribbon tab, select the **Beam Tools** > **Check Zero Length Members** tool in the Geometry Tools group.
   The **Zero length beams** dialog opens.
2. (Optional) Type a different tolerance in the **Enter Tolerance** field.

   Since duplicate nodes can be two or more nodes separated by any user-defined distance, zero length members too can be defined as members of any user defined length. This distance is the tolerance used in the process of detection.
3. Click **OK**.

   If no members of zero length are detected a message is displayed.

   If one or more zero length members are detected, the **Detect Zero Length Member** dialog opens.
4. Select a zero length member in the list and then click **Delete** to remove the beam.

   Tip: You can click **Highlight** to select the member in the View window (though most zero length members are difficult to see unless at a very high magnification). Then select the **Properties** tool on the **Beam** ribbon tab to open the **Beam** dialog for the zero length member.
5. Repeat Step four as necessary until all zero length members are removed.
6. Click **Close**.

M. To check for overlapping collinear members

1. On the Utilities ribbon tab, select the **Beam Tools** > **Check Overlapping Colinear Members** tool in the **Geometry Tools** group.

   If no overlapping collinear members are detected, then a message dialog opens with that message.

   If one or more overlapping collinear members are detected, the **Overlapping Collinear Beams** dialog opens.
2. Select the overlapping members pair and the Highlight option to select the beam(s) in the Active View window.
3. The members in the Active View window can be manipulated or deleted as necessary while the **Overlapping Collinear Beams** dialog is open.

An example of 2 members which would qualify as overlapping collinear are:

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 13 10 0; 6 -4 10 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
101 5 6
FINISH
```

Here, members 2 and 101 are overlapping collinear. Member 2 is entirely confined within the span of member 101, and collinear, but they are not attached to each other.

Another example is:

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 13 10 0; 6 -4 10 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
101 2 5
FINISH
```

Here, again, members 2 and 101 are overlapping collinear. But even though they are connected to each other at node 2, again member 2 is entirely confined within the span of member 101, and collinear.

M. To change a beam incidence

To change the incidences of a selected set of beams, use the following procedure.

1. Select one or more members in the view window.
2. On the **Utilties** ribbon tab, select the **Beam Tools** > **Beam Incidence** tool in the Geometry Tools group.

   The **Redefine Incidence** dialog opens.
3. Select an incidence definition to use:

   Switch incidences of selected beams

   or

   Set the incidence order of the seleted beams so that the start node is close to the origin than end node

   or

   Set the incidence order of the seleted beams so that the start node is farther from the origin than end node
4. Click **OK**.

Related Links

- *M. Member Orientation* (on page 799)

M. To detect and remove orphan nodes

To detect and then remove nodes which are not connected to any other model entity (i.e., “orphan nodes”), use the following procedure.

In the process of modeling, disconnected nodes can be created in your model.

1. On the **Utilities** ribbon tab, select the **Node Tools** > **Oprhan Nodes** tool in the **Geometry Tools** group.

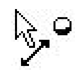
Any detected orphan nodes are selected in the view window.

2. To remove these orphan nodes, on the **Utilities** ribbon tab, select the **Node Tools** > **Remove Oprhan Nodes** tool in the **Geometry Tools** group.
   The detected orphan nodes are deleted.

M. To display the distance between two nodes

To temporarily display the dimension between two nodes in the active View window, use the following procedure.

1. On the **Utilities** ribbon tab, select the **Node Tools** > **Node to Node Distance** tool in the **Geometry Tools** group.

   The mouse pointer changes to the Node to Node Distance cursor ().
2. Click on any two nodes in succession in the active View window..
   A dimension call out is added between these two nodes.
3. Repeat step 2 to dimension as many distances as needed.
4. Either:

   Press the **<Esc>** key

   or

   Select the **Node to Node Distance** tool again

   The mouse pointer returns to the normal selection mode.

M. To display beam lengths

To display the lengths of each analytical member, use the following procedure.

To dimension only the members in a selection, select those members before starting this procedure.

1. On the **Utilties** ribbon tab, select the **Beam Tools** > **Dimension Beams** tool in the **Geometry Tools** group.

   The **Display/Remove Dimensions** dialog opens.
2. Select the option for which beams to be dimensioned:

To...	Select...
dimension all the beams in the current view window	**Dimension to View**
dimension only the beams you selected before step 1	**Dimension to Selected Beams**
dimension a comma separated list or range of beams	**Dimension to List**

3. Click **Display**.
4. (Optional) To remove the dimensions, select the **Remove** option and then click **Remove**.
5. Click **Close**.

Any new members will *not* be dimensioned. You must repeat this procedure to display their dimensions.

Related Links

- *Display/Remove Dimensions dialog* (on page 2949)

M. To check for negative volume solids

To check for solids that would result in a negative volume in the analysis, do the following.

This check uses the Jacobian matrix determinate to determine if any of the solids have a negative or indeterminate volume, which would result in an analysis error.

1. On the **Utilities** ribbon tab, select the **Solid Tools** > **Negative Volume** tool in the **Geometry Tools** group.

   The **List of Solids** dialog opens.
2. Select any solid number in the list to select it in the view window.
3. Click **Close**.

Related Links

- *G.5.2 Solid Elements* (on page 2110)

M. Physical Modeling workflow

You can use STAAD.Pro Physical Modeler to model a “physical” structure and then re-import that data back into the STAAD.Pro environment for loading, analysis, and design.

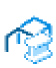
M. Using the Physical Modeler

You can model structure geometry, specifications, and many loads with the STAAD.Pro Physical Modeler.

You must being a physical model with an empty STAAD project. Existing analytical models cannot be opened in the physical modeler.

When you open a model that was created using the physical modeler, you will prompted if you want to open the model in the physical modeler environment.

1. Select the **Physical Modeling** workflow.

   The STAAD.Pro Physical Modeler window opens.

Refer to the STAAD.Pro Physical Modeler help for details on how to use that module to model your structure.

M. To drop the associated physical model

To remove the associated physical model from the current STAAD input file, use the following procedure.

Caution: This procedure cannot be undone. Once a physical model has been dropped, that model can no longer be opened within the **Physical Modeling** workflow.

When a physical model is associated with a STAAD input file, many tools in the **Analytical Modeling** workflow as well as portions of the input file in the STAAD.Pro Editor are disabled. This is intended to prevent making changes which would potentially corrupt the association between the physical and analytical models. However, there are times when you may need to drop this association to make changes to the analytical model.

1. Select the **Analytical Modeling** workflow in STAAD.Pro.
2. On the **Utilities** ribbon tab, select the **Drop Physical Model** tool in the **Physical Model** group.

   A warning dialog opens asking you to confirm you want to break this link.
3. Click **Yes**.

The input file is re-opened in STAAD.Pro as a analytical model only.

M. Building Planner workflow

The **Building Planner** workflow integrates STAAD Building Planner into STAAD.Pro. This is used to generate concrete building models which can then be analyzed in STAAD.Pro and detailed using the **Advanced Concrete Design** workflow.

M. To start a STAAD Model in the Building Planner workflow

1. On the Start page, select **New**.
   The New page opens to the **Model Info** tab.
2. Specify the **File Name**, **Location**, and **Units** as necessary.
3. Select the **Building** option for the **Type** of model.
4. (Optional) Select the **Job Info** tab to add project member names, dates, project description data, etc.

   You can also associate your STAAD project with a ProjectWise Project here.
5. Click **Create**.

   The STAAD.Pro window closes and the STAAD Building Planner application opens. The **Start** dialog opens.
6. Enter the project details, including **No of Levels** and **Founding Depth**.
7. Click **Create Project**.
   The **New Plan** dialog opens.

You are now ready to start adding details for the first floor plan and modeling your building structure.

Related Links

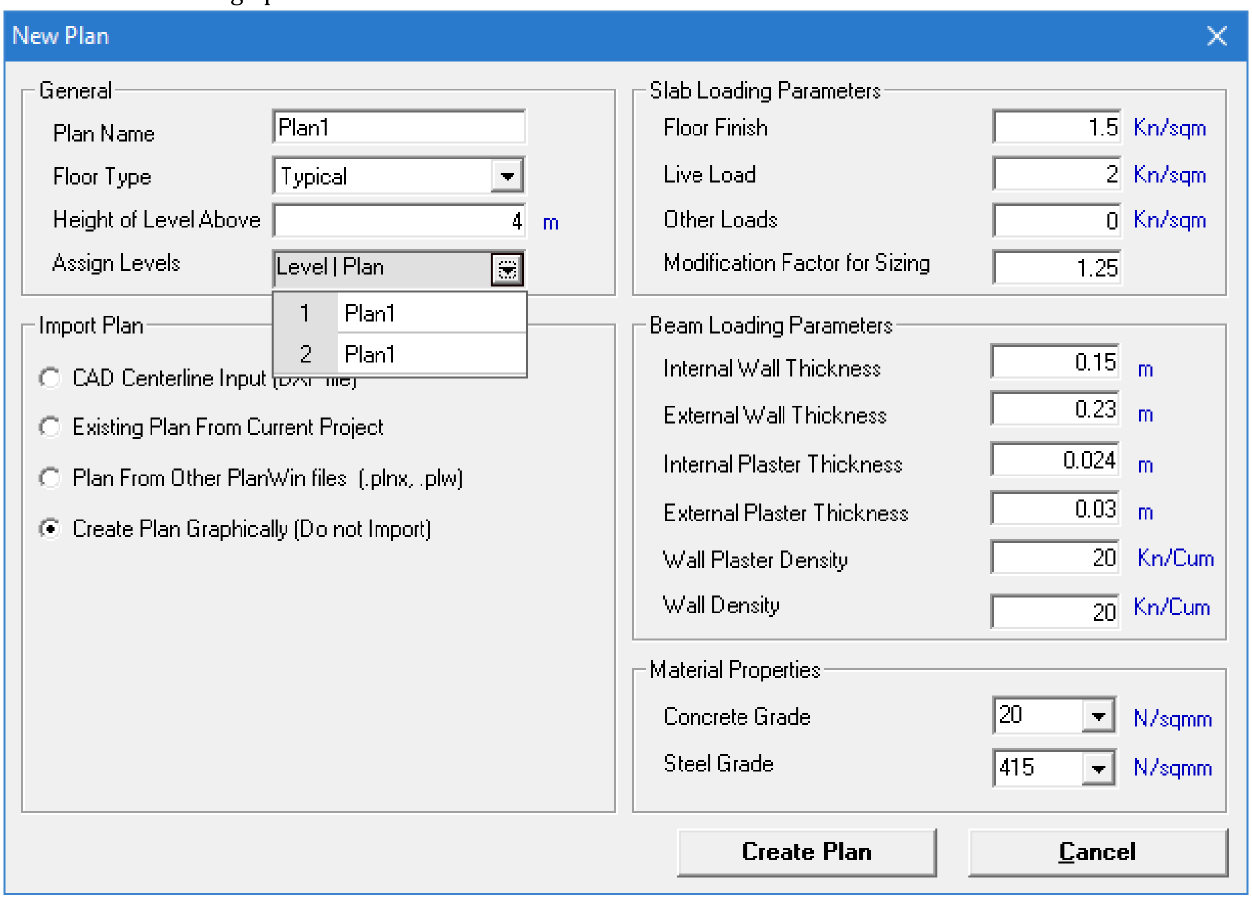
- *GS. Start Page* (on page 54)

M. Plans

Plans are building floor layouts (slabs, columns, and beams) which can be used for one or more floors in your building model. This allows you to rapidly model a building in which multiple floors have the same geometry, structure, and loads.

M. To Create a New Building Plan

1. Select **Workspace** > **New Plan**.
   The **New Plan** dialog opens.

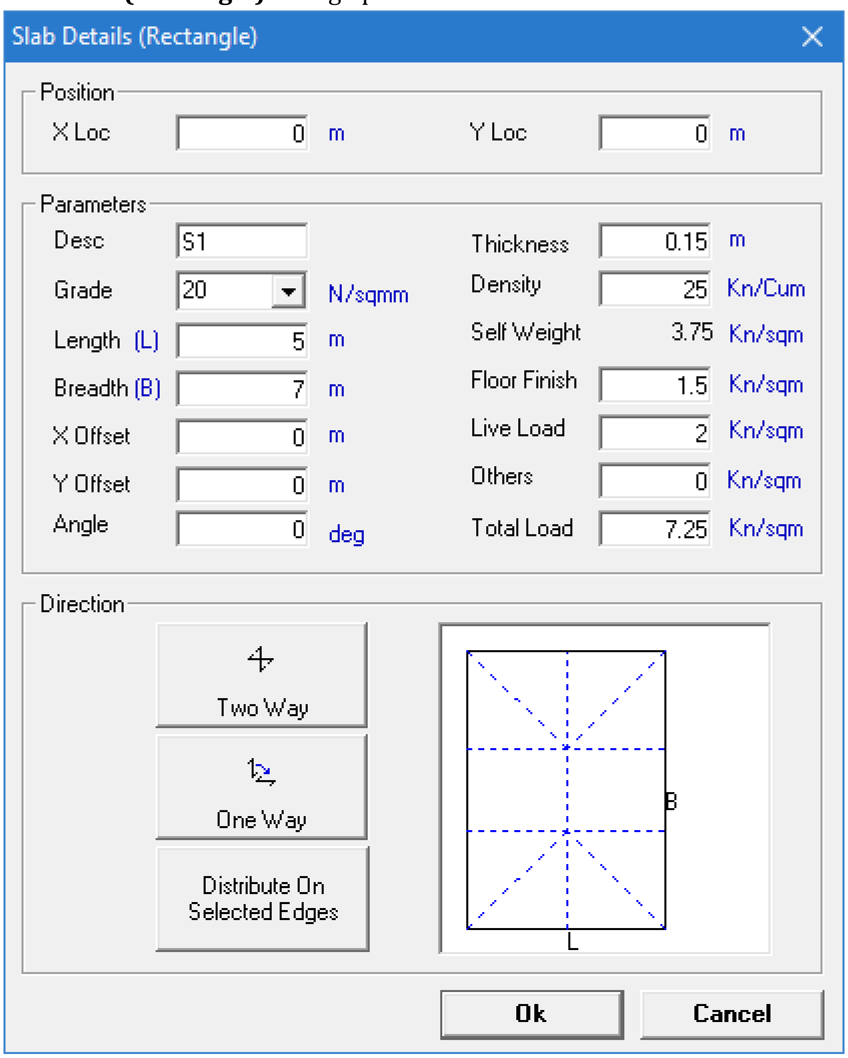


2. Specify the general details:
   a. Type a **Plan** name.
   b. (Optional) Select a **Floor type**
   c. Type the **Height of Level Above** (floor-to-floor height).
   d. Click the **Assign Levels** drop-down to assign plan names to each level of the building.

Note: The number of levels is specified in the **Start** dialog.

3. Select the **Create Plan Graphically (Do not import)** option to create a new plan

   Optionally, you may M. To Import a Building Plan (on page 923), a PlanWin file, or from another plan in this project (if there is another).
4. Specify the plan details and source (i.e., CAD file, drawing an plan in STAAD, or PlanWin plan).
5. Specify the loads on this plan:
   a. Enter the Slab Loading Parameters.
   b. Enter the Beam Loading Parameters.
6. Select the **Concrete Grate** and **Steel Grade** used in this plan.
7. Click **Create Plan**.
   If the plan was not assigned to any levels in Step 2d, then a warning message displays. Click **Yes** to proceed. Otherwise, the **Slab Details (Rectangle)** dialog opens.



8. Specify the slab details:
   a. Type coordinates for the Position

Tip: Leave default to accept origin.

  b. Type material, geometry, and loading parameters.
  c. Specify the direction the slab spans: Two Way, One Way, or Distribute on Selected Edges.

  Tip: The Length and Breadth values specified determine the recommended default direction.

9. Click **OK**.
   The slab is drawing in the graphical view.

M. To Import a Building Plan

The Building Planner can import plan data from other sources, such as PlanWin files or CAD drawings.

Note: The plan data must be in the correct format in a CAD drawing (`.dxf` file format). The correct units and scale must used in the drawing.

- Slab data must be in a layer named "Slab". Only closed polylines or light weight polylines can be imported.
- Column data must be in a layer named "Column". Use circles of any appropriate radius to mark the columns.
- Beam data must be in a layer named "Beam". Use lines to mark the beam centerlines.

1. Select **Workspace** > **New Plan**.

   If you have not yet specified project details, the **Start** dialog opens. Otherwise, the **New Plan** dialog opens.

   Note: Alternately, you can select **Edit** > **Import Plan File** to replace the current plan with imported plan data using the **Import Plan File** dialog.
2. Specify the general details:
   a. Type a **Plan** name.
   b. (Optional) Select a **Floor type**
   c. Type the **Height of Level Above** (floor-to-floor height).
   d. Click the **Assign Levels** drop-down to assign plan names to each level of the building.

   Note: The number of levels is specified in the **Start** dialog.
3. Select either:

   CAD Center line Input (DXF)

   or

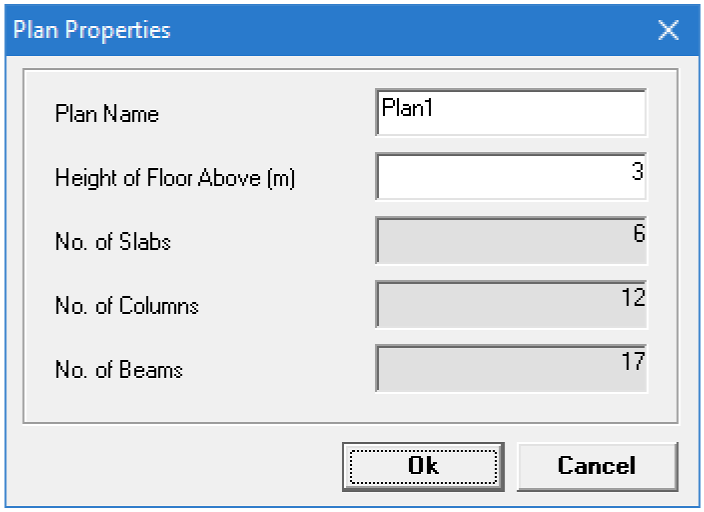
   Plan from PlanWin file

   or

   Existing plan from current project
4. Navigate to and select the file you want to use for either CAD or PlanWin file options and click **Open**.
5. If the selected PlanWin file has multiple plans, select the plan name you want to import.
6. Click **OK**.

M. To Edit Plan Properties

1. Select **View** > **Plan Properties**.
   The **Plan Properties** dialog opens.



2. Type a new **Plan Name** or **Height of Floor Above** value.
3. Click **OK**.

M. Slabs

Parametric slabs are used to model portions of physical slabs in a building floor. This allows you to model thickened slabs or other complex layout geometry easily.

M. To Add a Rectangular Slab

To add a rectangular slab to the current plan, use the following steps.

1. In a plan workspace, select **Slab** > **Create Slab Rectangle**.
2. Click a start and end corner for the slab in the graphics window.

   The start corner “snaps” to the nearest existing slab corner. You can input any corner using the X and Y offsets in t

   A gray rectangle connects the initial point to your mouse pointer to indicate the general direction of the slab. Exact slab dimensions are specified in the next steps, though.

   The **Slab Details (Rectangle)** dialog opens.
3. Specify the slab details:
   a. (Optional) Type **X Offset Y Offset**

   Tip: You may use negative numbers here.

   b. Type material, geometry, and loading parameters.
   c. Specify the direction the slab spans: **Two Way**, **One Way**, or **Distribute on Selected Edges**.

   Tip: The Length and Breadth values specified determine the recommended default direction.

   Tip: Cantilevered slabs can be specified using the **Distribute on Selected Edges** option and clicking a single, supporting edge.
4. Click **OK**.

The slab is drawn in the Plan | Slab page graphical view.

Tip: If the slab has been drawn beyond the extents of the window, right-click anywhere *not* on a slab and select **Zoom Extend** from the pop-up menu.

Describe the results of the action.

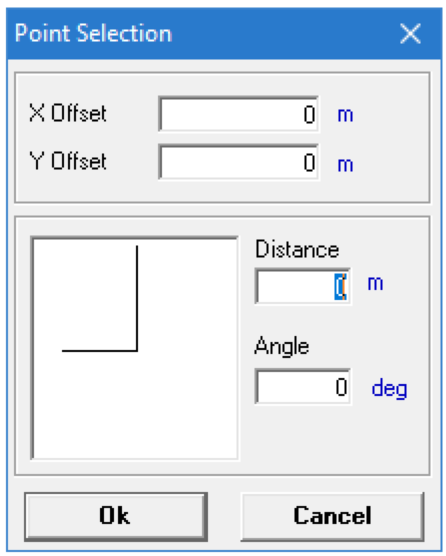
5. Repeat Steps 2 through 4 as necessary to continue drawing slabs.

Tip: Press **<Esc>** to quit using this tool.

M. To Add an Irregular Shape Slab

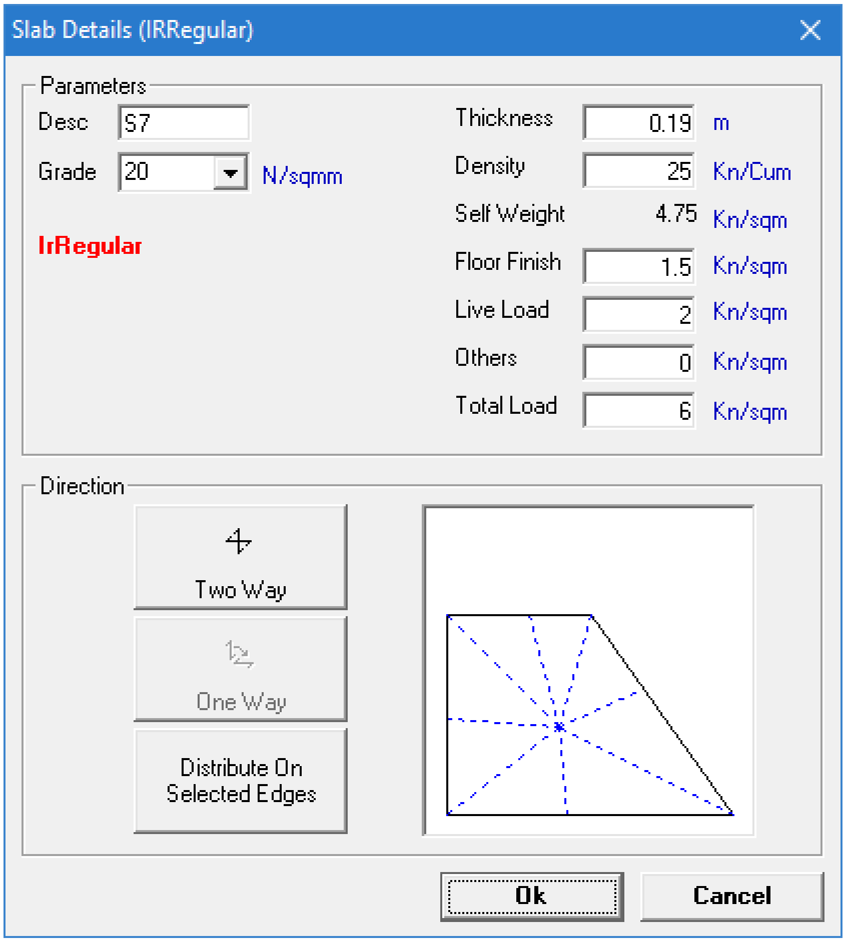
To add an irregularly shaped slab to the current plan, use the following steps.

1. In a plan workspace, select **Slab** > **Create Irregular**.
2. Click an initial corner for the slab.
3. Click another corner in either a clockwise or counterclockwise progression from the previous corner.
   The **Point Select** dialog opens.



Tip: The **Point Selection** dialog will not open if the program detects you have indicated an existing slab corner.

4. (Optional) Specify an X Offset and Y Offset value (relative to the point highlighted in a red circle), Distance, and Angle value to place the point at another location.
5. Click **OK**.
6. Repeat steps 3 through 5 to select each additional vertex point on the slab.
7. Right-click to stop adding corners.
   The **Slab Details (Irregular)** dialog opens.



8. Specify the slab details:
   a. Type label, material, geometry, and load parameters.
   b. Specify the direction the slab spans: **Two Way**, **One Way**, or **Distribute on Selected Edges**.

   Tip: The resulting dimensions determine the recommended default direction. If one or more slab spanning directions is determined to be unsuitable based on the dimensions, it is disabled.

   Tip: Cantilevered slabs can be specified using the **Distribute on Selected Edges** option and clicking a single, supporting edge.

9. Click **OK**.
   The slab is drawn in the Plan | Slab page graphical view.
10. Repeat Steps 2 through 9 to continue drawing irregularly shaped slabs.

Tip: Press **<Esc>** to quit using this tool.

M. To Edit Slab Properties

To edit the parameters of an existing slab, use the following steps.

Note: The label or material and load parameters of a slab may be edited. To edit they geometry, you must deleted and redraw a slab.

1. In a plan workspace, select **Slab** > **Select/Unselect**.

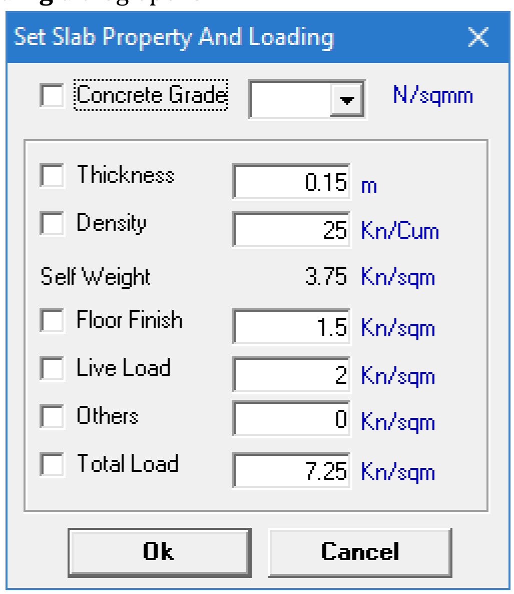
   Tip: You can use some of the other selection tools to refine your selection of slabs as necessary.
2. Select a slab in the graphical view.

   Tip: Hold **<Ctrl>** to select multiple slabs.
3. Either:

   Select **Slab** > **Set Property and Loads for Selected Slab/s**

   or

   Right-click and select **Set Property and Loads for Selected Slab/s** from the pop-up menu.
   The **Set Slab Property and Loading** dialog opens.



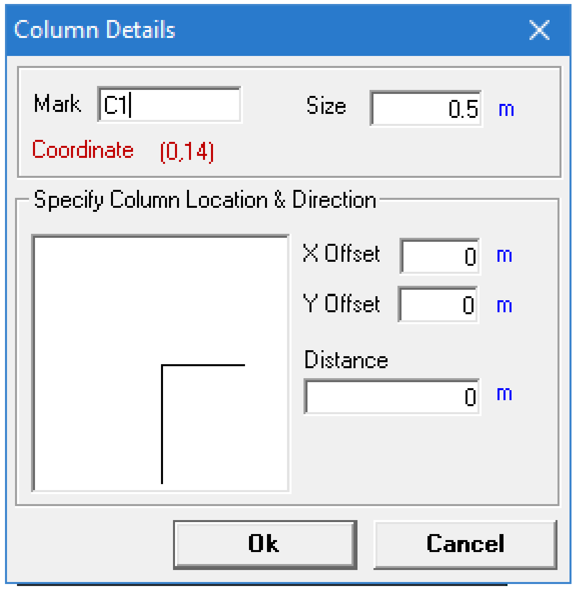
4. Check the options for each parameter you want to edit for all selected slabs and then type or select the value.
5. Click **OK**.

M. Columns

M. To Add a Column

To add a single column, use the following steps.

1. In a plan workspace, select **Column** > **Create Column**.
2. Click the nearest slab corner or edge where you want to place a column.
   The nearest slab corner is highlighted with a red circle. The **Column Details** dialog opens.



3. Type column details:
   a. (Optional) Type a custom **Mark**

   > **Note:** This must be unique to the plan.

   b. Type a face **Size** (square column).

      All initial column sizes are assumed to be square. Columns can later be resized manually or designed.

   c. (Optional) Type an **X Offset Y Offset**

      The coordinates of the column are displayed for reference.

   d. (Optional) Type an **Angle Distance**
4. Click **OK**.
   The column is drawn in dark blue on the graphical view.
5. Repeat Steps 2 through 4 to continue placing columns.

M. To Add Columns Automatically to the Entire Plan

To place columns automatically at detected points on the entire plan, use the following steps.

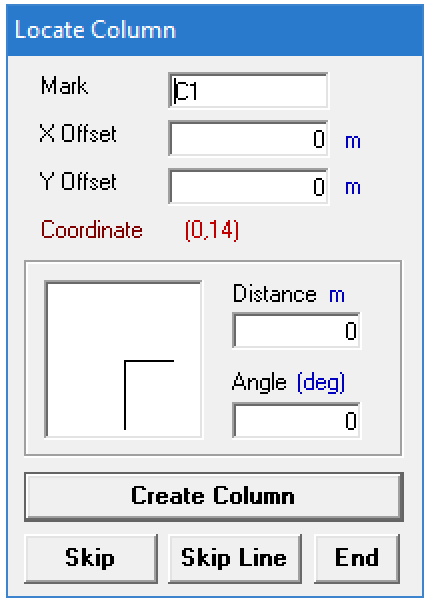
1. In a plan workspace, either:

   select **Column** > **Auto Column**, or

   or

   select **Column** > **Locate Column**.

   Columns are added at all slab corners on the floor, unless you selected the Locate Column option. In that case, the first detected column location is selected and the Locate Column dialog opens.



2. (Optional) If you selected the Locate Column feature, either:

   Specify the current column mark, offset, and orientation, or

   or

   click **Skip** to skip to the next detected column location, or

   or

   click **Skip Line** to skip all the detected locations in the current column line, or

   or

   click **End** to stop adding columns.
3. Repeat Step 2 as needed until you have reached the end of the detected column locations.

If you used the auto column feature or if you neglected to skip a location, you may need to delete any unwanted columns (e.g., those added at the end of a slab intended to cantilever), move columns, or edit the parameters of columns which are different than the default values for the plan.

M. Beams

M. To Create a Beam

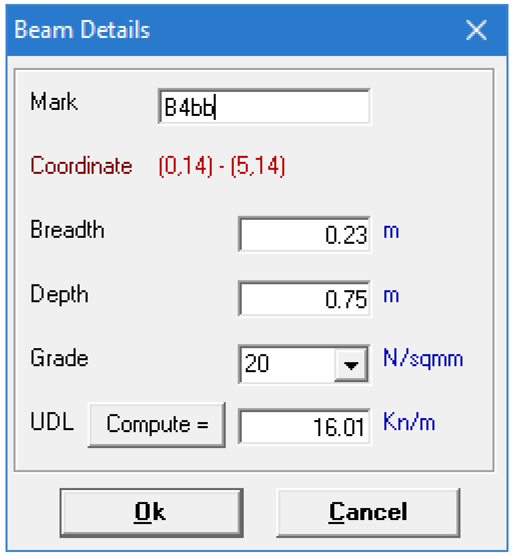
To add a single beam, use the following steps.

> **Note:** The **Auto Beam** tool will delete any existing beams on the plan. You may want to use use the Auto Beam feature prior to drawing individual beams.

1. In a plan workspace, select **Beam** > **Create Beam**.
2. Select the start and end support points for the beam.

If beams are drawing over columns or existing beams, the new beam will be broken into separate objects once placed.

The nearest slab corners are highlighted with red circles. The **Beam Details** dialog opens.

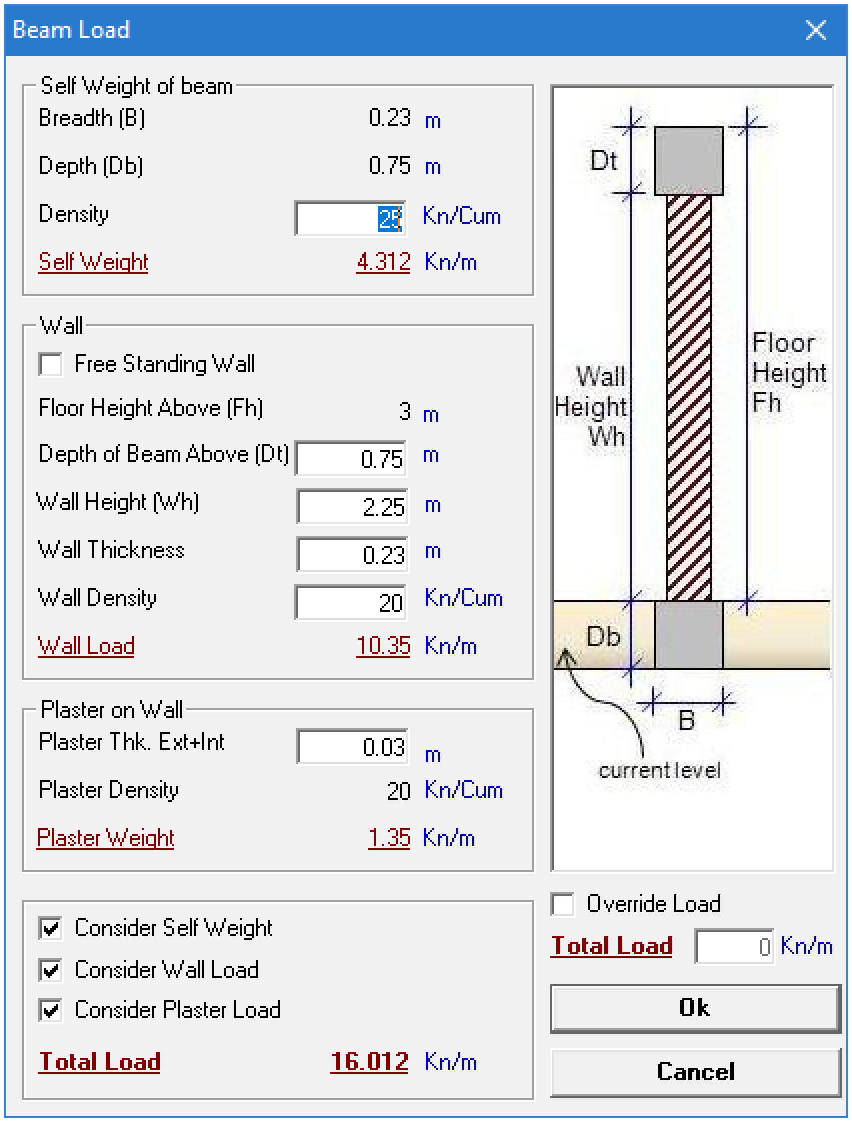


3. Type the beam details:
   a. (Optional) Type a custom **Mark**

   > **Note:** This must be unique to the plan. If you are drawing multiple beams over columns or existing beams, the mark will be incremented for the additionally created beams.

   b. Type a **Breadth** and **Depth** for the beam size.
   c. Select the **Grade** of material.
   d. (Optional) To include the uniform dead load for the beam, type a value or click **Compute Beam Load**

   > **Note:** If Compute is used, the Beam Load dialog opens with the specified beam dimensions. You can specify a material Density and include options for walls supported by the beam.



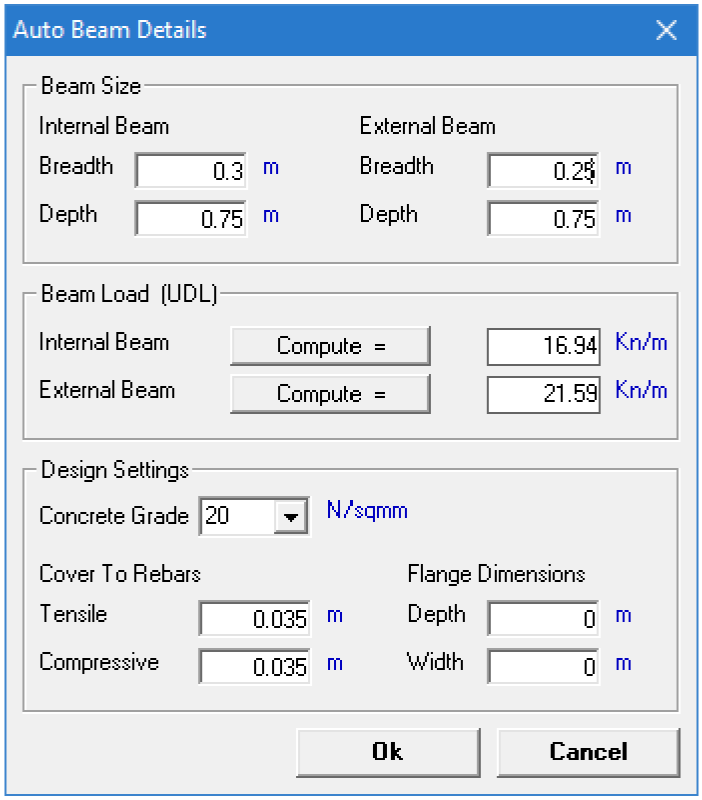
4. Click **OK**.
   The beam is drawn in green on the graphical view.
5. Repeat Steps 2 through 4 to continue placing beams.

M. To Add Beams Automatically to an Entire Plan

To place beams automatically at detected points on the entire plan, use the following steps.

1. In a plan workspace, select **Beam** > **Auto Beam**.
   If there are any existing beams, a warning message that they will be deleted opens.
2. Click **OK**.

   The **Auto Beam Details** dialog opens.

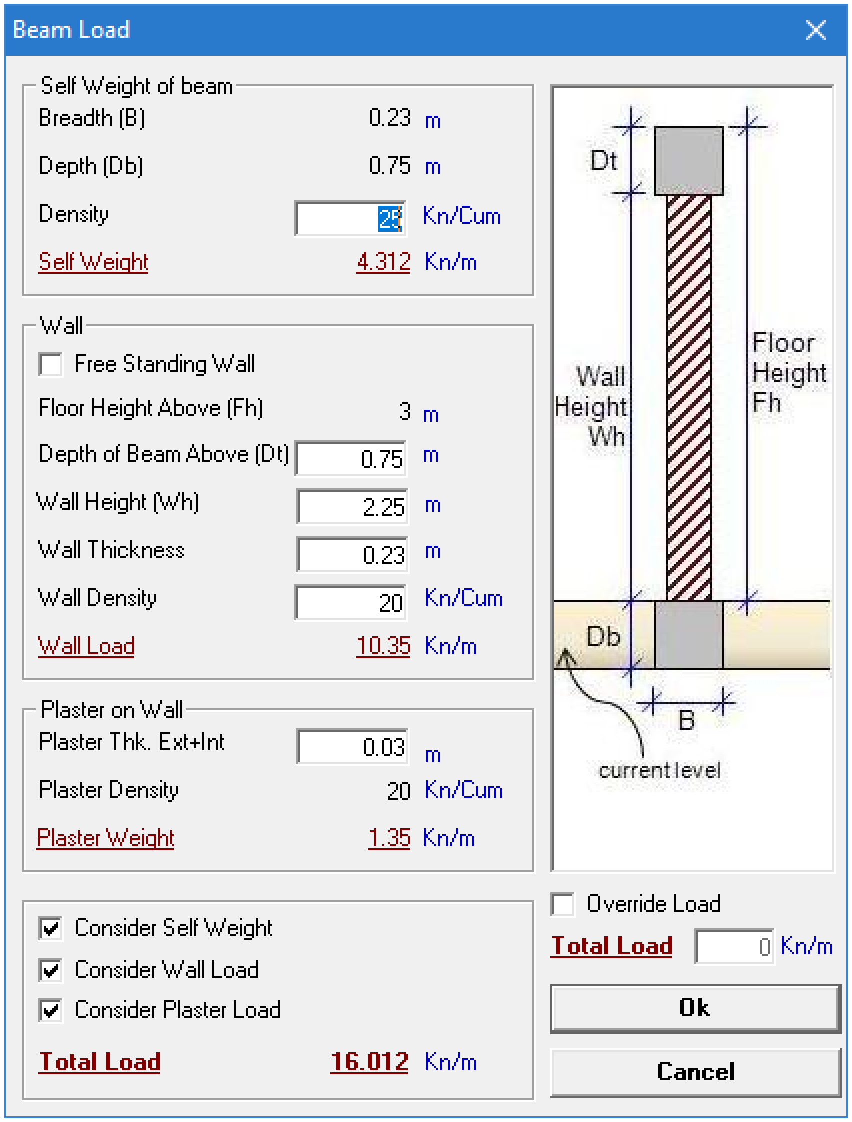


3. Type the beam details:

Note: The internal and external (exterior slab edge) beams can be specified separately.

- **a.** Type a **Breadth** and **Depth** for the beam sizes.
- **b.** (Optional) To include the uniform dead load for the beam, type a value or click **Compute Beam Load**

Note: If Compute is used, the Beam Load dialog opens with the specified beam dimensions. You can specify a material Density and include options for walls supported by the beam.



c. Select the **Grade** of material.

d. Type the **Cover to Rebars** and **Flange Dimensions** used for design.

4. Click **OK**.
   Beams are added at all slab edges on the floor.

M. To Specify Beam Continuity

To specify continuous beams over supports, use the following steps.

Building Planner uses a window to specify continuous beams for the entire plan. It is typically best practice to draw all beams for the plan before performing this procedure.

1. Select **Beam** > **Modify Beam Continuity**.
   The **Continuous Beam** window opens.
2. You can specify continuous beams individual for finer control or allow the program to autodetect all continuous beams:

To...	Select...
select individual beam segments to create continuous beams manually	**Modify Data** > **Mark/Unmark**
allow the program to detect all continuous beams automatically	**Modify Data** > **Auto Mark Continuous Beam**

If you used the auto mark option, then all beams that the program detects as having possible continuity over supports are marked in light purple. Proceed to Step 5.

3. Click a series of beams you want to mark as continuous.
   Each beam selected is highlighted in light red. You can click a beam segment again to unmark it.
4. Select **Modify Data** > **Create**.
   The selected beam segments are marked in light purple.
5. Repeat Steps 2 through 4 as necessary to continue to mark individual beam lines as continuous.
6. When you are finished marking continuous beams, either:

   click the **X** in the top,right-corner of the **Continuous Beam** window, or

   or

   press **<Alt+F4>**

If you have marked beams as continuous, you must finalize the plan including continuity.

M. To Edit Beam Properties

To change beam properties, use the following steps.

> **Tip:** If the **Beam Loading Errors** window indicates that beams intended to be cantlievered are not supported, simply right-click on the beam and select **Cantilever** from the pop-up menu.

1. Select **Beam** > **Select/Unselect**.

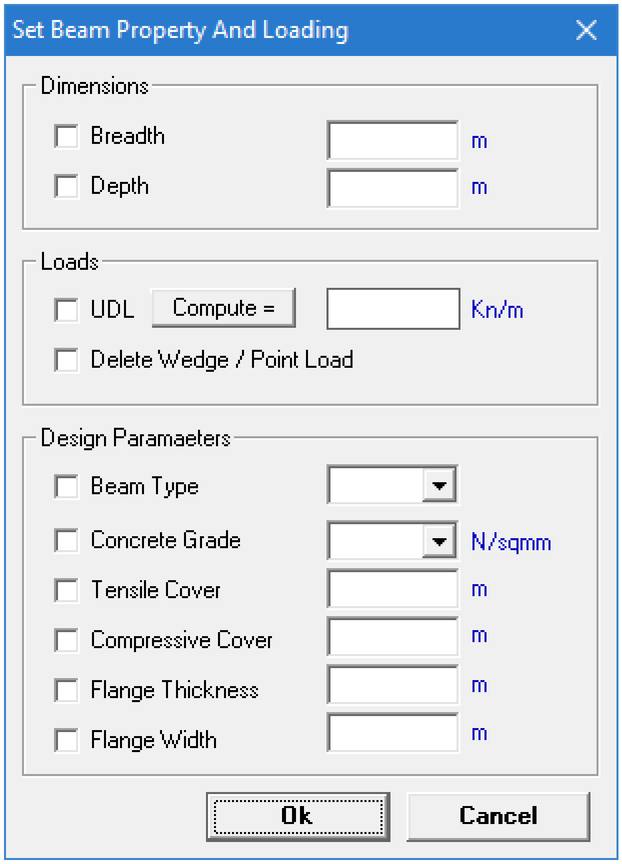
   > **Tip:** You can use some of the other selection tools to refine your selection of beams as necessary.
2. Select a beam in the graphical view.

   > **Tip:** Hold **<Ctrl>** to select multiple beams.
3. Either:

   select **Beam** > **Set Property and Loads for Selected Beam/s**, or

   or

   right-click and select **Set Property and Loads for Selected Beam/s** from the pop-up menu
   The **Set Beam Property and Loading** dialog opens.



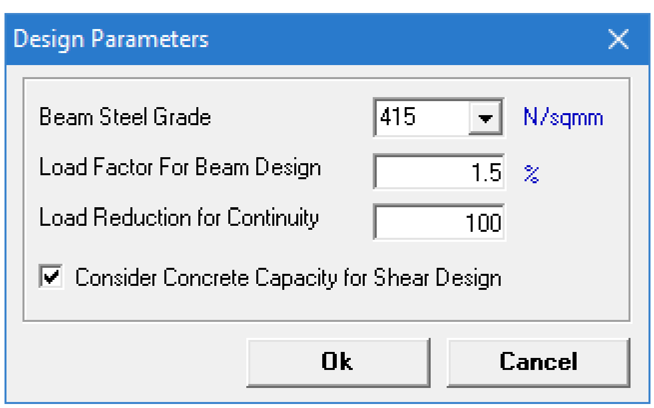
4. Check the options for each parameter you want to edit for all selected beams and then type or select the value.
5. Click **OK**.

M. To Specify Beam Design Parameters

To control some of the various design parameters, use the following procedure.

1. In a plan workspace, select **Analysis** > **Design Parameters**.

   The **Design Parameters** dialog opens.



2. Make changes to the design parameters as necessary.

3. Click **OK**.

M. Frames

M. To Finalize a Plan

To finalize a plan for use in a building frame, use the following steps.

In order to complete a 3D frame model using the plan data, you must first finalize the layout of a plan. if you select the Frame page without having finalized plans, a warning message is displayed.

1. In the plan workspace, either:

   select **Analysis** > **Finalize Plan (with Continuity)**, or

   or

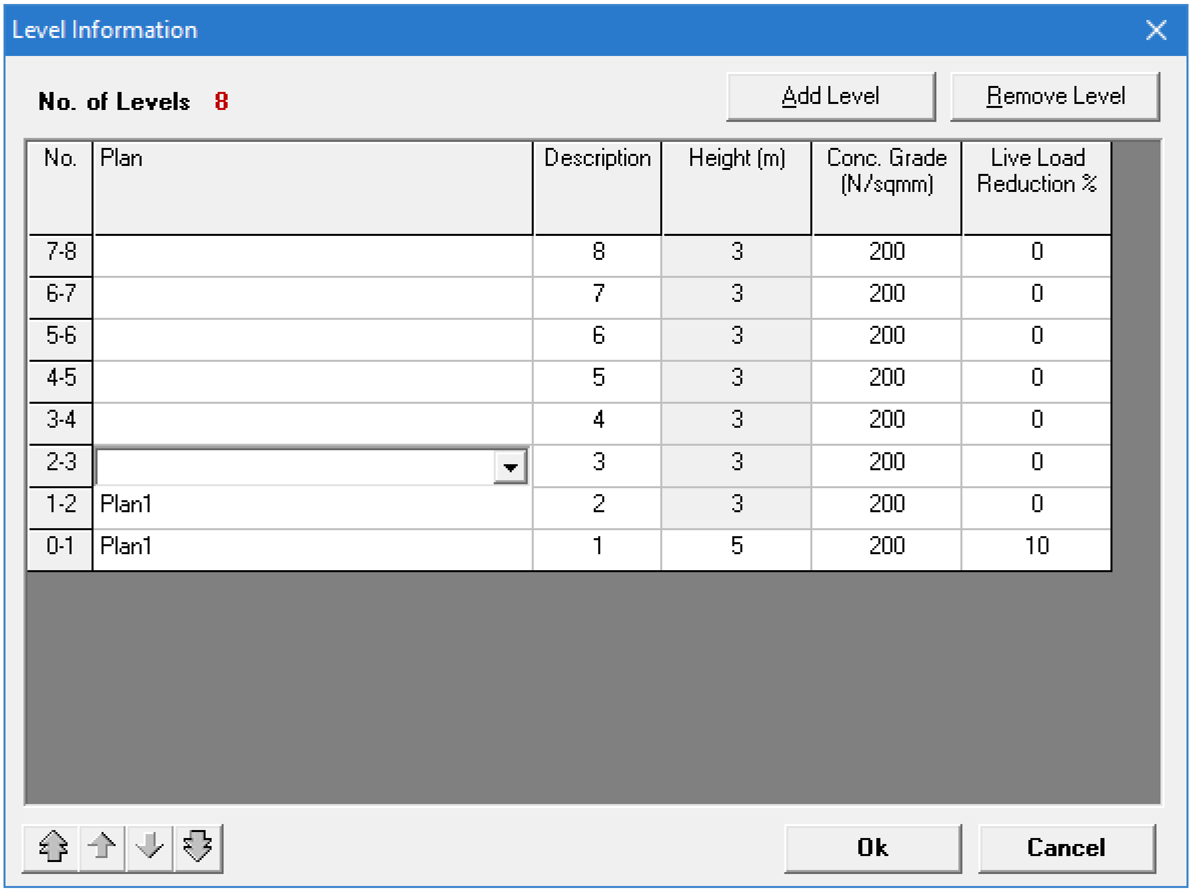
   select **Analysis** > **Finalize Plan (without Continuity)** to ignore continuity between beams, or
   A message dialog opens to indicate the plan was finalized. If any errors are detected, they will be highlighted in a pop-up window.

If any changes are made to the plan, you must repeat this procedure.

M. To Create a 3D Frame

In order to create or edit 3D frames, any changes to plans must first be finalized.

1. In the 3D Frame workspace, select **Assign** > **Level Properties**.
   The **Level Information** dialog opens.



2. Click **Add Level** to add another level and then click **OK** to confirm.
3. For each level, click in the Plan column to select a plan name from the drop-down list.

   Tip: Use the autofill arrows below the table to copy the plan name to rows above or below a selected row, or to all rows above or below.

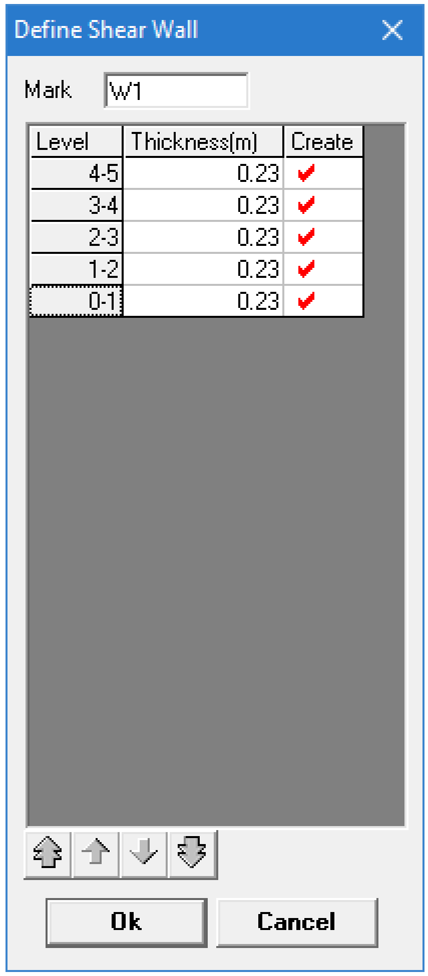
4. Click **OK**.
   The 3D Frame is created.
5. (Optional) Select **View** > **3D View**.

M. To Create a Shear Wall

Shear walls created by the Building Planner are analytically columns with the dimensions of the wall they represent.

1. Select **Assign** > **Define Shear Wall**.
2. Click the columns or slab corners to define the start and end of the shear wall in plan.

   The points are highlighted with gray-filled, red circles in the view window. The **Define Shear Wall** dialog opens.



3. Specify the shear wall parameters: as necessary
   a. Type a Mark to label the wall.
   b. If a shear wall is *not* continuous from base to roof, clear the check mark in the Create column for that level.
   c. Type an X Offset and Y Offset value to move the wall with respect to the selected start point.
   d. Type B and D dimensions to specify a wall size.
4. Click **OK**.
   The shear wall is drawn in dark red in the plan.
5. Repeat Steps 2 through 4 as necessary to continue adding shear walls.

Tip: Press **<Esc>** to quit using this tool.

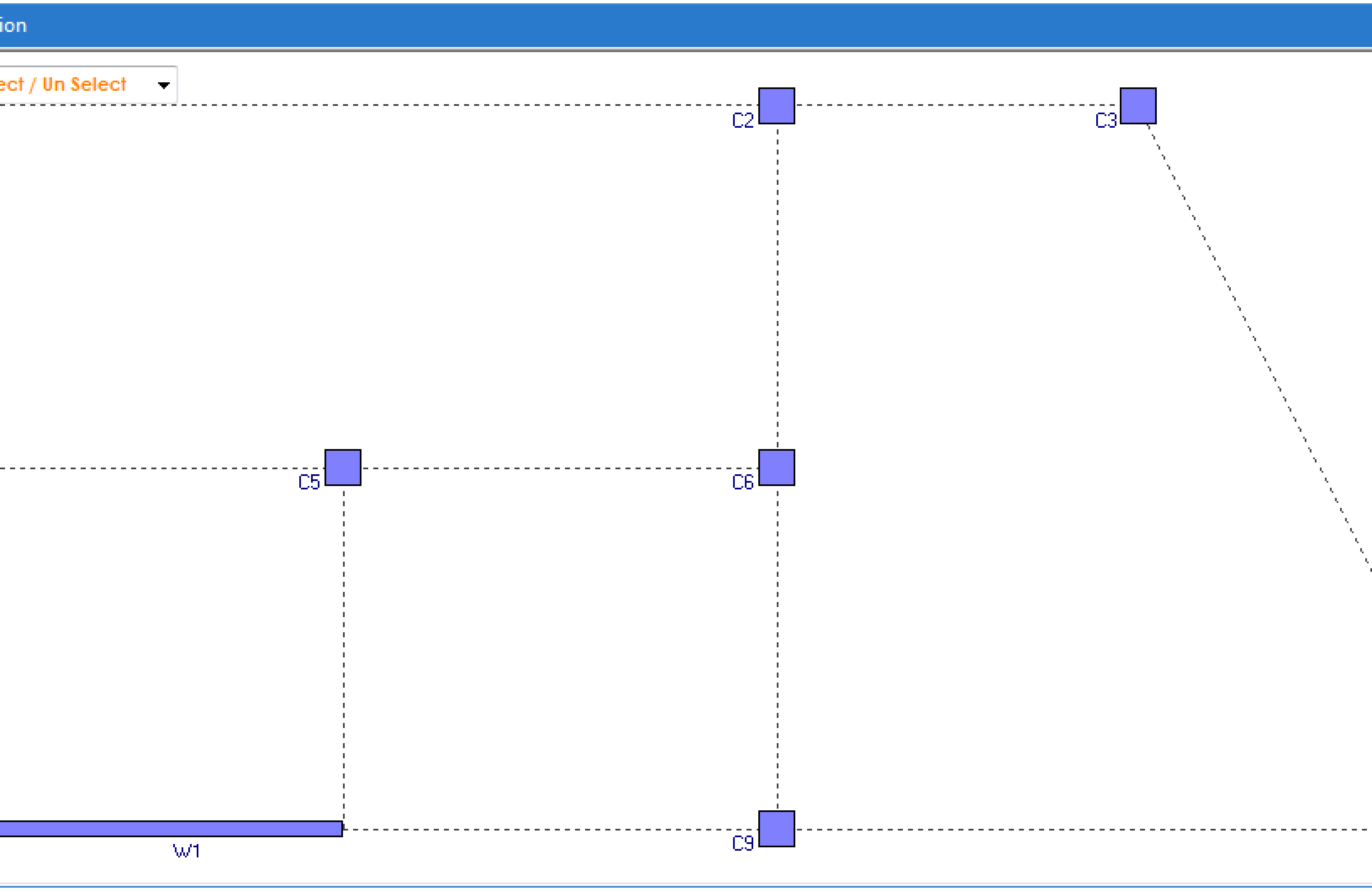
M. To Change Supports

To change boundary conditions for the 3D frame, use the following steps.

The supports are at each column or wall base at the first (lowest) level in the building frame. Initially, all supports are assumed as Fixed.

1. Select **Assign** > **Support Specification**

The **Support Specification** dialog opens.

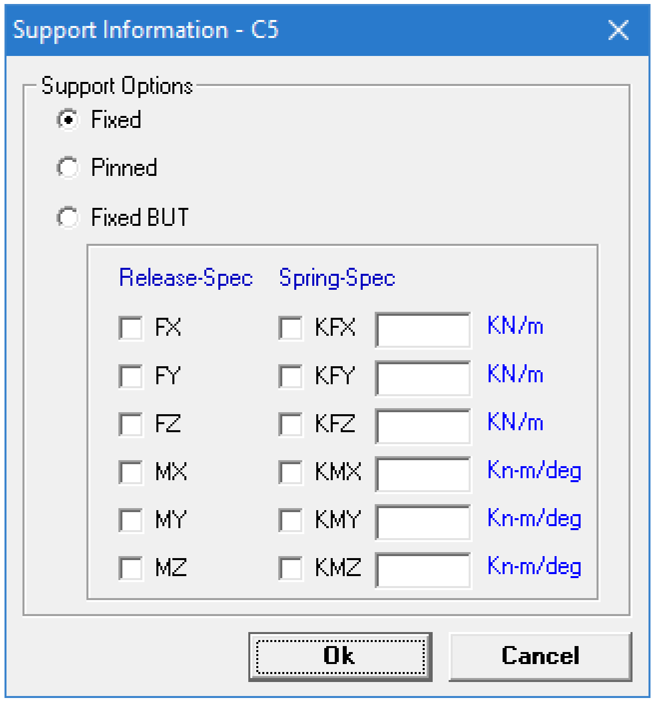


2. Select the one or more columns or walls.

Tip: Use the Selection drop-down list to change the selection method.

3. Right-click on a column or wall and select **Support Condition -** ***mark*** from the pop-up menu.

   The **Support Information** dialog opens.



4. Select the **Support Options** to use.
5. For **Fixed But** option, either:

   specify the **Release-Spec**

   or

   Spring-Spec and spring constant for each degree of freedom as necessary.
6. Click **OK**.
7. Repeat Steps 3 through 6 for each support you want to change.

Tip: The support type is color-coded in the **Support Specification** dialog.

8. Click the **X** in the top,right-hand corner of the **Support Specification** dialog when you are finished making changes to the supports.
   The **Support Details** table updates with the changes.

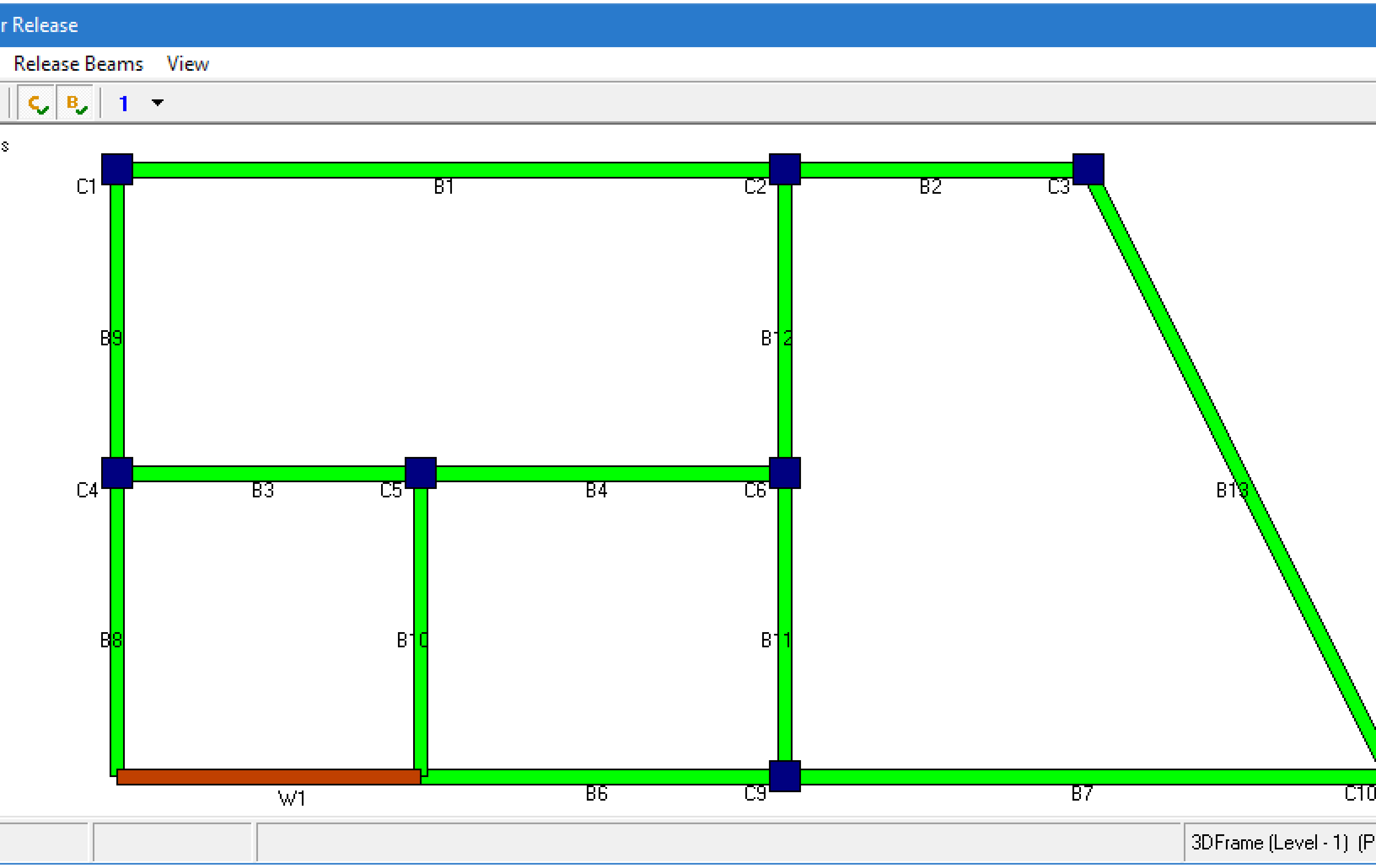
M. To Specify Member Releases

To change member end release details for the 3D frame, use the following steps.

All beams are initially assumed as fully fixed at both ends.

1. Select **Assign** > **Member Release (Beams)**.

   The **Member Release** dialog opens.



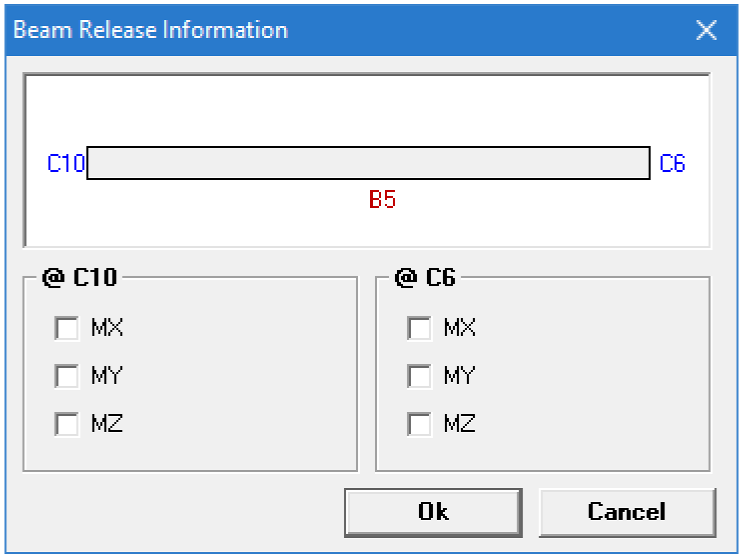
2. (Optional) Select the level from the drop-down list of level numbers in the tool bar.

   Tip: The current level and plan name are displayed in the status bar.

3. Right-click on a beam and select **Release Details-** ***mark*** from the pop-up menu.

   Tip: The **Member Release** dialog contains some tools to select and make release changes to multiple members, such as all primary or secondary beams or all beams along a column line. These tools allow you to select the levels to which you want to apply these release specifications as well.

   The **Beam Release Information** dialog opens.



4. Check the boxes for each degree of freedom to release at either end of the member and type a percent release value for each release direction.

    Tip: The default release of 99% is intended to represent a full member release.

5. Click **OK**.
6. Repeat Steps 2 through 6 for each member you want to edit release specifications.

    Tip: The member release specifications are color-coded in the **Member Release** dialog and indicated with dots on each member.

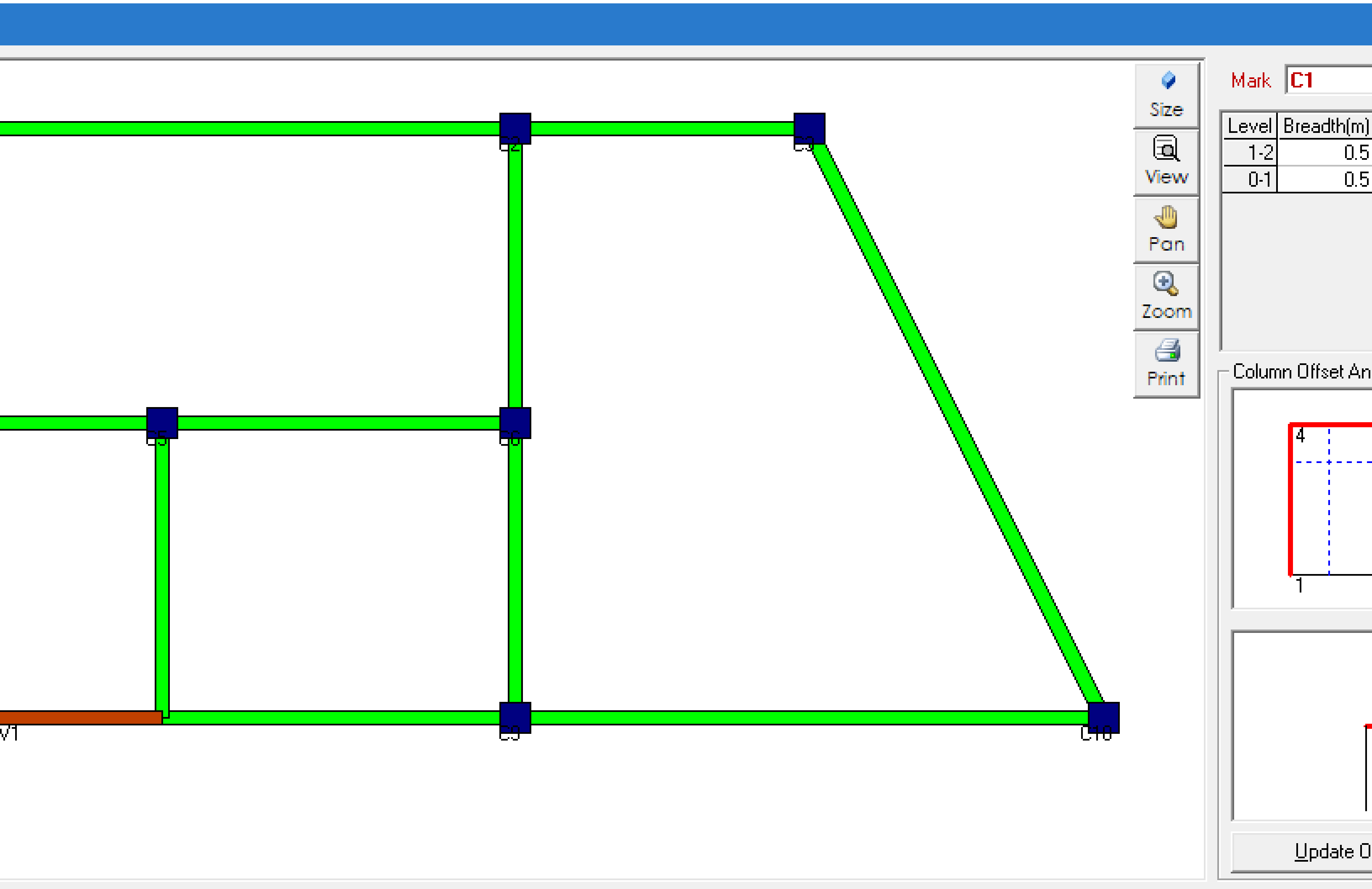
7. Click the **X** in the top,right-hand corner of the **Member Release** dialog when you are finished making changes to the member releases.
    The **Beam Release Details** table updates with the changes.

M. To Change Column Size, Orientation, and Alignment

To change a column size and orientation, use the following steps.

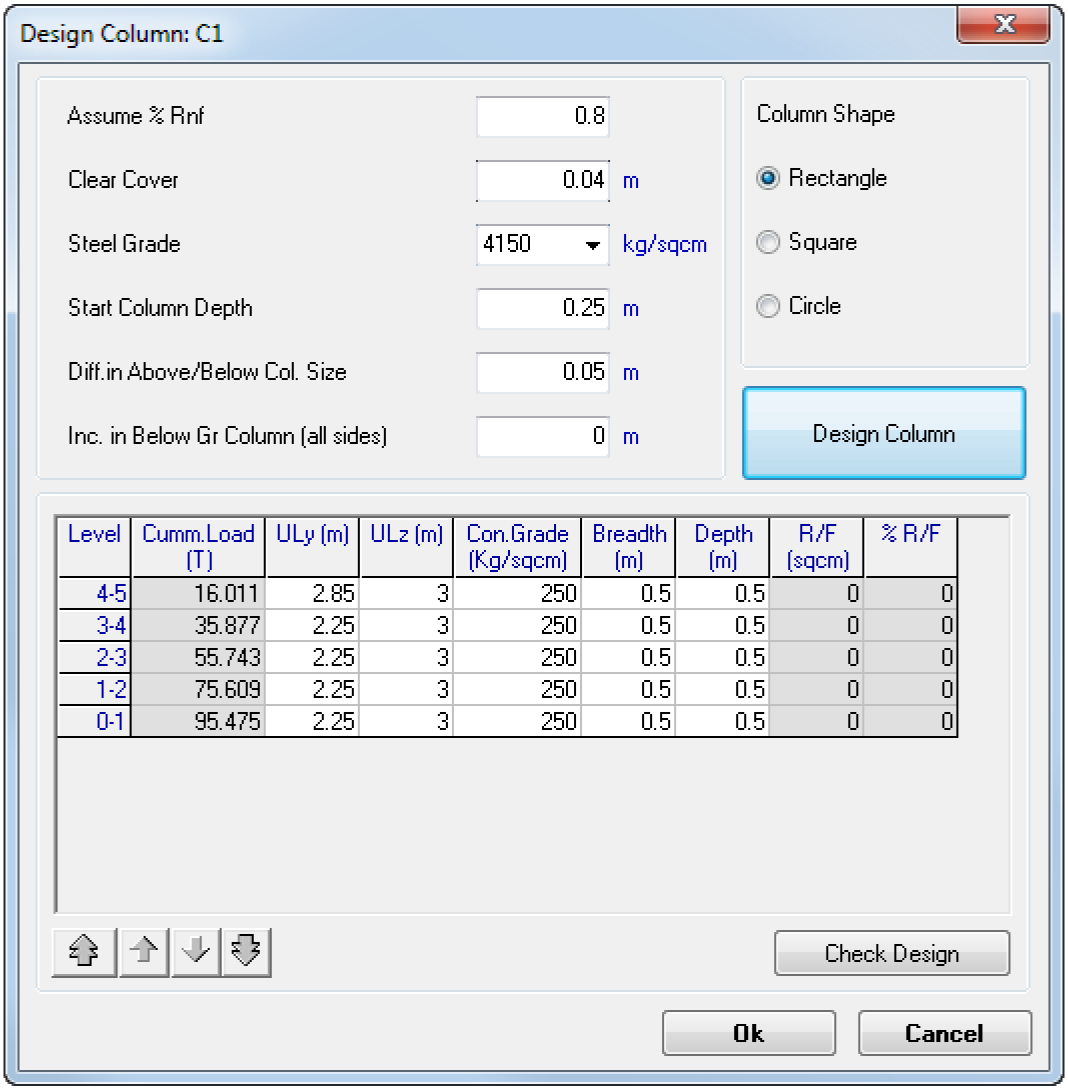
1. Select **Assign** > **Column Sizing and Orientation**.

    The **Column Sizing** dialog opens.



2. Select a column in the main graphical view.
   The selected column is highlighted in a red circle and the Mark is displayed. The column orientation and offset are displayed in the right-hand views.
3. To change the column orientation (i.e., Beta angle):
   - **a.** Click a different axis of orientation in the lower, right-hand graphic or click **+90** to rotate the column 90°
   - **b.** Click **Update Offset Data & Beta Angle**.
4. To change the column offset:
   - **a.** Click one of the column faces in the middle, right-hand graphic.
   - **b.** Type an offset value in the indicated units.
   - **c.** Repeat steps 4a and 4b to specify offsets in a different axis.
   - **d.** Click **Update Offset Data & Beta Angle**.
5. To change a column size:
   - **a.** Click **Re-Size Column.**

     The **Design Column: <mark>** dialog opens.



b. Type initial design parameters in the fields.

> **Tip:** You can use the **Check Column** feature to perform a design check on the current column size.

c. Select a **Column Shape**.

d. Click **Design Column**.
The column size (Breadth and Depth) at each floor are updated based on the design parameters and specified building loads.

e. (Optional) You can manually type changes to the table values to further refine the design.

> **Tip:** Use the automatic fill buttons below the table to copy values up or down the currently selected row.

f. Click **OK**.
The column diagrams are updated in the **Column Sizing** dialog.

6. Repeat Steps 2 through 5 for each column you want to change.
7. Click the **X** in the top,right-hand corner of the **Column Sizing** dialog when you are finished making changes to the column sizes and orientation.
The **Beam Release Details** table updates with the changes.

M. To Modify Wind Parameters

To specify parameters for lateral wind loads, use the following steps.

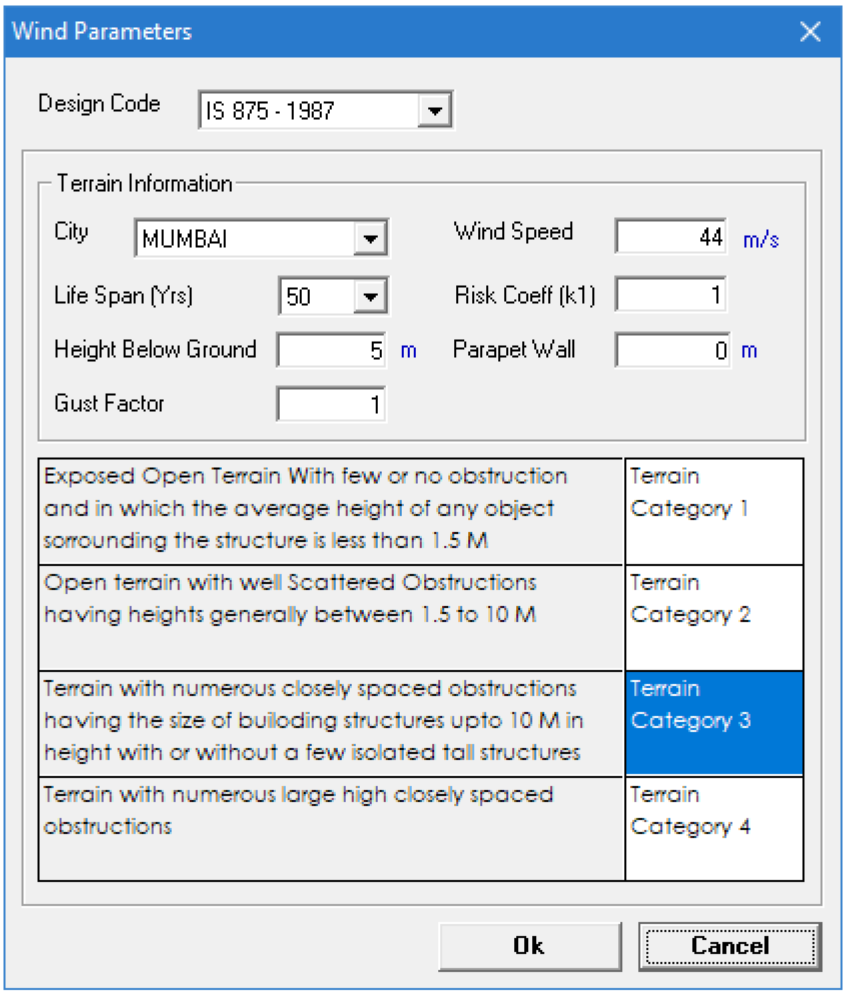
The wind load parameters are input per the following codes:

- UBC 97
- IS 875-1987
- ASCE7

1. Select **Assign** > **Wind Parameters**.

   Tip: You can also double-click any of the wind load parameters in the **Lateral Load** dialog.

   The **Wind Parameters** dialog opens.



2. Select the **Design Code** from the drop-down list.
3. Either:

   Select the appropriate **City** in the drop-down list (recommended)

   or

Type a **Wind Speed** value directly.

4. Either:

   Select the **Life Span (yrs)** for which the building is to be designed from the drop-down list (recommended)

   or

   Type a **Risk Coeff (K1)** value directly.
5. Type **Height Below Ground** and **Parapet Wall** values, in the units indicated.
6. Type a **Gust Factor** value.
7. Select the Terrain Category from the description list.
8. Click **OK**.

You can now review the calculated wind loads by selecting **Frame** > **Wind Effect**. In the Wind Effect dialog, select the different directions to see the distributed wind intensity on each floor. Selecting a column in the table displays the wind intensity values at the top and bottom of each floor along that column.

M. To Modify Seismic Parameters

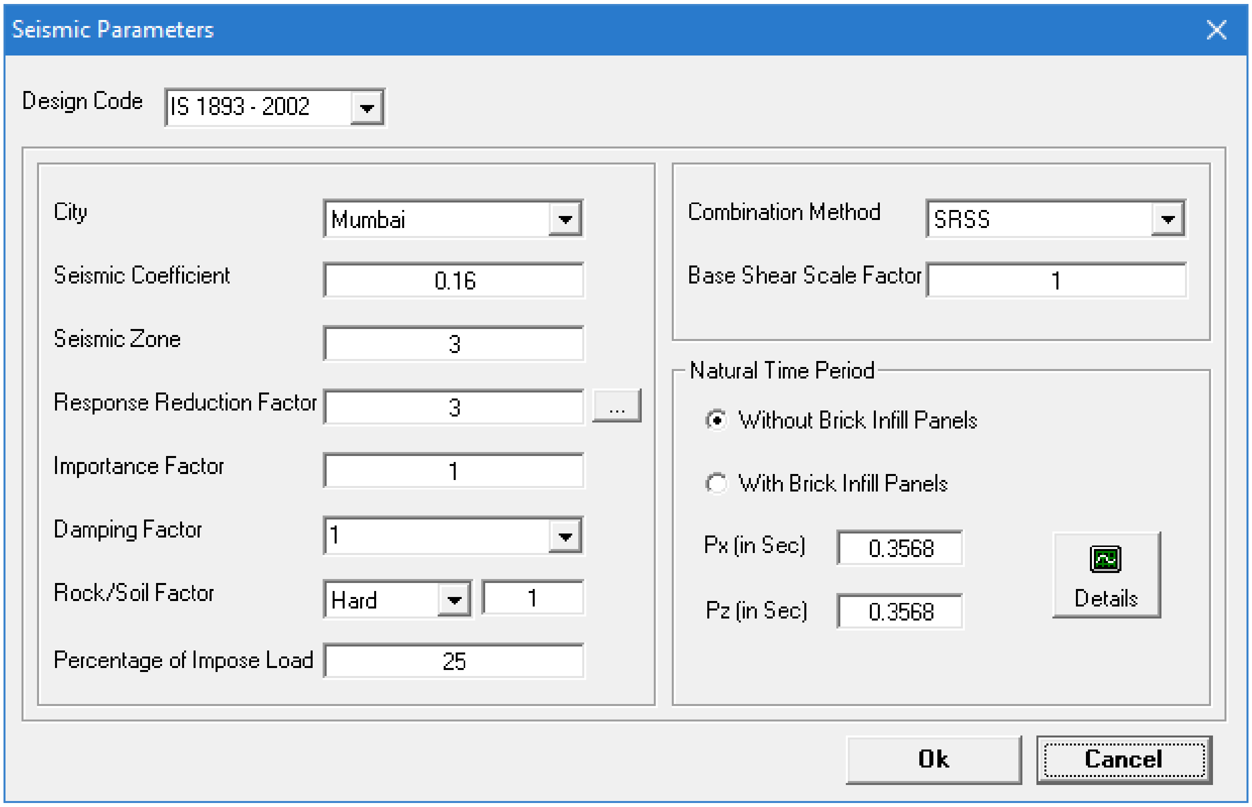
To specify parameters for lateral seismic loads, use the following steps.

The seismic load parameters are input per the following codes:

- UBC 97
- IS 1893-2002

1. Select **Assign** > **Seismic Parameters**.

   The **Seismic Parameters** dialog opens.



2. Either:

   Select the appropriate **City** in the drop-down list (recommended)

   or

   Type **Seismic Coefficient** and **Seismic Zone** values directly.
3. Specify a Response Reduction Factor for the building:
   a. Click **[...]** adjacent to the Response Reduction Factor field.
      The Response Reduction Factor dialog opens.
   b. Select the appropriate row in the **Building Frame Systems** description list.
   c. Click **OK**.
      The Response Reduction Factor is updated.
4. (Optional) Specify or select additional seismic parameters as necessary:
   a. Type an **Importance Factor** value.
   b. Type a **Damping Factor** value.
   c. Select the type of soil on site from the **Rock/Soil Factor** drop-down list.
   d. Select the **Combination Method** from the drop-down list.
   e. Type the **Base Shear Scale Factor** value.
5. Select if brick infill panels are present in the building structure to determine the **Natural Time Period**.

   Tip: Click **Details** to see the values used in evaluating Px and Pz.
6. Click **OK**.

M. To Modify the Load Combinations

The Building Planner mode uses building combinations per the Indian code by default. You can modify, remove, or add individual load combinations as necessary.

1. Select **Tools** > **Load Combination**.
   The **Loading Combination** dialog opens.
2. Make any changes to the load combinations necessary:

To…	Do the following…
change the load factor on a load type for a particular load combination	type the new load factor in the corresponding cell
add a new load combination	click **Add** and type the load factors for each load type
remove an existing load combination	select the load combination row and click **Remove**
set the current load combinations as the default in Building Planner mode	click **Set as Default**
restore the default load combinations (i.e., the code load combinations)	click **Load Default Combinations**

3. Click **Update**.
4. Click **[X]** to close the **Loading Combination** dialog.

M. Analysis and Design

M. To Generate a STAAD.Pro Model

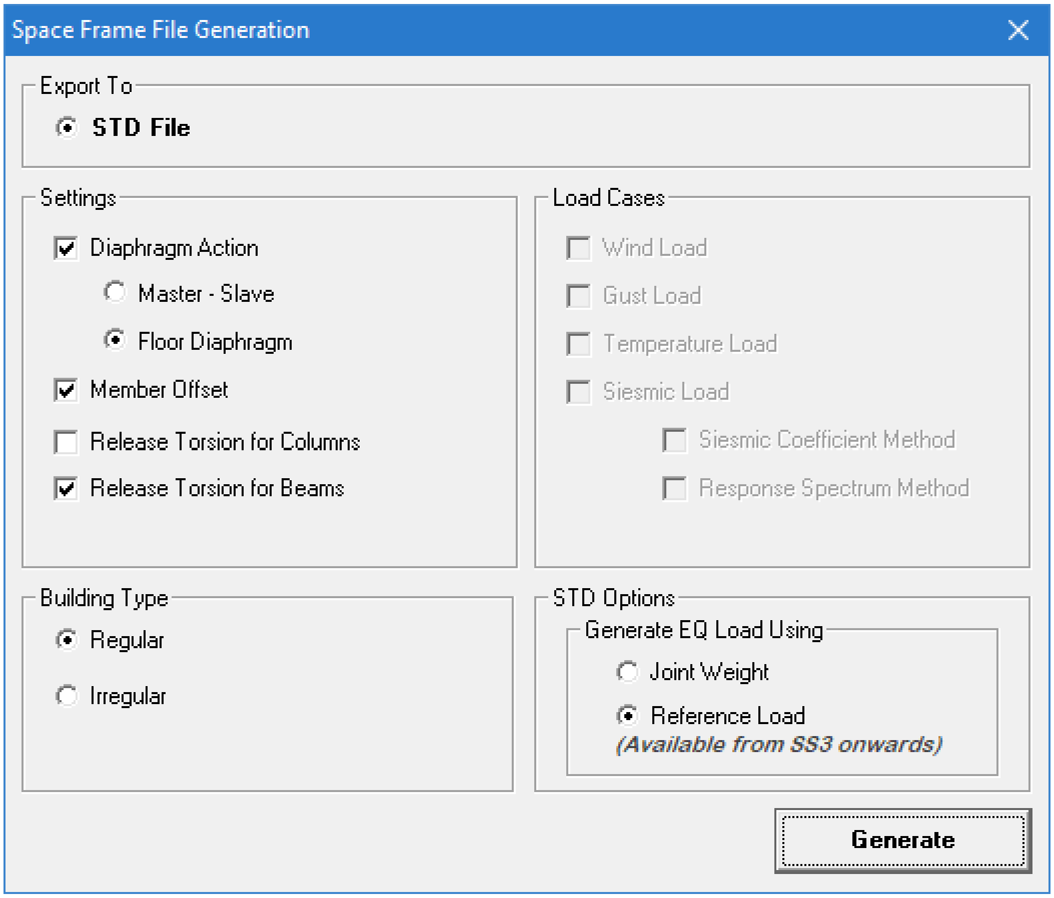
To generate a STAAD.Pro input file from your Building Planner workflow physical model, use the following procedure.

The physical model created using the Building Planner workflow can export an equivalent STAAD.Pro input file for use with the analysis and design capabilities in STAAD.Pro.

Tip: You must perform an analysis using STAAD.Pro in order to perform design in the Building Planner or Advanced Concrete Design workflows.

1. In the 3D Frame workspace, select **Analysis** > **Generate Analysis File**.
2. Click **OK**.

   The **Space Frame File Generation** dialog opens.



Caution: Do *not* close the dialog by clicking X as this will result in an invalid STAAD input tile.

3. Specify the parameters to use for the STAAD input file:
   a. Select the **Settings** to use for diaphragms, beams, and columns.
   b. Select the load cases to use, including if the seismic load should be (static) **Seismic Coefficient Method** or **Response Spectrum Method**.
   c. Select the **Building Type**.
      For **Irregular** buildings, both positive and negative direction load cases are generated.
   d. Select whether to **Generate EQ Load** Using by Joint Weight or Reference Load.
4. Click **Generate**.
   A warning message opens to inform you that the existing STAAD.Pro model will be overwritten.
5. Click **Yes**.
   The STAAD.Pro input file is created and opens in the Analytical Modeling workflow.

You can now make changes to the STAAD.Pro file as necessary. You can perform an analysis by selecting the **Run Analysis** tool on the **Analysis** ribbon tab or by pressing **<Ctrl+F5>**.

Related Links

- *Space Frame File Generation dialog* (on page 950)

Space Frame File Generation dialog

Used to specify analytical model settings, building details, and load cases to use for a STAAD input file (file extension `.std`) generated from the 3D building frame in Building Planner.

Dialog Controls

Control	Description
Settings group	**Control** / **Description** **Diaphragm Action** — Check this option to include a diaphragm definition in the analysis model. Select the method to use for defining the diaphragm: • Control - Dependent (formerly referred to as Master / Slave) • Floor Diaphragm **Member Offset** — Used to consider Member offset in the model. **Release Torsion for Columns** — Releases columns for torsion. **Release Torsion for Beams** — Releases beams for torsion.
Load Cases group	**Control** / **Description** **Wind Load** — Wind load will be applied to the analytical model using the load parameters indicated in the **Wind Parameters** dialog. **Gust Load** — Gust (wind) load will be applied to the analytical model using the load parameters indicated in the **Wind Parameters** dialog. **Temperature Load** — **Note:** This option is currently inactive. **Seismic Load** — Earthquake Load per IS 1893 will be applied to the analytical model using the load parameters in the **Seismic Parameters** dialog.
Building Type	This option is used to create appropriate load combinations. • For **Irregular** buildings, both positive and negative direction load cases are generated. • For **Regular** buildings, only positive direction load cases are generated.
Generate EQ Load Using	Specifies that earthquake load are created using either **Reference Load** or by **Joint Weight**.
Generate	Click generate the STAAD input file and open in Modeling mode.

Related Links

- *M. To Generate a STAAD.Pro Model* (on page 948)

M. To Design Slabs

You must have STAAD RCDC installed to perform slab design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC** > **Slab**.
   STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the slab and perform the design.

   Tip: Refer to the STAAD RCDC help for details.
3. Close STAAD RCDC when you have completed the slab designs for the selected building level.

M. To Design Beams

You must have STAAD RCDC installed to perform beam design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC** > **Beams**.
   STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the beam and perform the design.

   Tip: Refer to the STAAD RCDC help for details.
3. Close STAAD RCDC when you have completed the beam designs for the selected building levels.

M. To Design Columns

You must have STAAD RCDC installed to perform column design from the Building Planner mode.

You must perform a successful analysis of the generated STAAD.Pro input file prior to performing design.

1. Select **RCDC** > **Columns**.
   STAAD RCDC opens and reads the analysis results for the associated STAAD.Pro input file.
2. In STAAD RCDC, select the column and perform the design.

   Tip: Refer to the STAAD RCDC help for details.
3. Close STAAD RCDC when you have completed the column designs.

5
Analysis

This section of the help describes the analysis methods, data print commands, and how to perform analysis in STAAD.Pro

A. Types of Analysis

A. To specify a linear elastic analysis

To instruct the program to perform a first-order, linear elastic analysis of the structure, use the following procedure.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Analysis** tab.
3. Select the **Print Option** you want to use.
4. Click **Add**.
   The perform analysis command is added to the input file.

Related Links

- *TR.37.1 Linear Elastic Analysis* (on page 2620)
- *Perform Analysis tab* (on page 2917)

A. To specify a P-Delta analysis

To instruct the program to perform a second-order, P-Delta analysis, use the following procedure.

1. Either:

   select the **Analysis** page

or

on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

The **Analysis/Print Commands** dialog opens.
2. Select the **PDelta Analysis** tab.
3. Type the **Number of Iterations** to use.
4. (Optional) Check the **Use Geometric Stiffness (Kg)** option to include stiffening effects of the KG matrix.

   Refer to Notes for Stress Stiffening Matrix (Option 2) (on page 2623) for details on this option.
5. Check the **Small Delta** option to include effects of member deflection (i.e., P-δ effects).

   This option is recommended. Clear this option to only consider effects of structure drift (i.e., P-Δ effects).
6. Select the **Print Option** you want to use.
7. Click **Add**.
   The PDetla Analysis command is added to the input file.

Related Links
- *G.17.2.1 P-Delta Analysis* (on page 2141)
- *P Delta Analysis tab* (on page 2917)
- *TR.37.2 P-Delta Analysis Options* (on page 2621)

A. To specify a direct analysis

To instruct the program to perform a direct analysis as per AISC 360, use the following procedure.

Note: You must have one or more members with Direct Analysis parameters defined in order to use a direct analysis.

This is a non-linear, iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate in each step, changing the member characteristics until the maximum change in any τ_b is less than the tau tolerance (τ_{tol}), If the maximum change in any τ_b is less than $100 \times \tau_{tol}$ *and* the maximum change in any displacement degree of freedom is less than the displacement tolerance (δ_{tol}); then the solution has converged for this case.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Direct Analysis** tab.
3. Select to use either the **LRFD** or **ASD** design method.

4. Type values for both **Tau** and **Displacement** Tolerances.

   Note: Leave either field empty to use the default tolerances of 0.01 for Tau and 0.01 inches and 0.01 radians for displacement.
5. Type the **Number of Iterations** to use.

   Leave this field empty (zero) to use the default of one iteration.
6. Select the number of **PDelta Iterations** to perform in the iterative PDelta with small delta analysis procedure with a direct analysis.

   The default value of 15 is recommended.
7. (Optional) Clear the **Reduced EI** option to use the full EI for member section moment and section displacement calculations.
8. (Optional) Clear the **Perform Tau-b Iteration** option prevent the program from iterating τ_b.
9. Select the **Print Option** you want to use.
10. Click **Add**.
    The perform direct analysis command is added to the input file.

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *Perform Direct Analysis tab* (on page 2920)
- *TR.37.5 Direct Analysis* (on page 2630)

A. To specify a nonlinear analysis

To instruct the program to perform a static, geometric nonlinear analysis of the structure, use the following procedure.

Note: This requires a STAAD.Pro Advanced license.

The following limitations should be noted regarding static, geometrically nonlinear analyses:

- Large rotations in one step should be avoided by using more steps.
- Very large displacements, unstable structures, and/or post-buckling should be avoided.
- Geometrically nonlinear only. No tension/compression or contact is considered. No yield, plastic moment hinges or bilinear behavior is considered.
- Solids cannot be used for this analysis method.
- Temperature loads are *not* supported for geometric nonlinear analysis.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Nonlinear Analysis** tab.

3. (Optional) Specify the nonlinear analysis parameters as necessary:
   a. Type a absolute displacement limit for the first analysis step in the **ARC** field.
   b. Type the maximum number of **Iterations** to use to achieve equilibrium in the deform position within the specified tolerance value.
   c. Type the **Tolerance** to use for achieving convergence between two sequential iterations.
   d. Type the number of **Load Steps** to use for applying load in stages.
   e. Type frequency of **Rebuild Steps** of the tangent K matrix.
4. Check the **KG** option to add the geometric stiffness to the stiffness matrix.
5. (Optional) Check the **Set Displacement Limit** option to specify a target displacement:
   a. Either:

      type a **Node** number

      or

      click **[...]** to select a node in the view window
   b. Select a **DOF** (degree of freedom from the drop-down list).
   c. Type a **Target Value** distance for the displacement limit.
6. Select the **Print Option** you want to use.
7. Click **Add**.
   The perform nonlinear analysis command is added to the input file.

The maximum displacement should be reviewed for nonlinear analyses because this analysis type may result in buckling or large displacements.

Related Links

- *G.17.2.3 Static Geometrically Nonlinear Analysis* (on page 2147)
- *Nonlinear Analysis tab* (on page 2922)
- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *Nonlinear Analysis tab* (on page 2922)

A. To specify a nonlinear cable analysis

To specify a nonlinear cable analysis using the advanced features in STAAD.Pro, use the following steps.

You must specify at least one member with a nonlinear cable specification.

Note: Use of the Advanced Cable analysis feature requires the STAAD.Pro Advanced License.

Only one primary load case is permitted for a nonlinear cable analysis. A `CHANGE` command must be used with a new nonlinear cable analysis for each additional primary load case to be analyzed.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Cable Analysis** tab.
3. Select the **Advanced Cable Analysis** option.
4. (Optional) Set the options for **Use Modified Newton-Raphson** method and **Use Geometric Matrix (Kg)** as needed.
5. (Optional) Type values for the **Steps**, **Eq-Iterations**, and **Eq-tolerance** fields as needed.
   Default values are assumed for each when not specified.
6. Select the **Print Option** you want to use.
7. Click **Add**.
   The advanced cable analysis command is added to the input file.

A. To add a change command (on page 958) and M. To create a new primary load case (on page 830) to analyze additional load cases using a nonlinear cable analysis. Then repeat this procedure to specify another nonlinear cable analysis.

Tip: You can select to add the cable sag values for a nonlinear cable analysis to the output file in the post-analysis **Analysis/Print Commands** dialog.

Related Links
- *Perform Cable Analysis tab* (on page 2918)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)
- *Perform Cable Analysis tab* (on page 2918)

A. To specify an imperfection analysis

To instruct the program to perform an imperfection analysis, use the following procedure.

This analysis is used when member imperfection specifications are included in the model to define camber or drift.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Imperfection Analysis** tab.
3. Select the **Print Option** you want to use.
4. Click **Add**.
   The perform imperfection analysis command is added to the input file.

Related Links

- *G.17.2.4 Imperfection Analysis* (on page 2148)
- *TR.37.9 Imperfection Analysis* (on page 2657)
- *Perform Imperfection Analysis tab* (on page 2924)

A. To specify buckling analysis

To instruct the program to perform a buckling analysis on the structure, use the following procedure.

This analysis type is used to calculate the buckling factor for a load case. This is the load case multiplier at which global buckling of the structure would occur for that load case.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Buckling Analysis** tab.
3. Select the method for the buckling analysis:

   Iterative method

   or

   Eigen method - available for STAAD.Pro Advanced only
4. (Optional) If the iterative method is used, type the maximum **Number of Iterations** to use in the analysis.

   It not specified (i.e., left as zero), 20 iterations are used as the default value. 15 is recommended.

   Note: MAXSTEPS input is ignored when using the eigen method.
5. Select the **Print Option** you want to use.
6. Click **Add**.
   The perform buckling analysis command is added to the input file.

Related Links

- *G.17.2.2 Buckling Analysis* (on page 2145)
- *TR.37.4 Buckling Analysis* (on page 2628)
- *Perform Buckling Analysis tab* (on page 2925)

A. To specify a pushover analysis

To instruct the program to perform a pushover analysis, use the following procedure.

You mush first define pushover data and the pushover loading.

Note: This requires a STAAD.Pro Advanced license.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Pushover Analysis** tab.
3. Click **Add**.
   A message dialog opens to confirm you want to add this command.

Once you have successfully performed a pushover analysis, you can review the pushover results in the Postprocessing workflow.

Related Links

- *TR.37.7 Pushover Analysis* (on page 2643)
- *G.17.4 Pushover Analysis* (on page 2168)
- *M. Pushover Loads* (on page 881)
- *Perform Pushover Analysis tab* (on page 2926)

A. To add a change command

To add a change command which is required for performing a subsequent analysis command, use the following procedure.

A change command is only added *after* an analysis command.

In addition to being used for a new analysis, the change command is used to change the support conditions, active/inactive member status, load conditions, etc. The stiffness matrix being solved will be reset when a change command is used.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Change** tab.
3. (Optional) Check the **After Current** option if you previously selected the analysis command in the input file list within the **Analysis - Whole Structure** dialog.

This option allows you to specify specifically where the command is added. Otherwise it will be added to the end of the input file.

4. Click **Add**.
   The change command is added to the input file.

You can now add an additional analysis command to the input file. A change command with no subsequent analysis command has no effect.

Related Links

- *G.19 Multiple Analyses* (on page 2194)
- *TR.38 Change Specification* (on page 2660)
- *Change tab* (on page 2920)

A. To generate a floor spectrum

To calculate the floor spectra from time history acceleration, use the following procedure.

This procedure should only be used if a time history load case is being solved by the analysis engine.

Note: The floor response spectrum command must immediately follow an analysis command. Therefore, it is helpful to select the corresponding analysis command prior to starting this procedure.

Note: That analysis can only contain a single time history load case.

This command is used to specify the calculation of floor and/or joint spectra from time history results.

Note: This requires a STAAD.Pro Advanced license.

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Generate Floor Spectrum** tab.
3. For every group and direction that is necessary:
   a. Check one or more global directions in a table row: **GX**, **GY**, and **GZ**.
   b. Type a **Title** used to identify the spectrum.
   c. Use the drop-down to select the **Node Groups** which are included in this direction.

   You may include as many directions as needed.
4. (Optional) Type frequency options to use for the spectra (all are optional inputs):
   a. Type the **Lowest Frequency** (in Hz) to be in the calculated spectrum.
      This should be at least 0.0 Hz.
   b. Type the **Highest Frequency** (in Hz) to be in the calculated spectrum

   c. Type the number of **Frequency Interval** to use between the lowest and highest frequency.
5. (Optional) Type up to ten **Damping Ratios** to use as a ratio.
   (e.g., 3% damping is `0.03`)

   One spectrum will be generated for each damping ratio for each global direction requested for each floor defined. The spectrum will be based on these damping ratios. Is not entered, the default of 5% (0.05) damping will be used as a single damping ratio.
6. Check the **Relative Acceleration?** option if there is ground motion defined and you wan tthe spectrums based on the relative acceleration of the floor to the ground acceleration.

   Otherwise, the acceleration is assumed to be an absolute acceleration.
7. (Optional) Select the print parameters as needed (both are optional):
   a. Check the **Print Time History Acceleration used?** option to include this in the output.
   b. Check the **Print Calculated Spectrum?** option to include this in the output.
8. Select the **Print Option** you want to use.
9. Click **Add**.

> **Tip:** The **After Current** option is helpful to make sure this is added immediately following the analysis command if you selected prior to this procedure.

   The generate floor spectrum command is added to the input file.

Related Links

- *TR.37.10 Floor Spectrum Command* (on page 2657)
- *TR.37.10 Floor Spectrum Command* (on page 2657)
- *Generate Floor Spectrum tab* (on page 2921)

A. To specify pre-analysis commands

To instruct the program to include model input data in the STAAD output file (`.anl`), use the following procedure.

1. On the **Analysis and Design** ribbon tab, select the **Pre Analysis Commands** tool in the **Analysis Data** group.

   The **Pre Analysis Print - Whole Structure** dialog opens.
2. Click **Define Commands**.
   The **Analysis/Print Commands** dialog opens.
3. Select the tab corresponding to the print command you want to add to the output file.

   Problem Statistics
   Joint Coordinates
   Member Information
   Material Properties
   Support Information
   All
   Element Information

Solid Information
Member Properties

4. Click **Add**.

> **Note:** If you selected a portion of the model prior to opening the **Analysis/Print Commands** dialog, you can click **Assign** to assign the command to those model objects.

The selected print command is added to the input file.

5. Repeat Steps 3 and 4 to add additional print commands.
6. Click **Close**.

You must assign most print commands to portions of the model using one of the standard assignment methods.

Related Links

- *TR.42 Print Specifications* (on page 2666)
- *Analysis/Print Commands dialog (Pre Print)* (on page 2926)

A. To create a load list

To create an active list of load cases for use with subsequent print and design commands, use the following procedure.

A load list is used to control the loads that are included with following print and design specifications.

> **Note:** All load cases are considered by analysis commands. You just a `CHANGE` command to make changes to the loads considered for analysis.

1. On the **Analysis and Design** ribbon tab, select the **Load List** tool in the **Analysis Data** group.

The **Load List** dialog opens.

2. Either:

select one or more items in the **Load Cases** list and then click **[>]** to include them in the **Load List**

or

click **[>>]** in include all load cases

> **Tip:** Including all load cases is the same as not using a load list at all.

3. Click **OK**.

Related Links

- *Load List dialog* (on page 2931)
- *TR.39 Load List Specification* (on page 2662)

A. To check for soft stories and seismic code irregularities

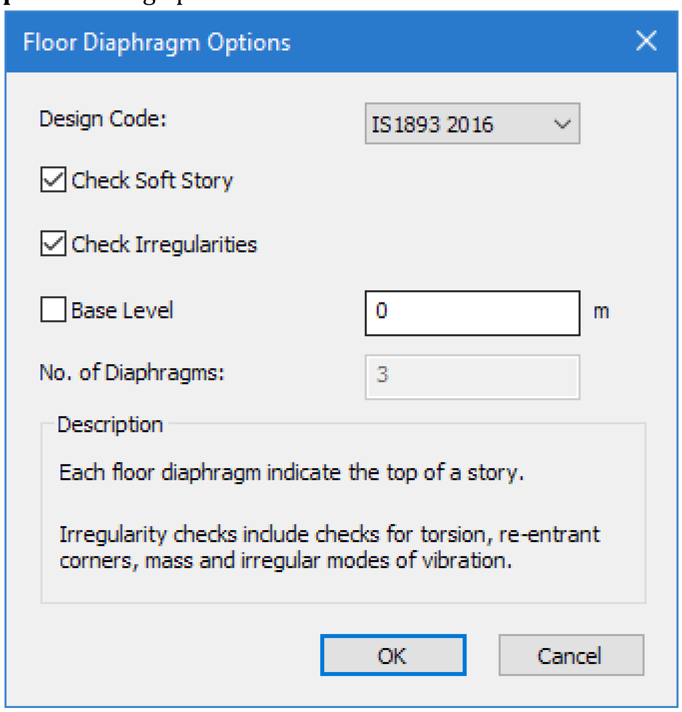
To check for soft stories and other seismic code irregularities in rigid diaphragm structures, use the following procedure.

Note: Soft story checks and plan irregularities are applicable to structures with rigid floor diaphragm definitions. Refer to M. To assign nodes to a floor diaphragm (on page 818).

The `CHECK SOFT STORY` option may added to a `FLOOR DIAPHRAGM` command set to check for soft stories per either IS 1893 2002, IS 1893 2016, or ASCE 7-05 codes. The `CHECK IRREGULARITIES` command may be added to check plan irregularities for the IS 1893 2016 or ASCE 7-05/10/16 codes.

Tip: You can check plan and vertical irregularities and structure stiffness per EC8 using P. Using the Earthquake Workflow (on page 2032).

1. On the **Analysis and Design** ribbon tab, select the **Miscellaneous Commands** > **Floor Diaphragm Options** tool in the **Analysis Data** group.
   The **Floor Diaphragm Options** dialog opens.



2. Select the building code for which you want to perform earthquake checks:

   IS1893 2002

   or

   IS1893 2016

   or

   ASCE7-05/10/16
3. Check the option **Check Soft Story** to perform this check.

4. (IS 1893 - 2016 or ASCE 7 only) To check for plan and other vertical irregularities, check the option **Check Irregularities**.

   For IS 1893 2016, the program can check horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6.

   For ASCE 7-05/10/16, the program can check horizontal irregularities (torsional and reentrant corners) and vertical irregularies (mass). Irregular modes of oscillation are not considered for this code.
5. (Optional) If the base of the structure is not the minimum global Y coordinate defined in the structure, then check the **Base Level** option and then type the height of the structure base in the adjacent field.
6. Click **Add**.

Note: Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command `PRINT STORY STIFFNESS` in the post analysis print commands section.

Related Links

- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.2 Check Irregularities* (on page 2333)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.2 Check Irregularities* (on page 2333)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *Floor Diaphragm Options dialog* (on page 2930)
- *TR.28.2.1 Soft Story Checking* (on page 2331)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.42 Print Specifications* (on page 2666)

A. To specify post-analysis print commands

To include additional analysis results in the STAAD output (`.anl`) file, use the following procedure.

Note: All output values will be available in the postprocessing workflow, regardless of whether you add them to the output file. Similarly, you can add post-analysis output to a report even if not included in the output file.

1. Select the **Post Analysis Commands** tool in the **Analysis Data** group on the **Analysis and Design** ribbon tab.

   The **Analysis/Print Commands** dialog for post-analysis commands opens.

> **Tip:** If the dialog does not immediately open, click **Define Commands** on the **Post Analysis Print - Whole Structure** dialog.

2. Select the tab pertaining to the output you want to add to the output file.
3. Check the **After Current** option to insert the print command following the currently selected command within the **Post Analysis Print - Whole Structure** dialog.

   Otherwise, the print command will be added to the end of the STAAD input file.
4. Click **Add**.
   The command is added to the STAAD Input file.
5. (Optional) Repeat steps 2 through 4 to add additional print commands as needed.
6. Click **Close**.

> **Note:** Some print commands require a assignment. These will be marked with question mark (?) in **Post Analysis Print - Whole Structure** dialog. Use the **Assignment Method** tools to assign these to the appropriate model objects.

Related Links

- *Analysis/Print Commands dialog (Post Print)* (on page 2927)
- *TR.42 Print Specifications* (on page 2666)

A. To output the center of rigidity

To instruct the program to include the center of rigidity of each rigid diaphragm in the analysis output, use the following procedure.

The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

1. Select the **Post Analysis Commands** tool in the **Analysis Data** group on the **Analysis and Design** ribbon tab.

   The **Analysis/Print Commands** dialog for post-analysis commands opens.

   > **Tip:** If the dialog does not immediately open, click **Define Commands** on the **Post Analysis Print - Whole Structure** dialog.

2. Select the **Dia CR** tab in the **Analysis/Print Commands** dialog.
3. Check the **After Current** option to insert the print command following the currently selected command within the **Post Analysis Print - Whole Structure** dialog.

   Otherwise, the print command will be added to the end of the STAAD input file.
4. Click **Add**.
   The command is added to the STAAD Input file.
5. Click **Close**.

Related Links

- *TR.42 Print Specifications* (on page 2666)

A. To report cable sag from an advanced cable analysis

To include the cable sag in the output from an advanced cable analysis, use the following procedure.

Tip: Cable sag is only available for advanced nonlinear analysis. If requested for a standard cable analysis, a warning message will be reported in the output.

An advanced cable analysis can also report the deflected shape of the sag in the cable using the `PRINT CABLE SAG` command.

1. On the **Analysis and Design** ribbon tab, select the **Post Analysis Commands** tool in the **Analysis Data** group.

   The **Post Analysis Print - Whole Structure** dialog opens.
2. Click Define Commands in the **Post Analysis Print - Whole Structure** dialog.
   The **Analysis/Print Commands** dialog opens.
3. Select the **Cable Sag** tab.
4. Click **Add**.
   The `PRINT CABLE SAG` command is added to the input file.

The output of a successful advanced cable analysis will report the cable sag in local XYZ coordinates. Post-analysis print will calculate the actual, nonlinear cable displacements along the length of cable.

A. To check for inter-story drift

To instruct the program to check the drift between adjacent stories against a code-defined ratio, use the following procedure.

Story drift is calculated as the relative horizontal displacement of two adjacent floors in a building. Inter-story drift can also be expressed as some factor times the story height. The allowable factor generally varies from one country code to another and may also vary depending on the type of loading. For example, in IS 1893: 2002 for seismic loading the allowable limit for inter-story drift is 0.004 times the story height whereas in IS 875 for wind loading it is 0.002 times the story height.

The drift at a particular story level in either lateral direction is calculated as the average of all the joint displacements present in that floor level. However if floor diaphragm is present, the drift is calculated at the center of mass (i.e,. control joint) of the floor.

Note: Additionally, the story stiffnesses used in calculating story drift or checking soft stories can be added to STAAD.Pro output by including the command `PRINT STORY STIFFNESS` in the post analysis print commands section.

Refer to TR.42 Print Specifications (on page 2666) for additional information.

Note: For dynamic IS 1893: 2002 response spectrum, the story drift check is performed by adding a command line within the load case, rather than in the post-analysis print commands. Refer to TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534) for additional information.

1. Select the **Post Analysis Commands** tool in the **Analysis Data** group on the **Analysis and Design** ribbon tab.

   The **Analysis/Print Commands** dialog for post-analysis commands opens.

   Tip: If the dialog does not immediately open, click **Define Commands** on the **Post Analysis Print - Whole Structure** dialog.
2. Select the **Story Drift** tab in the **Analysis/Print Commands** dialog.
3. Enter an **Allowable Drift Factor** value.

   This is the drift ratio value as specified by the applicable building code.
4. Check the **After Current** option to insert the print command following the currently selected command within the **Post Analysis Print - Whole Structure** dialog.

   Otherwise, the print command will be added to the end of the STAAD input file.
5. Click **Add**.
   The command is added to the STAAD Input file.
6. Click **Close**.

Related Links
- *TR.42 Print Specifications* (on page 2666)

A. To perform an analysis in STAAD.Pro

Once you have completed the input file, use the following procedure to perform an analysis and optional design.

The STAAD analysis engine performs analysis and design sequentially with a single click. In order to carry out the design, these design parameters must be specified along with geometry, properties, etc. in the input file (this is referred to as a “batch” design). Also, note that you can change the design code used for design and code check before performing the analysis and design.

The Analytical Modeling and Physical Modeling of the STAAD.Pro user interface are used to prepare the structural input data which is then passed to the STAAD analysis engine for general purpose structural analysis and design.

1. Either:

   Select the **Run Analysis** tool in the **Analysis** group on the **Analysis and Design** ribbon tab.

   or

Press **<CTRL+F5>**

The **STAAD Analysis and Design** dialog opens.

During the analysis (and design, if specified), an output file is generated. This file may contain selected input data items, results and error messages. Optional print specifications can be used to include additional information in the output file.

2. Select an option for what action occurs when the dialog is closed:

   Open the output file

   or

   go to the post-processing mode

   or

   remain in the analytical modeling mode
3. Click **Done**.
4. (Optional) To review the output file is this option was not selected in the **STAAD Analysis and Design** dialog, either:

   Select the **STAAD Output** tool in the **Utilities** group on the **Utilities** ribbon tab

   or

   Select the **View STAAD Output File** tool on the Quick Start toolbar.

Related Links

- *STAAD Analysis and Design dialog* (on page 2932)
- *STAAD Analysis and Design dialog* (on page 2932)

A. To run analysis on the cloud

To use Bentley's Scenario Services to perform analysis and design on your model, use the following procedure.

Note: Requires you to be logged into your valid CONNECT account.

Technical Preview:  This feature is part of a Technical Preview. It has not been reviewed per Bentley's standard quality assurance program and should be considered for review only.

1. On the **Analysis and Design** ribbon tab, select the **Run Cloud Analysis** tool in the **Cloud Analysis Services** group.

   The **Run Cloud Analysis** dialog opens.

2. If your STAAD model is not already associated with a CONNECT Project, you will be prompted to select one now.
3. (Optional) To create a new solution for use with the analysis:
   a. Click **Create**.
      The **Create New Solution** dialog opens.
   b. Type a **Solution Name**.
   c. (Optional) Type a **Solution Description**.
   d. Select the **Unit Convention** to use for this solution.
   e. Select the **Analysis Engine** to use.

      The various releases of STAAD.Pro are included in the drop-down list.
   f. Select the **Summary Indicators** you want to use for ranking the performance of this scenario.
   g. Select any optional **Other Indicators** you would also like to use for scenario comparisons.
   h. Click **Create Solution**.
      You are notified once the solution has been successfully created.
   i. Click **OK**.
4. Select the solution to use from the **Select Solution** drop-down list.

   The details of the selected scenario are displayed.
5. Type a **Scenario Name**.

   This must be a unique name.
6. Click **Submit**.
   You are notified once the analysis job has been successfully created and queued.

Once the analysis job is complete, you will be notified with a message from the Windows task area.

You can use the **Download Results** tools to either open the results for postprocessing STAAD.Pro or to save to your local computer for review or other postprocessing later.

6 Design

This section contains information on designing structural elements, organized by material type.

D. Batch Design versus Interactive Design Workflows

STAAD.Pro has two means by which structural members can be designed.

Batch Design

Using this method, code checks and/or member selection is performed directly by the analysis and design engine when an analysis is performed.

Interactive Design Workflow

Concrete code checks and member selection are performed in a post-processing workflow Advanced Concrete Design. This workflow is available in the Workflows panel.

Related Links

- *D. Available Steel Design Codes* (on page 969)
- *D. Available Concrete Design Codes* (on page 1067)
- *D. Available Aluminum Design Codes* (on page 1081)
- *D. Available Timber Design Codes* (on page 1084)

D. Steel Design

D. Available Steel Design Codes

All steel design codes are available using the batch design method in the Analytical Modeling workflow (e.g., via the STAAD input file) where not specifically stated otherwise in the workflow column.

Note: Design per the Chinese steel code GB 50017-2017 is available through the **Chinese Steel Design** workflow.

Batch Design

Table 41: Steel design codes available in batch design

Country/Region	Code	Title
Australia	D2.B. Australian Codes - Steel Design per AS 4100 - 1998 (on page 1356)	Standards Australia - Steel Structural Design, including Amendment 1 (2012)
Belgium	D5.D.10 Belgian National Annex to EC3 (on page 1563)	Belgian National Annex (EC3)
Canada	D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01 (on page 1419)	Limit States Design of Steel Structures
	D4.C. Canadian Codes - Cold Formed Steel Design per S136-94 (on page 1434)	Specification for the Design of Cold-Formed Steel Structural Members
	D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19 (on page 1447)	Limit States Design of Steel Structures (2009, 2014, and 2019 editions)
Europe	D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478)	Eurocode 3: Design of steel structures Part 1.1 General rules and rules for buildings
Finland	D5.D.7 Finnish National Annex to EC3 (on page 1543)	Finnish National Annex (EC3)
France	D6.A. French Codes - Steel Design per CM66-1977 (French) (on page 1592)	Regulations for the calculation of steel structures and Design rules for structural steelwork, Addendum 80
	D5.D.6 French National Annex to EC3 (on page 1537)	French National Annex (EC3)
Germany	D7.A. German Codes - Steel Design per DIN 18800 Code (on page 1600)	Structural steelwork. Safety against buckling of linear members and frames
	D5.D.12 German National Annex to EC3 (on page 1575)	German National Annex (EC3)
India	D8.B. Indian Codes - Steel Design per IS 800 - 1984 (on page 1625)	Code of Practice for General Construction in Steel

Country/Region	Code	Title
	D8.A. Indian Codes - Steel Design per IS 800 - 2007 (on page 1607)	General Construction in Steel - Code of Practice (Third Revision)
	D8.A. Indian Codes - Steel Design per IS 800 - 2007 (on page 1607)	General Construction in Steel - Code of Practice (Third Revision)
	D8.C. Indian Codes - Cold Formed Steel Design per IS 801 - 1975 (on page 1640)	Code of Practice for Use of Cold-Formed Light Gauge Steel Structural Members in General Build Construction
	D8.D. Indian Codes - Steel Design per IS 802 - 1995 (on page 1652)	Use of Structural Steel in Overhead Transmission Line Towers - Code of Practice
Japan	D9.C. Japanese Codes - Steel Design per 2002 AIJ (on page 1728)	Design Standard for Steel Structures
	D9.B. Japanese Codes - Steel Design per 2005 AIJ (on page 1719)	Design Standard for Steel Structures
Malaysia	D5.D.11 Malaysian National Annex to EC3 (on page 1570)	Malaysian National Annex (EC3)
Mexico	D10.B. Mexican Codes - Steel Design per NTC 1987 (on page 1754)	Normas Técnicas Complementarias para Diseño y construcción de Estructuras Metálicas - LRFD
Netherlands	D5.D.3 Dutch National Annex to EC3 (on page 1518)	Dutch National Annex (EC3)
New Zealand	D11.A. New Zealand Codes - Steel Design per NZS 3404-1997 (on page 1765)	New Zealand Standard for Steel Structures, Parts 1 & 2, including Amendments 1 & 2.
Norway	D12.A. Norwegian Codes - Steel Design per NS 3472 / NPD (on page 1790)	Veiledning om utforming, beregning og dimensjonering av stalkonstruksjoner. Sist enderet 1.
	D12.A. Norwegian Codes - Steel Design per NS 3472 / NPD (on page 1790)	Steel structures. Design rules (3rd Edition)
	D12.B. Norwegian Codes - Steel Design per NORSOK N-004 (on page 1837)	NORSOK N-004 Rev 2, October 2004
	D5.D.4 Norwegian National Annex to EC3 (on page 1527)	Norwegian National Annex

Country/Region	Code	Title
Russia	D13.B. Russian Codes - Steel Design Per SNiP 2.23-81* (Edition 1990) (on page 1890)	Design Standards for Steel Construction
	D13.C. Russian Codes - Steel Design Per SP 16.13330.2011 (on page 1905)	Code of Practice: Steel Structures
	D13.E. Russian Codes - Steel Design per SP 16.13330.2017 (on page 1932)	Code of Practice: Steel Structures
Poland	D5.D.8 Polish National Annex to EC3 (on page 1550)	Polish National Annex (EC3)
Singapore	D5.D.9 Singaporean National Annex to EC3 (on page 1556)	Singaporean National Annex (EC3)
South Africa	AD.2006.6.6 South African Steel Design (on page 355)	The structural use of steel Part 1: Limit-states design of hot-rolled steelwork
	D14.C. South African Codes - Steel Design Per SANS 10162-1:2011 (on page 1966)	Design of steel structures
Sweden	D5.D.13 Swedish National Annex to EC3 (on page 1579)	Swedish National Annex (EC3)
United Kingdom	D3.B. British Codes - Steel Design per BS5950:2000 (on page 1378)	Structural use of steelwork in building - Part 1: Code of practice for design - Rolled and welded sections, Incorporating Corrigendum No. 1
	D3.E. British Codes - Design per British Cold Formed Steel Code (on page 1405)	Structural use of steelwork in building - Part 5. Code of practice for design of cold formed thin gauge sections
	D3.C. British Codes - Design per BS5400 (on page 1399)	Steel, concrete and, composite bridges Part 3. Code of practice for design of steel bridges and Amd No. 4051 and Amd No. 6488
	D5.D.5 UK National Annex to EC3 (on page 1528)	British National Annex (EC3)

Country/Region	Code	Title
United States	D1.B.1 Working Stress Design (on page 1130)	Specification for Structural Steel Buildings. Allowable Stress Design and Plastic Design (1989, 9th Edition)
	D1.C. American Codes - Steel Design per AISC LRFD Specification (on page 1163)	LRFD Specification for Structural Steel Buildings (1999, 3rd Edition) Load and Resistance Factor Design Specification for Structural Steel Buildings (1993, 2nd Edition)
	D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1086)	Specification for Structural Steel Buildings (Unified specification), 2016 Edition and Design Guide #9 and AISC 341-16 Seismic Provisions. Both customary and metric units are supported.
	D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1086)	Specification for Structural Steel Buildings (Unified specification), 2010 Edition and Design Guide #9 and AISC 341-10 Seismic Provisions. Both customary and metric units are supported.
	D1.A. American Codes - Steel Design per AISC 360 Unified Specification (on page 1086)	Specification for Structural Steel Buildings (Unified specification), 2005 Edition and Supplement 1, April 2002 and AISC 341-05 Seismic Provisions. Both customary and metric units are supported.
	D1.B.2 Castellated Beams (on page 1147)	AISC ASD 1989 9th edition Castellated
	D1.E. American Codes - Steel Design per AISI Cold Formed Steel Code (on page 1186)	North American Specification for the Design of Cold Formed Steel Structural Members
	D1.D.1 AASHTO (ASD) (on page 1176)	Standard Specifications for Highway Bridges, 17th Edition (2002); Chapter 10, Part C

Country/Region	Code	Title
	D1.D.2 AASHTO (LRFD) (on page 1180)	LRFD Design Specification, 2nd Edition (1998)
	D1.J. American Codes - Steel Design per API 2A-WSD 2000 (on page 1283)	Recommended Practice for Planning, Design and Constructing Fixed Offshore Platforms-Working Stress Design , 21st Edition
	D1.I.1. American Transmission Tower Code - Steel Design per ASCE 10-97 (on page 1274)	Design of Latticed Steel Transmission Structures

Nuclear Codes

Note: These codes require a separate Nuclear Design Codes license.

Code	Title
D1.L.1. ASME NF 3000 - 1974 & 1977 Codes (on page 1310)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.L.1. ASME NF 3000 - 1974 & 1977 Codes (on page 1310)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.L.2. ASME NF 3000 - 1989 Code (on page 1319)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.L.3. ASME NF 3000 - 1998 Code (on page 1329)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.L.4. ASME NF 3000 - 2001 & 2004 Codes (on page 1338)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.L.4. ASME NF 3000 - 2001 & 2004 Codes (on page 1338)	ASME Boiler and Pressure Vessel Code, Section III Rules for Construction of Nuclear Power Plant Components, Division 1 - Appendices - Subsection NF
D1.K.1. ANSI/AISC N690-1994 Code (on page 1296)	Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities and Supplement No. 1
D1.K.2. ANSI/AISC N690-1984 Code (on page 1303)	Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities

Related Links

- *D.Steel Design Overview* (on page 975)
- *D. To specify steel design code and parameters* (on page 976)
- *D. Batch Design versus Interactive Design Workflows* (on page 969)

D.Batch Steel Design Operations

STAAD.Pro contains a broad set of facilities for designing structural steel members as individual components of an analyzed structure. The member design facilities provide you with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem.

D.Steel Design Overview

The design of steel members in STAAD.Pro can be achieved in the following three methods:

- check that the member meets the requirements of the design code using the Code Check command.
- for the forces resulting from the current set of loading, determine the most efficient size of section that would meet the requirements for the design code using the Select command.
- determine the optimum size for all members in the model that would meet the requirements for the design code for the current loading set using the Select Optimized command.

Each design follows the rules as defined in a published design code specification. The list of available codes currently supported can be found at D. Available Steel Design Codes (on page 969).

In order to determine the design code and code edition to which the members will be designed, the design command is preceded by a definition of the code and a series of code-related parameters. Each collection of these parameters is referred to as a “parameter block.” In the STAAD.Pro input file, this is initiated with the command `PARAMETER` and then typically terminated with the design command (e.g., `CHECK CODE`).

The parameter block provides the means by which you specify design-related member attributes required for the specified design code that have not been specified in the analytical model thus far. Each parameter has a default value which is ascribed to members to which a design command applies. When you assign a parameter to a given member, any design command that follows this assignment will use the specified parameter in the design. The default value of each design code parameter is listed in the appropriate section of the documentation.

```
PARAMETER
CODE design code/version
parameters modified from default values
design command
```

It is possible to define multiple design command and parameter blocks :

```
PARAMETER 1
CODE design code/version
parameters modified from default values
design command 1
design command 2
design command 3
...
PARAMETER 2
CODE design code/version
parameters modified from default values
```

```
design command 4
design command 5
design command 6
```

Also, each design command can be defined for different sets of members and different design commands:

```
PARAMETER 1
CODE design code/version
parameters modified from default values
CHECK CODE MEMBER 1 TO 10
SELECT MEMBERS 5 TO 20
```

Related Links

- *D. Available Steel Design Codes* (on page 969)
- *Steel Design - Whole Structure dialog* (on page 2932)
- *Design Parameters dialog* (on page 2934)
- *TR.48.1 Parameter Specifications* (on page 2676)
- *D. To specify steel design code and parameters* (on page 976)

D. To specify steel design code and parameters

To initiate the design of steel members and specify the code parameters, use the following procedure.

Note: To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the **Analysis and Design** ribbon tab, select **Steel** in the **Design** group gallery.
   The **Steel Design - Whole Structure** dialog opens.
2. In **Steel Design - Whole Structure** dialog, select the applicable steel design code from the **Current Code** drop-down list.
3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Specify a value or option for each required parameter for a set of members and then click **Add**.

   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

   Note: Different parameters can be used for different member type designs (e.g., columns, beams, composite floor girders, braces, etc.). Alternatively, you can use a separate set of parameters for different member types.
5. Click **Close**.

   The design parameters are marked with an  icon. This indicates that the need to be assigned to members.
6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.

Related Links

- *D.Steel Design Overview* (on page 975)
- *D. Available Steel Design Codes* (on page 969)
- *Steel Design - Whole Structure dialog* (on page 2932)
- *Design Parameters dialog* (on page 2934)

- *TR.48.1 Parameter Specifications* (on page 2676)

D. To design steel members in groups

To design members as a group, use the following procedure.

In many situations, it is preferable to have only a few different member sizes for many members rather than each member size be different, even if this would result in some members being underutilized. You can group members in the batch design to instruct the program to replace the member sizes in the group with the highest capacity member of the existing sizes in that group. You can also instruct the program to use a specified section property for the grouping or even group based on a specific member number, regardless of property specifications.

For example, in a floor bay with 5 beam members the edge members were selected as W12x14 shapes and the interior members were selected as W14x22 shapes. Grouping these members will immediately instruct the program to use W14x22 for all members for any subsequent instructions (i.e., analysis, design, take-off, etc.).

Note: All members within a group must be of the same cross section type. If one or more of the members has a different cross-setion type, the grouping is ignored by the program.

1. On the **Steel Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens. The tabs (if any) available in this dialog are dependent on the current design code selection. Not all commands are available for every design code.
2. If you want to retain the member groups for all subsequent commands, select the **FIXED GROUP** tab and then click **Add**.

   If you do not use the FIXED GROUP option, then the members are grouped once and then this grouping is ignored for subsequent commands. If you do add the FIXED GROUP command, then all subsequent member selection commands will apply to the group again.
3. Select the **GROUP** tab.
4. (Optional) Select the **Property Specification** to use for grouping shapes.
5. (Optional) Check the **Same as Beam #** option and then select a member number from the drop-down list.
6. Click **Add**.
7. Click **Close**.
8. Assign the group command to all the members you want to group together.

You can repeat this procedure to create as many design groups as necessary.

Once you have created a group, you will typically want to perform a code check or member selection on the members.

Related Links

- *TR.48 Steel and Aluminum Design Specifications* (on page 2675)
- *Design Commands dialog* (on page 2935)
- *Steel Design - Whole Structure dialog* (on page 2932)
- *TR.50 Group Specification* (on page 2680)

D. To specify steel design commands

To specify the code checking or design commands to be used for the steel design operation, use the following procedure.

1. On the **Steel Design - Whole Structure** dialog, click **Commands**.

The **Design Commands** dialog opens. The tabs (if any) available in this dialog are dependent on the current design code selection. Not all commands are available for every design code.

2. To perform a design action, select one of the following options (as they are available for the code selection):

To...	Select...
check the capacity of the current member sizes	the **CHECK CODE** tab
select a member size based on the last analysis results	the **SELECT** tab
select a member size based on iterating the analysis for updated member sizes	the **SELECT OPTIMIZED** tab

3. Click **Add**.
4. Click **Close**.
5. Assign the design commands to the appropriate members.

Related Links

- *TR.48 Steel and Aluminum Design Specifications* (on page 2675)
- *Design Commands dialog* (on page 2935)
- *Steel Design - Whole Structure dialog* (on page 2932)
- *TR.49 Code Checking Specification* (on page 2677)
- *TR.49.1 Member Selection Specification* (on page 2678)
- *TR.49.2 Member Selection by Optimization* (on page 2679)

D. To generate steel take off

To generate a summary of all steel sections with their lengths and weights for the whole structure or a list of members, use the following procedure.

1. On the **Steel Design - Whole Structure** dialog, click **Commands**.
The **Design Commands** dialog opens. The tabs (if any) available in this dialog are dependent on the current design code selection. Not all commands are available for every design code.
2. Select the take off table you want to include:

To...	Select...
generate a list of sections used including total length and weight	the **TAKE OFF** tab
generate a list sorted by members including the length and weight of the member and section used	the **MEMBER TAKE OFF** tab

3. Click **Add**.
4. Click **Close**.
5. Assign the take off commands to members or to a named group of members.

Related Links

- *TR.51 Steel and Aluminum Take Off Specification* (on page 2682)
- *Design Commands dialog* (on page 2935)
- *P. Steel AutoDrafter Workflow* (on page 2018)

D. Chinese Steel Design

The SSDD application for steel design to the Chinese code is integrated as the Chinese Steel Design workflow.

D. To open a model in the Chinese Steel Design workflow

To design steel members per the Chinese GB 50017-2017 code, use the following procedure.

In order to use the Chinese steel design workflow, the steel sections must be from the steel section table or from a user-provided table. Refer to Member Properties (on page 985) for details.

Note: In previous versions, wind and response spectra load cases per the Chinese code were applied within the SSDD application. These loads are now applied in the analytical modeling workflow within STAAD.Pro:

- M. To add a GB50009 wind load definition (on page 848)
- M. To add a seismic load definition (on page 857) (static load per GB50011-2010)
- M. To add a GB 50011-2010 response spectrum (on page 869)

Load combinations should be generated per the GB 50009-2012 specification.

- M. To automatically generate load combinations (on page 888)

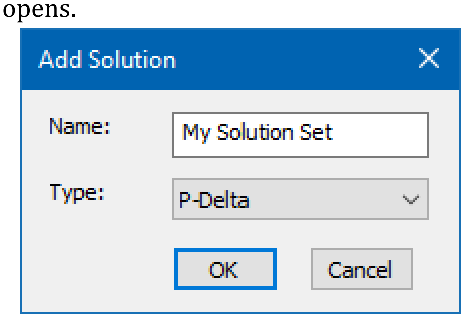
1. Open an input file in STAAD.Pro which contains the steel sections to be designed.
2. Perform a successful analysis.
3. Select **Chinese Steel Design** in the Workflows panel.

D. To add a new solution set

To add a new set of parameters in a solution set for comparing different designs, use the following procedure.

1. On the **Chinese Steel Design** ribbon tab in the **Solution** group, select the **Configuration** tool.

   The **Configure Solution** dialog opens.
2. Add a new solution set:
   a. Click **Add**.
      The **Add Solution Name** dialog opens.



b. Type a **Solution Name**.
c. Select the analysis **Type** to use with this solution set from the drop-down list.

> **Note:** The solution set is bound to an analysis method: **First-order** (elastic) or **P-Delta**.

d. Click **OK**.
The dialog closes.

The new solution set is added and is selected in the **Select Solution** drop-down list.

3. Click **OK**.
The dialog closes.
4. In the **Chinese Steel Design** ribbon tab **Solution** group, select the new solution set name from the drop-down list.
5. Add one or more design parameters.

> **Note:** The name of the current selection solution is displayed in the **Design Parameters** dialog. The analysis type associated with the solution set is also displayed in this dialog.

6. On the **Chinese Steel Design** ribbon tab in the **Solution** group, select the **Configuration** tool.

The **Configure Solution** dialog opens.
7. Select the new solution set name from the **Select Solution** drop-down list.
8. Add parameter sets to the **Used** list by either:

To...	Do the following...
add a single parameter	select a parameter name in the **Unused** list and then click **>**
add all available parameters	click **>>**

9. Click **OK**.

To design with a solution set, select the solution set name from the drop-down list in the **Chinese Steel Design** ribbon tab **Solution** group.

Related Links

- *RR 22.08.00-4.4 Multiple Parameters in Chinese Steel Design* (on page 100)
- *Configure Solution dialog* (on page 3019)

D. To add a parameter set

To a set of design parameters to one or more members, use the following procedure.

A parameter set is specific to a solution set and the associated analysis type. Select the correct solution set before adding a parameter set.

A parameter set is used to assign a collection of steel design parameters (grade, seismic detailing, etc.) to one or more members.

> **Note:** Many of the parameters do have default values, so they do not all need to be specified for every member. However, any member to be designed in the Chinese Steel Design workflow does require a parameter set assigned to it.

1. In the Chinese Steel Design workflow, select the **Parameters** page.

The **<solution set name> - Design Parameters** dialog opens.

> **Note:** The type of analysis associated with the current solution set is displayed at the top of the dialog.

2. In the **<solution set name> - Design Parameters** dialog, click **Add**.
   The **Add New Chinese Steel Design Parameters** dialog opens.
3. Select and specify the parameters on each of the dialog tabs for this set as needed.
4. Click **OK**.

Parameter sets associated with a STAAD input file are stored in a `.gsp` with the same file name as the input file and in the same file location. These are plain text files.

Related Links

- *Chinese Steel Design Parameters dialog* (on page 3020)
- *Design Parameters* (on page 988)

D. To assign secondary members

To assign members to be considered as "secondary" members per the Chinese steel code, use the following procedure.

The members into which a secondary member frame will *not* be considered as restrained when determining automatic effective length calculations.

1. On the **Chinese Steel Design** ribbon tab, select the **Secondary Member** tool in the **Settings** group.

   The **Assign Secondary Member** dialog opens.
2. Use one of the following methods to select members for assignment:

Options	Description
To assign to a selection	select one or more members in the model view window and then select the **Assign to Selected Beams** option
To assign to all the members in the current view	select the **Assign to View** option
To assign to a list of members	select the **Assign to Edit List** option and then type the member numbers in the text field

> **Tip:** You can use tools found in the **Beams** group on the **Select** ribbon tab (e.g., **Parallel** tools) to easily select members for the Assign to Selected Beams option.

3. Click **Assign**.

> **Note:** You can remove unwanted secondary member assignments by repeating step1 and then using the **Remove** button to remove assignments for all or selected members.

Related Links

- *Assign Secondary Member dialog* (on page 3012)

D. To specify brace angle threshold values

To specify angle values used as the limit when auto-detecting brace members, use the following procedure.

Note: The default brace angle values (1 = 60°, 2 = 60°, and 3 = 10°) are recommended. These values typically only need to be changed when the program does not automatically identify a brace member as such.

1. Select the **Parameters** page.
2. On the **Chinese Steel Design** ribbon tab, select the **Secondary Member** tool in the **Settings** group.

   The **Brace Angle** dialog opens.
3. Type the values for the angle thresholds as necessary:

   1 and 2 are the angle between the brace and the major or minor axis of the member, respectively.

   3 is the angle between adjacent brace members considered to be collinear.
4. Click **OK**.

Re-perform the design procedure to have the changes reflected in the brace detection.
Related Links

- *Brace Angle dialog* (on page 3013)

D. To add a custom material definition

To add a user-defined steel material definition for use with the Chinese Steel Design workflow, use the following procedure.

The standard steel values for Chinese steel design from Table 4.4.1 of GB50017-2017 are included in the STAAD.Pro material definitions for use in the analytical modeling workflow. The material yield stress (*fy*) used for the design parameter should match that in the STAAD.Pro analytical model. Otherwise, a warning is produced in the member check report.

If a custom material definition is used in the analytical model, then you can add a custom material definition for the Chinese steel design workflow as well.

1. On the **Chinese Steel Design** ribbon tab in the **Settings** group, select the **Material** tool.

   The **Material Parameter** dialog opens.
2. Click **Add**.
   A new row labeled "USERDEFINE*n*" is added to the bottom of the table.
3. Select the new row.

   Note: The default material values are "read only" (marked in gray). These cannot be edited or deleted.

   The material values are populated with the defaults.

4. Type a **Material Name**.
5. Type a **Max Thickness** for which these values will apply.

   The same material name can have different Max T values.
6. Type the strength values which apply for this material name and max. thickness.

   Yield Strength (fy)
   Tensile Strength (f)
   Shear Strength (fv)
   Section Bearing Strength (fce)
   Ultimate Tensile Strength (fu)
7. Repeat steps 2 through 6 to add additional material definitions.
8. Click **Close**.

You can now select this material name in the parameter definition.

Related Links

- *Material Parameter dialog* (on page 3013)

D. To perform steel design per the Chinese code

To perform steel design per the GB 500017 code, use the following procedure.

1. On the **Chinese Steel Design** ribbon tab, select the **Design** tool in the **Design** group.

   The **Chinese Steel Design** dialog opens.
2. (Optional) On the **Structure** tab, set the following options as needed:
   a. Click **Default** to make any changes to the default design parameter.

      This parameter set is used for any members which do not have a named parameter set assigned.

      Tip: Any members without a parameter set name associated with it in the view window will use the default parameter set.
   b. Select the **Design Range** option: all members or on those selected.
   c. Check the **Seismic Design** option to include seismic design and then select the **Seismic Precautionary Intensity** level from the drop-down list.
   d. Select the optimization options applicable.
3. Click **Design**.
   The design and optimization process is displayed.
4. Click **Done**.
   The **Results** page opens. The members are color-coded by design status in the view window as well as in the Summary Results table.

   Tip: If you would like to change or add to the design result columns displayed in the Summary Results table, select the **Table Settings** tool in the **Results** group on the **Chinese Steel Design** ribbon tab (active when the Results page is selected). From there you can make changes to the table columns.

5. Select a member either in the view window or in the table to display design results in the Member Detail Result table.

Related Links

- *Chinese Steel Design dialog* (on page 3017)

D. Chinese Steel Design Technical Reference

General Description

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and optimization designed taking into consideration the limit states, and two major categories of limit-state are recognized: ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and the design strength of each structural component should equal or exceed the required strength determined according to the load combinations.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Accordingly, the most economic section is selected based the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of GB 50017-2017. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. Therefore, it is your responsibility to enter all necessary loads and load combination factors for design in accordance with the Building Structure Loadings Code GB 50009-2012, Building Structures Seismic code GB 50011-2010, or other relevant design codes. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. This process is completed in the STAAD.Pro Analytical Modeling or Physical Modeling workflows.

Depending upon the analysis requirements, a regular (elastic) stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Related Links

- *M. To add a GB50009 wind load definition* (on page 848)
- *M. To add a seismic load definition* (on page 857)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *A. To specify a linear elastic analysis* (on page 952)
- *A. To specify a P-Delta analysis* (on page 952)

Member Properties

For specification of member properties, either the steel section library available in STAAD.Pro or user-provided table sections may be used.

For code check and Section optimization, the Chinese Steel Design workflow supports:

- Chinese - Rolled Steel table
  - I shapes
  - H shapes
  - T shapes
  - Channels
  - Double Channels
  - Angles
  - Double Angles
  - Tube
  - Pipe
- User-defined table
  - composite sections

Note: Prismatic sections and other special composite section can only be used with the Code Check facility. Furthermore, the design of laced and battened members is not considered. The program does not support the design of tapered section profiles per Chinese code.

Related Links

- *M. To add a new table section property* (on page 715)
- *M.To create a wide-flange user table section* (on page 731)

Section Classification

The occurrence of local buckling of the compression elements of a cross-section prevents the development of full section capacity. It is therefore imperative to establish this possibility prior to determining the section capacities. Cross sections are classified in accordance with their geometrical properties and the stress pattern on the compression elements. For each load case considered in the design process, the program determines the section class and calculates the capacities accordingly. It is worth noting that the section class reported in the design output corresponds to the most critical load case among those being considered for design.

The program can design members with all section profiles that are of Class 1, 2, 3, or 4 as defined in section 3.5.1 of the code. However, the design of members that have a Class 5 section profile are limited to:

- H-sections
- Wide flange (with equal flanges or unequal flanges)
- Tube sections

Member Capacities and Deflection Check

Member Capacities

The bearing capacity of a member is the allowable stress under various load conditions, such as allowable tensile stress, allowable compressive stress, etc. The bearing capacity of a member depends on various factors, such as the cross-section characteristics, slenderness ratio and the ratio of unsupported width to thickness.

Due to the limitation of space and the complexity of GB 50017-2017, we can only briefly introduce the main specification inspection conditions for the above types of components in the program.

Strength and Stability Checks of Various Components

For different types of components, the program will check the strength and stability of components and select the section according to the relevant provisions of GB 50017-2017. The following will introduce the relevant code terms considered in the inspection of various types of components in the program.

In the following clauses, the numbers in the opening brackets are consistent with the corresponding clauses in the original GB 50017-2017 specification. In the following formula, the meaning of each symbol is the same as that in GB 50017-2017.

Flexural Members

1. The flexural strength of the solid web member bending in the main plane is as follows:

   $$\frac{M_z}{r_z W_{nz}} + \frac{M_y}{r_y W_{ny}} \le f \tag{6.1.1}$$

   In code inspection, the default component is static load or indirect dynamic load. In the calculation and test of the program, the values of parameters and forces are obtained from the data structure of components.
2. The shear strength of the solid web member bending in the main plane is as follows:

   $$\tau = \frac{VS}{It_w} \le f_v \tag{6.1.3}$$

3. 1. At the edge of the calculation height of the web of composite beam, if there are large normal stress, shear stress and local compressive stress at the same time, or bear large normal stress and shear stress at the same time (such as the support of continuous beam or the change of flange section of beam, etc.), the converted stress is as follows:

   $$\sqrt{\sigma^2 + \sigma_c^2 - \sigma\sigma_c + 3\tau^2} \le \beta_1 f \tag{6.1.5}$$

4. The global stability of the I-section bending in two main planes is calculated as follows:

   $$\frac{M_z}{\phi_b W_x} + \frac{M_y}{r_y W_y} \le f \tag{6.2.3}$$

5. Height thickness ratio of Web:

   Calculated value = web height / web thickness (I-beam, H-beam, channel steel...)

   =Diameter / wall thickness (round pipe)

   =Short leg length / short leg thickness (angle steel)

   See table 3.5.1 of GB 50017-2017 for calculation limit.
6. Flange width thickness ratio:

   Calculated value = flange width / flange thickness (I-beam, H-beam, channel steel...)

   =Diameter / wall thickness (round pipe)

   =Long leg length / short limb thickness (angle steel)

See table 3.5.1 of GB 50017-2017 for calculation limit.

Axially Loaded Members

1. The strength calculation formula of axially loaded members is as follows:

$$\sigma = \frac{N}{A} \le f \tag{7.1.1-1}$$

$$\sigma = \frac{N}{A_n} \le 0.7 f_u \tag{7.1.1-2}$$

2. The stability checking formula of axial compression member is as follows:

$$\frac{N}{\phi A} \le f \tag{7.2.1}$$

   For the stability factor ϕ, the program is obtained according to “Appendix D stability coefficient of axial compression members”in the code.
3. Slenderness ratio of compression member

   Refer to article 7.2.2 of GB 50017-2017 for calculation value. After calculation of Y-axis and z-axis, take the maximum value.

   If the program automatically calculates the limit value, the default value is 150, and then according to the seismic code, the requirements of axial force members are adjusted and modified.
4. Slenderness ratio of tension member

   Refer to article 7.2.2 of GB 50017-2017 for calculation value. After calculation of Y-axis and z-axis, take the maximum value.

   If the program automatically calculates the limit value, the default value is 300, and then according to the seismic code, the requirements of axial force members are adjusted and modified.
5. Height thickness ratio of Web:

   The calculated value is the same as that of flexural member.

   For the calculation limit, see article 7.3.1 of GB 50017-2017.
6. Flange width thickness ratio

   The calculated value is the same as that of flexural member.

   For the calculation limit, see article 7.3.1 of GB 50017-2017.

Members Subject to Combined Axial Loading and Bending

1. The strength of tension (compression) bending members with bending moment acting on the main plane is as follows:

$$\frac{N}{A_n} \pm \frac{M_z}{r_z W_{nz}} \pm \frac{M_y}{r_y W_{ny}} \le f \tag{8.1.1}$$

2. Stability:

   A. The stability of solid web compression and bending members with bending moment acting in the plane of symmetry axis is considered as follows.

      a. When the bending moment acts on the plane, the stability calculation formula is as follows:

$$\frac{N}{\psi_z A} + \frac{\beta_{mz} M_z}{r_z w_{1z}\left(1 - 0.8\frac{N}{N'_{Ez}}\right)} \le f \tag{8.2.1-1}$$

When the bending moment acts in the plane of the symmetrical axis and the larger flange is pressed, it should be calculated according to the above formula, and also meet the following requirements:

$$\left| \frac{N}{A} - \frac{\beta_{mz} M_z}{\gamma_z w_{2z}\left(1 - 1.25\frac{N}{N'_{Ez}}\right)} \right| \le f \qquad (8.2.1\text{-}4)$$

b. When the bending moment acts outside the plane, the stability calculation formula is as follows:

$$\frac{N}{\phi_y A} + \frac{\beta_{mz} M_z}{\phi_b w_{1z}} \le f \qquad (8.2.1\text{-}3)$$

B. When the bending moment acts on the imaginary axis direction of the lattice compression bending member, the calculation formula of the overall stability in the bending moment action plane is as follows:

$$\frac{N}{\psi_z A} + \frac{\beta_{mz} M_z}{w_{1z}\left(1 - \frac{N}{N'_{Ez}}\right)} \le f \qquad (8.2.2\text{-}1)$$

C. The stability of biaxially symmetric solid web I-shaped and box shaped members with bending moment acting on two main planes is calculated according to the following formula:

$$\frac{N}{\phi_z A} + \frac{\beta_{mz} M_z}{\gamma_z w_{1z}\left(1 - 0.8\frac{N}{N_{Ez}}\right)} + \frac{\beta_{ty} M_y}{\phi_{by} W_{1y}} \le f \qquad (8.2.5\text{-}1)$$

$$\frac{N}{\phi_y A} + \frac{\beta_{my} M_y}{\gamma_y w_{1y}\left(1 - 0.8\frac{N}{N_{Ey}}\right)} + \frac{\beta_{tz} M_z}{\phi_{bz} W_{1z}} \le f \qquad (8.2.5\text{-}2)$$

D. For the double leg lattice beam columns with bending moment acting on two main planes, the overall stability is calculated by the following formula:

$$\frac{N}{\phi_z A} + \frac{\beta_{mz} M_z}{w_{1z}\left(1 - \frac{N}{N'_{Ez}}\right)} + \frac{\beta_{ty} M_y}{W_{1y}} \le f \qquad (8.2.6\text{-}1)$$

3. Shear strength

The calculation method is the same as that of flexural member.

4. Local stability (width thickness ration / height thickness ratio)

The calculated value is the same as that of flexural member.

See table 3.5.1 of GB 50017-2017 for calculation limit.

5. Slenderness ratio

The calculation method is coaxial center stress member

Deflection Check

In the process of deflection test, various conditions in Chapter 3, section 3 of the code are considered. If you need to consider the provisions of member deflection in the process of specification inspection and component selection, it is also necessary to consider the control by design parameters `DFF`, `DJ1`, and `DJ2`.

Design Parameters

You can completely control the whole design process by modifying the design parameters.

These parameters are used by design engineers to communicate their design decisions to computer programs. The default values of all these parameters are selected as the common values which are more inclined to the security design. For a specific design requirement, some or all of the parameters need to be changed to meet the actual design requirements.

Related Links

- *D. To add a parameter set* (on page 980)

Member Type

In GB 50017-2017, members are divided into three categories: flexural members, axially loaded members, and tension (compression)+ bending members. You can force some components to be any of the above types. In this case, the program will check according to the type specified regardless of the geometry or stress. You can also let the program judge automatically. When the program judges automatically, there are two judging principles: according to the geometric position or according to the force characteristics. If you do not specify the component type, the default setting of the program is automatic judgment. The classification principles of various components are described below.

1. According to the geometric conditions

The flexural member	member is located in the horizontal plane of XZ
The axial force member	member is defined as the member truss
Tension (compression) + bending member	other members

2. According to the actual stress situation of members

The flexural member	member with axial force = zero
The axial force member	member with bending moment = zero
Tension (compression) + bending member	other members

Flexural Member

The flexural members are beam members. The judgment principle used in the program is: if the vertical coordinate difference between the two ends of the component is zero, it is regarded as a flexural member; if the axial force of the component is zero, it is regarded as a flexural member. The clause of flexural member shall be used in code inspection.

Axially Loaded Members

The axially loaded members are column members or truss members. The judgment principle adopted in the program is: when judging by geometry, the axial force member will not be generated; if the bending moment of the member is zero, that is, it is not subjected to bending moment, it is regarded as the axial force member. In the standard inspection, the axial force member clause is used.

Tension (Compression) Bending Member

In addition to the above-mentioned flexural members and axially loaded members, all other members in geometric or stress states are regarded as tension (compression) bending members in the program. In the standard inspection, the terms of tension bending and compression bending members are used.

Custom Component

You can choose the inspection terms freely by crossing, and can mix different types of component terms.

Steel Grade

From the following table, the design values of tensile, compressive and bending strength f and shear strength V_f are obtained respectively:

Table 42: Design value of steel strength (N/mm²)

Steel Grade		Thickness or diameter (mm)	Design Value of Strength			Yield strength f_y	Tensile strength f_u
			Tension, compression ,bending f	Shear f_v	End bearing (planed and closely fitted) f_{ce}		
Carbon structural steel	Q235	≤16	215	125	320	235	370
		>16~40	205	120		225	
		>40~100	200	115		215	
Low alloy high strength structural steel	Q345	≤16	305	175	400	345	470
		>16~40	295	170		335	
		>40~63	290	165		325	
		>63~80	280	160		315	
		>80~100	270	155		305	
	Q390	≤16	345	200	415	390	490
		>16~40	330	190		370	
		>40~63	310	180		350	
		>63~100	295	170		330	
	Q420	≤16	375	215	440	420	520
		>16~40	355	205		400	
		>40~63	320	185		380	
		>63~100	305	175		360	
	Q460	≤16	410	235	470	460	550
		>16~40	390	225		440	
		>40~63	355	205		420	

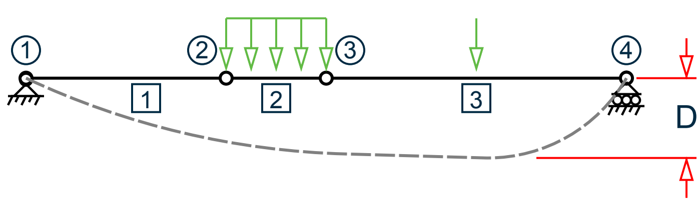
Steel Grade		Thickness or diameter (mm)	Design Value of Strength			Yield strength f_y	Tensile strength f_u
			Tension, compression ,bending f	Shear f_v	End bearing (planed and closely fitted) f_{ce}		
		>63~100	340	195		400	

In each inspection clause in the specification, the value of material strength used is extracted from the data specified here.

You can define a certain steel type manually, as long as the corresponding strength index can be specified. If you do not define and select the steel type, the program default value is set to Q235 steel.

Deformation Parameters

In the process of deflection inspection, various clauses in Appendix B of GB 50017-2017 are considered. If you need to consider the provisions of member deflection in the process of specification inspection and component selection, it can be controlled by three parameters `DFF`, `DJ1`, and `DJ2`. The `DFF` parameter takes the ratio of span to maximum allowable deformation as the limit value of deformation, which is called “deflection length”. The “flexure length” is the length of the member used to calculate the local deformation between the two ends. In most cases, the length of the member is equal to the length of the member. However, in some special cases, the “flexural length” may be different from the member length. For example, the beam shown in the following figure has 4 nodes and 3 members (in this case, two nodes are added in the middle of the component). In this case, for all three beam members, the “flexural length” is defined as the total length of the beam. Parameters `DJ1` and `DJ2` can be used to define this situation. At the same time, the connecting line between `DJ1` and `DJ2` is also the reference line for measuring the local deflection of the rod. Therefore, for all three members, `DJ1` should be node 1 and `DJ2` should be node 4. If `DJ1` and `DJ2` are not defined, the default value of “flexural length” is the length of the member (`DJ1` and `DJ2` take the end node of the member respectively), and the local displacement will be calculated along the line (i.e. axis) of the two ends of the member.



D = Maximum local deflection for members 1, 2, and 3.

The default setting value of the program is DFF = 400; DJ1 and DJ2 are the end nodes of components respectively. For DFF, you should take the denominator value of the corresponding type of allowable deflection in Appendix B according to the provisions of the code GB 50017-2017, and consider the provisions of other clauses.

Plastic Development Coefficients γ_x, γ_y, γ_{sharp}

The plastic development coefficient is a coefficient related to the shape of the section. The coefficient can be according to "specification" the provisions of article 6.1.2 8.1.1, table, through the STAAD/CHINA specification of CHINA steel structure design parameters γ_z, γ_y, γ_{limb} pointed to determine.

The program can automatically determine these coefficients according to the section type of the designed member, namely:

For I - section, take $\gamma_z = 1.05$, $\gamma_y = 1.20$;
For box section, take $\gamma_z = 1.05$, $\gamma_y = 1.05$;
For T steel limb tip parts $\gamma_{sharp} = 1.2$;
For other sections, use table 8.1.1.

If you enter the design parameters γ_z, γ_y, γ_{sharp}, the program will utilize those values. If you select to use automatic calculation, the program will be in accordance with your choice of sectional form, automatic computing (select) γ_z, γ_y, γ_{sharp}. Parameter γ_z, γ_y, γ_{sharp} the default value for the automatic calculation.

Stability Coefficient

The stability coefficient is divided into overall stability coefficient of flexural member (beam) and stability coefficient of axial compression member (truss). The program can automatically judge the coefficient according to the section type and force characteristics of the designed member. The built-in judgment principles in the program are as follows.

1. The overall stability coefficient of flexural members (beams) is calculated by the approximate calculation method in Appendix C of the code.

   For I-section and $\lambda_y > 120\varepsilon_k$

   $$\phi_b = \beta_b \frac{4,\ 320}{\lambda_y} \frac{Ah}{W_x} \left[\sqrt{1 + \left(\frac{\lambda_y t_1}{4.4h} \right)^2} + \eta_b \right] \varepsilon_k$$

   For I-section and $\lambda_y \le 120\varepsilon_k$

   a. When the cross section is biaxially symmetric:

   $$\phi_b = 1.07 - \frac{\lambda_y^2}{44,\ 000} \frac{f_y}{235}$$

   b. When the cross section is uniaxially symmetric:

   $$\phi_b = 1.07 - \frac{W_{1z}}{(2a_b + 0.1)Ah} \frac{\lambda_y^2}{14,\ 000} \frac{f_y}{235}$$

   T-section (moment acting on the plane of symmetry axis):

   a. When the flange is in compression by bending moment: double angle steel

   $$\phi_b = 1 - 0.0017\lambda_y \sqrt{\frac{f_y}{235}}$$

   Two board combination

   $$\phi_b = 1 - 0.0022\lambda_y \sqrt{\frac{f_y}{235}}$$

   b. When the flange is tensioned by bending moment: $\phi_b = 1.0$

When the value calculated by the formula is greater than 1.0, take 1.0.

2. For the axial compression member (truss), the calculation method of Appendix D in the code is used. It is related to the section type and slenderness ratio. The classification of section types is shown in table 7.2.1-1. You can also input data to specify the stability coefficient of the component. The default value of the program is automatic calculation, that is, the program automatically calculates the stability coefficient of the component according to the section type and force characteristics of the designed member.

Equivalent Moment Coefficient

The equivalent moment coefficient is used to “correct” the bending moment when checking the stability of the solid web bending member with bending moment acting on the symmetry axis plane, so it is divided into in-plane equivalent moment coefficient and out of plane equivalent moment coefficient. These coefficients can be calculated automatically by program or manually input.

In the dialog box of the program, the default value is 1.0. You can also specify them as other values or as automatic calculation. If the you specify automatic calculation, the program will automatically calculate the equivalent moment coefficient of the member according to the specification in 8.2.2.

Note: When members are pure and weakly braced frames without considering second-order analysis, the coefficient should be taken as 1 without reduction.

At the same time, it is necessary to specify whether there is lateral load in the direction of the main shaft.

Unsupported Effective Length and Effective Length Coefficient

The unsupported effective length and effective length coefficient are the calculation parameters needed to check the stability of members.

The direction of the strong axis and the weak axis of the principal inertial axis should be considered respectively. For complex spatial structures, it is very difficult to select the effective length and effective length coefficient of members. For the convenience of users, the function of automatically calculating these coefficients is set up in the program. Of course, users can also input these coefficients themselves. When these coefficients are automatically calculated by the program, they are calculated according to the following principles.

The general calculation length formula of members is as follows:

$$l_o = \mu l$$

For the principal inertial axis (the strong axis is the local Z-axis and the weak axis is the local Y-axis), the effective length of the member can be expressed as follows:

$$l_{zo} = l_z \mu_z$$

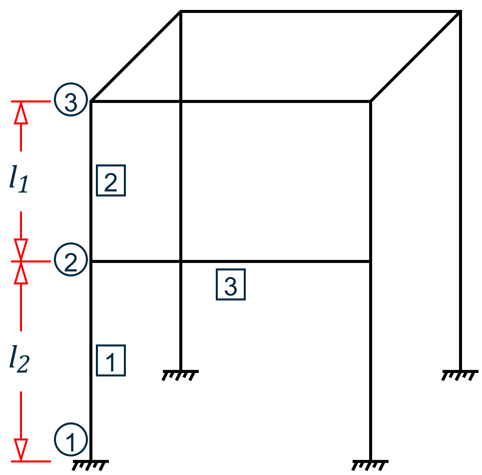
$$l_{yo} = l_y \mu_y$$

where

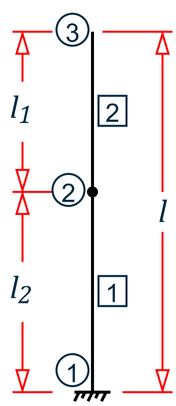
l_{zo} and l_{yo}	=	the effective lengths of the stability analysis along the strong axis (local Z axis) and weak axis (local Y axis) around the principal inertial axis, respectively;
μ_z and μ_y	=	the effective length coefficients in the strong axis direction and the weak axis direction, respectively; For P-Delta solution sets, the effective length coefficients are always taken as 1.0.

l_z and l_y	=	the unsupported lengths in the strong axis direction and the weak axis direction, respectively.

In the Chinese Steel Design workflow, the structure is divided into element and member system, and analyzed and calculated by finite element method. In some cases, it is possible that only one direction (such as the main axis direction) is supported at the node, while the other direction (such as the weak axis direction) is not supported. In this case, l_z and l_y are not equal, and they are not necessarily the distance between two nodes. For example, for the element [1] in the figure below, there is a horizontal member (unit 3) supporting at node 2 in the direction of the main axis, while there is no support at node 2 in the direction of the secondary axis. Therefore, the unsupported length of the member [1] in the direction of the principal axis $l_z = l_1$ and the unsupported length $l_y = l_1 + l_2$ in the secondary axis direction.



In some cases, you may add several nodes between members, and these nodes are only used for calculation, not nodes in the geometric sense. For example, as shown in the figure below, if a node (node 2) is added in the middle of a vertical column, then the unsupported length of element [1] and element [2] should be l instead of l_1 or l_2 during stability check.



For this kind of component, there is no problem in the structural analysis. But for the component design, it must be considered.

According to the provisions of GB 50017-2017, the members requiring stability analysis and calculation can be divided into three categories (the following are divided into three categories A, B and C).

Truss Chords and Single Web Members

For this kind of member, the provisions in table 7.4.1-1 in the specification can be adopted μ_z and μ_y.

For example, the Chord in the plane of the truss, $\mu_z = \mu_y = 1$,and web in the plane of the truss $\mu_z = \mu_y = 0.8$.

For this kind of component, you need to input the calculated length.

Uniform Section Column of Single or Multi Story Frame

For frame columns, the ratios K1 and K2 of the sum of the linear stiffness of the beams intersecting the upper and lower ends of the column and the sum of the linear stiffness of the column are required, and then the following equation is solved

For non-sway frame columns:

$$\mu = \sqrt{\frac{7.5K_1K_2 + 4(K_1 + K_2) + 1.52}{7.5K_1K_2 + K_1 + K_2}}$$

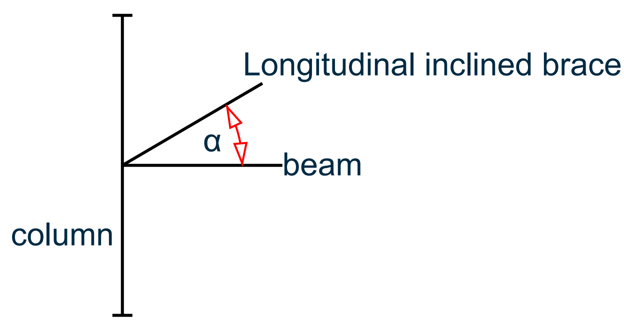
For sway frame columns:

$$\mu = \sqrt{\frac{(1 + 0.41K_1)(1 + 0.41K_2)}{(1 + 0.82K_1)(1 + 0.82K_2)}}$$

In the Chinese Steel Design workflow, you can input the length coefficient and unsupported length of members, and it can also be calculated automatically by the program. In this case, it is necessary to specify the supporting condition of the component (whether the two principal axes belong to the component with or without lateral displacement respectively).

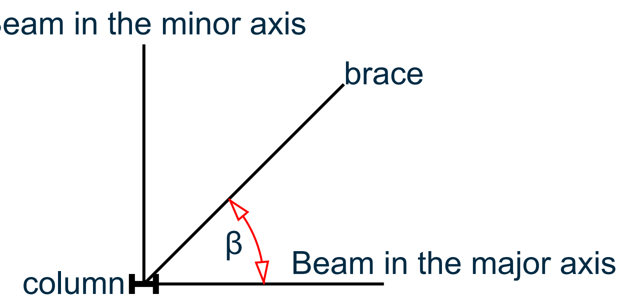
For the calculation of unsupported length, there are several cases as follows:

1. The longitudinal inclined strut is shown in the figure below.



In this case, the program will judge whether the constraint effect of the bar on the column should be considered according to the size of α angle. If $\alpha \leq 60$ ° (this can be specified), the program considers that the strut has constraint on the corresponding main axis direction of the column, and the linear stiffness of the strut is added to the total linear stiffness of the beam in proportion to calculate the value of K1 or K2.

2. Horizontal non principal inertial axis direction strut, as shown in the following figure:



In this case, the program will also automatically judge whether the constraint of the bar to the main axis of the column should be considered according to the size of the β angle.

If β is less than 60 °, which you can be specify, the strut is considered as a constraint on the strong axis direction of the column, and the linear stiffness of the strut is added to the total linear stiffness of the beam in the strong axis direction of the column.

3. The column extension angle is used to allow the engineer to set the maximum angle if the column is not on the same line but has an included angle when calculating the unsupported length of the column. Note that the angle should not exceed 30 degrees.

   Finally, the unsupported length (l_z and l_y) and the effective length coefficient of the column in the strong axis and weak axis direction ($\mu_z = \mu_y$) are calculated automatically

Rigid Fixed Stepped Column at the Lower End of Frame of Single Story Workshop

This kind of column is mostly used in the concrete column of heavy workshop. Generally, there are single and double order columns. Because in the steel structure, this kind of stepped column is not very useful. Therefore, for this kind of component, the user should input it manually after calculation according to 8.3.2 and 8.3.3 of the specification.

For single angle steel, when the length coefficient is calculated automatically, you can select to calculate by parallel limb (Y-Z axis) or strong weak axis (U-V axis) in the Design Parameters dialog box.

Allowable Slenderness Ratio

The allowable slenderness ratio of a member is the limit of the ratio between the effective length and the radius of rotation. The actual slenderness ratio of a member must be less than or equal to the allowable slenderness ratio.

7.2.2 the slenderness ratio of solid web member shall be determined by the following formula according to its instability mode:

1. For members whose section centroid coincides with shear center:
   a. When calculating the bending buckling, the slenderness ratio is calculated as follows:

$$\lambda_x = \frac{l_{ox}}{i_x} \qquad (7.2.2\text{-}1)$$

$$\lambda_y = \frac{l_{oy}}{i_y} \qquad (7.2.2\text{-}1)$$

where

l_{ox} and l_{oy}	=	the effective lengths of members to the section principal axis X and Y respectively, in accordance with section 7.4 of this specification;

i_x and i_y	=	the radius of rotation of the member section to the principal axis X and Y respectively.

b. When calculating torsional buckling, the slenderness ratio shall be calculated according to the following formula. If the width thickness ratio of biaxially symmetric cross-section plate does not exceed $15\varepsilon_k$, torsional buckling may not be calculated.

$$\lambda_z = \sqrt{\frac{I_0}{I_t / 25.7 + I_\omega / l_\omega^2}} \quad (7.2.2\text{-}3)$$

where

I_0, I_t, and I_ω	=	polar moment of inertia (MM4), free torsion constant (MM4) and sectorial moment of inertia (MM6) of the shear center of the member's gross section, respectively. For the cross section, $I_\omega = 0$ can be approximately taken;
l_ω	=	the effective length of torsional buckling, if both ends are hinged and the end section can be warped freely, the geometric length l is taken; if both ends are clamped and the warpage of end section is completely constrained, it is taken as $0.5l$.

2. Members with uniaxial symmetry section:

a. The slenderness ratio should be determined by formula (7.2.2-1) and formula (7.2.2-2) when calculating the bending buckling around the asymmetric principal axis. The slenderness ratio should be determined according to the following formula when calculating the bending torsional buckling around the symmetric principal axis:

$$\lambda_{yz} = \frac{1}{\sqrt{2}}\left[(\lambda_y^2 + \lambda_z^2) + \sqrt{(\lambda_y^2 + \lambda_z^2)^2 - 4\left(1 - \frac{y_s^2}{i_0^2}\right)\lambda_y^2\lambda_z^2}\right] \quad (7.2.2\text{-}4)$$

where

y_s	=	the distance from the centroid of the section to the shear center;
i_0	=	the polar radius of rotation of the cross-section of the shear center, for uniaxially symmetric section, $i_o^2 = i_x^2 + i_y^2$
λ_z	=	the equivalent slenderness ratio of torsional buckling is determined by formula (7.2.2-3).

b. When the effective length of the equal leg single angle steel axial compression member is equal, the bending torsion buckling can be omitted. The single angle strut of tower shall comply with the relevant provisions in section 7.6 of this specification.

c. The equivalent slenderness ratio λ_{yz} of T-section members with double angle steel combination around the axis of symmetry can be determined by the following simplified formula:

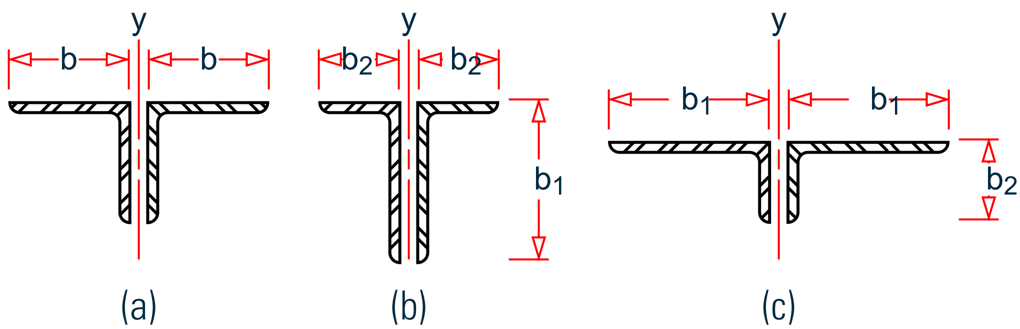


Figure 123: Double angle steel combined T-section (Figure 7.2.2.-1)

where

b	=	the leg width of equal angle;
b_1	=	the long leg width of unequal angle;
b_2	=	the short leg width of unequal angle;

Equal double angle steel [Fig. 7.2.2-1 (a)]

- When $\lambda_z \geq \lambda_y$

$$\lambda_{yz} = \lambda_y\left[1 + 0.16\left(\frac{\lambda_z}{\lambda_y}\right)^2\right] \quad (7.2.2\text{-}5)$$

- When $\lambda_z < \lambda_y$

$$\lambda_{yz} = \lambda_z\left[1 + 0.16\left(\frac{\lambda_y}{\lambda_z}\right)^2\right] \quad (7.2.2\text{-}6)$$

$$\lambda_z = 3.9\frac{b}{t} \quad (7.2.2\text{-}7)$$

Unequal double angle steel with parallel long legs [Fig. 7.2.2-1 (b)]:

- When $\lambda_z \geq \lambda_y$

$$\lambda_{yz} = \lambda_y\left[1 + 0.25\left(\frac{\lambda_z}{\lambda_y}\right)^2\right] \quad (7.2.2\text{-}8)$$

- When $\lambda_z < \lambda_y$

$$\lambda_{yz} = \lambda_z\left[1 + 0.25\left(\frac{\lambda_y}{\lambda_z}\right)^2\right] \quad (7.2.2\text{-}9)$$

$$\lambda_z = 5.1\frac{b_2}{t} \quad (7.2.2\text{-}10)$$

Unequal double angle steel with parallel short legs [Fig. 7.2.2-1 (c)]:

- When $\lambda_z \geq \lambda_y$

$$\lambda_{yz} = \lambda_y\left[1 + 0.06\left(\frac{\lambda_z}{\lambda_y}\right)^2\right] \quad (7.2.2\text{-}11)$$

- When $\lambda_z < \lambda_y$

$$\lambda_{yz} = \lambda_z\left[1 + 0.06\left(\frac{\lambda_y}{\lambda_z}\right)^2\right] \tag{7.2.2-12}$$

$$\lambda_z = 3.7\frac{b_1}{t} \tag{7.2.2-13}$$

d. The members without symmetrical axis and the shear center and centroid do not coincide are not considered in the current program, and are treated as (7.2.2-1) and (7.2.2-2)

e. The conversion slenderness ratio of axially compressed members with unequal angle steel can be determined according to the following simplified formula (Fig. 7.2.2-2)

- When $\lambda_z \geq \lambda_y$

$$\lambda_{xyz} = \lambda_v\left[1 + 0.06\left(\frac{\lambda_z}{\lambda_v}\right)^2\right] \tag{7.2.2-20}$$

- When $\lambda_z < \lambda_y$

$$\lambda_{xyz} = \lambda_v\left[1 + 0.06\left(\frac{\lambda_v}{\lambda_z}\right)^2\right] \tag{7.2.2-21}$$

$$\lambda_v = 4.21\frac{b_1}{t} \tag{7.2.2-22}$$

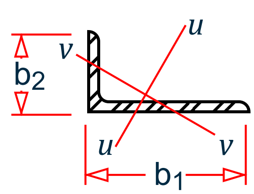


Figure 124: Unequal angle steel (Figure 7.2.2-2)

Note: V axis is the weak axis of angle steel, b_1 is the long leg width of angle steel.

The allowable slenderness ratio is the allowable slenderness ratio of compression member and tension member. You should select the correct value according to the provisions in 7.4.6 and 7.4.7 of the specification. The default values of the program are: allowable slenderness ratio of 150 under compression and 300 under tension. The slenderness ratio of components mainly includes the following situations:

1. Non-seismic:

   a. The slenderness ratio of compression members should not exceed the allowable values given in Table 7.4.6

Table 43: Allowable values of slenderness ratio for compression members (Table 7.4.6)

Nomenclature of members	Allowable values
Columns, members of trusses and monitors.	150
Lacing of columns, column bracings beneath crane girders or crane trusses	150
Bracings	200

Nomenclature of members	Allowable values
Members used to reduce the slenderness ratio of compression members	200

Notes:

i. A slenderness ratio of 200 may be allowed for compression web members in trusses (including space trusses) when they are stressed to or under 50% of their capacities.
ii. For single angle compression members, the least radius of gyration shall be used for calculation of slenderness ratio, but the radius of gyration about the axis parallel to the leg may be used for calculation of the out-of-plane slenderness ratio in the case of cross diagonals interconnected at the crossing point.
iii. For trusses with span length equal to or larger than 60m, the allowable slenderness ratio should be taken as 100 for compression chords and end posts, and 150 (when subjected to static or indirect dynamic load), or 120 (when subjected to direct dynamic load) for other web members in compression.
iv. In case member section is governed by allowable slenderness ratio, the effect of twisting may be neglected in calculating the slenderness ratio.

b. The slenderness ratio of tension members should not exceed the allowable values given in table 7.4.7.

Table 44: Allowable values of slenderness ratio for tension members (Table 7.4.7)

Nomenclature of members	Structures subject to static or indirect dynamic loading			Structures subject to direct dynamic loading
	Common buildings	chord that provides out of plane support for web	Mill buildings with heavy duty crane	
Members of Trusses	350	250	250	250
Column bracings beneath crane girders or crane trusses	300	-	200	-
Other tension members, bracings and ties (except pretensioned round bars)	400	-	350	-

Notes:

i. For structures subject to static loading, slenderness ratio of tension members may be checked only in vertical planes.
ii. For structures subject to direct or indirect dynamic loading, the slenderness ratio of a single angle tension member is calculated similarly to Note 2 of Table 7.4.6.
iii. The slenderness ratio of bottom chord of crane trusses for medium and heavy duty cranes should not exceed 200.

- **iv.** In mill buildings equipped with soaking pit cranes and stripper cranes or rigid claw cranes, the slenderness ratio of the bracings (except Item No.2 in the Table) should not exceed 300.
- **v.** When tension members change into compression ones under the combined action of dead and wind loads, their slenderness ratio should not exceed 250.
- **vi.** For trusses with span length equal to or larger than 60m, the slenderness ratio of tension chords and tension web members should not exceed 300 (when subjected to static or indirect dynamic loading) or 250 (when subjected to direct dynamic loading).

2. Seismic code clause 8.3, 8.4:

Section 8.3.1 states that the slenderness ratio of frame columns shall not be greater than $60\sqrt{235 / f_{ay}}$ for first grade seismic, shall not be greater than $80\sqrt{235 / f_{ay}}$ for second grade seismic, shall not be greater than $100\sqrt{235 / f_{ay}}$ for third grade seismic, and shall not be greater than $120\sqrt{235 / f_{ay}}$ for fourth grade seismic.

Section 8.4.1 The slenderness ratio and the width-to-thickness ratio of the epicenter diagonal brace shall comply with following requirements:

The slenderness ratio of the epicenter diagonal brace which is designed as compress member shall not be greater than $120\sqrt{235 / f_{ay}}$, and the epicenter diagonal brace of the first, second and third seismic grade shall not be designed as tension member, and the slenderness ratio of the epicenter diagonal brace which is designed as tension member shall not be greater than 80 for fourth seismic grade.

3. Seismic code clause 9.2.13:

The slenderness ratio of the frame column of the single-story steel factory, which should not be greater than 150 when the axial compression ratio is less than 0.2, and it should not be greater than $120\sqrt{235 / f_{ay}}$ when the axial compression ratio is not less than 0.2.

The limit of slenderness ratio of the supporting members between columns should meet the requirements of the steel code GB 50017-2017.

4. Seismic code clause H.2.8, for multistory factory structures

H.2.8 the basic anti-seismic structural measures of multi-story steel structure workshop should also meet the following requirements:

- **I.** The slenderness ratio of frame column should not be greater than 150; when the axial compression ratio is greater than 0.2, it should not be greater than $125(1 - 0.8N / A_f)\sqrt{235 / f_{ay}}$.
- **II.** The width thickness ratio of the frame columns and beams of the workshop shall meet the following requirements:
  - **i.** The single-layer part and the multi-layer part with height greater than 40m can be implemented according to Section 9.2 of this specification;
  - **ii.** When the total height of the multi-layer part is greater than 40m, the provisions in section 8.3 of this specification can be followed.
- **III.** In the maximum stress area of frame beam and column, the flange section shall not be changed suddenly, and the upper and lower flange shall be provided with lateral support. The relationship between the support point and the adjacent support point shall meet the relevant requirements of plastic design in steel structure code GB 50017-2017.
- **IV.** The supporting members between columns should meet the following requirements:
  - **i.** The column bracing of multi-storey frame should form X-shape or other anti-seismic forms with frame beam, and its slenderness ratio should not be greater than 150;

ii. The width thickness ratio of the supporting member shall meet the requirements of Section 9.2 of this specification.

When the program is selected to automatically calculate the slenderness ratio limit, the program will calculate according to the rules in the dialog box. You can select all of them to meet the requirements, or you can specify the filtering calculation rules.

Construction Requirements

The structural requirement parameters specified here refer to the limits of flange width thickness ratio and web height thickness ratio, as well as the section type of axial compression members. Generally, this parameter is automatically calculated by the program. However, in order to make it easier for users to design the program more flexibly, you can also set the parameters required here.

Flange Width Thickness Ratio / Web Height Thickness Ratio

The local stability inspection of components must meet the requirements of table 3.1.5 and 7.3.1 of steel code (GB 50017-2017).

When considering earthquake resistance, it should also meet the seismic code.

For the framed structures, the width-to-thickness ratio of the elements of beam and column shall conform to the requirements in Table 8.3.2.

Table 45: Width-to-thickness ratio limit of the elements of beam and column (Table 8.3.2)

Element Name		Grade 1	Grade 2	Grade 3	Grade 4
column	Flange overhanging part of I-section	10	11	12	13
	Web of I-section	43	45	48	52
	Plates of box-section	33	36	38	40
Beam	Flange overhanging part of I-section or box- section	9	9	10	11
	Flanges between both webs of box-section	30	30	32	36
	Webs of I-section or box-section	72-120Nb /(Af) £60	72-100Nb /(Af) £65	80-110Nb /(Af) £70	85-120Nb /(Af) £75

Note:

i. the values listed in the table is applicable to Q235 steel, other steel types shall multiply by $\sqrt{235 / f_{ay}}$.

ii. $N_b / (Af)$ is the axial compression ratio of the beam.

In the program, it is necessary to specify the seismic design or seismic grade to consider the restrictive provisions of the seismic code. For the members using seismic design (specified in the **Chinese Steel Design** dialog), their strength and stability should be reduced according to the code, and specified in the **Reduction Factor of Bearing Capacity**.

Note: When the circular pipe is a beam, the diameter thickness ratio is not specified, and the value shall be taken according to the column in table 3.5.1.

When the seismic option is selected, the program will check the provisions in the Table 8.3.2 and 8.4.1 of the seismic code.

When checking the seismic height thickness ratio of beam webs, when the tested load combination does not include seismic conditions, the seismic height thickness ratio limits of beam webs shall meet the requirements of grade I ≤ 60, grade II ≤ 65, grade III ≤ 70 and grade IV ≤ 75 respectively.

Section Coefficient

The net section coefficient refers to the ratio of net section to gross section area and section modulus (temporarily replaced by net section modulus coefficients an, WNZ and wny). The default value is 1.0.

The reduction coefficient of seismic bearing capacity of bracing is to reduce the bearing capacity of members in the presence of seismic load. The effective section coefficient of axially loaded members refers to (considering article 7.1.3 of code GB 50017-2017) that for axial tension and axial compression members, the area of dangerous section is multiplied by the effective section coefficient h in case of indirect force transmission.

D. Chinese Steel Design Parameter File Format

The parameters for steel design per GB 50017-2017 are stored in a separate file (file extension `.gsp`).

Example GSP File

Click the **Edit Parameters (Text)** button in the **Chinese Steel Design Parameters - Whole Structure** dialog to open and edit this file.

Note: No data validation of any kind is performed in the plain text editor. Therefore, the accuracy and validity of these changes is dependent upon the user to verify.

```
[version=2207]

*{ The below data is for code check general information, please do not modify it.

[CheckOptionAngle]

PrimaryAxis=60.000000

SecondaryAxis=60.000000

ExtendLine=10.000000

[CodeCheck]

SelectAll=1
```

```
GroupOptimize=0

FastOptimize=0

Iteration=0

SeismicGrade=None

BeamBendingStrength=1

BeamShearStrength=1

BeamEquivalentStress=1

BeamOverallStability=1

BeamSlendernessWeb=1

BeamSlendernessFlange=1

TrussStrength=1

TrussStability=1

TrussShearStrength=1

ColumnStrength=1

ColumnStabilityMzMy=1

ColumnStabilityMyMz=1

PressedTrussSlenderness=1

TensionTrussSlenderness=1

ColumnSlendernessFlange=1

ColumnSlendernessWeb=1

BeamDeflection=1

SecondaryMembers=

SectCollectionOrder=0

*{ The above data is for code check general information, please do not modify
it.



[GROUP=1]

Name(Parameter Name)=SUBBEAM

Type(Member Type)=1
```

```
Principle(Principle Rules)=0

SteelNo()=Q235

SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3

Fatigue(Fatigue Calculation)=0

Optimization(Perform optimized design)=0

MaxFailure(Failure Ratio)=1

MinTooSafe(Safety Ratio)=0.3

CheckLoadCase(Force Loads Case No.)=ALL

CheckDispLoadCase(Displacement Loads Case No.)=ALL

BeamBendingStrength()=1

BeamShearStrength()=1

BeamEquivalentStress()=1

BeamOverallStability()=1

BeamSlendernessFlange(b/t on beam)=1

BeamSlendernessWeb(h0/tw on beam)=1

TrussStrength(Axial Force Strength)=1

SecondaryMoment(Secondary Moment of Truss)=0

TrussStability(Solid-web Axial Compression Stability)=1

TrussShearStrength(Axial Shear Strength)=1

PressedTrussSlenderness(Pressed Member Slenderness)=1

TensionTrussSlenderness(Tension Member Slenderness)=1

ColumnStrength(Column Member Strength)=1

ColumnStabilityMzMy(Column Stability In-plane)=1

ColumnStabilityMyMz(Column Stability Out-plane)=1

ColumnSlendernessFlange(b/t on column)=1

ColumnSlendernessWeb(h0/tw on column)=1

CheckItemAPPENDIX_B11(Beam Deflection)=1

UseAntiSeismic(Use Seismic Adjusting Factor)=0

GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
```

```
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0

SLevel(Grade of Seismic Resistance)=0

lmdc(Slenderness Limit of Compression Member)=0

lmdt(Slenderness Limit of Tension Member)=0

Lmd831(Slenderness of Seismic Column)=0

Lmd841(Slenderness of Seismic Brace)=0

Lmd9213(Slenderness of Seismic Single-story Plant)=0

LmdH28(Slenderness of Seismic Multi-story Plant)=0

rz(Plastic Development Factor in Major Axis)=0

ry(Plastic Development Factor in Minor Axis)=0

gamaSharp(Plastic Development Factor of sharp side)=0

betamz(the equivalent moment factor in Major Axis plane)=0

betamy(the equivalent moment factor in Minor Axis plane)=0

betatz(the equivalent moment factor out Major Axis plane)=0

betaty(the equivalent moment factor out Minor Axis plane)=0

HasHorLoadZ(Has Horizontal Load in Z-Axis)=0

HasHorLoadY(Has Horizontal Load in Y-Axis)=0

DFF(Deflection Limit of Beam)=350

DJ1(Start Node Number in Major Axis)=0

DJ2(End Node Number in Major Axis)=0

Horizontal(Check for Deflection in Minor Axis)=0

Cantilever(Cantilever Member)=0

fabz(Overall Stability Factor in Major Axis of Bending Member)=0

faby(Overall Stability Factor in Minor Axis of Bending Member)=0

StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1

faz(Overall Stability Factor in Major Axis of Axial Compression Member=0

fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0

lz(Unbraced Length in Major Axis)=0

ly(Unbraced Length in Minor Axis)=0
```

```
miuz(Effective Length Factor for Column in Major Axis)=0

miuy(Effective Length Factor for Column in Minor Axis)=0

Lateral(Member in Frame Without Sidesway or not)=0

APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0

rFlange(Limit Ratio of Width to Thickness for Flange)=0

rWeb(Limit Ratio of High to Thickness for Web)=0

BucklingStrength(Axis forced member bulking strength)=0

ZSectType(Section Type in Z-Axis)=0

YSectType(Section Type in Y-Axis)=0

HSectWebInTrussPlane(Web of H in Truss Plane)=0

rAn(Net Factor of Section Area)=1

rWnz(Net Factor of Resistance Moment in Z-Axis)=1

rWny(Net Factor of Resistance Moment in Y-Axis)=1

CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1

AngleReduce(Angle Strength Reduce)=0

LAglConSta(Connect Type of unequal single angle)=0

LAngleStrength(Reduction Factor of Angle Strength)=0

LAngleStability(Reduction Factor of Angle Stability)=0

rTrussSectReduce(Effective Factor of Axial Force Section)=1

Members(Member Number)=23 TO 25 48 TO 53


[GROUP=2]

Name(Parameter Name)=MAINBEAM

Type(Member Type)=1

Principle(Principle Rules)=0

SteelNo()=Q235

SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3

Fatigue(Fatigue Calculation)=0

Optimization(Perform optimized design)=0
```

```
MaxFailure(Failure Ratio)=1

MinTooSafe(Safety Ratio)=0.3

CheckLoadCase(Force Loads Case No.)=ALL

CheckDispLoadCase(Displacement Loads Case No.)=ALL

BeamBendingStrength()=1

BeamShearStrength()=1

BeamEquivalentStress()=1

BeamOverallStability()=1

BeamSlendernessFlange(b/t on beam)=1

BeamSlendernessWeb(h0/tw on beam)=1

TrussStrength(Axial Force Strength)=1

SecondaryMoment(Secondary Moment of Truss)=0

TrussStability(Solid-web Axial Compression Stability)=1

TrussShearStrength(Axial Shear Strength)=1

PressedTrussSlenderness(Pressed Member Slenderness)=1

TensionTrussSlenderness(Tension Member Slenderness)=1

ColumnStrength(Column Member Strength)=1

ColumnStabilityMzMy(Column Stability In-plane)=1

ColumnStabilityMyMz(Column Stability Out-plane)=1

ColumnSlendernessFlange(b/t on column)=1

ColumnSlendernessWeb(h0/tw on column)=1

CheckItemAPPENDIX_B11(Beam Deflection)=1

UseAntiSeismic(Use Seismic Adjusting Factor)=0

GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0

GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0

SLevel(Grade of Seismic Resistance)=0

lmdc(Slenderness Limit of Compression Member)=0

lmdt(Slenderness Limit of Tension Member)=0

Lmd831(Slenderness of Seismic Column)=0
```

```
Lmd841(Slenderness of Seismic Brace)=0

Lmd9213(Slenderness of Seismic Single-story Plant)=0

LmdH28(Slenderness of Seismic Multi-story Plant)=0

rz(Plastic Development Factor in Major Axis)=0

ry(Plastic Development Factor in Minor Axis)=0

gamaSharp(Plastic Development Factor of sharp side)=0

betamz(the equivalent moment factor in Major Axis plane)=0

betamy(the equivalent moment factor in Minor Axis plane)=0

betatz(the equivalent moment factor out Major Axis plane)=0

betaty(the equivalent moment factor out Minor Axis plane)=0

HasHorLoadZ(Has Horizontal Load in Z-Axis)=0

HasHorLoadY(Has Horizontal Load in Y-Axis)=0

DFF(Deflection Limit of Beam)=400

DJ1(Start Node Number in Major Axis)=0

DJ2(End Node Number in Major Axis)=0

Horizontal(Check for Deflection in Minor Axis)=0

Cantilever(Cantilever Member)=0

fabz(Overall Stability Factor in Major Axis of Bending Member)=0

faby(Overall Stability Factor in Minor Axis of Bending Member)=0

StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1

faz(Overall Stability Factor in Major Axis of Axial Compression Member=0

fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0

lz(Unbraced Length in Major Axis)=0

ly(Unbraced Length in Minor Axis)=0

miuz(Effective Length Factor for Column in Major Axis)=0

miuy(Effective Length Factor for Column in Minor Axis)=0

Lateral(Member in Frame Without Sidesway or not)=0

APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
```

```
rFlange(Limit Ratio of Width to Thickness for Flange)=0

rWeb(Limit Ratio of High to Thickness for Web)=0

BucklingStrength(Axis forced member bulking strength)=0

ZSectType(Section Type in Z-Axis)=0

YSectType(Section Type in Y-Axis)=0

HSectWebInTrussPlane(Web of H in Truss Plane)=0

rAn(Net Factor of Section Area)=1

rWnz(Net Factor of Resistance Moment in Z-Axis)=1

rWny(Net Factor of Resistance Moment in Y-Axis)=1

CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1

AngleReduce(Angle Strength Reduce)=0

LAglConSta(Connect Type of unequal single angle)=0

LAngleStrength(Reduction Factor of Angle Strength)=0

LAngleStability(Reduction Factor of Angle Stability)=0

rTrussSectReduce(Effective Factor of Axial Force Section)=1

Members(Member Number)=4 5 32 33 36 TO 47


[GROUP=3]

Name(Parameter Name)=COLUMN

Type(Member Type)=3

Principle(Principle Rules)=0

SteelNo()=Q235

SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3

Fatigue(Fatigue Calculation)=0

Optimization(Perform optimized design)=0

MaxFailure(Failure Ratio)=1

MinTooSafe(Safety Ratio)=0.3

CheckLoadCase(Force Loads Case No.)=ALL

CheckDispLoadCase(Displacement Loads Case No.)=ALL
```

```
BeamBendingStrength()=1

BeamShearStrength()=1

BeamEquivalentStress()=1

BeamOverallStability()=1

BeamSlendernessFlange(b/t on beam)=1

BeamSlendernessWeb(h0/tw on beam)=1

TrussStrength(Axial Force Strength)=1

SecondaryMoment(Secondary Moment of Truss)=0

TrussStability(Solid-web Axial Compression Stability)=1

TrussShearStrength(Axial Shear Strength)=1

PressedTrussSlenderness(Pressed Member Slenderness)=1

TensionTrussSlenderness(Tension Member Slenderness)=1

ColumnStrength(Column Member Strength)=1

ColumnStabilityMzMy(Column Stability In-plane)=1

ColumnStabilityMyMz(Column Stability Out-plane)=1

ColumnSlendernessFlange(b/t on column)=1

ColumnSlendernessWeb(h0/tw on column)=1

CheckItemAPPENDIX_B11(Beam Deflection)=1

UseAntiSeismic(Use Seismic Adjusting Factor)=0

GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0

GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0

SLevel(Grade of Seismic Resistance)=0

lmdc(Slenderness Limit of Compression Member)=0

lmdt(Slenderness Limit of Tension Member)=0

Lmd831(Slenderness of Seismic Column)=0

Lmd841(Slenderness of Seismic Brace)=0

Lmd9213(Slenderness of Seismic Single-story Plant)=0

LmdH28(Slenderness of Seismic Multi-story Plant)=0

rz(Plastic Development Factor in Major Axis)=0
```

```
ry(Plastic Development Factor in Minor Axis)=0

gamaSharp(Plastic Development Factor of sharp side)=0

betamz(the equivalent moment factor in Major Axis plane)=0

betamy(the equivalent moment factor in Minor Axis plane)=0

betatz(the equivalent moment factor out Major Axis plane)=0

betaty(the equivalent moment factor out Minor Axis plane)=0

HasHorLoadZ(Has Horizontal Load in Z-Axis)=0

HasHorLoadY(Has Horizontal Load in Y-Axis)=0

DFF(Deflection Limit of Beam)=400

DJ1(Start Node Number in Major Axis)=0

DJ2(End Node Number in Major Axis)=0

Horizontal(Check for Deflection in Minor Axis)=0

Cantilever(Cantilever Member)=0

fabz(Overall Stability Factor in Major Axis of Bending Member)=0

faby(Overall Stability Factor in Minor Axis of Bending Member)=0

StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1

faz(Overall Stability Factor in Major Axis of Axial Compression Member=0

fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0

lz(Unbraced Length in Major Axis)=0

ly(Unbraced Length in Minor Axis)=0

miuz(Effective Length Factor for Column in Major Axis)=0

miuy(Effective Length Factor for Column in Minor Axis)=0

Lateral(Member in Frame Without Sidesway or not)=0

APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0

rFlange(Limit Ratio of Width to Thickness for Flange)=0

rWeb(Limit Ratio of High to Thickness for Web)=0

BucklingStrength(Axis forced member bulking strength)=0

ZSectType(Section Type in Z-Axis)=0

YSectType(Section Type in Y-Axis)=0
```

```
HSectWebInTrussPlane(Web of H in Truss Plane)=0

rAn(Net Factor of Section Area)=1

rWnz(Net Factor of Resistance Moment in Z-Axis)=1

rWny(Net Factor of Resistance Moment in Y-Axis)=1

CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1

AngleReduce(Angle Strength Reduce)=0

LAglConSta(Connect Type of unequal single angle)=0

LAngleStrength(Reduction Factor of Angle Strength)=0

LAngleStability(Reduction Factor of Angle Stability)=0

rTrussSectReduce(Effective Factor of Axial Force Section)=1

Members(Member Number)=1 2 7 8 30 31 34 35
```

Code Checking and Member Selection

Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate to carry the forces transmitted to it by the loads on the structure. The adequacy is checked per the GB 50017-2017 requirements.

Code checking is done using forces and moments at specified sections of the members. General they are calculated at every twelfth point along the beam. The code checking output labels the members as PASSED or FAILED. In addition, the critical condition, governing load case, location (distance from start joint) and magnitudes of the governing forces and moments are also printed.

Member Selection

The member selection process basically involves determination of the least weight member that PASS the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table.

Note: Member selection can not be performed on TUBES, PIPES, or members listed as PRISMATIC.

Use of Design Parameters

Users can modify the design parameters and control the whole design process by checking the specified parameters before. These parameters are used by design engineers to communicate their design decisions to computer programs. The default values of all these parameters are selected as the common values which are more inclined to the security design. For a specific design requirement, some or all of the parameters need to be changed to meet the actual design requirements. Therefore, specifying design parameters is often an indispensable step.

Assumptions

There are some default assumptions in the program when performing specification checking.

1. Biaxial symmetry and uniaxial symmetry are specified in the code. When different terms are adopted, the proportion of area difference between upper and lower flanges shall be taken as the judgment basis for H-section program judgment (mainly for H-section with unequal upper and lower flange and H-section with adhesive plate). When the area difference is less than 30%, the cross section is considered to be biaxially symmetric.
2. When calculating the post buckling strength of H-section with plate, the program considers that the laminate on the compression side is invalid, and the effect of the veneer is not considered when calculating the effective section.

Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

Member No.	refers to the member number for which the design is performed
Solution	Lists the solution set associated with this design.
Check	refers to steel design limit state, which has been checked against the steel code or has been selected.
Actual Value	prints the actual stresses for the critical condition, which is calculated by the analysis results and design parameters.
Failure Ratio Above	prints the allowable stresses for the critical condition. Normally which is related to the steel grade.
Normalized Ratio	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
Load case	provides the load case number, which governed the design.

D. Steel Connection Design

D. Connection Design workflow

D. Getting Started

The following concepts will help you understand how to use Connection Design in STAAD.Pro for the design of steel connections in your engineering practice.

D. Overview

Purpose

The Connection Design mode in STAAD.Pro is used to dynamically link structural model data —including section properties and analysis results— to the Connection Design application in order to check connection designs for code compliance. The resulting data and diagrams of the connection can also be included in the User Report.

Note: Not all options described here will be available for versions of Connection Design prior to release 7.0.

Note: Full use of the Connection Design mode in STAAD.Pro requires a valid RAM Connection license. The RAM Connection mode will run with a limited set of steel connections available with only a standard STAAD.Pro license. Please contact your Bentley account manager to have a Connection Design license added to your SELECT licenses.

Description

Connections are designed in the Connection Design Mode by creating “Joints” from the geometry and section properties. Forces resulting from the analysis are used in this mode by assigning a load envelope. A set of connection templates is then assigned to the joint. A suitable connection design, if one is available, will be reported once you have selected the appropriate connection templates.

Any number of joints may be selected and designed. Further, connection design is performed automatically for you once appropriate templates have been selected for the selected joints. These enhancements greatly reduce the time required for connection design in models of all sizes.

Tip: The selected load envelope is now used for all connection designs, instead of per design brief as in previous versions of STAAD.Pro.

Refer to the following procedures and interface elements used in designing steel connections with RAM Connection within STAAD.Pro. For specific information regarding RAM Connection templates, refer to the documentation with RAM Connection.

D. Full vs. Free Connection Sets

Basic connection templates are available to users with a valid STAAD.Pro license. The following connections templates are available when using a standard STAAD.Pro license.

- Basic DA BCF Bolted
- Basic DA BCF Welded
- Basic EP BCF Bolted
- Basic EP BCF Welded
- Basic DA BCW Welded
- Basic EP BCW Welded
- Basic SP BCF
- Basic SP BCW

Note: In order to use all connection templates (including Smart and Gusset connections), a valid RAM Connection license is required.

For a detailed explanation of connections available, refer to AD.2007-1001.5.1 in the What's New section.

D.Steel Connection Design Codes and Connection Types

The following tables indicate the allowable member types per connection for supported steel design codes in RAM Connection.

Steel design codes in RAM Connection:

Table 46: AISC 360 (on page 1017)
Table 47: BS5950 (on page 1020)
Table 48: GB 50017-03 (on page 1022)
Table 49: EN 1993-2005 (on page 1023)
Table 50: IS 800-2007 (on page 1025)
Table 51: AS 4100-1998 (on page 1027)
Table 52: NZS 3404-1997 (on page 1029)
Table 53: CSA S16-14 (on page 1030)

RAM Connection Section Type	STAAD.Pro Section Type	Notes
I	Wide Flange[a]	
T	Wide Flange (cut into tee; T type specification)	Tee sections
HSS_Rect	Tube	Rectangular hollow sections
HSS_Circ	Pipe	Circular hollow sections
C	Channel	
I2C	Channel (double back-to-back; D type specification)[b]	
L	Angle	Single angles
T2L	Angle (back-to-back; LD or SD type specification)[b]	Double angles

a. Wide flange sections that have been assigned a cover plate (top, bottom or both, with the options: TC, BC, or TB), as a composite beam with the CM option, or a double profile with the D option will be designed using the base profile only.

b. Channel sections and angle sections are only designed with the options defined above. Channels front-to-front (type specification FR) and double angle star arrangement (heel-to-heel; type specification SA) are not designed.

Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2257) for details on STAAD.Pro steel section options.

Table 46: AISC 360

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	I, I2C, HSS_RECT, HSS_CIRC
	CB	Tapered	YES	YES	NO	I
Bent plate	BCF	Prismatic	YES	NO	NO	I
	BCW	Prismatic	YES	NO	NO	I
	BG	Prismatic	YES	NO	NO	I
Bracket	Bracket plate	Prismatic	YES	NO	NO	I
	Tee bracket	Prismatic	YES	NO	NO	T
BS4Angles	BS	Prismatic	YES	YES	NO	I
Cap plate	CP	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
Clip angle	BCF	Prismatic	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	YES	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	YES	NO	I
Directly welded	BCF	Prismatic	YES	YES	Column "YES"	I
	BCW	Prismatic	YES	YES	NO	Beam "I" - Column "I, HSS_RECT"
End plate	BCF	Prismatic	YES	YES	Column "YES"	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Flange plate	CS	Prismatic	YES	YES	Top Column "YES"	I

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCF	Prismatic	YES	YES	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	YES	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	YES	NO	I
	BS	Prismatic	YES	YES	NO	I
Gusset base plate	GBP	Prismatic	YES	Column "YES" - Braces "NO"	Column "YES"	Column "I, HSS_RECT" - Braces "I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC"
Gusset chevron	CVR	Prismatic	YES	BEAM "YES" - Braces "NO"	NO	Beam "I" - Braces "I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC"
Gusset column beam brace	CBB	Prismatic	YES	Column "YES" - Beams "YES" - Braces "NO"	Column "YES"	Column "I, HSS_RECT" - Beam "I" - Braces "I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC"
Gusset VXB	VXB	Prismatic	YES	NO	NO	Braces "I, T, C, I2C, L, T2L, HSS_RECT, HSS_CIRC"
Moment end plate	BS	Prismatic	YES	YES	NO	I
	BS	Tapered	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Column, Beam "Prismatic"	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT"

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCF	Tapered "Beam"	YES	YES	Column "YES"	Beam "I"
	BCF	Haunched "Beam"	YES	YES	Column "YES"	Haunch "I, T"
	BCW	Column, Beam "Prismatic"	YES	YES	NO	Beam "I" - Column "HSS_RECT"
	BCW	Tapered "Beam"	YES	YES	NO	Beam "I"
	BCW	Haunched "Beam"	YES	YES	NO	Haunch "I, T"
Moment end plate Knee	BCF	Tapered	YES	YES	Column "YES"	I
PRConnector	PR	Prismatic	YES	NO	NO	I
Single plate	BS	Prismatic	YES	YES	NO	I
	BCF	Prismatic	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BCW	Prismatic	YES	YES	NO	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BG	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	Top Column "YES"	I
Standard tee	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	NO	NO	I

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Stiffened seated	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
Through plate	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "HSS_RECT, HSS_CIRC"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "HSS_RECT, HSS_CIRC"
Unstiffened seated	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
Tubular truss connections	CHB	Prismatic	NO	NO	NO	"HSS_RECT, HSS_CIRC"

Table 47: BS5950

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Bolted end plate	BS	Prismatic	YES	NO	NO	I
	BS	Tapered	YES	NO	NO	I
	BS	Haunched	YES	NO	NO	I, T
Cleat angle	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	NO	NO	I
Fully welded	BCF	Prismatic	YES	NO	NO	I
	BCW	Prismatic	YES	NO	NO	I
Flanges welded	BCF	Prismatic	YES	NO	NO	I
	BCW	Prismatic	YES	NO	NO	I
Fin plate	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BG	Prismatic	YES	NO	NO	I
Flange cover plate	BS	Prismatic	YES	NO	NO	I
	CS	Prismatic	YES	NO	NO	I
Flexible end plate	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	NO	NO	I
Moment end plate	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I" - Hauch "I, T"
	BCF	Haunched	YES	NO	NO	Beam "I" - Column "I" - Hauch "I, T"

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Web cover plate	BS	Prismatic	YES	NO	NO	I
	CS	Prismatic	YES	NO	NO	I

Table 48: GB 50017-03

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	NO	NO	Column "I, I2C, HSS_RECT, HSS_CIRC"
		Tapered	YES	NO	NO	I
Clip angle	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	NO	NO	Beam "I" - Girder "I, HSS_RECT"
Directly welded	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT"
Flange plate	BCF	Prismatic	YES	NO	NO	I
	BCW	Prismatic	YES	NO	NO	I
	BG	Prismatic	YES	NO	NO	I
Moment end plate	BCF	Column, Beam "Prismatic"	YES	YES	NO	Beam "I" - Column "I, HSS_RECT"

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCF	Tapered "Beam"	YES	YES	NO	Beam "I"
	BCF	Haunched "Beam"	YES	YES	NO	Haunch "I, T"
	BCW	Column, Beam "Prismatic"	YES	YES	NO	Beam "I" - Column "HSS_RECT"
	BCW	Tapered "Beam"	YES	YES	NO	Beam "I"
	BCW	Haunched "Beam"	YES	YES	NO	Haunch "I, T"
Single plate	BS	Prismatic	YES	NO	NO	I
	BCF	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BCW	Prismatic	YES	NO	NO	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BG	Prismatic	YES	NO	NO	I

Table 49: EN 1993-2005

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	Column "I, I2C, HSS_RECT, HSS_CIRC"
Bolted end plate	BS	Prismatic	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Column, Beam "Prismatic"	YES	YES	Column "YES"	I

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCF	Haunched "Beam"	YES	YES	Column "YES"	Haunch "I, T"
Cleat angle	BCF	Prismatic	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	YES		Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	YES		I
Cleat angle	CS	Prismatic	YES	YES	Top Column "YES"	I
Fully welded BCF	BCF	Prismatic	YES	YES	Column "YES"	I
Fully welded BCW	BCW	Prismatic	YES	YES		I
Flanges welded BCF	BCF	Prismatic	YES	YES	Column "YES"	I
Flanges welded BCW	BCW	Prismatic	YES	YES		I
End plate	BCF	Prismatic	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT"
	BCW	Prismatic	YES	YES		Beam "I" - Column "I, HSS_RECT"
	BG	Prismatic	YES	YES		I
	CS	Prismatic	YES	NO		HSS_RECT, HSS_CIRC
Fin plate	BCF	Prismatic	YES	YES	Column "YES"	Beam "I" - Column "I, HSS_RECT, HSS_CIRC"

Connection	Family	Member Types	Built-up Symmetric	Built-up Nonsymmetric	Member Flange Rotation	Sections Allowed
	BCW	Prismatic	YES	YES		Beam "I" - Column "I, HSS_RECT, HSS_CIRC"
	BG	Prismatic	YES	YES		I
Flange cover plate	BS	Prismatic	YES	YES		I
	CS	Prismatic	YES	YES	Top Column "YES"	I
Gusset column beam brace	CBB	Prismatic	YES	Column "YES" - Beams "YES" - Braces "NO"	Column "YES"	Column "I" - Beam "I" - Braces "C, I2C, L, T2L, HSS_RECT, HSS_CIRC"
Web cover plate	BS	Prismatic	YES	YES		I
	CS	Prismatic	YES	YES	Top Column "YES"	I

Table 50: IS 800-2007

Connection	Family	Member Types	Built-Up Symmetric	Built-Up Asymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	I, I2C, HSS_RECT, HSS_CIRC(Column)
Moment end plate	BS	Prismatic	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Prismatic (Column, Beam)	YES	YES	YES(Column)	I
	BCF	Haunched (Beam)	YES	YES	YES(Column)	I, T (Haunch)

Connection	Family	Member Types	Built-Up Symmetric	Built-Up Asymmetric	Member Flange Rotation	Sections Allowed
Cleat angle	BCF	Prismatic	YES	YES	YES(Column)	I (Beam)I, HSS_RECT (Column)
	BCW	Prismatic	YES	YES	NO	I (Beam)I, HSS_RECT (Column)
	BG	Prismatic	YES	YES	NO	I
Flange angles	BCF	Prismatic	YES	YES	YES(Column)	I
Seating angles	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
End plate	BCF	Prismatic	YES	YES	YES(Column)	I (Beam)I, HSS_RECT (Column)
	BCW	Prismatic	YES	YES	NO	I (Beam)I, HSS_RECT (Column)
	BG	Prismatic	YES	YES	NO	I
Web side plate	BCF	Prismatic	YES	YES	YES(Column)	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BCW	Prismatic	YES	YES	NO	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BG	Prismatic	YES	YES	NO	I
Flange cover plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Web cover plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I

Table 51: AS 4100-1998

Connection	Family	Member Types	Built-Up Symmetric	Built-Up Asymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	I, I2C, HSS_RECT, HSS_CIRC (Column)
Bolted end plate	BS	Prismatic	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Prismatic (Column, Beam)	YES	YES	YES (Column)	I
	BCF	Haunched (Beam)	YES	YES	YES (Column)	I, T (Haunch)
Mitred Knee	BCF	Prismatic	Yes	Yes	YES (Column)	I
Angle cleat	BCF	Prismatic	YES	YES	YES (Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Flange plate	BCF	Prismatic	YES	YES	YES (Column)	I
	BCW	Prismatic	YES	YES	NO	I
Seating connections	BCF	Prismatic	YES	YES	YES (Column)	I, HSS_RECT (Column)
	BCW	Prismatic	YES	YES	NO	I, HSS_RECT (Column)
End plate	BCF	Prismatic	YES	YES	YES (Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Web side plate	BCF	Prismatic	YES	YES	YES (Column)	I (Beam) I, HSS_RECT, HSS_CIRC (Column)

Connection	Family	Member Types	Built-Up Symmetric	Built-Up Asymmetric	Member Flange Rotation	Sections Allowed
	BCW	Prismatic	YES	YES	NO	I (Beam) I, HSS_RECT, HSS_CIRC (Column)
	BG	Prismatic	YES	YES	NO	I
Bolted flange plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Bolted web plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Bearing pad	BCF	Prismatic	YES	YES	YES (Top Column)	I (Beam) I, HSS_RECT, HSS_CIRC (Column)
	BCW	Prismatic	YES	YES	YES (Top Column)	I (Beam) I, HSS_RECT, HSS_CIRC (Column)
Directly welded splices	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Welded beam to column	BCF	Prismatic	Yes	Yes	YES (Column)	I
Column beam braces connections	CBB	Prismatic	YES	Column "YES" - Beams "YES" - Braces "NO"	Column "YES"	Column "I, HSS_RECT" - Beam "I" - Braces "I, I2C, L, T2L, HSS_RECT, HSS_CIRC"

Table 52: NZS 3404-1997

Connection	Family	Member Types	Built-up Symmetric	Built-up Asymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	I, I2C, HSS_RECT, HSS_CIRC(Column)
Bolted end plate	BS	Prismatic	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Prismatic (Column, Beam)	YES	YES	YES(Column)	I
	BCF	Haunched (Beam)	YES	YES	YES(Column)	I, T (Haunch)
Mitred Knee	BCF	Prismatic	Yes	Yes	YES(Column)	I
Angle cleat	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Flange plate	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
End plate	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Web side plate	BCF	Prismatic	YES	YES	YES(Column)	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BCW	Prismatic	YES	YES	NO	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BG	Prismatic	YES	YES	NO	I

Connection	Family	Member Types	Built-up Symmetric	Built-up Asymmetric	Member Flange Rotation	Sections Allowed
Bolted flange plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Bolted web plates	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Welded beam to column	BCF	Prismatic	Yes	Yes	YES(Column)	I

Table 53: CSA S16-14

Connection	Family	Member Types	Built-up Symmetric	Built-up Asymmetric	Member Flange Rotation	Sections Allowed
Base plate	CB	Prismatic	YES	YES	NO	I, I2C, HSS_RECT, HSS_CIRC(Column)
Bolted end plate	BS	Prismatic	YES	YES	NO	I
	BS	Haunched	YES	YES	NO	I, T
	BCF	Prismatic (Column, Beam)	YES	YES	YES(Column)	I
	BCF	Haunched (Beam)	YES	YES	YES(Column)	I, T (Haunch)
Mitred Knee	BCF	Prismatic	Yes	Yes	YES(Column)	I
Angle cleat	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Flange plate	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I

Connection	Family	Member Types	Built-up Symmetric	Built-up Asymmetric	Member Flange Rotation	Sections Allowed
End plate	BCF	Prismatic	YES	YES	YES(Column)	I
	BCW	Prismatic	YES	YES	NO	I
	BG	Prismatic	YES	YES	NO	I
Web side plate	BCF	Prismatic	YES	YES	YES(Column)	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BCW	Prismatic	YES	YES	NO	I (Beam)I, HSS_RECT, HSS_CIRC (Column)
	BG	Prismatic	YES	YES	NO	I
Flange plate splices	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Web cover plates splices	BS	Prismatic	YES	YES	NO	I
	CS	Prismatic	YES	YES	YES (Top Column)	I
Welded beam to column	BCF	Prismatic	Yes	Yes	YES(Column)	I

D. Using the RAM Connection mode

This section provides you with some common procedures used in design steel connections in the Connection Design mode.

D. To assign member type attributes

To assign the member type to consider for steel connection design in the Connection Design workflow, use the following procedure.

The member type attribute is *only* used for steel connection design in RAM Connection. It has no effect for members of other materials or other workflows.

Note: To assign a member as a truss type for analysis, you must M. To assign axial action members (on page 803). This process only affects the member type used for connection design.

1. Select one or members that will share the same attribute.

2. Either:

   on the **Beam Tools** ribbon tab, select the **Add Attribute** tool from the **Plugins** group

   or

   right-click and then select **Add Member Attribute** from the pop-up menu
   The **Member Attribute** dialog opens.
3. Select **MEMBTYPE** from the **Attribute Name** drop-down list.
4. Select the most appropriate member type from the **Attribute Value** drop-down list.
   - column
   - primary beam
   - brace
   - rafter
   - girt
   - purlin
   - eave-strut
   - secondary-beam
   - tertiary-beam
   - chord
   - branch
5. Click **Apply**.

Related Links
- *TR.29.3 Member Type Attribute* (on page 2343)
- *Member Attribute dialog* (on page 3057)

D. To edit the RAM Connection settings

1. Click the **RAM Settings** tool 
   The **RAM Connection Settings** dialog opens.
2. Select the default **Design Code** from the drop down list.
3. (Optional) For AISC codes, select the **Consider AISC 341-05 and AISC 358-05 Seismic Provision** option to include these seismic checks in design of connections by default for AISC codes.
4. (Optional) Select the **Design multiple selected connections individually** option to disable grouping of connections into same templates.
5. Check the country names to include various section catalogs in the **Sections Add to RAM** list.
6. Click **OK** to save the changes.

Related Links
- *D. RAM Connection Settings dialog* (on page 1043)
- *D. RAM Connection Settings dialog* (on page 1043)

D. To design steel connections

Before performing a connection design, you should select the appropriate design code and options in the **RAM Connection Settings** dialog.

1. Run an analysis.

   Tip: The Connection Design workflow is only available after the completion of a successful analysis.
2. Select **Connection Design** in the Workflows panel.

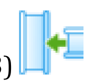
   

   The **RAM Connection Validation** dialog opens.
3. Click **Close**.
   The **RAM Connection Input** dialog opens.
4. Select the **Add Load Envelope** tool 
   The D. Load Envelope dialog (on page 1042) opens.
5. Select the loads you wish to use for design and then click **OK**.

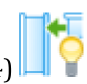
   This can be an existing load envelope or a selection of loads and load combinations.
6. Select the joint(s) you wish to add connections to.

   Note: Use the selection tools on the **Connection Design** ribbon tab in the **Assign Connections** group to assist selecting the intended joints.

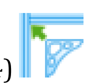
   Selected joints are highlighted with red dots.
7. In the **Connection Design** ribbon tab in the **Assign Connections** group, select the appropriate tool corresponding to the connection assignment you want to make:

   D. To select a basic connection template (on page 1033) 

   or

   D. To Select a Smart Connection Template (on page 1034) 

   or

   D. To Select a Gusset Connection Template (on page 1034) 
   In the corresponding Connection dialog, you can select which design code and the connection template you wish to use for the selected joint's design brief.
8. Click **OK** to assign the templates and design the connections.

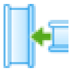
   The Connection Design dialog will display a list of all connection assignments.
9. Click **Close** to dismiss the dialog.
   A list of connections and basic information is displayed in the D. RAM Connection Input table (on page 1040).

D. To select a basic connection template

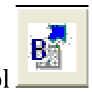
An appropriate set of joints must be selected in order to assign basic connections.

1. Either:

on the **Connection Design** ribbon tab, select the **Basic Connection** tool in the **Assign Connection** group



or

in the **RAM Connection Input** dialog, select the **Basic Connections** tool 

The **Basic Connection** dialog opens.

2. You may specify design code or grouping overrides here.
3. Select a connection class from the drop-down list.
   All available connections are displayed in the **Available** list box below.
4. Select one or more connections and add them to the **Selected** list box by clicking the **[>]** (add) button.

> **Tip:** You may add all connections in the list by clicking the **[>>]** (add all) button.

5. Click **OK** to accept and design the selected joints using these connections.

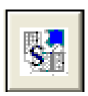
Related Links

- *Basic Connections dialog* (on page 3032)

D. To Select a Smart Connection Template

An appropriate set of joints must be selected in order to assign smart connections.

1. Either:

   Click the **Smart Connections** tool 

   or

   Select **Connection Design** > **Assign Smart Connection**

   The Smart Connections (on page 3035) opens.
2. You may specify design code or grouping overrides here.
3. (Optional) For AISC codes, select the option for **Consider AISC 341-05 Seismic Provisions**
   Select a load cases from the drop-down list which represents the gravity load.
4. (Optional) For BS5950 code, select the option to **Consider Structural Integrity** if necessary.
5. (Optional) For Base Plates, select a **Seismic Category** site class from the drop down list.
6. Select a connection class from the drop-down list.

   All available connections are displayed in the **Available** list box below.
7. Select the connection you wish to use and add it to the Selected list box by clicking the Add (**>**) button.
8. Click **OK** to accept and design the selected joints using these connections.

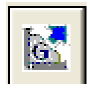
Related Links

- *Smart Connections* (on page 3035)

D. To Select a Gusset Connection Template

An appropriate set of joints must be selected in order to assign gusset connections.

1. Either:

Click the **Gusset Connections** tool 

or

Select **Connection Design** > **Assign Gusset Connection**

The Gusset Connections dialog (on page 3037) opens.

2. You may specify design code or grouping overrides here.
3. Select a connection class from the drop-down list.
   All available connections are displayed in the **Available** list box below.
4. Select the connection you wish to use and add it to the Selected list box by clicking the Add (>) button.
5. Click **OK** to accept and design the selected joints using these connections.

Related Links

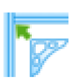
- *Gusset Connections dialog* (on page 3037)

D. To design an HBBB connection
To design a horizontal brace-beam-beam gusset connection, use the following procedure.

HBBB connections apply to joints with the following conditions:

- no column at the joint
- two perpendicular beams with H section and one brace with an angle section
- beams should be parallel with the ground (X-Z plane when Y is up)
- the brace should be in the same plane with the beams, with the angle between the brace and either beam between 30° and 60°

1. Start the **Connection Design** workflow and select the applicable load envelope.
2. Select a joint that matches the joint requirements.
3. On the **Connection Design** ribbon tab, select the **Gusset Connection** tool in the **Assign Connections** group.

   The **Gusset Connection** dialog opens.
4. Select an AISC design code from the **Design Code** drop-down list.
5. Select **Gusset Plate HBBB** from the connection template drop-down list.
6. Double-click the **HBBB_CA** template to add it to the selected list.
7. Click **OK**.
8. Double-click the connection list view to review the connection design.
   The connection pad opens.
9. (Optional) If the girder and beam have been incorrectly specified, they can be changed:
   a. On the **Connection Design** ribbon tab, select the **Beam and Girder Identification** tool in the **Frames** group.
      The **Beam-Girder Identification** dialog opens.
   b. Check the Swap option for the beam-girder connection you want to switch.
   c. Click **Switch**.
   d. Click **OK**.
   e. Repeat step 8 to open the connection design again.

D. To design an HCBB connection
To design a horizontal brace-column-beam gusset connection, use the following procedure.

HCBB connections apply to joints with the following conditions:

- at least one column with an H section at the joint
- column is vertical (perpendicular to beams)
- two perpendicular beams with H section and one brace with an angle section
- beams should be parallel with the ground (X-Z plane when Y is up)
- the brace should be in the same plane with the beams, with the angle between the brace and either beam between 30° and 60°

1. Start the **Connection Design** workflow and select the applicable load envelope.
2. Select a joint that matches the joint requirements.
3. On the **Connection Design** ribbon tab, select the **Gusset Connection** tool in the **Assign Connections** group.

   The **Gusset Connection** dialog opens.
4. Select an AISC design code from the **Design Code** drop-down list.
5. Select **Gusset Plate HCBB** from the connection template drop-down list.
6. Double-click the **HCBB_CA** template to add it to the selected list.
7. Click **OK**.
8. Double-click the connection list view to review the connection design.
   The connection pad opens.
9. (Optional) If the girder and beam have been incorrectly specified, they can be changed:
   a. On the **Connection Design** ribbon tab, select the **Beam and Girder Identification** tool in the **Frames** group.
      The **Beam-Girder Identification** dialog opens.
   b. Check the Swap option for the beam-girder connection you want to switch.
   c. Click **Switch**.
   d. Click **OK**.
   e. Repeat step 8 to open the connection design again.

D. To edit steel connections

1. Either:

   On the **Connection Design** ribbon tab, select the **Joint Cursor** tool in the **Assign Connections** group and then double-click on any connection in the View window.

   .

   or

   Select the entry for the connection row in the **RAM Connection Input** table and then click **Edit**

   The **Connection Pad** opens displaying the design information for this joint.

2. Make the desired changes to the connection input data.
3. Select the **Save** tool to save any changes .
   Changes made to the connection pad are saved back to the connection design in STAAD.Pro.
4. Click the **[X]** to close the Connection Pad window.

D. Selecting Joints & Connections

The Select Nodes tool is used to select nodes in the View window for assigning connection templates.

The **Select Joints** tool  (found in the Selection toolbar or from the **Select** menu) is used to select connections in the View window. Typical STAAD.Pro graphical selection rules apply.

The Select menu contains tools to select different connections and joints based on logical criteria:

- The **Select** > **Select Joints** sub-menu contains tools which logically select joints based on the connecting members.
- The **Select** > **Select Connections** sub-menu contains tools which aide in selecting joints for grouping.

D. Design Connections Individually

Any time multiple connections fitting the same template are selected and a template is applied, these will be grouped to the template meeting the minimum requirement of the entire group.

If you wish to design each connection to an optimized (and likely different) template, then select the **Design Connections Individually** option in the Connection assignment dialog before clicking **OK.**

This option can be selected in the D. RAM Connection Settings dialog (on page 1043) to disable the grouping feature.

Related Links

- *Basic Connections dialog* (on page 3032)
- *Gusset Connections dialog* (on page 3037)
- *Smart Connections* (on page 3035)

D. To delete steel connections

1. Select the connection(s) you wish to remove in the **RAM Connection Input** table.
2. Either:

   press **<Delete>**

   or

   click **Delete**
3. Click **Yes** to confirm the deletion.

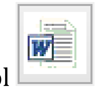
Tip: You may need to Refresh the display (press **<F5>**) to see the connection icon removed in the View window.

D. To Export Connection Designs to a Report

To export the connection design results to a Microsoft® Office Word® document report, use the following procedure.

You must perform connection design on one or more connections to generate a report.

1. Either:

   On the **RAM Connection Input** Table, select the **Export Connection Reports** tool 

   or

   Select **Connection Design** > **Export Connections report**.

   The **RAM Report Export** dialog opens.
2. Select the connections you want to include in the report.

   > **Tip:** Click **Select All** to quickly include all connection design into the report.
3. (Optional) Select or clear options to include **Data Report Result Report forumla**
4. Select to output multiple connections to either **Individual Reports** (separate file for each connection) or a **Merged Report** (single file containing all selected connections).
5. Click **OK**.
   The report is generated and saved in the same folder as the STAAD.Pro input file.

Related Links

- *RAM Report Export dialog* (on page 3041)

D. To add connection designs to your report

To add a summary of steel connection design results to your STAAD.Pro report, use the following procedure.

1. Select the **Postprocessing** workflow.
2. Either:

   select the **Reports** page

   or

   select the**File** ribbon tab and then select **Report** > **Setup** in the backstage view

   The **Report Setup** dialog opens.
3. Select the **Items** tab.
4. In the **Available** topics drop-down list, select **RAM Connection Summary Report**.
5. Double-click the **RCNX Report** entry to add it to the **Selected** list.
   The connection design summary is now included in the report.
6. Either:

   click **Print** to generate a copy of the report

   or

   click **OK** to dismiss the dialog and review the report on screen.

D. Custom Connection Templates

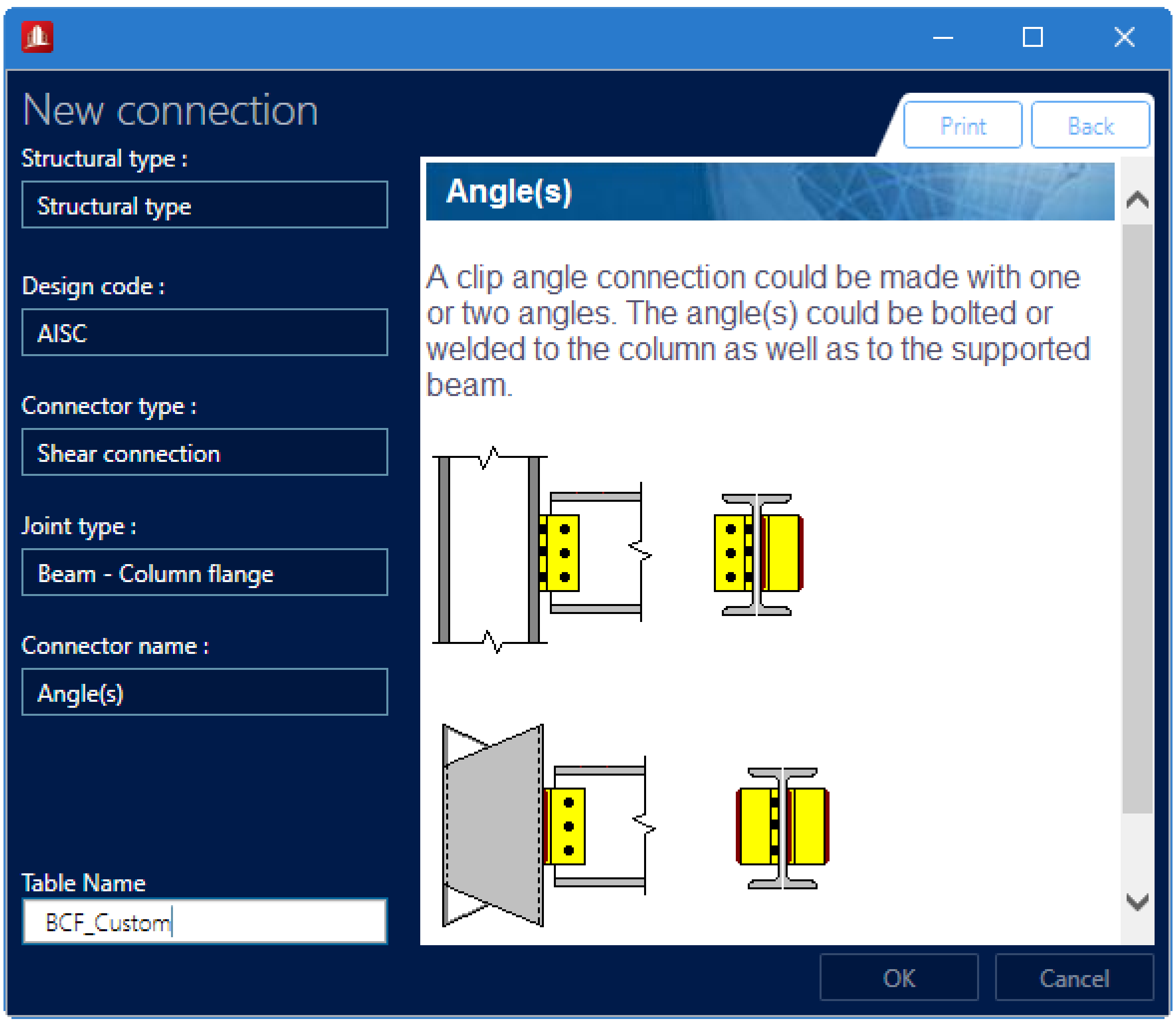
You can create and use your own connection templates.
D. To create a custom template file

Custom template files are created in the **Connection Design** workflow.

1. On the **Connection Design** ribbon tab, select the **Connection Database** tool in the **Configure** group.

The connection database dialog opens.

2. Click **New group**.
3. Type a New group name then click the X to close the dialog.
   The new group is created and selected as the current group.
4. Click **New table**.
   The new table dialog opens.
5. Specify the table parameters:
   a. Select the **Design code** to use for these connections.
   b. Select the **Connector type** from the drop-down list.
   c. Select the **Joint type** for the connections.
   d. Select the **Connector name** for the connection.
   e. The context-sensitive help panel provides additional information on the connector name selection.
   f. Click **OK**.



   The new table is saved to the custom group.
6. (Optional) Repeat Steps 4 and 5 to create additional tables as needed.

D. To add a connection template to a table
To add a custom connection template to a database table, use the following procedure.

> **Tip:** One of the simplest methods to get started is to copy factory installed templates to your new group and edit those to suite your needs.

1. On the **Connection Design** ribbon tab, select the **Connection Database** tool in the **Configure** group.
   The connection database dialog opens.
2. Select **Custom** in the **Groups** drop-down list.
3. Click **New Item**.
   The **Connection Pad** window opens.
4. Specify the connection parameters:
   a. Type the connection **Name**.
   b. Update the connection parameters for this connection type.
   c. Click **Save**.
   d. Close the window.

   The new connection name is added to the items list. A list of the connection tables where you can add this connection type are also listed.
5. Click **>>** to add the new connection to the connection database file.

The new connection type is available for use in design in this connection database.

> **Note:** When the connection database is selected in the connection database dialog, you can click **Reset** to remove any custom items and restore the database to the default values.

D. Pages in the Connection Design workflow

The page control bar in Connection Design workflow contains the following pages:

D. Connections page

Used to select, assign, design, and edit steel connections. The Connection page is where connections are defined and designed.

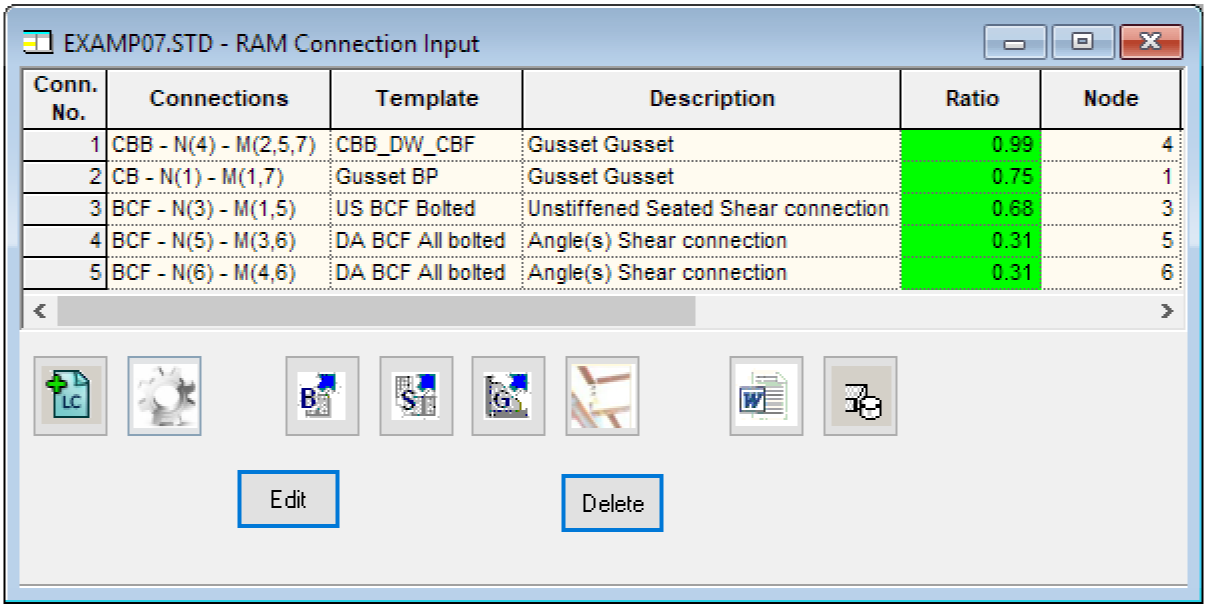
When the **Connections** page is selected, the **RAM Connection Input** table opens.

The **Whole Structure** view window displays the structure (or a portion thereof), which is used to graphically select joints for connection assignment and connections for editing.

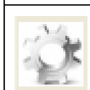
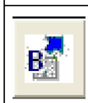
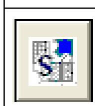
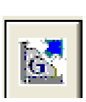
> **Tip:** When a table row(s) is selected in the **RAM Connection Input** table, the corresponding Joint(s) will be highlighted in the view window.

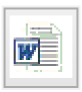
D. **RAM Connection Input** table
Contains tools to generate new connection designs and contains all previously designed connections.

Opens when the D. Connections page (on page 1040) is selected.



Tools

Icon	Description	Same Effect as Selecting
 **Load Envelope**	Opens the D. Load Envelope dialog (on page 1042), which is used to select load cases and combinations to include in the load envelope for steel connection design.	**Connection Design > Load Envelope For Connection**
 **RAM Connection Settings**	Opens the D. RAM Connection Settings dialog (on page 1043), which is used to set the default design code and grouping toggle.	**Connection Design > Assign RAM Connection Settings**
 **Assign Basic Connections**	Opens the Basic Connections dialog (on page 3032), which is used to assign basic connection templates to selected joints.	**Connection Design > Assign Basic Connections**
 **Assign Smart Connections**	Opens the Smart Connections (on page 3035), which is used to assign smart connection templates to selected joints.	**Connection Design > Assign Smart Connections**
 **Assign Gusset Connections**	Opens the Gusset Connections dialog (on page 3037), which is used to assign gusset connection templates to selected joints.	**Connection Design > Assign Gusset Connections**

Icon	Description	Same Effect as Selecting
 **Show RAM Database**	Opens the RAM Connection Material Database dialog (on page 3042), which is used to review additional material data required by RAM Connection for the design of steel connections.	**Connection Design** > **Assign RAM Materials**
 **Identify Beam-Girder**	Opens the Beam-Girder Identification dialog (on page 3040), which is used to switch beam and girder assignments in a Beam-Girder connection.	**Connection Design** > **Identify Beam and Girder**
 **Export Connection Reports**	Opens RAM Report Export dialog (on page 3041), which is used to manage connections and other report details to include in an report.	**Connection Design** > **Export Connections report**

Table

Once a Connection is established the following information will be displayed in the Input grid:

- Connection Name
- Template used for the Connection
- Description of the Connector
- Node (where the Joint is created)
- Members which participated in the connection

D. **Load Envelope** dialog
Used to specify the loads which will be included in the Load Envelope used for steel connection design.

Opens when the **Add Load Envelope** tool is selected in the **Connection Design** workflow.

Loads Select the source of connection design load envelopes:

- **Defined Envelopes -** Select this option to use a load envelope which has been previously defined for this model. A list of all defined envelopes is then available in the drop-down list.
- **Envelope of Load Cases in Selected List -** The option will be inactive if no load envelopes have been defined.

Loads Use the controls to populate the selected load list for a new load envelope.

Click this button...	to...
>	Add the selected load case to the Selected list
>>	Add all load cases to the Selected list.
<<	Remove all entries in the Selected list, load combination is placed back in the Available list.
<	Remove the selected entry from the Selected list.

The **Selected** list displays load cases that have been selected.

OK Click this button to accept changes made to the Load Envelope.

Cancel Click this button to close the dialog with no changes saved.

Apply Click to apply the changes without closing the dialog.

Help This button opens the online documentation.

D. **RAM Connection Settings** dialog
Used to set the default design code and grouping toggle.

Opens when the **RAM Settings** tool  is selected in the RAM Connection input dialog.

Design Code Specify the default design code to be used from the drop-down list. The following codes are currently available in Connection Design in STAAD.Pro:

AISC 360-10 (ASD) — Allowable Stress Design per AISC 360-10
AISC 360-10 (LRFD) — Load Resistance Factor Design per AISC 360-10
AISC 360-05 (ASD) — Allowable Stress Design per AISC 360-05
AISC 360-05 (LRFD) — Load Resistance Factor Design per AISC 360-05
AS 4100 1998
BS 5950-01:2000
CSA S16-14 — Limit States Design of Steel Structures (2014 editions)
EN 1993-1-7 — Eurocode 3
IS 800-2007
GB50017-2003 — "Design of Steel Structures" Chinese code
NZS 3404-1997

Consider AISC 341-05 and AISC 358-05 Seismic Provision (AISC codes only) Select this option to direct the program to consider seismic provisions per AISC 341-05 “Seismic Provisions for Structural Steel Buildings” and AISC 358-05 “Prequalified Connections for Special and Intermediate Steel Moment Frames for Seismic Applications ” in the design of connections of members in a designated seismic frame.

Design multiple selected Select this option to disable grouping of connections into same templates

connections individually	
Sections Add to RAM	Check the boxes associated with country section catalog to include in the RAM Connection catalog.

OK	Accepts the settings changes and closes the dialog.
Cancel	Closes the dialog without saving changes.
Help	Opens the Help window.

Related Links

- *D. To edit the RAM Connection settings* (on page 1032)
- *D. To edit the RAM Connection settings* (on page 1032)

D. Results page

Used to review steel connection designs. The **Results** page displays the summary, layout, and results of the connection designs.

View Window

When a row of a RAM Connection Result table is selected, the DXF drawing and the Report will be displayed in the View window.

RAM Connection Results table

Contains the following items for each connection (upon successful connection design):

- Joint Name
- Template
- Design Code
- Ratio (Critical Strength Ratio)
- Status — Possible values:

    OK (Critical Strength Value < 1),
    No Good (Critical Strength Value > 1), or
    Warning (This connection may require additional review)

D. Seismic Frames page

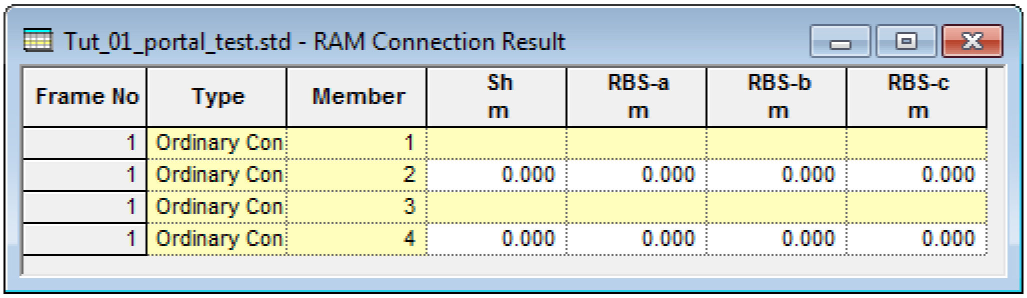
Used to assign and review seismic frames for connection design.

When the **Seismic Frame** page is selected in the **Connection** workflow, the **RAM Connection Result frames** table opens.

D. **RAM Connection Result** frames table
Used to review frame member assignments and specify reduced section (induced plastic hinge) parameters for beam members within a frame.

Note: A column member may be used in multiple frames but a beam member can only be used in a single frame definition.

Opens when the D. Seismic Frames page (on page 1044) is selected in the **Connection** workflow.



Frame No	Type	Member	Sh m	RBS-a m	RBS-b m	RBS-c m
1	Ordinary Con	1				
1	Ordinary Con	2	0.000	0.000	0.000	0.000
1	Ordinary Con	3				
1	Ordinary Con	4	0.000	0.000	0.000	0.000

The table rows are grouped first by **Frame No** and then by **Member** number. The **Type** of lateral seismic resisting is also displayed. These elements (shaded in yellow) may not be edited. The location and dimensions of reduced beam sections (RBS) for forming plastic hinges may be specified here for beam members.

Where:

Sh is the distance to the Hinge Location
RBS-a is the horizontal distance to locate the reduce beam section
RBS-b is the length of the reduced section of the flange of the beam
RBS-c is the depth of the reduced section of the flange

D. Application Window Layout

After entering the Connection Design Mode is entered, some of the menus change to offer some additional tools.

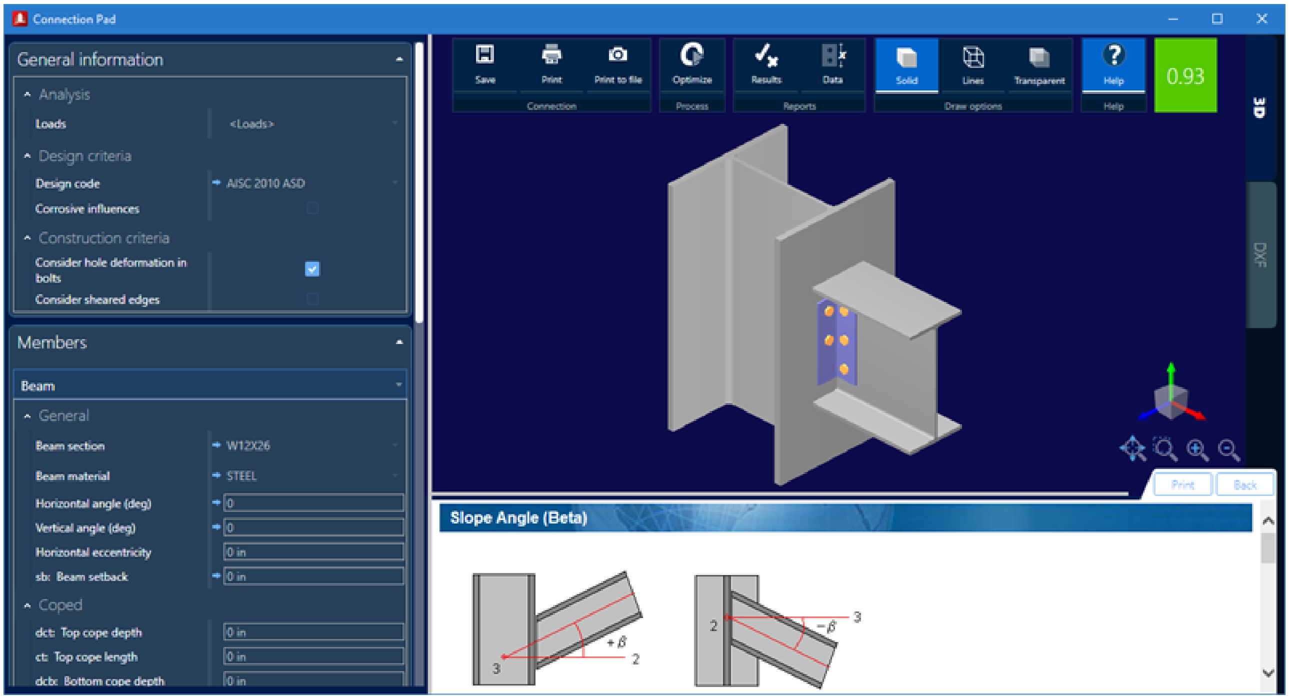
Note: For help with all other application window elements, please refer to the M. Creating Model Objects (on page 653) .

D. RAM Connection pad

Used to inspect connection details, edit a connections input, and access steel connection design results.

Opens when a connection icon is double clicked in the View window.

Note: The appearance and layout of the Connection pad varies depending on which version of **RAM Connection** is installed. The tools and operation remain essentially the same.

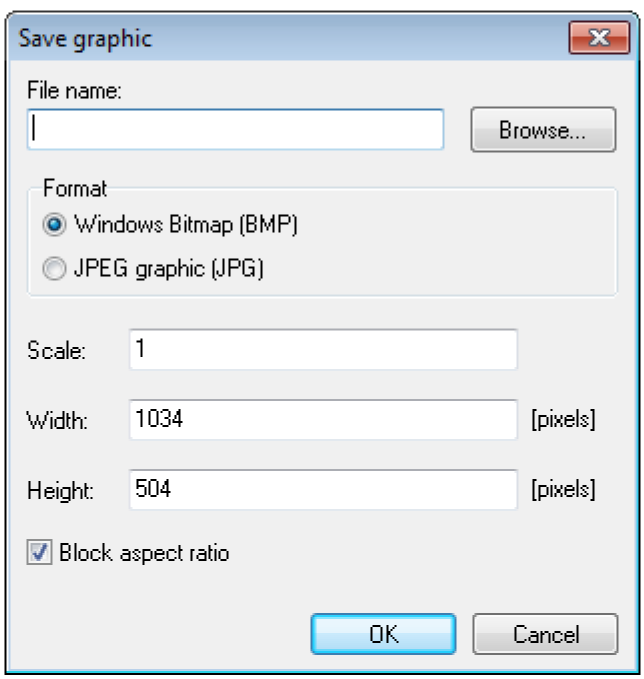


The connection pad is required for the creation of templates for the database and also to review/edit a model connection, or many model connections with the connections detailer. It is accessed when any template of the database is edited, when the user double clicks a model connection, or when several identical model connections are selected and the detailing command is invoked from the main menu.

D. **Save Graphic** dialog

Used to save a connection drawing to an external image file.

Opens when the **Print to File** tool is selected in the **Connection Pad Drawing** window.



File name Specify a name for the image file.

Browse Opens a Windows Save As dialog, which is used to specify a drive, directory, and file name for the image file.

Format Select one of the following common image file formats:

- Windows Bitmap (BMP) - An uncompressed image file format. These are very common and readable by most image editing tools. Many word processing or spreadsheet programs can import these image files for re-use. These files can be very large for even relatively short reports (approximately 20x the size of a JPEG).
- JPEG graphic (JPG) - A compressed image format commonly used for photographic images. These are also readable by a wide variety of programs, including web browsers, image editors, and office programs. JPEG is not recommended for line drawings or text due to the blurring that may result from its compression algorithm.

Scale Specify a scale factor to be used with the image file.

Width Specify a width (in pixels) for the image file.

Height Specify a height (in pixels) for the image file.

Block aspect ratio Selecting the is option will scale the height or width such that the aspect ratio of the original drawing is constant in the image file.

OK Create the image file with the selected options and close the Save Graphic dialog.

Cancel Close the dialog without saving an image file.

D. Connection Tags

Connection tags are used to associate connection data included in an XML file with members. This data can be exchanged with other programs via the ISM link. The connections can also be checked by defined capacities in the XML file.

Connection tags consist of two pieces of data:

i. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to D. Connection Tags XML File Schema (on page 1052) for additional information on the required structure of this XML file.

ii. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the `DEFINE MEMBER ATTRIBUTE` command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names in the associated XML file. Refer to TR.29.2 Connection Tag Member Attribute (on page 2341) for additional information on this command.

D. To create a connection tag

To create a new connection tag for use in the current STAAD model, use the following steps.

Note: You must have a Connection Tags XML file containing appropriate connection capacities with members used in the model. A default XML file is included with the program that contains a minimal amount of connection data.

A connection tag is added to the model to define the type of connection (e.g., shear or moment) and the associate attributes of that connection (e.g., end releases or capacities associated with the connection).

1. Select one or more members in the view window.
2. Either:

   on the **Beam Tools** ribbon tab, select the **Assign** tool in the **Connection Tags** group

   or

   right-click and select **Connection Tags** > **Assign Connection Tags** from the pop-up menu

   Note: If you have not specified a Connection Tags XML data file, then the Select Connection Tag File dialog opens prompting you to select an appropriate XML file.

   The **Assign Connection Tags** dialog and **New Connection Tag** dialog open.
3. If the **New Connection Tag** dialog did not already open, then click the **New** tool in the dialog toolbar 
4. On the **New Connection Tag** dialog, select the category to use from the **Select Categories** drop-down list.
5. Select the Tag to use from the selected category from the **Select Tags** drop-down list.
6. Either:

   set the option to **Assign Beam End Releases** to the use the end releases in the connection tag definition

   or

   clear the **Assign Beam End Release** option and manually select the end releases you want for this connection
7. Select either the **Start** or **End** Location option (or both).

   Note: Selecting both Start and End locations creates two distinct connections, which must both be assigned.
8. Either:

   click **Add** to add this connection to the connection tags list in the Assign Connection Tags dialog

   or

   click **Assign** to assign the connection tag to the current member selection

Related Links

- *Assign Connection Tags dialog* (on page 3053)
- *New Connection Tag dialog* (on page 3055)

D. To delete a connection tag

To remove a connection tag entirely from the STAAD model, use the following steps.

This procedure will remove the connection from the model. To remove the assignment of a connection tag from one or more members, see D. To remove connection tag assignments (on page 1049).

1. Select one or more members in the view window.
2. Either:

   on the **Beam Tools** ribbon tab, select the **View** tool in the **Connection Tags** group

   or

   on the **Utilities** ribbon tab, select the **Connection Tags** > **View Tags** tool in the **Tools** group

   or

   right-click and select **Connection Tags** > **View Connection Tags** from the pop-up menu.

   The **Assign Connection Tags** dialog opens.
3. Select the row in the Connection Tags table for the connection tag you want to remove.
4. Select the **Remove** tool in the dialog toolbar. 
   A confirmation dialog opens to confirm you want to delete the connection tag.
5. Click **OK**.
   A confirmation dialog opens in the STAAD.Pro user interface window to confirm you want remove the associated STAAD input command for the member attribute definition.
6. Click **Yes**.
   The row is removed from the Connection Tags table.

D. To remove connection tag assignments

To remove the connection tag assignments from members, use the following steps.

This procedure will disassociate a connection tag with a member end. If you want to complete remove a connection tag from the STAAD.Pro model, see D. To delete a connection tag (on page 1048).

1. Select one or more members in the view window.
2. Either:

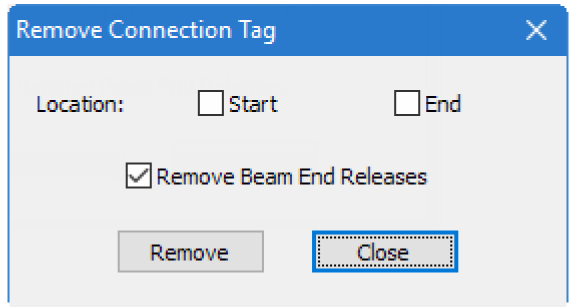
   on the **Beam Tools** ribbon tab, select the **Remove** tool in the **Connection Tags** group

   or

   right-click and select **Connection Tags** > **Remove Connection Tags** from the pop-up menu

   The **Assign Connection Tags** and the **Remove Connection Tag** dialogs open.



3. Select either the **Start** or **End** (or both) corresponding to the member end from which you want the connection tag removed.
4. (Optional) If you do not want the beam end releases for this beam end changed, then clear the option **Remove Beam End Releases**.

> **Tip:** By default, this option is set so the member end releases for this end are returned to their defaults (i.e., fully fixed).

5. Click **Remove**.
   A confirmation dialog opens.
6. Click **OK**.
   A confirmation dialog opens in the STAAD.Pro user interface window to confirm you want remove the associated STAAD input command for the member attribute definition.
7. Click **Yes**.

   The member number is removed from the Assigned Beams cell of the Connection Tags table.

> **Note:** If the tag selected was only assigned to a single member end, then the row is removed from the Connection Tags table.

Related Links

- *Remove Connection Tags dialog* (on page 3057)

D. To check connection tags

To check the assigned connection tags capacity using the STAAD.Pro analysis results, use the following steps.

Prior to checking connections, you must perform a successful analysis on the model so analysis results at member ends can be compared to connection capacities.

Additionally, each beam and connecting member (beam or column) must have a connection capacity for the sections used in the associated Connection Tags XML file to facilitate checking connections.

1. Select one or more members in the view window.
2. Either:

   on the **Beam Tools** ribbon tab, select the **Check** tool in the **Connection Tags** group

   or

on the Utilities ribbon tab, select the **Connection Tags** > **Check Tags** in the Tools group

or

right-click and select **Connection Tags** > **Check Connection Tags** from the pop-up menu

The **Assign Connection Tags** dialog and **Check Connection Tags** dialog open.

3. Select the load case types you want to use in the **Select Load Case Types** drop-down list.
   The load cases list displays only the type or types selected.
4. Either:

   select load cases individually by setting their check boxes on in the list

   or

   set the **Select All** option check box on.
5. Click **Check**.
   The **Connection Tag Check Results** table opens. This table displays the connection tag data from the Connection Tag XML file capacity used checked against the critical load case at each member end with a connection tag. The status of the check (i.e., Pass or Fail) is listed in the Remarks column.

Related Links

- *D. Connection Tags Capacity Checks* (on page 1051)
- *Check Connection Tags dialog* (on page 2952)

D. Connection Tags Capacity Checks

The rules for checking the connections are dependent on the whether the connection is typed as moment or shear.

Moment Connections

Shear Capacity Check **-** The shear capacity for a given beam/column combination present in the `ConnectionTagFile.xml` is checked against the demand taken as the absolute value of the major axis shear value, Fy, from each of the load cases included and the maximum value reported.

If the shear capacity in the XML file is 207.345 kN, then:

- when Fy = ±208.0 kN > 207.345 kN, the connection check is failed.
- when Fy = ±200.0 kN < 207.345 kN, the connection check is passed.

Moment Capacity Check **-** The moment capacity check for a given beam/column combination is based on both the major axis moment and axial force. For each load case, the maximum demand is taken from the end moment plus the axial force multiplied by a specified alpha factor and compared against the given moment capacity.

If the moment capacity in the XML file is 35.951 kN·m and the tension force factor, alpha, is 0.09565, then demand = $|Mz| + a \cdot |Fx|$:

- when Mz = 30 kN·m and Fx = -50 kN, then demand is 30 + 0.09565(50) = 34.7825 kN·m < 35.951 kN·mm, the connection check is passed.
- when Mz = 32 kN·m and Fx = -50 kN, then demand is 32 + 0.09565(50) = 36.7825 kN·m > 35.951 kN·mm, the connection check is failed.

Shear Connections

The shear capacity for a given beam/column combination present in the `ConnectionTagFile.xml` is checked against the demand taken as the major axis shear value, Fy, and a proportion of the axial force, Fx, from each of the load cases included and the maximum value reported.

If the shear capacity in the XML file is 15.538 kN and the tension force factor, alpha, is 0.2016, then demand = $|Fy| + a \cdot |Fx|$:

- when Fy = 12 kN and Fx = -25 kN, then demand = 12 + 0.2016(25) = 17.04 kN > 15.538 kN, the connection is failed.
- when Fy = 10 kN and Fx = 25 kN, then demand = 10 + 0.2016(25) = 15.04 kN < 15.538 kN, the connection is passed.

Note: There are no moment capacity checks for a shear connection.

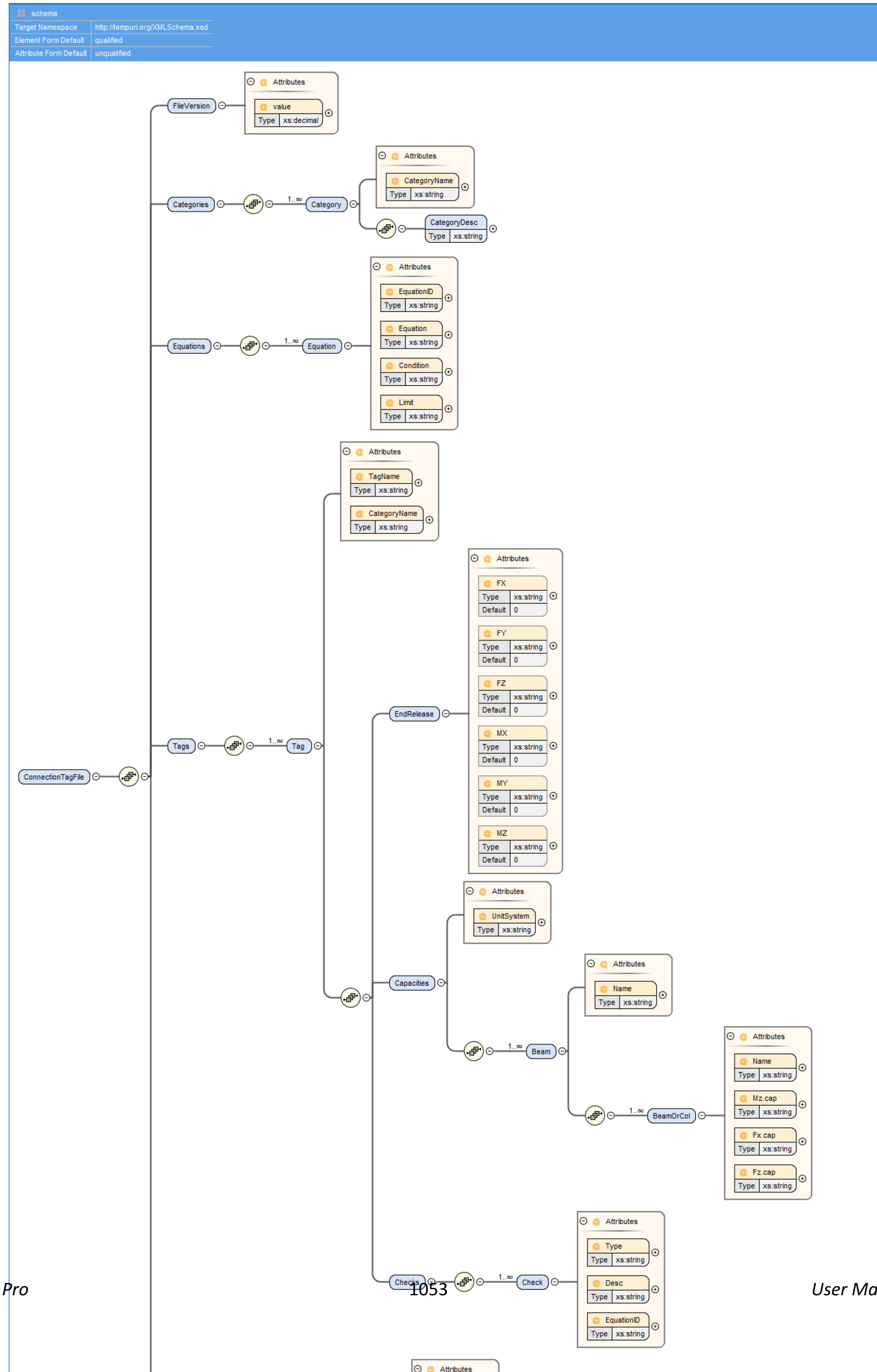
Related Links

- *D. To check connection tags* (on page 1050)
- *Check Connection Tags dialog* (on page 2952)

D. Connection Tags XML File Schema

A simple schema is used to define the data for a Connection Tags XML file used for storing connection tag data for STAAD.Pro.

When you initiate a connection tags action for the first time from the right-click pop-up menu, you will be prompted to select a Connection Tags XML file for use with creating and checking connection tags.


schema
Target Namespace http://tempuri.org/XMLSchema.xsd
Element Form Default qualified
Attribute Form Default unqualified
ConnectionTagFile
FileVersion
Attributes
value
Type xs:decimal
Categories
Category
CategoryName
Type xs:string
CategoryDesc
Equations
Equation
EquationID
Condition
Limit
Tags
Tag
TagName
EndRelease
FX
FY
FZ
MX
MY
MZ
Default 0
Capacities
UnitSystem
Beam
Name
BeamOrCol
Mz.cap
Fx.cap
Fz.cap
Checks
Check
Type
Desc
1..∞

1. GUID-4AA84EEA-C19A-4440-AC79-E4953BBDAEF1-LOW.PNG

Figure 125: A diagram representation of the XML schema used

ConnectionTagFile

The `<ConnectionTagFile>` element is the root element for a STAAD.Pro Connection Tags file. Within a ConnectionTagFile element two types of data: connection categories and connection tags.

Table 54: ConnectionTagFile model details

Contains	(File Version then Categories then Equations then Tags then RCNXTemplates)
Contained by	n/a (*root element*)
Attributes	

FileVersion

The `<FileVersion>` element is indicate the file version used.

Table 55: FileValue model details

Contains	n/a (*empty element*)
Contained by	ConnectionTagFile
Attributes	value (required, decimal)

Categories

The `<Categories>` element contains individual connection categories. Typically a connection tags file contains both Moment and Shear categories.

Table 56: Categories model details

Contains	(Category) (*one or more*)
Contained by	ConnectionTagFile
Attributes	none

Category

The `<Category>` element defines a connection category.

Table 57: Category model details

Contains	(CategoryDesc)

Contained by	Categories
Attributes	CategoryName (required, restricted string data, no default); values are: MOMENT SHEAR

CategoryDesc

The `<CategoryDesc>` element is used to provide a human readable description for the parent connection category. It can contain any alpha-numeric string.

Table 58: CategoryDesc model details

Contains	text data
Contained by	Category
Attributes	none

Equations

The `<Equations>` element contains individual equations.

Table 59: Categories model details

Contains	(Equation) (*one or more*)
Contained by	ConnectionTagFile
Attributes	none

Equation

The `<Equation>` element contains a user-defined equation.

Table 60: Capacity model details

Contains	n/a (*empty element*)
Contained by	Equations

Attributes	EquationID (required, string), Equation (required, string), Condition (required, string), Limit(required, decimal) See " Equations Guidelines (on page 1059) " for details on writing equation and condition values.

Tags

The `<Tags>` element contains the connection tag data in a series of child elements.

Table 61: Tags model details

Contains	(Tag) (*one or more*)
Contained by	ConnectionTagFile
Attributes	

Tag

The `<Tag>` element contains an end release definition and a set of connection capacities to be used for checking connection tags.

Table 62: Tag model details

Contains	(EndRelease then Capacities)
Contained by	Tags
Attributes	TagName (required, string, no default), CategoryName (required, restricted string data, no default); values are: MOMENT SHEAR

EndRelease

The `<Tag>` element is used to specify the end release conditions to be used with a connection tag.

Table 63: EndRelease model details

Contains	n/a (*empty element*)
Contained by	Tag

Attributes	FX (required, restricted integer), FY (required, restricted integer), FZ (required, restricted integer), MX (required, restricted integer), MY (required, restricted integer), MZ (required, restricted integer) The range for these attributes is: 0 - restrained 1 - full release

Capacities

The `<Capacities>` element contains one or more connection capacity value sets for a specific system of units.

Table 64: Capacities model details

Contains	(Capacity) (*one or more*)
Contained by	Tag
Attributes	UnitSystem(required, restricted string data, no default); values are: METRIC IMPERIAL

Beam

The `<Beam>` element contains one or more beam or column sections.

Table 65: Beam model details

Contains	BeamOrCol (*one or more*)
Contained by	Capacities
Attributes	Name (required, string)

BeamOrCol

The `<BeamOrCol>` element is used to specify connection capacity bending and shear capacities for a specific member, or for the default member.

Table 66: BeamOrCol model details

Contains	n/a (*empty element*)
Contained by	Beam
Attributes	Name (required, string), Mz.cap (required, decimal), Fx.cap (required, decimal), Fz.cap (required, decimal) alpha (required, decimal) See D. Connection Tags Capacity Checks (on page 1051) for details on how these values are used. See " Use of Wild Cards (on page 1059) " for details on using defaults.

Checks

The `<Checks>` element contains one or more check.

Table 67: Checks model details

Contains	(Check) (*one or more*)
Contained by	Tag
Attributes	none

Check

The `<Check>` element is used to specify connection capacity between specified members required for checking connections.

Table 68: Check model details

Contains	n/a (*empty element*)
Contained by	Checks
Attributes	Type (required, string), Desc (required, string), EquationID (required, string) - reference to equation IDs.

RCNXTemplates

The `RCNXTemplates` element contains one or more RAM Connection template.

Table 69: RCNXTemplates model details

Contains	(RCNXTemplate) (*one or more*)
Contained by	Tag
Attributes	none

RCNXTemplate

The `RCNXTemplate` element is used to map a connection tag to a RAM Connection template.

Table 70: RCNXTemplate model details

Contains	n/a (*empty element*)
Contained by	RCNXTemplates
Attributes	TagName (required, string), Code (required, string), Template (required, string) - the RAM Connection template name, File (required, string) - the RAM Connection template file name

Use of Wild Cards

Instead of specifying the capacities and other details for each individual beam or column section, “wild card” entries can be used. This is particularly useful when the remaining attributes are the same for multiple sections.

Example using the `Default` wild card:

```
<Beam  Name="UB203x102x23">
      <BeamOrCol Name="Default" Mz.cap="35.9156" Fx.cap="207.3451" Fz.cap="" />
</Beam>
```

Equations Guidelines

The following table contains the expressions that can be evaluated for user-defined equations:

Digits/ Characters	0	1	2	3	4	5	6	7	8	9	.	,

Binary Operators	+	-	*	/								
Unary Operators	+	-										
Parenthesis	(	)										
Symbols	PI											
Functions	ABS	POW	ROUND	SQRT								
	MAX	MIN	SIN	COS	TAN	ASIN	ACOS	ATAN	LN	LOG	EXP	
Reserved Keywords (variables)	[MZ.CAP]	[FX.CAP]	[FZ.CAP]	[FX]	[FY]	[FZ]	[MX]	[MY]	[MZ]			

Notes:

- "," (comma) can only be used with the POW function
- The ROUND function rounds off to the nearest integer, not to a certain decimal place
- Condition attributes should include any one of the following:
  - EQ - equal to
  - LT - less than
  - LE - less than or equal to
  - GT - greater than
  - GE - greater than or equal to
- Reserved keywords should be specified with square brackets
- Reserved keywords, symbols, functions, and conditions are *not* case sensitive.

Example of an equation:

```
<Equation EquationID="Eq1" Equation="Abs([MZ])/[MZ.CAP]+Abs([FX])/[FX.CAP]"
Condition="LT" Limit="1.0" />
```

Related Links

- *D. Sample Connection Tags XML File* (on page 1061)

D. Sample Connection Tags XML File

This example file is installed in `%LocalAppData%\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default\Plugins\ConnectionTagLink`. You can copy this file and use it as a template. Copies can be saved in any location and loaded through the user interface.

```
<?xml version="1.0" encoding="utf-8" ?>
  <ConnectionTagFile>
   <FileVersion value="1.0" />
   <Categories>
    <Category CategoryName="MOMENT">
     <CategoryDesc>End Moment Connection</CategoryDesc>
    </Category>
   <Category CategoryName="SHEAR">
     <CategoryDesc>Single Shear Connection</CategoryDesc>
   </Category>
  </Categories>
  <Equations>
   <Equation EquationID="Eq1" Equation="(abs([Mz])+([alpha]*abs([Fx])))/[Mz.cap]"
Condition="LT" Limit="1.0" />
   <Equation EquationID="Eq2" Equation="abs([Fy])/[Fy.cap]" Condition="LT"
Limit="1.0" />
   <Equation EquationID="Eq3" Equation="(abs([Fy])+([alpha]*abs([Fx])))/[Fy.cap]"
Condition="LT" Limit="1.0" />
  </Equations>
  <Tags>
   <Tag TagName="EM" CategoryName="MOMENT">
    <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
    <Capacities UnitSystem="METRIC">
     <Beam Name="UB203x102x23">
      <BeamOrCol Name="UC152x152x23" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC152x152x30" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC152x152x37" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC203x203x46" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC203x203x60" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC254x254x73" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC254x254x89" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC305x305x97" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC305x305x118" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC305x305x137" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x368x153" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x368x177" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x368x202" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x368x235" Mz.cap="35.9156214893831"
```

```
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x406x287" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
      <BeamOrCol Name="UC356x406x340" Mz.cap="35.9156214893831"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.09695" />
    </Beam>
    <Beam Name="UB203x133x25">
      <BeamOrCol Name="UC152x152x23" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC152x152x30" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC152x152x37" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC203x203x46" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC203x203x60" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC254x254x73" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC254x254x89" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC305x305x97" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC305x305x118" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC305x305x137" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x368x153" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x368x177" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x368x202" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x368x235" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x406x287" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
      <BeamOrCol Name="UC356x406x340" Mz.cap="42.5258304852143"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.0977"/>
    </Beam>
   </Capacities>
   <Checks>
    <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
    <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
   </Checks>
  </Tag>
  <Tag TagName="EMH" CategoryName="MOMENT">
   <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
   <Capacities UnitSystem="METRIC">
    <Beam Name="UB203x102x23">
     <BeamOrCol Name="UC152x152x23" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
     <BeamOrCol Name="UC152x152x30" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
     <BeamOrCol Name="UC152x152x37" Mz.cap="61.8354487089892"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
     <BeamOrCol Name="UC203x203x46" Mz.cap="62.3614827257097"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
```

```
      <BeamOrCol Name="UC203x203x60" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC254x254x73" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC254x254x89" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC305x305x97" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC305x305x118" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC305x305x137" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x368x153" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x368x177" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x368x202" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x368x235" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x406x287" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
      <BeamOrCol Name="UC356x406x340" Mz.cap="80.1535632590318"
Fy.cap="207.345115136926" Fx.cap="" alpha="0.19695" />
     </Beam>
    </Capacities>
    <Checks>
     <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
     <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
    </Checks>
   </Tag>
   <Tag TagName="SS" CategoryName="SHEAR">
    <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="1" MZ="1" />
    <Capacities UnitSystem="METRIC">
     <Beam  Name="UB203x102x23">
      <BeamOrCol Name="Column" Mz.cap="" Fy.cap="77.0472" Fx.cap="" alpha="1" />
      <BeamOrCol Name="UB203x102x23" Mz.cap="" Fy.cap="53.3250236369366" Fx.cap=""
alpha="0.692108520970738" />
      <BeamOrCol Name="UB203x133x25" Mz.cap="" Fy.cap="42.947706898342" Fx.cap=""
alpha="0.557420735579515" />
      <BeamOrCol Name="UB254x146x31" Mz.cap="" Fy.cap="77.0472" Fx.cap="" alpha="1" />
      <BeamOrCol Name="UB254x146x43" Mz.cap="" Fy.cap="77.0472" Fx.cap="" alpha="1" />
      <BeamOrCol Name="UB305x165x40" Mz.cap="" Fy.cap="75.7216934250438" Fx.cap=""
alpha="0.982796174618205" />
      <BeamOrCol Name="UB305x165x46" Mz.cap="" Fy.cap="75.4377877721379" Fx.cap=""
alpha="0.979111346968324" />
      <BeamOrCol Name="UB356x171x51" Mz.cap="" Fy.cap="71.3487080904668" Fx.cap=""
alpha="0.9260389487284" />
      <BeamOrCol Name="UB356x171x57" Mz.cap="" Fy.cap="71.0965925141754" Fx.cap=""
alpha="0.922766726294731" />
      <BeamOrCol Name="UB406x178x60" Mz.cap="" Fy.cap="71.1684435424868" Fx.cap=""
alpha="0.923699284886236" />
      <BeamOrCol Name="UB406x178x67" Mz.cap="" Fy.cap="70.8462523996888" Fx.cap=""
alpha="0.919517547681017" />
      <BeamOrCol Name="UB457x191x74" Mz.cap="" Fy.cap="64.3115752377084" Fx.cap=""
alpha="0.834703600360667" />
      <BeamOrCol Name="UB457x191x82" Mz.cap="" Fy.cap="64.0483627878951" Fx.cap=""
alpha="0.831287350973107" />
```

```
      <BeamOrCol Name="UB533x210x92" Mz.cap="" Fy.cap="61.209191556098" Fx.cap=""
alpha="0.794437585740922" />
      <BeamOrCol Name="UB610x229x101" Mz.cap="" Fy.cap="56.2244909263798" Fx.cap=""
alpha="0.729740872171601" />
      <BeamOrCol Name="UB610x229x113" Mz.cap="" Fy.cap="56.0901432778102" Fx.cap=""
alpha="0.72799716638385б" />
      <BeamOrCol Name="UB610x229x125" Mz.cap="" Fy.cap="55.9120086425492" Fx.cap=""
alpha="0.725685146800263" />
      <BeamOrCol Name="UB610x229x140" Mz.cap="" Fy.cap="55.6469181235012" Fx.cap=""
alpha="0.722244521845066" />
      <BeamOrCol Name="UB610x305x149" Mz.cap="" Fy.cap="42.4479655728093" Fx.cap=""
alpha="0.550934564433352" />
      <BeamOrCol Name="UB610x305x179" Mz.cap="" Fy.cap="42.1557467137328" Fx.cap=""
alpha="0.547141839206783" />
      <BeamOrCol Name="UB610x305x238" Mz.cap="" Fy.cap="41.620079719439" Fx.cap=""
alpha="0.540189386758233" />
      <BeamOrCol Name="UB686x254x152" Mz.cap="" Fy.cap="49.7324681393296" Fx.cap=""
alpha="0.645480538414499" />
      <BeamOrCol Name="UB762x267x173" Mz.cap="" Fy.cap="47.8567277507922" Fx.cap=""
alpha="0.621135197006409" />
      <BeamOrCol Name="UB838x292x194" Mz.cap="" Fy.cap="44.7544803846779" Fx.cap=""
alpha="0.580870951633258" />
      <BeamOrCol Name="UB914x305x201" Mz.cap="" Fy.cap="43.3227775363216" Fx.cap=""
alpha="0.562288798766491" />
      <BeamOrCol Name="UC203x203x46" Mz.cap="" Fy.cap="29.4320332925698" Fx.cap=""
alpha="0.382000037542828" />
      <BeamOrCol Name="UC203x203x60" Mz.cap="" Fy.cap="29.1497731650909" Fx.cap=""
alpha="0.378336567261249" />
      <BeamOrCol Name="UC254x254x73" Mz.cap="" Fy.cap="50.553607239979" Fx.cap=""
alpha="0.656138149601529" />
      <BeamOrCol Name="UC254x254x89" Mz.cap="" Fy.cap="50.247003128998" Fx.cap=""
alpha="0.652158717370625" />
      <BeamOrCol Name="UC305x305x97" Mz.cap="" Fy.cap="42.6924370326103" Fx.cap=""
alpha="0.554107573443426" />
      <BeamOrCol Name="UC305x305x118" Mz.cap="" Fy.cap="42.4223945092113" Fx.cap=""
alpha="0.550602676141525" />
      <BeamOrCol Name="UC305x305x137" Mz.cap="" Fy.cap="42.1936338437931" Fx.cap=""
alpha="0.54763357842716" />
     </Beam>
    </Capacities>
    <Checks>
      <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq3" />
    </Checks>
   </Tag>
   <Tag TagName="DS" CategoryName="SHEAR">
    <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="1" MZ="1" />
    <Capacities UnitSystem="METRIC">
     <Beam  Name="UB203x102x23">
      <BeamOrCol Name="Column" Mz.cap="" Fy.cap="15.5367929094832" Fx.cap=""
alpha="0.201652920670487" />
      <BeamOrCol Name="UB203x102x23" Mz.cap="" Fy.cap="17.213427830897" Fx.cap=""
alpha="0.223414060873036" />
      <BeamOrCol Name="UB203x133x25" Mz.cap="" Fy.cap="17.1973773492168" Fx.cap=""
alpha="0.223205740756533" />
      <BeamOrCol Name="UB254x146x31" Mz.cap="" Fy.cap="17.181355989296" Fx.cap=""
alpha="0.222997798613006" />
      <BeamOrCol Name="UB254x146x43" Mz.cap="" Fy.cap="17.1175602321941" Fx.cap=""
alpha="0.222169789845629" />
```

```
      <BeamOrCol Name="UB305x165x40" Mz.cap="" Fy.cap="17.181355989296" Fx.cap=""
alpha="0.22299798613006" />
      <BeamOrCol Name="UB305x165x46" Mz.cap="" Fy.cap="17.1440856356919" Fx.cap=""
alpha="0.222514064569406" />
      <BeamOrCol Name="UB356x171x51" Mz.cap="" Fy.cap="17.1069724435092" Fx.cap=""
alpha="0.22203237033285" />
      <BeamOrCol Name="UB356x171x57" Mz.cap="" Fy.cap="17.0700154484705" Fx.cap=""
alpha="0.22155270338793" />
      <BeamOrCol Name="UB406x178x60" Mz.cap="" Fy.cap="17.0805587073497" Fx.cap=""
alpha="0.221689544945821" />
      <BeamOrCol Name="UB406x178x67" Mz.cap="" Fy.cap="17.0332136939237" Fx.cap=""
alpha="0.2210750513182" />
      <BeamOrCol Name="UB457x191x74" Mz.cap="" Fy.cap="17.0227272874638" Fx.cap=""
alpha="0.220938947651099" />
      <BeamOrCol Name="UB457x191x82" Mz.cap="" Fy.cap="16.975693734259" Fx.cap=""
alpha="0.220328496483441" />
      <BeamOrCol Name="UB533x210x92" Mz.cap="" Fy.cap="16.4598373205167" Fx.cap=""
alpha="0.213633166688948" />
      <BeamOrCol Name="UB610x229x101" Mz.cap="" Fy.cap="15.5149040096276" Fx.cap=""
alpha="0.201368823391733" />
      <BeamOrCol Name="UB610x229x113" Mz.cap="" Fy.cap="15.4887168929929" Fx.cap=""
alpha="0.201028939312434" />
      <BeamOrCol Name="UB610x229x125" Mz.cap="" Fy.cap="15.453935064702" Fx.cap=""
alpha="0.200577503980703" />
      <BeamOrCol Name="UB610x229x140" Mz.cap="" Fy.cap="15.4020481619935" Fx.cap=""
alpha="0.199904060913227" />
      <BeamOrCol Name="UB610x305x149" Mz.cap="" Fy.cap="12.6138802017488" Fx.cap=""
alpha="0.163716270049383" />
      <BeamOrCol Name="UB610x305x179" Mz.cap="" Fy.cap="12.5472910390491" Fx.cap=""
alpha="0.162852005511545" />
      <BeamOrCol Name="UB610x305x238" Mz.cap="" Fy.cap="12.4246407638615" Fx.cap=""
alpha="0.1612601205996" />
      <BeamOrCol Name="UB686x254x152" Mz.cap="" Fy.cap="14.2038160959675" Fx.cap=""
alpha="0.184352138636674" />
      <BeamOrCol Name="UB762x267x173" Mz.cap="" Fy.cap="13.806921607951" Fx.cap=""
alpha="0.179200822456248" />
      <BeamOrCol Name="UB838x292x194" Mz.cap="" Fy.cap="13.1316837120063" Fx.cap=""
alpha="0.17043687132052" />
      <BeamOrCol Name="UB914x305x201" Mz.cap="" Fy.cap="12.8118907361867" Fx.cap=""
alpha="0.166286260061192" />
      <BeamOrCol Name="UC203x203x46" Mz.cap="" Fy.cap="16.6033632740696" Fx.cap=""
alpha="0.215495998220177" />
      <BeamOrCol Name="UC203x203x60" Mz.cap="" Fy.cap="16.494259319733" Fx.cap=""
alpha="0.214079931778611" />
      <BeamOrCol Name="UC254x254x73" Mz.cap="" Fy.cap="14.3749412117765" Fx.cap=""
alpha="0.186573181267801" />
      <BeamOrCol Name="UC254x254x89" Mz.cap="" Fy.cap="14.3112296483847" Fx.cap=""
alpha="0.185746265255385" />
      <BeamOrCol Name="UC305x305x97" Mz.cap="" Fy.cap="12.6694166452628" Fx.cap=""
alpha="0.164437080714975" />
      <BeamOrCol Name="UC305x305x118" Mz.cap="" Fy.cap="12.6080621808817" Fx.cap=""
alpha="0.163640757624959" />
      <BeamOrCol Name="UC305x305x137" Mz.cap="" Fy.cap="12.5559372281328" Fx.cap=""
alpha="0.16296422489244" />
    </Beam>
   </Capacities>
   <Checks>
     <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq3" />
```

```
   </Checks>
   </Tag>
  </Tags>
  <RCNXTemplates>
    <RCNXTemplate TagName="EM" Code="EN 1993-1-8" File="Flanges welded BCF.con"
Template="Flanges_Welded_BCF" />
    <RCNXTemplate TagName="EMH" Code="EN 1993-1-8" File="Bolted end plate BCF.con"
Template="BEP EN BCF Flush" />
    <RCNXTemplate TagName="SS" Code="EN 1993-1-8" File="End Plate BCW.con"
Template="EP EN BCW" />
    <RCNXTemplate TagName="DS" Code="EN 1993-1-8" File="End Plate BCW.con"
Template="EP EN BCW Full depth" />
  </RCNXTemplates>
</ConnectionTagFile>
```

Related Links

- *D. Connection Tags XML File Schema* (on page 1052)

D. Connection Tags sub menu

These sub-menu items are available under **Connection Tags** on the right-click pop-up menu when a member is selected.

Menu item	Description
Assign	Opens the **Assign Connection Tags** dialog and **New Connection tag** dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.
Remove	Opens the **Assign Connection Tags** dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.
View	Opens the **Assign Connection Tags** dialog.
Check	(Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

Related Links

- *G.9 Connection Tags* (on page 2125)

D. Concrete Design

D. Available Concrete Design Codes

The following concrete design codes are available in the batch design mode and in the interactive design mode.

Batch Design

Table 71: Concrete design codes available in Batch design

Country	Code	Title
Australia	D2.A. Australian Codes - Concrete Design per AS 3600 - 2001 (on page 1350)	Australian Standard - Concrete Structures
Canada	D4.A. Canadian Codes - Concrete Design per CSA Standard A23.3-94 (on page 1413)	Design of Concrete Structures
India	D8.E. Indian Codes - Concrete Design per IS 456 (on page 1669)	Code of Practice for Plain and Reinforced Concrete (2000 edition)
	D8.G. Indian Codes - Concrete Design per IS 13920-1993 (on page 1697)	Code of Practice for Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces (1993 edition)
	D8.F. Indian Codes - Concrete Design per IS 13920-2016 (on page 1683)	Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces - Code of Practice
Japan	D9.A. Japanese Codes - Concrete Design per 1991 AIJ (on page 1713)	Architectural Institute of Japan Standards for Structural Calculation of Steel Reinforced Concrete Structures
Mexico	D10.A. Mexican Codes - Concrete Design per MEX NTC 1987 (on page 1743)	Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto
Russia	D13.A. Russian Codes - Concrete Design Per SNiP 2.03.01-84* (on page 1866)	Building Regulations: Concrete and Reinforced Concrete Construction
	D13.D. Russian Codes - Concrete Design Per SP 63.1330.2012 (on page 1918)	Concrete and Reinforced Concrete Structures. Basic Provisions
South Africa	D14.A. South African Codes - Concrete Design per SABS-0100-1 (on page 1949)	The structural use of concrete Part 1:Design
United Kingdom	D3.D. British Codes - Design per BS8007 (on page 1403)	Design of concrete structures for retaining aqueous liquids

Country	Code	Title
United States	D1.F. American Codes - Concrete Design per ACI 318 (on page 1198)	Building Code for Structural Concrete (2014 edition). Building Code for Structural Concrete (2011 edition). Building Code for Structural Concrete (2008 edition). Building Code for Structural Concrete (2005 edition). Building Code for Structural Concrete (2002 edition). Building Code for Structural Concrete (1999 edition).

Advanced Concrete Design Workflow

Table 72: Available Concrete Design Codes in RCDC

Country	Code	Title
Europe	EN 02 2004	Eurocode 2: Design of concrete structures. General rules and rules for buildings
India	IS 456 + IS 13920 - 2016	Code of Practice for Plain and Reinforced Concrete & Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces (2016 edition)
	IS 456 + IS 13920 - 1993	Code of Practice for Plain and Reinforced Concrete & Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces (1993 edition)
Malaysia	EN 02 2004 + MS National Annex	Malaysian National Annex to Eurocode 2: Design of concrete structures
Singapore	EN 02 2004 + SS National Annex	Singaporean National Annex to Eurocode 2: Design of concrete structures
United Kingdom	BS 8110 97	Structural use of concrete. Code of practice for design and construction

Country	Code	Title
	EN 02 2004 + UK National Annex	UK National Annex to Eurocode 2: Design of concrete structures
United States	ACI 318 2014	Building Code for Structural Concrete (2014 edition), Imperial and Metric
	AC 318 2011	Building Code for Structural Concrete (2011 edition), Imperial and Metric

Related Links

- *D. To specify concrete design code and parameters* (on page 1069)
- *D. Batch Design versus Interactive Design Workflows* (on page 969)

D. Batch Member and Element Design Operations

How to perform concrete member and slab design in the Analytical Modeling workflow using batch operations.

D. To specify concrete design code and parameters

To initiate the design of concrete members and specify the code parameters, use the following procedure.

Note: To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the **Analysis and Design** ribbon tab, select **Concrete** in the **Design** group gallery.
   The **Concrete Design - Whole Structure** dialog opens.
2. In the **Concrete Design - Whole Structure** dialog, select the applicable concrete design code from the **Current Code** drop-down list.
3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Specify a value or option for each required parameter for a set of members and then click **Add**.

   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

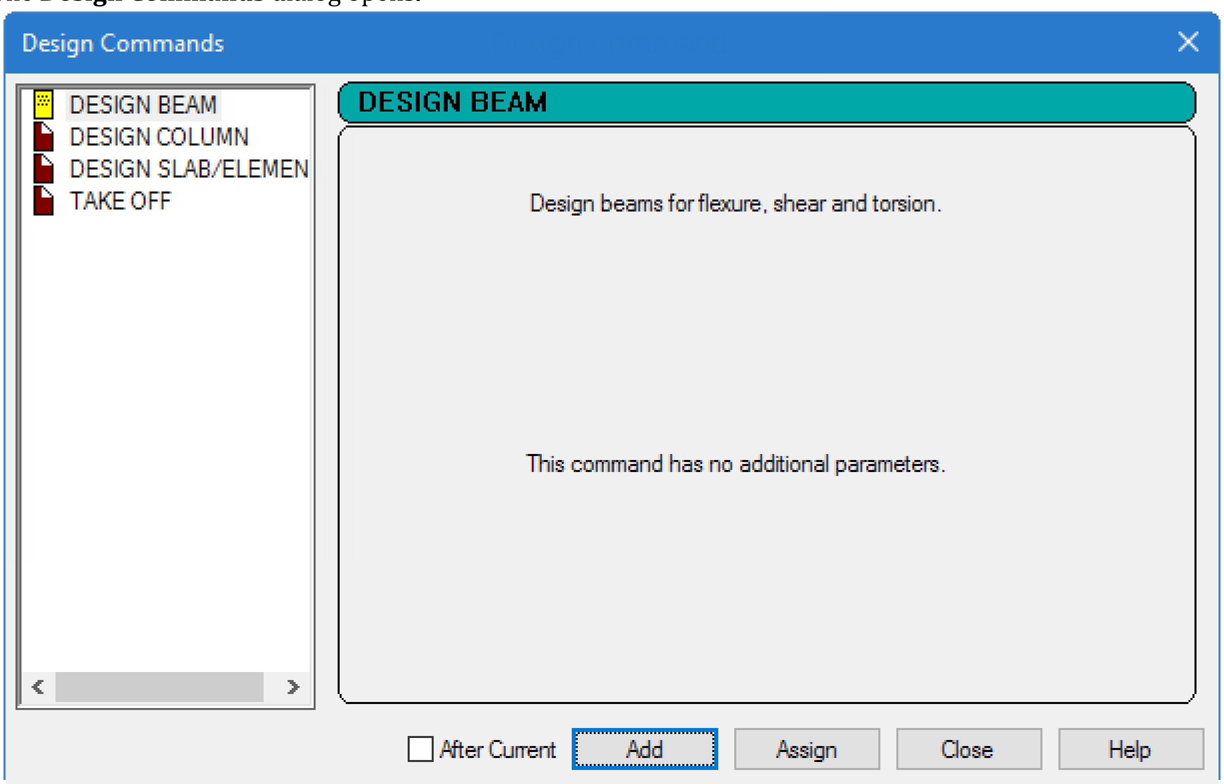
   Note: Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.
5. Click **Close**.
   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.
Related Links

- *D. Available Concrete Design Codes* (on page 1067)
- *Concrete Design - Whole Structure dialog* (on page 2936)
- *Design Parameters dialog* (on page 2934)
- *TR.53.1 Design Initiation* (on page 2684)
- *TR.53.2 Concrete Design-Parameter Specification* (on page 2685)
- *TR.53.5 Concrete Design Terminator* (on page 2686)

D. To specify concrete beam design command

To specify the design commands for concrete beams, use the following procedure.

1. On the **Concrete Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens.
   

2. To perform a design action for concrete members, select one of the following options based on the member type:

To...	Select...
design members acting as beams	the **DESIGN BEAM** tab
design members acting as columns	the **DESIGN COLUMN** tab

3. Click **Add**.
4. Click **Close**.
5. Assign the design commands to the appropriate members.

Related Links

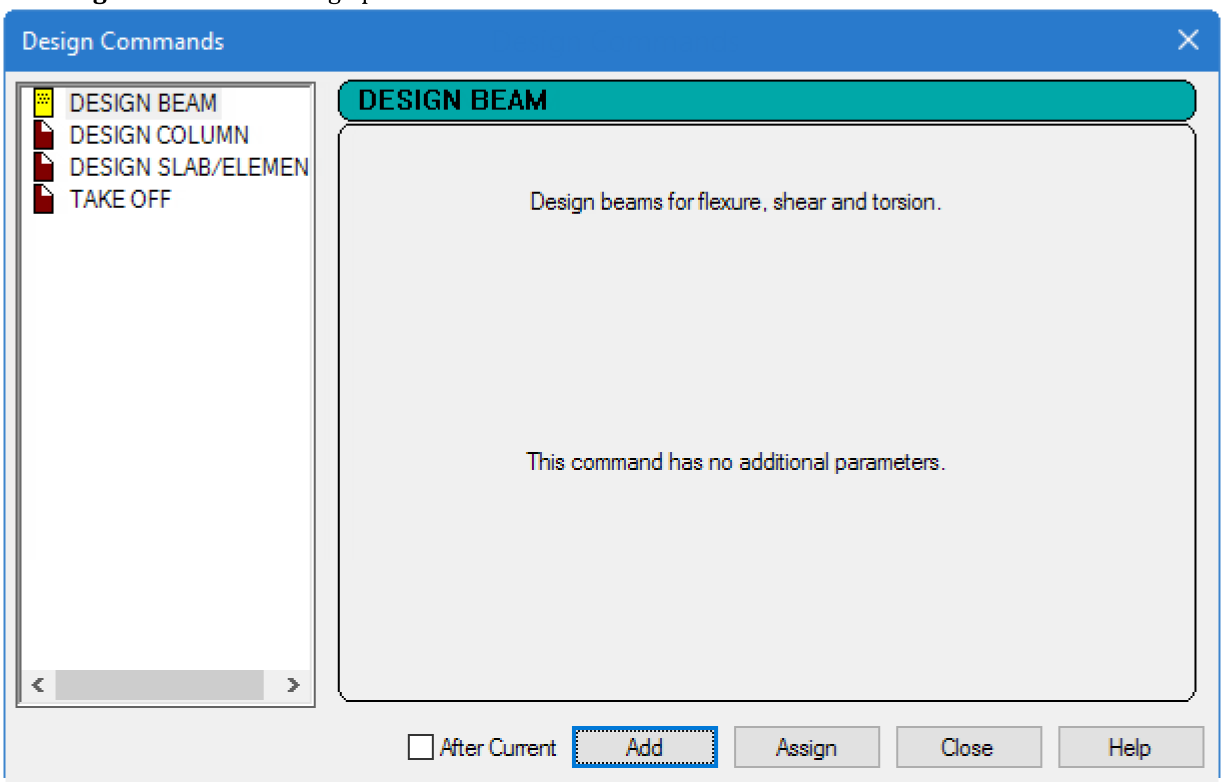
- *Concrete Design - Whole Structure dialog* (on page 2936)

- *Design Commands dialog* (on page 2935)
- *TR.53.3 Concrete Design Command* (on page 2685)

D. To generate concrete take off

To generate a summary of the total concrete volume along with the reinforcing steel bar numbers and weights for the design concrete members, use the following procedure.

1. On the **Concrete Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens.



2. Select the **TAKE OFF** tab.
3. Click **Add**.
4. Click **Close**.
5. Assign the take off commands to members or to a named group of members.

Related Links

- *Concrete Design - Whole Structure dialog* (on page 2936)
- *Design Commands dialog* (on page 2935)
- *TR.53.4 Concrete Take Off Command* (on page 2686)

D. Advanced Concrete Design

This workflow launches the STAAD.Pro Advanced Concrete Design (RCDC) program, which is used for the design and detailing of reinforced concrete building structures.

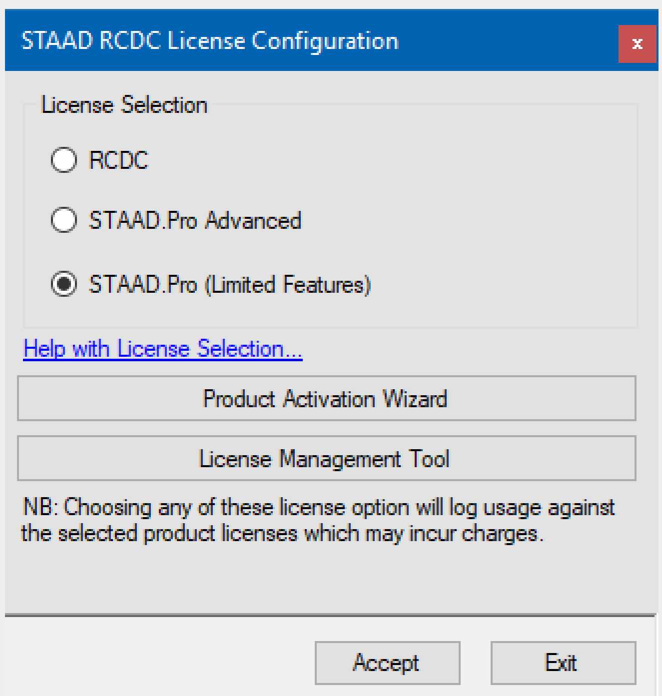
The Advanced Concrete Design workflow provides direct access for STAAD.Pro models to leverage the power of the RCDC application. This is a standalone application, which is operated outside the STAAD.Pro environment, but requires a model and results data from a suitable analysis.

The model should typically be formed from beams and columns (plates are currently not supported). RCDC can be used to design the following objects:

- Pile Caps
- Footings
- Columns and walls (note walls are determined from a STAAD model as very wide columns, see the RCDC documentation for more details)
- Beams
- Slabs (zones defined by a loop of beams, not plates, see RCDC documentation for more details)

As projects progress, each design created in RCDC is retained and displayed when RCDC is reentered so that previous designs can be recalled and/or continued. If any analysis is performed on the STAAD.Pro model from when the RCDC document was last edited, this is identified as an old document and should be reviewed again to ensure that the results of the design are still valid.

Note: When using the model in STAAD.Pro Advanced, access to RCDC will be provided using the same STAAD.Pro Advanced license and no other license is required. If using STAAD.Pro (basic), then the Advanced Concrete Design workflow will still allow access to RCDC, but this will allow only a limited feature set without the use of an additional license. This could either be a STAAD.Pro Advanced license or a STAAD Advanced Concrete license. Please check which licenses are available to you before attempting to use this and ensure that you use the correct license.



D. Advanced Slab Design

The STAAD.Pro Advanced Slab Design workflow is an integrated tool that works from within the STAAD.Pro environment. Concrete slabs can be defined and the data can be transferred to RAM Concept .

The data passed into RAM Concept includes the geometry, section and material properties, loads and combination information, and analysis results.

Note: Beginning with STAAD.Pro CONNECT Edition, direct export to ADAPT-Builder® has been deprecated.

D. Using the Advanced Slab Design workflow

Note: A successful analysis of a STAAD.Pro input file is required before initiating and using the Advanced Slab Design workflow.

D. To open the Advanced Slab Design workflow

1. Open an input file (containing the slab to be designed) in STAAD.Pro.
2. Perform a successful analysis.
3. Select **Advanced Slab Design** in the **Workflows** panel.

D. To create a load envelope

1. Select the **Envelopes** page in the **Advanced Slab Design** page control bar.
   The **Envelopes** table.

   Tip: This page is typically open by default.
2. Click **New Envelope** on the **Envelopes** table.

   workflow.

   The **Load Envelope** dialog opens.
3. (Optional) Type a title for the load envelope.

   A Load Envelope name should be entered, being defined by the chosen load cases by clicking the check boxes corresponding to each load case.
4. Select the load cases or combinations to be included in the design load envelope.

   Tip: Check the **Select All Load Cases Shown Below** option to add all available load cases to the envelope.
5. For each load case included, select a **Load Type** to describe the nature of that load.
6. Click **OK**.
   The new load envelope is added to the **Envelopes** table.

Related Links

- *D. Envelopes table* (on page 1075)

- *D. Load Envelope dialog* (on page 1076)

D. To create a slab definition

Note: A maximum of 10 holes can be defined per slab when exporting to RAM Concept.

1. Select the **Slab Design** page in the **Advanced Slab Design** page control bar.
   The **Geometry Cursor** tool is selected.
2. Select the structure geometry which will form the slab.

   Note: The slab geometry should include all plate elements which form the slab, as well as column members both above and below the slab, beam members framing the slab, and wall plate elements which are connected both above and below to the slab.
3. Click **New Slab** on the **Slabs** table.
   The **Slab Definition** dialog opens.
4. Select the **Load Envelope** with which to associate the slab definition.
5. Click **OK**.

   The new slab definition is added to the **Slabs** table.

   Note: If the selection of plates is such that the plates form two separated entities (e.g., plates that form parallel floors) then STAAD.Pro will create multiple separate slab entities.

Related Links
- *D. Slabs table* (on page 1076)
- *D. Slab Definition dialog* (on page 1077)

D. To export slab definitions to RAM Concept

1. Select the slab design definition(s) in the **Slabs** table.
2. Select the **Export** > **Export RAM Concept Information** tool in the **RAM Concept** group on the **Advanced Slab Design** ribbon tab.

   A dialog opens to confirm that the data files were created.

D. To open the STAAD.Pro slab data in RAM Concept

1. Select the slab design definition(s) in the **Slabs** table.
2. Either:

   Select the **Export** > **Run RAM Concept** tool in the **RAM Concept** group on the **Advanced Slab Design** ribbon tab.

or

Right-click on the **Slabs** table and select **Run** > **RAM Concept** from the pop-up menu.

RAM Concept launches and The **Concept import** dialog opens.

3. Leave all structure geometry element types selected (default) and click **OK.**

   Upon the completion of a successful import, a message dialog opens to inform you of the total number of structural elements and loads created.

Refer to the RAM Concept documentation for assistance performing analysis and design operations.

D. Pages in the Advanced Slab Design workflow

The Pages in the Advanced Slab Design workflow are described below in brief.

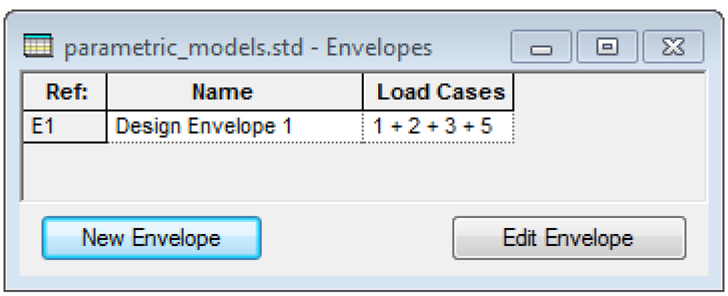
Table 73: Page Controls in the Advanced Slab Design workflow

Page	Purpose
Envelopes	Used to create set(s) from available Primary and Combination Loads defined in the current STAAD.Pro file for which analysis results are already available. When the **Envelopes** page is selected, the **Envelopes** table opens.
Slab Design	Used to create Slab definitions for slab design. When the **Slab Design** page is selected, the **Slabs** table opens.

D. **Envelopes** table

Used to view a table containing all the load envelopes created by the user for the current model. This table will also open even if there is no load envelope created yet.

Opens when the **Envelopes** page is selected in the **Advanced Slab Design** workflow.



Envelopes list Contains a list of all Envelopes available for Slab Design, along with the component load cases.

New Envelope Opens the D. Load Envelope dialog (on page 1076), which is used to create a load envelope from defined loads.

Edit Envelope Opens the **Load Envelope** dialog for editing the selected envelope.

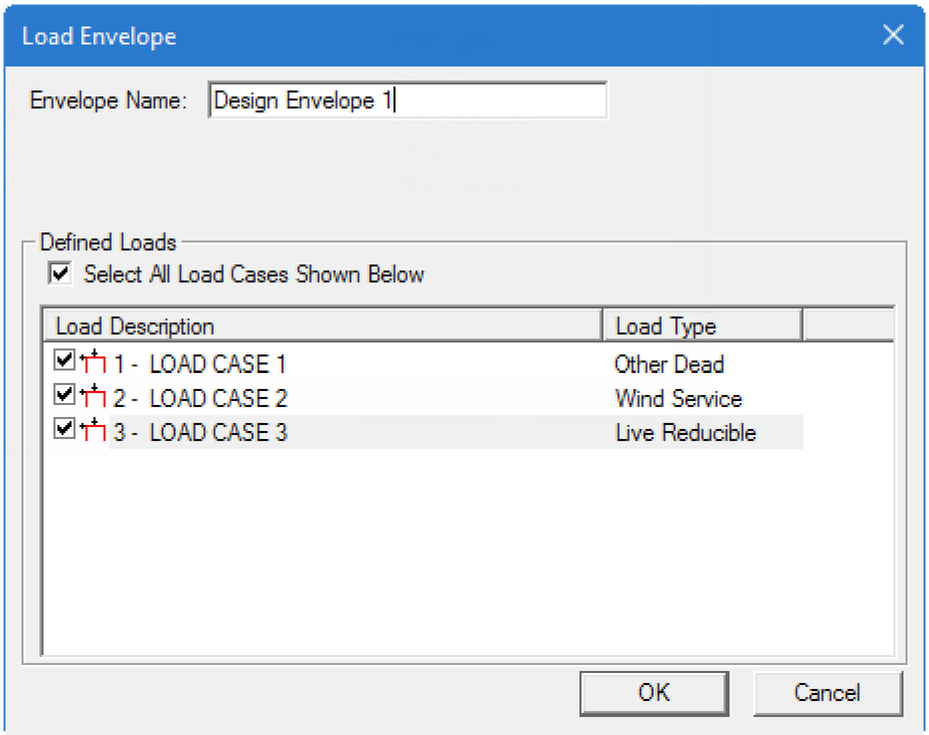
Related Links

- *D. To create a load envelope* (on page 1073)

D. **Load Envelope** dialog

Used to define an envelope of primary and combined load cases used in the analysis that will then be used by RAM Concept for performing code checks or member selections

Opens when **New Envelope** is clicked on the D. Envelopes table (on page 1075) .



Envelope Name	Specify a title for the load envelope. The default title uses an incremented id number.
Select All Load Cases Shown Below	Select this option to add all load cases in the Defined Loads list to the new load envelope.
Defined Loads list	Lists all previously defined load cases in the input file. For each load selected for the Load Envelope, the Load Type must be defined. The Load Type describes the nature of the load. Clicking in the cell for a Load Case opens a drop-down list containing all load types. Select the most appropriate type to make the assignment.
OK	Creates a new Load Envelope with the selected parameters and closes the dialog.
Cancel	Closes the dialog without creating a new load envelope.

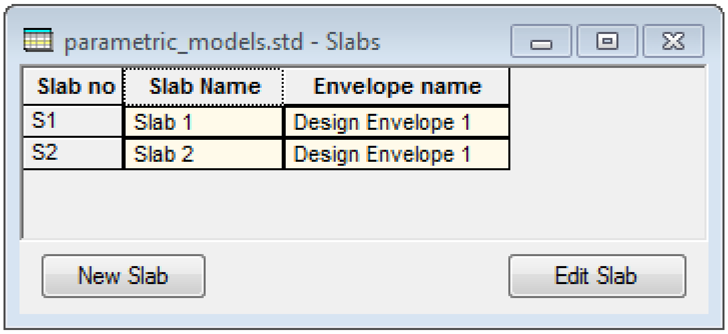
Related Links

- *D. To create a load envelope* (on page 1073)

D. **Slabs** table

Contains the slabs already defined along with the associated design load envelope.

Opens when the **Slab Design** page is selected in the **Advanced Slab Design** workflow.



Slab no	Lists sequential slab reference numbers (S1, S2, etc.)
Slab Name	Name of the Slab definition.
Envelope name	The load envelope associated with the slab.

New Slab Opens the D. Slab Definition dialog (on page 1077), which is used to create a slab definition from a set of selected plate elements.

Edit Slab Opens the **Slab Definition** dialog to edit or rename the currently selected slab.

Related Links

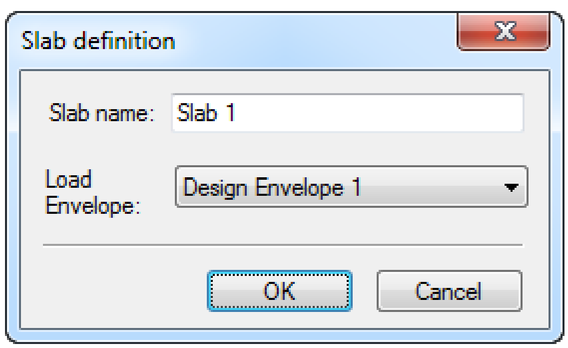
- *D. To create a slab definition* (on page 1074)

D. **Slab Definition** dialog

Used to create a slab definition from a set of selected geometry.

Opens when **New Slab** is clicked on the D. Slabs table (on page 1076) .

Note: A maximum of 10 holes can be defined per slab when exporting to RAM Concept.



Slab name	(Edit mode only) Enter the name of the Slab definition. Slab definitions are automatically named with incrementing numbers when they are created.
Load Envelope	Select a previously defined design load envelope from the list.
OK	Closes the dialog and creates a slab from the selected geometry.
Cancel	Closes the dialog without creating a slab definition.

Related Links

- *D. To create a slab definition* (on page 1074)

D. Foundation Design

The Foundation Design workflow integrates STAAD Foundation Advanced into STAAD.Pro. In this workflow, you can select loads and geometry to pass to a new STAAD Foundation Advanced project.

Note: This feature requires STAAD.Pro V8i (SELECTseries 2) (release 20.07.07) or higher.

Tip: Models containing a large number of supported nodes or load cases may result in slow performance on older computer hardware. Exporting a limited set of data can be used to improve performance in STAAD Foundation Advanced in these cases.

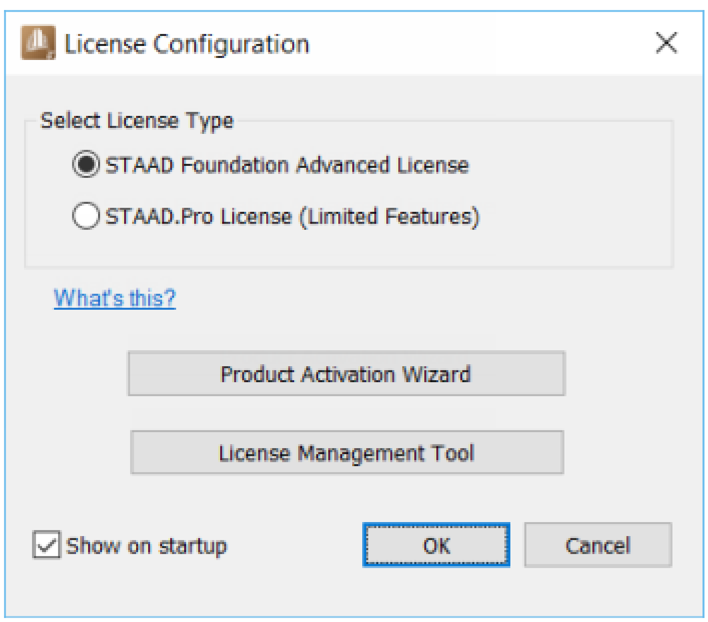
D. Using the Foundation Design workflow

Note: See Limited Versus Full Licensed Versions of STAAD Foundation Advanced (on page 1078) for details on the two license options available and the foundations that can be designed using those licenses.

Limited Versus Full Licensed Versions of STAAD Foundation Advanced

STAAD Foundation Advanced is available in two forms:

- licensed version
- a limited version which may be used with a STAAD Foundation Advanced license



Note: This limited version requires a license for STAAD.Pro but not a separate license for STAAD Foundation Advanced.

When you launch the program with STAAD Foundation Advanced installed, you are given a choices to select one of these in the **License Configuration** dialog. If you select the limited version option, then you only have access to the items as indicated in the following table. If you have purchased a license then you will have access to all the modules in STAAD.Pro.

Note: All available design codes are included for each feature.

Table 74: Items included in the free, limited version of STAAD Foundation Advanced

Items indicated with a "✔" are available with a free, limited version license. Items indicated with an "X" require a license to use.

Mode	Feature Name	Limited Version (license free)
Toolkit	All modules	X
General	Isolated footings	✔
General	Combined footings	✔
General	Pilecaps	✔
General	Mat foundations	X
General	Octagonal footings	X
General	Strap footings	X
General	Vibrating machine foundations	X
General	Combined with rib	X
Plant	All modules	X

Tip: If you uncheck the **Show on startup** option but need to access the **License Configuration** dialog, you can select the **License Configuration** tool in the **Settings** group on the **Home** ribbon tab. 

D. To open the Foundation Design workflow

1. Perform a successful analysis.
2. Select **Foundation Design** in the **Workflows** panel.

   The **Foundation Design** dialog opens.

D. To export all of the structure data to a STAAD Foundation Advanced project

This is the typical method of transferring data. Keep in mind that you can set up multiple jobs in STAAD Foundation Advanced, each containing some or all of the STAAD.Pro supports or load cases.

1. In the **Foundation Design Options** dialog, select the **All Supports** option is it is not already selected.
2. Click **Include All** to transfer all available load cases to the list of load cases To be included in design.
3. Click **STAAD Foundation Advanced**.
   STAAD Foundation Advanced opens and the STAAD.Pro support data and results are imported.

D. To export a limited set of structure data to a STAAD Foundation Advanced project

1. Either:

   Use the **Node Cursor** tool (selected by default with then Foundation Design workflow is selected) to graphically select the support nodes you want to transfer. The **Selected Supports** option will be selected automatically in the **Foundation Design** dialog.

   or

   Select the **Listed Supports** option in the **Foundation Design** dialog and then type a list of node numbers.
2. Either:

   Select one or more load cases in the **Excluded from design** list and click **Include** to transfer them to the **To be included in design** list.

   or

   Select the **Select Envelope** option and then select a load envelope from the drop-down list.
3. Click **STAAD Foundation Advanced**.
   STAAD Foundation Advanced opens and the STAAD.Pro support data and results are imported.

Tip: To change the loads or update the STAAD Foundation Advanced model per any geometry changes to your STAAD.Pro model, simply re-perform the steps involved in exporting the data.

D. **Foundation Design** dialog

Used to specify the supports and results to be exported to a STAAD Foundation Advanced project.

Opens when the **Foundation Design** workflow is selected.

Supports for Foundation Design

- **All Supports** – All supports in the structure will be exported to the STAAD Foundation Advanced project.
- **Selected Supports** – Only supports which are selected in the Active view window will be included. Use the **Nodes Cursor** tool to make selections.
- **Listed Supports** – List the support numbers of supports to included.

Load Cases and Combinations

Select the load cases to be included in the export to STAAD Foundation Advanced project:

- **Select Envelope** – select a load envelope from the drop-down list.

  Tip: You can create envelopes in the Analytical Modeling workflow. Refer to M. To create a load envelope (on page 901) for details.

- **Select Load Cases and Combinations** – select this option to choose load cases and combinations to include in the design.

  Excluded from design — All primary load cases and load combinations are listed here. Any that remain in this list are not exported to the

Table 75: Load Selector tools

Click this button	To
Include	Include load cases selected in the Excluded from design list in the To be included in design list below.
Include All	Include all available load cases in the To be included in design list below.
Exclude	Remove the selected load cases from the To be included in design list below.
Exclude All	Remove all load cases in the To be included in design list below.

To be included in design Loads included here will be exported to the STAAD Foundation Advanced project.

STAAD Foundation Advanced Starts STAAD Foundation Advanced and imports the structural data selected in this dialog into a new project file.

D. Aluminium Design

D. Available Aluminum Design Codes

The following aluminum design code is available in batch mode in STAAD.Pro.

Table 76: Aluminum Design codes available in STAAD.Pro

Country	Code	Title
US	D1.H. American Codes - Aluminum Design per 1994 ADM (on page 1264)	Specifications for Aluminum structures

Related Links

- *D. To specify aluminum design code and parameters* (on page 1083)
- *D. Batch Design versus Interactive Design Workflows* (on page 969)

D.Aluminum Design Overview

The design of aluminum members in STAAD.Pro can be achieved in the following two methods:

- check that the member meets the requirements of the design code using the Code Check command.
- for the forces resulting from the current set of loading, determine the most efficient size of section that would meet the requirements for the design code using the Select command.

Each design follows the rules as defined in a published design code specification. The list of available codes currently supported can be found at D. Available Aluminum Design Codes (on page 1081).

In order to determine the design code and code edition to which the members will be designed, the design command is preceded by a definition of the code and a series of code-related parameters. Each collection of these parameters is referred to as a “parameter block.” In the STAAD.Pro input file, this is initiated with the command `PARAMETER` and then typically terminated with the design command (e.g., `CHECK CODE`).

The parameter block provides the means by which you specify design-related member attributes required for the specified design code that have not been specified in the analytical model thus far. Each parameter has a default value which is ascribed to members to which a design command applies. When you assign a parameter to a given member, any design command that follows this assignment will use the specified parameter in the design. The default value of each design code parameter is listed in the appropriate section of the documentation.

```
PARAMETER
CODE design code/version
parameters modified from default values
design command
```

It is possible to define multiple design command and parameter blocks :

```
PARAMETER 1
CODE design code/version
parameters modified from default values
design command 1
design command 2
design command 3
...
PARAMETER 2
CODE design code/version
parameters modified from default values
design command 4
design command 5
design command 6
```

Also, each design command can be defined for different sets of members and different design commands:

```
PARAMETER 1
CODE design code/version
parameters modified from default values
CHECK CODE MEMBER 1 TO 10
SELECT MEMBERS 5 TO 20
```

Related Links

- *TR.49 Code Checking Specification* (on page 2677)

- *TR.49.1 Member Selection Specification* (on page 2678)
- *TR.49.2 Member Selection by Optimization* (on page 2679)
- *TR.49.1 Member Selection Specification* (on page 2678)
- *D. To specify aluminum design commands* (on page 1083)

D. To specify aluminum design code and parameters

To initiate the design of aluminum members and specify the code parameters, use the following procedure.

Note: To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the **Analysis and Design** ribbon tab, select **Aluminium** in the **Design** group gallery.
   The **Aluminum Design - Whole Structure** dialog opens.

   Note: Only the US Aluminum code is available for design of aluminum members.
2. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
3. Specify a value or option for each required parameter for a set of members and then click **Add**.

   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

   Note: Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.
4. Click **Close**.

   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
5. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.
Related Links

- *D. Available Aluminum Design Codes* (on page 1081)
- *Aluminum Design - Whole Structure dialog* (on page 2937)
- *Design Parameters dialog* (on page 2934)
- *TR.48.1 Parameter Specifications* (on page 2676)

D. To specify aluminum design commands

To specify the code checking or design commands to be used for the aluminum design operation, use the following procedure.

1. On the **Aluminum Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens.
2. To perform a design action, select one of the following options (as they are available for the code selection):

To...	Select...
check the capacity of the current member sizes	the **CHECK CODE** tab

To...	Select...
select a member size based on the last analysis results	the **SELECT** tab

3. Click **Add**.
4. Click **Close**.
5. Assign the design commands to the appropriate members.

Related Links

- *Aluminum Design - Whole Structure dialog* (on page 2937)
- *Design Commands dialog* (on page 2935)
- *D.Aluminum Design Overview* (on page 1082)
- *TR.49 Code Checking Specification* (on page 2677)
- *TR.49.1 Member Selection Specification* (on page 2678)
- *TR.49.2 Member Selection by Optimization* (on page 2679)
- *TR.49.1 Member Selection Specification* (on page 2678)

D. To generate aluminum take off

To generate a summary of all aluminum sections with their lengths and weights for the whole structure or a list of members, use the following procedure.

1. On the **Aluminum Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens.
2. Select the **TAKE OFF** tab.

   This will generate a list of sections used including total length and weight.
3. Click **Add**.
4. Click **Close**.
5. Assign the take off command to members or to a named group of members.

Related Links

- *TR.51 Steel and Aluminum Take Off Specification* (on page 2682)
- *Design Commands dialog* (on page 2935)

D. Timber Design

D. Available Timber Design Codes

The following timber design codes are available in batch design in STAAD.Pro.

Table 77: Timber Design codes available in STAAD.Pro

Country	Code	Title
Canada	D4.D. Canadian Codes - Timber Design per CAN/CSA-086-01 (on page 1439)	Wood Design Standard

Country	Code	Title
Europe	D5.E. European Codes - Timber Design Per EC 5: Part 1-1 (on page 1582)	Eurocode 5: Design of timber structures - Part 1.1: General-Common rules and rules for buildings
United States	D1.G. American Codes - Timber Design per AITC Code (on page 1248)	Timber Construction Manual - AITC 1984
	D1.G. American Codes - Timber Design per AITC Code (on page 1248)	Timber Construction Manual - AITC 1994

Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *D. To specify timber design code and parameters* (on page 1085)
- *D. Batch Design versus Interactive Design Workflows* (on page 969)

D. To specify timber design code and parameters

To initiate the design of timber members and specify the code parameters, use the following procedure.

Note: To reduce the number of load cases used in design operations, you may want to create a load envelope or load list prior to specifying the design.

Batch mode design is specified and performed in the Analytical Modeling workflow.

1. On the **Analysis and Design** ribbon tab, select **Timber** in the **Design** group gallery.
   The **Timber Design - Whole Structure** dialog opens.
2. In the **Timber Design - Whole Structure** dialog, select the applicable concrete design code from the **Current Code** drop-down list.
3. Click **Define Parameters**.
   The **Design Parameters** dialog opens.
4. Specify a value or option for each required parameter for a set of members and then click **Add**.

   You only need to specify parameters that require a different value from the default value. Repeat this step for all non-default parameters.

   Note: Different parameters can be used for different member type designs (e.g., columns, beams, etc.). Alternatively, you can use a separate set of parameters for different member types.
5. Click **Close**.

   The design parameters are marked with an icon. This indicates that the need to be assigned to members.
6. Use one of the STAAD.Pro assignment methods to assign each parameter to the applicable members.

You will now need to instruct the program to perform a design command on these members.
Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *D. Available Timber Design Codes* (on page 1084)
- *Timber Design - Whole Structure dialog* (on page 2939)
- *Design Parameters dialog* (on page 2934)

D. To specify timber design commands

To specify the code checking or design commands to be used for the timber design operation, use the following procedure.

1. On the **Timber Design - Whole Structure** dialog, click **Commands**.
   The **Design Commands** dialog opens. The tabs (if any) available in this dialog are dependent on the current design code selection. Not all commands are available for every design code.
2. To perform a design action, select one of the following options (as they are available for the code selection):

To...	Select...
check the capacity of the current member sizes	the **CHECK CODE** tab
select a member size based on the last analysis results	the **SELECT** tab (AITC 1984 code only)

3. Click **Add**.
4. Click **Close**.
5. Assign the design commands to the appropriate members.

Related Links

- *Timber Design - Whole Structure dialog* (on page 2939)
- *Design Commands dialog* (on page 2935)

D. Design Codes

This section contains engineering reference material and input commands for structural design using batch mode. The codes are organized by country, material, and edition.

Effort has been made to provide some basic information about the analysis considerations and the logic used in the design approach. A brief outline of the factors affecting the design along with references to the corresponding clauses in the codes is also provided. Examples are provided at the appropriate places to facilitate ease of understanding of the usage of the commands and design parameters. You are urged to refer to the Examples Manual for solved problems that use the commands and features of STAAD.Pro. Since the STAAD.Pro output contains references to the clauses in the code that govern the design, we recommend that you consult the documentation of the code of that country for additional details on the design criteria.

D1. American Codes

D1.A. American Codes - Steel Design per AISC 360 Unified Specification

Steel member design per ANSI/AISC 360-05, 360-10, and 360-16 *Specifications for Structural Steel Buildings*, is available in STAAD.Pro. These specifications are published as part of the AISC *Steel Construction Manual*. Since the ASD and the LRFD method are both addressed in those specifications, they are referred to as `UNIFIED`.

To use the 2016 edition (default), specify the command:

```
CODE AISC UNIFIED
```

or

```
CODE AISC UNIFIED 2016
```

To use the 2010 edition, specify the command:

```
CODE AISC UNIFIED 2010
```

To use the 2005 edition, specify the command:

```
CODE AISC UNIFIED 2005
```

Tip: Either method may be selected in the user interface using the **Steel Design - Whole Structure** dialog.

Design can be performed according to the provisions for **Load and Resistance Factor Design** (LRFD) or to the provisions for **Allowable Strength Design** (ASD), as per section B3 of the code. This selection of the design methodology can be done through the `METHOD` parameter. The full list of parameters is given in D1.A.6 Design Parameters (on page 1100).

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

D1.A.1 General Comments on Design as per AISC Unified Code

Both Allowable Stress Design and Load Resistance Factor Design methods are implemented in STAAD. The selection of the method can be done through the `METHOD` parameter explained in the parameter list. This Unified Code allows the designer to design the member as per LRFD as well as ASD method.

Design for Strength Using Load and Resistance Factor Design (LRFD)

Design according to the provisions for Load and Resistance Factor Design (LRFD) satisfies the requirements of the AISC 360 Unified Code Specification, when the design strength of each structural component equals or exceeds the required strength determined on the basis of the LRFD load combinations.

Design shall be performed in accordance with Equation B3-1 of the Code:

$$R_u \le \phi R_n$$

where

R_u = required strength (LRFD)
R_n = nominal strength
φ = resistance factor
φR_n = design strength

Design for Strength Using Allowable Strength Design (ASD)

Design according to the provisions for Allowable Strength Design (ASD) satisfies the requirements of the AISC 360 Unified Code Specification when the allowable strength of each structural component equals or exceeds the required strength determined on the basis of the ASD load combinations.

Design shall be performed in accordance with Equation B3-2 of the Code:

$$R_a \le R_n/\Omega$$

where

R_a = required strength (ASD)
R_n = nominal strength
Ω = safety factor

R_n/Ω	=	allowable strength

D1.A.2 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, non-compact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro is capable of determining the section classification for the standard shapes and design accordingly.

The Section Classification is done as per section B4 and Table B4.1, for Stiffened and Un-Stiffened Elements of a section.

D1.A.3 Member Properties

For specification of member properties of standard American steel sections, the steel section library available in STAAD.Pro may be used. The syntax for specifying the names of built-in steel shapes is described in the next section.

D1.A.4 Built-in Steel Section Library

The following sections describe specification of steel sections from the AISC Steel Tables.
Related Links
- *G.6.2 Built-In Steel Section Libraries* (on page 2116)

D1.A.4.1 AISC Steel Table

Almost all AISC steel shapes are available for input. Following are the descriptions of all the types of sections available:

Wide Flanges (W shapes)

All wide flange sections as listed in AISC are available the way they are written, e.g., W10X49, W21X50, etc.

```
20 TO 30 TA ST W10X49
33 36 TA ST W18X86
```

C, MC, S, M, HP Shapes

The above shapes are available as listed in AISC (9th Edition) without decimal points. For example, C8X11.5 will be input as `C8X11` and S15X42.9 will be input as `S15X42`, omitting the fractional portion of the weight past the decimal.

Note: Exception: `MC6X151` for MC6X15.1 and `MC6X153` for MC6X15.3.

```
10 TO 20 BY 2 TA ST C15X40
1 2 TA ST MC8X20
```

Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel.

```
21 22 24 TA D MC9X25
55 TO 60 TA D C8X18
```

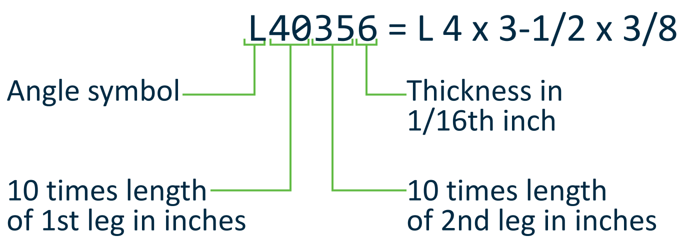
Front to front channel steel sections can be defined by using the FR (Double Channel – Front to Front) option and any spacing required between the channels are specified with the SP designation.

```
61 62 TABLE FR C4X5 SP 0.5
```

Note: The SP parameter is optional, but if it is not set, the section will not be assumed to be a closed box for torsional calculations.

Angles

Angle specifications in STAAD.Pro are different from those in the AISC manual. The following example illustrates angle specifications.



Similarly, `L505010` = L 5 x 5 x 5/8 and `L904016` = L 9 x 4 x 1

At present, there are two ways to define the local y and z-axes for an angle section. To make the transition from the AISC Manual to the program data easy, the standard section for an angle is specified:

```
51 52 53 TA ST L40356
```

This specification has the local z-axis (i.e., the minor axis) corresponding to the Z-Z axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

```
54 55 56 TA RA L40356
```

Note: RA denotes reverse angle

Double Angles

Short leg back to back or long leg back to back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose.

```
14 TO 20 TA LD L35304 SP 0.5
```

Long leg back to back L3-1/2x3x1/4 with 0.5 space.

```
23 27 TA SD L904012
```

Short leg back to back L 9x4x3/4

Tees

Tees are not input by their actual names, as they are listed in the AISC manual, but instead by designating the beam shapes (W and S) from which they are cut. For example

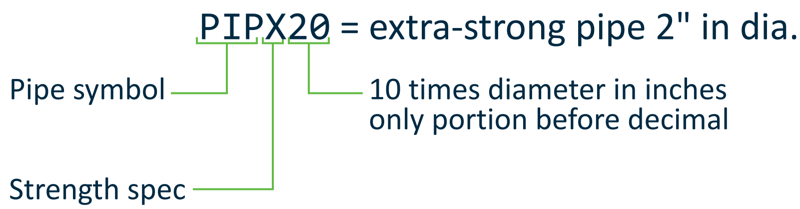
```
1 2 5 8 TA T W8X24
```

Tee cut from W8x24, or a TW4x12.

Pipes

Two types of specifications can be used for pipe sections. Pipe sections listed in the AISC manual can be specified as follows.

```
5 TO 10 TA ST PIPX20
```



Where the strength spec is one of the following:

S = standard
X = extra-strong
D = double extra-strong

General pipe sections may be input by their outer and inner diameters. For example,

```
1 TO 9 TA ST PIPE OD 2.0 ID 1.875
```

Indicates a pipe with an outer diameter of 2.0 and inner diameter of 1.875 in current input units.

Round HSS

Round hollow structural sections listed in the AISC manual can be specified as follows.

```
5 TO 10 HSS16X0.438
11 TO 12 HSS14X0.5_A1085
```

Indicates members 5 through 10 as 16" outer diameter and a nominal wall thickness of 7/16" (0.4375"). Members 11 and 12 have a 14" outer diameter and a 1/2" nominal wall thickness and use A1085 grade material.

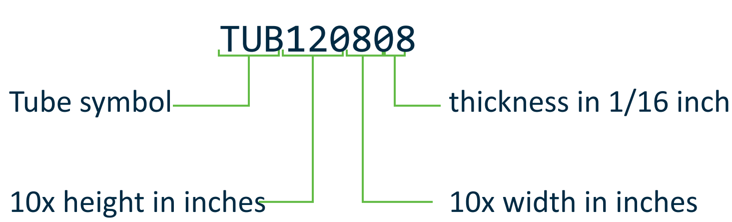
Notes:

- AISC codes require that the design of HSS sections use section properties calculated based on a reduced wall thickness. The STAAD.Pro database values incorporated a design thickness of 0.93× the nominal thickness.
- HSS sections using A1085 grade material per the AISC 360-16 and 360-10 codes are designed using section properties with the full nominal wall thickness. Thus, a separate set of tables is available to specify this material grade as these use different section properties.

Tubes

Two types of specifications can be used for tube sections. Tube sections from the AISC tables can be specified as follows.

```
5 TO 10 TA ST TUB120808
```



General tube sections may be input by their dimensions (height, width, and thickness) as follows.

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

Indicates a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 in the current input units.

Note: Member Selection cannot be performed on tubes specified by their dimensions. Only code checking can be performed on these sections.

Rectangular HSS

Rectangular hollow structural sections listed in the AISC manual can be specified as follows.

```
5 TO 10 HSST14X10X0.313
11 TO 12 HSST6X3X0.375_A1085
```

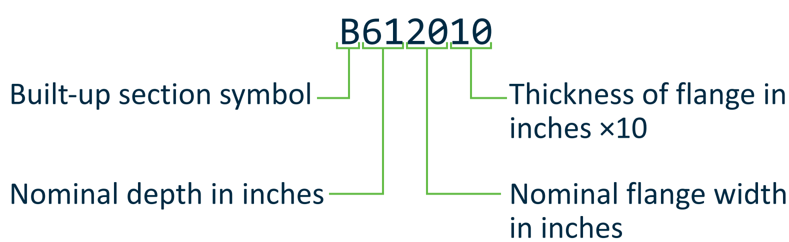
Indicates members 5 through 10 as a 14" x 10" tube with a nominal wall thickness of 5/16" (0.3125"). Members 11 and 12 are a 6" x 3" tube with a 3/8" nominal wall thickness and use A1085 grade material.

D1.A.4.2 Welded Plate Girders

The AISC welded plate girder shapes (pages 2-230 and 2-231 – AISC 9th edition) are available in the Steel Section library of the program.

```
1 TO 10 TA ST B612010
15 16 TA ST B682210
```

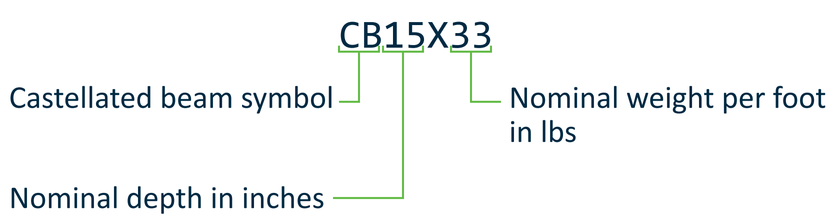
Nomenclature



D1.A.4.3 Castellated Beams Section Sizes

STAAD.Pro incorporates the non-composite castellated beam tables.

All castellated beams are listed with their nominal depth and their nominal weight per foot (i.e., similar to the nomenclature used for American wide flange shapes).



Tip: You can click **View Table** in the **Section Profile Tables** dialog to view the properties table, which includes the root beam profile size used for each castellated beam.

```
10 TO 20 TA ST CB15X33
2 TA ST CB27x40
```

Related Links

- *D1.B.2 Castellated Beams* (on page 1147)

D1.A.5 Limit States

Slenderness

The slenderness check is *not* used as a critical member ratio check. If a slenderness check is included and exceed unity (1.0), then it may be reported as the governing criteria. However, if it is less than unity, it will be reported but will not be used as the governing ratio.

D1.A.5.1 Axial Tension

The criteria governing the capacity of tension members are based on:

- Tensile Yielding in Gross Section and
- Tensile Rupture of Net Section.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section D2-(a) of the code.

The second limit state involves fracture at the section with the minimum effective net area, and the corresponding check is done as per section D2-(b) of the code.

STAAD.Pro calculates the tension capacity of a given member based on these two limit states.

The Net Section Area may be specified by the user through the use of the parameter `NSF` (see D1.B.1.2 Design Parameters (on page 1134)). The Effective Net Area of tension members can be determined by using the Shear Lag Factor. You can also input the shear lag factor through the use of the parameter `SLF`.

Related Links

- *V.AISC 360-16 C Tension ASD* (on page 5013)
- *V.AISC 360-16 C Tension LRFD* (on page 5022)
- *V. AISC 360-16 I Tension LRFD* (on page 5031)
- *V.AISC 360-16 L Tension ASD* (on page 5040)
- *V.AISC 360-16 L Tension LRFD* (on page 5048)
- *V. AISC 360-10 W D.1* (on page 5348)
- *V. AISC 360-10 - D-2* (on page 5354)
- *V. AISC 360-10 WT D.3* (on page 5361)
- *V. AISC 360-10 HSST D.4* (on page 5367)
- *V. AISC 360-10 HSSP D.5* (on page 5374)
- *V. AISC 360-10 2L D.6* (on page 5380)
- *V. AISC 360-05 Tension* (on page 5398)

D1.A.5.2 Axial Compression

The Design Compressive Strength (LRFD), $\phi_c \times P_n$, and the Allowable Compressive Strength (ASD), P_n / Ω_c , are calculated by the program.

The Nominal Compressive Strength, P_n, shall be the minimum value obtained according to the Limit States of:

- Flexural Buckling,
- Torsional Buckling, and
- Flexural-Torsional Buckling.

The Nominal Compressive Strength, P_n, for a particular member is calculated by STAAD.Pro according to the procedure outlined in Chapter E, section E3 to E5, of the unified code specifications. For slender elements, the procedure described in section E7 is used.

For single angle members in compression designed per AISC 360-16, the conditions in E5 for use of the modified slenderness ratio in Cl. E5.(a) are assumed to be met by default. However, the engineer can specify these conditions by use of the `E5P` parameter. Further, the modified slenderness ratio for box or space trusses can be utilized by using the `IMM` parameter.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY and KZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD.Pro immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

Related Links

- *D1.A.6 Design Parameters* (on page 1100)
- *V.AISC 360-16 Double L E.5* (on page 4816)
- *V. AISC 360-16 I Compression LRFD* (on page 4832)
- *V.AISC 360-16 Pipe E.11* (on page 4842)
- *V.AISC 360-16 Rect HSS E.9* (on page 4857)
- *V.AISC 360-16 W E.1A* (on page 4872)
- *V.AISC 360-16 W E.1B* (on page 4888)
- *V. AISC 360-10 W E.1C* (on page 5158)
- *V. AISC 360-10 W E.1D* (on page 5164)
- *V. AISC 360-10 - E-2* (on page 5171)
- *V. AISC 360-10 Built up I E.3* (on page 5178)
- *V. AISC 360-10 Double L E.5* (on page 5187)
- *V. AISC 360-10 Double L E.6* (on page 5193)
- *V. AISC 360-10 WT E.7* (on page 5200)
- *V. AISC 360-10 Rect HSS E.9* (on page 5206)
- *V. AISC 360-10 Pipe E.11* (on page 5213)
- *V. AISC 360-10 Built Up I E 12* (on page 5220)
- *V. AISC 360-05 Compression* (on page 5392)

D1.A.5.3 Flexural Design Strength

The Design Flexural Strength (LRFD), $\phi_b \cdot M_n$, and the Allowable Flexural Strength (ASD), M_n / Ω_b , are being calculated by the program.

The Nominal Flexural Strength, M_n, is determined according to Sections F2 through F12 of unified code specifications, for different types of rolled sections.

Doubly symmetric, I-shaped sections with slender flanges are also checked, as per Section F3 of AISC 360-10.

The Nominal Flexural Strength of a member is determined by the limit states of Yielding (Y), Lateral-Torsional Buckling (LTB), Flange Local Buckling (FLB), Web Local Buckling (WLB), Tension Flange Yielding (TFY), Leg Local Buckling (LLB), and Local Buckling (LB).

The program internally calculates the Lateral-Torsional Buckling Modification Factor (C_b) for non-uniform moment diagrams when both ends of the unsupported segment are braced. The purpose of this factor is to account for the influence of the moment gradient on lateral-torsional buckling. Alternatively, this can be directly specified using the CB parameter.

To specify laterally unsupported length of member flanges, the parameters UNB, UNL, UNR, and UNT can be used. By default, these take the value of the member length.

Related Links

- *V.AISC 360-16 Angle F.11A* (on page 4564)
- *V.AISC 360-16 C Flex Mem F.2-1A* (on page 4586)
- *V.AISC 360-16 C LTB Test F.2B* (on page 4602)
- *V.AISC 360-16 HSST Compact Flange F.6* (on page 4618)
- *V.AISC 360-16 HSST NonCompact Flange F.7* (on page 4635)
- *V.AISC 360-16 HSST Slender Flange F.8* (on page 4653)
- *V.AISC 360-16 I Minor Axis Bending F.5* (on page 4670)
- *V.AISC 360-16 Pipe F.9* (on page 4686)
- *V.AISC 360-16 W Flex Memb F.1-1A* (on page 4701)
- *V. AISC 360-16 W Flexural Check LRFD* (on page 4715)
- *V.AISC 360-16 W Local Buckling F.3A* (on page 4725)
- *V.AISC 360-16 W LTB Test F.1-2B* (on page 4740)
- *V.AISC 360-16 W Member Selection F.4* (on page 4756)
- *V.AISC 360-16 WT Shape F.10* (on page 4784)
- *V. AISC 360-05 Bending* (on page 5386)

D1.A.5.4 Design for Shear

The Design Shear Strength (LRFD), $Q_v \times V_n$, and the Allowable Shear Strength (ASD), V_n / Ω_v , are calculated by the program, as per section G2 of the unified code specifications.

The Nominal Shear Strength, V_n , of un-stiffened or stiffened webs, is calculated taking care of limit states of shear yielding and shear buckling. The sections G4 to G7 of the code specifications are used to evaluate Nominal Shear Strength, V_n for different types of rolled sections.

Shear Capacity Along the Major Axis

For determining shear capacity, Vc in the major axis Clause G2, G3, G4 are followed based on the different shapes.

Note: For user-defined sections the value of shear area will be used instead of the term 'Aw' in the equation in the above-mentioned clauses.

Shear Capacity Along the Weak Axis

The nominal shear strength, V_n, for each shear resisting element in doubly symmetric and singly symmetric shapes loaded in the weak axis (minor axis, or along the flanges) without torsion in is determined per AISC 360-16 G6 as follows:

$$V_n = 0.6F_y b_f\, t_f C_{v2} \tag{G6-1}$$

The use of the section dimensions[1] bf& tf in the above equation will be based on the section profile type. The following table lists the value of the terms bf, tf , kv, Cv2 that are used in the calculation of shear capacities for various section profile shapes, when subject to shear along the z-axis of the section.

Shape	Use of terms $b_f t_f$ in eqn G6-1	Dims used to calculate shear buckling coeff. Cv2 (Ref Cl G.2)	Ceoff kv used to calculate Cv2
Built Up Box	bf = 2.0 × (B – 2×wall thickness) & tf = flange thickness	h = (B – 2×wall thickness) & t = flange thickness	5.0
HSS Box	bf = 2.0 × Width between fillets & tf = wall thickness	h = Width between fillets & t = wall thickness	5.0
Rolled/ Channel	bf = 2.0 × Flange width & tf = flange thickness	h = Flange Width & t = Flange thickness	1.2
Rolled / Built Up I	bf = 2.0 × Flange width & tf = flange thickness	h = Flange Width & t = Flange thickness	1.2
I With Cover Plate(s)[2]:	Bf×tf = 2.0 × Flange width × flange thickness + width of top cover plate × thickness of top plate + width of bottom plate × thickness of bottom plate	h = Flange Width & t = Flange thickness for the base I shape & h = width of plate and t = thickness of plate	1.2
L- Section	bf = Length of leg along the X -axis & tf = thickness	h = Length of leg along the X -axis & t = thickness	1.2
T – Section	bf = Flange width & bf = flange thickness	h = Flange width (bf) & t = flange thickness (tf)	1.2
Solid bar / rod[3]	Bf×tf = 0.5 × section area	1.0	-

Notes:

1. For the shear in the weak axis, the value is determined using Bf and Tf alone. AZ cannot be used.
2. The shear capacities of the base I section and the top and / or bottom cover plates are evaluated separately using the dims shown and a Kv = 1.2. These values are then added to get the final section shear strength.

3. For solid rectangular bars and rods, half the total section area is assumed to resist the shear along the vertical axis and the other half to resist the shear along the horizontal axis. This could lead to conservative results in some cases.

Related Links

- *V. AISC 360-16 Shear Strong Axis* (on page 4903)
- *V. AISC 360-16 Shear Weak Axis* (on page 4918)
- *V. AISC 360-10 W Shape Strong Axis Shear G.1* (on page 5257)
- *V. AISC 360-10 C Strong Axis Shear G.2* (on page 5263)
- *V. AISC 360-10 L Shear Capacity G.3* (on page 5269)
- *V. AISC 360-10 HSST Shear Capacity G.4* (on page 5275)
- *V. AISC 360-10 HSSP Shear Capacity G.5* (on page 5282)
- *V. AISC 360-10 - G-6* (on page 5288)
- *V. AISC 360-10 C Weak Axis Shear G.7* (on page 5341)

D1.A.5.5 Design for Combined Forces

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by sections H1 and H3. These interaction formulas cover the general case of biaxial bending combined with axial force and torsion. They are also valid for uniaxial bending and axial force.

Related Links

- *V. AISC 360-10 - H.1B* (on page 5228)
- *V. AISC 360-10 W Tens BM H3* (on page 5234)

D1.A.5.6 Design for Torsion

Stresses due to torsion in non-HSS sections are considered per AISC 360-10 Section H3.3. This section states that the available torsional strength for non-HSS members shall be the least value obtained according to the limit states of yielding under normal stress, shear yielding under shear stress, or buckling:

$$\phi_t = 0.9 \text{ (LRFD)}, \Omega_t = 1.67 \text{ (ASD)}$$

For the limit state of yielding under normal stress (H3-7):

$$F_n = F_y$$

For the limit state of shear yielding under shear stress (H3-8):

$$F_v = 0.6F_y$$

The calculation of F_v and F_n is based on AISC Design Guide 9 *Torsional Analysis of Structural Steel Members* (DG-9). In general terms, in case of shear stress, F_v will comprise of components of shear stress due to shear about both axes, warping shear stress and shear stress due to pure torsion. In case of normal stress F_n , stress due to axial force and stress due to flexure about both axes is considered. For some sections, like Single Angles and Tees, the component due to warping is negligible with respect to stress for pure torsion (Ref. Section 4.2 and 4.3 of Design Guide 9).

Notes

- STAAD.Pro will perform these torsion design checks when the `TORSION` parameter has been set to `1` (these are *not* checked by default).
- When torsion checks are performed, `TRACK 3` output may be used to provide detailed torsion design output for Design Guide 9 checks.

- The torsion checks per Design Guide 9 require additional analysis to calculate Θ (rotation of the element due to applied torsion; refer to the following sections) at the 13 design segments along the member. Thus there is a performance cost for each torsion check. Therefore, it is recommended that torsion checks only be performed on the necessary members (rather than all members).

Pure Torsional Shear Stress

These shear stresses are always present in the cross-section of a member subjected to torsional moment. They are in plane shear stresses which vary linearly along the thickness of an element.

$$\tau_t = Gt\Theta'$$

where

where

τ_t	=	the pure torsional shear stress at the element edge
G	=	the shear modulus of elasticity of steel
t	=	thickness of the element
Θ'	=	first derivative of rotation expressed as a function of local x (distance from start end to point where rotation is calculated)

In the case of an element with a rectangular cross section:

$$\tau_t = T_u t/J$$

In the case of a hollow circular element or a pipe section with an inner radius of R:

$$\tau_t = T_u R/J$$

In the case of a tube section:

$$\tau_t = T_u/(2bht)$$

where

where

T_u	=	the total torsional moment action at any location along the beam

Shear Stress Due to Warping

When warping in a member is restrained, in plane shear stresses are developed which are constant along thickness of the element but vary along the length of the element.

$$\tau_{ws} = -ES_{ws}\Theta'''/t$$

where

where

τ_{ws}	=	shear stress at a point,s, due to warping
E	=	the modulus of elasticity of steel
t	=	thickness of the element
S_{ws}	=	warping statical moment at a point, s
Θ'''	=	third derivative of rotation expressed as a function of local x (distance from start end to point where rotation is calculated)

Note: The shear stress due to warping is neglected for angle, tee, tube, or pipe sections.

Normal Stress Due to Warping

When warping in a member is restrained, direct stress acting perpendicular to the cross-section of the element is generated. These stresses are constant along the cross-section but vary along the length of the member.

$$\sigma_{ns} = EW_{ns}\Theta''$$

where

where

σ_{ns}	=	normal stress at a point,s, due to warping
E	=	the modulus of elasticity of steel
W_{ns}	=	normalized warping function at a point, s
Θ''	=	second derivative of rotation expressed as a function of local x (distance from start end to point where rotation is calculated)

Note: Point s refers to a point on the cross section area of a particular section as explained in Section 3.2.2 of Design Guide 9.

Combined Stresses due to Axial, Bending, and Torsional Stresses

Under section 4.6 of Design Guide 9, the combined stress in a section due all the stresses as explained in sections 4.1, 4.2, 4.3, and 4.4 is

$$f_n = \sigma_a + \sigma_{bz} + \sigma_{by} + \sigma_s$$

$$f_v = \tau_{sz} + \tau_{sy} + \tau_t + \tau_{ws}$$

These stresses are calculated at 13 sections along the beam length.

Considered Loads

Only member loads of the following types are considered in these checks:

- concentrated torque (moment about local x axis)
- concentrated force eccentric from the member shear center
- uniformly distributed torque (full or partial)
- uniformly distributed force eccentric to the member shear center
- end torques (only considered when end supports are fixed)

Linearly varying torque is *not* considered in the torsion checks. Joint loads are also *not* considered in the torsion checks.

The boundary conditions for torsional analysis and the method to calculate rotation,Θ , and its derivatives are used as described in DG-9.

STAAD.Pro calculates the stresses due to flexure, pure torsion, and warping torsion at 13 different sections along the member length. The total stress is the vector summation at each location.

Related Links

- *V. AISC 360-16 - Torsion* (on page 5058)
- *V. AISC 360-10 HSST Torsional Strength H.5A* (on page 5240)
- *V. AISC 360-10 HSSP Torsional Strength H.5B* (on page 5249)

D1.A.5.7 Design of Web-Tapered Members

AISC 360 05/10 specifications have been incorporated into STAAD.Pro to perform code checking on web tapered wide flange, square, and round shapes.

Note: Member selection cannot be performed on web-tapered members. That is, `SELECT ALL`, `SELECT OPTIMIZE`, and `PROFILE` are *not* applicable to web-tapered members.

The section properties used for web-tapered members are based on interpolated values between the start and depths of the member. Similar interpolation of values from start and end values is done for square and round tapered members.

Related Links

- *TR.20.3 Tapered Member Specification* (on page 2266)

D1.A.5.8 Design of I-Section with Cover Plates

STAAD.Pro can design I-sections with cover plates per AISC 360-05 and AISC 360-10.

According to B4.2(c) (p.16.1-15) of the AISC 360-05 specification, the cover plate is taken as a stiffened element and the width of a flange plate in a built-up section is taken between as the distance between lines of fasteners or welds. For the compression flange, the appropriate classification in Table B4.1 is that of case 12, which explicitly includes "flange cover plates and diaphragm plates between lines of fasteners or welds." The section classification limits for this case are:

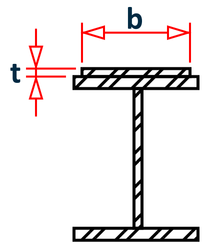
$$\lambda = b/t$$

$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}}$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}}$$

where

b	=	width of the cover plate
t	=	thickness of the cover plate



Similarly, according to B4.2(c) of the AISC 360-10 specification, the cover plate is taken as a stiffened element and the width of the flange plate in a built-up section is taken as the distance between lines of fasteners or welds. For the compression flange, case 7 is used.

$$\lambda = b/t$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}}$$

For flexure, case 18 is used.

$$\lambda = b/t$$

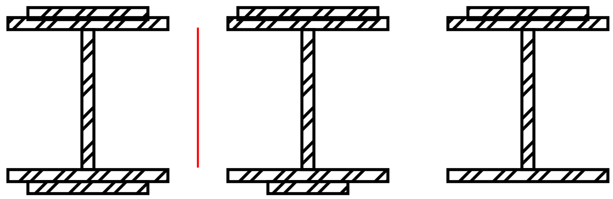
$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}}$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}}$$

The section classification is included in the STAAD.Pro output listed as UNSTIFFENED / STIFFENED elements of the shape, along with the classification limits calculated as above and the element slenderness ratio, λ.

In addition to the limit states described elsewhere in this section, I-sections with cover plates have the following considerations for code checking per AISC 360-05/10:

- For Section E4, the program determines if the section is doubly or singly symmetric. Based on this, E.4-4 or E. 4-5 is used for the compression code check, respectively. An I-section with the same cover plate top and bottom remains doubly symmetric. An I-section with differing top and bottom plates or only a top or a bottom cover plate is singly symmetric.



- For lateral-torsional buckling calculations, the values of r_t and a_w are calculated as described in F4.2 for the effective radius of gyration in section ii.
- For shear, the minimum of all b/t ratios - including the flanges and top and bottom cover plates - is used for the web slenderness in the horizontal direction.
- For shear, the cover plates are checked for tension field action.
- The cover plates are considered in the seismic classification of the section.

It is assumed that the section classifications are for I-section flanges, web, and cover plates but not for parts of elements (i.e., outstanding flange).

Note: The following two items are *not* checked for I-sections with cover plates:

- The effects of the cover plates for flange local buckling of the section. Only the I-section flanges are considered in the FLB calculation.
- AISC Design Guide 9 is not incorporated.

D1.A.6 Design Parameters

Design per AISC 360-05, 360-10, and 360-016 (Unified) specifications is requested by using the CODE parameter. Other applicable parameters are summarized in the following Table. These parameters communicate design decisions from the engineer to the program and thus allow you to control the design process.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: For AISC 360-2016, a maximum of nine (9) design commands may be specified in a parameter block.

Table 78: AISC 360-05, 360-10, and 360-16 Design Parameters

Parameter Name	Default Value	Description

CODE	AISC UNIFIED	Used to designate this code (default is the 2016 edition).
		`CODE AISC UNIFIED (2016)`
		`CODE AISC UNIFIED 2010`
		`CODE AISC UNIFIED 2005`

ALH (AISC 360-16 only)	0.5	Distance of applied point torsion from start of member as a fraction of member length. Represented by " a " in the torsional case options in Appendix B of AISC Design Guide 9. To be used with `TND` values of 3, 6, or 9. (0<ALH<1). Set `TORSION 1` to enable Torsion check as per DG9.
BEAM	1.0	See Note 9 below. 0.0 = design at start and end nodes and those locations specified by the `SECTION` command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes.
BRC (AISC 360-05 and 360-10 only)	1	Specifies the bracing type for the member used for seismic provision checks: 1 = Relative bracing 2 = Nodal bracing
CAN	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see D1.B.1.2 Design Parameters (on page 1134))
CB [2]	1.0	Coefficient C_b per Chapter F. If C_b is set to 0.0, it will be calculated by the program. Any other value will be directly used in the design. See Note 2 below.
CSPACING	12 in	Spacing between connectors in current length units. Refer to Section E6.1 and E6.2 of AISC 360.
DFF	none (mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))

Parameter Name	Default Value	Description
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
DMAX	1000.0 mm	Maximum allowable depth for member selection.
DMIN	0.0 mm	Minimum allowable depth for member selection.
DUCT (AISC 360-16 only)	0	The ductile category of the member as per AISC 341-16: 0 = non-ductile 1 = moderately ductile 2 = highly ductile
E5PROVISION (AISC 360-16 only)	0	Specifies whether the requirements for single angle members in compression in sections E5.(1) to E5.(3) are met for this member. 0) All requirements are satisfied. Thus the modified slenderness approach of E5 is used. 1) One or more are not satisfied See Note 11 below for details.
FLX	1	Parameter for specifying the lateral-torsional restraint condition for a single angle. Refer to Section F10 of AISC 360-05, 360-10, and 3601-16. 1 = Member does not have continuous lateral-torsional restraint along the length. 2 = Member has continuous lateral-torsional restraint along the length. 3 = Lateral-torsional restraint is provided at the point of maximum moment only. **AISC 360-16:** If FLX 2 is used, then any values assigned to parameters UNB, UNL, UNR, or UNT are ignored and zero is used for the member length for lateral torsional buckling.
FRM (AISC 360-05 and 360-10 only)	0	Specifies the seismic force-resisting system used in seismic provision checks: 0 = Ordinary Moment Frame (OMF) 1 = Intermediate Moment Frame (IMF) 2 = Special Moment Frame (SMF)
FU	400 MPa	Ultimate strength of steel. **AISC 360-16:** This value is ignored if SGR specified other than 0. Refer to Note 10 for details.

Parameter Name	Default Value	Description
FYLD	250 MPa	Yield strength of steel. The program considers a valid range of input values between 10 ksi - 100 ksi (69 MPa - 689 MPa). **AISC 360-16:** This value is ignored if SGR specified other than 0. Refer to Note 10 for details.
IMM (AISC 360-16 only)	0	Truss type of single angle members: 0) E5(a): Individual / web member of a planar truss 1) E5(b): Web member of a box or space truss **Note:** For the default case, a single angle is assumed to be an individual or web member of a planar truss [E5(a)] with adjacent web members attached to the same side of the gusset plate or chord. See Note 11 below for additional details.
KX	1.0	K value for flexural-torsional buckling.
INTERACTION	0	Directs the program which interaction equations to check per section H1: 0 = Checks both H1.1 and H1.3 and reports the lower ratio as critical 1 = Check both H1.1 and H1.3 and reports the higher ratio as critical 2 = Always checks per H1.1 even when H1.3 may be applicable 3 = Checks H1.3 in lieu of H1.1 when applicable
KY	1.0	Effective length factor to calculate slenderness ratio for compression buckling about local y-axis. Usually this is the minor axis.
KZ	1.0	Effective length factor to calculate slenderness ratio for compression buckling about local z-axis. Usually this is the major axis.
LBRC (AISC 360-16 only)	1	Type of flange lateral bracing: 0 = none 1 = panel bracing 2 = point bracing 3 = special bracing Used to calculate bracing requirements as per seismic provisions in AISC 341-16.

Parameter Name	Default Value	Description
LEG	0	This parameter is meant for plain angles (Section E5). 0 = The angle is connected by the longer leg. 1 = The angle is connected by the shorter leg.
LX	Member Length	Length for flexural-torsional buckling. See Note 8 below.
LY	Member Length	Length to calculate slenderness ratio for buckling about local y-axis.
LZ	Member Length	Length to calculate slenderness ratio for buckling about local z-axis.
MAIN	200	Allowable slenderness limit for compression members. A value of 1 suppresses this check. Any value greater than 1 is used as the compression slenderness check value.
METHOD	LRFD	Used to specify LRFD or ASD design methods.
MTYP	1	Specifies whether the member is a beam or column. Used for seismic provisions checks. 1 = Beam 2 = Column For AISC 360-16 only: 3 = Beam-column
NBRC (AISC 360-16 only)	1	Number of braced points within the span. Represented by "n" in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions.
NSF	1.0	Net Section Factor for tension members, equal to A_n/A_g ,used to account for reduction in section used for tension checks (clause B 4.3b.) combined with the SLF parameter to determine the rupture strength. (see also SLF parameter)
PROFILE		Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of actual load to allowable strength.

Parameter Name	Default Value	Description

SEISMIC	0	Used to instruct the program to add additional checks per the AISC 341, *Seismic Provisions for Structural Steel Buildings*.
		
		0 = Do not check seismic provisions
		
		AISC 360 – 05, check according to AISC 341 – 05
		
		AISC 360 – 10, check according to AISC 341 – 10
		
		AISC 360 – 16, check according to AISC 341 – 16
		
		See section D1.A. American Codes - Steel Design per AISC 360 Unified Specification for more details.
		
		1 = Check seismic provisions
		
		See D1.A.9 Seismic Provision Checking per AISC 341 (on page 1112) for details.
		
		Note: These additional parameters that should be set when using the seismic option:
		• AISC 360-05 and AISC 360-10
		  • MTYPE
		  • FRM
		  • BRC
		• AISC 360-16
		  • MTYPE
		  • DUCT

<table>
<tr><th>Parameter Name</th><th>Default Value</th><th>Description</th></tr>
<tr><td><u>SGR</u>

(AISC 360-16 only)</td><td></td><td>Select ASTM steel grades:

Custom
1 = A36
2 = A53 Gr.B
3 = A500 Gr.B (HSSRect)
4 = A500 Gr.B (HSSRound)
5 = A500 Gr.C (HSSRect)
6 = A500 Gr.C (HSSRound)
7 = A501 Gr.A
8 = A501 Gr.B
9 = A529 Gr.50
10 = A529 Gr.55
11 = A709 Gr.36
12 = A1043 Gr.36
13 = A1043 Gr.50
14 = A572 Gr.42
15 = A572 Gr.50
16 = A572 Gr.55
17 = A572 Gr.60
18 = A572 Gr.65
19 = A618 Gr.I(a)/Gr.I(b)/Gr.II
20 = A618 Gr.III
21 = A709 Gr.50
22 = A709 Gr.50S
23 = A709 Gr.50W
24 = A913 Gr.50
25 = A913 Gr.60
26 = A913 Gr.65
27 = A913 Gr.70
28 = A992
29 = A588
30 = A847
31 = A1085

The yield stress and ultimate stress will be auto-calculated based on the grade selected. Note that any SGR value greater than 0 will take priority when calculating the yield stress and ultimate stress over any supplied FYLD and FU value.

AISC 360-16: Refer to Note 10 for details.

Note: “HSS Rectangle A1085” and “HSS Round A1085” profiles steel grade will always be considered as A1085 irrespective of any value assigned to SGR.</td></tr>
</table>

Parameter Name	Default Value	Description
SLF	1.0	Shear Lag Factor, value "U" normally taken from table D3.1, combined with the NSF parameter to determine the net effective area used to calculate the section rupture strength. (see also NSF parameter)
SNUG	1	Type of connection for the built-up members: 0 = Welded or pretensioned bolts 1 = Bolted snug-tight
SOE (AISC 360-16 only)	0	Second Order Effects have been considered in analysis forces or not: 0 = Have not been considered 1 = Have been considered By default, Second Order Effects are not considered in the analysis forces. This is related to Torsion checks as per DG9. Set `TORSION 1` to enable Torsion check as per DG9.
SRT (AISC 360-16 only)	0	Used for tension slenderness checking. 0 = Checks slenderness design for tension member in conventional way. 1= Checks slenderness design for tension member as per CL D1. For default case, tension slenderness check is made by dividing the TSL value by Rmin. For SRT value 1, slenderness design for tension member will be done by taking maximum length out of LZ, LY and member length.
STFB (AISC 360-16 only)	0.0	Stiffener width for one-sided web stiffeners, twice the individual stiffener width for pairs of stiffeners. Represented by "bs" in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions.
STFT (AISC 360-16 only)	0.0	Thickness of web stiffeners. Represented by "tst" in Appendix 6.3.2(a) of AISC 360-16. Required for Seismic Provisions.
STIFF	Member Length or depth of beam, whichever is greater	Spacing of stiffeners for plate girder design.

Parameter Name	Default Value	Description
STP	1.0	Section Type used for design 1 = Rolled section 2 = Welded section **Note:** If a UPT Wide flange section with different top & bottom flange dimensions have been specified for a member, the AISC360-16 module will ignore the value of STP and consider the section as a Welded/Built-Up section.
TBRC (AISC 360-16 only)		Type of torsional bracing: 0 = None 1 = continuous bracing 2 = point bracing 3 = special bracing Used to calculate bracing requirements as per seismic provisions in AISC 341-16.
TFA (AISC 360-16 only)	0	Tension field action to be considered in shear design: 0 = do not consider 1 = consider
TSL (AISC 360-16 only)	Member Length	The length used in tension slenderness checks.
TMAIN	300	Allowable slenderness limit for tension members. A value of 1 suppresses this check. Any value greater than 1 is used as the tension slenderness check value.

Parameter Name	Default Value	Description
TND (AISC 360-16 only)	1	Torsion loading and end condition as in Table in Appendix C.4 of AISC Design Guide 9: 1 = Equal concentrated end torques. Both ends free. 2 = Equal concentrated end torques. Both ends fixed. 3 = Concentrated Torque. Both ends pinned. 4 = Uniformly distributed torque. Both ends pinned. 6 = Concentrated Torque. Both ends fixed. 7 = Uniformly distributed torque. Both ends fixed. 9 = Concentrated Torque. One end fixed, another end free. 10 = Partial uniformly distributed torque. One end fixed, another end free. 12 = Uniformly distributed torque. One end fixed, another end pinned. The number corresponds to the case number of the case chart in Appendix B of DG9. Set `TORSION 1` to enable Torsion check as per DG9.
TORSION (AISC 360-10 and 360-16 only)	0	Specifies design for torsion per AISC Design Guide 9. See D1.A.5.6 Design for Torsion (on page 1096) 0 = Do not perform torsion checks 1 = Perform torsion checks **Note:** When torsion checks are performed, `TRACK 3` output may be used to provide detailed torsion design output for Design Guide 9 checks.
TRACK	0	Specifies the amount of detail included in design output 0 = Suppress all member capacities 1 = Print all member capacities 2 = Print full member design details
UNB	Member Length	Unsupported length of the bottom flange for calculating flexural strength. Will be used only if compression is in the bottom flange. See Note 3 below.
UNL	Member Length	Unsupported length of left extreme flange for LTB that will be used as lateral-torsional buckling length for the section with the vertical axis as major principal axis and where the left extreme fiber is in compression. If member is assigned with any value of UNL and FLX=2 concurrently, LTB length of the member will be treated as zero.

Parameter Name	Default Value	Description
UNR	Member Length	Unsupported length of right extreme flange for LTB that will be used as Lateral-torsional buckling length for the section with the vertical axis as major principal axis and where the right extreme fiber is in compression. If member is assigned with any value of UNR and FLX=2 concurrently, LTB length of the member will be treated as zero.
UNT	Member Length	Unsupported length of the top flange for calculating flexural strength. Will be used only if compression is in the top flange. See Note 3 below.
WTYPE (AISC 360-05 and 360-10 only)	0	Weld type for HSS per Sect. B3.12 (AISC 360-05) or Sect. B4.12 (AISC 360-10): 0 = Electric resistance welding 1 = Submerged arc welding For HSS Rectangle and Round profiles from AISC databases, the weld type will always be determined based on profile table (grade A1085 or not). `WTYP` will have no effect on these. For any other hollow profiles (pipe, tube, box, etc.) from AISC databases, all hollow profiles from any other country databases, and User Provided tables, the weld type must be specified using the `WTYP` parameter.

Notes

1. For the AISC 360 unified code, an angle is automatically checked for geometric axis bending (in addition to principal axis bending) provided one of the following conditions is met:
   i. The `FLX` parameter is set to 2 for that member. The section could be an equal or an unequal legged angle.
   ii. The angle is equal-legged, has bending moment only about one of its geometric axes, and is not subjected to axial compression.

   The `AXIS` parameter is only used by the deprecated AISC 360-05 code checking method (`CODE AISC UNIFIED OLD`). If this code is used, then AXIS 1 specifies design based on principle axes, where AXIS 2 specifies design based on geometric axes.
2. Non-default values of `CB` must be re-entered before every subsequent `CHECK CODE` or `SELECT` command.
3. Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if `SET Z UP` is used).
4. For a description of the deflection check parameters `DFF`, `DJ1`, `DJ2` see the Notes section of D1.B.1.2 Design Parameters (on page 1134) of this manual.
5. `NSF` is the Net Section Factor as used in most of the steel design codes in STAAD.Pro. It is defined as the Ratio of “Net cross section area” / “Gross section area” for tension member design. The default value is 1.0. For the AISC 360 code, it is described in section D.3.2.
6. `SLF` is the Shear Lag Factor, as used in Section D.3.3 of the AISC 360-05 code. This factor is used to determine the effective net area by multiplying this factor with net area of the cross section. Please refer to Table D3.1 of the 360 code for a list of acceptable `SLF` values. In STAAD.Pro, the default value for `SLF` is 1.0. The effective net area is used to determine the tensile strength for tensile rupture in the net section, as per equation D.2.2.

7. To summarize, the "Gross Area" (Ag) is multiplied by `NSF` to get the "Net Area" (An) of the section. The "Net Area" (An) is again multiplied by `SLF` to get the "Effective Net Area" (Ae) of the section.
8. For the design of a single angle for flexure, the parameter `LX` should be used to specify the value of the term "L" in equations F10-4a, F10-4b, F10-5 and F10-6 of AISC 360-05 and the term "Lb" in equations F10-4, F10-5, F10-6a, and F10-6b of AISC 360-10.
9. When `BEAM` is 1.0 (default), the design is performed at 13 evenly spaced points along the length of the beam, including start and end points (i.e., 1/12th points or at ends of 12 equal length segments).

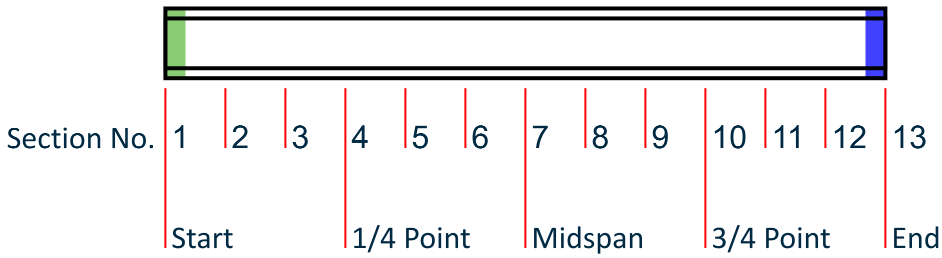


Figure 126: The default sections for design when BEAM 1.0 is used

When `BEAM` is 0.0, the start and ends along with up to three locations specified in TR.41 Section Specification (on page 2665) are designed.

10. For AISC 360-16, the material strength values are first taken from the `SGR` parameter if specified (and not zero). If `SGR` has not been specified, then the values of the `FYLD` and `FU` parameters will be used if specified. If no design parameters have been specified for material strengths, then the values in the material definition are used. If no material definition has been assigned, then the STAAD.Pro default values of Fy = 36 ksi and Fu = 58 ksi are assumed.
11. For AISC 360-16, single angle members in compression can be designed using a modified slenderness method provided a set of conditions are met for that member. It is left to the engineer to confirm that conditions E5.(1) through E5.(3) are met and to specify this for single angle member using the `E5P` parameter. Those conditions are:

    1) Members are loaded at the ends in compression through the same leg.
    2) Members are attached by welding or by connections with a minimum of two bolts.
    3) There are no intermediate transverse loads.

    If one or more of these conditions are not met, then the `E5P 1` parameter value should be assigned to those members.

    Note: Conditions E5.(4) and E5.(5) are checked by the program:

    4) Slenderness ratio, $L_c/r \leq 200$
    5) For unequal leg angles, the ratio of leg widths is < 1.7.

    The effective slenderness ratio method used for single angles also depends on if those angle members are:

    1. individual members or are web members of planar trusses (in which case, Cl. E5(a) applies - `IMM 0`)
    2. web members of box or space trusses (in which case, Cl. E5(b) applies - `IMM 1`)

    A message is issued if the `E5P` or `IMM` parameter is used with any section type other than a single angle.

Note: The `E5P` and `IMM` parameters are only applicable to standard catalog or UPT angles which also are assigned as truss members.

Related Links

- *RR 22.09.00-3.1 Tension Slenderness Checks in AISC 360-16* (on page 96)

D1.A.7 Code Checking and Member Selection

Code Checking and Member Selection options are both available in the AISC 360 Unified Code implementation in STAAD.Pro.

For additional information on code checking, refer to:

- D1.B.1.3 Code Checking (on page 1141) for general information and
- TR.49 Code Checking Specification (on page 2677) for input command details

For addiitional information on member selection refer to:

- D1.B.1.4 Member Selection (on page 1142) for general information and
- TR.49.1 Member Selection Specification (on page 2678) for input command details

D1.A.8 Tabulated Results of Steel Design

Results of Code Checking and Member Selection are presented in the output file. The output is clearly marked for the selected specification (ASIC 360), edition used (2010 or 2005), and the design method (LRFD or ASD).

The following details are presented on Code Checking of any member:

- Result of Code Checking (Pass / Fail) for the member Number.
- Critical Condition which governed the design and the corresponding Ratio and Location.
- Loads corresponding to the Critical Condition at the Critical Location.
- Section Classification
- Slenderness check report
- Section Capacities in Axial Tension, Axial Compression, Bending, and Shear in both the directions.

  If Seismic provisions are to be checked, these follow as described in D1.A.9 Seismic Provision Checking per AISC 341 (on page 1112)

Note: An asterisk following a critical load case number indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2616) for additional information.

D1.A.9 Seismic Provision Checking per AISC 341

Additional parameters may be specified for member checking or design per AISC 360-05, 360-10, or 360-16 to perform seismic provision checks per the corresponding edition of AISC 341.

The `SEISMIC` parameter is used to specify seismic provision checks for members. For those members in the AISC 360-05 or 360-10 editions, the `MTYP` (member type), `FRM` (frame type), and `BRC` (bracing type) parameters are used to specify the structure and member details. For the AISC 360-16 edition, only the `MTYP` (member type) parameter is used to specify the member details. See D1.A.6 Design Parameters (on page 1100) for details on the values used for these parameters.

Provisions Checked

The following checks are performed with the `SEISMIC 1` parameter for the AISC 341-05/10 specifications:

- seismic classification of the member - the “worst case” classification is considered among all elements for the member. For
  - For members in an ordinary moment frame (OMF) or intermediate moment frame (IMF), the general section classification of AISC 360-05/10 is used.
  - For members in a special moment frame (SMF), the classification per table I-8-1 of AISC 341-05 or table D1.1 of AISC 341-10, respectively, are used.
- required flexural strength of the member -
  - OMF - $M_r = M_z$ (moment about major axis), per Sec. 11.8 of AISC 341-05 or Sec. E.1.4 of AISC 341-10
  - IMF - $M_r = M_u = R_y Z F_y$ (LRFD) or $= R_y Z F_y / 1.5$ (ASD) per Sec. 10.8 of AISC 341-05 or Sec. D.1.2a of AISC 341-10
  - SMF - $M_r = M_u = R_y Z F_y$ (LRFD) or $= R_y Z F_y / 1.5$ (ASD) per Sec. 9.8 of AISC 341-05 or Sec. D.1.2b of AISC 341-10
- required bracing strength
  - Relative bracing for beams - $P_{br} = 0.008 M_r C_d / h_0$ per eq. A-6-5 of AISC 360 05/10
  - Nodal bracing for beams - $P_{br} = 0.02 M_r C_d / h_0$ per eq. A-6-7 of AISC 360 05/10
  - Relative bracing for columns - $P_{br} = 0.004 P_r$ per eq. A-6-1 of AISC 360 05/10
  - Nodal bracing for columns - $P_{br} = 0.01 P_r$ per eq. A-6-3 of AISC 360 05/10
- required bracing stiffness: each of the following are multiplied by $1/\phi$ for LRFD or by Ω for ASD:
  - Relative bracing for beams - $\beta_{br} = 4 M_r C_d / (L_b h_0)$ per eq. A-6-6 of AISC 360 05/10
  - Nodal bracing for beams - $\beta_{br} = 10 M_r C_d / (L_b h_0)$ per eq. A-6-8 of AISC 360 05/10
  - Relative bracing for columns - $\beta_{br} = 2 P_r / L_b$ per eq. A-6-2 of AISC 360 05/10
  - Nodal bracing for columns - $\beta_{br} = 8 P_r / L_b$ per eq. A-6-4 of AISC 360 05/10
- required bracing spacing
  - OMF - no requirement
  - IMF - $L_b = 0.17 r_y E / F_y$ per Sec 10.8 of AISC 341-05 or Sec. D.1.2a of AISC 341-10
  - SMF - $L_b = 0.086 r_y E / F_y$ per Sec 9.8 of AISC 341-05 or Sec. D.1.2b of AISC 341-10

Provisions Checked for AISC 341-16

The following checks are performed with the `SEISMIC 1` parameter for the AISC 341-16 specification:

- Section seismic classification against ductility – the section is evaluated to be seismically compact or seismically non-compact for either highly or moderately ductile members (specified using the `DUCT` parameter)
- Required flexural strength of member checked per D1-1
- Stability bracings of beam
  - Maximum allowed spacing of bracings checked per D1-2
  - Required strength of lateral beam bracing at panel type flange lateral bracing (A-6-5 of AISC 360-16), point type flange lateral bracing (A-6-7 of AISC 360-16), and special flange lateral bracing at plastic hinge locations (D1-4)
  - Required strength of torsional bream bracing at point torsional bracing (A-6-9 of AISC 360-16), continuous torsional bracing, and special torsional bracing at plastic hinge locations (D1-5)

  - Required stiffness of beam bracing at panel type flange lateral bracing and point type flange lateral bracing (A-6-6 of AISC 360-16)
  - Required stiffness of torsional bracing at point torsional bracing (A-6-10 and A-6-11 of AISC 360-16) and continuous torsional bracing (A-6-13 of AISC 360-16)
  - Special bracing at plastic hinge locations (D1-6)
- Stability bracings of column
  - Required strength of column bracing at panel type bracing (A-6-1 of AISC 360-16) and point lateral bracing (A-6-3 of AISC 360-16)
  - Required stiffness of column bracing at panel type bracing and point lateral bracing (A-6-2 of AISC 360-16)
- Stability bracing of beam-column
  - Required strength of lateral beam bracing at panel type flange lateral bracing and point type flange lateral bracing (A-6-4 of AISC 360-16)
  - Combination of lateral and torsional bracing

Output

The output for the AISC 341-05/10/16 checks follow the checks for AISC 360-05/10/16 for a member. No seismic provision checks are printed for `TRACK 0` output. For `TRACK 1` output, the seismic classification and the requirements are given. `TRACK 2` adds the details of the classification limits.

Track 1 example for AISC 341-10:

```
|-----------------------------------------------------------------------|
| SEISMIC PROVISION(AISC 341:10):-                                      |
|                                                                       |
| MOMENT FRAME TYPE:  IMF                                               |
|                                                                       |
| SEISMIC CLASSIFICATION:  FLANGE:  Compact           WEB:  Non-Compact |
|                                                                       |
| REQUIREMENTS:-                      VALUE         CRITERIA            |
| EXPECTED FLEXURAL STRENGTH        5.05E+02        Sec. D.1.2a.        |
| BRACING STRENGTH                  2.78E+00        Eq. A-6-5           |
| BRACING STIFFNESS                 2.67E+06        Eq. A-6-6           |
| BRACING SPACING                   1.36E+01        Sec. D.1.2a.        |
|-----------------------------------------------------------------------|
```

Track 2 example for AISC 341-10:

```
|-----------------------------------------------------------------------|
| SEISMIC PROVISION(AISC 341:10):-                                      |
|                                                                       |
| MOMENT FRAME TYPE:  IMF                                               |
|                                                                       |
| SEISMIC CLASSIFICATION:    CLASS:            l :         l md:       |
| FLANGE:                    Compact          6.575       9.152         |
| WEB:                       Compact         47.465      35.884         |
|                                                                       |
| REQUIREMENTS:-                      VALUE         CRITERIA            |
| EXPECTED FLEXURAL STRENGTH        5.05E+02        Sec. D.1.2a.        |
| BRACING STRENGTH                  2.78E+00        Eq. A-6-5           |
| BRACING STIFFNESS                 2.67E+06        Eq. A-6-6           |
| BRACING SPACING                   1.36E+01        Sec. D.1.2a.        |
|-----------------------------------------------------------------------|
```

Track 2 example for AISC 341-16:

```
|-------------------------------------------------------------------------------|
|                        CHECKS FOR 341-16 SEISMIC CRITERIA                     |
|-------------------------------------------------------------------------------|
| FLEXURAL STRENGTH X                                                           |
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC       |
|            -183.8       280.6       0.655     Eq.D1-1        4     17.50     |
|                                                                               |
| Intermediate Results :                                                        |
|  Overstrength factor       : Ry     =  1.0000                TableA3.1        |
|  Allow. Brac Spc           : Lb     =  15.167     ft         Eq.D1-2          |
|-------------------------------------------------------------------------------|
| FLEXURAL STRENGTH Y                                                           |
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC       |
|             0.000       46.11       0.000     Eq.D1-1        4      0.00     |
|                                                                               |
| Intermediate Results :                                                        |
|  Overstrength factor       : Ry     =  1.0000                TableA3.1        |
|  Allow. Brac Spc           : Lb     =  15.167     ft         Eq.D1-2          |
|-------------------------------------------------------------------------------|
```

D1.A.10 AISC 360-16 Design

The checking or selection of members is performed per the following sections of AISC 360-16 and AISC 341-16:

- Section Classification: Chapter B
- Tension Chapter D
- Compression: Chapter E
- Flexure: Chapter F
- Shear: Chapter G
- Torsion (Including checks as per DG-9 for certain profile shapes – see below): Chapter H & Design Guide 9 for Torsion design
- Combined actions – Chapter H for:
  - Axial and/or
  - Flexure and/or
  - Shear and/or
  - Torsion
- Checks for Seismic suitability – AISC 341-16

The following section profile shapes are allowed for design:

- Rolled Shapes
  - I-Shape
  - Channel
  - Single angle
  - Solid Circular bar
  - Solid Rectangular bar
  - Tee
  - HSS rectangular
  - HSS round

- Built Up Shapes
  - Built Up Channel
  - Built Up I
  - Double Channels – back to back
  - Double Channels – face to face
  - Double Angles
  - Built Up Box
  - I With Cover Plates (Top and/or bottom)
- Tapered Shapes
  - I with web tapered
  - Square hollow shape with wall tapered
  - Circular hollow shape diameter tapered

Note: Certain checks (e.g., DG-9 torsion checks) will only be performed for certain profile shapes (non-HSS shapes).

Seismic Design per AISC 341-16

Refer to Provisions Checked for AISC 341-16 (on page 1113) for additional details.

D1.A.10.1 Tension

Design of the members subject to axial tension is performed per chapter D of AISC 360-16.

All the cross-section profiles described in section 1A.10 are considered for tension checks.

Eye bars and pin-connected members are *not* checked per clauses D5 and D6.

Limit States

The tensile strength is calculated with due consideration to both tensile yielding and tensile rupture effects as per D.2 of the code.

The F_y and F_u values would be chosen based on the grade of steel associated with the member, specified by `SGR` parameter. Note that the implementation also provides a means to create custom steel grades. If `SGR` is set to `0` ("custom steel"), then the values assigned for `FYLD` and `FU` is used respectively.

Slenderness Limits

The code does not mandate the maximum slenderness limit for members in tension. However, the program will check against a maximum slenderness ratio (L/r) which is specified using `TMAIN` parameter. The length considered to calculate the tension slenderness will be the full member length.

Effective Net Area

The gross area, A_g , and net area, A_n , of tension members shall be determined in accordance with the provisions of Section B4.3 of AISC 360-16.

The effective net area is the net area multiplied by the parameter `SLF` (shear lag factor).

The net area is the gross area multiplied by the parameter `NSF` (net section factor).

D1.A.10.2 Compression

Design of the members subject to compression is performed per chapter E of AISC 360-16.

All the cross-section profiles described in section 1A.10 are considered for compression.

The design compressive strength is calculated based on the applicable limit states of flexural buckling, torsional buckling, and flexural-torsional buckling modes as per Cl.E.1.

The nominal compressive strength, P_n , shall be the lowest value obtained based on the applicable limit states of flexural buckling, torsional buckling, and flexural-torsional buckling. In general, Table.E1.1 is followed for the choice of applicable sections/clauses for the various cross section shapes.

Limit States

Following limit states are checked for flexure per Equations E3-1, E3-2, and E3-3 for without slender elements:

- Flexural buckling about X axis
- Flexural buckling about Y axis
- Flexural buckling about U axis (for unsymmetrical shapes only)
- Flexural buckling about V axis (for unsymmetrical shapes only)
- Flexural-torsional buckling

Flexural-torsional buckling is checked per Equations E4-1, E4-2, and E4-3, E4-4, E4-7. Note the following:

- Check no performed for solid rect or circ bars, thin-walled rect or circ hollow sections, double-channel face-to-face (only when channels are connected by full weld or tension bolts), or single angles with compact long legs.

The gross area is used for members with non-slender elements whereas the effective area, Ae, is used for slender elements. The effective width of a slender element of any section other than round HSS is calculated for each limit state of compression as per eq. E7-2 or eq. E7-3 depending on the condition mentioned in the specification. The effective area (A_e) for a slender HSS round section is calculated as per cl. E7.2.

Parameters

The design parameters that affect compression are:

- `LEG`
- `LZ`
- `LY`
- `LX`
- `KZ`
- `KY`
- `KZ`
- `SNU`
- `CSP`

Slenderness Limits

The code does not mandate the maximum slenderness limit for members in compression. However, the program will check against a maximum slenderness ratio (L/r) which is specified using `MAIN` parameter whose default value is 180. The length considered to calculate the compression slenderness will be the full member length.

D1.A.10.3 Flexure

Design strength in flexure is calculated as per clause F1. The

Limit States

The nominal flexural strength is calculated for different limit states as per sections F2 through F13.

The following limit states are checked for flexure:

- Flexural Yielding about X and Y axis
- Flexural Yielding about U and V axis (for single angle only)
- Lateral-torsional Buckling about X and Y axis
- Lateral-torsional Buckling about U axis (for single angle only)
- Flange Local Buckling
- Web Local Buckling
- Wall Local Buckling (for HSS Round only)
- Compression Flange Yielding
- Tension Flange Yielding

Parameters

The design parameters that affect flexure are:

- `UNT`
- `UNB`
- `UNL`
- `UNR`
- `FLX`
- `SNU`
- `CB`

D1.A.10.4 Shear

Design of the members subject to shear is performed per chapter G of AISC 360-16.

All the cross-section profiles described in section 1A.10 are considered for shear.

An option to consider tension field action is available. Also, the option to consider the shear capacity of I sections or channel shapes with transverse web stiffeners.

The program does not consider any specific checks for web openings in clause G7.

The design of transverse web stiffeners is not performed.

Limit States

The design shear strength is calculated per G1.

The nominal shear strength is calculated for the following sections:

- I-Shapes and Channels G2
- Single Angles and Tees G3
- Rect HSS, Box Sections, and other Doubly Symmetric Members G4
- Round HSS G5
- Shear capacity along weak axis in Doubly Symmetric and Singly Symmetric Shapes G6

D1.A.10.5 Combined Forces and Torsion

The effects of combined actions of axial force with major and minor axis moments are accounted for the following:

Doubly and Singly symmetric members subject to flexure and compression - H1.1

- Rolled I
- Rolled Channel
- Solid circular bar
- Solid rectangular bar
- Tee
- HSS Rectangle
- HSS Round
- Built Up Channel
- Built Up I
- Double Channels – back to back
- Double Angles
- Built Up Box
- I Profile With Cover Plates (Top and/or bottom)

Doubly and Singly symmetric members subject to flexure and tension - H1.2

- Rolled I
- Rolled Channel
- Solid circular bar
- Solid rectangular bar
- Tee
- HSS Rectangle
- HSS Round
- Built Up Channel
- Built Up I
- Double Channels – back to back
- Double Angles
- Built Up Box
- I Profile With Cover Plates (Top and/or bottom)

Doubly symmetric rolled compact members subject to single axis flexure and compression – H1.3

If you prefer to use only cl. H1.1 even when cl. H1.3 is applicable, this may be done with the `INT` parameter.

Unsymmetrical and other members subject to Flexure and Axial Force - H2

This section is used to check single angles.

Members Subject to Torsion and combined Torsion, Flexure, Shear, and Axial Force - H3

Checks for members subject to pure torsion or subject to torsion combined with other effects such as flexure or axial loads will be performed based on the profile shape of the member. Only the following section profiles are considered when performing torsion checks:

- Closed profiles
    - HSS Rectangle

- HSS Round
- Solid Circular bar
- Solid Rectangular bar
- Open profiles
  - Rolled I
  - Rolled Channel
  - Tee
  - Single angle
  - Built-up I
  - Built-up Channel (Only if symmetric about X-X)
  - I Profile With Cover Plates (Top and/or bottom)

The implementation will perform the AISC 360-16 code checks as per clauses H.3.1 and H.3.2 for circular & rectangular hollow shapes. For all other shapes listed above, the component will follow the design methods as per AISC *Design Guide -9: Torsional Analysis of Structural Steel Members* (DG-9).

For non-HSS members subject to torsion and combined stress, the program cannot detect the load-end restraint condition automatically from the model. You can specify this through TND parameter. Also, for some cases the distance where the torque is applied must be provided through the ALH parameter.

Related Links
RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

D1.A.10.6 Updates to AISC 360-16

This section highlights the changes implemented from AISC 360-16 from AISC 360-10.

Section Classification

The width to thickness ratio limits for members in flexure have been updated for the following element types as per AISC 360-16:

- Stems of tees (14) are considered slender when $d / t > 1.52\sqrt{\frac{E}{F_y}}$
- Flanges of box sections (21) are considered slender when $b / t > 1.49\sqrt{\frac{E}{F_y}}$

CHAPTER E Compression Checks: Torsional & Flexural-Torsional Buckling Calculations

Design for the following profile shapes/conditions have been updated to use the corresponding clauses/ equations from AISC 360-16:

<u>2.a Cl. E4: Non-Slender Double Angles & Tee Shapes</u>

- Calculation of Fez and Fey to use (E4-9) and (E4-8).
- Calculation of Fe to use (E4-3).
- Calculation of Fcr to use (E3-2) or (E3-3).
- Calculation of Pn to use (E4-1).

<u>2.b Cl. E5: Single-Angle Compression Members</u>

- ·Flexural torsional buckling limit for single angles changed to $b / t \leq 0.71\sqrt{\frac{E}{F_y}}$.

2.c Cl. E7: Compression Members with Slender Elements

- Calculation of nominal compressive strength and critical stress F_{cr} updated to follow the updated criteria from Clause E7.

CHAPTER F Flexure Checks

Design for Flexural members has been updated to use the following:
3.a Cl.F4 : Non-Standard(other) I Shaped Members

Lateral Torsional Buckling:

The calculation of effective radius of gyration, r_t , updated to use the AISC 360-16 equation F4-11 for I-shapes with a rectangular compression flange:

$$r_t = \frac{b_{fc}}{\sqrt{12\left(1 + \frac{1}{6}a_w\right)}} \tag{F4-11}$$

3.b Cl.4 : Tension Flange Yielding

The tension flange yielding checks have been updated to use the updated web plastification factor calculations as per equations F4-16a to F4-17.

1. When $I_{yc}/I_y > 0.23$
   i. When $\frac{h_c}{t_w} \le \lambda_{pw}$

   $$R_{pt} = \frac{M_p}{M_{yt}} \tag{F4-16a}$$

   ii. When $\frac{h_c}{t_w} > \lambda_{pw}$

   $$R_{pt} = \left[\frac{M_p}{M_{yt}} - \left(\frac{M_p}{M_{yt}} - 1\right)\left(\frac{\lambda - \lambda_{pw}}{\lambda_{rw} - \lambda_{pw}}\right)\right] \le \frac{M_p}{M_{yt}} \tag{F4-16b}$$

2. When $I_{yc}/I_y \le 0.23$

   $$R_{pt} = 1.0 \tag{F4-17}$$

3.c Cl.F7 : Flexural checks for Square and Rectangular HSS & Box Sections

- Cl.F7.2: Flange local buckling checks updated to use F7-4 & F7-5 of AISC 360-16 to calculate the effective width of slender elements.
  - For HSS

  $$b_e = 1.92t_f\sqrt{\frac{E}{F_y}}\left(1 - \frac{0.38}{b/t_f}\sqrt{\frac{E}{F_y}}\right) \le b \tag{F7-4}$$

  - For box sections

  $$b_e = 1.92t_f\sqrt{\frac{E}{F_y}}\left(1 - \frac{0.34}{b/t_f}\sqrt{\frac{E}{F_y}}\right) \le b \tag{F7-5}$$

- Cl.F7.3: Web local buckling checks updated to use F7-6 through to F7-9 of AISC 360-16.
  - For sections with noncompact webs

$$M_n = M_p - (M_p - F_y S)\left(0.305\frac{h}{t_w}\sqrt{\frac{F_y}{E}} - 0.738\right) \le M_p \quad \text{(F7-6)}$$

where

h = depth of web, as defined in Section B4.1b, in. (mm)

- For sections with slender webs
  - compression flange yielding

    $$M_n = R_{pg} F_y S \quad \text{(F7-7)}$$

  - compression flange local buckling

    $$M_n = R_{pg} F_{cr} S_{xc} \quad \text{(F7-8)}$$

    and

    $$F_{cr} = \frac{0.9 E k_c}{\left(\frac{b}{t_f}\right)^2} \quad \text{(F7-9)}$$

    where

    R_{pg} = defined by Equation F5-6 with $a_w = 2ht_w/(bt_f)$
    k_c = 4.0
- Cl.F7.4: Lateral torsional buckling checks updated to use F7-6 through to F7-13 of AISC 360-16.

<u>3.d Cl.F9 : Flexural checks for Tees and Double Angles</u>

- Yielding checks for Tee sections updated to use Eqn F9-4 of AISC 360-16.
- Yielding checks for Double Angle sections updated to use Eqn F9-5 of AISC 360-16.

  $$M_p = 1.5 M_y \quad \text{(F9-5)}$$
- Lateral torsional buckling checks updated to use Eqns F9-6 through to F9-13 of AISC 360-16
- Flange Local Buckling calculations of Tees and double angle legs updated to include criteria as per CL. F9.3 (b) of AISC 360-16
- Web local buckling checks for Stem of Tee sections updated to use Cl F9.4 - Eqns F9-17 through to F9-19 of AISC 360-16.
- Web local buckling checks for legs of double angle sections updated to use Cl F9.4 (b) of AISC 360-16.

<u>3.e Cl.F10 : Flexural checks for Single Angles</u>

- Lateral Torsional Buckling checks updated to use Eqns F10-4 through to F10-5b of AISC 360-16.

CHAPTER G Shear Checks

4.a Cl.G2: I Shapes and Channels

- Calculation of Shear strength coefficient Cv updated to use Eqns G2-2 to G2-4 of AISC 360-16
- Calculation of Shear buckling coefficient kv updated to use Eqns from section G2 of AISC 360-16.
- Criteria for consideration of Tension field action updated as per Section G2.2 of AISC 360-16.
- Shear strength calculations for cases with tension field action updated to use Eqns G2-6 through to G2-11 of AISC 360-16

D1.A.10.7 Output

Preceding the member output will be a series of design statements regarding axis conventions, nomenclature, notes, and abbreviations.

TRACK 0 Output

This format reports the design summary for a member indicating the pass/fail status, the critical design ratio, the most critical load case ID, and the code clause that produced the critical design ratio.

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                     *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W18X50          (AISC SECTIONS)
|
|  Status:        PASS       Ratio:         0.703       Loadcase:        3
|
|  Location:     17.50       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -266.4
|
|-----------------------------------------------------------------------------
|
```

TRACK 1 Output

This format produces a more detailed report for the member. In addition to the items from a `TRACK 0` output, this will report a summary of the checks for each of the individual effects considered during design. This will also provide the section properties and design criteria that were used for the design process. A summary of AISC341-16 checks, if performed, will also be included with the `TRACK 1` output. Refer to Provisions Checked for AISC 341-16 (on page 1113) for additional details.

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                     *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W18X50          (AISC SECTIONS)
|
|  Status:        PASS       Ratio:         0.703       Loadcase:        3
|
|  Location:     17.50       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -266.4
|
|-----------------------------------------------------------------------------
|
| SLENDERNESS
|
| Actual Slenderness Ratio    :    254.294
|
```

```
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.703(PASS)
|
|          Loc  :   17.50            Condition :    Cl.F2.1
|
|----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
|
| Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
|
| Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
|
| Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
|
| Cw  :   3.046E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   420.00)
|
|                            l         lp        lr       CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         47.49       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   420.00)
|
|                            l         lp        lr       CASE
|
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         47.49      90.55    137.27     Table.4.1b.Case15
|
```

```
|--------------------------------------------------------------------------------
|
```

TRACK 2 Output

This format expands on the `TRACK 1` format and will provide the final capacities along with the primary intermediate values that were considered during the design for each effect considered. This will also provide the section properties and design criteria that were used for the design process.

```
       STAAD SPACE                                              -- PAGE
NO.    4

                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                     *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     1
|--------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W18X50         (AISC SECTIONS)
|
|  Status:        PASS       Ratio:         0.703       Loadcase:        3
|
|  Location:     17.50       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -266.4
|
|--------------------------------------------------------------------------------
|
| SLENDERNESS
|
| Actual Slenderness Ratio    :    254.294
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|--------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3            Ratio     :        0.703(PASS)
|
|          Loc  :   17.50           Condition :    Cl.F2.1
|
|--------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
|
| Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
|
| Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
|
| Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
|
```

```
| Cw  :   3.046E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   420.00)
|
|                            l         lp        lr       CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         47.49       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   420.00)
|
|                            l         lp        lr       CASE
|  |
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         47.49      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                              -- PAGE NO.    5

                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                   *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       661.5       0.000     Cl.D2          3      0.00
|
```

```
|
| Intermediate Results :
|  Nom. Ten. Yld Cap          : Pn     =  735.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              0.000       716.6       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     = 0.10208     ft2        Eq.D3-1
|  Nom. Ten. Rpt Cap          : Pn     =  955.50     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              0.000       498.5       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  56.933                Cl.E2
|  Elastic Buckling Stress   : Fex    =  12716.     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  5680.7     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  553.88     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              0.000       51.36       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
```

```
|
| Effective Slenderness     : Lcy/ry =  254.29               Cl.E2
|
| Elastic Buckling Stress   : Fey    =  637.37     kip/ft2    Eq.E3-4
|
| Crit. Buckling Stress     : Fcry   =  558.97     kip/ft2    Eq.E3-3
|
| Nom. Flexural Buckling    : Pny    =  57.061     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEX-TOR-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       259.3       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
| Elastic F-T-B Stress      : Fe     =  3217.8     kip/ft2    Eq.E4-2
|
| Crit. F-T-B Stress        : Fcr    =  2822.3     kip/ft2    Eq.E3-2
|
| Nom. Flex-tor Buckling    : Pn     =  288.11     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|

   STAAD SPACE                                              -- PAGE NO.    6

                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                     *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       230.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
| Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
| Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
```

```
|  Nom. Shear Along X        : Vnx    =  256.50     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             30.45       191.7       0.159     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  191.70     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|

    STAAD SPACE                                               -- PAGE NO.    7

                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                        *****************************************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEX YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -266.4       378.8       0.703     Cl.F2.1        3     17.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  420.83     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEX. YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       62.25       0.000     Cl.F6.1        3      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  69.167     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|

    STAAD SPACE                                               -- PAGE NO.    8

                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.0)
                    *****************************************************

ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| CLAUSE H1
|
|                       RATIO          CRITERIA           L/C      LOC
|
|                           0.703      Eq.H1-1b            3       17.50
|
|
|
| Intermediate Results :
|
|  Modification Factor       : Cb     =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  661.50     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  378.75     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  62.250     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

D1.B. American Codes - Steel Design per AISC 9th Edition

D1.B.1 Working Stress Design

D1.B.1.1 Allowables per AISC Code

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the American Institute of Steel Construction (AISC) Code. The ninth edition of the AISC Code, as published in 1989, is used as the basis of this design (except for tension stress). Because of the size and complexity of the AISC codes, it would not be practical to describe every aspect of the steel design in this manual. Instead, a brief description of some of the major allowable stresses are described herein.

Related Links

- *V. AISC ASD - Column Compression Capacity 1* (on page 5700)
- *V. AISC ASD - Column Compression Capacity 2* (on page 5703)
- *V. AISC ASD - Column Compression Capacity 3* (on page 5705)
- *V. AISC ASD - Square Tube Compression Capacity* (on page 5708)
- *V. AISC ASD - Rectangular Tube Compression Capacity* (on page 5710)
- *V. AISC ASD - Tee Compression Capacity* (on page 5713)
- *V. AISC ASD - Beam Load Capacity 3* (on page 5715)
- *V. AISC ASD - Wide Flange Beam Load Capacity 1* (on page 5717)
- *V. AISC ASD - Wide Flange Beam Load Capacity 2* (on page 5720)
- *V. AISC ASD - MC Beam Load Capacity* (on page 5722)
- *V. AISC ASD - Wide Flange Beam Load Capacity 3* (on page 5725)
- *V. AISC ASD - Select Wide Flange Beam 1* (on page 5727)
- *V. AISC ASD - Select Wide Flange Beam 2* (on page 5729)
- *V. AISC ASD - Select Wide Flange Beam 3* (on page 5732)
- *V. AISC ASD - Compression and Biaxial Bending* (on page 5734)
- *V. AISC ASD - Angle in Compression* (on page 5737)
- *V. AISC ASD - 2D Frame Validation* (on page 5739)

D1.B.1.1.1 Tension Stress

Allowable tensile stress on the net section is calculated as;

$$F_t = 0.60\ F_y$$

D1.B.1.1.2 Shear Stress

Allowable shear stress on the gross section,

$$F_v = 0.4\ F_y$$

D1.B.1.1.3 Stress Due To Compression

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula E-1 in the AISC Code, when the largest effective slenderness ratio (Kl/r) is less than C_c . If Kl/r exceeds C_c , allowable compressive stress is decreased as per formula 1E2-2 of the Code.

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

D1.B.1.1.4 Bending Stress

Allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Section 1.5.1.4 is:

$$F_b = 0.66F_y$$

If meeting the requirements of this section of:

a. $b_f/2t_f \le 65/\surd(F_y)$
b. $b_f/t_f \le 190/\surd(F_y)$
c. $d/t \le 640(1 - 3.74\ (f_a/F_y))/\surd(F_y)$ when $(f_a/F_y) < 0.16$, or than $257/\surd(F_y)$ if $(f_a/F_y) > 0.16$
d. The laterally unsupported length shall not exceed $76.0\ b_f/F_y$ (except for pipes or tubes), nor $20{,}000/(d\ F_y/A_f)$
e. The diameter:thickness ratio of pipes shall not exceed $3{,}300/\surd(F_y)$

If for these symmetrical members, $b_f/2t_f$ exceeds $65/\surd(F_y)$, but is less than $95/\surd(F_y)$, $F_b = F_v(0.79 - 0.002(b_f/2t_f)\surd(F_y)$

For other symmetrical members which do not meet the above, Fb is calculated as the larger value computed as per AISC formulas F1-6 or F1-7 and F1-8 as applicable, but not more than 0.60Fy. An unstiffened member subject to axial compression or compression due to bending is considered fully effective when the width-thickness ratio is not greater than the following:

- $76.0/\surd(F_y)$, for single angles or double angles with separators
- $95.0/\surd(F_y)$, for double angles in contact
- $127.0/\surd(F_y)$, for stems of tees

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by B5 of the AISC code.

Tension and compression for the double symmetric (I & H) sections with $b_f/2t_f$ less than $65/\surd(F_y)$ and bent about their minor axis, $F_b = 0.75\ F_y$. If $b_f/2t_f$ exceeds $65/\surd(F_y)$, but is less than $95/\surd(F_y)$, $F_b = F_y(1.075 - 0.005(b_f/2t_f)\surd(F_y)$.

For tubes, meeting the subparagraphs b and c of this Section, bent about the minor axis, $F_b = 0.66F_y$; failing the subparagraphs B and C but with a width:thickness ration less than $238/\surd(F_y)$, $F_b = 0.6F_y$.

D1.B.1.1.5 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy AISC formula H1-1 and H1-2 when fa/Fa is greater than 0.15, otherwise formula H1-3 is used. It should be noted that during code checking or member selection, if fa/Fa exceeds unity, the program does not compute the second and third part of the formula H1-1, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for sidesway and $0.6 - 0.4\ (M1/M2)$, but not less than 0.4 for no sidesway.

D1.B.1.1.6 Singly Symmetric Sections

For double angles and Tees which have only one axis of symmetry, the KL/r ratio about the local Y-Y axis is determined using the clauses specified on page 3-53 of the AISC ASD 9th ed. Manual.

D1.B.1.1.7 Torsion per Publication T114

The AISC 89 code of specifications for steel design currently does not have any provisions specifically meant for design of sections for Torsion. However, AISC has published a separate document called *Torsional Analysis of Steel Members* which provides guidelines on transforming torsional moments into normal stresses and shear stresses which can then be incorporated into the interaction equations explained in Chapter H of the AISC 89 code. The guidelines of the publication have now been incorporated into the AISC-89 steel design modules of STAAD.

To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter `TORSION` with a value of 1.0. See D1.B.1.2 Design Parameters (on page 1134) for more details.

Methodology

If the user were to request design for torsion, the torsional properties required for calculating the warping normal stresses, warping shear stresses and pure shear stresses are first determined. These depend of the "boundary" conditions that prevail at the ends of the member. These boundary conditions are defined as "Free", "Pinned" or "Fixed". They are explained below:

Free represents the boundary condition such as that which exists at the free end of a cantilever beam. It means that there is no other member connected to the beam at that point.

Pinned represents the condition that corresponds to either a pinned support defined at the joint through the Support command or a release of any of the moments at the joint through a Member Release Specification.

Fixed represents the condition where a fixed support exists at the joint. In the absence of a support at that joint, it represents a condition where a rigid frame connection exists between the given member and at least one other member connected to that joint. Also, no member releases should be present at that joint on the given member.

After the boundary conditions are determined, the normal and shear stresses are determined. The guidelines specified in the publication T114 for concentrated torsional moments acting at the ends of the member are used to determine these stresses.

The warping normal stresses are added to the axial stresses caused by axial load. These are then substituted into the interaction equations in Chapter H of the AISC 89 code for determining the ratio. The plane shear and warping shear stresses are added to the shear stresses causes by actual shear forces and compared against the allowable shear stresses on the cross section.

Torsional boundary conditions at a joint where a FIXED BUT type of support is specified

If the end of a the member is declared a `FIXED BUT` type of support, the torsional boundary conditions at that end are determined in the following manner.

If the member framing into that support does not have any "member releases" specified at that node, then,

a. If all of the 3 translational degrees of freedom at that support are either free to displace, or have a spring, then, that end of the member is considered torsionally `FREE`.

Example:

```
45 FIXED BUT MX MY MZ KFX 75 KFY 115
```

In this example, at joint 45, a spring has been specified along KFX and KFY, and, no restraint is provided for translation along global Z. So, the member which has joint 45 as one of its nodes is considered torsionally free at joint 45.

b. If any of the 3 translational degrees of freedom at that support are restrained, and, any of the moment degrees of freedom are unrestrained or have a spring, then, that end of the member is considered torsionally PINNED.

Examples:

```
78 FIXED BUT FX MZ
```

In this example, joint 78 is prevented from translation along global Y and Z, and free to rotate about global Z. So, the member which has joint 78 as one of its nodes is considered torsionally PINNED at joint 78.

```
17 FIXED BUT MX MY
```

In this example, joint 17 is prevented from translation along global X, Y and Z, and free to rotate about global X and Y. So, the member which has joint 17 as one of its nodes is considered torsionally PINNED at joint 17.

```
85 FIXED BUT FZ MZ KFY 1.0E8 KMX 1.6E6
```

In this example, the joint is prevented from translation along global X, has a rotational spring for resisting moments about global X and is free to rotate about global Z. So, the member which has joint 85 as one of its nodes is considered torsionally PINNED at joint 85.

Restrictions

This facility is currently available for Wide Flange shapes (W, M & S), Channels, Tee shapes, Pipes and Tubes. It is not available for Single Angles, Double Angles, members with the `PRISMATIC` property specification, Composite sections (Wide Flanges with concrete slabs or plates on top), or Double Channels. Also, the stresses are calculated based on the rules for concentrated torsional moments acting at the ends of the member.

D1.B.1.1.8 Design of Web Tapered Sections

Appendix F of AISC-89 provides specifications for design of Web-Tapered members. These specifications have been incorporated into STAAD to perform code checking on web tapered wide flange shapes. Please note that member selection cannot be performed on web-tapered members.
D1.B.1.1.9 Slender Compression Elements
For cross sections with elements which fall in the category of slender as per Table B5.1 of the AISC ASD code (the others being compact and non-compact), the rules of Appendix B of the code have been implemented. For stiffened compression elements, the effective cross section properties are calculated and used. For unstiffened compression elements, the allowable stresses are reduced per the Appendix.

D1.B.1.2 Design Parameters

The program contains a large number of parameter names which are needed to perform designing and code checking. These parameter names, with their default values, are listed in the following table. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the `KZ` (k value in local z-axis) value of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instructions. Similarly, the `TRACK` value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the `TRACK` value must be set to 1.0.

The parameters `PROFILE`, `DMAX`, and `DMIN` are only used for member selection.

Table 79: AISC (9th Ed.) Design Parameters

Parameter Name	Default Value	Description
AXIS	1	Select axis about which single angles are design 1) Design single angles for bending about their principle axis. 2) Design single angles for bending about their geometric axis.
BEAM	1.0	Used to specify the number of sections at which the member design is evaluated. 0.0 = design at start and end nodes and those locations specified by the `SECTION` command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. **Note:** See D1.A.6 Design Parameters (on page 1100).
BMAX	83.3333 ft	Maximum allowable width of the flange. Used in the design of tapered sections.

Parameter Name	Default Value	Description
CAN	0	Specifies the method used for deflection checks 0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see note 1)
CB	1.0	Cb value as used in Section 1.5 of AISC. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.
CDIA	0.0	The diameter of circular openings. If a member has more than one circular opening, they can have different diameters.
CHOLE	NONE	Section locations of circular openings along the length of the member. Maximum three locations can be specified for each member when there is no rectangular opening.
CMP	0	Composite action with connectors 0) design as non-composite beam 1) design as a composite beam if the slab is in bending compression throughout the span, design as a non-composite beam if the slab is in tension anywhere along the span 2) design as a composite beam only. Ignore moments which cause tension in the slab.
CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y and z axes, respectively.
CYC	500,000	Cycles of maximum stress to which the shear connectors are subject.
DFF	none (mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DIA	0.625 in.	Diameter of the shear connectors
DINC	1 in	Incremental depth value used in the design of tapered sections.
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (see note 1)
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see note 1)

Parameter Name	Default Value	Description
DMAX	1000 in.	Maximum allowable section depth.
DMIN	0.0 in.	Minimum allowable section depth.
DR1	0.4	Ratio of moment due to dead load applied before concrete hardens to total moment.
DR2	0.4	Ratio of moment due to dead load applied after concrete hardens to total moment.
ELECTRODE	1	Weld material to be used for reinforced opening. 0) E60XX 1) E70XX 2) E80XX 3) E90XX 4) E100XX 5) E110XX
FBINC	0	Incremental bottom flange width used in the design of tapered sections. In this case, the top flange width will remain unchanged.
FLX	1	Single angle member bracing 1) Single angle member is *not* fully braced against lateral torsional buckling. 2) Single angle member is fully braced against lateral torsional buckling. 3) Single angle member is braced against lateral torsional buckling at the point of maximum moment.
FPC	3.0 ksi	Compressive strength of concrete at 28 days
FSS	1	Is the full section to be used for shear design? 0) No (False) 1) Yes (True)
FTBINC	0	Incremental flange width (top and bottom) used in the design of tapered sections.
FTINC	0	Incremental top flange width used in the design of tapered sections. In this case, the bottom flange width will remain unchanged.

Parameter Name	Default Value	Description
FU	Depends on FYLD	Ultimate tensile strength of steel in current units. If FYLD < 40 KSI, then FU = 58 KSI If 40 KSI ≤ FYLD ≤ 50 KSI, then FU = 60 KSI If FYLD > 50 KSI, then FU = 65 KSI
FYLD	36 KSI	Yield strength of steel in current units.
HECC	0.0	Eccentricity of opening with respect to the centerline of the member.
KX	1.0	K value used in computing KL/r for flexural torsional buckling for tees and double angles.
KY	1.0	Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.
KZ	1.0	Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.
LX	Member Length	Length value used in computing KL/r for flexural torsional buckling for tees and double angles.
LY	Member Length	Length used to calculate slenderness ratio for buckling about the local y-axis.
LZ	Member Length	Same as LY, but in the local z-axis.
MAIN	0.0	Toggles the slenderness check 0.0) check for slenderness 1.0) suppress slenderness check Any value greater than 1 = Allowable KL/r in compression.
NSF	1.0	Net section factor for tension members.
OVR	1.0	Overstress factor. All the allowable stress are multiplied by this number. It may be assigned any value greater than 0.0. It is used to communicate increases in allowable stress for loads like wind and earthquake.
PLTHICK	0.0	Thickness of cover plate welded to the bottom flange of the composite beam.
PLWIDTH	0.0	Width of cover plate welded to the bottom flange of the composite beam.

Parameter Name	Default Value	Description
PROFILE		Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of actual to allowable stress.
RDIM	0.0	Dimensions of rectangular openings (at each section, `RDIM` has a length term and a depth term – see syntax below). If a member has more than one rectangular opening they can have different dimensions.
RHOLE	None	Section locations of rectangular openings along the length of the member. Maximum three locations can be specified for each member when there is no circular opening.
RBHEIGHT	0.0	Height of ribs in the form steel deck.
RBWIDTH	2.5 in.	Width of ribs in the form steel deck.
SHE	0	Option for calculating actual shear stress. 0) Compute the shear stress using VO/Ib 1) Computer the shear stress based on the area of the section element.
SHR	0	Indicates use of temporary shoring during construction. 0) Without shoring 1) With shoring
SSY	0.0	Sidesway 0.0) Sidesway in local y-axis. 1.0) No sidesway
SSZ	0.0	Same as SSY, but in local z-axis.
STIFF	Member Length or depth of beam, whichever is greater	Spacing of stiffeners for plate girder design.
STP	1	Section type as defined in ASD Manual table. 1) Rolled 2) Welded

Parameter Name	Default Value	Description
TAPER	1.0	Design basis for tapered members 0.0) Design tapered I-section based on rules of Chapter F and Appendix B of AISC only. Due not use the rules in Appendix F of AISC-89. 1.0) Design tapered I-sections based on the rules of Appendix F of AISC-89.
THK	4.0 in.	Thickness of concrete slab or the thickness of concrete slab above the form steel deck.
TMAIN	300	Any value greater than 1 = Allowable KL/r in tension.
TORSION	0.0	Toggles the check for torsion 0.0) No torsion check is performed 1.0) Perform torsion check based on rules of AISC T114.
TRACK	0.0	Controls the level of detail to which results are reported: 0) minimum detail 1) intermediate detail 2) maximum detail (see Figure 2.1)
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the bottom flange
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the top flange.
WELD	1 for closed sections, 2 for open sections	Weld type as described in D1.B.1.10 Weld Design (on page 1145): 1) welding is on one side only except for wide-flange or tee sections, where the web is always assumed to be welded on both sides. 2) welding on both sides. For closed sections like a pipe or tube, the welding will be on one side only.
WIDTH	0.25 times the member length	Effective width of the concrete slab.
WMAX		Maximum welding thickness.
WMIN		Minimum welding thickness.

Parameter Name	Default Value	Description
WSTR	0.4 × FYLD	Allowable weld stress. Refer to D1.B.1.10 Weld Design (on page 1145) for how WELD, WMAX, WMIN, and WSTR parameters are used in weld design.

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if `SET Z UP` is used).

Notes

1. When performing the deflection check, you can choose between two methods. The first method, defined by a value `0` for the `CAN` parameter, is based on the local displacement. See TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement.

   If the `CAN` parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by `DJ1` (or in the absence of `DJ1`, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at `DJ2` or the end node of the member.

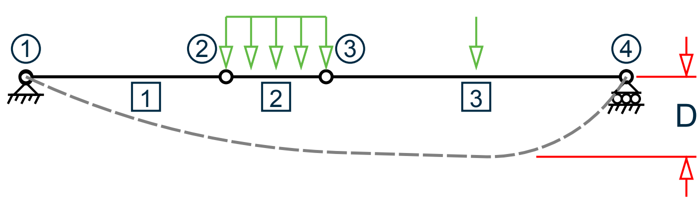
   Compute Delta = $\sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}$

   Compute Length = distance between `DJ1` and `DJ2` or, between start node and end node, as the case may be.

   Then, if `CAN` is specified a value 1, dff = L/Delta

   Ratio due to deflection = `DFF`/dff
2. If `CAN = 0`, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different.

   For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Also the straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured. Thus, for all three members here, `DJ1` should be "1" and `DJ2` should be "4".

   D is equal to the maximum local deflection for members 1, 2, and 3.

   ```
   PARAMETERS
   DFF 300. ALL
   DJ1 1 ALL
   DJ2 4 ALL
   ```
3. If `DJ1` and `DJ2` are not used, “Deflection Length” will default to the member length and local deflections will be measured from original member line.
4. It is important to note that unless a `DFF` value is specified, STAAD.Pro will not perform a deflection check. This is in accordance with the fact that there is no default value for `DFF`.

5. A critical difference exists between the parameters UNT/UNB and the parameters LY and LZ. Parameters UNT and UNB represent the laterally unsupported length of the compression flange. It is defined in Chapter F, page 5-47 of the specifications in the AISC 1989 ASD manual as the distance between cross sections braced against twist or lateral displacement of the compression flange. These parameters are used to calculate the allowable compressive stress (FCZ and FCY) for behavior as a beam. Parameters LY and LZ are the unbraced lengths for behavior as a column and are used to calculate the KL/r ratios and the allowable axial compressive stress FA.
6. Parameters SSY and CMY are based upon two values defined in page 5-55, Chapter H of the AISC 9th ed. manual. SSY is a variable which allows you to define whether or not the member is subject to sidesway in the local Y direction. CMY is a variable used for defining the expression called Cm in the AISC manual. When SSY is set to 0 (which is the default value), it means that the member is subject to sidesway in the local Y direction. When SSY is set to 1.0, it means that the member is not subject to sidesway in the local Y direction. The only effect that SSY has is that it causes the program to calculate the appropriate value of CMY. If SSY is set to 0 and CMY is not provided, STAAD.Pro will assume CMY as 0.85. If SSY is set to 1 and CMY is not provided, STAAD.Pro will calculate CMY from the equation on page 5-55. However, if you provide CMY, the program will use that value and not calculate CMY at all, regardless of what you defines SSY to be.

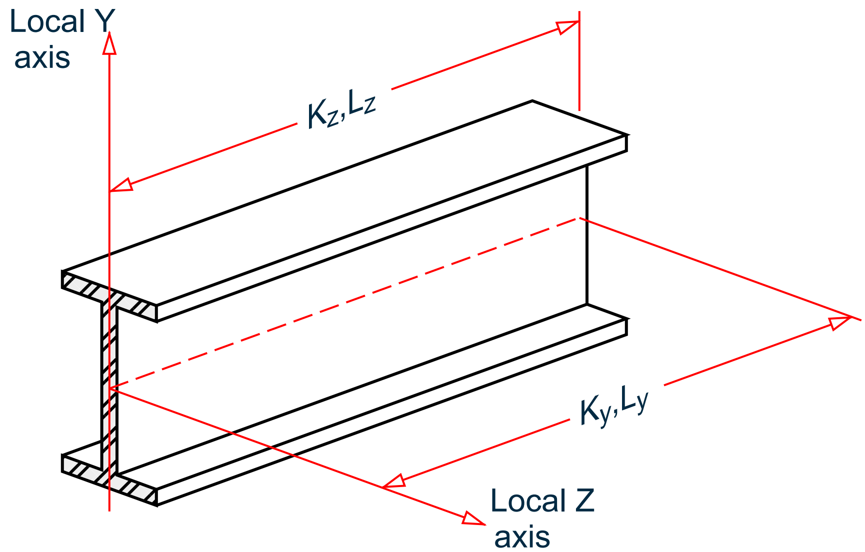


Figure 127: Terms used in calculating slenderness ratios KL/r for local Y and Z axes

7. For a T shape which is cut from a parent I, W, S, M or H shapes, the PROFILE parameter should be assigned a value corresponding to the parent shape. For example, if the T desired is an American WT6, specify W12 for the PROFILE parameter.

D1.B.1.3 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs.

Code checking can be done with any type of steel section listed in D1.A.4 Built-in Steel Section Library (on page 1088) of this manual.

Related Links

- *TR.49 Code Checking Specification* (on page 2677)
- *TR.49.1 Member Selection Specification* (on page 2678)
- *TR.49.2 Member Selection by Optimization* (on page 2679)

D1.B.1.4 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (i.e., the lightest section) which fulfills the code requirements for the specified member.

The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the `PROFILE` parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters `DMAX` and `DMIN` which limit the maximum and minimum depth of the members. If the `PROFILE` parameter is provided for specified members, `DMAX` or `DMIN` parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in D1.A.4 Built-in Steel Section Library (on page 1088). Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in that table.

Member selection *cannot* be performed on members whose section properties are input as prismatic.

D1.B.1.4.1 Member Selection by Optimization

Steel table properties of an entire structure can be optimized by STAAD.Pro. This optimization method involves the following steps.

1. `CHECK CODE ALL`
2. Modify the ratios
3. `SELECT ALL`
4. `PERFORM ANALYSIS`
5. `SELECT ALL`

An additional step of grouping may be performed if the `FIXED GROUP` and `GROUP` commands are provided (See TR.50 Group Specification (on page 2680)). After the last step, a re-analysis is not automatically performed, so users must ensure that they specify the analysis command following the `SELECT OPTIMIZE` command.

D1.B.1.4.2 Deflection Check With Steel Design

This facility allows the user to consider deflection as a criteria in the `CODE CHECK` and `MEMBER SELECTION` processes. The deflection check may be controlled using the three parameters `DJ1`, `DJ2`, and `DFF` which are described in D1.B.1.2 Design Parameters (on page 1134). Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

Related Links

- *TR.49.1 Member Selection Specification* (on page 2678)

D1.B.1.5 Truss Members

As described in G.8.1 Truss and Tension- or Compression-Only Members (on page 2121), a truss member is capable of carrying only axial forces. So during the design phase, no calculation time (or, matrix bandwidth) is wasted determining the allowable bending or shear stresses. Therefore, if there is any truss member in an analysis (such as a bracing or a strut, etc.), it is advisable to declare it as a truss member rather than as a regular frame member with both ends pinned.

D1.B.1.6 Unsymmetric Sections

For unsymmetric sections like single angles, STAAD.Pro considers the smaller section modulus for calculating bending stresses.

For single angles, the "specification for allowable stress design of single-angle members," explained in pages 5-309 to 5-314 of the AISC-ASD 9th edition manual has been incorporated.

D1.B.1.7 Composite Beam Design as per AISC-ASD

In G.6.7 Composite Beams and Composite Decks (on page 2119) of this manual, two methods of specifying the properties of a beam as a composite section (I-shaped beam with concrete slab on top) are described. Those members can be designed as composite beams in accordance with the AISC ASD code provisions. If the properties are assigned using the explicit method as defined in Section 1.7.7, the design parameters must be separately assigned. The `CMP` parameter in particular must be set a value of `1` or `2`. If the properties are derived from the composite decks, the design parameters are automatically generated during the deck creation phase, and hence no separate parameters need to be assigned.

Other parameters used in the design of composite members are: `DIA`, `DR1`, `DR2`, `FPC`, `HGT`, `PLTHICK`, `PLWIDTH`, `RBHEIGHT`, `RBWWIDTH`, `SHR`, `THK`, and `WIDTH`. Refer to D1.B.1.2 Design Parameters (on page 1134) for details on design parameters.

Example

```
UNIT INCH
PARAMETER
CODE AISC
BEAM 1 ALL
TRACK 2 ALL
DR1 0.3135 ALL
WID 69.525 ALL
FPC 3.0 ALL
THK 4.0 ALL
CMP 1 ALL
CHECK CODE ALL
SELECT ALL
```

D1.B.1.8 Plate Girders

The requirements of Chapter G – pages 5-51 through 5-53 of the AISC ASD 9th edition manual – are not implemented. Therefore, if the web slenderness ratio h/t_w of a section exceeds $\frac{970}{\sqrt{F_y}}$, STAAD will *not* perform a design for that member.

D1.B.1.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

a. `MEMBER` refers to the member number for which the design is performed.
b. `TABLE` refers to the AISC steel section name which has been checked against the steel code or has been selected.
c. `RESULT` prints whether the member has `PASS`ed or `FAIL`ed. If the `RESULT` is `FAIL`, there will be an asterisk (*) mark in front of the member number.
d. `CRITICAL COND` refers to the section of the AISC code which governed the design.
e. `RATIO` prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
f. `LOADING` provides the load case number which governed the design.
g. `FX`, `MY`, and `MZ` provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments to perform design, only FX MY and MZ are printed since they are the ones which are of interest, in most cases.
h. `LOCATION` specifies the actual distance from the start of the member to the section where design forces govern.
i. If the parameter `TRACK` is set to 1.0, the program will block out part of the table and will print the allowable bending stresses in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV), all in kips per square inch. In addition, member length, area, section moduli, governing KL/r ratio and CB are also printed.
j. In the output for `TRACK 2.0`, the items Fey and Fez are as follows:

$$F_{ey} = 12\pi^2 E/[23\cdot(K_y L_y/r_y)^2$$

$$F_{ez} = 12\pi^2 E/[23\cdot(K_z L_z/r_z)^2$$

Example Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************

  |--------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES   |
  |*************                                       |        IN INCH UNIT |
  |            *  |=============================|   ===|===     ------------ |
  |MEMBER   5  *  |  AISC SECTIONS              |      |        AX =   8.85  |
  |            *  | ST  W14X30                  |      |   --Z  AY =   3.39  |
  |DESIGN CODE *  |                             |      |        AZ =   3.47  |
  | AISC-1989  *  ===============================   ===|===     SY =   5.82  |
  |            *                                                SZ =  42.05  |
  |            *  |<---LENGTH (FT)=   10.00 --->|               RY =   1.49  |
  |*************                                                RZ =   5.73  |
  |                                                                          |
  |                163.3 (KIP-FEET)                                          |
  |PARAMETER        |                                      L1  STRESSES      |
  |IN KIP  INCH     |                              L1  L1      IN KIP  INCH  |
  |---------------  +                          L1              -------------|
  | KL/R-Y=  80.64  |                      L1                  FA  =  15.11  |
  | KL/R-Z=  20.93  +                  L1                      fa  =   0.30  |
  | UNL   = 120.00  |              L1                          FCZ =  19.95  |
  | CB    =   1.00  +                                          FTZ =  21.60  |
  | CMY   =   0.85  |       L1  L1                             FCY =  27.00  |
```

```
| CMZ   =   0.85  +   L1                                       FTY =  27.00 |
| FYLD  =  36.00  |L0                                          fbz =  46.60 |
| NSF   =   0.85  +---+---+---+---+---+---+---+---+---|        fby =   0.00 |
| DFF   =   0.00  -9.1                                         Fey =  22.97 |
| dff=      0.00                ABSOLUTE MZ ENVELOPE           Fez = 340.99 |
| (KL/R)max =  80.64              (WITH LOAD NO.)              FV  =  14.40 |
|                                                              fv  =   4.78 |
|                                                                           |
|                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
|                     -------------------------                             |
|                                                                           |
|               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
|                                                                           |
|     VALUE      -2.7        16.5        0.0         0.0       163.3        |
|  LOCATION      0.0         0.0        0.0         0.0        10.0        |
|   LOADING        1           1          0           0           1        |
|                                                                           |
|**************************************************************************|
|*                                                                        *|
|*                       DESIGN SUMMARY  (KIP-FEET)                       *|
|*                       --------------                                   *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX              MY             MZ          LOCATION              |
|        ======================================================            |
|          FAIL        AISC- H2-1        2.335             1               |
|          2.66 T          0.00      -163.32            10.00              |
|*                                                                        *|
|**************************************************************************|
|                                                                           |
|---------------------------------------------------------------------------|
```

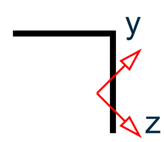
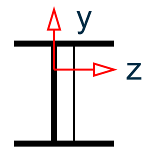
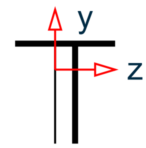
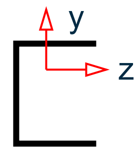
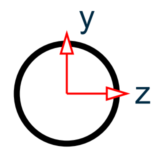
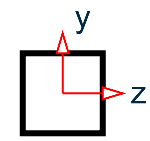
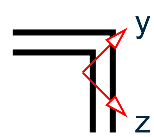
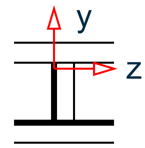
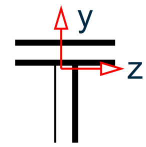
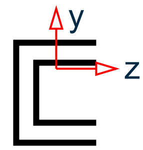
D1.B.1.10 Weld Design

Selected provisions of the AISC specifications for the Design, Fabrication and Erection of Steel for Buildings, 1999, and the American Welding Society D1.1 Structural Welding Code – Steel, 1998, have been implemented.

STAAD.Pro is able to select weld thickness for connections and tabulate the various stresses. The weld design is limited to the members having properties from wide flange, tee, single angle, single channel, pipe and tube section tables only. The parameters `WELD`, `WMAX`, `WMIN`, and `WSTR` (as explained in D1.B.1.2 Design Parameters (on page 1134)) govern the weld design.

Since the thickness of a weld is very small in comparison to its length, the properties of the weld can be calculated as line member. Therefore, the cross-sectional area (AZ) of the weld will actually be the length of the weld. Similarly, the units for the section moduli (SY and SZ) will be length-squared and for the polar moments of inertia (JW) will be length-cubed. The following table shows the different available weld lines, their type and their coordinate axes.

Table 80: WELD Parameter Values

Weld Type	Angle	Wide Flange	Tee	Channel	Pipe	Tube
1						
2					n/a	n/a

Actual stresses, calculated from the member forces, can be specified by three names, based on their directions.

Horizontal Stress as produced by the local z-shear force and torsional moment.

$$F_h = \frac{VZ}{AX} + \frac{CH \times MX}{JW}$$

where

VZ	=	Shear in local z-axis
MX	=	Torsional moment
CH	=	Distance of the extreme fiber for horizontal (local z) forces
AX	=	Area of the weld as the line member
JW	=	Polar moment of inertia

Vertical Stress as produced by the axial y-shear force and torsional moment.

$$F_v = \frac{VY}{AX} + \frac{CV \times MX}{JW}$$

where

VY	=	Shear in local y-axis
MX	=	Torsional moment
CY	=	Distance of the extreme fiber for vertical (local y) forces
AX	=	Area of the weld as the line member
JW	=	Polar moment of inertia

Direct Stress as produced by the axial force and bending moments in the local y and z directions.

$$F_d = \frac{FX}{AX} + \frac{|MZ|}{SZ} + \frac{|MY|}{SY}$$

where

FX	=	Axial force
MY	=	Bending in local y-axis
MZ	=	Bending in local z-axis
CH	=	Distance of the extreme fiber for horizontal (local z) forces
AX	=	Area of the weld as the line member

SY	=	= Section modulus around local y-axis
SZ	=	= Section modulus around local z-axis

Note: The moments MY and MZ are taken as absolute values, which may result in some conservative results for asymmetrical sections like angle, tee and channel.

Combined Stress calculated by the square root of the summation of the squares of the above three principal stresses.

$$F_{comb} = \sqrt{F_h^2 + F_v^2 + F_d^2}$$

The weld thickness required is then calculated by:

$$t_w = \frac{F_{comb}}{F_W}$$

where

F_W = Allowable weld stress, defined by the `WSTR` parameter.

The default value is $0.4 \times F_y$ (where F_y = `FYLD` parameter value).

The thickness, t_w, is rounded up to the nearest 1/16th of an inch and all the stresses are recalculated. The tabulated output prints the latter stresses. If the parameter `TRACK` is set to 1.0, the output will include the weld properties. The program does not calculate the minimum weld thickness as needed by some codes, but checks only against the minimum thickness as provided by the user (or 1/16th inch if not provided).

When the `TRUSS` qualifier is used with `SELECT WELD` command, the program will design the welds required for truss angle and double angle members that are attached to gusset plates. The program reports the number of welds (two for single angles, four for double angles), and the length required for each weld. The thickness of the weld is taken as 1/4 inch (6 mm) for members up to 1/4 inch (6 mm) thick, and 1/16 inch (1.5 mm) less than the angle thickness for members greater than 1/4 inch (6 mm) thick. The minimum weld length is taken as four times the weld thickness.

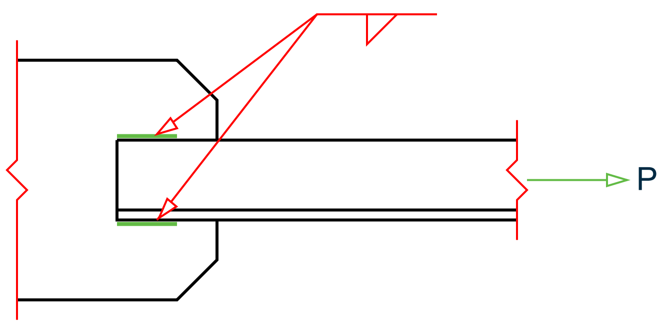


Figure 128: Weld design for `SELECT WELD TRUSS`

D1.B.2 Castellated Beams

STAAD.Pro comes with the non-composite castellated beam tables.

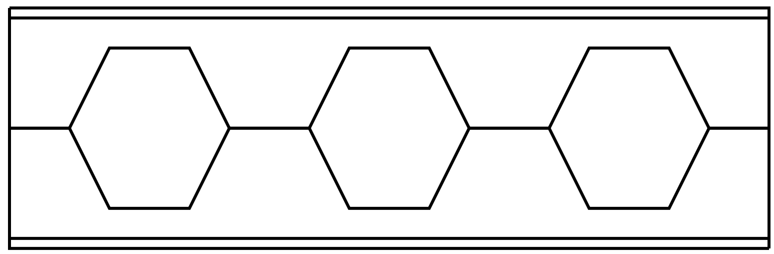


Figure 129: Castellated beam elevation

According to the manufacturer, castellated beams are manufactured by cutting a wide flange along the web in a "zig-zag" pattern, offsetting the two halves, and welding the two halves together, as shown in the next figure. As a result, the underlying steel section is a wide flange (W shapes) in the AISC table or a B shape. STAAD currently supports only the ones derived from W shapes.

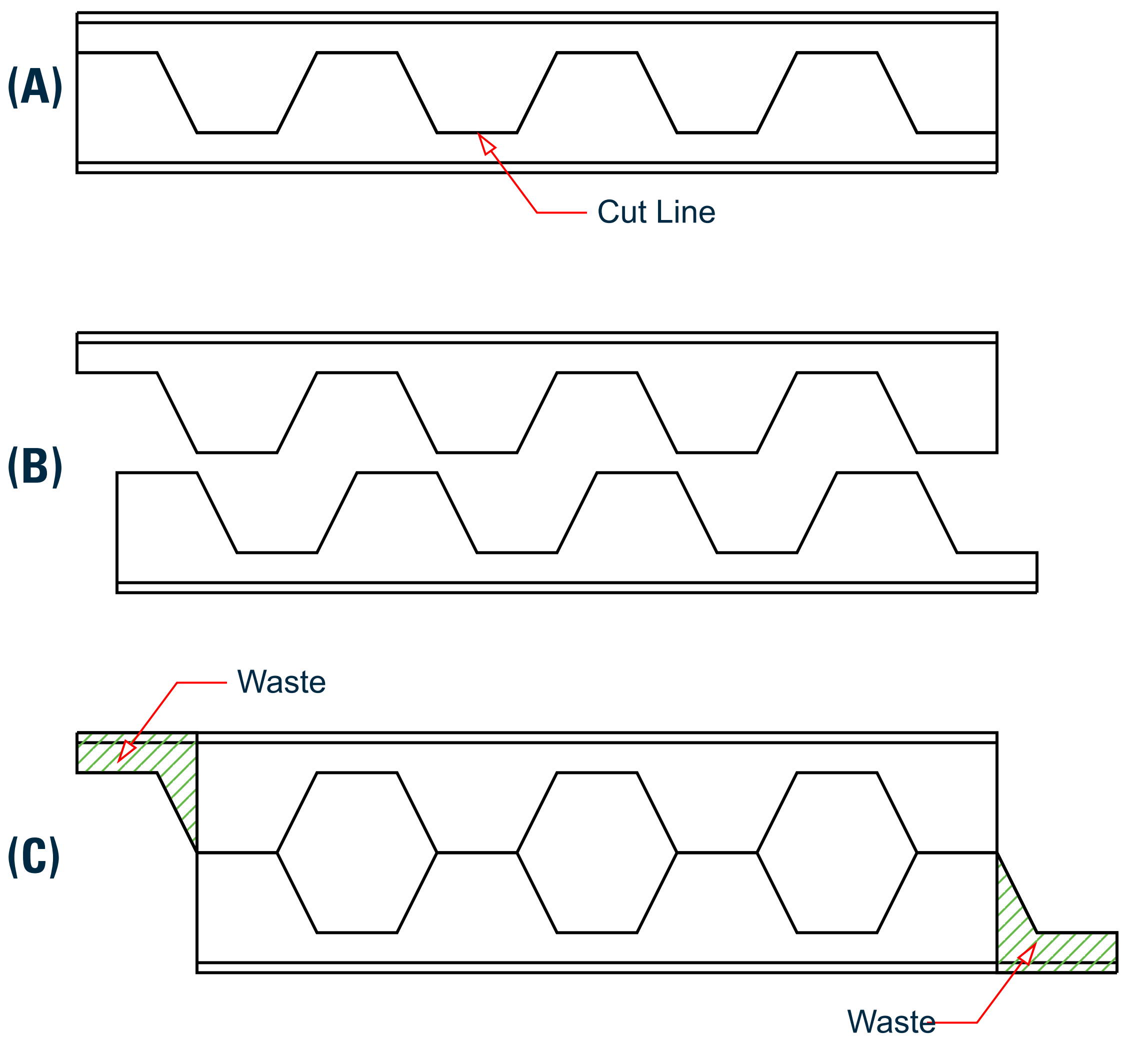


Figure 130: Manufacturing process for castellated beams: A) Cut hexagonal line, B) Stagger top and bottom sections, and C) discard waste at ends

Related Links

- *D1.A.4.3 Castellated Beams Section Sizes* (on page 1091)

D1.B.2.1 Analysis and Design Criteria

The local axis system (local X, local Y and local Z) of a castellated beam is identical to that for a wide flange, and is shown in G.4.2 Local Coordinate System (on page 2088).

It is important recognize that there are two basic issues to be understood with regard to these members a) analysis b) steel design

First, the design issues because only then will their relationship with the analysis issues become apparent. Design of a castellated beam is done only for FY (shear along the web) and MZ (moment about the major axis which is the Z axis). If at the start of the design process, the program detects that the beam has axial force (FX), shear along local-Z (FZ), torsion (MX) or moment about the minor axis (MY), design of that member will be terminated.

Next is how these design limitations have a bearing on the analysis issues. If you intend to design these members, as a result of the above restrictions, he/she must model it in such a way that none of the 4 unacceptable degrees of freedom end up with a non-zero value anywhere along the length of the member. That means, if the member ends are defined as supports, the support conditions must be defined with the above in mind. Similarly, if the castellated member is attached to other members, its end conditions (`MEMBER RELEASES`) must be modeled taking the above facts into consideration.

The design limitations also have a bearing on the type of loads that are applied to the member. Loads which cause any of the above-mentioned four degrees of freedom to end up with a non-zero value will cause the member design to be terminated.

However, if you wish to only analyze the structure, and are not interested in performing a steel design, the above described restrictions for supports, member end conditions or loading are not applicable.

The design method is the allowable stress method, using mainly the rules stated in the AISC ASD 9th edition code. Only code checking is currently available for castellated beams. Member selection is not.

Note: STAAD.Pro does not multiply the analysis moment by 1.7 for ASD method. It is up to you to multiply the dead and live loads by 1.7 in load combination and using this load case in design. The reason is that if program internally multiplies the analysis moment by 1.7 for ASD method (it is 1.2 for dead and 1.6 for live loads for LRFD method) then you must ensure that the analysis moment is the unfactored moment. If by mistake the 1.7 factor is used during load combination and the I-beam with web opening is designed with this load, the program will further increase the load by 1.7. Hence, it has been intentionally left to your engineering judgment to use whatever load factor you see fit before designing I-beam with opening with that factored load case.

D1.B.2.2 Design Parameters

The following table contains a list of parameters and their default values.

Table 81: American Castellated Beam Design Parameters

Parameter Name	Default Value	Description
CB	1.0	Cb value used for computing allowable bending stress per Chapter F of the AISC specifications.

Parameter Name	Default Value	Description
CMZ	0.85	Cm value in local z-axis. Used in the interaction equations in Chapter H of the AISC specifications.
EOPEN	1.5e + b	Distance from the ecenter of the last hole to the end of the member. 1.5e + b is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b.
FYLD	36 ksi	Yield stress of steel.
RATIO	1.0	Permissible maximum ratio of actual load to section capacity. Any input value will be used to change the right hand side of the governing interaction equations in Chapter H and elsewhere in the AISC specifications.
SOPEN	1.5e + b	Distance from the start of the member to the center of the first hole. 1.5e + b is the minimum allowable value. Any value greater than or equal to this minimum will be used by the program. See Figure 2.13 for the definition of e and b.
TRACK	0	Used to control the level of description of design output. 0 = Detailed output suppressed 1 = Detail output included
UNL	Member Length	Unsupported length of compression flange for calculating allowable bending stress.

D1.B.2.3 Design Procedure

Cross-Section Checks

The first check that is carried out is a verification whether the member properties satisfy certain basic requirements. If the member fails these checks, the remainder of the checks are not performed.

The cross section checks are the following:

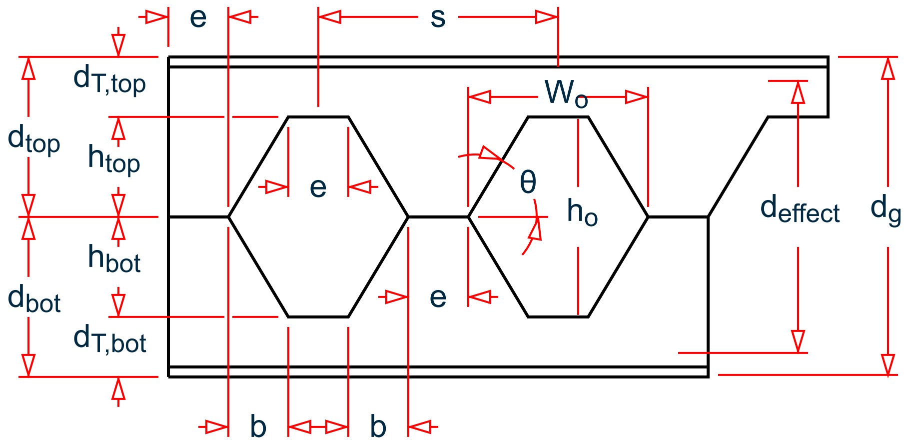


Figure 131: Castellated beam nomenclature

1. Web Post Width (e) should be at least 3.0 inches
2. Tee Depth (d_T-top and d_T-bot) should be greater than the thickness of flange plus one inch.
3. Angle θ should be between 45 and 70 degrees.
4. In order for the program to determine the number of holes which are admissible for the beam, the parameters SOPEN and EOPEN need to be assigned. In the figure above, there is a term shown as S. This value is part of the section tables supplied with STAAD.Pro, so it retrieves that value from there. It then computes the number of holes, and the remainder of the terms shown in the above diagram.
5. SOPEN and EOPEN (see the parameter table shown earlier) have to be at least 1.5e + b, with “e” and “b” as shown in the earlier figure. If you inputs a value less than these minima, the minimum values are used.

References

Design of castellated beams in STAAD.Pro is based on the information gathered from the following sources:

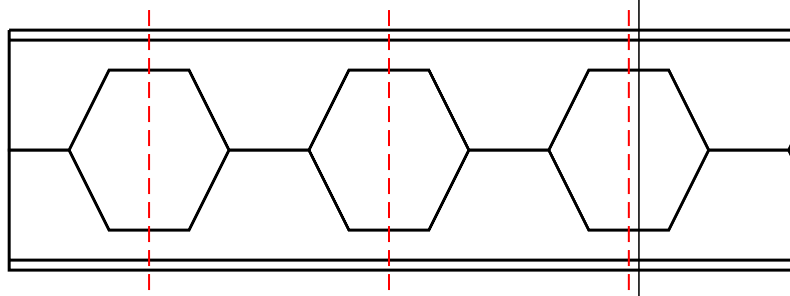
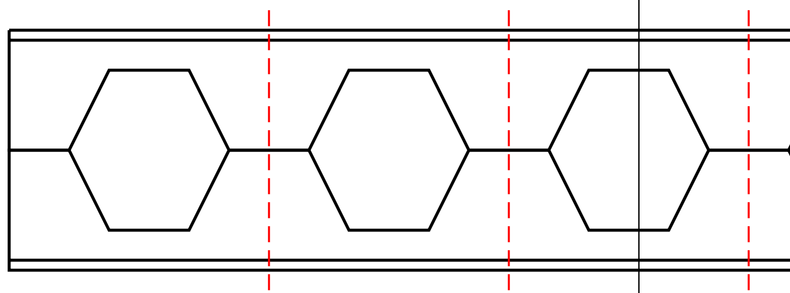
1. *Design of Welded Structures*, Omer W. Blodget, published by The James Lincoln Arc Welding Foundation, pages 4.7-8 and 4.7-9
2. AISC 9th edition manual – Allowable stress design
3. ASCE *Journal of Structural Engineering* 124:10, October 1998, “Castellated Beam Web Buckling in Shear,” R.G. Redwood and S. Demirdjian

Checking the member for adequacy in carrying the applied loading

This consists of five different checks:

1. Global Bending
2. Vierendeel Bending
3. Horizontal Shear
4. Vertical Shear
5. Web Post Buckling

Table 82: Cross section considered for limit states

Design For	Section Considered in the Design (shown with the vertical dotted lines)
Vierendeel Bending	
Global Bending Vertical Shear Horizontal Shear Web Post Buckling	

1. Global Bending:

   Global bending check is done at the web post section. This is the region of the member where the full cross section is active, without interference of the holes.

   The actual bending stress is computed at the middle of the web post location and is obtained by dividing the moment by the section modulus of the full section.

   For computing the allowable bending stress, the compactness of the section is first determined in accordance with Table B5.1 in the Chapter B of the AISC 9th edition specifications. The rules applicable to I-shaped sections are used for this. Following this, the allowable bending stress is computed per chapter F of the same.

   The ratio is computed by dividing the actual stress by the allowable stress.
2. Vierendeel Bending:

   This is checked at the middle of the hole locations. The effective cross section at these locations is a Tee. The overall moment (Mz) at the span point corresponding to the middle of the hole is converted to an axial force and a moment on the Tee.

   The actual stress is computed at the top and bottom of each Tee section.

   $$f_a = M / (d_{effect} \times A_t)$$

   where

   A_t = area of the Tee section

   $$f_b = V \times e \times a / (2 \times S)$$

   where

α = the area factor. For the top Tee section, α = Area of Top Tee / (Area of Top Tee + Area of Bottom Tee)

Allowable Stresses for vierendeel bending:

- Axial Stress: The allowable axial stress is computed as per the Chapter E of the AISC specifications. The unsupported length for column buckling is equal to *e*.
- Bending Stress: The allowable bending stress is computed for the top and bottom Tee section as per the Chapter F of the AISC manual.

The axial stress plus bending stress is computed at the top and bottom of each tee section. If it is compressive then it is checked against equations H1-1 and H1-2 of Chapter H of the AISC manual. If it is tensile then it is checked against equation H2-1

3. Horizontal Shear

   Allowable Shear stress is computed as $0.4\ Fy$.

   Actual Stress: Please refer to pages 4.7-8 and 4.7-9 of reference #1.

4. Vertical Shear

   Allowable Shear stress is computed as $0.4\ Fy$.

   The actual shear stress is computed at the middle of the web post location.

5. Web Post buckling

   Refer to pages 1202-1207 of reference #3.

D1.B.2.4 General Format

The command syntax in the STAAD input file for assigning castellated beams is:

```
MEMBER PROPERTY AMERICAN
```

```
Member-list TABLE ST section-name
```

Example

```
MEMBER PROPERTY AMERICAN
2 TABLE ST CB12X28
```

Assigning Design parameters

Under the PARAMETERS block on input, the code name must be specified as:

```
CODE AISC CASTELLATED
```

Example

```
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 25 31
FYLD 50 MEMB 25 31
SOPEN 11.124 MEMB 25 31
…
CHECK CODE MEMB 25 31
```

D1.B.2.5 Steel Design Output

The following is a typical `TRACK 2` level output page from a STAAD output file.

```
                       STAAD.PRO CODE CHECKING - (AISC CASTELLATED)   v1.0
                       ********************************************
 ALL UNITS ARE - Kip and Inches (UNLESS OTHERWISE NOTED)


   Castellated Steel Design for Member      2
   ==========================================

   Section Name  ST  CB27X40

   Design Results
   --------------
   Design Status: Pass   Critical Ratio: 0.96

   Check for Global Bending
   -------------------------
   Load =     3         Section =    260.874
   Fy =       0.76           Mz =   -3020.39
   Fb top =      33.00   Fb Bot =      33.00
   fb  =      26.83
   Ratio =  0.81

   Check for Vierendeel Bending
   ----------------------------
   Load =     3         Section =    214.624
   Fy =       4.61           Mz =   -2894.76
   Fa =      29.91           Fb =      30.00
   Klr =       1.46          Fe =   69606.88
   fa =      26.08           fb =       2.79
   Ratio =  0.96

   Check for Vertical Shear
   -------------------------
   Load =     3         Section =      0.000
   Fy =      22.50           Mz =       0.00
   Fv =      20.00           fv =       2.62
   Ratio =  0.13

   Check For Horizontal Shear ( Web Post )
   ----------------------------------------
   Load =     3         Section =    519.874
   Fy =     -20.82           Mz =    -415.10
   Fv =      20.00           fv =      14.73
   Ratio =  0.74

   Check for Web Post Buckling
   ---------------------------
   Load =     3         Section =    519.874
   Fy =     -20.82           Mz =    -415.10
   Mallow =     141.32     Mact =     189.47
   Ratio =  0.75
```

Viewing the design results in the GUI

1. After the analysis and design is completed, double click on the castellated member in the STAAD.Pro View window.
   The **Beam** dialog box.
2. Select the **Castellated Beam Design** tab.

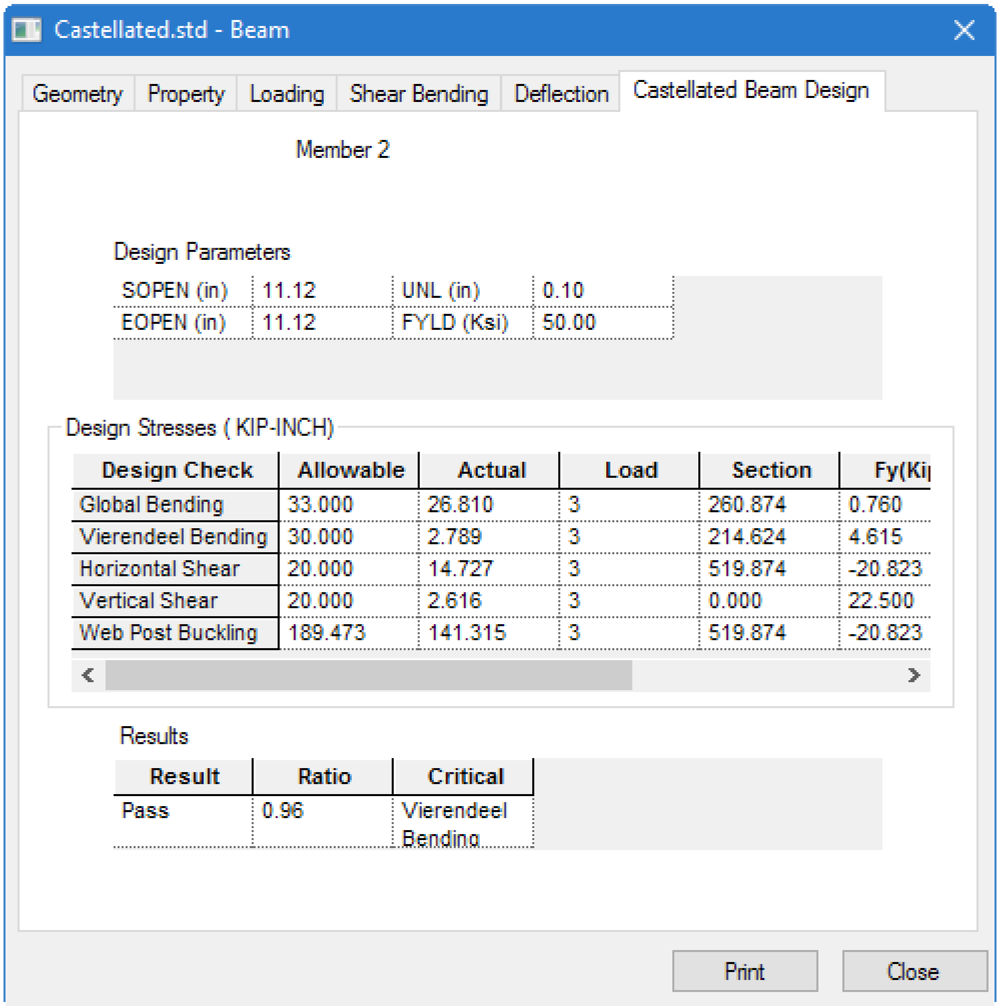


*Figure 132: **Beam** dialog box Castellated Beam Design tab*

3. (Optional) Click **Print** to create a hard copy of the Castellated Beam data for this beam.
4. Click **Close**.
   The dialog closes.

D1.B.2.6 Example

The following is an example STAAD Input for a portal frame with a castellated beam.

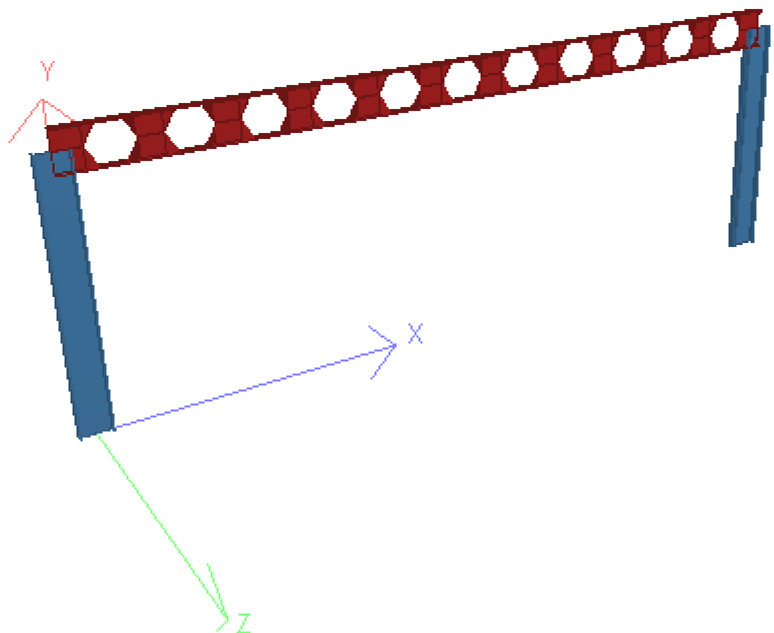


Figure 133: 3D Rendering of the example structure

Tip: You can copy the input code below and paste into the STAAD Editor or into a plain text editor program and save as an `.STD` file for use in STAAD.Pro.

```
STAAD PLANE EXAMPLE PROBLEM FOR
*CASTELLATED BEAM DESIGN
UNIT FT KIP
JOINT COORDINATES
1 0. 0. ; 2 45 0
3 0 15 ; 4 45 15
MEMBER INCIDENCE
1 1 3; 2 3 4; 3 4 2
MEMBER PROPERTY AMERICAN
2 TA ST CB27x40
1 3 TA ST W21X50
UNIT INCH
CONSTANTS
E STEEL ALL
DEN STEEL ALL
POISSON STEEL ALL
MEMBER RELEASE
2 START MX MY MZ
2 END MY MZ
UNIT FT
SUPPORT
1 2 FIXED
LOADING 1 DEAD AND LIVE LOAD
MEMB LOAD
2 UNI Y -0.4
LOADING 2 WIND FROM LEFT
MEMBER LOAD
2 UNI Y -0.6
LOAD COMB 3
1 1.0 2 1.0
```

```
PERFORM ANALYSIS
LOAD LIST 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
UNIT KIP INCH
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 2
FYLD 50 MEMB 2
CMZ 0.85 MEMB 2
CB 1.1 MEMB 2
TRACK 2.0 ALL
SOPEN 11.124 MEMB 2
EOPEN 11.124 MEMB 2
CHECK CODE MEMB 2
FINISH
```

D1.B.3 Design of Beams with Web Openings

Design of steel members with web openings per AISC Steel Design Guide 2 - ASD specifications may be performed in STAAD for members whose yield strength is 65 ksi or less.

Note: The web openings are given consideration only during the design phase. The reduction in section properties caused by the presence of the openings is not considered automatically during the analysis phase. Hence, the analysis is performed as if the full section properties are effective for such members.

During the design process, the program first determines the utilization ratio (U.R.) at the location of the opening as though it is an unreinforced opening. If the U.R. is less than 1.0, the member is presumed to have passed the requirements at that location. If the U.R. exceeds 1.0, then it determines the U.R. as though it is a reinforced opening. If it fails this too, the cause of the failure along with the associated numerical values is reported.

Related Links

- *V. AISC ASD - Design of Steel Beam with Web Opening* (on page 5747)
- *V. AISC ASD - Deflection Check for a Steel Beam with Web Opening* (on page 5754)

D1.B.3.1 Description

The following are the salient points of the design process.

A. Only a code check operation is permitted on members with web openings. The `MEMBER SELECTION` process will not be performed if a web opening is specified for the member.

B. The `CODE CHECK` operation is performed at the following locations along the member span:

   a. The 13 equally spaced points along the member span customary with the `BEAM 1` parameter

   or

   The section locations specified using the `SECTION` command, if the `BEAM` parameter is set to 0.0

   or

   The 2 member ends if `BEAM` parameter is set to 0, and the `SECTION` command is not specified.

   b. At the web openings locations defined using the `RHOLE` and `CHOLE` parameters (Refer to Table 2.18-1 below).

   If any of the locations defined under (a) above happen to coincide with those in (b), such locations are designed as places where openings are located, and not as an unperforated section location.

The utilization ratio (U.R.) is determined for all the locations in (a) above, as well as all the locations in (b) above. The highest value among these locations is deemed critical from the design standpoint.

The design output consists of the critical value obtained from checking the locations under (a), and each of the locations under (b).

The critical location among those in (b) is *not* displayed in the post-processing pages of the program such as the Beam-Unit Check page, or the Member Query-Steel Design tab.

Members declared as `TRUSS` (trusses) or `TENSION` (tension-only) are not designed for web openings.

D1.B.3.2 Design steps for Steel Beam with Web Opening

At the location of web holes, the capacity of the section is determined using the rules explained below.

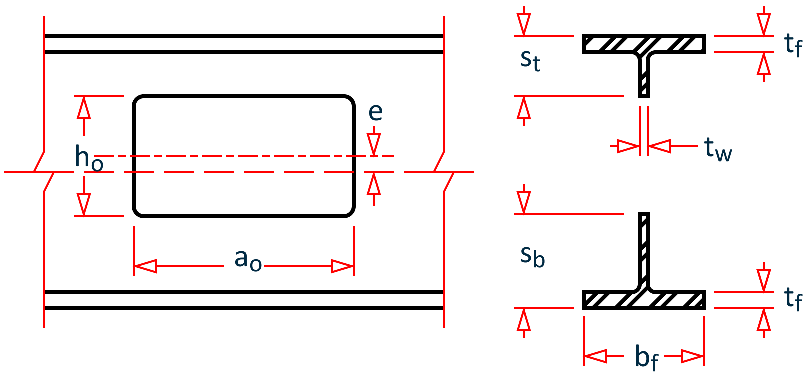


Figure 134: Opening configurations for steel beams with unreinforced opening

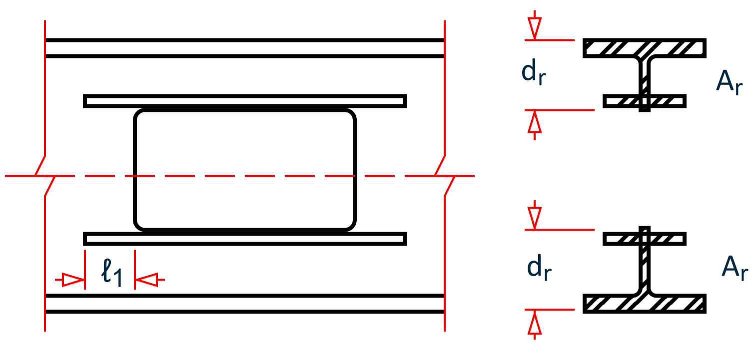


Figure 135: Opening configurations for steel beams with reinforced opening

Section Properties Required:

A_s = Cross-sectional area of steel in the unperforated member (at a section along the beam where there is no opening)
d = Depth of beam
t_w = Thickness of web
b_f = Width of flange
t_f = Thickness of flange
Z = Plastic section modulus of member without opening
J = Torsional constant of the beam

L = Length of the member
L_b = Unbraced length of compression flange

Opening Information Required:

e = Eccentricity of opening, specified using the HECC parameter (distanced from mid-depth of beam to mid-depth of the opening) = | e |
sign convention for eccentricity: upward eccentricity + e, downward eccentricity - e

Loading:

V_u = Factored shear at different opening locations
M_u = Factored bending moment at different opening locations

Calculated Parameters

Circular Opening Properties:

Unreinforced Opening Properties:

$h_o = D_o$ for bending
$h_o = 0.9\ D_o$ for shear
$a_o = 0.45\ D_o$

Reinforced Opening Properties:

$h_o = D_o$ for bending and shear
$a_o = 0.45\ D_o$

Tee Properties:

Refer to Figure 2.17(a), above

s_t = Depth of top tee = $d/2 - (h_o\ /\ 2 + e)$
s_b = Depth of bottom tee = $d/2 - (h_o\ /\ 2 - e)$

Reinforcement Properties:

A_r = Area of reinforcement on each side of the opening
t_r = Thickness of reinforcement bar
b_r = Width of reinforcement bar
D_r = Depth of reinforcement bar

D1.B.3.3 Calculation Steps

1. Check for local buckling of compression flange and reinforcement (if any)

   (AISC Design Guide 2 for Web Openings: section 3.7.a.1)

   Width to thickness ratio of compression flange, $F_1 = b_f / 2t_f$
   Width to thickness ratio of web reinforcement, $F_2 = b_r / t_r$
   Limiting width to thickness ratio, $B_1 = 65 / \sqrt{F_y}$
   F_1 and F_2 must not exceed B_1.
2. Check for web buckling

(AISC Design Guide 2 for Web Openings: section 3.7.a.2)

Width to thickness ratio of web, $W_1 = (d – 2t_f) / t_w$
W_1 must not exceed $520/\sqrt{F_y}$

3. Check for opening dimensions to prevent web buckling

(AISC Design Guide 2 for Web Openings: section 3.7.a.2 & section 3.7.b.1)

a. Limit on a_o / h_o as given below,

If W_1 is $\le 420/\sqrt{F_y}$ then web qualifies as stocky,

a_o / h_o must not exceed 3.0,

If W_1 is $> 420/\sqrt{F_y}$ but $\le 520/\sqrt{F_y}$ then

a_o / h_o must not exceed 2.2,

b. h_o / d must not exceed 0.7

c. The opening parameter, $p_o = (a_o / h_o) + (6 h_o / d)$ must not exceed 5.6

4. Check for tee dimensions

(AISC Design Guide 2 for Web Openings: section 3.7.b.1)

a. The maximum depth of the web opening is governed by the following rules:

Depth $s_t \ge 0.15d$, $s_b \ge 0.15d$

b. Aspect ratios of the tees ($v = a_0 / s$) should not be greater than 12

$a_0 / s_b \le 12$, $a_0 / s_t \le 12$

5. Check for buckling of tee-shaped compression zone

(AISC Design Guide 2 for Web Openings: section 3.7.a.3)

The tee, which is in compression, is investigated as an axially loaded column. For unreinforced members this is not done when the aspect ratio of the tee is less than or equal to four:

$$F_y t_w d/\sqrt{3} \le 4$$

For reinforced openings, this check is only done for large openings in regions of high moment.

6. Calculation for Maximum Moment Capacity, M_m

(AISC Design Guide 2 for Web Openings: section 3.5.a)

For unperforated section

$$M_p = F_y Z$$

$$\Delta A_s = h_o t_w - 2A_r$$

Unreinforced Opening:

$$M_m = M_p \left[1 - \frac{\Delta A_s \left(\frac{h_o}{4} + e \right)}{Z} \right]$$

Reinforced Opening:

a. If $t_w e < A_r$

$$M_m = M_p \left[1 - \frac{t_w \left(\frac{h_o^2}{4} + h_o e - e^2 \right) - A_r h_o}{Z} \right] \le M_p$$

b. If $t_w e \geq A_r$

$$M_m = M_p\left[1 - \frac{\Delta A_s\left(\frac{h_o}{4} + e - \frac{A_r}{2t_w}\right)}{Z}\right] \leq M_p$$

7. Calculation for Maximum Shear Capacity, V_m

(AISC Design Guide 2 for Web Openings: section 3.6.a)

V_{pb} and V_{pt} = Plastic shear capacity of the web of the tee

$$V_{pb} = F_y t_w s_b / \sqrt{3}$$

$$V_{pt} = F_y t_w s_t / \sqrt{3}$$

The values of aspect ratios ν_b and ν_t and factors μ_b and μ_t(which appear in the equations shown below) are different for reinforced and unreinforced openings.

Unreinforced Opening:

For the bottom tee, $\nu_b = a_o / s_b$ and $\mu_b = 0$
For the top tee, $\nu_t = a_o / s_t$ and $\mu_t = 0$

Reinforced Opening:

s_{t1} and s_{b1} are used to calculate for n reinforced opening.

$$s_{t1} = s_t - A_r / (2b_f)$$

$$s_{b1} = s_b - A_r / (2b_f)$$

P_r = Force in reinforcement along edge of opening = $F_y A_r \leq F_y t_w a_o / (2\sqrt{3})$
d_{rt} and d_{rb} = Distance from outside edge of flange to centroid of reinforcement.

$$d_{rt} = s_t - ½\, t_r$$

$$d_{rb} = s_b - ½\, t_r$$

For the bottom tee, $\nu_b = a_o / s_{b1}$ and $\mu_b = 2P_r d_{rb} / (V_{pb} s_b)$

For the top tee, $\nu_t = a_o / s_{t1}$ and $\mu_t = 2P_r d_{rt} / (V_{pt} s_t)$

General Equations:

Using equations given below for α_{vb} and α_{vt}

α_{vb} = Ratio of nominal shear capacity of bottom tee, V_{mb} to plastic shear capacity of the web of the tee = $(\sqrt{6} + \mu_b) / (\nu_b + \sqrt{3}) \leq 1$
α_{vt} = Ratio of nominal shear capacity of top tee, V_{mt} to plastic shear capacity of the web of the tee = $(\sqrt{6} + \mu_t) / (\nu_t + \sqrt{3}) \leq 1$

$$V_{mb} = V_{pb} a_{vb}$$

$$V_{mt} = V_{pt} a_{vt}$$

V_m = Maximum nominal shear capacity at a web opening = $V_{mb} + V_{mt}$

8. Check against Maximum Shear Capacity V_m

(AISC Design Guide 2 for Web Openings: section 3.7.a.2)

V_p = Plastic shear capacity of unperforated web = $F_y t_w d / \sqrt{3}$

If $W_1 \leq 420/\sqrt{F_y}$, V_m must not exceed 2/3 V_p
If $W_1 > 420/\sqrt{F_y}$ but $\leq 520/\sqrt{F_y}$, V_m must not exceed 0.45 V_p

9. Check against Moment Shear Interaction

AISC Design Guide 2 for Web Openings: section 3.2 & section 3.4)

$$R1 = V_u / V_m \leq 1.0$$

$$R2 = M_u / M_m \leq 1.0$$

$$R1^3 + R2^3 \leq R^3, R \leq 1.0$$

10. Corner Radii (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.2)

Minimum radii = the greater of $2t_w$ or 5/8 inch

11. Calculation of length of fillet weld (for reinforced opening only)

(AISC Design Guide 2 for Web Openings: section 3.7.b.5)

For reinforcing bars on both sides / on one side of the web:

Fillet welds should be used on both sides of the reinforcement on extensions past the opening. The required strength of the weld within the length of the opening is,

$$R_{wr} = 2P_r$$

Where:

R_{wr} = Required strength of the weld

The reinforcement should be extended beyond the opening by a distance

$$L_1 = a_o / 4 \text{ or } L_1 = A_r\sqrt{3} / (2t_w)$$

whichever is greater, on each side of the opening. Within each extension, the required strength of the weld is

$$R_{wr} = F_y A_r$$

Additional requirements for reinforcing bars on one side of the web:

A_f = area of flange = $b_f \cdot t_f$

a. $A_r \leq A_f / 3$

b. $a_o / h_o \leq 2.5$

c. $V1 = s_t / t_w$ or $V2 = s_b / t_w$

$V1$ and $V2 \leq 140 / \sqrt{F_y}$

d. $M_u / (V_u d) \leq 20$

12. Calculation for spacing of openings

(AISC Design Guide 2 for Web Openings: section 3.7.b.6)

Rectangular Opening:

$$S \geq h_o$$

$$S \geq a_o (V_u / V_p) / [1 - (V_u / V_p)]$$

Circular Opening:

$$S \geq 1.5 D_o$$

$$S \geq D_o (V_u / V_p) / [1 - (V_u / V_p)]$$

13. Check for deflection

The deflection check is performed using the approximate procedure described in section 6.2 of the AISC Design Guide 2 for Web Openings.

D1.B.3.4 General Format

```
RHOLE r1 r2 r3 Memb <list>
```

```
CHOLE c1 c2 c3 Memb <list>
```

Where:

r1, *r2*, and *r3* and *c1*, *c2*, and *c3* are the section locations of three rectangular and three circular openings respectively, along the length of the member in ascending order from the start of the member (i.e., r1< r2<r3 and c1< c2<c3)

Notes

The maximum number of openings allowed for each member is three. Thus there can be three rectangular openings, three circular openings, or a combination of three rectangular and circular openings for each member.

```
RDIM [l1 d1] [l2 d2] [l3 d3] Memb <list>
```

Where l1, l2, and l3 are the three different lengths and d1, d2 and d3 are the three different depths of the rectangular openings.

```
CDIA d1 d2 d3 Memb <list>
```

Where *d1*, *d2*, and *d3* are the three different diameters of the circular openings.

```
HECC e1 e2 e3 Memb <list>
```

If the eccentricity of the opening is in the negative local Y-axis of the member the sign should be negative.

```
ELECTRODE f Memb <list>
```

Where *f* is the weld material used to calculate size and length of fillet weld required to connect reinforcing bars on beam web at opening.

Example

```
UNIT INCH PARAMETER RHOLE 0.4 0.6 MEMB 5
RDIM 10.0 5.0 20.0 10.0 MEMB 5 CHOLE 0.8 MEMB 5 CDIA 10.0 MEMB 5 ELECTRODE 3
MEMB 5
```

The above example shows that member 5 contains two rectangular openings at sections 0.4 and 0.6 whereas one circular opening is located at section 0.8 of the member. The dimensions of rectangular openings are 10.0 × 5.0 and 20.0 × 10.0 inch respectively whereas diameter of circular opening is 10.0 inch.

D1.C. American Codes - Steel Design per AISC LRFD Specification

The 2nd and 3rd editions of the American AISC LRFD code have been implemented. The commands to access those respective codes are:

For the 3rd edition code:

```
PARAMETER
```

```
CODE LRFD
```

or

```
PARAMETER
```

```
CODE LRFD3
```

For the 2nd Edition:

```
PARAMETER
```

```
CODE LRFD2
```

D1.C.1 General Comments

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized--ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the LRFD specifications as implemented in STAAD steel design. A detailed description of the design process along with its underlying concepts and assumptions is available in the LRFD manual. However, since the design philosophy is drastically different from the conventional Allowable Stress Design (ASD), a brief description of the fundamental concepts is presented here to initiate the user into the design process.

D1.C.2 LRFD Fundamentals

The primary objective of the LRFD Specification is to provide a uniform reliability for all steel structures under various loading conditions. This uniformity can not be obtained with the allowable stress design (ASD) format.

The ASD method can be represented by the inequality

$$\Sigma Q_i < R_n/F.S.$$

The left side is the required strength, which is the summation of the load effects, Q_i (forces and moments). The right side, the design strength, is the nominal strength or resistance, R_n, divided by a factor of safety. When divided by the appropriate section property (area or section modulus), the two sides of the inequality become the actual stress and allowable stress respectively. ASD, then, is characterized by the use of unfactored "working" loads in conjunction with a single factor of safety applied to the resistance. Because of the greater variability and, hence, unpredictability of the live load and other loads in comparison with the dead load, a uniform reliability is not possible.

LRFD, as its name implies, uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible. The LRFD method may be summarized by the inequality

$$y_i Q_i < R_n Ø$$

On the left side of the inequality, the required strength is the summation of the various load effects, Q_i, multiplied by their respective load factors, y_i. The design strength, on the right side, is the nominal strength or resistance, R_n, multiplied by a resistance factor, Ø.

In the STAAD implementation of LRFD, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

D1.C.3 Analysis Requirements

The types of construction recognized by AISC specification have not changed, except that both “simple framing” (formerly Type 2) and “semi-rigid framing” (formerly Type 3) have been combined into the same category, Type PR (partially restrained). “Rigid Framing” (formerly Type 1) is now Type FR (fully restrained). Type FR construction is permitted unconditionally. Type PR construction may necessitate some inelastic, but self-limiting, deformation of a structural steel element. Thus, when specifying Type PR construction, the designer should take into consideration the effects of partial restraint on the stability of the structure, lateral deflections and second order bending moments. As stated in Sect. C1 of the LRFD specification, an analysis of second order effects is required. Thus, when using LRFD code for steel design, the user must use the P-Delta analysis feature of STAAD.

D1.C.4 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, noncompact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and user specified shapes and design accordingly.

D1.C.5 Limit States

Related Links

- *V. AISC LRFD - Wide Flange Tension Capacity* (on page 5760)
- *V. AISC LRFD - Angle Section Tension Capacity* (on page 5761)
- *V. AISC LRFD - Wide Flange Compression Capacity 1* (on page 5763)
- *V. AISC LRFD - Wide Flange Compression Capacity 2* (on page 5765)
- *V. AISC LRFD - Angle Section Compression Capacity* (on page 5767)
- *V. AISC LRFD - Tee Section Compression Capacity* (on page 5768)
- *V. AISC LRFD - Rectangular HSS Compression Strength* (on page 5770)
- *V. AISC LRFD - Double Angle Compression Capacity* (on page 5772)
- *V. AISC LRFD - Round HSS Compression Capacity* (on page 5774)
- *V. AISC LRFD - Wide Flange Flexural Strength 1* (on page 5775)
- *V. AISC LRFD - Wide Flange Flexural Strength 2* (on page 5778)

D1.C.5.1 Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter `NSF` (see D1.C.6 Design Parameters (on page 1167)). STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

D1.C.5.2 Axial Compression

The column strength equations have been revised in LRFD to take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD according to the procedure outlined in Chapter E of the LRFD specifications. For slender elements, the procedure described in Appendix B5.3 is used.

Singly symmetric and unsymmetric compression members are designed on the basis of the limit states of flexural-torsional and torsional buckling. The procedure of Appendix E3 is implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D1.C.5.3 Flexural Design Strength

In LRFD, the flexural design strength of a member is determined by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section.

The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length L_r and buckling moment M_r are functions of the section geometry and are calculated as per the procedure of Chapter F.

The purpose of bending coefficient C_b is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter CB (see D1.C.6 Design Parameters (on page 1167)) or may be calculated by the program (if CB is specified as 0.0). In the absence of the parameter CB, a default value of 1.0 will be used.

The procedure for calculation of design strength for flexure also accounts for the presence of residual stresses of rolling.

To specify laterally unsupported length, either or both of the parameters UNB and UNT (see D1.C.6 Design Parameters (on page 1167)) can be used.

D1.C.5.4 Combined Axial Force and Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1-1a and H1-1b. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

D1.C.5.5 Design for Shear

The procedure of Sect. F2 of the LRFD Specification is used in STAAD to design for shear forces in members. Shear strength as calculated in LRFD is governed by the following limit states: Eq. F2-1a by yielding of the web; Eq. F2-2a by inelastic buckling of the web; Eq. F2-3a by elastic buckling of the web. Shear in wide flanges and channel sections is resisted by the area of the web, which is taken as the overall depth times the web thickness.

D1.C.6 Design Parameters

Design per LRFD specifications is requested by using the CODE parameter. Other applicable parameters are summarized in Table 2-4. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters DMAX and DMIN are applicable for member selection only.

Table 83: AISC LRFD (2nd and 3rd Ed.) Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified LRFD or LRFD3 for 3rd Edition. Must be specified LRFD2 for 2nd Edition.

Parameter Name	Default Value	Description
AXIS	1	1 = Design single angles for bending about their principle axis. 2 = Design single angles for bending about their geometric axis.
BEAM	1.0	0.0 = design at start and end nodes and those locations specified by the `SECTION` command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. **Note:** See D1.A.6 Design Parameters (on page 1100).
CAN	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see D1.B.1.2 Design Parameters (on page 1134))
CB**	1.0	Coefficient Cb per Chapter F of AISC LRFD. If Cb is set to 0.0, it will be calculated by the program. Any of value will be used directly in design.
DFF	None (Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Joint of Member	Joint No. denoting starting point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
DJ2	End Joint of Member	Joint No. denoting end point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))

Parameter Name	Default Value	Description
DMAX	45.0 in.	Maximum allowable section depth.
DMIN	0.0 in.	Minimum allowable section depth
FLX	1	1 = Single angle member is *not* fully braced against lateral torsional buckling. 2 = Single angle member is fully braced against lateral torsional buckling. 3 = Single angle member is braced against lateral torsional buckling at the point of maximum moment.
FYLD	36.0 ksi	Yield strength of steel.
FU	60.0 ksi	Ultimate tensile strength of steel.
KX	1.0	K value for flexural-torsional buckling.
KY	1.0	Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.
KZ	1.0	Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.
LX	Member Length	Length used for flexural-torsional buckling.
LY	Member Length	Length to calculate the slenderness ratio for buckling about the local y-axis.
LZ	Member Length	Length to calculate the slenderness ratio for buckling about the local z-axis.
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check Any value greater than 1 = Allowable KL/r in compression.

Parameter Name	Default Value	Description
NSF	1.0	Net section factor for tension members.
PROFILE		Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of actual load effect to design strength.
STIFF	Member Length or depth, whichever is greater	Spacing of stiffeners for beams for shear design.
STP	1	Section type to determine F_r (compressive residual stress in flange) per 3rd Ed. LRFD spec., p 16.1-97. 1 = Rolled section (F_r = 10 ksi) 2 = Welded section (F_r = 16.5 ksi)
TMAIN	300	Any value greater than 1 = Allowable KL/r in tension.
TRACK	0.0	Specified the level of detail included in the output. 0.0 = Suppress all design strengths 1.0 = Print all design strengths 2.0 = Print expanded design output
UNB	Member Length	Unsupported length (L_h) of the bottom* flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.
UNT	Member Length	Unsupported length (L_h) of the top* flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if `SET Z UP` is used).

Note: For a description of the deflection check parameters DFF, DJ1, DJ2, and CAN, see the D1.B.1.2 Design Parameters (on page 1134) of this manual.

The STIFF parameter represents the term "a" as defined in Section F2, page 6-113 of the LRFD 2nd edition manual.

** Non-default values of CB must be re-entered before every subsequent CHECK CODE or SELECT command.

D1.C.7 Code Checking and Member Selection

Both code checking and member selection options are available in STAAD LRFD implementation. See D1.B.1.3 Code Checking (on page 1141) and D1.B.1.4 Member Selection (on page 1142) for general information on these options.

Example for the LRFD-2001 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

Example for the LRFD-1994 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD2
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

Related Links

- *TR.49 Code Checking Specification* (on page 2677)
- *TR.49.1 Member Selection Specification* (on page 2678)
- *TR.49.2 Member Selection by Optimization* (on page 2679)

D1.C.8 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. A detailed discussion of the format is provided in D1.B.1.9 Tabulated Results of Steel Design (on page 1144). Following exceptions may be noted: CRITICAL COND refers to the section of the LRFD specifications which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

D1.C.9 Composite Beam Design per the AISC LRFD 3rd edition code

The design of composite beams per the 3rd edition of the American LRFD code has been implemented. The salient points of this feature are as follows:

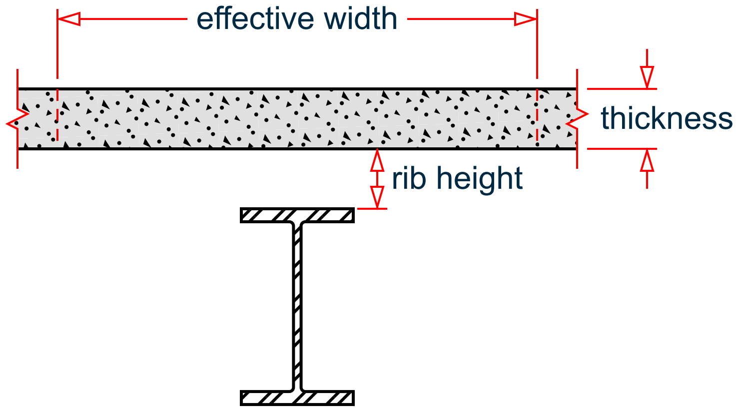


Figure 136: Nomenclature of composite beams

where

t	=	thickness of slab
A_s	=	area of steel beam
f_c	=	ultimate compressive strength of concrete
b	=	effective slab width
h_r	=	rib height

Table 84: Composite Beam Design Parameters for AISC-LRFD

Parameter Name	Default Value	Description
RBH	0.0 in.	Rib height for steel form deck.
EFFW	Value used in analysis	Effective width of the slab.
FPC	Value used in analysis	Ultimate compressive strength of the concrete slab.

Theoretical Basis

1. Find the maximum compressive force carried by concrete as:

   $0.85\ f_c \cdot b \cdot t$
2. Find the maximum tensile force carried by the steel beam as:

   $A_s\ .\ f_y$

   Tensile strength of concrete is ignored.
3. If step 1 produces a higher value than step 2, plastic neutral axis (PNA) is in the slab. Else, it is in the steel beam.

Location of the Plastic Neutral Axis (PNA) defines the moment capacity:

- Case 1: PNA in the slab

  Find the depth of the PNA below the top of the slab as:

  $$0.85f_c \cdot b \cdot a = A_s \cdot f_y$$

  Rearranging terms:

  $$a = A_s \cdot f_y/(0.85f_c \cdot b)$$

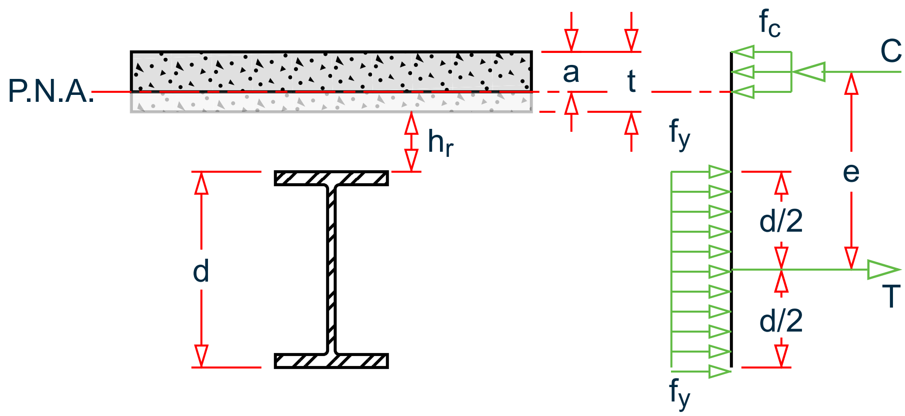


Figure 137: Plastic neutral axis in the concrete slab

  Lever arm

  $$e = d/2 + h_r + t - a/2$$

  Moment Capacity

  $$\phi_b(A_s \cdot f_y)e$$

- Case 2: PNA in Steel Beam

  $$C_s + C_b = T_b$$

  where

C_s	=	compressive force in slab = $0.85f_c \cdot b \cdot t$
C_b	=	compressive force in steel beam
T_b	=	tensile force in steel beam

  Since the magnitude of $C_b + |T_b| = A_s \cdot f_y$

  Substituting for $T_b = (A_s \cdot f_y - C_b)$ gives:

  $$C_s + C_b = A_s \cdot f_y - C_b$$

  Rearranging terms:

  $$C_b = (A_s \cdot f_y - C_s)/2$$

  Determine whether the PNA is within the top flange of the steel beam or inside the web:

  where

C_f	=	maximum compressive force carried by the flange = $A_f \cdot f_y$
A_f	=	area of the flange

  If $C_f \geq C_b$, the PNA lies within the flange (Case 2A)

  If $C_f < C_b$, the PNA lies within the web (Case 2B)

- Case 2A: PNA in Flange of Steel Beam

  Calculate:

  $$y = C_f/(b_f \cdot f_y)$$

  where

  b_f = width of the flange

  The point of action of the tensile force is the centroid of the steel are below the PNA. After find that point, e_1 and e_2 can be calculated.

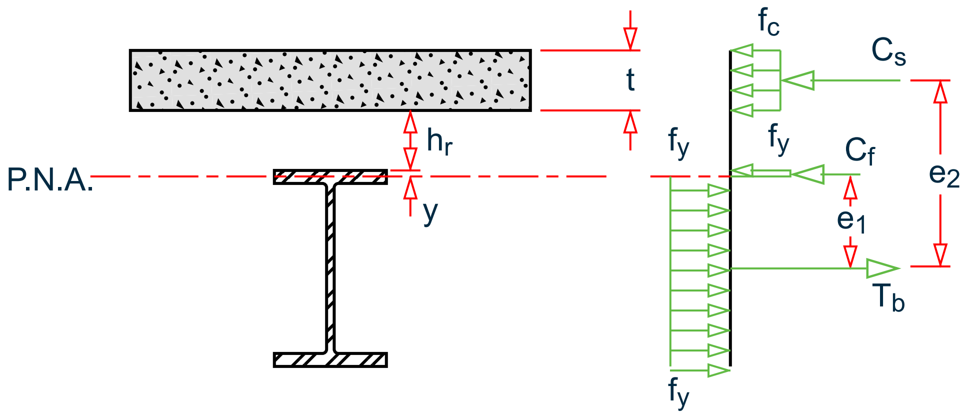


Figure 138: Plastic neutral axis falls within the top flange

  Moment Capacity

  $$\phi_b(C_f \cdot e_1 + C_s \cdot e_2)$$

- Case 2B: PNA in Web of Steel Beam

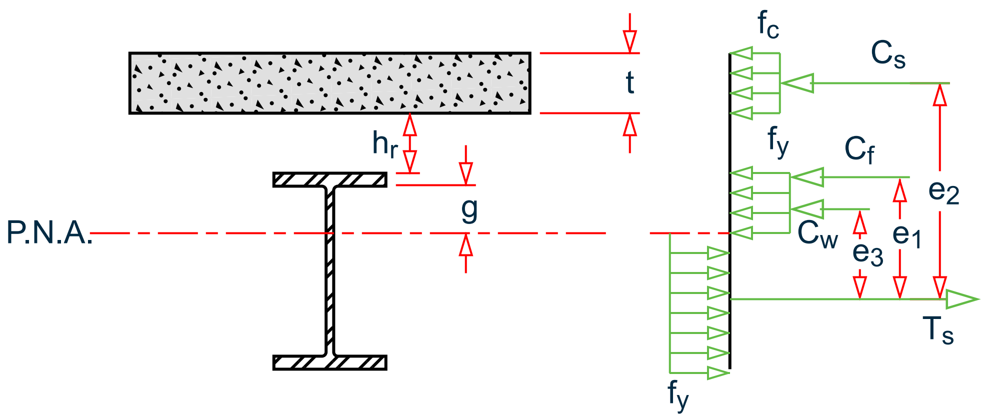


Figure 139: Plastic neutral axis falls within the web

  where

  C_w = compressive force in the web = C_b - C_f

  g = $C_w/(t_w \cdot f_y)$

  t_w = thickness of the web

The point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, e_1, e_2, and e_3 can be calculated.

Moment Capacity

$$\phi_b(C_s \cdot e_2 + C_f \cdot e_1 + C_w \cdot e_3)$$

Utilization Ratio = Applied Moment / Moment Capacity

Notes

1. Rib Height is the distance from top of flange of steel beam to lower surface of concrete.
2. If the slab is flush on top of the steel beam, set the Rib Height to zero.

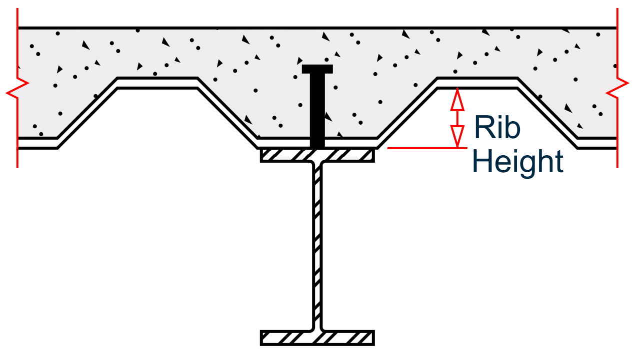


Figure 140: Steel deck form ribs

3. For moments which cause tension in the slab (called positive moments in STAAD.Pro convention), design of the beam is presently not carried out.
4. Shear connectors are presently not designed.
5. Member selection is presently not carried out.
6. In order to design a member as a composite beam, the member property specification during the analysis phase of the data must contain the `CM` attribute. See TR.20.1 Assigning Properties from Steel Tables (on page 2257) for details.

Example

```
STAAD SPACE
…
MEMBER PROPERTY
1 TA CM W12X26 CT 6.0 FC 4.0 CW 40.0
…
PERFORM ANALYSIS
…
PARAMETER
CODE LRFD
RBH 5.0 MEMB 1
CHECK CODE MEMB 1
FINISH
```

D1.D. American Codes - Steel Design per AASHTO Specifications

Design to AASHTO Standard Specifications for Highway Bridges utilizing the ASD and LRFD approaches are available in STAAD.Pro. These are described in the following two sections.

To utilize the ASD method, specify the commands

```
PARAMETER
CODE AASHTO
```

or

```
PARAMETER
CODE AASHTO ASD
```

To utilize the LRFD method, specify the commands

```
PARAMETER
CODE AASHTO LRFD
```

D1.D.1 AASHTO (ASD)

The design of structural steel members in accordance with the AASHTO *Standard Specifications for Highway Bridges*, 17th edition has been implemented.

Related Links

- *V. AASHTO 17th Ed ASD - Design Frame* (on page 4547)

D1.D.1.1 General

The section of the above code implemented in STAAD is Chapter 10, Part C – Service Load design Method, Allowable Stress design. Sections 10.32.1.A and 10.36 are implemented. As per the AASHTO committee, this is the last edition for this code (the ASD approach) and only technical errors will be fixed in the future for this code.

In general, the concepts followed in `MEMBER SELECTION` and `CODE CHECKING` procedures are similar to that of the AISC based design. It is assumed that you are familiar with the basic concepts of steel design facilities available in STAAD. Please refer to D. Steel Design (on page 969) for detailed information on this topic. This section specifically addresses the implementation of steel design based on the AASHTO specifications.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. The design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) are not supported.

D1.D.1.2 Allowable Stresses

The member design and code checking in STAAD.Pro is based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It is beyond the scope of this manual to describe every aspect of structural steel design per AASHTO specifications because of practical reasons. This section will discuss the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A of the AASHTO code specifies the allowable stresses.

Axial Stress

Allowable tension stress, as calculated in AASHTO is based on the net section. This tends to produce a slightly conservative result. Allowable tension stress on the net section is given by,

$$F_t = 0.55 \cdot F_y$$

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$F_a = F_y/FS \cdot [1-(Kl/r)^2 F_y]/(4\pi^2 E)$$

when $(Kl/r) \leq Cc$

$$Fa = \pi^2 E/[FS \cdot (Kl/r)^2]$$

when $(Kl/r) > Cc$

Where:

$$C_c = (2\pi^2 E/F_y)^{1/2}$$

It should be noted that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when 1/r exceeds a certain value.

Bending Stress

Allowable stress in bending compression for rolled shape girders and built-up sections whose compression flanges are supported laterally through their full length by embedment in concrete is given by:

$$F_b = 0.55 \cdot F_y$$

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

$$Fb = 0.55 \cdot F_y[1 - (1/r)^2 F_y/(4\pi^2 E)]$$

Where:

$$r^2 = b^2/12$$

Due to inadequate information in the AASHTO Code, the allowable tensile stresses due to bending for both axes are set to be the same as the corresponding allowable bending compressive stresses.

Shear Stress

Allowable shear stress on the gross section is given by:

$$F_v = 0.33\ F_y$$

For shear on the web, the gross section is defined as the product of the total depth and the web thickness. The AASHTO code does not specify any allowable stress for shear on flanges. The program assumes the same allowable for shear stress (0.33Fy) for both shear on the web and shear on the flanges. For shear on the flanges, the gross section is taken as 2/3 times the total flange area.

Bending-Axial Stress Interaction

Members subjected to both axial and bending stresses are proportioned according to section 10.36 of the AASHTO steel code. All members subject to bending and axial compression are required to satisfy the following formula:

$$f_a/F_a + C_{mx} \cdot f_{bx}/[(1 - f_a/F_{ex}) \cdot F_{bx}] + C_{my} \cdot f_{by}/[(1 - f_a/F_{ey}) \cdot F_{by}] < 1.0$$

at intermediate points, and

$$f_a/(0.472 \cdot F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at the ends of the member.

The start and end nodes of a member are treated as support points.

For members subject to axial tension and bending, the following equations are checked:

$$f_a/F_a + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at intermediate points, and

$$f_a/(0.472 \cdot F_y) + f_{bx}/F_{bx} + f_{by}/F_{by} < 1.0$$

at the ends of the member.

D1.D.1.3 AASHTO (ASD) Design Parameters

The following table outlines the parameters that can be used with the AASHTO (ASD) code along with the default values used if not explicitly specified.

Table 85: AASHTO (ASD) Design Parameters

Parameter Name	Default Value	Description
BEAM	1.0	0.0 = Design at ends and those locations specified by the SECTION command. 1.0 = Design at ends and every 1/12th point along the member length.
CB	1.0	Cb value as used in the calculation of Fb 0.0 = Cb value to be calculated Any other value will be used in the calculations.
CMY CMZ	0.85 for sideswayand calculatedfor no sidesway	Cm value in local y & z axes
DFF	None.(Mandatory for a deflection check)	“Deflection length” / Maximum allowable local axis deflection.
DJ1	Start joint of member	Joint No. denoting starting point for calculating “Deflection Length”.
DJ2	End joint of member	Joint No. denoting ending point for calculating “Deflection Length”.
DMAX	1000.0	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.

Parameter Name	Default Value	Description
DMIN	0.0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
FYLD	36 KSI	Yield strength of steel in current units.
KY	1.0	Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.
KZ	1.0	Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in local z-axis.
MAIN	0.0	0.0 =check for slenderness 1.0 =suppress slenderness check
NSF	1.0	Ratio of "Net cross section area" / 'Gross section area' for tension member design.
PROFILE	None	Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2676) for details.
PUNCH		1.0 = K-Overlap 2.0 = K-Gap 3.0 = T and Y 4.0 = Cross with diaphragms 5.0 = Cross without diaphragms
RATIO	1.0	Permissible ratio of the actual to allowable stresses.

Parameter Name	Default Value	Description
SSY	0.0	0.0 =Sidesway in local y-axis. 1.0 =No sidesway in local y-axis
SSZ	0.0	0.0 =Sidesway in local z-axis. 1.0 =No sidesway in local z-axis.
STIFF	Greater of member length or depth of beam.	Spacing of stiffeners for plate girder design in current length units.
TRACK	0	Level of detail in Output File: 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level.
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.
WSTR	0.4 x FYLD	Allowable welding stress

D1.D.2 AASHTO (LRFD)

The following outlines the implementation of the AASHTO *Standard Specifications for Highway Bridges* (LRFD, 1998) which has been implemented in STAAD.Pro.

Related Links

- *V. AASHTO 2nd Ed LRFD - Design Beam* (on page 4559)

D1.D.2.1 General

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of AASTHO-LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the AASTHO-LRFD specifications as implemented in STAAD steel design.

D1.D.2.2 Capacities per AASHTO (LRFD) Code

Axial Strength

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified through the use of the parameter NSF. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly

$$P_r = \varphi_y P_{ny} = \varphi_y F_y A_g$$

$$P_r = \varphi_u P_{nu} = \varphi_u F_u A_n U$$

where

P_{ny}	=	Nominal tensile resistance for yielding in gross section (kip)
F_y	=	Yield strength (ksi)
A_g	=	Gross cross-sectional area of the member (in^2)
P_{nu}	=	Nominal tensile resistance foe the fracture in the net section (kip)
F_u	=	Tensile strength (ksi)
A_n	=	Net area of the member
U	=	reduction factor to account for shear lag
φ_y	=	resistance factor for yielding of tension member
φ_u	=	resistance factor for fracture of tension members

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$\lambda = \left(\frac{Kl}{r_z \pi}\right)^2 \frac{F_y}{E}$$

if λ ≤ 2.25

Nominal compressive resistance,

$$P_n = 0.66^{\lambda} F_y A_s$$

if λ > 2.25

Nominal compressive resistance

$$P_n = 0.88 F_y A_s / \lambda$$

where

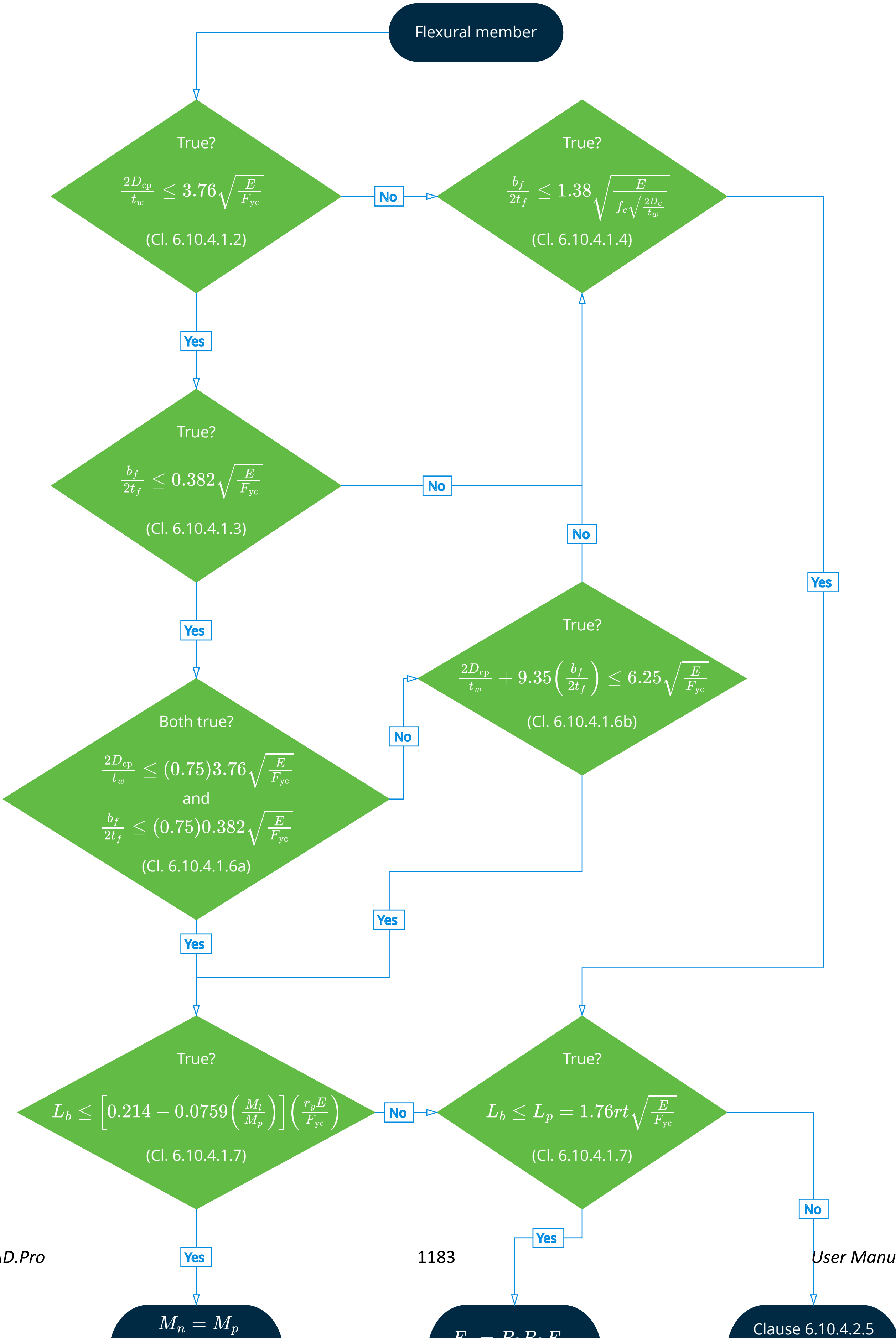
A_s	=	Gross sectional area

The Factored resistance

$$P_r = \varphi_c \, Pn$$

Bending Strength

The flow to calculate the allowable bending strength for rolled shape girders and built-up sections is given by the flowing flow chart.



Flexural member
True?
$\frac{2D_{cp}}{t_w} \leq 3.76\sqrt{\frac{E}{F_{yc}}}$
(Cl. 6.10.4.1.2)
No
True?
$\frac{b_f}{2t_f} \leq 1.38\sqrt{\frac{E}{f_c\sqrt{\frac{2D_c}{t_w}}}}$
(Cl. 6.10.4.1.4)
Yes
True?
$\frac{b_f}{2t_f} \leq 0.382\sqrt{\frac{E}{F_{yc}}}$
(Cl. 6.10.4.1.3)
No
No
Yes
Yes
True?
$\frac{2D_{cp}}{t_w} + 9.35\left(\frac{b_f}{2t_f}\right) \leq 6.25\sqrt{\frac{E}{F_{yc}}}$
(Cl. 6.10.4.1.6b)
No
Both true?
$\frac{2D_{cp}}{t_w} \leq (0.75)3.76\sqrt{\frac{E}{F_{yc}}}$
and
$\frac{b_f}{2t_f} \leq (0.75)0.382\sqrt{\frac{E}{F_{yc}}}$
(Cl. 6.10.4.1.6a)
Yes
Yes
True?
$L_b \leq \left[0.214 - 0.0759\left(\frac{M_l}{M_p}\right)\right]\left(\frac{r_y E}{F_{yc}}\right)$
(Cl. 6.10.4.1.7)
No
True?
$L_b \leq L_p = 1.76rt\sqrt{\frac{E}{F_{yc}}}$
(Cl. 6.10.4.1.7)
No
Yes
Yes
$M_n = M_p$
Clause 6.10.4.2.5

Shear Strength

The nominal shear resistance of un-stiffened webs of homogeneous girders shall be calculated as.

If $D/t_w \le 2.46\sqrt{(E/F_y)}$, then

$$V_n = V_p = 0.58 F_{yw} D t_w$$

If $2.46\sqrt{(E/F_y)} < D/t_w \le 3.07\sqrt{(E/F_y)}$, then

$$V_n = 1.48 \cdot t_w^{\ 2} \sqrt{(E \cdot F_y)}$$

If $D/t_w > 3.07\sqrt{(E/F_y)}$, then

$$V_n = 4.55 \cdot t_w^{\ 3} \cdot E/D$$

Bending-Axial Interaction

Members subjected to both axial forces and bending moments are proportioned according to section 6.9.2.2 of the AASHTO steel code. All members subject to bending and axial compression or axial tension are required to satisfy the following formula:

If $P_u/P_r < 0.2$, then

$$\frac{P_u}{2.0 P_r} + \left(\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \le 1.0$$

If $P_u/P_r \ge 0.2$, then

$$\frac{P_u}{P_r} + \frac{8}{9}\left(\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \right) \le 1.0$$

D1.D.2.3 AASHTO (LRFD) Design Parameters

The following table outlines the parameters that can be used with the AASHTO (LRFD) code along with the default values used if not explicitly specified.

Table 86: AASHTO (LRFD) Design Parameters

Parameter Name	Default Value	Description
BEAM	1.0	Identify where beam checks are performed: 0 = Perform design at ends and those locations specified in the SECTION command. 1 = Perform design at ends and 1/12th section locations along member length.
DMAX	1000	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.

Parameter Name	Default Value	Description
DMIN	0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
DFF	0	"Deflection Length"/Max allowable local deflection If set to 0, (default) then no deflection check is performed.
DJ1	Start joint of member	Joint No. denoting starting point for calculating "Deflection Length".
DJ2	End joint of member	Joint No. denoting ending point for calculating "Deflection Length".
GRADE	1	Grade of Steel: 1: Grade 36 2: Grade 50 3 Grade 50W 4: Grade 70W 5: Grade 100/100W Refer to AASHTO LRFD, Table 6.4.1-1
KY	1.0	Effective length factor to calculate slenderness ratio for buckling about local y-axis. Usually this is the minor axis.
KZ	1.0	Effective length factor to calculate slenderness ratio for buckling about local z-axis. Usually this is the major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about the local Y axis.
LZ	Member Length	Same as above except in local z-axis.

Parameter Name	Default Value	Description
MAIN	0.0	Flag for checking slenderness limit: 0.0 =check for slenderness 1.0 =suppress slenderness check
NSF	1.0	Net Section Factor. Ratio of (Net Area)/(Gross Area)
NSF	1.0	Net section factor for tension members.
TRACK	0	Level of detail in Output File: 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level..
UNB	Member Length	Unsupported length of bottom flange. Used for calculating the moment of resistance when the bottom of beam is in compression.
UNT	Member Length	Unsupported length of top flange. Used for calculating the moment of resistance when top of beam is in compression.

D1.E. American Codes - Steel Design per AISI Cold Formed Steel Code

Provisions of the AISI S100 *North American Specification for the Design of Cold-Formed Steel Structural Members*, 2016 Edition have been implemented. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations using the LRFD Method. For flexural members, the Nominal Section Strength is calculated on the basis of initiation of yielding in the effective section (Procedure I).

Note: Design per the AISC 1996 code has been deprecated.

D1.E.1 Cross-Sectional Properties

You specify the geometry of the cross-section by choosing one of the section shape designations from the STAAD.Pro Steel Tables for cold-formed sections, which mirror the Gross Section Property Tables published in AISI S100-16.

The following section profiles are supported in this implementation:

- Standard cold-formed sections from the section database:
  - angle (with or without lips)
  - channel (with or without lips) or eave strut
  - double-channels, arranged back-to-back or toe-to-toe

    Note: All double channels are treated as being directly welded together. Spacing is not considered for back-to-back or toe-to-toe arrangements.
  - zee or zee purlin (with or without lips)
  - hat
  - tube (SHS and RHS)
  - pipe and CHS
- The following UPT sections are supported:
  - angle
  - channel
  - tube
  - pipe

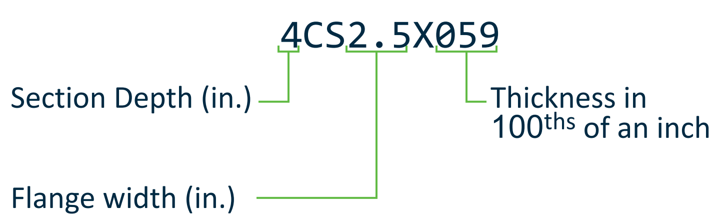
Shape selection may be done using the member property pages of the graphical user interface or by specifying the section designation symbol in the input file.

Standard hot-rolled sections, prismatic sections, and UPT sections other than those listed above are not supported.

D1.E.2 The AISI Steel Section Library

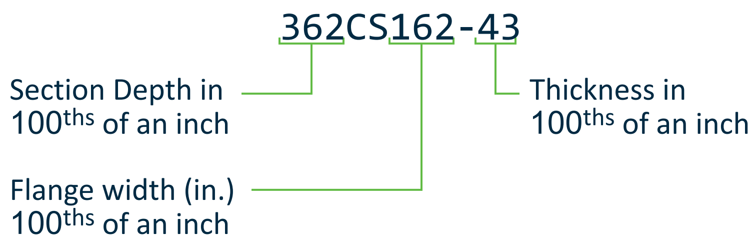
The command-line syntax for assigning steel sections from the AISI library is as explained below.

C-Section with Lips



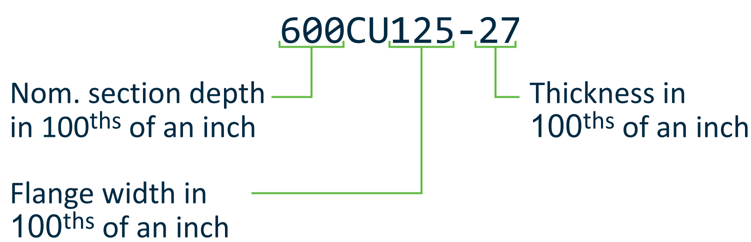

```
20 TO 30 TABLE ST 14CS3.75X135
33 36 TABLE ST 12CS1.625X102
42 43 TABLE ST 4CS4X060
```

Additionally, SSMA stud sections from the AISI tables are included in STAAD.Pro using a similar nomenclature, where CS is used instead of the AISI "S" designation.




```
44 TO 47 TABLE ST 362CS162-43
```

C-Section without Lips

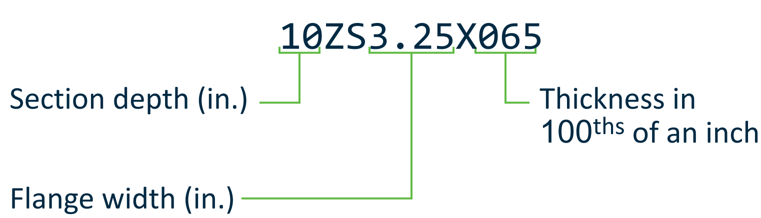


SSMA track sections from the AISI tables are included in STAAD.Pro using a similar nomenclature, where CU is used instead of the AISI "T" designation.

```
50 TO 60 TABLE ST 600CU125-27
32 33 TABLE ST 162CU125-18
21 28 TABLE ST 1000CU125-68
```

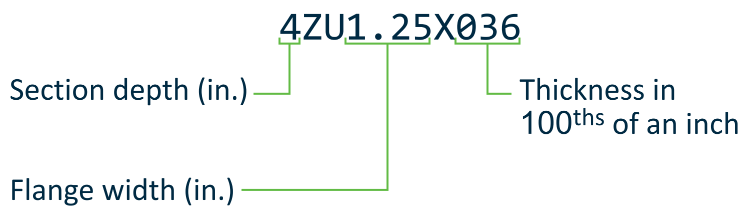
Note: The C-Sections without lips in the format of 3CU1.25X057 can be found in the **US/AISI Coldformed Legacy Profiles** database.

Z-Section with Lips



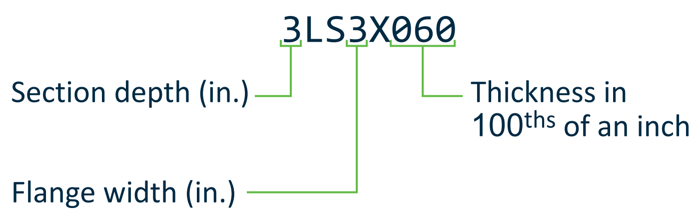

```
1 3 4 TABLE ST 12ZS3.25X105
33 45 TABLE ST 10ZS3.25X065
12 13 TABLE ST 6ZS2.25X059
```

Z-Section without Lips



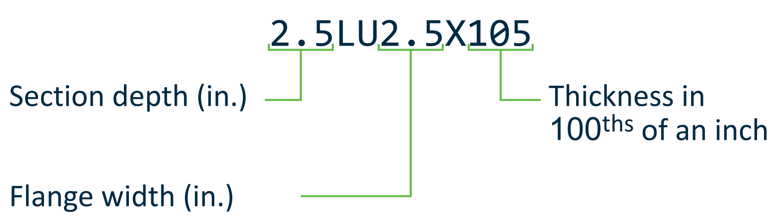

```
2 3 TABLE ST 8ZU1.25X105
4 5 TABLE ST 4ZU1.25X036
6 7 TABLE ST 1.5ZU1.25X048
```

Equal Leg Angles with Lips



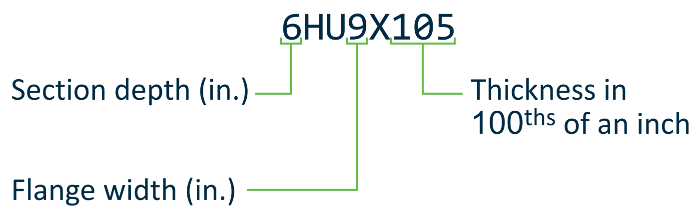

```
8 9 TABLE ST 4LS4X105
10 11 TABLE ST 3LS3X060
12 13 TABLE ST 2LS2X075
```

Equal Leg Angles without Lips




```
1 5 TABLE ST 4LU4X135
7 8 TABLE ST 2.5LU2.5X105
4 9 TABLE ST 2LU2X060
```

Hat Sections without Lips




```
4 8 TABLE ST 10HU5X075
5 6 TABLE ST 6HU9X105
1 7 TABLE ST 3HU4.5X135
```

D1.E.3 Design Procedure

Design Methodology

The AISI S100-16 follows a capacity-based design procedure. LRFD and ASD are two methods specified in the specification.

Structural members are designed to have strength such that the available strength (factored resistance) Ra, equals or exceeds the required strength (effect due to factored loads), R. Use the `METHOD` parameter to indicate which design method the program should use (the default method is LRFD).

- Allowable Strength Design (ASD): $R \leq R_a\ ,\ \ R_a = \ R_n\ /\ \Omega$
- Load and Resistance Factor Design (LRFD): $R_u \leq R_a\ ,\ \ R_a = \ \phi R_n$

Table 87: Saftey and Resistance factors in AISI 2016

Force	LRFD: ϕ Factor	ASD: Ω Value
Tension	0.90	1.67
Compression	0.85	1.80
Flexure	0.95 for box sections 0.90 for other sections	1.67
Shear	0.95	1.60

Tension Resistance

Design of the members subject to axial tension is performed as per Chapter D of AISI S100-16. The tensile strength is calculated with due consideration to both tensile yielding and tensile rupture effects. The design tensile strength, $\phi_t T_n$,(LRFD) and the allowable tensile strength, T_n/Ω_t, (ASD) will be calculated as per Chapter D2 and D3 of the code. The nominal tensile strength, T_n shall be the lower value obtained per the limit states of tensile yielding in the gross section and tensile rupture in the net section.

You can control the net area used to calculate the tensile rupture strength by assigning the `NSF` parameter.

Compression Resistance

Design of the members subject to axial compression is performed as per chapter E of AISI S100-16. The nominal compressive strength is calculated considering compression yielding, global (flexural, flexural-torsional, and torsional) buckling, local buckling, and distortional buckling. The minimum of these three criteria will be used to calculate the compression capacity or strength. The design compressive strength, $\phi_c P_n$, (LRFD) and the allowable compressive strength, P_n / Ω_c, (ASD) will be calculated as per chapter E2, E3 and E4 of the specification.

For the calculation of global compression capacities, section E2.1 is applicable for double channels front-to-front. Since this is *not* a pure box (but like a box), the program does not use E2.1.1. Rather, this built-up shape is treated like a typical section that is not subject to FTB and uses Cl. E2.1-1.

Torsional and flexural torsional buckling are *not* considered in the determination of compression capacity for L sections (with and without lips) designed about geometric axes. Rather, it is assumed that these members have restraints against torsional and flexural torsional buckling, which is typical in practice.

Flexure Resistance

Design of the members subject to bending about principal axes is performed as per Chapter F of AISI S100-16. The specification applies for bending about one principal axis only.

Note: For Z-shapes (zee and zee with lips), bending is checked about centroidal axes, axes passing through or perpendicular to the web

It is assumed that the member is loaded in a plane parallel to the axis that passes through the shear center or is restrained against twisting.

The nominal flexure resistance is calculated considering flexural yielding, global (lateral-torsional) buckling, local buckling, and distortional buckling. The smallest of these criteria will be used to calculate the flexural capacity or strength. The design flexural strength, $\phi_b M_n$, (LRFD) and the allowable flexural strength, M_n / Ω_b, (ASD) will be calculated as per chapter F2, F3 and F4 of the specification.

For calculation of yielding and global (lateral-torsional) buckling, Cl. F2.1.4 is used for double channel front-to-front section.

Shear Resistance

Design of the members subject to shear is dealt with by the module as per chapter G of AISI S100-16. The nominal shear strength, V_n, is calculated for flexural members without transverse stiffeners. The shear buckling force, V_{cr}, is calculated for for the web of members with transverse stiffeners.

The spacing of transverse stiffeners, if present, is given using the `STIFF` parameter.

The program ignores any spacing values specified for closed sections such as double channel front-to-front.

Interaction Checks

The program checks members subject to axial force and flexure about one or both axes as well as flexure and shear.

Excluded Checks

The following check are *not* made as part of the AISI S100-16 design:

- connection design

- design for torsion
- design for stability, ponding, fatigue, or corrosion effects
- web crippling checks
- combined checks for bending and web crippling
- combined checks for bending and torsion
- design for serviceability

Related Links

- *V. AISI 2016 CU Nominal and Local Axial Capacity* (on page 5917)
- *V. AISI 2016 CU Nominal Moment Capacity* (on page 5926)
- *V. AISI 2016 Cylindrical Tubular Section* (on page 5934)
- *V. AISI 2016 Hat Section* (on page 5941)

D1.E.4 Code Checking and Member Selection

The following two design modes are available:

Code Checking

The program compares the resistance of members with the applied load effects, in accordance with the LRFD and ASD methods of the AISI code. Code checking is carried out for specified locations via the `SECTION` command or the `BEAM` parameter. The results are presented in a form of a PASS/FAIL identifier and a ratio of load effect to resistance for each member checked compared to the `RATIO` parameter. You may choose the degree of detail in the output data by setting the `TRACK` parameter.

Member Selection

You may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

D1.E.5 Design Parameters

The following table contains the input parameters for specifying values of design variables and selection of design options.

Tip: When using the STAAD.Pro Editor, you can right-click and select **Insert Snippet** from the pop-up menu and then select **AISI 2016 Design** from the snippets list. This will insert the Parameter block with the commonly used parameters set to their defaults for all members. Refer to I. To insert a code snippet (on page 2051) for details on using snippets.

Table 88: AISI S100-16 Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description

CODE	-	Must be specified as one of the following to invoke design per AISI S100-16.
		`AISI`
		`AISI 2016`
		`AISI S100 2016`
		Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

AXIS	0	This flag allows you to perform the design of members about the local axes otherwise the design will be performed for the principal axes. 0) design considering principal axes 1) design considering local axes
BEAM	1.0	0.0 = design at start and end nodes and those locations specified by the `SECTION` command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. **Note:** See D1.A.6 Design Parameters (on page 1100).
CAN	0	0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see Note 9 in D1.A.6 Design Parameters (on page 1100))

Parameter Name	Default Value	Description
DFF	none (mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
FLX	0	Parameter for specifying the lateral-torsional restraint condition for a single angle. 0) Restraint not provided 1) Restraint provided
FU	58 ksi	Ultimate tensile strength of steel in current units.
FYLD	36 ksi	Yield strength of steel in current units.
KT	1.0	K value for flexural-torsional buckling.
KY	1.0	Effective length factor for overall column buckling about the local y-axis; used to compute the KL/r ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value.

Parameter Name	Default Value	Description
KZ	1.0	Effective length factor for overall column buckling about the local z-axis; used to compute the KL/r ratio for determining the capacity in axial compression. Values can range from 0.01 (for a column completely restrained against buckling) to any large value.
LT	Member Length	Unbraced length used in computing KL/r for twisting, in current units of length.
LY	Member Length	Maximum distance between the lateral restrained points in Local Y axis. Length used to calculate slenderness ratio for buckling about the local y-axis.
LZ	Member Length	Maximum distance between the lateral restrained points in Local Z axis. Same as LY, but in the local z-axis.
METHOD	LRFD	Used to specify LRFD or ASD design methods. The various design factors considered are summarized in Table 31 for LRFD and in Table 32 for ASD method.
NSF	1.0	NSF is the Net Section Factor as used in most of the steel design codes in STAAD.Pro. It is defined as the Ratio of “Net cross section area” / “Gross section area” for tension member design.
RATIO	1.0	Permissible ratio of the actual to allowable capacities.
SSY	0	0) Sway or Unbraced along local Y axis 1) Braced along local Y axis
SSZ	0	0) Sway or Unbraced along local Z axis 1) Braced along local Z axis

Parameter Name	Default Value	Description
STIFF	Member Length	Spacing of transverse stiffeners **Note:** This will be used for shear capacity calculation and combine shear and bending interaction. Only used if TSA is set to 1.
TRACK	0	Specifies the amount of detail included in design output. 0) Suppress all member capacities. 1) Print all member capacities. 2) Print full member design details.
TSA	0	Specifies whether the bearing and intermediate transverse stiffeners are present. If set to 1, the program uses more liberal and set of interaction equations found in AISI S100 2016 i.e. Eq.H 2-1. 0) Beams with unreinforced webs 1) Beams with transverse web stiffeners The program uses the distance given in the STIFF parameter.
UNB	Member Length	Unsupported length of the bottom flange for calculating flexural strength. Will be used only if compression is in the bottom flange. See Note 2 below.
UNT	Member Length	Unsupported length of the top flange for calculating flexural strength. Will be used only if compression is in the top flange. See Note 2 below.

Notes

1. When BEAM is 1.0 (default), the design is performed at 13 evenly spaced points along the length of the beam, including start and end points (i.e., 1/12th points or at ends of 12 equal length segments).

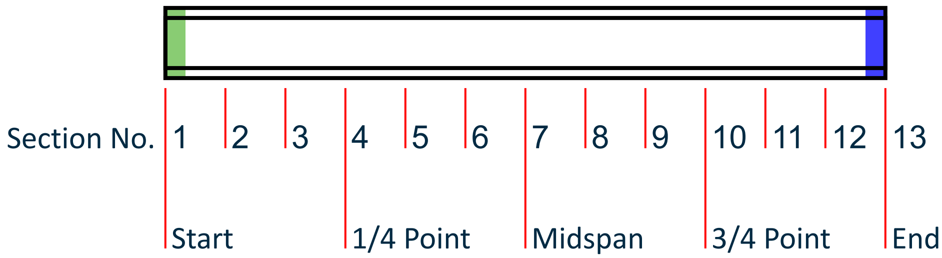


Figure 142: The default sections for design when BEAM 1.0 is used

When `BEAM` is 0.0, the start and ends along with up to three locations specified in TR.41 Section Specification (on page 2665) are designed.

2. Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if `SET Z UP` is used).
3. For description of the deflection check parameters `DFF`, `DJ1`, and `DJ2`, refer to the Notes section of D1.B.1.2 Design Parameters (on page 1134).
4. The program will calculate the distortional capacities if that is applicable for assigned section type.
5. The Cb value will be calculated based on full span of an analytical member irrespective of unrestrained length specified by the user.
6. When performing the deflection check, you can choose between two methods. The first method, defined by a value `0` for the `CAN` parameter, is based on the local displacement. See TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement.

   If the `CAN` parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by `DJ1` (or in the absence of `DJ1`, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at `DJ2` or the end node of the member.

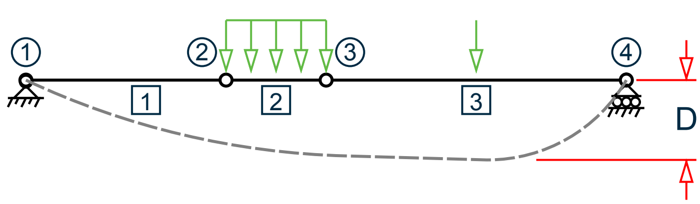
   Compute Delta = $\sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}$

   Compute Length = distance between `DJ1` and `DJ2` or, between start node and end node, as the case may be.

   Then, if `CAN` is specified a value 1, dff = L/Delta

   Ratio due to deflection = `DFF`/dff
7. If `CAN = 0`, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different.

   For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Also the straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured. Thus, for all three members here, `DJ1` should be "1" and `DJ2` should be "4".



D is equal to the maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

D1.F. American Codes - Concrete Design per ACI 318

Concrete member and floor design per ACI 318-14, ACI 318-11, ACI 318-08, ACI 318-05 ACI 318-02 and ACI 318-99 *Building Code Requirements for Structural Concrete* is available in STAAD.Pro.

Note: To use the ACI 318 design code for a specific publication year, refer to Design Operations.

D1.F.1 Design Operations

STAAD.Pro has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current edition of ACI 318 (unless a previous version is specified).

The design requirements need to be contained within a concrete design block, include the code reference, any modifications to design parameters and instructions as to which objects are to be designed.

```
START CONCRETE DESIGN
CODE code_ref
parameters values
design_instructions
END CONCRETE DESIGN
```

The versions of the ACI 318 code implemented can be referred to as follows:

Code Edition	*code_ref*	Design Beams	Design Columns	Design Slabs
2014	ACI ACI 2014	Yes	Yes	
2011	ACI 2011	Yes	Yes	
2008	ACI 2008	Yes	Yes	Yes
2005	ACI 2005	Yes	Yes	Yes
2002	ACI 2002	Yes	Yes	Yes
1999	ACI 1999	Yes	Yes	Yes

For a list of the *parameters* that can be used for design, see D1.F.3 Design Parameters (on page 1201).

Note: Not all parameters apply to all versions of the ACI 318 design code.

design_instructions:

```
DESIGN BEAM { member_list | ALL }
```

```
DESIGN COLUMN { member_list | ALL }
```

```
DESIGN PLATE { element_list | ALL }
```

See also TR.53 Concrete Design Specifications (on page 2684).

Example

An partial input with an example of beam design input for ACI 318-14:

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2014
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN BEAM 1 7 10
END CONCRETE DESIGN
```

D1.F.2 Section Types for Concrete Design

The current version of ACI 318 in STAAD.Pro supports the design of beams and columns defined as follows.

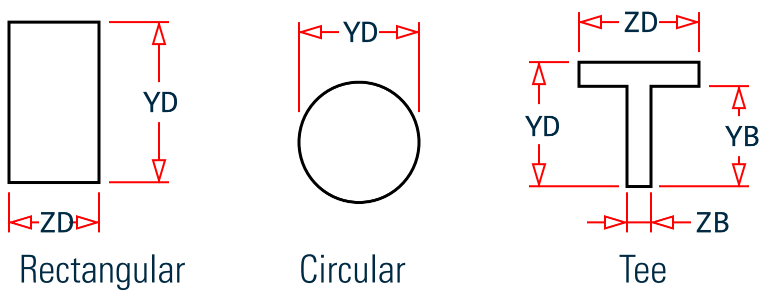


Figure 143: Section types for concrete design

Beam Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic
2. Tee Section
   a. Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)

Column Design – uses a 2D beam element if specified with the following properties:

1. Rectangular
   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic
2. Tee Section

   a. Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)

3. Circular

   a. Defined as Prismatic section with YD (circumference)

Related Links

- *TR.20.2 Prismatic Property Specification* (on page 2263)

D1.F.2.1 Section Types Supported for ACI 318-99 – ACI 318-11

Design using ACI 318-11, ACI 318-08, ACI 318-05, ACI 318-02, or ACI 318-99 in STAAD.Pro supports the design of beams, columns, or slabs defined as follows.

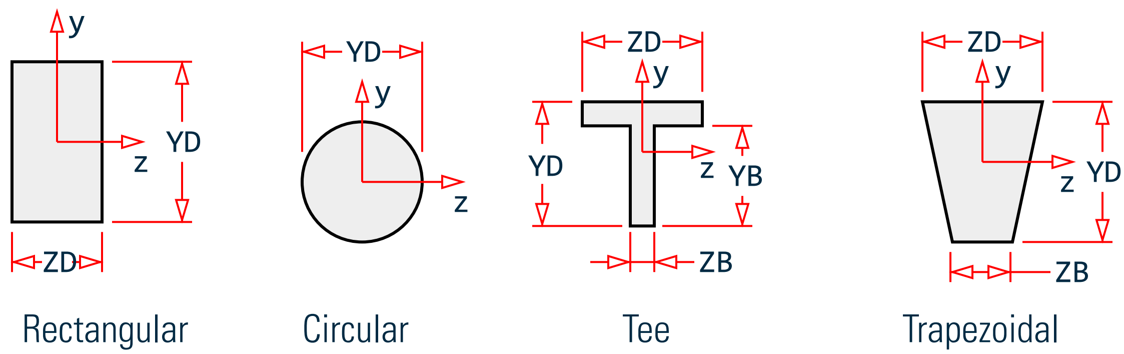


Figure 145: Section types for concrete design

Beam Design – uses a 2D beam element if specified with the following properties:

1. Rectangular

   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic

2. Tee Section

   a. Defined as a Prismatic section with YD (overall depth), YB (depth of stem), ZD (width of flange), and ZB (width of stem)

3. Trapezoidal

   a. Defined as Prismatic section with YD (depth), ZD (width at top), and ZB (width at bottom).

Column Design – uses a 2D beam element if specified with the following properties:

1. Rectangular

   a. Defined as a Prismatic section with YD (depth) and ZD (width)
   b. Defined in a User Table, type: prismatic

2. Circular

   a. Defined as Prismatic section with YD (circumference)

Floor Slab Design - uses a 2D plate element

1. 3-noded plate with thickness of element is equal to the thickness of the slab

2. 4-noded plate with thickness of element is equal to the thickness of the slab. All four nodes should be co-planar.

D1.F.3 Design Parameters

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-14 code. The commands that initiate the design of these is outlined in TR.53.3 Concrete Design Command (on page 2685). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

Note: Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.

Table 89: ACI 318 Design Parameters

Parameter Name	Default Value	Description
BDY	1.0	Column stiffness reduction factor for bending about the local Y axis. **Note:** Not used if parameter `MMAG` > 0 **Note:** Only used with ACI 318-11 and ACI 318-14.
BDZ	1.0	Column stiffness reduction factor for bending about the local Z axis. **Note:** Not used if parameter `MMAG` > 0 **Note:** Only used with ACI 318-11 and ACI 318-14.
BRDO	0	Bottom Rebar Detail Options 0) Increase number of bars, then increase bar size, then add layers 1) Increase the bar size, then increase the number of bars, then add layers **Note:** Only used with ACI 318-14.
CLB	1.5 in. (38 mm) for members*	Minimum cover from the base of the section to the bottom longitudinal reinforcement.
CLS	1.5 in. (38 mm)*	Minimum cover from the sides of the section to the side longitudinal reinforcement. See Note a (on page 1205) and Note c (on page 1206) below.

Parameter Name	Default Value	Description
CLT	1.5 in. (38 mm) for members*	Minimum cover from the top of the section to the top longitudinal reinforcement.
CRDO	0	Design options for main reinforcement in columns 0) Increase the number of bars and the bar size 1) Increase the bar size and then the number of bars **Note:** Only used with ACI 318-14.
FC	4,000 psi (28 N/ mm^2)*	Compressive strength of concrete.
FYMAIN	60,000 psi (420 N/mm^2)*	Yield stress for main longitudinal reinforcing steel.
FYSEC	60,000 psi (420 N/mm^2)*	Yield stress for transverse reinforcing steel.
LATR	0 (No)	Is this member subject to lateral loading? 0) No 1) Yes This will be used in determining the development length requirements, in ACI 318-14 clause 9.7.3.8. This parameter is only applicable for beams. **Note:** Only used with ACI 318-14.
LWF	0	Specifies the aggregate composition of the concrete. 0) Normal weight 1) Sand light weight 2) All light weight
MAXMAIN	#18 (Imperial), 57M (Metric)	Maximum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** When using ACI 318-14, the maximum size used in specific positions can be individually set. See detailing parameters below.
MIMB	#4 (Imperial), 13M (Metric)	Minimum main rebar number (size) at the bottom face **Note:** Only used with ACI 318-14.

Parameter Name	Default Value	Description
MIMS	#4 (Imperial), 13M(Metric)	Minimum main rebar number (size) at for skin (side) bars **Note:** Only used with ACI 318-14.
MIMT	#4 (Imperial), 13M (Metric)	Minimum main rebar number (size) at the top face **Note:** Only used with ACI 318-14.
MINMAIN	#4 (Imperial), 13M (Metric)	Minimum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below.
MINSEC	#4 (Imperial), 13M (Metric)	Minimum size of reinforcing bar to be used for transverse reinforcement.
MMAG	1.0	Factor to magnify the analysis moments used for column design. **Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ.
MXMB	#18 (Imperial), 57M (Metric)	Maximum main rebar number (size) at the bottom face **Note:** Only used with ACI 318-14.
MXMS	#18 (Imperial), 57M (Metric)	Maximum main rebar number (size) for skin (side faces) bars **Note:** Only used with ACI 318-14.
MXMT	#18 (Imperial), 57M (Metric)	Maximum main rebar number (size) at the top face **Note:** Only used with ACI 318-14.
RCOAT	0	Rebar coating type: 0) No coating 1) Epoxy coating 2) Galvanized **Note:** Only used with ACI 318-14.
REINF	0	Column transverse reinforcement type. 0) links (tied column) 1) spiral reinforcement

Parameter Name	Default Value	Description
RHOMIN	0.01	Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code. **Note:** This parameter is not used with ACI 318-14.
SKY	1.0	Effective length factor for moment magnification procedure calculations for columns, in the local Y direction. When the SWY parameter is 0 (i.e., not braced), then $SKY \geq 1.0$. When SWY is 1 (i.e., braced), then $0.5 \leq SKY \leq 1.0$. **Note:** Only used with ACI 318-11 and ACI 318-14.
SKZ	1.0	Effective length factor for moment magnification procedure calculations for columns, in the local Z direction. When the SWY parameter is 0 (i.e., not braced), then $SKZ \geq 1.0$. When SWY is 1 (i.e., braced), then $0.5 \leq SKZ \leq 1.0$. **Note:** Only used with ACI 318-11 and ACI 318-14.
SLY	n/a	Number of the load case that applies sidesway load in the local Y axis for column design. The moments from this case are used to magnify the local Y axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting. **Note:** This parameter is not used if MMAG has been specified. **Note:** Only used with ACI 318-11 and ACI 318-14.
SLZ	n/a	Number of the load case that applies sidesway load in the local z axis for column design. The moments from this case are used to magnify the local Z axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the SWY parameter setting. **Note:** This parameter is not used if MMAG has been specified. **Note:** Only used with ACI 318-11 and ACI 318-14.

Parameter Name	Default Value	Description
SQY	0.2	Stability index, Q, in the local Y axis as defined in ACI 318 section 6.6.4.3(b). **Note:** Only used with ACI 318-11 and ACI 318-14.
SQZ	0.2	Stability index, Q, in the local Z axis as defined in ACI 318 section 6.6.4.3(b). **Note:** Only used with ACI 318-11 and ACI 318-14.
SWY	0	Braced against sidesway: 0) Not braced 1) Braced **Note:** Use only when `MMAG` parameter is equal 0. **Note:** Only used with ACI 318-11 and ACI 318-14.
TRACK	0.0	Output detail 0) Design and report only the steel area requirements 1) Perform detailed design and report bar layouts 2) Perform detailed design and report all details
TRDO	0	Top Rebar Detail Options 0) Increase number of bars, then increase the bar size, then add layers 1) Increase the bar size, then increase the number of bars, then add layers
TRN	0	Specifies whether a member is subject to transverse loads between supports. This parameter influences the moment magnification calculations per CL 6.6.4.5.3. 0) True 1) False

Notes

a. The value used when specifying the `CLS` parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is `CLS` plus half the diameter of the main bar.

b. The following parameters are *not* applicable to the ACI 318-14 (only to ACI 318-11 and earlier):

DEPTH
EFACE
RHOMAIN
SFACE
WIDTH
NSECTION

Note: In ACI 318-14, members are always designed with five sections. The parameter `NSECTION` used with older editions of the code has no effect.

c. `MAXMAIN` and `MINMAIN` are now supplemented by new set of parameters like `MXMB`, `MIMC` etc. You can assign values either in the old parameters (like `MAXMAIN`) or in the new parameters (like `MXMT`, `MXMB`, `MXMC`). If you choose to use older parameters like `MAXMAIN` and `MINMAIN`, the program will store this input into newer parameters like `MXMB`, `MIMC` etc. internally. However, if you use both older and newer parameters, only the newer parameter values are accepted by the program.

Related Links
TR.31.2.16 IBC 2015 Seismic Load Definition

- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *TR.53.2 Concrete Design-Parameter Specification* (on page 2685)

D1.F.3.1 ACI 318-2011 Design Parameters

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-11 code. The commands that initiate the design of these is outlined in TR.53.3 Concrete Design Command (on page 2685). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

Note: Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.

Table 90: ACI 318 2011 Design Parameters

Parameter Name	Default Value	Description
ALPHA	1.5 for rectangular sections, 2.0 for square sections	Exponent for Bresler Load Contour Method for Bixaial Interaction. **Note:** The value should in theory be specified differently for each load case. This can be done by performing multiple designs with different load cases and choosing a value that is appropriate for all those cases.

Parameter Name	Default Value	Description
BDY	1.0	Column stiffness reduction factor for bending about the local Y axis. **Note:** Not used if parameter MMAG > 0 **Note:** Only used with ACI 318-11 and ACI 318-14.
BDZ	1.0	Column stiffness reduction factor for bending about the local Z axis. **Note:** Not used if parameter MMAG > 0 **Note:** Only used with ACI 318-11 and ACI 318-14.
CLB	1.5 in. (38 mm) for beams* 0.75 in. (19 mm) for plate elements*	Minimum cover from the base of the section to the bottom longitudinal reinforcement. See Note c (on page 1211) below.
CLS	1.5 in. (38 mm)*	Minimum cover from the sides of the section to the side longitudinal reinforcement. See Note a (on page 1205) and Note c (on page 1211) below.
CLT	1.5 in. (38 mm) for beams* 0.75 in. (19 mm) for plate elements*	Minimum cover from the top of the section to the top longitudinal reinforcement. See Note c (on page 1211) below.
DEPTH	YD*	Depth of the member measured in the local Y axis. **Note:** This parameter is not used with ACI 318-14.
EFACE	0.0*	Face of support location at end of beam. If specified, the shear force at end is computed at a distance of EFACE + d from the end joint of the member. **Note:** See D1.F.4.3 Shear and Torsion Design (on page 1217) for additional information. **Note:** This parameter is not used with ACI 318-14.
FC	4,000 psi (28 N/mm^2)*	Compressive strength of concrete.
FYMAIN	60,000 psi (420 N/mm^2)*	Yield stress for main longitudinal reinforcing steel.

Parameter Name	Default Value	Description
FYSEC	60,000 psi (420 N/mm^2)*	Yield stress for transverse reinforcing steel.
LWF	1.0	Modification factor, λ, for lightweight concrete as specified in ACI i318-08, cl. 8.6.1. Valid entries are between 0.75 and 1.0, inclusive. Used as a reduction factor for the mechanical properties of lightweight concrete. **Note:** This parameter is used in ACI 318-2008, 2001, and 2014 only.
MAXMAIN	#18 bar (57 mm)	Maximum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** When using ACI 318-14, the maximum size used in specific positions can be individually set. See detailing parameters below. **Note:** See Note b (on page 1211) below.
MINMAIN	#4 bar (13 mm)	Minimum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below. **Note:** See Note b (on page 1211) below.
MINSEC	#4 bar (13 mm)	Minimum size of reinforcing bar to be used for transverse reinforcement. **Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below. **Note:** See Note b (on page 1211) below.
MMAG	1.0	Factor to magnify the analysis moments used for column design. **Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ.

Parameter Name	Default Value	Description
NSECTION	12	Number of equally spaced sections to be considered in finding critical moments for beam design. The valid range is from 12 to 20. This parameter applies to the design process rather than individual members and thus should not have any assigned member list. If more than one `NSECTION` parameter is defined, then highest value will be used.
REINF	0	Column transverse reinforcement type. 0) links (tied column) 1) spiral reinforcement
RHOMIN	0.01	Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code. **Note:** This parameter is not used with ACI 318-14.
SFACE	0.0*	Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE + d from the start joint of the member. **Note:** See D1.F.4.3 Shear and Torsion Design (on page 1217) for additional information. **Note:** This parameter is not used with ACI 318-14.
SKY	1.0	Effective length factor for moment magnification procedure calculations for columns, in the local Y direction. When the `SWY` parameter is 0 (i.e., not braced), then $SKY \geq 1.0$. When `SWY` is 1 (i.e., braced), then $0.5 \leq SKY \leq 1.0$. **Note:** Only used with ACI 318-11 and ACI 318-14.
SKZ	1.0	Effective length factor for moment magnification procedure calculations for columns, in the local Z direction. When the `SWY` parameter is 0 (i.e., not braced), then $SKZ \geq 1.0$. When `SWY` is 1 (i.e., braced), then $0.5 \leq SKZ \leq 1.0$. **Note:** Only used with ACI 318-11 and ACI 318-14.

Parameter Name	Default Value	Description
SLY	n/a	Number of the load case that applies sidesway load in the local Y axis for column design. The moments from this case are used to magnify the local Y axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the `SWY` parameter setting. **Note:** This parameter is not used if `MMAG` has been specified. **Note:** Only used with ACI 318-11 and ACI 318-14.
SLZ	n/a	Number of the load case that applies sidesway load in the local z axis for column design. The moments from this case are used to magnify the local Z axis moments for all design load cases as per the sway or non sway requirements. Calculations are dependent on the `SWY` parameter setting. **Note:** This parameter is not used if `MMAG` has been specified. **Note:** Only used with ACI 318-11 and ACI 318-14.
SQY	0.2	Stability index, Q, in the local Y axis as defined in ACI 318 section 6.6.4.3(b). **Note:** Only used with ACI 318-11 and ACI 318-14.
SQZ	0.2	Stability index, Q, in the local Z axis as defined in ACI 318 section 6.6.4.3(b). **Note:** Only used with ACI 318-11 and ACI 318-14.
SWY	0	Braced against sidesway: 0) Not braced 1) Braced **Note:** Use only when `MMAG` parameter is equal 0. **Note:** Only used with ACI 318-11 and ACI 318-14.

Parameter Name	Default Value	Description
TRACK	0.0	Output detail • Beam Design: 0) Critical moment will not be printed out with beam design report. 1) Critical moment will be printed out with beam design report 2) Print out required steel areas for all intermediate sections specified by `NSECTION`. • Column Design: 0) Prints out detailed design reports 1) Prints out column interaction analysis results in addition to TRACK 0.0 output. 2) Prints out a schematic interaction diagram and intermediate interaction values in addition to TRACK 2.0 results.
TRN	0	Transverse loads between supports 0 - False 1 - True **Note:** Only used with ACI 318-14.
WIDTH	ZD*	Width of the member measured in the local Z axis. **Note:** This parameter is not used with ACI 318-14.

Note: * These values must be provided in the current unit system being used.

Notes

a. The value used when specifying the `CLS` parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is `CLS` plus half the diameter of the main bar.

b. When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm, and 60 mm.

c. Clear cover values are set for the concrete Not exposed to weather or in contact with ground. Plates or slabs are assumed to be reinforced with bar number 11(in imperial units) or smaller.

d. Required for bar detailing in a physical member which might contain intermediate members along with cantilevers.

For internal members both `STRS` and `EDSP` should be set to 0.

e. In ACI 318-14, members are always designed with 5 sections. The parameter `NSECTION` used with older editions of the code has no effect.

D1.F.3.2 Pre ACI 318-2011 Design Parameters

The following outlines the parameters that are available to control the design of members as beams or columns per the ACI 318-99 through ACI 318-2008 codes. The commands that initiate the design of these is outlined in TR.53.3 Concrete Design Command (on page 2685). If any parameter is not specified and its value is required in the design, then it will use that specified as the default value in the tables below.

Note: Not all the parameters are used in all versions of the ACI 318 code. Ensure that all the necessary parameters for the version of the code are properly specified. Where practical, parameters used in earlier versions of the code will be supported in the newer versions. Some old parameters are not supported in newer versions of the code and new parameters have been added in newer versions of the code that provide greater flexibility in the design.

Table 92: ACI 318 2008, 2005, 2002, and 1999 Design Parameters

Parameter Name	Default Value	Description
ALPHA	1.5 for rectangular sections, 2.0 for square sections	Exponent for Bresler Load Contour Method for Bixaial Interaction. **Note:** The value should in theory be specified differently for each load case. This can be done by performing multiple designs with different load cases and choosing a value that is appropriate for all those cases. **Note:** This parameter is not used with ACI 318-1999.
CLB	1.5 in. (38 mm) for beams* 0.75 in. (19 mm) for plate elements*	Minimum cover from the base of the section to the bottom longitudinal reinforcement. See Note c (on page 1215) below.
CLS	1.5 in. (38 mm)*	Minimum cover from the sides of the section to the side longitudinal reinforcement. See Note a (on page 1205) and Note c (on page 1215) below.
CLT	1.5 in. (38 mm) for beams* 0.75 in. (19 mm) for plate elements*	Minimum cover from the top of the section to the top longitudinal reinforcement. See Note c (on page 1215) below.
DEPTH	YD*	Depth of the member measured in the local Y axis. **Note:** This parameter is not used with ACI 318-14.

Parameter Name	Default Value	Description
EFACE	0.0*	Face of support location at end of beam. If specified, the shear force at end is computed at a distance of EFACE + d from the end joint of the member. **Note:** See D1.F.4.3 Shear and Torsion Design (on page 1217) for additional information. **Note:** This parameter is not used with ACI 318-14.
FC	4,000 psi (28 N/ mm^2)*	Compressive strength of concrete.
FYMAIN	60,000 psi (420 N/mm^2)*	Yield stress for main longitudinal reinforcing steel.
FYSEC	60,000 psi (420 N/mm^2)*	Yield stress for transverse reinforcing steel.
LWF	1.0	Modification factor, λ, for lightweight concrete as specified in ACI i318-08, cl. 8.6.1. Valid entries are between 0.75 and 1.0, inclusive. Used as a reduction factor for the mechanical properties of lightweight concrete. **Note:** This parameter is used in ACI 318-2008, 2001, and 2014 only.
MAXMAIN	#18 bar (57 mm)	Maximum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** See Note b (on page 1215) below.
MINMAIN	#4 bar (13 mm)	Minimum size of reinforcing bar to be used for longitudinal reinforcement. **Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below. **Note:** See Note b (on page 1215) below.
MINSEC	#4 bar (13 mm)	Minimum size of reinforcing bar to be used for transverse reinforcement. **Note:** When using ACI 318-14, the minimum size used in specific positions can be individually set. See detailing parameters below. **Note:** See Note b (on page 1215) below.

Parameter Name	Default Value	Description
MMAG	1.0	Factor to magnify the analysis moments used for column design. **Note:** For ACI 318-14, if this is set to 0, then the moments for the local Y and Z axis will be calculated independently using the parameters BDY and BDZ.
NSECTION	12	Number of equally spaced sections to be considered in finding critical moments for beam design. The valid range is from 12 to 20. This parameter applies to the design process rather than individual members and thus should not have any assigned member list. If more than one NSECTION parameter is defined, then highest value will be used.
REINF	0	Column transverse reinforcement type. 0) links (tied column) 1) spiral reinforcement
RHOMIN	0.01	Minimum reinforcement required in a concrete column. Enter a value between 0.0 and 0.08, where 0.08 = 8% reinforcement; the maximum allowed by the ACI code. **Note:** This parameter is not used with ACI 318-14.
SFACE	0.0*	Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE + d from the start joint of the member. **Note:** See D1.F.4.3 Shear and Torsion Design (on page 1217) for additional information. **Note:** This parameter is not used with ACI 318-14.

Parameter Name	Default Value	Description
TRACK	0.0	Output detail • Beam Design: 0) Critical moment will not be printed out with beam design report. 1) Critical moment will be printed out with beam design report 2) Print out required steel areas for all intermediate sections specified by NSECTION. • Column Design: 0) Prints out detailed design reports 1) Prints out column interaction analysis results in addition to TRACK 0.0 output. 2) Prints out a schematic interaction diagram and intermediate interaction values in addition to TRACK 2.0 results.
WIDTH	ZD*	Width of the member measured in the local Z axis. **Note:** This parameter is not used with ACI 318-14.

Notes

a. The value used when specifying the CLS parameter for column design is taken to be the clear cover for the longitudinal bars in a column. It is not taken as the clear cover for the tie bars. Therefore, the distance from the edge of the column to the centerline of the first row of longitudinal bars is CLS plus half the diameter of the main bar.

b. When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm, and 60 mm.

c. Clear cover values are set for the concrete Not exposed to weather or in contact with ground. Plates or slabs are assumed to be reinforced with bar number 11(in imperial units) or smaller.

d. Required for bar detailing in a physical member which might contain intermediate members along with cantilevers.

For internal members both STRS and EDSP should be set to 0.

D1.F.4 Beam Design

Beams are designed at a series of locations along their length for the moment about the local Z axis (MZ), shear force in the local Y and Z axes (FY and FZ), torsional moment (MX), and axial force (FX) for an envelope of the forces from a specified collection of load cases and combinations.

The envelope is created at each design section from a collection of the max/min forces in all degrees of freedom for all the load cases and/or combinations that have been included in the design. As there are 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) and to capture the maximum and minimum effects, a total of 12 sets of loads are designed for. Each set will include the maximum or minimum of one force such as Max FX along with all the other associated forces in the load case/combination.

For each member that is designed as a beam, the forces from the current `LOAD LIST` (see TR.39 Load List Specification (on page 2662)) will be used to create an envelope of forces.

This envelope of forces is then used to determine a required area of longitudinal rebar at each of the key section locations. Five sections are defined on each member: the start, quarter point, mid-point, three-quarter point, and end point. This required area is then used determine an optimized bar arrangement that meets the code requirements, including detailing rules. Once the longitudinal bars have been designed and detailed, the program then designs the transverse steel based on the code provisions. Note that in case of torsion loads, the effect of the applied torsion will be catered for while designing the longitudinal steel as well as the transverse steel.

The required area of reinforcements at every cross section is first calculated. In order to then arrive at an optimized detailed bar layout, the program starts with the bar size defined in the `MINMAIN` parameter (or `MIMT` parameter for top bars and `MIMB` parameter for bottom bars, if specified)

The program will then iterate though increasing the number of bars or the size of bars dependent on the values of the `BRDO` and `TRDO` parameters.

If additional skin bars (i.e., side face bars) are needed for spacing requirements, these will be added.

Any detailing rules such as development lengths etc will be considered and the final design output would indicate whether a given set of bars are not be anchored or not.

Assumed Values Used

Most of the values are determined from the beam data, however, some values of note include:

- Young's modulus of rebar (E_s) is taken as 29,000 ksi
- Young's modulus of concrete is calculated from the given value of F'_c . But if $F'_c > 10{,}000$, then E_c is limited to 3,605 psi.
- Density of concrete is taken as 145 pcf (0.083912 kip-in^3)

Scope

Note that deep beams –where the span < 4× the beam depth– are not designed. A warning is reported in the output.

Related Links

- *V. ACI 318-14 Tee Beam* (on page 6192)
- *V. ACI 318-14 Rectangular Beam without Torsion* (on page 6196)

D1.F.4.1 Strength and Ductility Design

The strength design is performed in two steps considering the longitudinal reinforcement design for axial-bending in an initial step, and the shear and torsion design including the longitudinal reinforcement for torsion tension in a latter pass together with the transverse design.

The member resistance is calculated based on strain compatibility and interaction surface calculations.

The following conditions should be checked (9.5.1.1)

$$\phi Mn \geq Mu$$
$$\phi Vn \geq Vu$$

$\phi Tn \geq Tu$
$\phi Pn \geq Pu$

ϕ is determined according to 21.2

The ratio of neutral axis depth to the depth of the furthest rebar in tension is limited such that the section strain at the location of the maximum rebar depth is a minimum of 0.004.

Application of the ductility checks are limited to cross sections with net axial load (compression) less than $0.10f´c\ Ag$, in accordance with section 9.3.3.1.

The maximum strain in the tension reinforcement is calculated and compared with the Code limit. In order to reduce the time involved in the axial-bending calculations, the approximation may currently have a maximum difference with the more precise values of 12% mostly in biaxial cases, the approximation will be much better for other cases.

D1.F.4.2 Minimum Flexural Reinforcement

The code specifies to dispose a minimum area of flexural reinforcement where tension reinforcement is required.

As a minimum bound, the condition that is given in 9.6.2.1is used. Although it is intended only for prestressed beams, it may be used also in reinforced concrete beams and control in odd circumstances such as where the specified cover is extremely large. The bending strength of the section is designed to be at least $1.2\ Mcr$, where the cracking moment is assumed to be applied to each axis separately as uniaxial bending. No simultaneous biaxial moments are considered. The cracking moments about each axis are calculated to cause the stress in the extreme tension fiber to reach fr. In order to account for a possible varying of elastic modulus (Ec) values, the following formula is used ignoring the effect of the inertia product (Ixy):

$$M_{cr(region)} = \frac{\sigma_{cr(region)} E_c I_c}{E_{c(region)} c_{(region)}}$$

where

$\sigma_{cr(region)}$	=	the flexural tension strength for a region
$E_{c(region)}$	=	the modulus of elasticity for a specific region
$c_{(region)}$	=	the distance from the neutral axis to the extreme fiber of a region
$E_c I_c$	=	the integrated product of the inertia and elastic modulus for the entire section
fr	=	the modulus of rupture calculated in accordance with Clause 19.2.3.1
λ	=	calculated according to Clause 19.2.4.2

The cracking moment is individually calculated for each axis and only considers a particular axis if there is a considerable bending moment about that axis. The sign of the acting moment is also considered.

This minimum bound is considered only when the 9.6.1.2 or 9.6.1.3 is also applied

D1.F.4.3 Shear and Torsion Design

In the absence of torsion, the required shear bar density at cross section at each cross section will initially be calculated for shear along both the local Z and Y axes. Based on these required shear densities, the program will divide the member into one or more shear zones such that each zone (between the start and end cross sections) will require the same shear density. Once the shear zones have been identified, transverse bar links will be provided based on the bar size specified using the `MINSEC` parameter. In order to meet the required density demand, the program will start with the minimum number of legs and calculate the most optimum spacing for the transverse links. If the minimum / maximum spacing criteria cannot be met during this process, additional legs will be added and the spacings adjusted to optimize the design. Note that the program will use the same bar size for links to cater for shear along both the X a nd Z directions to maintain a practical design.

Note that the program will use the same bar size for all transverse bars to cater for shear requirements in both axes to maintain a practical design .

If the beam is also subject to torsional forces (MX), both the longitudinal steel as well the transverse steel will be designed to cater for the applied torsion, in addition to any longitudinal or transverse steel that would have been required for the bending/ shear effects. The torsion force increases the demand in the transverse due to pure shear and longitudinal direction due to warping effects.

See section F.7 for the clauses considered for shear and torsion design.

D1.F.4.4 Definition of Bar Positions

Both input parameters and design output refer to the positions of bars in beam sections as follows.

Note: The bars shown in these figures are representational only; they do not correspond to any examples in this section.

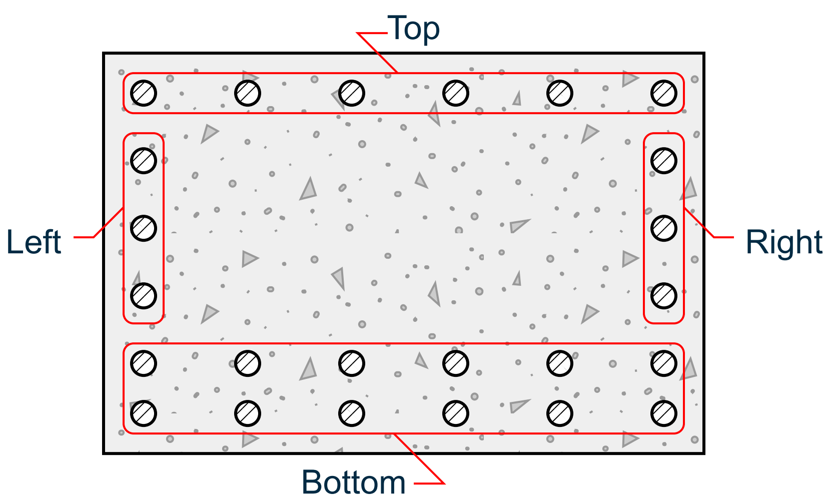


Figure 146: ACI 318-14 rectangular beam bar positions

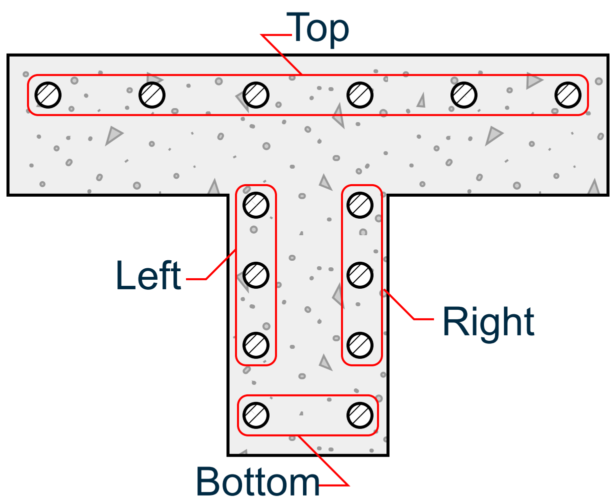


Figure 147: ACI 318-14 Tee beam bar positions

D1.F.4.5 Beam Design Output

The following report options are provided for the output for a beam design.

TRACK 0 Output

Setting the option `TRACK 0` will cause the program to produce the required area of longitudinal as well as transverse steel at 5 equally spaced locations –including member ends– along the member length.

Note: This option is meant to be considered for quick area of steel (As) requirement analysis and should *not* be considered as a safe, acceptable design.

Sample `TRACK 0` output:

Member	member number
Type	Beam
Shape	Either rectangular or Tee
As	the area of longitudinal steel required at each cross section
Asv/sv	the transverse steel density required at each cross section

```
                              STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                              ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)
   |-----------------------------------------------------------|
   |  Member :     12                                          |
   |  Type   : Beam                Shape  : Rectangular        |
   |-----------------------------------------------------------|
   |  AREA OF STEEL REQUIRED                                   |
   |-----------------------------------------------------------|
   |  Section -         |   1   |   2   |   3   |   4   |   5   |
   |-----------------------------------------------------------|
   |  Location          |  0.00 | 60.00 |120.00 |180.00 |240.00 |
   |  As(Longitudinal)|  7.17 |  4.46 |  4.94 |  4.46 |  3.95 |
```

```
 |  As/sv(Trans Y)  |  0.09 |  0.05 |  0.01 |  0.02 |  0.07 |
 |  As/sv(Trans Z)  |  0.00 |  0.00 |  0.00 |  0.00 |  0.00 |
 |-----------------------------------------------------------|
```

TRACK 1 Output

Setting the program to output the beam design with a `TRACK 1` option will perform full design and detailing operations and will report a summary of the design. A `TRACK 1` report will include the following sections:

Design Summary — This section provides a summary of the design including the design status (i.e., PASS/FAIL), member type, length, the critical ratio, the critical criteria, and relevant clauses.

Cross section details — This section provides shape type and dimensions of the cross section profile.

Longitudinal Bar Layout — The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of “yes” in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of “no” indicates that no additional extension to the bar is needed beyond the given dimension. The value given in “Distance from face”is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

Transverse Bar Layout — The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1217)) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value “Nums” is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

Note: This output will *not* report any warnings or errors from the design and member analysis. These are only reported in the `TRACK 2` output.

Sample beam design `TRACK 1` output:

```
                             STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                             ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)

Member :     13

DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Fail          Type    : Beam          Length:  216.000 |
| Critical Ratio :   1.000         Criteria: Torsion                        |
| Critical Clause:   9.5.3/9.5.4                                            |
| All Failures   :   N.A                                                    |
-----------------------------------------------------------------------------

CROSS SECTION
```

```
-------------------------------------------------------
| Shape: Rectangular | Width:   16.00 | Depth:   21.00 |
-------------------------------------------------------

LONGITUDINAL BAR LAYOUT
-----------------------------------------------------------------------
|                 |    Bars    |    Location     | Distance  |   Anchor  |
|        Position | Nums | Size | Start|    End   | From Face | Start End |
-----------------------------------------------------------------------
|           Right |    2 | #11  |   0.00| 216.00 |      2.20 |  Yes  Yes |
|             Top |    2 | #11  |   0.00| 216.00 |      2.20 |  Yes  Yes |
|             Top |    1 | #11  |121.73| 216.00 |      2.20 |   No  Yes |
|          Bottom |    2 | #11  |   0.00| 216.00 |      2.20 |  Yes  Yes |
|            Left |    2 | #11  |   0.00| 216.00 |      2.20 |  Yes  Yes |
-----------------------------------------------------------------------

TRANSVERSE BAR LAYOUT
----------------------------------------------------------------------------
|                              |  Asv Density  |      Rebar Specification    |
| Zone | Dir.|  From  |  To    | Reqd. | Prov. | Nums | Size | Spacing | Legs |
----------------------------------------------------------------------------
|    1 |  Y  |  0.00  | 54.00  |  0.06 |  0.07 |   10 |  # 4 |   6.00  |   2  |
|    1 |  Z  |  0.00  | 54.00  |  0.02 |  0.05 |    8 |  # 4 |   7.71  |   2  |
|    2 |  Y  | 54.00  |108.00  |  0.03 |  0.05 |    8 |  # 4 |   7.71  |   2  |
|    2 |  Z  | 54.00  |108.00  |  0.02 |  0.05 |    8 |  # 4 |   7.71  |   2  |
|    3 |  Y  |108.00  |216.00  |  0.10 |  0.12 |   34 |  # 4 |   3.27  |   2  |
|    3 |  Z  |108.00  |216.00  |  0.02 |  0.09 |   26 |  # 4 |   4.32  |   2  |
----------------------------------------------------------------------------

------------------------ Member:    13 Design Ends ------------------------
```

TRACK 2 Output

`TRACK 2` will produce the results of `TRACK 1` along with the details of the bars and their distribution at each cross section. This option will also report any of the relevant detailed design messages. The longitudinal bar details at each cross section will have information on the setting out of each bar at the specific cross section. This should serve as a useful tool to inspect the design in detail. A `TRACK 2` output would contain:

Design Summary	This section provides a summary of the design including the design status, i.e. PASS/FAIL, member type and length, the critical ratio, the critical criteria and relevant clauses.
Cross Section Details	This section provides the shape type and dimension(s) of the cross-section
Design Inputs	This section reports the material data for the steel and concrete including the yield strength for steel (FY), compressive strength of concrete (FC) and modulus of elasticity for both (Es and Ec) This section additionally includes details of the cover to the longitudinal bars from the top, bottom and side faces.
Critical Strength Results	This section reports the demand determined from the design envelope as described in D1.F.4 Beam Design (on page 1215), along with the capacity for forces applied in positive and negative directions. The ratio is then the value of the demand divided by the appropriate capacity.

Longitudinal Bar Details at Cross Sections

For each of the design cross sections, the longitudinal bars are reported for each face where they are required, top bottom and skin. Area of steel is reported as the optimum required from the design and that provided after additionally meeting the detailing requirements.

The total quantity of bars on the given face are reported as “No(s) bars,” the size of longitudinal bars and quantity of layers that the bars are to be distributed on. For example, if the number of bars reported is 10 and in 2 layers, then there will be 5 bars in each layer.

Longitudinal Bar Layout

The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of “yes” in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of “no” indicates that no additional extension to the bar is needed beyond the given dimension. The value given in “Distance from face”is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

Transverse Bar Layout

The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1217)) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value “Nums” is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

Note: This output will additionally report any warnings or errors from the design and member analysis.

Sample beam design TRACK 2 output:

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)

Member :     14

DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length:  240.000 |
| Critical Ratio :   0.832        Criteria: Shear Y                         |
| Critical Clause:   9.5.3/9.5.4                                            |
-----------------------------------------------------------------------------

CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   16.00 | Depth:   21.00 |
-------------------------------------------------------

DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc           4.000 |                      | Ec     0.360E+04 |
```

```
| Steel     | Fy(main)    60.000 | Fy(trans)    60.000 | Es     0.290E+05 |
| Cover     | Top          1.500 | Bottom        1.500 | Sides      1.500 |
-------------------------------------------------------------------------


CRITICAL STRENGTH RESULTS
-----------------------------------------------------------
| Category |   Demand   |Min Capacity|Max Capacity|  Ratio  |
-----------------------------------------------------------
| Axial    |       9.117|   -961.359 |    673.920 |   0.014 |
| Flexure  |   -4462.279|  -5808.514 |   5086.883 |   0.768 |
| Shear Y  |      97.255|   -116.842 |    116.842 |   0.832 |
| Shear Z  |      -0.636|    -22.857 |     22.857 |   0.028 |
| Torsion  |       0.000|      0.000 |      0.000 |   0.000 |
-----------------------------------------------------------

LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-----------------------------------------------------------------------------
| Distance | Position | Ast-reqd | Ast-prov | No(s)bars| Size | No of Layers |
-----------------------------------------------------------------------------
|    0.000 |      Top |    4.572 |    4.680 |    3     | #11 |       1       |
|          |   Bottom |    0.672 |    3.120 |    2     | #11 |       1       |
|          |     Left |    0.966 |    3.120 |    2     | #11 |       1       |
|          |    Right |    0.966 |    3.120 |    2     | #11 |       1       |
|   60.000 |      Top |    1.524 |    4.680 |    3     | #11 |       1       |
|          |   Bottom |    1.002 |    3.120 |    2     | #11 |       1       |
|          |     Left |    0.966 |    3.120 |    2     | #11 |       1       |
|          |    Right |    0.966 |    3.120 |    2     | #11 |       1       |
|  120.000 |      Top |    1.524 |    3.120 |    2     | #11 |       1       |
|          |   Bottom |    1.484 |    3.120 |    2     | #11 |       1       |
|          |     Left |    0.966 |    3.120 |    2     | #11 |       1       |
|          |    Right |    0.966 |    3.120 |    2     | #11 |       1       |
|  180.000 |      Top |    1.524 |    3.120 |    2     | #11 |       1       |
|          |   Bottom |    1.002 |    3.120 |    2     | #11 |       1       |
|          |     Left |    0.966 |    3.120 |    2     | #11 |       1       |
|          |    Right |    0.966 |    3.120 |    2     | #11 |       1       |
|  240.000 |      Top |    1.524 |    3.120 |    2     | #11 |       1       |
|          |   Bottom |    0.495 |    3.120 |    2     | #11 |       1       |
|          |     Left |    0.966 |    3.120 |    2     | #11 |       1       |
|          |    Right |    0.966 |    3.120 |    2     | #11 |       1       |
-----------------------------------------------------------------------------

LONGITUDINAL BAR LAYOUT
-----------------------------------------------------------------------
|                 |    Bars     |   Location     | Distance  |  Anchor   |
|        Position | Nums | Size | Start|    End  | From Face | Start End |
-----------------------------------------------------------------------
|           Right |    2 | #11  |  0.00| 240.00  |    2.20   |  Yes  Yes |
|             Top |    2 | #11  |  0.00| 240.00  |    2.20   |  Yes  Yes |
|             Top |    1 | #11  |  0.00| 108.54  |    2.20   |  Yes   No |
|          Bottom |    2 | #11  |  0.00| 240.00  |    2.20   |  Yes  Yes |
|            Left |    2 | #11  |  0.00| 240.00  |    2.20   |  Yes  Yes |
-----------------------------------------------------------------------

TRANSVERSE BAR LAYOUT
----------------------------------------------------------------------------
|                               |  Asv Density  |    Rebar Specification     |
| Zone | Dir.|  From |  To     | Reqd. | Prov. | Nums | Size | Spacing | Legs |
```

```
-------------------------------------------------------------------------------
|     1 |   Y  |   0.00 |120.00 |  0.12 |  0.14 |   42 |  # 4 |    2.93  |   2  |
|     1 |   Z  |   0.00 |120.00 |  0.03 |  0.10 |   32 |  # 4 |    3.87  |   2  |
|     2 |   Y  |120.00 |240.00 |  0.10 |  0.10 |   32 |  # 4 |    3.87  |   2  |
|     2 |   Z  |120.00 |240.00 |  0.03 |  0.10 |   32 |  # 4 |    3.87  |   2  |
-------------------------------------------------------------------------------

------------------------ Member:    14 Design Ends ------------------------
```

D1.F.4.6 Beam Design per ACI 318-11 and Earlier

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an NSECTION parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

Related Links

- *V. ACI 318-11 Rectangular Singly Reinforced Beam* (on page 6206)
- *V. ACI 318-08 Rectangular Singly Reinforced Beam* (on page 6210)
- *V. ACI 318-05 Rectangular Singly Reinforced Beam* (on page 6215)
- *V. ACI 318-02 Rectangular Beam* (on page 6221)
- *V. ACI 318-99 Beam and Column Reinforcement* (on page 6229)

D1.F.4.6.1 Cracked Moment of Inertia - ACI Beam Design

When beam design is done per ACI 318, STAAD will report the moment of inertia of the cracked section at the location where the design is performed. The cracked section properties are calculated in accordance with the equations shown below.

Rectangular Sections

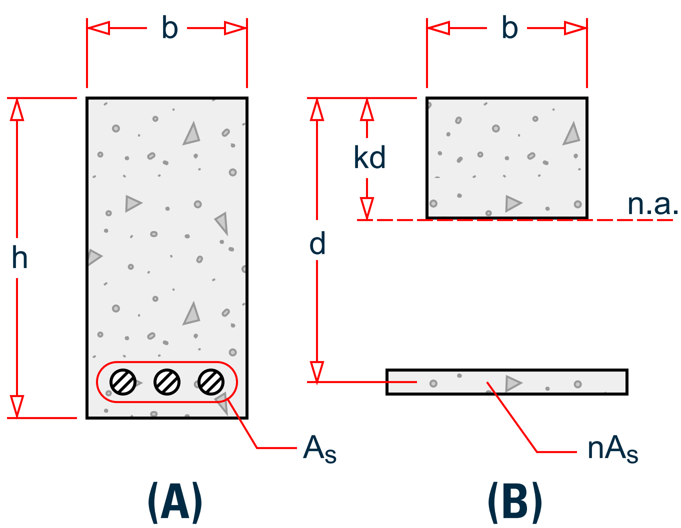


Figure 148: Gross section (A) and cracked transform section (B) for rectangular shapes

Without compression steel

$$n = E_s/E_c$$

$$B = \frac{b}{nA_I}$$

$$I_g = \frac{b \times h^3}{12}$$

$$kd = \frac{\sqrt{2d \times B + 1} - 1}{B}$$

$$I_{cr} = \frac{b(kd)^3}{3} + nA_s(d - kd)^2$$

Tee Shaped Sections

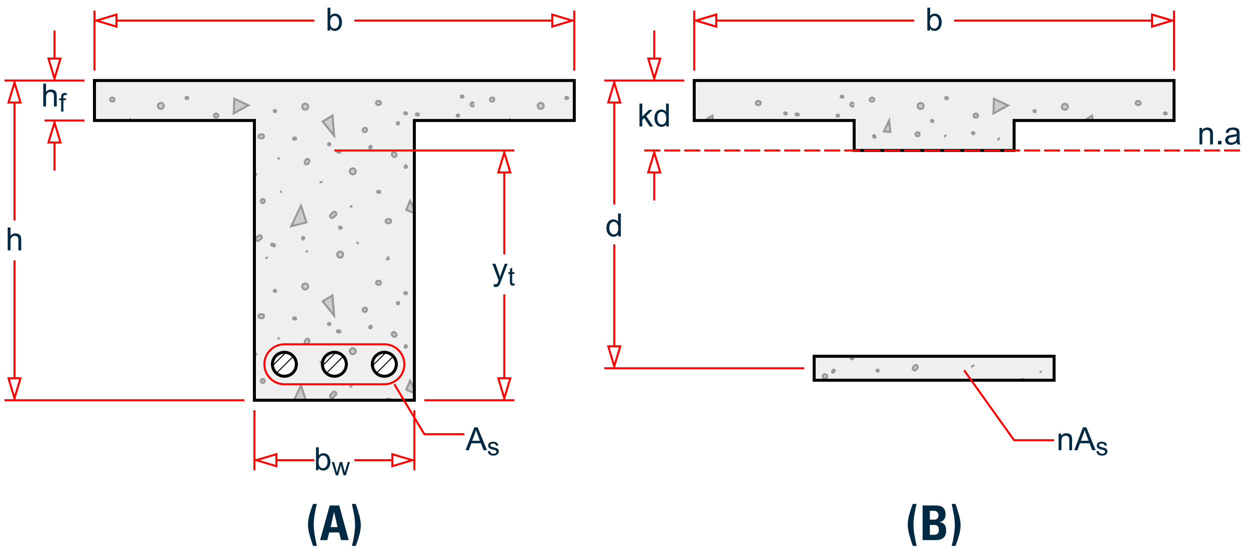


Figure 149: Gross and cracked transform sections for tee shapes without compression steel

Without compression steel

$$C = \frac{b_w}{nA_s}$$

$$f = \frac{h_f\left(b - b_w\right)}{nA_s}$$

$$y = h - \frac{1}{2}\frac{\left(b - b_w\right) \times h_f^2 + b_w \times h^2}{\left(b - b_w\right) \times h_f + b_w \times h}$$

$$kd = \frac{\sqrt{C\left(2d + h_f \times f\right) + (1 + f)^2} - \left(1 + f\right)}{C}$$

$$I_{cr} = \frac{\left(b - b_w\right) \times h_f^3}{12} + \frac{b_w(kd)^3}{3} + \left(b - b_w\right) \times h_f \times \left(kd - \frac{h_f}{2}\right)^2 + nA_s \times (d - kd)^2$$

See D1.F.4.5 Beam Design Output (on page 1219) for an example of output including the calculated cracked moment of inertia.

D1.F.4.6.2 Design of I-shaped beams per ACI-318

I-shaped sections can be designed as beams per the ACI 318 code. The property for these sections must be defined through a user table, I-section, or using the tapered specification. Information on assigning properties in this manner is available in TR.19 User Steel Table Specification (on page 2241) (I-section type) and TR.20.3 Tapered Member Specification (on page 2266) (Tapered I shape) of the Technical Reference manual.

From the standpoint of the analysis – determining member forces, nodal displacements and support reactions – the same set of facilities and rules which are applicable for any normal reinforced concrete frames or other structures can be used when I-sections or tapered concrete members are specified. In other words, there isn't anything unique or special to account for in the analysis model simply because I-shaped concrete beams are part of it.

From the standpoint of design, the following rules are applicable:

1. The member can be designed as a beam using the general principles explained in D1.F.4 Beam Design (on page 1215). It currently cannot be designed as a column. Design as a beam is done for flexure (MZ), shear (FY) and torsion (MX) just like that for rectangular, tee or trapezoidal beams. Axial forces (FX) are used during the capacity computations in shear and torsion. At each section along the length that the member is designed at, the depth at that section location is used for effective depth computation.
2. The program performs the following tests on the section dimensions before starting the design:
   - If the thickness of the web is the same as the width of the top and bottom flanges, the member is designed as a rectangular section.
   - If the thickness of the web is the same as the width of one of the flanges but not the other, the member is designed as a T-section or a rectangular section, depending on which side the compression due to bending is at.
   - If the web thickness does not match the width of either flange, design is done using the rules applicable for T-beams – one flange is in compression, the other in tension, and tensile capacity of concrete on the tensile side of the neutral axis is ignored.
   - The program is also able to design the beam as a doubly reinforced section if it is unable to design it as a single-reinforced section.
3. The parameters for designing these members are as shown in D1.F.3 Design Parameters (on page 1201) of this manual. Detailed output on design at individual section locations along the member length may be obtained by setting the TRACK parameter to 3.0.

An example for I-beam design is shown below.

```
STAAD PLANE I BEAM CONCRETE DESIGN PER ACI-318
UNIT FEE KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0
MEMBER INCIDENCES
1 1 2
UNIT INCHES KIP
MEMBER PROPERTY
1 TAPERED 18 10 18 15 2.5
CONSTANTS
E 3300 ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL
SUPPORTS
1 2 PINNED
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -5.76
```

```
LOAD 2 LIVE LOAD
1 UNI GY -7.04
LOAD COMB 3 ACI 318-02
1 1.4 2 1.7
PERFORM ANALYSIS
LOAD LIST 3
START CONCRETE DESIGN
CODE ACI 2002
UNIT INCHES KIP
MINMAIN 9 ALL
FC 4 ALL
FYMAIN 60 ALL
TRACK 2.0 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

D1.F.4.6.3 ACI 318-11 and Earlier Beam Design Output

The following annotations apply to the output:

LEVEL	Serial number of bar level which may contain one or more bar group
HEIGHT	Height of bar level from the bottom of beam
BAR INFO	Reinforcement bar information specifying number of bars and bar size
FROM	Distance from the start of the beam to the start of the reinforcement bar
TO	Distance from the start of the beam to the end of the reinforcement bar
ANCHOR(STA/END)	States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end
ROW	Actually required flexural reinforcement (As/bd) where b = width of cross section (ZD for rectangular and square section) and d = effective depth of cross section (YD - distance from extreme tension fiber to the c.g. of main reinforcement).
ROWMN	Minimum required flexural reinforcement (Amin/bd)
ROWMX	Maximum allowable flexural reinforcement (Amax/bd)
SPACING	Distance between centers of adjacent bars of main reinforcement
Vu	Factored shear force at section
Vc	Nominal shear strength provided by concrete
Vs	Nominal shear strength provided by shear reinforcement
Tu	Factored torsional moment at section
Tc	Nominal torsional moment strength provided by concrete
Ts	Nominal torsional moment strength provided by torsion reinforcement

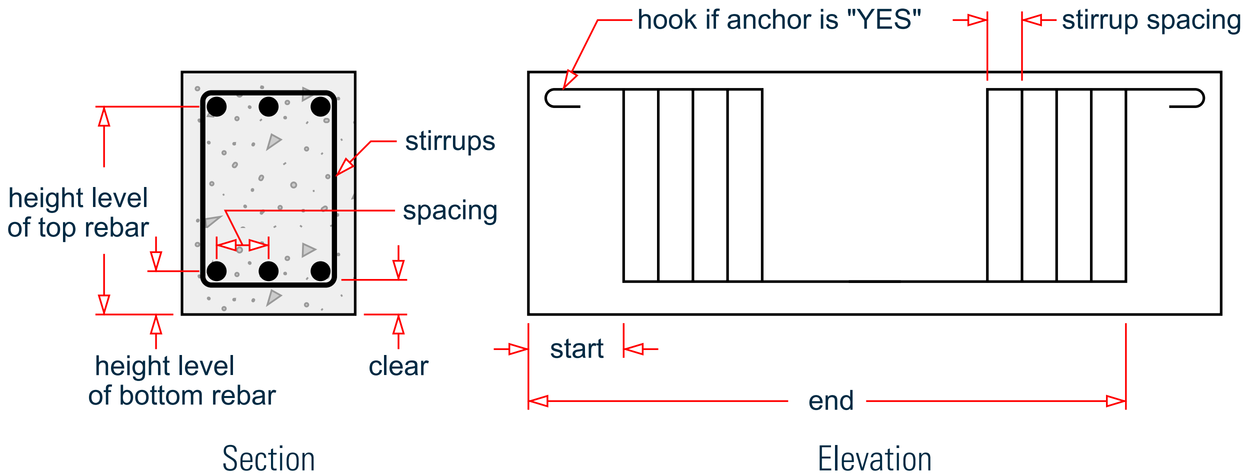


Figure 150: Nomenclature used in output

The following is a sample `TRACK 1.0` output from concrete beam design per ACI (from the file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US.8. Concrete Design for a Space Frame.std`):

```
  =====================================================================

         BEAM  NO.    14 DESIGN RESULTS - FLEXURE PER CODE ACI 318-05

  LEN -  20.00FT.  FY - 60000.  FC - 4000.  SIZE - 16.00 X 21.00 INCHES

  LEVEL    HEIGHT     BAR INFO       FROM            TO          ANCHOR
         FT.    IN.                FT.     IN.     FT.     IN.    STA  END
  _____________________________________________________________________

    1    0 + 2-3/8    8-NUM.5      2 + 2-5/8     20 + 0-0/0     NO   YES
   |----------------------------------------------------------------|
   |   CRITICAL POS MOMENT=    191.70 KIP-FT  AT 11.67 FT, LOAD    1|
   |   REQD STEEL=  2.47 IN2, RHO=0.0083, RHOMX=0.0214 RHOMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 1.62/ 1.62 INCH           |
   |   REQD. DEVELOPMENT LENGTH = 27.37 INCH                        |
   |----------------------------------------------------------------|

    Cracked Moment of Inertia Iz at above location =    4320.43 inch^4

    2    1 + 6-1/8    4-NUM.11     0 + 0-0/0     16 + 2-0/0     YES  NO
   |----------------------------------------------------------------|
   |   CRITICAL NEG MOMENT=    371.81 KIP-FT  AT  0.00 FT, LOAD    1|
   |   REQD STEEL=  5.39 IN2, RHO=0.0184, RHOMX=0.0214 RHOMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 2.82/ 3.53 INCH           |
   |   REQD. DEVELOPMENT LENGTH = 80.14 INCH                        |
   |----------------------------------------------------------------|

    Cracked Moment of Inertia Iz at above location =    8050.77 inch^4

    3    1 + 6-3/8    3-NUM.6     16 +10-1/4     20 + 0-0/0     NO   YES
   |----------------------------------------------------------------|
   |   CRITICAL NEG MOMENT=    104.94 KIP-FT  AT 20.00 FT, LOAD    1|
   |   REQD STEEL=  1.30 IN2, RHO=0.0044, RHOMX=0.0214 RHOMN=0.0033 |
```

```
  |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 1.75/ 5.62 INCH           |
  |   REQD. DEVELOPMENT LENGTH = 21.35 INCH                        |
  |----------------------------------------------------------------|

    Cracked Moment of Inertia Iz at above location =    2602.81 inch^4



         B E A M  N O.    14 D E S I G N  R E S U L T S - SHEAR

   AT START SUPPORT - Vu=   84.31 KIP  Vc=   35.77 KIP  Vs=   76.64 KIP
   Tu=    0.23 KIP-FT  Tc=   5.70 KIP-FT  Ts=   0.00 KIP-FT  LOAD     1
   NO STIRRUPS ARE REQUIRED FOR TORSION.
   REINFORCEMENT IS REQUIRED FOR SHEAR.
   PROVIDE NUM. 5 2-LEGGED STIRRUPS AT  4.7 IN. C/C FOR  102. IN.
   AT END   SUPPORT - Vu=   57.62 KIP  Vc=   35.77 KIP  Vs=   41.06 KIP
   Tu=    0.23 KIP-FT  Tc=   5.70 KIP-FT  Ts=   0.00 KIP-FT  LOAD     1
   NO STIRRUPS ARE REQUIRED FOR TORSION.
   REINFORCEMENT IS REQUIRED FOR SHEAR.
   PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  9.3 IN. C/C FOR  102. IN.
 ___  11J____________________  240.X  16.X  21_____________________  12J____
|                                                                           |
||=========================================================  ===========||
| 4#11H |18.| |0.TO |194. | | | | | | | | | | | |  |3#6 H| 18. 202.TO| 240. |
| 23#5 C/C  5 | | | | | | | | | | | | | | | | | |  |  |  |  |12#4 C/C  9|   |
|       |8#5|H| |2.  |27.TO| 240.| | | | | | | | |  |  |  |  |      |  |  |   |
|       ================================================================||
|                                                                           |
|___________________________________________________________________________|
 ____________    ____________    ____________    ____________    ____________
|            |  |            |  |            |  |            |  |            |
|   OOOO     |  |   OOOO     |  |   OOOO     |  |   OOOO     |  |    ooo     |
| 4#11       |  | 4#11       |  | 4#11       |  | 4#11       |  | 3#6        |
|            |  |            |  |            |  |            |  |            |
|            |  | 8#5        |  | 8#5        |  | 8#5        |  | 8#5        |
|            |  | oooooooo   |  | oooooooo   |  | oooooooo   |  | oooooooo   |
|            |  |            |  |            |  |            |  |            |
|____________|  |____________|  |____________|  |____________|  |____________|
```

D1.F.5 Column Design

Columns design as per ACI 318-14 is based on a full 3D interaction surface for a given cross section. Columns will be designed for the combined effects of axial loads and bi-axial bending moments. Other effects such as torsion and shear will also be accounted for in the design. The program then provides a final detailed design followed by a cross section wise analysis to ensure that the design is satisfactory. During the analysis stage the strengths of the cross section for each of these effects will also be calculated. The design process works on the basis of being able to find a point on or very close to the interaction surface so as to ensure the most efficient design.

As with beam design, an envelope is created at each design section from a collection of the max/min forces in all degrees of freedom for all the load cases and combinations that have been included in the design. As there are six degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) and to capture the maximum and minimum effects, a total of 12 sets of loads are designed for. Each set will include the maximum or minimum of one force such as Max FX along with all the other associated forces in the load case/combination.

The following section shapes will be considered for column design with ACI 318-14:

1. square
2. rectangular
3. circular
4. tee-shaped

All column designs will be based on a uniform distribution of reinforcement around the column perimeter. The design options to control a column design will be based on the CRDO parameter setting.

Related Links

- *V. ACI 318-14 Circular Column* (on page 6183)
- *V. ACI 318-14 Rectangular Column* (on page 6186)

D1.F.5.1 Strength Design

All column designs will be based on a uniform distribution of reinforcement around the column perimeter. The design options to control a column design will be based on the CRDO parameter setting.

Column resistance is calculated based on strain compatibility and interaction surface calculations.

The following conditions should be checked (10.5.1.1)

$$\phi Pn \geq Pu$$
$$\phi Mn \geq Mu$$
$$\phi Vn \geq Vu$$
$$\phi Tn \geq Tu$$

$\phi P_n \geq P_u$

$\phi M_n \geq M_u$

$\phi V_n \geq V_u$

$\phi\ T_n \geq T_u$

Where the nominal axial compressive strength, P_n , shall not exceed $P_{n,max}$ in accordance with Table 22.4.2.:

$P_{n,max} = 0.80\ P_o$ for ties
$P_{n,max} = 0.85\ P_o$ for spirals
$P_o = 0.85\ f'_c\ (A_g - A_{st}) + f_y A_{st}$

where

A_{st}	=	the total area (of nonprestressed) longitudinal reinforcement
Ag	=	the gross area of the concrete section
f'_c	=	the specified compressive strength of concrete
f_y	=	the specified yield strength of passive reinforcement
M_n	=	the nominal flexural strength
V_n	=	the nominal shear strength
T_n	=	the nominal torsional moment

The design is performed in two steps considering the longitudinal reinforcement design for axial-bending in an initial step, and the shear and torsion design including the longitudinal reinforcement for torsion tension in a latter pass together with the transverse design.

The minimum area of longitudinal reinforcement is taken as $0.01Ag$ and the maximum as $0.08Ag$. In agreement with the commentary a warning is triggered when the reinforcement ratio is over 4 percent due to possible lap splices zones.

D1.F.5.2 Slenderness Effects and Analysis Considerations

Slenderness effects are extremely important in designing compression members. The ACI 318-14 code specifies two options by which the slenderness effect can be accounted for. One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro uses both these options. To perform the first type of analysis, use the command `PDELTA ANALYSIS` instead of `PERFORM ANALYSIS`. This analysis method will accommodate the requirements as specified in Section 6.7 of the ACI 318-14Code (Section 10.10 in previous editions), except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD.Pro can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief that a `PDELTA ANALYSIS`, as performed by STAAD.Pro , is most appropriate for the design of concrete members. However, you should note that to take advantage of this analysis, all combinations of loadings must be provided as repeat load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a repeat load case consists of one or more primary load case which are revised during the PDelta analysis based on the deflections. Also note that the you must provide the proper factored loads (e.g., 1.4 for DL etc.). STAAD.Pro does *not* factor the loads automatically.

D1.F.5.3 Moment Magnification

If the moment magnification method is used in lieu of an exact analysis, columns designed per ACI 318-14 in STAAD.Pro can consider sway using the moment magnification procedure. The second order effects are approximated by magnified moments for slender columns (as determined by clause 6.2.5/6.6.4.6) in sway and non-sway frames.

The following parameters apply to these calculations:

For sway frames, moments should be amplified twice, once for the member end sway (P-Δ effect) provided in clause 6.6.4.6 and using those amplified moments, again amplify them for member curvature along the length effects (P-δ effect) based on clause 6.6.4.5.

For non-sway frames moments need only be amplified once for the member curvature along the length effects (P-δ effect) based on clause 6.6.4.5

STAAD Parameters Required to Perform the Design and Their Functionality

SWY	Is member braced against side-sway: This parameter checks whether the column being designed is braced structurally against side sway for lateral loads. It will affect the slenderness limit of the structure. This parameter is based on clause 6.2.5
BDY / BSZ	β_{dns} factor required for Eq. 6.6.4.4.4(a) for stiffness reduction in case of non-sway case.
SQY / SQZ	Stability index value which is required for two main purpose; one is for checking whether the moments need to be magnified for sway or non-sway case, Clause 6.6.4.3(b), and for the calculation of δ_s give in Eq. 6.6.4.6.2(a) for moment amplification of a sway case.

SKY / SKZ Effective length factor which is based on Fig. R6.2.5 to be used to calculate member slenderness in Eq. 6.2.5(a)/(b)

SLY / SLZ The external load case number for the sway case which is used to extract moments M_{1s} and M_{2s} in equation 6.6.4.6.1(a)/(b). Incase this parameter is 0, it indicates to the program you do not want to include a load case which based on their judgement will not produce side-sway and hence needs to be amplified.

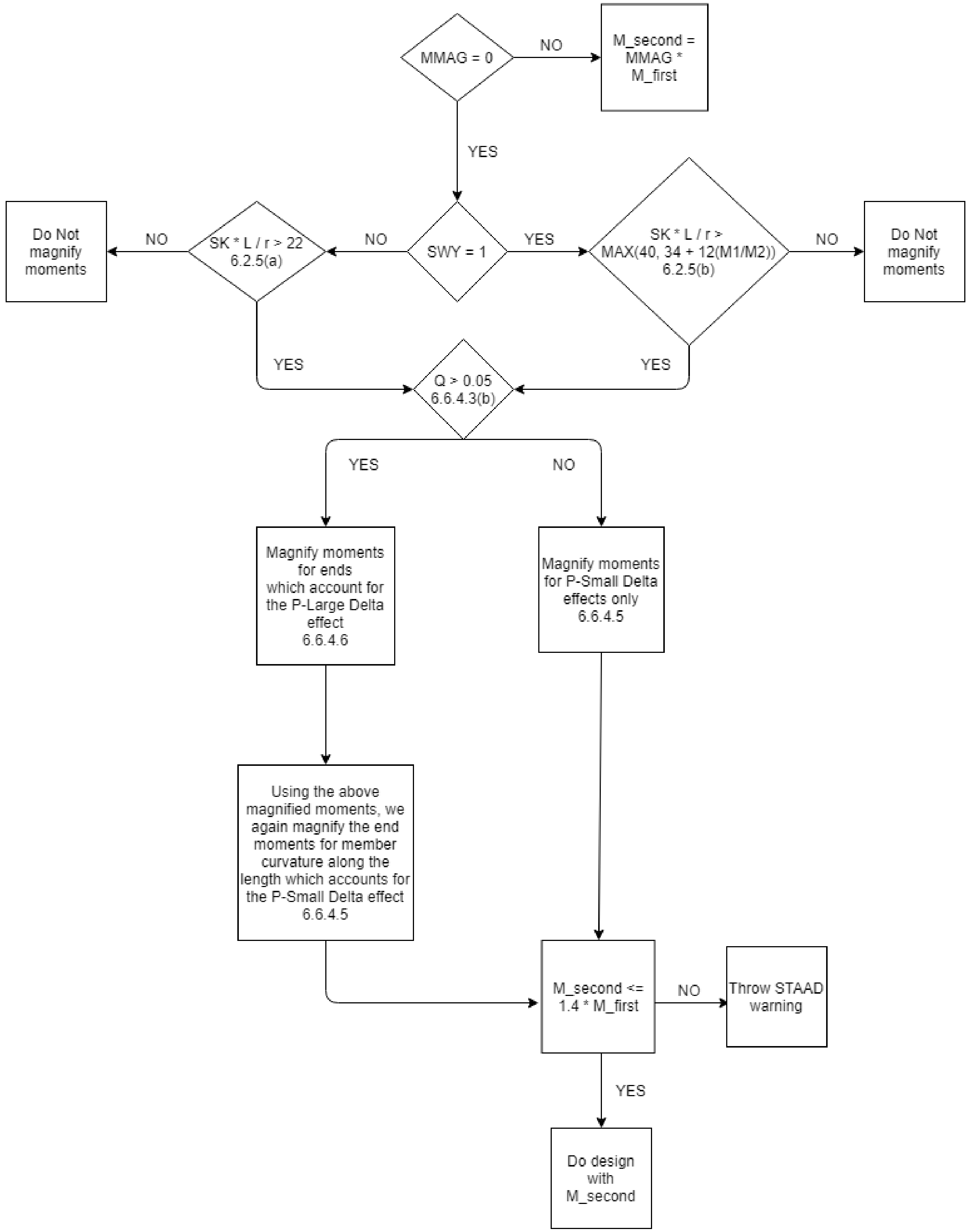
TRN This parameter asks from the user whether there are in transverse loads in the column between its two ends. It affects the calculation on Cm based on Eq. 6.6.4.5.3(a)/(b) which is required for amplification of moments for non-sway case.

MMAG This is the starting point parameter for the program. A non-zero value represents that the program will ignore all the moment amplification algorithm and just amplify the moments by multiplying this MMAG user-defined factor.

Note: The program will only go through with the moment magnification algorithm if and only if `MMAG` is specified as 0.

Flowchart

Given below is a flowchart of the moment amplification algorithm which is being incorporated in STAAD.Pro.


MMAG = 0
NO
M_second = MMAG * M_first
YES
Do Not magnify moments
NO
SK * L / r > 22 6.2.5(a)
NO
SWY = 1
YES
SK * L / r > MAX(40, 34 + 12(M1/M2)) 6.2.5(b)
NO
Do Not magnify moments
YES
Q > 0.05 6.6.4.3(b)
YES
YES
NO
Magnify moments for ends which account for the P-Large Delta effect 6.6.4.6
Magnify moments for P-Small Delta effects only 6.6.4.5
Using the above magnified moments, we again magnify the end moments for member curvature along the length which accounts for the P-Small Delta effect 6.6.4.5
M_second <= 1.4 * M_first
NO
Throw STAAD warning
YES
Do design with M_second

D1.F.5.4 Definition of Bar Positions

Both input parameters and design output refer to the positions of bars in column sections as follows.

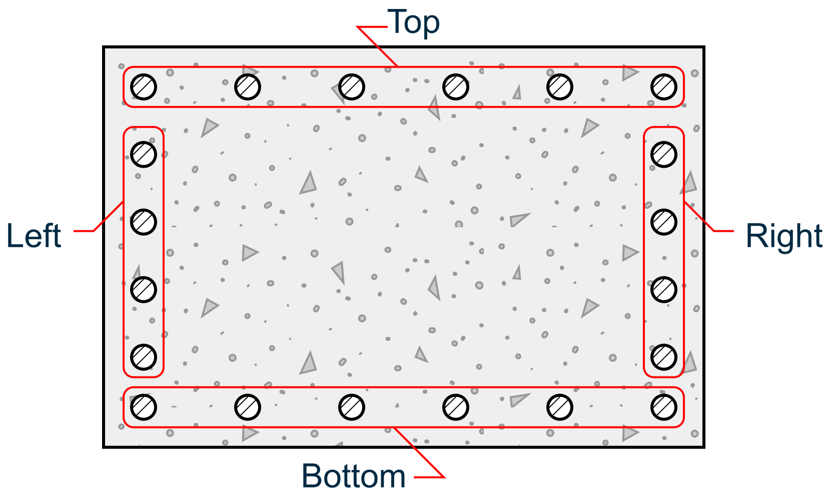


Figure 151: ACI 318-14 Rectangular column bar positions

Bars are evenly distributed around circumference

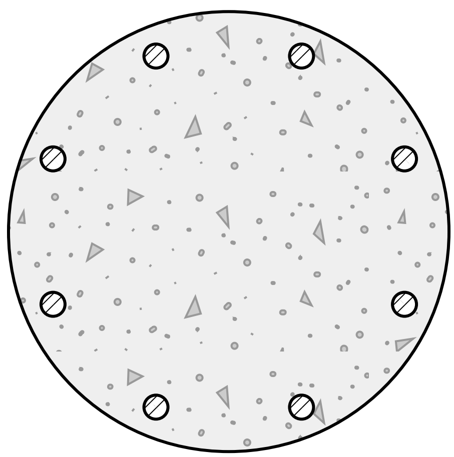


Figure 152: ACI 318-14 Circular column bar positions

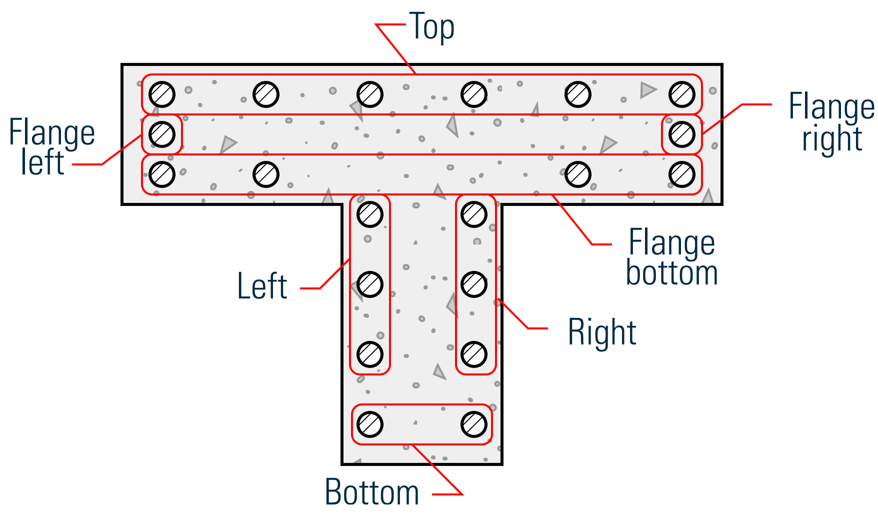


Figure 153: ACI 318-14 Tee column bar positions

D1.F.5.5 Column Design Output

The following report options are provided for the output for a column design.

TRACK 0 Output

Setting the option `TRACK 0` will cause the program to produce the required area of longitudinal as well as transverse steel at 5 equally spaced locations –including member ends– along the member length.

Note: This option is meant to be considered for quick area of steel (As) requirement analysis and should *not* be considered as a safe, acceptable design.

Sample `TRACK 0` output:

Member	member number
Type	Column
Shape	Either rectangular, Tee, or circular
As	the area of longitudinal steel required at each cross section
Asv/sv	the transverse steel density required at each cross section

```
                     STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                     ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)
  |---------------------------------------------------------|
  |  Member :      8                                        |
  |  Type   : Column           Shape  : Rectangular         |
  |---------------------------------------------------------|
  |  AREA OF STEEL REQUIRED                                 |
  |---------------------------------------------------------|
  |  Section -        |   1   |   2   |   3   |   4   |   5   |
  |---------------------------------------------------------|
  |  Location         |  0.00 | 36.00 | 72.00 |108.00 |144.00 |
  |  As(Longitudinal)|  3.29 |  1.44 |  1.44 |  1.73 |  5.45 |
  |  As/sv(Trans Y)   |  0.03 |  0.03 |  0.03 |  0.03 |  0.03 |
```

```
  |  As/sv(Trans Z)  |  0.03 |  0.03 |  0.03 |  0.03 |  0.03 |
  |----------------------------------------------------------|
```

TRACK 1 Output

Setting the program to output the beam design with a `TRACK 1` option will perform full design and detailing operations and will report a summary of the design. A `TRACK 1` report will include the following sections:

Design Summary	This section provides a summary of the design including the design status (i.e., PASS/FAIL), member type, length, the critical ratio, the critical criteria, and relevant clauses.
Cross section details	This section provides shape type and dimensions of the cross section profile.
Longitudinal Bar Layout	The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of "yes" in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of "no" indicates that no additional extension to the bar is needed beyond the given dimension. The value given in "Distance from face"is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.
Transverse Bar Layout	The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1217)) within the span for both the local Y & Z axes. Each zone is identified by the unique start and end location reported as the distances "From" and "To." This provides shear densities as minimum required and as provided by the given bar details. The value "Nums" is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

Note: This output will *not* report any warnings or errors from the design and member analysis. These are only reported in the `TRACK 2` output.

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)

Member :      9

DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Fail         Type    : Column         Length:  144.000 |
| Critical Ratio :   1.095        Criteria: See Below                       |
| Critical Clause:   10.6.1                                                 |
| All Failures   :   10.6.1                                                 |
-----------------------------------------------------------------------------

CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   12.00 | Depth:   12.00 |
-------------------------------------------------------
```

```
LONGITUDINAL BAR LAYOUT
----------------------------------------------------------------------
|                 |    Bars      |   Location     | Distance  |  Anchor  |
|        Position | Nums | Size  | Start|    End  | From Face | Start End |
----------------------------------------------------------------------
|             Top |    3 | #11   |  0.00| 144.00  |     2.20  |  Yes  Yes |
|          Bottom |    3 | #11   |  0.00| 144.00  |     2.20  |  Yes  Yes |
|            Left |    1 | #11   |  0.00| 144.00  |     2.20  |  Yes  Yes |
|           Right |    1 | #11   |  0.00| 144.00  |     2.20  |  Yes  Yes |
----------------------------------------------------------------------

TRANSVERSE BAR LAYOUT
---------------------------------------------------------------------------
|                             |  Asv Density  |     Rebar Specification      |
| Zone | Dir.|   From |  To   | Reqd. | Prov. | Nums | Size | Spacing | Legs |
---------------------------------------------------------------------------
|    1 |  Y  |  0.00 |144.00 |  0.03 |  0.08 |   32 |  # 4 |   4.65  |   2  |
|    1 |  Z  |  0.00 |144.00 |  0.03 |  0.08 |   32 |  # 4 |   4.65  |   2  |
---------------------------------------------------------------------------

------------------------ Member:     9 Design Ends ------------------------
```

TRACK 2 Output

`TRACK 2` will produce the results of `TRACK 1` along with the details of the bars and their distribution at each cross section. This option will also report any of the relevant detailed design messages. The longitudinal bar details at each cross section will have information on the setting out of each bar at the specific cross section. This should serve as a useful tool to inspect the design in detail. A `TRACK 2` output would contain:

Design Summary — This section provides a summary of the design including the design status, i.e. PASS/FAIL, member type and length, the critical ratio, the critical criteria and relevant clauses.

Cross Section Details — This section provides the shape type and dimension(s) of the cross-section

Design Inputs — This section reports the material data for the steel and concrete including the yield strength for steel (FY), compressive strength of concrete (FC) and modulus of elasticity for both (Es and Ec) This section additionally includes details of the cover to the longitudinal bars from the top, bottom and side faces.

Critical Strength Results — This section reports the demand determined from the design envelope as described in D1.F.4 Beam Design (on page 1215), along with the capacity for forces applied in positive and negative directions. The ratio is then the value of the demand divided by the appropriate capacity.

Longitudinal Bar Details at Cross Sections — For each of the design cross sections, the longitudinal bars are reported for each face where they are required, top bottom and skin. Area of steel is reported as the optimum required from the design and that provided after additionally meeting the detailing requirements.

The total quantity of bars on the given face are reported as “No(s) bars,” the size of longitudinal bars and quantity of layers that the bars are to be distributed on. For example, if the number of bars reported is 10 and in 2 layers, then there will be 5 bars in each layer.

Longitudinal Bar Layout — The position and extent of the longitudinal bars will be reported along with an indication if additional anchorage is required at the start or the end of the bars. A value of “yes” in the Anchor column indicates that the bar or set of bars will need to be anchored beyond the given section distances. This can be done using hooks/bends or physical anchors as deemed fit. A value of “no” indicates that no additional extension to the bar is needed beyond the given dimension. The value given in “Distance from face”is the measurement from the face defined by the position, to the center line of the bar, measured perpendicular to the face.

Transverse Bar Layout

The transverse bar table reports the details of various shear zones (see D1.F.4.3 Shear and Torsion Design (on page 1217)) within the span for both the local Y & Z axes.

Each zone is identified by the unique start and end location reported as the distances “From” and “To.”

This provides shear densities as minimum required and as provided by the given bar details.

The value “Nums” is the number of transverse groups in the zone (this equates to the zone length/spacing +1), the center to center spacing between adjacent transverse groups along the local X axis, the size of the bar used in the transverse group, and the leg count for the bars in a transverse group. A transverse group is the collection of transverse bars at a given cross section.

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************

Units: Kip and Inch (Unless Noted Otherwise)

Member :     10

DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column         Length:  144.000 |
| Critical Ratio :   0.942        Criteria: Flexure                         |
| Critical Clause:   10.5.2                                                 |
-----------------------------------------------------------------------------

CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   12.00 | Depth:   12.00 |
-------------------------------------------------------

DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc           4.000 |                      | Ec     0.360E+04 |
| Steel    | Fy(main)    60.000 | Fy(trans)    60.000 | Es     0.290E+05 |
| Cover    | Top          1.500 | Bottom        1.500 | Sides      1.500 |
-------------------------------------------------------------------------

Design Messages
-------------------------------------------------------------------------

   WARNINGS: DESIGN FOR MEMBER    10
   --------------------------------
    1) Ast Prov > 4 % ,  Cl - 10.6.1.1(commentary)
    2) Incorrect shear min density or max spacing. Density may be ignored


CRITICAL STRENGTH RESULTS
-----------------------------------------------------------
| Category |   Demand   |Min Capacity|Max Capacity|  Ratio  |
-----------------------------------------------------------
```

```
| Axial    |    -91.718|   -438.248 |    336.960 |   0.209 |
| Flexure  |    957.058|  -1016.395 |   1016.395 |   0.942 |
| Shear Y  |     13.584|    -24.847 |     24.847 |   0.547 |
| Shear Z  |      5.534|    -20.846 |     20.846 |   0.265 |
| Torsion  |      0.000|      0.000 |      0.000 |   0.000 |
-----------------------------------------------------------

LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
------------------------------------------------------------------
| Distance | Ast-reqd | Ast-prov | No(s)bars| Size | No of Layers |
------------------------------------------------------------------
|    0.000 |    3.465 |    6.240 |     4    |  #11 |       1      |
|   36.000 |    1.440 |    6.240 |     4    |  #11 |       1      |
|   72.000 |    1.440 |    6.240 |     4    |  #11 |       1      |
|  108.000 |    1.753 |    6.240 |     4    |  #11 |       1      |
|  144.000 |    5.650 |    6.240 |     4    |  #11 |       1      |
------------------------------------------------------------------

LONGITUDINAL BAR LAYOUT
----------------------------------------------------------------------
|                 |     Bars      |    Location     | Distance  |  Anchor  |
|        Position | Nums | Size   | Start|   End    | From Face | Start End |
----------------------------------------------------------------------
|             Top |    2 | #11  |  0.00| 144.00 |     2.20  |  Yes  Yes |
|          Bottom |    2 | #11  |  0.00| 144.00 |     2.20  |  Yes  Yes |
----------------------------------------------------------------------

TRANSVERSE BAR LAYOUT
---------------------------------------------------------------------------
|                              |  Asv Density  |      Rebar Specification      |
| Zone | Dir.|  From |  To     | Reqd. | Prov. | Nums | Size | Spacing | Legs |
---------------------------------------------------------------------------
|    1 |  Y  |  0.00 |144.00 |  0.03 |  0.10 |   38 |  # 4 |   3.89   |   2  |
|    1 |  Z  |  0.00 |144.00 |  0.03 |  0.10 |   38 |  # 4 |   3.89   |   2  |
---------------------------------------------------------------------------

------------------------ Member:    10 Design Ends ------------------------
```

D1.F.5.6 Column Design in Pre-2014 Codes

Columns design in STAAD.Pro per the ACI code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the `MMAG` parameter is specified, the column moments are multiplied by the `MMAG` value to arrive at the ultimate moments on the column. Since the ACI code no longer requires any minimum eccentricity conditions to be satisfied, such checks are not made.

Method used

Bresler Load Contour Method

Known Values

Pu, Muy, Muz, B, D, Clear cover, Fc, Fy

Ultimate Strain for concrete : 0.003

Steps involved

1. Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.
2. Find an approximate arrangement of bars for the assumed reinforcement.
3. Calculate PNMAX = 0.85 Po, where Po is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed PNMAX. If PNMAX is less than Pu/PHI, (PHI is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 8%, the column cannot be designed with its current dimensions.
4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as MYCAP and MZCAP respectively.
5. Solve the interaction equation:

$$\left(\frac{M_{ny}}{M_{ycap}}\right)^{a} + \left(\frac{M_{nz}}{M_{zcap}}\right)^{a} \leq 1.0$$

where

a	=	The ALPHA parameter used in ACI 318 2011, 2008, 2005, and 2002. Refer to D1.F.3.1 ACI 318-2011 Design Parameters (on page 1206) or D1.F.3.2 Pre ACI 318-2011 Design Parameters (on page 1212) for details.

If the column is subjected to a uniaxial moment, a is chosen as 1.0

6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 8%) and steps 2 to 6 are repeated.
8. The maximum spacing of reinforcement closest to the tension force, for purposes of crack control, is given by

$$s = 15\left(40,\ 000\frac{40,\ 000}{f_s}\right) - 2.5cc \leq 12\left(\frac{40,\ 000}{f_s}\right)$$

with f_s in psi and is permitted to be taken equal to (2/3) f_y, rather than 60 percent of f_y, as in ACI 318-02.

9. Section 10.9.3 has been modified to permit the use of spiral reinforcement with specified yield strength of up to 100,000 psi. For spirals with fyt greater than 60,000 psi, only mechanical or welded splices may be used.

Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

P0	Maximum purely axial load carrying capacity of the column (zero moment).
Pnmax	Maximum allowable axial load on the column (Section 10.3.5 of ACI 318).
P-bal	Axial load capacity at balanced strain condition.
M-bal	Uniaxial moment capacity at balanced strain condition.
e-bal	M-bal / P-bal = Eccentricity at balanced strain condition.
M0	Moment capacity at zero axial load.

P-tens Maximum permissible tensile load on the column.

Des. Pn Pu/PHI where PHI is the Strength Reduction Factor and Pu is the axial load for the critical load case.

Des. Mn $Mu \times MMAG / PHI$ where PHI is the Strength Reduction Factor and Mu is the bending moment for the appropriate axis for the critical load case.

For circular columns,

$$M_u = \sqrt{M_{uy}{}^2 + M_{uz}{}^2}$$

$$e/h = (M_n/P_n)/h$$

where

h = the length of the column

Example

Column design per the ACI 318-2005 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2005
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example

Column design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example

Column design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Column Design Output

The samples illustrate different levels of the column design output. The following output is generated without any TRACK definition (i.e., using the default of TRACK 0.0):

```
=====================================================================

     COLUMN  NO.     5  DESIGN PER ACI 318-05 - AXIAL + BENDING

FY - 60000  FC - 4000 PSI,  SQRE SIZE - 12.00 X 12.00 INCHES, TIED
        AREA OF STEEL REQUIRED =   7.589  SQ. IN.

BAR CONFIGURATION        REINF PCT.   LOAD   LOCATION   PHI
-----------------------------------------------------------

  8 - NUMBER  9            5.556        2      STA     0.650
(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
TIE BAR NUMBER    4 SPACING   8.00 IN
```

TRACK 1.0 generates the following additional output:

```
    COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)

 --------------------------------------------------------
   P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
  942.40    753.92    179.59    170.75   11.41
    M0        P-tens.   Des.Pn    Des.Mn     e/h
  148.52   -480.00    350.15     10.47    0.00249
 --------------------------------------------------------

    COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)

 --------------------------------------------------------
   P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
  942.40    753.92    179.59    170.75   11.41
    M0        P-tens.   Des.Pn    Des.Mn     e/h
  148.52   -480.00    350.15    136.51    0.03249
 --------------------------------------------------------
```

TRACK 2.0 generates the following output in addition to the above examples:

```
                             Pn        Mn        Pn        Mn    (@ Z )
            |                695.93     77.23    347.96    148.53
         P0 |*               637.93     93.16    289.97    157.71
            | *              579.94    107.06    231.98    164.41
     Pn,max|__*              521.94    118.23    173.98    170.18
            |   *            463.95    129.01    115.99    163.66
         Pn |    *           405.96    139.03     57.99    156.37
    NOMINAL|      *          Pn        Mn        Pn        Mn    (@ Y )
      AXIAL|       *         695.93     77.23    347.96    148.53
COMPRESSION|        *        637.93     93.16    289.97    157.71
         Pb|-------*Mb       579.94    107.06    231.98    164.41
            |      *         521.94    118.23    173.98    170.18
___________|____*_______     463.95    129.01    115.99    163.66
            |  * M0   Mn,    405.96    139.03     57.99    156.37
            | *   BENDING
     P-tens|*      MOMENT
            |
```

Related Links

- *V. ACI 318-14 Square Column* (on page 6189)

- *V. ACI 318-11 Circular Column* (on page 6201)
- *V. ACI 318-11 Square Column* (on page 6203)
- *V. ACI 318-02 Square Column* (on page 6219)
- *V. ACI 318-99 Square Column* (on page 6225)
- *V. ACI 318-99 Circular Column* (on page 6228)

D1.F.5.6.1 Slenderness Effects and Analysis Consideration
Slenderness effects are extremely important in designing compression members. The ACI 318 code specifies two options by which the slenderness effect can be accommodated (Section 10.10 & 10.11 ACI-318). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro uses both these options. To perform the first type of analysis, use the command `PDELTA ANALYSIS` instead of `PERFORM ANALYSIS`. This analysis method will accommodate the requirements as specified in Section 10.10 of the ACI-318 Code, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD.Pro can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief that a `PDELTA ANALYSIS`, as performed by STAAD.Pro , is most appropriate for the design of concrete members. However, you should note that to take advantage of this analysis, all combinations of loadings must be provided as repeat load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a repeat load case consists of one or more primary load case which are revised during the PDelta analysis based on the deflections. Also note that the you must provide the proper factored loads (e.g., 1.4 for DL etc.). STAAD.Pro does *not* factor the loads automatically.

Moment Magnification per the 2011 Editions

If the moment magnification method is used in lieu of an exact analysis, columns designed per ACI 318-11 in STAAD.Pro can consider sway using the moment magnification procedure. The second order effects are approximated by magnified moments for slender columns (as determined by clause 10.10.1) in sway and non-sway frames. The moment magnification procedure described in clause 10.10.6 of the code is used for nonsway frame columns and the procedure in clause 10.10.7 is used for sway frame columns.

The parameters `BDY`, `BDZ`, `SKY`, `SKZ`, `SLY`, `SLZ`, `SQY`, `SQZ`, `SWY`, and `TRN` apply to these calculations.

Note: You must set the `MMAG` parameter to 0 in order to consider sidesway for slender columns. Otherwise, the `MMAG` value is used directly.

D1.F.6 Slab Design

Slab design is performed only for the moments MX and MY at the center of an element. Design will not be performed for SX, SY, SXY, SQX, SQY, or MXY. Also, design is not performed at any other point on the surface of the element.

Note: Element design is only supported for ACI 318-08 and earlier editions of the code.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement (D1.F.2 Section Types for Concrete Design (on page 1199)). The parameters FYMAIN, FC, and CLEAR listed in D1.F.3 Design Parameters (on page 1201) are relevant to slab design. Other parameters mentioned in Table 3.1 are not applicable to slab design. Please note that the default value of clear cover - parameters CLT and CLB - for plate elements is 0.75 inches, as shown in D1.F.3 Design Parameters (on page 1201).

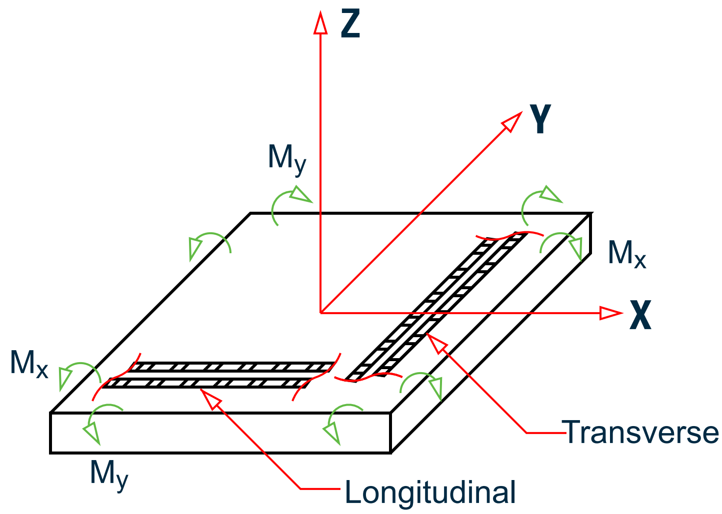


Figure 154: Sign convention of loaded plate element

Element design for flexure is done using the same rules that are used for beam design. A unit width (1 meter) is assumed as the width of the beam. This suits plate elements as MX and MY are in units of moment per unit width. Reinforcement is reported in units of mm^2 per unit width. Longitudinal direction corresponds to the direction of reinforcement for MX, transverse corresponds to that for MY. Longitudinal reinforcement is the outer layer of reinforcement and transverse is the inner layer.

Elements are *not* designed for shear forces or axial stress.

Example Element Design Output

```
   ELEMENT DESIGN SUMMARY
   ----------------------

   ELEMENT    LONG. REINF      MOM-X /LOAD    TRANS. REINF      MOM-Y /LOAD
              (SQ.IN/FT)     (K-FT/FT)         (SQ.IN/FT)     (K-FT/FT)

        FY:    60.000 KSI FC:     4.000 KSI COVER (TOP):     0.750 IN
        COVER (BOTTOM):     0.750 IN TH:     6.000 IN
     47 TOP : Longitudinal direction - Only minimum steel required.
     47 TOP : Transverse direction   - Only minimum steel required.
     47 TOP :      0.130          0.00 /    0        0.130          0.00 /    0
        BOTT:      0.562         11.60 /    1        0.851         14.83 /    1
```

Example

Element design per the ACI 318-2008 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2008
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

Example

Element design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

D1.F.7 Scope of ACI 318 Code Implemented

The following describes the code clauses from ACI 318-14 implemented for beams and columns.

Note: Plate elements and deep beams cannot be designed per ACI 318-14. Use an older edition of the ACI code to design these elements if needed.

Materials

- Beams

  19.2.2
  19.2.3.1
  19.2.4.2
- Columns

  19.2.2
  19.2.4.2

Minimum Reinforcement

- Beams

  9.6.1.1
  9.6.1.2
  9.6.1.3

9.7.2.1
9.7.6.4.2
9.7.6.4.3

- Columns

  10.6.1
  10.7.2.1
  10.7.3.1
  10.7.6.1
  10.7.6.2
  25.2.3

Clauses 9.7.6.4.2 and 9.7.6.4.3 are not included in the standard design criteria of the minimum reinforcement since the design of these criteria should consider the rebar including the bar detailing and thus they should be applied in a second design after performing the strength design and the bar detailing.

Clause 9.6.1.3 is also not included in the standard criteria since the reduction of the minimum reinforcement is optional.

Service

- Beams

  9.7.2.2
  9.7.2.3

Strength

- Beams

  9.5
  9.5.1
  9.5.2
  9.5.3
  9.5.4
  9.6.3.1
  9.6.3.3
  9.6.4.2
  9.7.5.2
  9.7.6.2.2
  9.7.6.3.1
  9.7.6.3.3
  9.7.3.3
  9.6.4.3
  20.2.2.4a
  20.2.2.1
  20.2.2.2
  21.2.1
  22.2.2.2
  22.5.5

22.2.2.3
22.7.4
22.5.1.2
22.7.6.1
22.7.7.1

- Columns

  10.5
  10.5.1
  10.5.2
  10.5.3
  10.6.2
  10.7.6.5.2
  20.2.2.4a
  20.2.2.1
  20.2.2.2
  22.5
  22.2.2.2
  22.5.5
  22.2.2.3
  22.7.4
  22.5.1.2
  22.7.6.1

Ductility

- Beams

  9.3.3.1

Development Length

- Beams

  25.4.1.4
  25.4.2
  25.4.3
- Columns

  25.4.1.4
  25.4.2
  25.4.3

Span Detailing

- Beams

  9.7.3.3
  9.7.3.8

9.7.7

D1.G. American Codes - Timber Design per AITC Code

D1.G.1 Design Operations

STAAD.Pro supports timber design per two versions of the AITC code: 1985 and 1994. The implementation of both the codes is explained below.

To access the 1994 edition, specify the commands:

```
CODE AITC
```

or

```
CODE AITC 1994
```

To access the 1984 edition, specify the commands:

```
CODE AITC 1984
```

or

```
CODE TIMBER
```

D1.G.1.1 1994 AITC Code Implementation

The salient aspects of design in accordance with the 4th edition (1994) of the *Timber Construction Manual* published by the American Institute of Timber Construction are:

1. Design can be performed for two types of timber sections: dimensional timber sections (i.e., sawn lumber) and glulaminated sections.
2. The program includes a database of dimensional timber sections with this code.

Implementation of Dimensional Lumber Properties

The database of sawn lumber sections, listed in Table 8.1 of the 1994 AITC Manual, is implemented in the program. Some of the key aspects of this implementation are:

In the property tables in the AITC manual, one will find that, for any particular species of timber, the Modulus of Elasticity (E) and allowable stresses may vary with the cross-section size. For example, a 2x4 Douglas Fir-Larch, Select Structural member has an E of 1900 ksi and an allowable bending stress, F_b, of 1450 psi. A 5x5 Douglas Fir-Larch, Select Structural, Beam or Stringer member has an E of 1600 ksi and an allowable bending stress, F_b, of 1600 psi. And a 5x5 Douglas Fir-Larch, Select Structural, Post or Timbers member has an E of 1600 ksi and an allowable bending stress, F_b, of 1750 psi.

So, in the STAAD timber database for sawn lumber, for each species and grade of timber, the section size, or properties are associated with the Modulus of Elasticity and allowable stresses for the cross-section. When a section is assigned, its E and allowable stresses are automatically fetched along with its properties. The material properties of Southern Pine members were taken from Table 8.4 of the 1994 AITC manual. For all other species with section sizes 2”-4” wide, the material properties have been taken from Table 8.3. For all non-Southern Pine species with section sizes greater than 5”x5”, the material properties are obtained from Table 8.6 of the 1994 AITC Manual.

Please note that not all section sizes listed in Table 8.1 are available in every species. Some sizes are not produced for particular species. For example, the Aspen species only produces sizes from 2”-4” wide. It does not produce sizes 5”x5” and larger. This can be observed by comparing Table 8.3, where Aspen is listed as an available species, to Table 8.6, where Aspen is not listed as an available species. Also note that although 1” wide members are listed in Table 8.1, there are no values available in the species properties tables; Table 8.4, Table 8.5, and Table 8.6. AITC does not allow for the structural design of these small members.

D1.G.1.2 1985 AITC Code Implementation

STAAD's Timber design module per the 1985 AITC code (*Timber Construction Manual*, 3rd. Edition, 1985) allows design of Glulam timber sections. It also conforms to the National Design Specification for Wood Construction and Supplement (NDS) and building codes like Uniform Building Code (UBC), Basic/National Building Code and Standard Building Code. Some of the main features of the program are:

1. This feature is for Glulam Timber only (design of dimensional lumber are available for 1994 AITC only).
2. Code check and design of members as per TCM - AITC.
3. Design values for Structural Glued Laminated Timber tables are in-built into the program. The program accepts Table no., Combination and Species specifications as inputs (e.g., 1:16F-V3-SP/SP) and reads design values from in-built tables.
4. Incorporates all the following Allowable stress modifiers:

   i. Duration of Load Factor
   ii. Size Factor
   iii. Form Factor
   iv. Lateral stability of Beams and Columns
   v. Moisture Content Factor
   vi. Temperature and Curvature factors.

   The allowable stresses for bending, tension, compression, shear and Moduli of elasticities are modified accordingly.
5. Determines slenderness for beams and columns (Short, intermediate and long) and checks for min. eccentricity, lateral stability, buckling, bending and compression, bending and tension and horizontal shear against both axes.
6. The output results show sections provided or chosen, actual and allowable stresses, governing condition and ratios of interaction formulae and the relevant AITC clause nos. etc for each individual member.

Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *TR.52.1 Timber Design Parameter Specifications* (on page 2683)

D1.G.2 Allowable Stress per AITC Code

Explanation of terms and symbols used in this section

Table 93: Timber design nomenclature

Symbols	Description
f_a	Actual compression or tension stress (in PSI). For tension, the axial load is divided by net sectional area (i.e, NSF x X-area).
FA	Allowable design value for compression or tension (in PSI) modified with applicable modifiers or calculated based on slenderness in case of compression.

Symbols	Description
f_{bz} f_{by}	Actual bending stresses about local Z and Y axis (in PSI).
FBZ FBY	Allowable design values for bending stresses about local Z and Y axis (in PSI) modified by the applicable modifiers.
JZ JY	Modifier for P-DELTA effect about the Z and Y axis respectively as explained in formula 5-18 of TCM.
f_{vz} f_{vy}	Actual horizontal shear stresses.
FVZ FVY	Allowable horizontal shear stresses.
VZ VY	Shear in local Z and local Y direction.
ZD YD	Depth of section in local Z and Y axis.
EZ EY	Minimum eccentricity along Z and Y axis.
CFZ CFY	Values of the size factors in the z-axis and y-axis, respectively.
CLZ CLY	Represent the factors of lateral stability for beams about the z-axis and y-axis, respectively.
RATIO	Permissible ratio of stresses. The default value is 1.0.

D1.G.2.1 Combined Bending and Axial Tension

The following interaction formulae are checked :

$$f_a/FA + f_{bz}/(FBZ \times CFZ) + f_{by}/(FBY \times CFY) \leq$$

Lateral stability check with Net compressive stress:

$$f_a/FA + f_{bz}/(FBZ \times CLZ) + f_{by}/(FBY \times CLY) \leq$$

D1.G.2.2 Combined Bending and Axial Compression

$$f_a/FA + f_{bz}/(FBZ\text{-}JZ \times f_a) + f_{by}/(FBY\text{-}JY \times f_a) \leq$$

Applicability of the size factor:

a. When CF < 1.00,

if fa > FBZ x (1-CFZ), FBZ is not modified with CFZ. if fa > FBY x (1-CFY), FBY is not modified with CFY.

if fa < FBZ x (1-CFZ) FBZ is taken as FBZ x CFZ + fa but shall not exceed FBZ x CLZ

if fa < FBY x (1-CFY) FBY is taken as FBY x CFY + fa but shall not exceed FBY x CLY

b. When CF ≥ 1.00, the effect of CF and CL are cumulative FBZ is taken as FBZ x CFZ x CLZ FBY is taken as FBY x CFY x CLY

D1.G.2.3 Minimum Eccentricity

The program checks against min. eccentricity in following cases:

a. The member is a FRAME member and not a truss member and under compression.
b. The value of actual axial compressive stress does not exceed 30% of the allowable compressive stress.
c. The actual moments about both axes are less than moments that would be caused due to min. eccentricity. In this approach, the moment due to min. eccentricity is taken as the compressive load times an eccentricity of 1 in. or 0.1 x depth whichever is larger.

In case of min. eccentricity,

$$f_{bz} = f_a \times (6+1.5 \times JZ)/(EZ/ZD)$$

$$f_{by} = f_a \times (6+1.5 \times JY)/ (EY/YD)$$

the following conditions are checked :

$$f_a/FA + f_{bz}/(FBZ\text{-}JZ \times f_a) \leq$$

$$f_a/FA + f_{by}/(FBY\text{-}JY \times f_a) \leq$$

D1.G.2.4 Shear Stresses

Horizontal stresses are calculated and checked against allowable values:

$$f_{vz} = 3 \times VY /(2 \times Area \times NSF) \leq FVZ$$

$$f_{vy} = 3 \times VZ /(2 \times Area \times NSF) \leq FVY$$

Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *TR.52.1 Timber Design Parameter Specifications* (on page 2683)

D1.G.3 Input Specification

A typical set of input commands for STAAD.Pro timber design per AITC 1984 is listed below:

```
UNIT KIP INCH
PARAMETER
CODE TIMBER
GLULAM  1:16F-V3-DF/DF MEMB 1 TO 14
GLULAM  1:24F-V5-SP/SP MEMB 15 TO 31
```

```
GLULAM  20F-V1-DF/WW MEMB 32 TO 41
LAMIN  1.375 LY 168.0 MEMB 5 9 15 TO 31
LZ  176.0 MEMB 1 TO 4 6 7 8 10 TO 14
LUZ 322.6 ALL
LUY 322.6 ALL
WET  1.0 ALL
CDT  1.33
NSF  0.85
BEAM  1.0 ALL
CHECK  CODE 1 TO 14
SELECT  MEMB 15 TO 31
```

D1.G.3.1 Explanation of Input Commands and Parameters

Specify `PARAMETER` and then `CODE TIMBER` to start `TIMBER DESIGN` before specifying the input parameters. The user must provide the timber grade (`GLULAM GRADE`) for each member he intends to design. The parameters can be specified for all or specified list of members. If a parameter is not specified, the default value is assigned to it. See following `INPUT PARAMETERS LIST TABLE` for description and default values of the parameters.

D1.G.3.2 Glulam Grade and Allowable Stresses from Table

The allowable stresses for `GLULAM` members are read in from Table-1 and Table-2 of AITC for design values for Structural Glued Laminated Timber. The structural members are to be specified in the following manner:

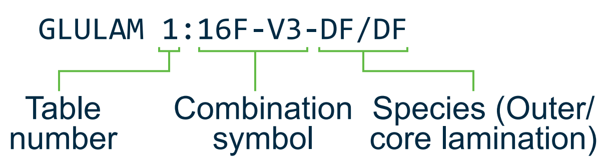


Figure 155: AITC Table-1 glulam members

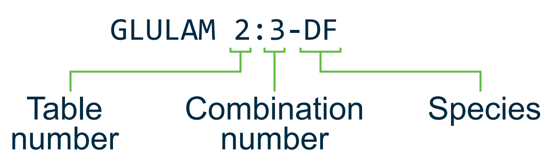


Figure 156: AITC Table-2 glulam members

For TABLE-2 members, the applicable stress values are selected based on the depth and the number of laminations.

Note: The lamination thickness (in inches) can be specified. Typical values are 1 3/8 inches or 1 1/2 inches. If not specified, a default value of 1 1/2 inches assumed by the program.

Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *TR.52.1 Timber Design Parameter Specifications* (on page 2683)

D1.G.4 Naming Conventions for Sections

The following conventions are used to describe timber sections in STAAD.Pro

D1.G.4.1 Dimensional Lumber sections

As can be seen from Tables 8.3 through 8.6 of the AITC 1994 manual, one or more of the following attributes have to be considered while choosing a section :

- Species
- Commercial Grade
- Size classification
- Nominal size of the section
- Grading rules agency

STAAD.Pro uses a naming convention that incorporates all of the above. Shown below is the name of a section that has characteristics as shown. It may be found on page 8-637 of the AITC 1994 manual.

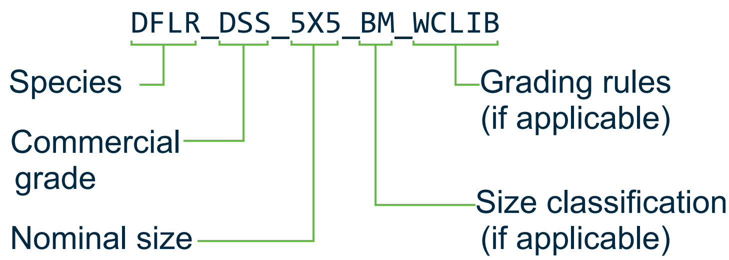


Figure 157: Timber naming conventions

Species: Douglas Fir Larch
Commercial Grade: Dense Select Structural
Size Classification: Beams
Nominal size: 5"x 5"
Grading Rules Agency: WCLIB

D1.G.4.2 Implementation of Glulam Properties

For glulam sections, each glulam designation has an associated value of modulus of elasticity and a set of allowable stresses. However, these values are not dependent on the size of the cross section. For example, a 3-1/8"x 6" 24F-V8 DF/DF beam and a 6-3/4" x 30" 24F-V8 DF/DF beam both have an E of 1,600 ksi and an allowable bending stress in the tension zone, F_{bx}, of 2,400 psi.

Therefore, in STAAD.Pro’s glulam database, the section sizes are not linked to the glulam type. Users may specify any cross-section size they choose and pick the desired glulam type. The Modulus of Elasticity and allowable stresses associated with that glulam are assigned to the member. The material properties for the glulam database are taken from Table 1 of AITC 117-93 – *Design Standard Specifications for Structural Glued Laminated Timber of Softwood Species*. This publication has been reproduced in the AITC 1994 manual starting from page 8-843.

Example for Dimensional Timber

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC DFLN_SS_4X4
```

```
E 273600
POISSON 0.15
DENSITY 0.025
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
3 4 7 8 TABLE ST DFLN_SS_4X4
CONSTANTS
MATERIAL DFLN_SS_4X4 MEMB 3 4 7 8
```

D1.G.4.3 Glulam Sections

The STAAD.Pro name for glulam sections incorporates

- Combination Symbol
- Species-Outer Laminations/Core Laminations

Shown here is a typical section listed in page 8-854 of the AITC manual.

```
GLT-24F-V11_DF/DFS
```

Implementation of Material Constants

As explained in the previous paragraphs, for sawn lumber as well as glulam sections, E is built into the database and gets automatically assigned to the member along with the section dimensions. Density, Poisson's ratio and Alpha (coefficient of thermal expansion) have to be assigned separately. If they are not assigned, the analysis engine will use default values for those.

Example for Glulam Timber

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_WET_DF/DF
E 191923
POISSON 0.15
DENSITY 0.025
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
8 PRIS YD 1.5 ZD 0.427083
CONSTANTS
MATERIAL GLT-24F-V8_WET_DF/DF MEMB 8
```

D1.G.5 Design Parameters

The timber design parameters for the AITC codes.

D1.G.5.1 AITC 1994 Parameters

Table 94: AITC 1994 Timber Design Properties

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
CB	C_b	1.0	Bearing Area Factor, Table 4.13
CFB	C_F	1.0	Size Factor for Allowable Bending Stress, see Table 8.3, 8.4, 8.5, 8.6, 8.7
CFC	C_F	1.0	Size Factor for Allowable Compression Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
CFT	C_F	1.0	Size Factor for Allowable Tension Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
CFU	C_{fu}	1.0	Flat Use Factor, see Table 4.9
CSS	C_H	1.0	Shear Stress Factor, Section 4.5.14
CMB	C_M	1.0	Wet service Factor for Allowable Bending Stress, see Table 4.8
CMC	C_M	1.0	Wet service Factor for Allowable Compression Parallel to Grain, see Table 4.8
CME	C_M	1.0	Wet service Factor for Modulus of Elasticity, see Table 4.8
CMP	C_M	1.0	Wet service Factor for Allowable Compression Perpendicular to Grain, see Table 4.8

Parameter Name		Default Value	Description
STAAD	**AITC 1994 Code**		
CMT	C_M	1.0	Wet service Factor for Allowable Tension Parallel to Grain, see Table 4.8
CMV	C_M	1.0	Wet service Factor for Allowable Shear Stress Parallel to Grain, see Table 4.8
CR	C_r	1.0	Repetitive Member Factor, see Section 4.5.10
CSF	C_F	1.0	Form Factor, see Section 4.5.12
CTM	C_t	1.0	Temperature Factor, see Table 4.11
CTT	C_T	1.0	Buckling Stiffness Factor, see Section 4.5.15
KB	K_b	1.0	Buckling Length Coefficient to calculate Effective Length
KBD	K_{bd}	1.0	Buckling Length Coefficient for Depth to calculate Effective Length
KBE	K_{bE}	0.609	Euler Buckling Coefficient for Beams, see Section 5.4.11
KCE	K_{cE}	1.0	Euler Buckling Coefficient for Columns, see Section 5.8.2
KEY	K_{ey}	1.0	Buckling Length Coefficient in Y Direction
KEZ	K_{ez}	1.0	Buckling Length Coefficient in Z Direction
KL	K_l	1.0	Load Condition Coefficient, Table 4.10

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
LZ	LZ	Member Length	Effective shear length in the z direction for Column Stability Check, Le=Ke*L
LY	LY	Member Length	Effective shear length in the y direction for Column Stability Check, Le=Ke*L
LUZ	LUZ	Member Length	Member length in the z direction for Beam Stability Check, Lu=Kb*l +Kbd*d
LUY	LUY	Member Length	Member length in the y direction for Beam Stability Check, Lu=Kb*l +Kbd*d
CDT	CDT	1.0	Load Duration Factor
CCR	CCR	1.0	Curvature factor (Section 4.5.11)
INDEX	INDEX	10	Exponent value in the Volume Factor Equation (Section 4.5.6)
CV	CV	1.0	Volume Factor (Section 4.5.6)
CC	CC	0.8	Variable in Column Stability Factor, Cp (Section 5.8.2, Eqn 5-14)
SRC	SRC	1.0	Slenderness ratio of Compression member
SRT	SRT	1.0	Slenderness ratio of Tension member
RATIO		1.0	Permissible ratio of actual to allowable stress

Parameter Name		Default Value	Description
STAAD	AITC 1994 Code		
BEAM		1.0	0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. **Note:** See D1.A.6 Design Parameters (on page 1100).

D1.G.5.2 AITC 1984 Parameters

Table 95: AITC 1985 Timber Design Parameters

Parameter Name	Default Value	Description
BEAM	1.0	0.0 = design at start and end nodes and those locations specified by the SECTION command. 1.0 = design at 13 evenly spaced points (i.e., 1/12th points) along member length, including start and end nodes. **Note:** See D1.A.6 Design Parameters (on page 1100).
CCR	1.0	Curvature factor
CDT	1.0	Duration of load factor
CSF	1.0	Form factor
CTM	1.0	Temp. factor
LAMINATION	1.50 inch	Thickness of lamination in inch (1.50 or 1.375)

Parameter Name	Default Value	Description
LUZ	1.92*L	Unsupported effective length for beam in z.
LUY	1.92*L	Unsupported effective length for beam in y.
LY	Member Length	Same as above in y-axis.
LZ	Member Length	Effective length of the column in z-axis.
NSF	1.0	Net section factor for tension members. (both shear and tension stresses are based on sectional area x nsf)
RATIO	1.0	Permissible ratio of actual to allowable stresses.
WET	0.0	0.0 - dry condition 1.0 - wet condition wet use factors are in-built

D1.G.6 Member Design Capabilities

STAAD.Pro is capable of performing member design functions for both 1984 and 1994 editions of AITC.

Tip: The User Interface can be used to easily assign design commands to members.

D1.G.6.1 Code Checking

The `CHECK CODE` command enables the user to check the adequacy of the size (`YD` X `ZD`) provided in the `MEMBER PROPERTIES` for the most critical forces and moments. The program prints whether the member has passed or failed, the critical conditions and the value of the ratio.

D1.G.6.2 Member Selection

Member selection is limited to AITC 1994.

The `SELECT MEMBER` command starts with the min. permissible depth (or min. depth provided thru `DMIN` parameter) and checks the code. If the member fails with this depth, the thickness is increased by one lamination thickness and the code requirements are checked again. The process is continued till the section passes all the code requirements. This ensures the least weight section for the member. If the depth of the section reaches max. allowable or available depth and the member still fails, you can have the following options for redesign:

a. Change the width or increase the max. allowable depth (`DMAX`)
b. Change the timber grade
c. Change the design parameters

Related Links

- *TR.52.2 Code Checking Specification* (on page 2684)

D1.G.7 Orientation of Lamination

Laminations are always assumed to lie along the local Z-plane of the member. In the `MEMBER PROPERTIES` section, YD always represents the depth of the section across the grain and ZD represents the width along the grain. This is in accordance with the sign convention conforming to `SET Y UP` (default).

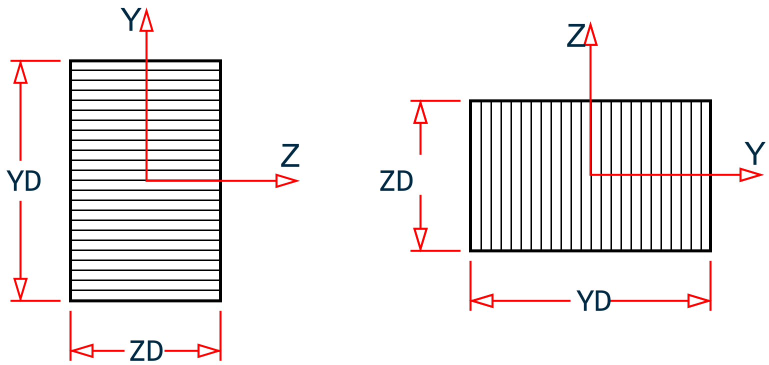


Figure 158: Orientation of lamination

D1.G.8 Tabulated Results of Member Design

For `CODE CHECKING` and/or `MEMBER SELECTION` the output results are printed as shown in the previous section. The items are explained as follows:

a. MEMBER refers to the member number for which the design is performed.

b. TABLE refers to the size of the `PRISMATIC` section (B X D or ZD X YD).

c. RESULT prints whether the member has PASSed or FAILed.

d. CRITICAL COND refers to the CLAUSE or FORMULA NO. from the TIMBER CONSTRUCTION MANUAL (3rd. Edition, AITC-1985) which governed the design. See following table:

Table 96: Critical conditions evaluated per AITC code

Critical Condition	Governing Criteria
Clause 5-19	Axial Compression and Bending with MINIMUM ECCENTRICITY.
Clause 5-18	Axial Compression and Bending
Clause 5-42	Axial Tension and Bending
Clause 5-24	Horizontal Shear
Clause 5.40	Lateral stability for net compressive stress in case of Tension and Bending.

e. RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. This ratio is usually the cumulative ratio of stresses in the interaction formula. In case of shear governing the design, it means the ratio of the actual shear stress to allowable shear stress. If this value exceeds the allowable ratio (default 1.0) the member is FAILed.

f. LOADING provides the load case number that governed.
g. FX, MY and MZ provide the design axial force, moment in local Y axes and moment in local Z axes respectively. FX value is followed by a letter C or T to denote COMPRESSION or TENSION.
h. LOCATION specifies the actual distance from the start of the member to the section where design forces govern in case BEAM command or SECTION command is specified.

OUTPUT parameters that appear within the box are explained as follows:

a. MEMB refers to the same member number for which the design is performed.
b. GLULAM GRADE refers to the grade of the timber.
c. LAM refers to lamination thickness provided in the input or assumed by the program. See INPUT PARAMETERS section.
d. LZ, LY, LUZ and LUY are the effective lengths as provided or calculated. See INPUT PARAMETERS section.
e. JZ and JY are the modifiers for the P-DELTA effect about Z-axis and Y-axis respectively. These are calculated by the program.
f. CDT, CSF, WET, CCR, CTM are the allowable stress modifiers explained in the INPUT PARAMETERS section.
g. CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively. CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively. These values are printed to help the user see the intermediate design values and re-check the design calculations.
h. f_a, f_{bz}, f_{by}, f_{vz} and f_{vy} are the actual axial stress, bending stresses about Z and Y axes and horizontal shear stresses about Z and Y axes respectively. If the bending moments about both axes are less then the eccentric moments based on min. eccentricity then bending stresses are calculated based on the min. eccentricity . Refer DESIGN OPERATIONS section for details.
i. FA, FBZ, FBY, FVZ, and FVY are the final allowable axial, bending (Z and Y axes) and horizontal shear (Z and Y axes) stresses. See D1.G.2 Allowable Stress per AITC Code (on page 1249) for details.

D1.G.8.1 Example Glulam Member Design

```
                         STAAD.Pro CODE CHECKING - (AITC)
                         ***********************


 ALL UNITS ARE - POUN FEET (UNLESS OTHERWISE NOTED)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================

     1    10.750X16.500 GLULAM GRADE:GLT-24F-V8_DF/DF
                         PASS      CL.5.9.2           0.014         3
                     4583.17 C          0.00       1310.87    12.0000
   |-----------------------------------------------------------------------|
   |   LEZ = 144.000  LEY = 144.000  LUZ = 144.000  LUY = 144.000 INCHES   |
   |                                                                       |
   | CD  = 1.000 CMB = 1.000 CMT = 1.000 CMC = 1.000 CMP = 1.000           |
   | CMV = 1.000 CME = 1.000 CFB = 1.000 CFT = 1.000 CFC = 1.000           |
   | CFU = 1.000 CR  = 1.000 CTT = 1.000 CC  = 1.000 CF  = 1.000           |
   | CT  = 1.000 CH  = 1.000 CB  = 1.000 CI  = 1.000 CV  = 0.000           |
   | CLY = 0.999 CLZ = 0.997 CP  = 0.934 c   = 0.900 E'  =  1600000.122 PSI |
   |                                                                       |
   | ACTUAL STRESSES : (POUND INCH)                                        |
   |                         fc    =       25.839 ft    =       0.000      |
```

```
|                      f_cby =        0.000 f_cbz =       32.249          |
|                      fv    =        0.000                               |
| ALLOWABLE STRESSES: (POUND INCH)                                        |
|                      FC    =     1541.320 FT    =        0.000          |
|                      FCBY =     1448.972 FCBZ  =     2340.789           |
|                      FCEY =        0.000 FCEZ  =     8780.903           |
|                      FBE   =     3727.248                               |
|                      FTB   =        0.000 F**TB =        0.000          |
|                      FV    =        0.000 SLENDERNESS =       50.000    |
|-------------------------------------------------------------------------|
```

D1.G.9 Examples

The following conventions are used to describe timber sections in STAAD.Pro

D1.G.9.1 Example for dimensional lumber

```
STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0;
5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7 8; 8 8 5;
9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;
UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC DFLR_SS_2X4
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
ISOTROPIC DFLR_SS_3X6
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
1 TO 4 9 TO 11 TABLE ST DFLR_SS_2X4
5 TO 8 12 13 TABLE ST DFLR_SS_3X6
CONSTANTS
MATERIAL DFLR_SS_2X4 MEMB 1 TO 4 9 TO 11
MATERIAL DFLR_SS_3X6 MEMB 5 TO 8 12 13
MEMBER RELEASE
9 TO 13 START MP 0.99
9 TO 13 END   MP 0.99
6 END MP 0.99
7 START MP 0.99
SUPPORTS
1 PINNED
5 FIXED BUT FX MZ
UNIT FEET POUND
LOAD 1 DEAD+LIVE LOAD
SELFWEIGHT Y -1
MEMBER LOAD
```

```
1 TO 4 UNI GY -30
5 TO 8 UNI GY -40
LOAD 2 SNOW LOAD
MEMBER LOAD
5 TO 8 UNI GY -50
LOAD 3 WIND LOAD
MEMBER LOAD
5 6 UNI Y -30
7 8 UNI Y 25
LOAD COMB 11 D+L+SNOW
1 1.0 2 1.0
LOAD COMB 12 D+L+SNOW+WIND
1 1.0 2 1.0 3 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
*Design per AITC
PARAMETER
CODE AITC
BEAM 1.0 ALL
CHECK CODE ALL
FINISH
```

D1.G.9.2 Example for Glulaminated lumber

```
STAAD PLANE EXAMPLE FOR GLULAM DESIGN
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 12 0 0; 3 24 0 0; 4 36 0 0; 5 0 12 0; 6 6 10 0; 7 18 6 0; 8 30 2 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 8 4; 8 1 5; 9 2 6; 10 3 7; 11 1 6;
12 2 7; 13 3 8;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_DF/DF
E 1600
POISSON 0.15
DENSITY 1.44676e-005
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 7 PRIS YD 16.5 ZD 10.75
8 TO 13 PRIS YD 10.5 ZD 8.75
CONSTANTS
MATERIAL GLT-24F-V8_DF/DF MEMB 1 TO 13
SUPPORTS
1 4 PINNED
UNIT POUND FEET
LOAD 1  DEAD
SELFWEIGHT Y -1
LOAD 2  LIVE
MEMBER LOAD
1 TO 3 UNI GY -100
4 TO 7 UNI GY -100
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER
```

```
CODE AITC
CMT 1 ALL
RATIO 0.9 ALL
CHECK CODE ALL
FINISH
```

D1.H. American Codes - Aluminum Design per 1994 ADM

STAAD.Pro is capable of performing aluminum member design based on the ASD 1994 *Specifications for Aluminum Structures*, Sixth Edition (October, 1994) included in the *Aluminum Design Manual* 1994.

D1.H.1 Member Properties

In order to do this design in STAAD.Pro, the members in the structure must have their properties specified from Section VI of the above-mentioned manual. The section names are mentioned in Tables 5 through 28 of that manual. All of those tables except Table 10 (Wing Channels) and Table 20 (Bulb Angles) are available in STAAD.Pro.

Described below is the command specification for various sections:

D1.H.1.1 Standard single section

```
memb-list  TA ST section-name
```

Example

```
1 TO 5 TA ST CS12X11.8
9 TA ST I8.00X13.1
11 33 45 67 TA ST LS8.00X8.00X0.625
18 TA ST 1.50PipeX160
15 TA ST T(A-N)6.00X8.00X11.2
23 25 29 TA ST 20X12RectX.500Wall
```

D1.H.1.2 Double channel back-to-back

```
memb-list  TA BACK section-name  SPACING value
```

Example

```
3 TA BACK C(A-N)7X3.61 SPACING 1.5
5 TA BACK C15X17.33 SP 0.75
```

D1.H.1.3 Double channel front-to-front

```
memb-list  TA FRONT section-name  SPACING value
```

Example

```
2 TA FRONT CS12X10.3 SP 1.0
4 TA FR CS10X10.1 SP 0.5
```

D1.H.1.4 Double angle long leg back-to-back

```
memb-list  TA LD section-name  SPACING value
```

Example

```
14 TA LD LS4.00X3.00X0.375 SP 1.5
```

D1.H.1.5 Double angle short leg back-to-back

```
memb-list  TA SD section-name  SPACING value
```

Example

```
12 TA SD L3.5X3X0.5 SP 0.25
13 TA SD L8X6X0.75 SP 1.0
```

D1.H.2 Design Procedure

The design is done according to the rules specified in Sections 4.1, 4.2 and 4.4 on pages I-A-41 and I-A-42 of the Aluminum code. The allowable stresses for the various sections are computed according to the equations shown in Section 3.4.1 through 3.4.21 on pages I-A-27 through I-A-40. The adequacy of the member is checked by calculating the value of the left-hand side of equations 4.1.1-1, 4.1.1-2, 4.1.1-3, 4.1.2-1, 4.4-1 and 4.4-2. This left-hand side value is termed as RATIO. If the highest RATIO among these equations turns out to be less than or equal to 1.0, the member is declared as having PASSed. If it exceeds 1.0, the member has FAILed the design requirements.

Note: The check for torsion per Clause 4.3 for open sections is currently not implemented in STAAD.Pro.

D1.H.3 Design Parameters

The following are the parameters for specifying the values for variables associated with the design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 97: Aluminum Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as ALUMINUM Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
ALCLAD	0	Defines if material is Alclad. 0 - Material used in the section is not an Alclad. 1 - Material used in the section is an Alclad.

Parameter Name	Default Value	Description
ALLOY	34	This variable can take on a value from 1 through 40. The default value represents the alloy 6061-T6. See Table 14A.2 below for a list of values for this parameter and the alloy they represent. Table 3.3-1 in Section I-B of the Aluminum specifications provides information on the properties of the various alloys.
BEAM	0.0	If this parameter is set to 1.0, the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details). If neither the BEAM parameter nor any SECTION command is specified, STAAD will terminate the run and ask the user to provide one of those 2 commands. This rule is not enforced for TRUSS members.
DMAX	1000 in.	Maximum depth permissible for the section during member selection. This value must be provided in the current units.
DMIN	0.0 in	Minimum depth required for the section during member selection. This value must be provided in the current units.

Parameter Name	Default Value	Description
KT	1.0	Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the allowable stress in axial compression. See Equation 3.4.7.2-6 on page I-A-28 of the Aluminum specifications for details.
KY	1.0	Effective length factor for overall column buckling in the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.
KZ	1.0	Effective length factor for overall column buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.
LT	Member length	Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the allowable stress in axial compression. See Equation 3.4.7.2-6 on page I-A-28 of the Aluminum specifications for details.

Parameter Name	Default Value	Description
LY	Member length	Effective length for overall column buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.
LZ	Member length	Effective length for overall column buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.
PRODUCT	1	This variable can take on a value from 1 through 4. They represent: 1 - All 2 - Extrusions 3 - Drawn Tube 4 - Pipe The default value stands for All. The PRODUCT parameter finds mention in Table 3.3-1 in Section I-B of the Aluminum specifications.

Parameter Name	Default Value	Description
SSY	0.0	Factor that indicates whether or not the structure is subjected to sidesway along the local Y axis of the member. The values are: 0 - Sidesway is present along the local Y-axis of the member 1 - There is no sidesway along the local Y-axis of the member. The sidesway condition is used to determine the value of Cm explained in Section 4.1.1, page I-A-41 of the Aluminum specifications.
SSZ	0.0	Factor that indicates whether or not the structure is subjected to sidesway along the local Z axis of the member. The values are: 0 - Sidesway is present along the local Z-axis of the member 1 - There is no sidesway along the local Z-axis of the member. The sidesway condition is used to determine the value of Cm explained in Section 4.1.1, page I-A-41 of the Aluminum specifications.
STIFF	Member length	Spacing in the longitudinal direction of shear stiffeners for stiffened flat webs. It is input in the current units of length. See section 3.4.21 on page I-A-40 of the Aluminum specifications for information regarding this parameter.

Parameter Name	Default Value	Description
STRUCTURE	1	In Table 3.4-1 in Section I-A of the Aluminum specifications, it is mentioned that the value of coefficients nu, ny and na are dependent upon whether the structure being designed is a building or a bridge. Users may convey this information to STAAD using the parameter STRUCTURE. The values that can be assigned to this parameter are: 1 - Buildings and similar type structures 2 - Bridges and similar type structures
TRACK	2	This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 1 - Prints only the member number, section name, ratio, and PASS/FAIL status. 2 - Prints the design summary in addition to that printed by TRACK 1 3 - Prints the member properties and alloy properties in addition to that printed byTRACK 2. 4 - Prints the values of variables used in design in addition to that printed by TRACK 3.
UNL	Member length	Distance between points where the compression flange is braced against buckling or twisting. This value must be provided in the current units. This value is used to compute the allowable stress in bending compression.

Parameter Name	Default Value	Description
<u>WEL</u>D	0	In Table 3.4-2 in Section I-A of the Aluminum specifications, it is mentioned that the value of coefficients Kt and Kc are dependent upon whether or not, the location of the section where design is done is within 1.0 inch of a weld. The WELD parameter is used in STAAD for this purpose. The values that can be assigned to this parameter are: 0 - Region is farther than 1.0in from a weld 1 - Region is within 1.0in from a weld

D1.H.3.1 Aluminum Alloys available in STAAD

Table 98: Alloy Parameters

Value	Name
1	1100-H12
2	1100-H14
3	2014-T6
4	2014-T6510
5	2014-T6511
6	2014-T651
7	3003-H12
8	3003-H14
9	3003-H16
10	3003-H18
11	3004-H32
12	3004-H34
13	3004-H36

Value	Name
14	3004-H38
15	5005-H12
16	5005-H14
17	5005-H32
18	5005-H34
19	5050-H32
20	5050-H34
21	5052-H32
22	5052-H34
23	5083-H111
24	5086-H111
25	5086-H116
26	5086-H32
27	5086-H34
28	5454-H111
29	5454-H112
30	5456-H111
31	5456-H112
32	6005-T5
33	6105-T5
34	6061-T6
35	6061-T6510
36	6061-T6511
37	6061-T651
38	6063-T5
39	6063-T6

Value	Name
40	6351-T5

D1.H.4 Code Checking

The purpose of code checking is to determine whether the initially specified member properties are adequate to carry the forces transmitted to the member due to the loads on the structure. Code checking is done at the locations specified by either the `SECTION` command or the `BEAM` parameter described above.

See T.1 – Steel Portal Frame (on page 387) for STAAD provides an example on the usage of the `CHECK CODE` command.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D1.H.4.1 Example

Sample input data for Aluminum Design

```
PARAMETER
CODE ALUMINUM
BEAM 1 ALL
KY 1.2 MEMB 3 4
ALLOY 35 ALL
PRODUCT 2 ALL
TRACK 3 ALL
SELECT ALL
PARAMETER
CODE ALUMINUM
ALCLAD 1 ALL
STRUCT 1 ALL
CHECK CODE ALL
```

D1.H.5 Member Selection

The member selection process involves the determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it.

See T.1 – Steel Portal Frame (on page 387) for STAAD provides an example on the usage of the `SELECT MEMBER` command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.I. American Codes - Steel Design per ASCE Transmission Tower Codes

D1.I.1. American Transmission Tower Code - Steel Design per ASCE 10-97

STAAD.Pro is capable of performing steel design based on the American Transmission Tower code ASCE 10-97 *Design of Latticed Steel Transmission Structures*.

D1.I.1.1 General Comments

The ASCE 10-97 code is meant to supercede the older edition of the code, available under the name ASCE Publication 52. However, in the interests of backward compatibility, both codes are currently accessible in STAAD.Pro.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. Design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) is not supported.

To use the ASCE 52 code, use the commands

```
PARAMETER
CODE ASCE 52
```

To use the ASCE 10-97 code, use the commands

```
PARAMETER
CODE ASCE
```

The detailing requirements, such as provisioning of stiffeners and checking the local effects like flange buckling, web crippling, etc. must be performed manually. It is assumed that you are familiar with the basic concepts of Steel Design facilities available in STAAD. Please refer to D. Steel Design (on page 969) for detailed information on this topic.

D1.I.1.2 Allowable Stresses per ASCE 10-97

Member selection and code checking operations in the STAAD implementation of ASCE 10-97 are done to resist loads at stresses approaching yielding, buckling, fracture and other limiting conditions specified in the standard. Those stresses are referred to in the standard as Design Stresses. The appropriate sections of the ASCE standard where the procedure for calculating the design stresses is explained are as follows.

D1.I.1.2.1 Design Axial Tensile Stress

Design tensile stresses are calculated on the basis of the procedure described in section 3.10. The NSF parameter (see the Parameters table shown later in this section) may be used if the section area needs to be reduced to account for bolt holes.

D1.I.1.2.2 Design Axial Compressive Stress

Design compressive stress calculation is based on the procedures of section 3.6 through 3.9. For angle members under compression, the procedures of sections 3.7 and 3.8 have been implemented. Capacity of the section is computed for column buckling and wherever applicable, torsional buckling. The user may control the effective lengths for buckling using the LT, LY, LZ and/or KT, KY, KZ parameters (see the Parameters table shown later in this section).

D1.I.1.2.3 Design Bending Compressive Stress

Calculations for design bending compressive stress about the major axis and minor axis are based on the procedures of section 3.14. Procedures outlined in sections 3.14.1 through 3.14.6 have been implemented.

D1.I.1.2.4 Design Bending Tensile Stress

Calculations for design bending tensile stress about the major and minor axis are based on the procedures of section 3.14.2.

D1.I.1.2.5 Design Shear Stress

Calculation of the design shear stress is based on the procedure outlined in section 3.15 of the ASCE 10-97. The procedure of section 3.15.2 is followed for angles and the procedure of section 3.15.1 is followed for all other sections.

D1.I.1.3 Critical Conditions used as criteria to determine Pass/Fail status

These are Clause 3.4 for slenderness limits, Clause 3.12 for Axial Compression and Bending, Clause 3.13 for Axial Tension and Bending, Clause 3.9.2 for Maximum w/t ratios and Clause 3.15 for Shear.

D1.I.1.4 Design Parameters

Design parameters are summarized in the table shown later in this section. These parameters may be used to control the design process to suit specific modeling needs. The default parameter values have been selected such that they are frequently used numbers for conventional design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 99: Steel Design Parameters for ASCE 10-97

Parameter Name	Default Value	Description
CODE	-	Must be specified as ASCE to design per ASCE 10-97. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	0 = Perform design at beam ends and section locations specified according to the SECTION command 1 = Perform design at the ends and eleven intermediate sections of the beam
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y and z axes as defined in equation 3.12-1 on p.10 of ASCE 10-97.
DMAX	45.0 in.	Maximum allowable depth for member selection

Parameter Name	Default Value	Description
DBL	0.75 in.	Diameter of bolt for calculation of number of bolts required and the net section factor.
DMIN	0.0 in.	Minimum allowable depth for member selection
ELA	4	Indicates what type of end conditions are to be used from among Equations 3.7-4 thru 3.7-7 to determine the KL/R ratio. 1. EQN.3.7-4, Page 4 2. EQN.3.7-5, Page 4 3. EQN.3.7-6, Page 4 4. EQN.3.7-7, Page 5 **Note:** Value 1 is valid for leg members only.
ELB	1	Indicates what type of end conditions are to be used from among Equations. 3.7-8 thru 3.7-10 and 3.7-12 thru 3.7-14 to determine the KL/R ratio. 1. EQN.3.7-8, Page 5, EQN. 3.7-12, Page 5 2. EQN.3.7-9, Page 5, EQN. 3.7-13, Page 5 3. EQN.3.7-10, Page 5, EQN. 3.7-14,Page 5
FVB	30 KSI	Shear strength of bolt.
FYB	36 KSI	Yield strength of bolt.
FYLD	36.0 KSI	Yield Strength of steel
KT	1.0	Effective length coefficient for warping restraint (clause 3.14.4, p. 11)
KY	1.0	Effective length factor (K) for compression buckling about the Y-axis (minor axis)

Parameter Name	Default Value	Description
KZ	1.0	Effective length factor (K) for compression buckling about the Z-axis (major axis)
LEG	0.0	This parameter is meant for plain angles. 0. indicates that the angle is connected by both legs and allowable stress in axial tension is 1.0FYLD. 1. indicates that the angle is connected only by the shorter leg and allowable tensile stress is computed per clause 3.10.2 as 0.9FYLD. 2. indicates that the angle is connected by the longer leg.
LT	Member Length	Effective length for warping.
LY	Member Length	Length to calculate slenderness ratio for buckling about the Y-axis (minor axis)
LZ	Member Length	Length to calculate slenderness ratio for buckling about the Z-axis (major axis)
MAIN	2	Parameter that indicates the member type for the purpose of calculating the KL/R ratio (see clause 3.4, PAGE 3, ASCE 10-97) 1. Leg member, KL/R ≤ 150 2. Compression member, KL/R ≤ 250 3. Tension member, KL/R ≤ 500 4. Hanger member, KL/R ≤ 375 (Clause 3C.4, page 31) 5. Redundant member, KL/R ≤ 250 10. Do not perform the KL/R Check

Parameter Name	Default Value	Description
NHL	0	Number of bolt holes on the cross section that should be used to determine the net section factor for tension capacity.
NSF	1.0	Net section factor for tension members
RATIO	1.0	Permissible ratio that determines the cut off point for pass/fail status. A value below this quantity indicates PASS while a value greater than this quantity indicates FAILURE.
SSY	0.0	0.0 = Sidesway in local y-axis 1.0 = No sidesway
SSZ	0.0	Same as above except in local z-axis
TRACK	0.0	0.0 = Suppresses printing of allowable stresses 1.0 = Prints all allowable stresses
UNB	Member Length	Unsupported length of the bottom flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.
UNF	1.0	Same as UNL, but provided as a fraction of the member length
UNL	Member Length	Unsupported length of member for calculation of allowable bending stress
UNT	Member Length	Unsupported length of the top flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.

Notes:

- All values must be provided in the current unit system.

- If ELA and ELB are both defined for a member, only one of them will be used for the slenderness calculation depending on the member type (as defined by the MAIN parameter) for that member.

D1.I.1.5 Code Checking and Member Selection

Both code checking and member selection options are available in the ASCE 10-97 implementation. In general, it may be noted that the concepts followed in MEMBER SELECTION and CODE CHECKING procedures are similar to that of the AISC based design.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.I.2. American Transmission Tower Code - Steel Design per ASCE Manuals and Reports

STAAD.Pro is capable of performing steel design based on the ASCE Manuals and Reports on Engineering Practice No. 52 – *Guide for Design of Steel Transmission Towers*, Second Edition

D1.I.2.1 General Comments

The design philosophy and procedural logistics for member selection and code checking is based upon the principles of allowable stress design. Two major failure modes are recognized: failure by overstressing and failure by stability considerations.

The following sections describe the salient features regarding the process of calculation of the relevant allowable stresses and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economical section is selected based on the least weight criteria. The code checking part of the program also checks the slenderness requirements, the minimum metal thickness requirements, and the width-thickness requirements.

The detailing requirements, such as provisioning of stiffeners and checking the local effects like flange buckling, web crippling, etc. must be performed manually. It is assumed that you are familiar with the basic concepts of Steel Design facilities available in STAAD. Please refer to D. Steel Design (on page 969) for detailed information on this topic.

D1.I.2.2 Allowable Stresses per ASCE (Pub. 52)

The member design and code checking in the STAAD implementation of ASCE (Pub. 52) is based upon the allowable stress design method. Appropriate sections of this publication are referenced below.

D1.I.2.2.1 Allowable Axial Tensile Stress

Allowable tensile stresses are calculated on the basis of the procedure described in section 4.10. The NSF parameter (Refer to D1.I.2.3 Design Parameters (on page 1280)) may be used if the net section area needs to be used.

D1.I.2.2.2 Allowable Axial Compressive Stress

Allowable compressive stress calculation is based on the procedures of section 4.6 through 4.9. For angle members under compression, the procedures of sections 4.7 and 4.8 have been implemented. Capacity of the section is computed for column buckling and wherever applicable, torsional buckling. The user may control the effective lengths for buckling using the LX, LY, LZ and/or KX, KY, KZ parameters (Refer to D1.I.2.3 Design Parameters (on page 1280)).

D1.I.2.2.3 Allowable Bending Compressive Stress

Calculations for allowable bending compressive stress about the major axis and minor axis are based on the procedures of section 4.14. Procedures outlined in sections 4.14.1 through 4.14.6 have been implemented.

D1.I.2.2.4 Allowable Bending Tensile Stress

Calculations for allowable bending tensile stress about the major and minor axis are based on the procedures of Section 4.14.2.

D1.I.2.2.5 Allowable Shear Stress

Calculation of the allowable shear stress is based on the procedure outlined in section 4.15 of the ASCE Pub. 52. The procedure of section 4.15.2 is followed for angles and the procedure of section 4.15.1 is followed for all other sections.

D1.I.2.2.6 Critical Conditions used as criteria to determine Pass/Fail status

These are Clause 4.4 for slenderness limits, Equation 4.12-1 for Axial Compression and Bending, Equation 4.13-1 for Axial Tension and Bending, Clause 4.9.2 for Maximum w/t ratios and Clause 4.15 for Shear.

D1.I.2.3 Design Parameters

These parameters may be used to control the design process to suit specific modeling needs. The default parameter values have been selected such that they are frequently used numbers for conventional design.

Table 100: Steel Design Parameters for ASCE (Pub. 52) Based Design

Parameter Name	Default Value	Description
CODE	-	Must be specified as ASCE 52. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BEAM	0.0	Specifies locations along member length at which member design is designed. 2.0 = use the section locations specified according to the SECTION command 3.0 = at the ends and eleven intermediate sections of the beam
DBL	0.75 in.	Diameter of bolt for calculation of number of bolts required and the net section factor.

Parameter Name	Default Value	Description
DMAX	45.0 in.	Maximum allowable depth for member selection
DMIN	0.0 in.	Minimum allowable depth for member selection
ELA	4	Indicates what type of end conditions are to be used from among Equations 4.7-4 thru 4.7-7 to determine the KL/R ratio. 1 = EQN.4.7-4, Page 26 (Valid for leg members only) 2 = EQN.4.7-5, Page 27 3 = EQN.4.7-6, Page 27 4 = EQN.4.7-7, Page 27
ELB	1	Indicates what type of end conditions are to be used from among Equations. 4.7-8 thru 4.7-10 to determine the KL/R ratio. 1 = EQN.4.7-8, Page 27, EQN. 4.7-12, Page 28 2 = EQN.4.7-9, Page 27, EQN. 4.7-13, Page 28 3 = EQN.4.7-10, Page 27, EQN. 4.7-14,Page28
FVB	30 KSI	Shear strength of bolt.
FYB	36 KSI	Yield strength of bolt.
FYLD	36.0 KSI	Yield Strength of steel
KT	1.0	Effective length coefficient for warping restraint (clause 4.14.4, pg 36)
KY	1.0	Effective length factor (K) for compression buckling about the Y-axis (minor axis)
KZ	1.0	Effective length factor (K) for compression buckling about the Z-axis (major axis)

Parameter Name	Default Value	Description
LEG	0.0	This parameter is meant for plain angles. 3.0 = the angle is connected by both legs and allowable stress in axial tension is 1.0Â·FYLD 4.0 = the angle is connected only by the shorter leg and allowable tensile stress is computed per Cl. 4.10.2 as 0.9Â·FYLD 5.0 = the angle is connected by the longer leg
LT	Member Length	Effective length for warping.
LY	Member Length	Length to calculate slenderness ratio for buckling about the Y-axis (minor axis)
LZ	Member Length	Length to calculate slenderness ratio for buckling about the Z-axis (major axis)
MAIN	2	Parameter that indicates the member type for the purpose of calculating the KL/R ratio (See Cl. 4.4, p. 25) 1 = Leg member (KL/r ≤ 150) 2 = Compression member (KL/r ≤ 200) 3 = Tension member (KL/r ≤ 500) 4 = Hanger member per Cl. 4C.4, p. 43 (KL/r ≤ 375) 5 = Redundant member (KL/r ≤ 250) 10 = Do *not* perform the slenderness (KL/r) check
NHL	0	Number of bolt holes on the cross section that should be used to determine the net section factor for tension capacity.
NSF	1.0	Net section factor for tension members

Parameter Name	Default Value	Description
RATIO	1.0	Permissible ratio that determines the cut off point for pass/fail status. A value below this quantity indicates PASS while a value greater than this quantity indicates FAILURE.
TRACK	0.0	Level of detail in output 0.0 = Suppresses printing of allowable stresses 1.0 = Prints all allowable stresses
UNF	1.0	Same as `UNL`, but provided as a fraction of the member length
UNL	Member Length	Unsupported length of member for calculation of allowable bending stress

D1.I.2.4 Code Checking and Member Selection

Both code checking and member selection options are available in the ASCE Pub. 52 implementation. In general, it may be noted that the concepts followed in `MEMBER SELECTION` and `CODE CHECKING` procedures are similar to that of the AISC based design.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.J. American Codes - Steel Design per API 2A-WSD 2000

The API Steel Design facility in STAAD.Pro is based on the API 2A-WSD standard, titled *Recommended Practice for Planning, Design and Constructing Fixed Offshore Platforms-Working Stress Design*, 21st Edition (December 2000). Joint checks includes “Errata and Supplements” 1, 2 & 3 of the code.

D1.J.1 Design Operations

STAAD.Pro contains a broad set of facilities for the design of structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design;
- Specify whether to perform code checking or member selection;

- Specify design parameter values, if different from the default values; and
- Specify design parameters to carry out joint checks.

These operations may be repeated any number of times depending upon the design requirements.

The basic process is as follows:

1. Define the STAAD model geometry, loading, and analysis.
2. Run the analysis and API design which creates the Geometry file (file extension `.PUN`) and give preliminary design results.
3. Check and modify the Geometry file as necessary.
4. Re-run the analysis to read the modified Geometry file for the final design results.

D1.J.1.1 Limitations

The parameter `SELECT 1.0` should not be used while carrying out punching shear checks. It can be used in initial runs for member selection.

No classification of the joint is performed using the loading. For the initial run of an API code check, all joints will be assumed to be a T/Y joint. Refer to D1.J.8 Joint Design (on page 1293) for details.

No hydrostatic checks are performed.

D1.J.1.2 Truss Members

A truss member is capable of carrying only axial force. So in design, no time is wasted calculating the allowable bending or shear stresses, thus reducing design time considerably. Therefore, if there is any truss member in an analysis (like bracing or strut, etc.), it is wise to declare it as a truss member rather than as a regular frame member with both ends pinned.

D1.J.2 Allowables per API Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Petroleum Institute (API-RP2A) Code. The 21st edition of API Code, as published in 2007, is used as the basis of this design (except for tension stress).

Note: Open sections are not applicable to API code. Therefore, any open sections added to API Code design block will be designed per the D1.B. American Codes - Steel Design per AISC 9th Edition (on page 1130).

D1.J.2.1 Tension Stress

Allowable tension stresses, as calculated in STAAD, are based on the API Code, clause (3.2.1-1).

Allowable tension stress on the net section

$$F_t = 0.60 \cdot F_y$$

D1.J.2.2 Shear Stress

Beam Shear Stress

Allowable beam shear stress on the gross section must conform to Clause 3.2.4-2 of the API code:

$$F_v = 0.4 \cdot F_y$$

The maximum applied beam shear stress is per Eqn 3.2.4-1:

$f_v = V / 0.5\ A$

Torsional Shear Stress

Allowable torsional shear stress per Eqn. 3.2.4-4:

$F_{vt} = 0.4 \cdot F_y$

F_{vt} is the maximum torsional shear stress per Clause 3.2.4-3 of the API code.

D1.J.2.3 Stress Due to Compression

The allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula 3.2.2-1 in the API Code when the largest effective slenderness ratio, Kl/r is less than or equal to C_c. If Kl/r exceeds C_c, then the allowable compressive stress is increased as per formula (3.2.2-2) of the Code.

Where:

$$C_c = \sqrt{2\pi^2 \frac{E}{F_y}}$$

For $D/t > 60$, the lesser of F_{xe} or F_{xc} is substituted for F_{xy}.

Where:

F_{xe} = the elastic local buckling stress calculated with C, the critical elastic buckling coefficient = 0.3 (3.2.2-3)
F_{xc} = the inelastic local buckling stress. (3.2.2-4)

D1.J.2.4 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy API formula 3.3.1-1 and 3.3.1-2 when $f_a/F_a > 0.15$, otherwise formula 3.3.1-3 applies. It should be noted that during code checking or member selection, if $f_a/F_a > 1.0$, the program does not compute the second 3.3.1-1/2.

D1.J.2.5 Bending Stress

The allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Clause 3.2.3 of the API code, is:

a. When $D/t \le 1{,}500/F_y$ (Imperial Units),

$F_b = 0.75F_y$

b. When $1{,}500/F_y < D/t \le 3{,}000/F_y$ (Imperial Units),

$F_b = [0.84 - 1.74\ F_y D/(Et)]F_y$

c. When $3{,}000/F_y < D/t \le 300$ (Imperial Units),

$F_b = [0.72 - 0.58\ F_y D/(Et)]F_y$

D1.J.2.6 Simple Joints: Capacity Checks

A typical joint and the terms involved with the joint checks are given below:

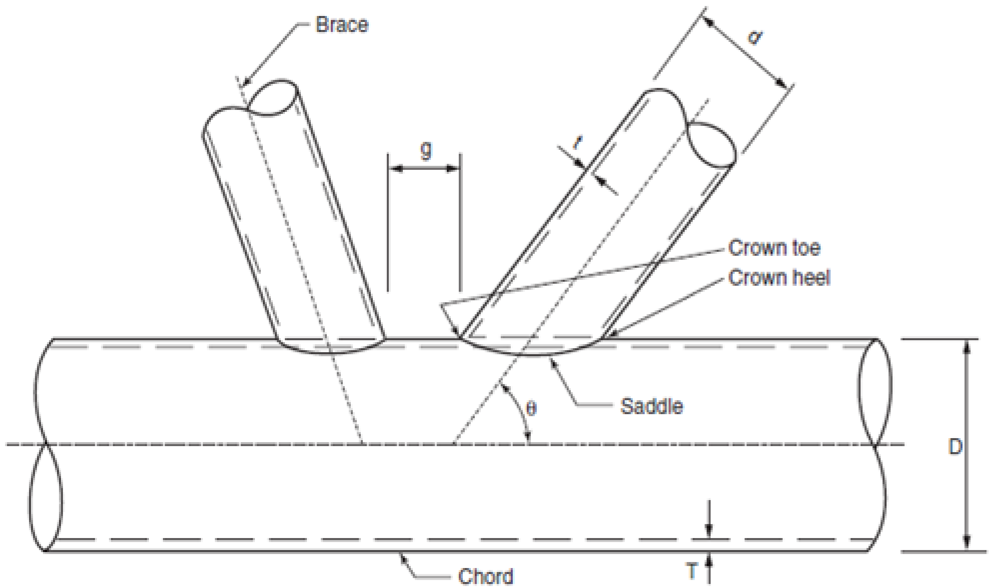


Figure 159: Simple joint diagram

Definitions

θ = Brace included rage
g = Gap between braces
t = Brace wall thickness at intersection
T = Chord wall thickness at intersection
d = Brace outside diameter
D = Chord outside diameter

$$\beta = d/D$$

$$\gamma = D/(2T)$$

$$\tau = t/T$$

Joint Validity

The validity range of the joints that are identified will be checked as per Cl. 4.3.1 of the code. The conditions to be checked for each joint are as given below:

$$0.2 \leq \beta \leq 1.0$$

$$10 \leq \gamma \leq 50$$

$$30° \leq \theta \leq 90°$$

F_y = 90 ksi (500 MPa)
g/D > -0.6 (for K joints)

If any of these conditions are not satisfied for the joint under consideration, the programissues a warning message corresponding to the invalid parameter(s). The program will, however, perform the joint checks as the code allows for the design of such joints with modified values of yield strength. You can use the FYLD parameter to reset the yield strength.

Joint Capacity

The capacity of the joint, both the axial capacity and the moment capacity is

The allowable capacity for brace axial load, P_a, is evaluated as:

$$P_a = Q_u Q_f \frac{F_{yc} T^2}{\mathrm{FSJ} \sin\theta}$$

The allowable capacity for brace bending moment, M_a, is evaluated as:

$$M_a = Q_u Q_f \frac{F_{yc} T^2 d}{\mathrm{FSJ} \sin\theta}$$

Where:

F_y = the yield stress of the chord member at the joint (or 0.8 of the tensile stress, if less)
FSJ = the factor of safety parameter (1.6 by default)

Q_u and Q_f are the strength factor and the Chord factor that are to be determined based on the joint type. The strength factor, Q_u, is to be determined as given in Section 4.3.3 of the code (ref. Table. 4.3-1 of the API code).

$$Q_f = \left[1 + C_1 \left(\frac{FSJP_c}{P_y} \right) - C_2 \left(\frac{FSJM_c}{M_p} \right) - C_3 A^2 \right]$$

$$A = \sqrt{ \left(\frac{FSJP_c}{P_y} \right)^2 + \left(\frac{FSJM_c}{M_p} \right)^2 }$$

P_c = axial load

$$M_c = \sqrt{M_{ipb}^2 + M_{opb}^2}$$

C_1, C_2, and C_3 are factors determined by the following table:

Joint Type		C_1	C_2	C_3
K joints under brace axial loading		0.2	0.2	0.3
T/Y joints under brace axial loading		0.3	0	0.8
X joints under brace axial loading	β ≤ 0.9	0.2	0	0.5
	β = 1.0	-0.2	0	0.4
All joints under brace moment loading		0.2	0	0.4

Note: For values of β between 0.9 and 1.0, coefficients are linearly interpolated between listed values.

For joints that are a mixture of K, X, or Y joints, the capacity of the joint is evaluated as a weighted average of the capacities of each joint.

In case the joint is subjected to combined axial load and bending moments (in-plane and/or out-of-plane), the program performs the following interaction check as given by Cl 4.3.6 of the code:

$$\left| \frac{P}{P_a} \right| + \left(\frac{M}{M_a} \right)_{ipb}^2 + \left| \frac{M}{M_a} \right|_{opb} \le 1.0$$

D1.J.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in Table 22A.1. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the `KZ` value (k value in local z-axis) of a member is set to 1.0, wile in the real structure it may be 1.5. In that case, the `KZ` value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the `TRACK` value of a member is set to 0.0, which means no allowable stresses of the member will be printed.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 101: American (API) Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `API` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	Beam parameter: 0.0 = design only for end moments or those at locations specified by the SECTION command. 1.0 = calculate moments at twelfth points along the beam, and use the maximum Mz location for design. 2.0 = Same for `BEAM 1.0`, but additional check is made at each end.
CB	1.0	Cb value as used in Section 1.5 of AISC 0.0 = Cb value to be calculated Any other value will mean the value to be used in design
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes

Parameter Name	Default Value	Description
DMAX	100.0 in	Maximum allowable depth
DMIN	0.0	Minimum allowable depth
FSJ	1.6	Factor of safety used for joint checks.
FYLD	36 ksi	Yield strength of steel.
KY	1.0	K value in local y-axis. Typically the minor axis.
KZ	1.0	K value in local z-axis. Typically the major axis.
LY	Member Length	Length in local Y-axis to calculate slenderness ratio.
LZ	Member Length	Length in local Z-axis to calculate slenderness ratio.
MAIN	0.0	Design for slenderness. 1.0 = Main member 2.0 = Secondary member
NSF	1.0	Net section factor for tension members.
RATIO	1.0	Permissible ratio of the actual to allowable stresses
SSY	0.0	Design for sidesway. 0.0 = Sidesway in local y-axis 1.0 = No sidesway
SSZ	0.0	Design for sidesway in local z-axis
TRACK	0.0	Controls the level of detail in the output: 0.0 = Print design output at the minimum level of detail. 1.0 = Print design output at an intermediate level of detail. 2.0 = Print design output at the maximum level of detail.

Parameter Name	Default Value	Description
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.
WELD	1	Weld type, as explained in section 3.1.1 of the API code. 1.0 = Welding is one side only except for wide flange or tee sections, where the web is always assumed to be welded on both sides. 2. 0 = Welding is both sides. For closed sections like pipe or tube, the welding will be only on one side.
WMIN	1.16 in.	Minimum thickness
WSTR	0.4 X FLYD	Allowable welding stress

Note: The parameters DMAX and DMIN are only used for member selection.

D1.J.4 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate as per API. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks, the critical condition of API code (like any of the API specifications for compression, tension, shear, etc.), the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value), the governing load case, and the location (distance from the start of the number of forces in the member) where the critical condition occurs.

Code checking can be done with any type of steel section listed in D1.A.3 Member Properties (on page 1088) .

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D1.J.5 Member Selection

The program is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters `DMAX` and `DMIN` which limits the maximum and minimum depth of the members.

- Member selection can be performed with all types of hollow steel sections.
- Selection of members whose properties are originally input from a user created table will be limited to sections in the user table.
- Member selection cannot be performed on members whose section properties are input as prismatic.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.J.6 Chord Selection and Q_f Parameter

Q_f is a factor to account for the presence of nominal longitudinal stress in the chord. When calculating Q_f for the joints, the moments used in the chord stress calculation will be from the computer node results and not the representative moments underneath the brace. If the moment varies significantly along the chord, it is more accurate to use the actual chord moment in the middle of the brace foot print. The tests reported in Reference I[1] were performed with a constant moment along the chord. Thus for a local joint check, the local chord moment (under the brace) should be used.

STAAD calculates Q_f based on the moment at the chord member. The chord member can be selected automatically by initial screening by the program (based on geometry and independent of loading) or specified in the External file.

In the automatic selection of the chord two collinear members (5 degree tolerance) are used to identify the chord. The chord is then selected from one of the two members based on the larger diameter then thickness or then by the minimum framing angle; for T joints the first member modeled will be selected as the chord.

You should confirm that the chord either be assigned by the program or the user is representative of the local chord moment for the brace in question.

D1.J.6.1 Reference

1 Ref I: Boone, TJ. Yura, JA. and Hoadley, PW. *Ultimate Strength if Tubular Joints – Chord Stress Effects*, OTC 4828, 1984

D1.J.7 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

Member	the member number for which the design is performed.
TABLE	AISC steel section name which has been checked against the steel code or has been selected.

RESULTS prints whether the member has PASSed or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.

CRITICAL COND the section of the AISC code which governs the design.

RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.

LOADING provides the load case number which governed the design.

FX, MY, and MZ provide the axial force, moment in local Y-axis, and the moment in local Z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.

LOCATION specifies the actual distance from the start of the member to the section where design forces govern.

Note: If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending stressed in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV).

D1.J.7.1 Example of Member Code Check output

For `TRACK 0.0` output:

```
                         STAAD.Pro CODE CHECKING - (API )
                         ***********************
 PROGRAM CODE REVISION V21_API_2000/1

ALL UNITS ARE - KN   METE (UNLESS OTHERWISE NOTED)

MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================

      6 ST PIP40610.0    (BRITISH SECTIONS)
                           PASS      API 3.3.1-2      0.024         2
                       2.76 T         0.00            5.12        3.00
      7 ST PIP40610.0    (BRITISH SECTIONS)
                           PASS      API 3.3.1-3      0.078         2
                      98.14 C         0.00            5.12        0.00
```

For `TRACK 1.0` or `TRACK 2.0` output:

```
     14 ST PIP1938.0     (BRITISH SECTIONS)
                           PASS      API 3.3.1-3      0.130         2
                      67.16 C         0.00            0.29        4.24
 |--------------------------------------------------------------------------|
 | MEMB=    14, UNIT NEW-MMS ,L= 4243. AX=  4670. SZ=  208157. SY=  208157.|
 | KL/R-Y=  64.6  CB=  1.00 YLD=  248.21 ALLOWABLE STRESSES:  FCZ=  186.2  |
 | FTZ=  186.2 FCY=  186.2 FTY=  186.2 FA=  117.6 FT=  148.9   FV=   99.3  |
 |--------------------------------------------------------------------------|
```

D1.J.7.2 Example of Joint Check output

For `TRACK 0.0` output:

```
          STAAD.Pro - API JOINT CHECKS TO 21st edition.
          ---------------------------------------------------

NODE NO:      7   CHORD NO:     7   BRACE NO:    10   RATIO:   0.049   PASS
NODE NO:      7   CHORD NO:     7   BRACE NO:    13   RATIO:   0.245   PASS
NODE NO:      7   CHORD NO:    11   BRACE NO:    14   RATIO:   0.222   PASS
```

For `TRACK 2.0` output:

```
          STAAD.Pro - API JOINT CHECKS TO 21st edition.
          ---------------------------------------------------

======================================================================
NODE NO :      7   CHORD NO:     7   BRACE NO:    13
======================================================================
DESIGN DATA : (Units    : N , mm  )
  Chord Memb :      D =  406.40  T = 10.01
  Brace Memb :      d =  193.70  t =  8.00
  Angle (THETA) =  45.0 deg  GAP   =   50.80   Fyc = 248.2
  BETA          =  0.48      GAMMA =   20.30   TAU =  0.80
JOINT CLASS :     X + Y     Contributions:   0.% K,  50.% X,  50.% Y
                Factors are not displayed for TRACK 1.0 output  FACTORS :
  ----------------------------------------------------------------------
  Joint   Load   Strength     Chord Load    C1        C2        C3
  Class   Cond   factor (Qu)  factor (Qf)
  ----------------------------------------------------------------------
    X     AX      10.962        0.988       0.200    0.000    0.500
    Y     AX      14.299        0.983       0.300    0.000    0.800
    X     IPB      7.896        0.989       0.200    0.000    0.400
    Y     IPB      7.896        0.989       0.200    0.000    0.400
                ----------------------------------------------------------------------
CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO   STATUS
   (Cl. 4.3)             (KN, m )         (KN, m )
----------------------------------------------------------------------
AXIAL        :            66.995     2   273.398      0.245   PASS
IP BENDING   :             0.080     2    33.228      0.002   PASS
INTERACTION  :                       2                0.245   PASS
----------------------------------------------------------------------
CRITICAL     :                       2                0.245   PASS
----------------------------------------------------------------------
```

D1.J.8 Joint Design

D1.J.8.1 Joint Checking

The design of joints is based on Section 4 of the API code.

The program only checks simple joints and overlapping joints formed between circular hollow section members. Any other type of joint within the structure or joint cans will not be considered for API joint checks. Other types of joints (such as grouted joints, joints with ring stiffeners, etc.) are not considered.

Material Strength

The API code states in Cl. 4.2.1 that the value of yield stress of the chord member to be used in the calculation of the joint capacity should be limited to 0.8 times the tensile strength of the chord for materials with a yield stress less than or equal to 500 MPa.

The yield stress to be used in the joint capacity checks value is specified in the joint data file (*filename* .PUN). For every joint, the value specified in the FYLD column will be used as the yield strength to be used for the joint capacity checks. When the file is created for the first time by the program, a default value of 36 ksi is used for all joints. The value used for each joint check will also be reported in the output file.

Note: All the fields in the joint data file (`*.PUN` file) are to be in imperial units.

Minimum Joint Capacity

Clause 4.2.3 of the code specifies a minimum capacity for any joint as follows:

The connections at the ends of a member should develop the strength required by the design loads, but should not be less than 50% of the effective strength of the member. The effective strength is defined as the buckling load for a compression member or the yield load for members in tension. You, however, must ensure that this condition is satisfied even if the joint strength indicates a PASS status.

The program checks to see if the capacity of a joint as calculated by the methods in the code satisfies this requirement. If not the program issues a warning to that effect and marks the joint as FAILED. The program calculates the axial and/or bending moment capacities of the joint and reports the load/capacity ratio for each condition. The program also reports a “critical ratio” along with the condition that induces this ratio. Note that the maximum among the various individual ratios will be reported as the ‘critical ratio’. The program also reports a PASS/FAIL status for the joint.

Refer to D1.J.2.6 Simple Joints: Capacity Checks (on page 1285) for details of capacity checks performed.

Joint Classification

Clause of 4.2.4 of the API code essentially classifies a joint into one of the three basic types: K, X, and Y. Joint classification is the process whereby the axial load in a given brace is subdivided into its K, X, and Y components corresponding to the three joint types. A joint—as considered in the code—is the connection between a "chord" and a "brace" that are in the same plane. The program considers any two members to be in the same plane if they lie in planes that are within ±15 degrees of each other. The classification of a joint can also be a mixture of any of the basic types mentioned above. Once the classification of a joint has been identified, the capacity of that joint is then evaluated per Section 4.3 of the code.

The program automatically identifies the joints in a structure and identifies the chord and the brace members. The program applies the ±15° rule to determine the members in a plane and then determines the joint as being the intersection point of these members. Since a joint is between a chord and a brace member, the program considers two members at a time and then proceeds to identify the chord and the brace member at that joint. The program assumes the member with the larger diameter among the two members as the chord member and the other is considered as the brace. If both members have the same diameter, the chord is assumed to be the member with the thicker wall. If both the diameter and thickness of the members are identical, the program will assume the most horizontal member to be the chord. To be automatically considered as a chord member, the member has to be continuous across the joint. The user can always edit the joint data file (*.PUN) to add or delete new BRACE-CHORD joints.

The chord and brace member numbers (from the STAAD input file) are saved under the `CHORD` and `BRACE` columns in the *filename* .PUN file.

When the joint data file (`.PUN`) is created by the program, a default joint Class Y is assumed for the initial joint checks. This is indicated by the `K`, `X`, and `Y` column values being set to 0, 0, and 1 respectively. Since the API code allows for a mixed joint classification, you must manually vary the contribution factors for K, X, and Y joint classes for a given joint. For example, if a joint is to be 25% K, 25% X, and 50% Y, then you must assign K column value of 0.25, X column value of 0.25, and a Y column value of 0.50 for that joint. The program will verify that the supplied contributions sum to 1.0.

If the joint has a gap (i.e., a K-GAP joint), the gap distance (in inches) must be supplied in the `GAP` column. The value to be provided will be the actual gap between the brace members at the joint. An overlap can be specified by setting the gap to a negative value. The overlapping brace in this case can then be indicated by specifying the member number at the `OBRACE` (Overlapping brace) column in the data file.

Overlapping Joints

Clause 4.4 of the API discusses overlapping joints. Checks for overlapping joints will be performed as described D1.J.2.6 Simple Joints: Capacity Checks (on page 1285). The difference will be in that the gap value, g, will be taken as negative in evaluating the various factors.

If the axial loads in the overlapping brace and the through brace have the same sign, the axial load in the through brace will be increased to allow for the loads in the overlapping brace. This will be achieved by allowing a portion of the overlapping brace load equal to the proportion of the overlapping brace area to be added to the axial load in the through brace.

Note: The program issues a warning for any joint overlap is less than $0.25 \cdot \beta \cdot D$.

Related Links

- *V.API K Joint* (on page 5978)
- *V.API Overlapping KJoint - Comp and Bend* (on page 5983)
- *V.API Overlapping K Joint - Tens and Bend* (on page 5989)
- *V.API X Joint* (on page 5996)
- *V.API Y Joint* (on page 6001)

D1.J.8.2 Joint File Format

The data contained in the *`filename`* `.PUN` file should meet the following format. The overall process of performing punching shear checks consists of two steps which are explained in D1.J.8.1 Joint Checking (on page 1293).

When the API design module is invoked, the program will initially check for the presence of a *`filename`* `.PUN` file (where *`filename`* is the name of the `.std` file) in the same folder as the input file. If the program does not find such a file, it assumes that the joint design is being run for the first time and will create this file. If the program does find this file, it will assume that the joint design has been run at least once and will attempt to read the input data from this file. Not that modifying and saving the main structure (i.e., any changes to the main model using GUI or text editor) will invalidate all design results and the program will automatically delete all design related files including the `*.PUN` file. Hence if the user wises to keep an existing version of the *.PUN file, he/she must make a separate copy of this file before making any changes to the model.

Note: Units used in this file must be kips and inches.

General Format

```
*BRACE CHORD K X Y D T d t GAP FYLD OBRACE TW SWAP
```

```
 b# c# K% X% Y% Dc Tc db tb gap fy ob tw swap
```

Where:

b# = the brace member number
c# = the chord member number

K%, X%, and Y% = The fractional contributions of K-type, X type and Y-type, respectively. Initially the joints will be classed as Y (i.e., K=0, X=0 and Y=1).

db, tb = Diameter and thickness of BRACE member

Dc, Tc = Diameter and thickness of CHORD member

gap = Distance required to calculate gap factor for K bracing. Initially, the value of GAP is assumed as 0. An overlap can be specified by setting the gap to a negative value.

fy = the yield stress to be used in the joint capacity checks

ob = member number of the overlapping brace in an overlap joint (i.e., a gap value less than zero)

tw = Used in overlap K-joint, taken as the lesser of the weld throat thickness or thickness t of the thinner brace in inches

swap = If parameter SWAP 0 is used then major moment Mz is taken for In Plane Bending (IPB). SWAP 1 uses the minor moment My as the IPB.

Example

```
*BRACE  CHORD   K      X      Y       D       T       d       t      GAP   FYLD OBRACE TW SWAP
    10      7  0.000  0.000  1.000  16.000  0.394  16.000  0.394   0.00  36.0  0.0  0.00 0
    13      7  0.000  0.000  1.000  16.000  0.394   7.626  0.315   0.00  36.0  0.0  0.00 0
    14     11  0.000  0.000  1.000  16.000  0.394   7.626  0.315   0.00  36.0  0.0  0.00 0
```

D1.K. American Codes - Steel Design per ANSI/AISC N690 Design Codes

D1.K.1. ANSI/AISC N690-1994 Code

STAAD.Pro is capable of performing steel design based on ANSI/AISC N690-1994 and as amended by *Supplement No. 2 to the Specification of the Design, Fabrication and Erection of Steel Safety-Related Structures for Nuclear Facilities* (ANSI/AISC N690 1994(R2004)s2).

Design of members per ANSI/AISC N690-1994 requires the *STAAD Nuclear Design Codes* SELECT Code Pack.

Related Links

- *V. AISC N690 1994 Angle* (on page 5820)
- *V. AISC N690 1994 Channel* (on page 5826)
- *V. AISC N690 1994 Pipe Section With SLC* (on page 5830)
- *V. AISC N690 1994 Pipe* (on page 5843)
- *V. AISC N690 1994 W Shaped* (on page 5846)

D1.K.1.1 General Comments

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by ANSI/AISC N690-1994 and as amended by ANSI/AISC N690 1994(R2004)s2.

All the design steps are done as described in D1.B.1.1 Allowables per AISC Code (on page 1130) except for allowable stress in compression for `AUSTENITIC STAINLESS STEEL`. Section Q1.5.9 is used to calculate allowable compressive stress for Austenitic Stainless Steel. Correction made in Supplementary s1 published in April 15, 2002 has been applied.

Note: By default, N690 code uses Stainless Steel material in the design. Care should be taken to assign the proper Stainless Steel material properties to the members for the analysis. There is a parameter – `STYPE` – to change material type to either Stainless Steel (STYPE=1) or Carbon Steel (STYPE=0).

D1.K.1.1.1 Design Process

Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

$$\frac{f_a}{\mathrm{SFC} \cdot F_a} + \frac{C_{my} f_{by}}{\mathrm{SMY} \cdot F_{by}\left(1 - \frac{f_a}{F'_{ey}}\right)} + \frac{C_{mz} f_{bz}}{\mathrm{SMZ} \cdot F_{bz}\left(1 - \frac{f_a}{F'_{ez}}\right)} \le 1.0$$

and Q1.6-1b:

$$\frac{f_a}{\mathrm{SFC} \cdot 0.6F_y} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

when, $f_a/F_a > 0.15$, as per section Q1.6.1 of the code.

Otherwise, equation Q1.6-2 must be satisfied:

$$\frac{f_a}{\mathrm{SFC} \cdot F_a} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $[0.6 - 0.4 \cdot (M1/M2)]$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q1.6-3:

$$\frac{f_a}{\mathrm{SFT} \cdot 0.6F_y} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

Where:

`SFC`, `SFT`, `SMZ`, and `SMY` are stress limit coefficient parameters used to control the components of the interaction equations. Refer to D1.K.1.2 Design Parameters (on page 1297) for details.

D1.K.1.2 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure

Table 102: Design Parameters for ANSI/AISC N690-1994

Parameter Name	Default Value	Description
CODE	-	Must be specified as `AISC N690` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	1	Beam parameter 0. Perform design at ends and those locations in the SECTION command. 1. Perform design at ends and at 1/12th section locations along the member length.
CAN	0	Used for Deflection Check only (i.e., when DFF is specified). 0. Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1. Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient, as specified in Chapter F of AISC ASD. 0.0 = CB is calculated itself Any other user-defined value is accepted.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
COMPOSITE	0	Composite action with connectors (CMP) 0. No composite action 1. Composite action 2. Ignore positive moments during design
CONDIA	0.625 in	Diameter of shear connectors (DIA), in current units.
CONHEIGHT	2.5 in	Height of shear connectors after welding (HGT), in current units.

Parameter Name	Default Value	Description
CYCLES	500,000	Cycles of maximum stress to which the shear connector is subject (CYC).
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length"
DLR2	0.4	Ratio of moment due to dead load applied after the concrete hardens to the total moment (DR2).
DLRATIO	0.4	Ratio of moment due to dead load applied before the concrete hardens to the total moment (DR1).
DMAX	45 inch	Maximum allowable depth
DMIN	0.0 inch	Minimum allowable depth
EFFWIDTH	1/4 Member Length	Effective width of concrete slab (WID).
FYLD	36 KSI	Yield strength of steel in current units.
FPC	3 KSI	Compressive strength of concrete at 28 days, in current units.
FSS	1	Full section shear for welding. 0. False 1. True
FU	60 KSI	Ultimate tensile strength of steel, in current units.
FYLD	46 KSI	Yield strength of steel, in current units.
KX	1.0	Effective length factor for flexural torsional buckling.

Parameter Name	Default Value	Description
KY	1.0	Effective Length Factor for Compression in local y-axis. Usually, this is minor axis.
KZ	1.0	Effective Length Factor for Compression in local z-axis. Usually, this is major axis.
LX	Member Length	Length for flexural torsional buckling.
LY	Member Length	Length to calculate slenderness ratio (KL/r) for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).
MAIN	0.0	Design for slenderness: 0. check for slenderness 1. suppress slenderness check
NSF	1.0	Net section Factor for tension members
OVR	1.0	Factor by which all allowable stresses/capacities should be multiplied. Default of 1.0 indicates that no overstressing is allowed.
PLTHICK	0	Thickness of the cover plate welded to the bottom flange of the composite beam (PLT), in current units.
PLTWIDTH	0	Width of the cover plate welded to the bottom flange of the composite beam (PLT), in current units.
PROFILE	None	Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.

Parameter Name	Default Value	Description
RIBHEIGHT	0	Height of ribs of form steel deck (RBH), in current units.
RIBWIDTH	0	Width of ribs of form steel deck (RBW), in current units.
SFC	1.0	Stress limit coefficient for compression (SLC) as found in Table Q 1.5.7.1.
SFT	1.0	Stress limit coefficient for tension (SLC) as found in Table Q 1.5.7.1.
SHE	0	Shear stress calculation option 0. Computes the actual shear stress using VQ/It 1. Computes the actual shear stress using V(Ay or Az)
SHORING	0	Temporary shoring during construction 0. Without shoring 1. With shoring
SLABTHICK	4 in	Thickness of concrete slab or thickness of concrete slab above the form steel deck (THK), in current units.
SMY	1.0	Stress limit coefficient for minor axis bending (SLC) as found in Table Q 1.5.7.1.
SMZ	1.0	Stress limit coefficient for major axis bending (SLC) as found in Table Q 1.5.7.1.
SSY	0	Design for sidesway in the local y axis. 0. Sidesway 1. No sidesway

Parameter Name	Default Value	Description
SSZ	0	Design for sidesway in the local z axis. 0. Sidesway 1. No sidesway
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design, in current units.
STYPE	0.0	Type of steel material 0. Normal Steel 1. Austenitic Stainless Steel
TAPER	1	Design for tapered member. 0. Design for tapered I-section based on rules in Chapter F and Appendix B. 1. Design for tapered section based on Appendix F.
TMAIN	240 for main member 300 for "Truss" member	Slenderness limit under tension
TORSION	0	Design for torsion. 0. Do not design for torsion. 1. Design for torsion.
TRACK	0.0	Controls the levels of detail to which results are reported. 0. Minimum detail 1. Intermediate detail level 2. Maximum detail
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.

Parameter Name	Default Value	Description
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.
WELD	1	Design for weld. 0. Closed sections. 1. Open sections.
WMAX	1 in	Maximum weld thickness, in current units.
WMIN	0.625 in	Minimum weld thickness, in current units.
WSTR	0.4·Fyld	Allowable welding stress, in current units.

D1.K.1.2.1 Notes

1. All values are entered in the current units
2. parameters `DMAX` and `DMIN` are only used with the `MEMBER SELECTION` command

D1.K.2. ANSI/AISC N690-1984 Code

For code checking of steel members, STAAD compares the actual stresses with the allowable stresses as defined by ANSI/AISC N690-1984: *Nuclear Facilities - Steel Safety-Related Structures for Design, Fabrication, and Erection.*

A brief description of some of the major allowable stresses is described herein.

Related Links

- *V. AISC N690 1984 Angle* (on page 5849)
- *V. AISC N690 1984 Channel* (on page 5855)
- *V. AISC N690 1984 Pipe Section With SLC* (on page 5858)
- *V. AISC N690 1984 Pipe* (on page 5865)
- *V. AISC N690 1984 Tee* (on page 5868)
- *V. AISC N690 1984 W Shaped* (on page 5872)

D1.K.2.1 Design Process

The following Checks are to be performed on a Steel Member as per this AISC N690 – 1984 Code. When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.K.2.1.1 Slenderness

The maximum allowable slenderness ratio in Compression (K·L/r_min), as per clause Q1.8.4 of the code shall not exceed 200. And the maximum allowable slenderness ratio in Tension (L/r_min) shall not exceed 240 for main members and 300 for bracing members and other secondary members.

This can be controlled by using the existing `MAIN` and `TMAIN` parameters respectively.

The default value of `MAIN` is `200` and for `TMAIN` is `240`.

D1.K.2.1.2 Check for Element Slenderness and Stress Reduction Factors

The permissible Width-to-Thickness Ratio of “Un-stiffened Elements under Compression” is determined as per section Q1.9.1 and that of “Stiffened Elements under Compression” is determined as per section Q1.9.2 of the code.

The permissible Width–Thickness Ratio of web is determined as per section Q1.10.2.

D1.K.2.1.3 Tension

Allowable tensile stress on the Net section is calculated as $0.60 \cdot F_y$, but not more than $0.5 \cdot F_u$ on the Effective Net area, as per section Q1.5.1.1.

The Net Area (An) shall be determined in accordance with Q1.14, and the `NSF` parameter can be utilized for that.

The Effective Net Area (Ae) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. Q1.14),

$$A_e = C_t \cdot A_n$$

Unless otherwise specified, the default value of the `CT` parameter is set as 0.75.

The value of `CT` parameter for other conditions is described at section Q1.14.

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.K.2.1.4 Compression

The allowable compressive stress for columns which meet the provisions of section Q1.9, except those fabricated from austenitic stainless steel shall be as required by Q1.5.1.3. The allowable compressive stress for columns fabricated from austenitic stainless steel shall be in accordance to section Q1.5.9.

A. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

  a. On gross section of axially loaded compression members, when $(Kl/r) \le C_c$,

$$F_a = [1 - (Kl/r)^2/(2 \cdot C_c{}^2)]F_y / \{5/3 + [3(Kl/r)/(8 \cdot C_c)] - [(Kl/r)^3/(8 \cdot C_c{}^3)]\}$$

  Where:

$$C_c = [(2 \cdot \pi^2 E)/F_y]^{1/2}$$

  b. When $(Kl/r) > C_c$,

$$F_a = 12 \cdot \pi^2 E/[23(kL/r)^2]$$

B. Gross sections of columns fabricated from Austenitic Stainless steel:

  a. When $(Kl/r) \le 120$,

$$F_a = F_y/2.15 - [(F_y/2.16 - 6)/120](kL/r)$$

  b. When $(Kl/r) > 120$,

$$F_a = 12 - (KL/r)/20$$

If the provisions of the section Q1.9 are not satisfied,

A. For un-stiffened compression element, a reduction factor Q_s is introduced. Detailed values of Qs for different shapes are given in Section QC2.

B. For stiffened compression element, a reduced effective width b_e is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = 253 \cdot t/\sqrt{F_y}\{1 - (50.3/[(b/t)\sqrt{F_y}]\} \leq b$$

b. For other uniformly compressed elements:

$$b_e = 253 \cdot t/\sqrt{F_y}\{1 - (44.3/[(b/t)\sqrt{F_y}]\} \leq b$$

Consequently, a reduction factor Q_a is introduced and is equal to the effective area divided by the actual area. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or unstiffened elements shall not exceed

$$F_a = Q_s Q_a[1 - (Kl/r)^2/(2 \cdot C_c^{\ 2})]F_y / \{5/3 + [3(Kl/r)/(8 \cdot C_c)] - [(Kl/r)^3/(8 \cdot C_c^{\ 3})]\}$$

Where:

$$C'_c = [(2 \cdot \pi^2 E)/(Q_s Q_a F_y)]^{1/2}$$

D1.K.2.1.5 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in section Q1.5.1.4 is:

A. Along Major Axis:

a. Tension and compression on extreme fibers of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection Q1.5.1.4.1.1 to 7, shall result in a maximum bending stress:

$$F_b = 0.66 \cdot F_y$$

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed $65/\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

$d/t = (640/\sqrt{F_y})[1 - 3.74(f_a/F_y)]$ when $f_a/F_y \leq 0.16$

$d/t = 257/\sqrt{F_y}$ when $f_a/F_y > 0.16$

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_f/\sqrt{F_y}$ nor $20000/(d/A_f)F_y$.

b. For noncompact and slender elements, section Q1.5.1.4.2 is followed.

c. For box-type flexural members, maximum bending stress is:

$$F_b = 0.60 \cdot F_y$$

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of section Q1.5.1.4.1, maximum tensile and compressive bending stress shall not exceed the following value as per section Q1.5.1.4.3:

$$F_b = 0.75 \cdot F_y$$

b. For doubly symmetrical members (I shaped) meeting the requirements of section Q1.5.1.4.1, except where $b_f/2t_f > 65/\sqrt{F_y}$ but is less than $95/\sqrt{F_y}$, maximum tensile and compressive bending stress shall not exceed:

$$F_b = F_y[0.79 - 0.002(b_f/2t_f)\sqrt{F_y}]$$

D1.K.2.1.6 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy equation Q1.6-1a:

$$\frac{f_a}{\mathrm{SFC} \cdot F_a} + \frac{C_{my} f_{by}}{\mathrm{SMY} \cdot F_{by}\left(1 - \frac{f_a}{F'_{ey}}\right)} + \frac{C_{mz} f_{bz}}{\mathrm{SMZ} \cdot F_{bz}\left(1 - \frac{f_a}{F'_{ez}}\right)} \le 1.0$$

and Q1.6-1b

$$\frac{f_a}{\mathrm{SFC} \cdot 0.6F_y} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

when, $f_a/F_a > 0.15$, as per section Q1.6.1 of the code.

Otherwise, equation Q1.6-2 must be satisfied:

$$\frac{f_a}{\mathrm{SFC} \cdot F_a} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $[0.6 - 0.4 \cdot (M1/M2)]$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy equation Q 1.6-1b:

$$\frac{f_a}{\mathrm{SFT} \cdot 0.6F_y} + \frac{f_{by}}{\mathrm{SMY} \cdot F_{by}} + \frac{f_{bz}}{\mathrm{SMZ} \cdot F_{bz}} \le 1.0$$

Where `SFC`, `SFT`, `SMZ`, and `SMY` are stress limit coefficient parameters used to control the components of the interaction equations. Refer to D1.K.2.3 Design Parameters (on page 1307) for details.

D1.K.2.1.7 Shear Stress

Allowable shear stress on the gross section [ref. section Q1.10.5.2] is calculated as

$$F_v = (F_y/2.89)C_v \le 0.4 \cdot F_y$$

Where:

$C_v = (45{,}000 \cdot k)/[F_y(h/t)^2]$, when $h/t \le 0.8$
$C_v = [190/(h/t)]\sqrt{(k/F_y)}$, when $h/t > 0.8$
$k = 4.00 + [5.34/(a/h)^2]$, when $a/h \le 1.0$
$k = 5.34 + [4.00/(a/h)^2]$, when $a/h > 1.0$

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.K.2.2 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used, namely: I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, and Double Channel sections.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D1.K.2.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure

Table 103: Design Parameters for ANSI/AISC N690-1984

Parameter Name	Default Value	Description
CODE	-	Must be specified as `AISC N690 1984` to use the ANSI/AISC N690-1984 code for checking purposes. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
CAN	0	Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
CT	0.75	Reduction Coefficient in computing net effective net area of an axially loaded tension member.
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection

Parameter Name	Default Value	Description
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length"
DMAX	45 inch	Maximum allowable depth
DMIN	0.0 inch	Minimum allowable depth
FU	60 KSI	Ultimate tensile strength of steel in current units.
FYLD	36 KSI	Yield strength of steel in current units.
KY	1.0	Effective Length Factor for Compression in local y-axis. Usually, this is minor axis.
KZ	1.0	Effective Length Factor for Compression in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).
MAIN	0.0	Design for slenderness. 0. Check for slenderness 1. Suppress slenderness check
NSF	1.0	Net section Factor for tension members
PROFILE	None	Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.

Parameter Name	Default Value	Description
SFC	1.0	Stress limit coefficient for compression (SLC) as found in Table Q 1.5.7.1.
SFT	1.0	Stress limit coefficient for tension (SLC) as found in Table Q 1.5.7.1.
SMY	1.0	Stress limit coefficient for minor axis bending (SLC) as found in Table Q 1.5.7.1.
SMZ	1.0	Stress limit coefficient for major axis bending (SLC) as found in Table Q 1.5.7.1.
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design
STYPE	0.0	Steel type: 0.0 = Normal Steel 1.0 = Austenitic Stainless Steel
TMAIN	240 for main member 300 for “Truss” member	Slenderness limit under tension
TRACK	0.0	Controls the levels of detail to which results are reported. 0 = Minimum detail 1 = Intermediate detail level 2 = Maximum detail
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

D1.K.2.3.1 Notes

1. All values are entered in the current units

2. parameters DMAX and DMIN are only used with the MEMBER SELECTION command

D1.K.2.4 Code Checking and Member Selection

Both code checking and member selection options are available with the AISC N690 1984 code.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.L. American Codes - Steel Design per ASME NF Codes

The following American Society of Mechanical Engineers – Nuclear Facility codes are available.

D1.L.1. ASME NF 3000 - 1974 & 1977 Codes

STAAD.Pro is capable of performing steel design based on the American Society of Mechanical Engineers *Nuclear Facility Code*, ASME NF 3000 - 1974 & 1977 .

Note: From design point of view, there are no major differences between NF-3000 1974 and NF-3000 1977 version of codes.

Design of members per ASME NF 3000 - 1974 & 1977 requires the *STAAD Nuclear Design Codes* SELECT Code Pack.

Related Links

- *V. ASME NF 3000 1974 Angle* (on page 6005)
- *V. ASME NF 3000 1974 Channel* (on page 6010)
- *V. ASME NF 3000 1974 Pipe* (on page 6014)
- *V. ASME NF 3000 1974 Tee* (on page 6018)
- *V. ASME NF 3000 1974 WShaped* (on page 6022)
- *V. ASME NF 3000 1977 Angle* (on page 6025)
- *V. ASME NF 3000 1977 Channel* (on page 6031)
- *V. ASME NF 3000 1977 Pipe* (on page 6034)
- *V. ASME NF 3000 1977 Tee* (on page 6038)
- *V. ASME NF 3000 1977 WShaped* (on page 6042)

D1.L.1.1 Design Process

The design process follows the following design checks

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.L.1.1.1 Slenderness
As per clause XVII-2223 of NF-3000 1974, the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

D1.L.1.1.2 Tension

Allowable tensile stress on the Net section is calculated as ($0.60 \times F_y$), but not more than ($0.5 \times F_u$) on the Net area.

The Net Area (A_n) shall be determined in accordance with the clause XVII-2283 of NF-3000 1974, and the `NSF` parameter can be utilized for that.

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.Pro.

D1.L.1.1.3 Compression

The allowable compressive stress for columns shall be as required by clause XVII-2213 of NF-3000 1974.

a. Gross Sections of Columns:

  a. On gross section of axially loaded compression members, when $(Kl/r) < C_c$,

$$F_a = F_y \frac{\left[1 - \frac{(KL/r)^2}{2C_c^2}\right]}{5\left/3 + \frac{3(KL/r)}{8C_c} - \frac{(KL/r)^3}{8C_c^3}\right.}$$

  Where:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

  b. When $(Kl/r) > C_c$,

$$F_a = \frac{12\pi^2 E}{23(KL/r)^2}$$

  c. When $(Kl/r) > 120$,

$$F_{as} = \frac{F_a\left[Eq.(a1) \text{ or } Eq.(a2)\right]}{1.6 - \left(\frac{l}{200r}\right)}$$

b. Member elements other than columns:

  a. For Plate Girder Stiffeners, $F_a = 0.60 \cdot F_y$

  b. For webs of rolled shapes, $F_a = 0.75 \cdot F_y$

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of clause XVII-2224 of NF-3000 1974.

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element, a reduction factor, Q_s, is introduced. Detailed values of Q_s for different shapes are given in the clause XVII-2225.2 of NF-3000 1974.

b. For stiffened compression element, a reduced effective width, b_e, is introduced.

  a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{50.3}{(b/t)\sqrt{f}}\right)\right] \le b$$

  b. For other uniformly compressed elements:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{44.3}{(b/t)\sqrt{f}}\right)\right] \le b$$

Consequently, a reduction factor, Q_a, equal to the effective area divided by the actual area is introduced.

Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

$$F_a = F_y \frac{Q_s Q_a \left[1 - \frac{(KL\ / r)^2}{2C'^2_c} \right]}{5 \Big/ 3 + \frac{3(KL\ / r)}{8C'_c} - \frac{(KL\ / r)^3}{8C'^3_c}}$$

Where:

$$C'_c = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}}$$

The section is also checked that the stress is less than 2/3× the critical buckling stress, which is taken as the Euler critical buckling load divided by the cross sectional area:

$$F = \frac{2}{3}\frac{P_{\text{cr}}}{A}$$

where

P_{cr} = the Euler critical buckling load of the column: $\frac{\pi^2 EI}{(KL\ )^2}$

D1.L.1.1.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in XVII-2214 of NF-3000 1974 is:

a. Along Major Axis:

b. For Compact Sections, tension and compression on extreme fibers of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

$$F_b = 0.66 \times F_y$$

If meeting the requirements of this member of:

a. Width-thickness ratio of un-stiffened projecting elements of the compression flange shall not exceed $52.2/\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

$$d/t = (412/\sqrt{F_y})[1 - 2.33(F_a/F_y)]$$

except that it need not be less than $257/\sqrt{F_y}$.

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_f/\sqrt{F_y}$ nor $20000/(d/A_f)F_y$.

e. For noncompact and slender elements, clause XVII-2214.2 and XVII-2214.5 of NF-3000 1974 are followed respectively.

f. For box-type flexural members, maximum bending stress is:

$$F_b = 0.60 \times F_y$$

g. Along Minor Axis:

For doubly symmetrical members (I shaped) meeting the requirements of XVII-2214.1(a) and (b) of NF-3000 1974, maximum tensile and compressive bending stress shall not exceed:

$$F_b = 0.75 \times F_y$$

D1.L.1.1.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - f_a / F'_{ex}\right) F_{bx}} + \frac{C_{my} f_{by}}{\left(1 - f_a / F'_{ey}\right) F_{by}} \le 1.0$$

and

$$\frac{f_a}{0.60 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

when $f_a / F_a > 0.15$, otherwise

$$\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a / F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $0.6 - 0.4 \cdot (M1/M2)$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

$$\frac{f_a}{0.60 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

D1.L.1.1.6 Shear Stress
Allowable shear stress on the gross section [ref. XVII-2263.2 of NF-3000 1974] is calculated as

$$F_v = \left(F_y / 2.89\right) C_v \le 0.4 F_y$$

where

C_v = $\frac{45,\ 000k}{F_y (h\ /\ t)^2}$ when $h/b \le 0.8$

= $\frac{190}{h\ /\ t} \sqrt{\frac{k}{F_y}}$ when $h/b > 0.8$

k = $4.00 + 5.34/(a/h)^2$ when $a/h \le 1.0$

= $5.34 + 4.00/(a/h)^2$ when $a/h > 1.0$

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.1.1 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D1.L.1.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

Note: Unlike many other design codes available in STAAD.Pro (which use the `BEAM` parameter), design per ASME NF 3000 codes in STAAD.Pro is *always* performed based on forces calculated at 13 sections, including ends.

Table 104: ASME NF 3000 Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `CODE NF3000 1974` or `CODE NF3000 1977` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
CAN	0	Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes

Parameter Name	Default Value	Description
CT	0.75	Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length"
DMAX	45 inch	Maximum allowable depth. Used only with the MEMBER SELECTION command.
DMIN	0.0 inch	Minimum allowable depth. Used only with the MEMBER SELECTION command.
FYLD	36 KSI	Yield strength of steel at temperature in current units.
FU	60 KSI	Ultimate tensile strength of steel in current units.
KBK	1.0	Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. **Note:** Ignored unless SRL is set to D.
KS	1.0	Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. **Note:** Ignored unless SRL is set to D.
KV	1.0	Stress Limit Factor applicable to the Design Allowable Shear Stresses. **Note:** Ignored unless SRL is set to D.

Parameter Name	Default Value	Description
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check
NSF	1.0	Net Section Factor for tension member.
PROFILE	None	Used in member selection. See TR. 48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SRL	A	Service level, which defines the service level factors to use for modifying stress values for the service level conditions. **A.** Normal Conditions **B.** Upset **C.** Emergency **D.** Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined. Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1348) for additional information.
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design
TMAIN	240 for main member 300 for "Truss" member	Slenderness limit under tension

Parameter Name	Default Value	Description
TRACK	0.0	Controls the levels of detail to which results are reported. 0. Minimum detail 1. Intermediate detail level 2. Maximum detail
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

Related Links

- *D1.L.5. ASME NF 3000 Service Level Conditions* (on page 1348)

D1.L.1.4 Code Checking and Member Selection

Both code checking and member selection options are available with the ASME NF-3000 1974 and ASME NF-3000 1977 codes.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.L.1.5 Example

A cantilever beam of length 30 inch is loaded at its free end with 5 kip compressive load and 5 kip lateral load. The beam is assigned with W24X104 steel member and is designed in accordance with ASME NF3000 1974.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
```

```
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X104
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5 FY -5
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT JOINT DISPLACEMENTS
PRINT MEMBER FORCES
PARAMETER 1
CODE NF3000 1974
FYLD 36 ALL
FU 58 ALL
KY 0.9 ALL
KZ 0.9 ALL
NSF 0.85 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

The corresponding TRACK 2 output is as follows:

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-74)   v2.0
            ********************************************




            ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

            MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
            FX             MY             MZ       LOCATION
            =======================================================================


            1  ST   W24X104                  (AISC SECTIONS)
            PASS      NF-74-EQN-21      0.032         1
            5.00 C          0.00         150.00        0.00

|-----------------------------------------------------------------------------|
            | SLENDERNESS CHECK:    ACTUAL RATIO:   9.30 ALLOWABLE RATIO:
200.00          |
            | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                     |
            | AXIAL: 2.07E+01  FCZ: 2.38E+01  FCY: 2.70E+01  FTZ: 2.38E+01  FTY: 2.70E
+01 |
            | SHEAR: 1.44E
+01                                                               |
```

```
              | ACTUAL STRESSES: (UNIT - KIP
INCH)                                            |
              | AXIAL: 1.63E-01  FBZ: 5.83E-01  FBY: 0.00E+00 SHEAR:
4.15E-01                  |

|-----------------------------------------------------------------------------|
              | SECTION PROPERTIES: (UNIT -
INCH)                                              |
              | AXX:  30.70  AYY:  12.05  AZZ:  12.80  RZZ: 10.05  RYY:
2.90                 |
              | SZZ:     257.26  SYY:
40.47                                                |

|-----------------------------------------------------------------------------|
              | PARAMETER: (UNIT - KIP
INCH)                                                  |
              | KL/R-Z:  2.69   KL/R-Y:  9.30   UNL:   30.0   CMZ:  1.00   CMY:
1.00        |
              | CB: 1.75   FYLD:    36.00   FU:    58.00   NET SECTION FACTOR:
0.85          |
              | KS:1.000  KV:1.000  KBK:
1.000                                                 |

|-----------------------------------------------------------------------------|
              | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                      |
              | CLAUSE        RATIO  LOAD       FX         VY         VZ         MZ
MY   |
              | TENSION       0.000    0    0.00E+00       -          -          -
-   |
              | COMPRESSION  0.008    1    5.00E+00       -          -          -
-   |
              | COMP&BEND     0.032    1    5.00E+00       -          -    1.50E+02 0.00E
+00  |
              | TEN&BEND      0.000    1    5.00E+00       -          -    1.50E+02 0.00E
+00  |
              | SHEAR-Y       0.029    1        -     5.00E+00        -          -
-   |
              | SHEAR-Z       0.000    0        -          -    0.00E+00         -
-   |

|-----------------------------------------------------------------------------|
```

D1.L.2. ASME NF 3000 - 1989 Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Society of Mechanical Engineers — Nuclear Facility (ASME NF) Code. The ASME NF-3000 1989 Code is used as the basis of this design.

A brief description of some of the major allowable stresses is described herein.

Related Links

- *V. ASME NF 3000 1989 Angle* (on page 6046)
- *V. ASME NF 3000 1989 Channel* (on page 6051)
- *V. ASME NF 3000 1989 Pipe* (on page 6055)
- *V. ASME NF 3000 1989 Tee* (on page 6058)
- *V. ASME NF 3000 1989 WShaped* (on page 6062)

D1.L.2.1 Design Process

The design process follows the following design checks.

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.L.2.1.1 Slenderness

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

D1.L.2.1.2 Tension

Allowable tensile stress on the Net section is calculated as ($0.60 \times F_y$), but not more than ($0.5 \times F_u$) on the Effective Net area.

The Net Area (A_n) shall be determined in accordance with NF-3322.8(c)(1) - (a), (b) and (c), and the NSF parameter can be utilized for that.

The Effective Net Area (A_e) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)),

$$A_e = C_t \times A_n$$

Unless otherwise specified, the default value of the CT parameter is set as 0.75.

The value of CT parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.L.2.1.3 Compression

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c)(3).

a. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

   a. On gross section of axially loaded compression members, when $(Kl/r) < C_c$,

$$F_a = F_y \frac{\left[1 - \frac{(KL / r)^2}{2C_c^2}\right]}{5/3 + \frac{3(KL / r)}{8C_c} - \frac{(KL / r)^3}{8C_c^3}}$$

   Where:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

   b. When $(Kl/r) > C_c$,

$$F_a = \frac{12\pi^2 E}{23(KL / r)^2}$$

   c. When $(Kl/r) > 120$,

$$F_{as} = \frac{F_a\left[Eq.(a1) \text{ or } Eq.(a2)\right]}{1.6 - \left(\frac{l}{200r}\right)}$$

b. Gross sections of columns fabricated from Austenitic Stainless steel:

  a. When $(Kl/r) \leq 120$,

$$F_a = F_y\left(0.47 - \frac{KL\ /r}{444}\right)$$

  b. When $(Kl/r) > 120$,

$$F_a = F_y\left(0.40 - \frac{KL\ /r}{600}\right)$$

c. Member elements other than columns:

  a. For Plate Girder Stiffeners, $F_a = 0.60 \cdot F_y$

  b. For webs of rolled shapes, $F_a = 0.75 \cdot F_y$

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of NF-3322.2(d).

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element, a reduction factor, Q_s, is introduced. Detailed values of Q_s for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).

b. For stiffened compression element, a reduced effective width, b_e, is introduced.

  a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{50.3}{(b\ /\ t)\sqrt{f}}\right)\right] \leq b$$

  b. For other uniformly compressed elements:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{44.3}{(b\ /\ t)\sqrt{f}}\right)\right] \leq b$$

Consequently, a reduction factor, Q_a, equal to the effective area divided by the actual area is introduced.

Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

$$F_a = F_y \frac{Q_s Q_a\left[1 - \frac{(KL\ /r)^2}{2C'^2_c}\right]}{5\Big/3 + \frac{3(KL\ /r)}{8C'_c} - \frac{(KL\ /r)^3}{8C'^3_c}}$$

Where:

$$C'_c = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}}$$

The section is also checked that the stress is less than 2/3× the critical buckling stress, which is taken as the Euler critical buckling load divided by the cross sectional area:

$$F = \frac{2}{3}\frac{P_{\text{cr}}}{A}$$

where

P_{cr} = the Euler critical buckling load of the column: $\frac{\pi^2 EI}{(KL)^2}$

D1.L.2.1.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:

a. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

$$F_b = 0.66 \times F_y$$

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed $65/\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

$$d \Big/ t = \left(\frac{640}{\sqrt{F_y}}\right)\left[1 - 3.74\left(\frac{f_a}{F_y}\right)\right] \text{ when } f_a/F_y \leq 0.16$$

$$d \Big/ t = \left(\frac{257}{\sqrt{F_y}}\right) \text{ when } f_a/F_y > 0.16$$

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_f/\sqrt{F_y}$ nor $20000/(d/A_f)F_y$.

b. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.

c. For box-type flexural members, maximum bending stress is:

$$F_b = 0.60 \times F_y$$

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

$$F_b = 0.75 \times F_y$$

b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), *except where* $b_f/2t_f$ exceeds $65/\sqrt{F_y}$ but is less than $95/\sqrt{F_y}$, maximum tensile and compressive bending stress shall not exceed:

$$F_b = F_y[1.075 - 0.005(b_f/2t_f)\sqrt{F_y}]$$

D1.L.2.1.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a / F'_{ex})F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a / F'_{ey})F_{by}} \leq 1.0$$

and

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \leq 1.0$$

when $f_a/F_a > 0.15$, otherwise

$$\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $0.6 - 0.4 \cdot (M1/M2)$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

D1.L.2.1.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

$$F_v = (F_y/2.89)C_v \le 0.4F_y$$

where

C_v = $\frac{45,\ 000k}{F_y(h\ /\ t)^2}$ when $h/b \le 0.8$

= $\frac{190}{h\ /\ t}\sqrt{\frac{k}{F_y}}$ when $h/b > 0.8$

k = $4.00 + 5.34/(a/h)^2$ when $a/h \le 1.0$

= $5.34 + 4.00/(a/h)^2$ when $a/h > 1.0$

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.2.2 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D1.L.2.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

Note: Unlike many other design codes available in STAAD.Pro (which use the BEAM parameter), design per ASME NF 3000 codes in STAAD.Pro is *always* performed based on forces calculated at 13 sections, including ends.

Table 105: ASME NF 3000 Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as NF3000 1989. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
CAN	0	Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.
CT	0.75	Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length"

Parameter Name	Default Value	Description
DMAX	45 inch	Maximum allowable depth, in current units. Used only with the MEMBER SELECTION command.
DMIN	0.0 inch	Minimum allowable depth, in current units. Used only with the MEMBER SELECTION command.
FYLD	36 KSI	Yield strength of steel at temperature in current units.
FU	60 KSI	Ultimate tensile strength of steel in current units.
KBK	1.0	Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. **Note:** Ignored unless SRL is set to D.
KS	1.0	Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. **Note:** Ignored unless SRL is set to D.
KV	1.0	Stress Limit Factor applicable to the Design Allowable Shear Stresses. **Note:** Ignored unless SRL is set to D.
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).

Parameter Name	Default Value	Description
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check
NSF	1.0	Net Section Factor for tension member.
PROFILE	None	Used in member selection. See TR. 48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SRL	A	Service level, which defines the service level factors to use for modifying stress values for the service level conditions. **A.** Normal Conditions **B.** Upset **C.** Emergency **D.** Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined. Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1348) for additional information.
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design
STYPE	0.0	0.0 = Normal Steel 1.0 = Austenitic Stainless Steel
TMAIN	240 for main member 300 for "Truss" member	Slenderness limit under tension
TRACK	0.0	Controls the levels of detail to which results are reported. 0 = Minimum detail 1 = Intermediate detail level 2 = Maximum detail

Parameter Name	Default Value	Description
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

Related Links

- *D1.L.5. ASME NF 3000 Service Level Conditions* (on page 1348)

D1.L.2.4 Code Checking and Member Selection

Both code checking and member selection options are available with the ASME NF-3000 1989 code.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.L.2.5 Example

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 1989.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
JOINT COORDINATES
1 0 0 0; 2 360 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST B571806
CONSTANTS
```

```
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5
MEMBER LOAD
1 UNI GY -1.0 0 100
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PARAMETER 1
CODE NF3000 1989
STYPE 1 ALL
FYLD 36 ALL
KY 0.75 ALL
KZ 0.75 ALL
FU 58 ALL
NSF 0.9 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

The corresponding TRACK 2 output is as follows:

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-89)   v2.0
                       ********************************************




 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================


      1  ST   B571806                  (AISC SECTIONS)
                            PASS      SHEAR Y           0.770         1
                        5.00 C          0.00       5000.00        0.00
  |-----------------------------------------------------------------------------|
  | SLENDERNESS CHECK:    ACTUAL RATIO:  75.08 ALLOWABLE RATIO: 200.00          |
  | ALLOWABLE STRESSES:   (UNIT - KIP  INCH)                                    |
  | AXIAL: 1.13E+01  FCZ: 2.08E+01  FCY: 2.31E+01  FTZ: 2.16E+01  FTY: 2.31E+01 |
  | SHEAR: 5.18E+00                                                             |
  | ACTUAL STRESSES: (UNIT - KIP  INCH)                                         |
  | AXIAL: 1.06E-01  FBZ: 5.86E+00  FBY: 0.00E+00 SHEAR: 3.99E+00              |
  |-----------------------------------------------------------------------------|
  | SECTION PROPERTIES: (UNIT - INCH)                                           |
  | AXX:  47.00  AYY:  25.08  AZZ:  15.00  RZZ: 22.80  RYY:  3.60              |
  | SZZ:     853.77  SYY:      67.54                                            |
  |-----------------------------------------------------------------------------|
  | PARAMETER: (UNIT - KIP  INCH)                                               |
  | KL/R-Z: 11.84   KL/R-Y: 75.08   UNL:  360.0   CMZ:  1.00   CMY:  1.00       |
  | CB: 1.75   FYLD:    36.00   FU:    58.00   NET SECTION FACTOR: 0.90         |
  | CT:  0.75   STEEL TYPE:  1.0  KS:1.000  KV:1.000  KBK:1.000                |
  |-----------------------------------------------------------------------------|
  | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -INCH)                      |
```

```
| CLAUSE        RATIO   LOAD        FX          VY          VZ          MZ          MY   |
| TENSION       0.000    0     0.00E+00        -           -           -           -    |
| COMPRESSION   0.009    1     5.00E+00        -           -           -           -    |
| COMP&BEND     0.290    1     5.00E+00        -           -     5.00E+03 0.00E+00   |
| TEN&BEND      0.000    1     5.00E+00        -           -     5.00E+03 0.00E+00   |
| SHEAR-Y       0.770    1         -      1.00E+02        -           -           -    |
| SHEAR-Z       0.000    0         -           -     0.00E+00        -           -    |
|-----------------------------------------------------------------------------|
```

D1.L.3. ASME NF 3000 - 1998 Code

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as defined by the American Society of Mechanical Engineers — Nuclear Facility (ASME NF) Code. The ASME NF-3000 1998 Code is used as the basis of this design.

A brief description of some of the major allowable stresses is described herein.

Related Links

- *V. ASME NF 3000 1998 Angle* (on page 6066)
- *V. ASME NF 3000 1998 Channel* (on page 6071)
- *V. ASME NF 3000 1998 Pipe* (on page 6075)
- *V. ASME NF 3000 1998 Tee* (on page 6079)
- *V. ASME NF 3000 1998 WShaped* (on page 6083)

D1.L.3.1 Design Process

The design process follows the following design checks.

Each one of the checks are described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.L.3.2.1 Slenderness

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

D1.L.3.2.2 Tension

Allowable tensile stress on the Net section is calculated as ($0.60 \times F_y$), but not more than ($0.5 \times F_u$) on the Effective Net area.

The Net Area (A_n) shall be determined in accordance with NF-3322.8(c)(1) - (a), (b) and (c), and the `NSF` parameter can be utilized for that.

The Effective Net Area (A_e) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)),

$$A_e = C_t \times A_n$$

Unless otherwise specified, the default value of the `CT` parameter is set as 0.75.

The value of `CT` parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.L.3.2.3 Compression

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c)(3).

a. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

  a. On gross section of axially loaded compression members, when $(Kl/r) < C_c$,

$$F_a = F_y \frac{\left[1 - \frac{(KL/r)^2}{2C_c^2}\right]}{5 / 3 + \frac{3(KL/r)}{8C_c} - \frac{(KL/r)^3}{8C_c^3}}$$

  Where:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

  b. When $(Kl/r) > C_c$,

$$F_a = \frac{12\pi^2 E}{23(KL/r)^2}$$

  c. When $(Kl/r) > 120$,

$$F_{as} = \frac{F_a\left[Eq.(a1) \text{ or } Eq.(a2)\right]}{1.6 - \left(\frac{l}{200r}\right)}$$

b. Gross sections of columns fabricated from Austenitic Stainless steel:

  a. When $(Kl/r) \le 120$,

$$F_a = F_y\left(0.47 - \frac{KL/r}{444}\right)$$

  b. When $(Kl/r) > 120$,

$$F_a = F_y\left(0.40 - \frac{KL/r}{600}\right)$$

c. Member elements other than columns:

  a. For Plate Girder Stiffeners, $F_a = 0.60 \cdot F_y$

  b. For webs of rolled shapes, $F_a = 0.75 \cdot F_y$

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of NF-3322.2(d).

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element, a reduction factor, Q_s, is introduced. Detailed values of Q_s for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).

In the case for angles or plates projecting from compression members and for projecting elements of compression flanges of girder,

When $95 / \sqrt{F_y / kc} < b / t < 195 / \sqrt{F_y / kc}$, $Q_s = 1.293 - 0.00309(b / t)\sqrt{F_y / kc}$

When $b / t > 195 / \sqrt{F_y / kc}$,

$$Q_s = \frac{26,\ 200kc}{F_y(b/t)^2}$$

Where:

$$kc = \frac{4.05}{(h\ /\ t)^{0.46}}$$ when $h/t > 70$, otherwise, kc = 1.0.

b. For stiffened compression element, a reduced effective width, b_e, is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{50.3}{(b/t)\sqrt{f}}\right)\right] \leq b$$

b. For other uniformly compressed elements:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{44.3}{(b/t)\sqrt{f}}\right)\right] \leq b$$

Consequently, a reduction factor, Q_a, equal to the effective area divided by the actual area is introduced.

Combining both these factors, allowable stress for axially loaded compression members containing stiffened or un-stiffened elements shall not exceed

$$F_a = F_y \frac{Q_s Q_a\left[1 - \frac{(KL\ /r)^2}{2C'^2_c}\right]}{5\Big/3 + \frac{3(KL\ /r)}{8C'_c} - \frac{(KL\ /r)^3}{8C'^3_c}}$$

Where:

$$C'_c = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}}$$

The section is also checked that the stress is less than 2/3× the critical buckling stress, which is taken as the Euler critical buckling load divided by the cross sectional area:

$$F = \frac{2}{3}\frac{P_{\mathrm{cr}}}{A}$$

where

P_{cr} = the Euler critical buckling load of the column: $\frac{\pi^2 EI}{(KL)^2}$

D1.L.3.2.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:

1. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

$$F_b = 0.66 \times F_y$$

If meeting the requirements of this member of:

   a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed $65/\sqrt{F_y}$.
   b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.
   c. The depth-thickness ratio of the web shall not exceed

      $d/t = (640/\sqrt{F_y})[1 - 3.74(f_a/F_y)]$ when $f_a/F_y \le 0.16$

      $d/t = 257/\sqrt{F_y}$ when $f_a/F_y > 0.16$

   d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_f/\sqrt{F_y}$ nor $20,000/(d/A_f)F_y$.

2. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.
3. For box-type flexural members, maximum bending stress is:

   $F_b = 0.60 \times F_y$

B. Along Minor Axis:

   a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

      $F_b = 0.75 \times F_y$

   b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), except where $b_f/2t_f$ exceeds $65/\sqrt{F_y}$ but is less than $95/\sqrt{F_y}$, maximum tensile and compressive bending stress shall not exceed:

      $F_b = F_y[1.075 - 0.005(b_f/2t_f)\sqrt{F_y}]$

D1.L.3.2.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a/F'_{ex})F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a/F'_{ey})F_{by}} \le 1.0$$

and

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

when $f_a/F_a > 0.15$, otherwise

$$\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $0.6 - 0.4 \cdot (M1/M2)$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

D1.L.3.2.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

$$F_v = (F_y/2.89)C_v \le 0.4F_y$$

where

C_v = $\frac{45,000k}{F_y(h/t)^2}$ when $h/b \leq 0.8$

$= \frac{190}{h/t}\sqrt{\frac{k}{F_y}}$ when $h/b > 0.8$

k = $4.00 + 5.34/(a/h)^2$ when $a/h \leq 1.0$

$= 5.34 + 4.00/(a/h)^2$ when $a/h > 1.0$

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.3.3 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D1.L.3.4 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

Note: Unlike many other design codes available in STAAD.Pro (which use the `BEAM` parameter), design per ASME NF 3000 codes in STAAD.Pro is *always* performed based on forces calculated at 13 sections, including ends.

Table 106: ASME NF 3000 1998 Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `NF3000 1998`. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
CAN	0	Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
CT	0.75	Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length"
DMAX	45 inch	Maximum allowable depth, in current units. Used only with the MEMBER SELECTION command.
DMIN	0.0 inch	Minimum allowable depth, in current units. Used only with the MEMBER SELECTION command.
FYLD	36 KSI	Yield strength of steel at temperature in current units.
FU	60 KSI	Ultimate tensile strength of steel in current units.

Parameter Name	Default Value	Description
KBK	1.0	Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. **Note:** Ignored unless SRL is set to D.
KS	1.0	Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. **Note:** Ignored unless SRL is set to D.
KV	1.0	Stress Limit Factor applicable to the Design Allowable Shear Stresses. **Note:** Ignored unless SRL is set to D.
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check
NSF	1.0	Net Section Factor for tension member.
PROFILE	None	Used in member selection. See TR. 48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.

Parameter Name	Default Value	Description
SRL	A	Service level, which defines the service level factors to use for modifying stress values for the service level conditions. **A.** Normal Conditions **B.** Upset **C.** Emergency **D.** Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined. Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1348) for additional information.
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design
STYPE	0.0	0.0 = Normal Steel 1.0 = Austenitic Stainless Steel
TMAIN	240 for main member 300 for "Truss" member	Slenderness limit under tension
TRACK	0.0	Controls the levels of detail to which results are reported. 0 = Minimum detail 1 = Intermediate detail level 2 = Maximum detail
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

Notes

1. All values are entered in the current units.

2. The parameters DMAX and DMIN are only used with the MEMBER SELECTION command.

Related Links

- *D1.L.5. ASME NF 3000 Service Level Conditions* (on page 1348)

D1.L.3.5 Code Checking and Member Selection

Both code checking and member selection options are available with the ASME NF-3000 1998 code.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.L.3.6 Example

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 1998.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 100 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST B571806
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5
MEMBER LOAD
1 UNI GY -1.0 0 100
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
STYPE 1 ALL
FYLD 36 ALL
KY 0.75 ALL
```

```
KZ 0.75 ALL
FU 58 ALL
NSF 0.9 ALL
CT 0.85 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

The corresponding TRACK 2 output is as follows:

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-98)   v2.0
                       ********************************************



 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================


      1  ST   B571806                 (AISC SECTIONS)
                            PASS      SHEAR Y           0.635         1
                        5.00 C          0.00        5000.00        0.00
 |-----------------------------------------------------------------------------|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  20.85 ALLOWABLE RATIO: 200.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP  INCH)                                    |
 | AXIAL: 1.20E+01  FCZ: 2.22E+01  FCY: 2.31E+01  FTZ: 2.22E+01  FTY: 2.31E+01 |
 | SHEAR: 6.28E+00                                                             |
 | ACTUAL STRESSES: (UNIT - KIP  INCH)                                         |
 | AXIAL: 1.06E-01  FBZ: 5.86E+00  FBY: 0.00E+00 SHEAR: 3.99E+00               |
 |-----------------------------------------------------------------------------|
 | SECTION PROPERTIES: (UNIT - INCH)                                           |
 | AXX:  47.00  AYY:  25.08  AZZ:  15.00  RZZ: 22.80  RYY:  3.60               |
 | SZZ:     853.77  SYY:      67.54                                            |
 |-----------------------------------------------------------------------------|
 | PARAMETER: (UNIT - KIP  INCH)                                               |
 | KL/R-Z:  3.29   KL/R-Y: 20.85   UNL:  100.0   CMZ:  1.00   CMY:  1.00       |
 | CB: 1.75   FYLD:    36.00   FU:     58.00   NET SECTION FACTOR: 0.90        |
 | CT:  0.85   STEEL TYPE:  1.0  KS:1.000  KV:1.000  KBK:1.000                 |
 |-----------------------------------------------------------------------------|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -INCH)                      |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ        MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -         -   |
 | COMPRESSION  0.009    1    5.00E+00       -         -         -         -   |
 | COMP&BEND    0.272    1    5.00E+00       -         -    5.00E+03 0.00E+00  |
 | TEN&BEND     0.000    1    5.00E+00       -         -    5.00E+03 0.00E+00  |
 | SHEAR-Y      0.635    1        -     1.00E+02       -         -         -   |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -         -    |
 |-----------------------------------------------------------------------------|
```

D1.L.4. ASME NF 3000 - 2001 & 2004 Codes

STAAD.Pro is capable of performing steel design based on the American Society of Mechanical Engineers *Nuclear Facility Code*, ASME NF 3000 - 2004.

Note: Use of 2004 edition requires STAAD.Pro V8i (SELECTseries 2) NRC (build 20.07.07.30) or higher. Use of 2001 edition requires STAAD.Pro V8i (SELECTseries 3) NRC (build 20.07.08.22) or higher.

Design of members per ASME NF 3000 - 2001 & 2004 requires the *STAAD Nuclear Design Codes* SELECT Code Pack.

Related Links

D1.L.4.1 Design Process

The design process follows the following design checks.

Each one of the checks is described in the following sections.

When a design is performed, the output file the reports the maximum utilization from all of the checks.

D1.L.4.1.1 Slenderness

As per NF-3322.2(c), the slenderness ratio KL/r of compression members shall not exceed 200, and the slenderness ratio L/r of tension members, preferably should not exceed 240 for main members and 300 for lateral bracing members and other secondary members. The default limit for TRUSS members in Tension is set at 300.

D1.L.4.1.2 Tension

Allowable tensile stress on the Net section is calculated as ($0.60 \times F_y$), but not more than ($0.5 \times F_u$) on the Effective Net area.

The Net Area (A_n) shall be determined in accordance with NF-3322.8(c)(1) - (a), (b) and (c), and the NSF parameter can be utilized for that.

The Effective Net Area (A_e) of axially loaded tension members, where the load is transmitted by bolts through some but not all of the cross-sectional elements of the member, shall be computed from the formula (ref. NF-3322.8(c)(1)(d)),

$$A_e = C_t \times A_n$$

Unless otherwise specified, the default value of the CT parameter is set as 0.75.

The value of CT parameter for other conditions is described at section NF-3322.8(c)(1)(d)(1), (2) and (3).

The provisions for Pin-connected and Threaded tensile member are not implemented in STAAD.

D1.L.4.1.3 Compression

The allowable compressive stress for columns, except those fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(1). The allowable compressive stress for columns fabricated from austenitic stainless steel shall be as required by NF-3322.1(c)(2). The allowable compressive stress for member elements other than columns constructed by any material, including austenitic stainless steel, shall be as required by NF-3322.1(c)(3).

A. Gross Sections of Columns, except those fabricated of austenitic stainless steel:

a. On gross section of axially loaded compression members, when $(Kl/r) < C_c$,

$$F_a = F_y \frac{\left[1 - \frac{(KL/r)^2}{2C_c^2}\right]}{5/3 + \frac{3(KL/r)}{8C_c} - \frac{(KL/r)^3}{8C_c^3}}$$

Where:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}}$$

b. When $(Kl/r) > C_c$,

$$F_a = \frac{12\pi^2 E}{23(KL/r)^2}$$

c. When $(Kl/r) > 120$,

$$F_{as} = \frac{F_a\left[Eq.(a1) \text{ or } Eq.(a2)\right]}{1.6 - \left(\frac{l}{200r}\right)}$$

B. Gross sections of columns fabricated from Austenitic Stainless steel:

a. When $(Kl/r) \leq 120$,

$$F_a = F_y[0.47 - (Kl/r)/444]$$

b. When $(Kl/r) > 120$,

$$F_a = F_y[0.40 - (Kl/r)/600]$$

C. Member elements other than columns:

a. For Plate Girder Stiffeners,

$$F_a = 0.60 \cdot F_y$$

b. For webs of rolled shapes,

$$F_a = 0.75 \cdot F_y$$

The above clauses are applicable only when the width-thickness ratio of the element satisfies all the sub-sections of NF-3322.2(d)..

If the above-mentioned clauses are not satisfied,

a. For un-stiffened compression element,

A reduction factor Q_s is introduced. Detailed values of Q_s for different shapes are given in NF-3322.2(e)(2)(a) to NF-3322.2(e)(2)(d).

In the case for angles or plates projecting from compression members and for projecting elements of compression flanges of girder,

When $95/(F_y/kc)^{1/2} < b/t < 195/(F_y/kc)^{1/2}$, $Q_s = 1.293 - 0.00309 \cdot (b/t) \cdot (F_y/kc)^{1/2}$

When $b/t > 195/(F_y/kc)^{1/2}$, $Q_s = 26{,}200 \cdot kc/[F_y(b/t)^2)]$

Where:

$kc = 4.05/[(h/t)^{0.46}]$ if $h/t > 70$, otherwise $kc = 1.0$.

b. For stiffened compression element,

A reduced effective width b_e is introduced.

a. For the flanges of square and rectangular sections of uniform thickness:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{50.3}{(b/t)\sqrt{f}}\right)\right] \leq b$$

b. For other uniformly compressed elements:

$$b_e = \frac{253t}{\sqrt{f}}\left[1 - \left(\frac{44.3}{(b/t)\sqrt{f}}\right)\right] \leq b$$

Consequently, a reduction factor Q_a is introduced and is equal to the effective area divided by the actual area. Combining both these factors, allowable stress for axially loaded compression members containing stiffened or unstiffened elements shall not exceed

$$F_a = F_y \frac{Q_s Q_a\left[1 - \frac{(KL/r)^2}{2C'^2_c}\right]}{5\Big/3 + \frac{3(KL/r)}{8C'_c} - \frac{(KL/r)^3}{8C'^3_c}}$$

Where:

$$C'_c = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}}$$

The section is also checked that the stress is less than 2/3× the critical buckling stress, which is taken as the Euler critical buckling load divided by the cross sectional area:

$$F = \frac{2}{3}\frac{P_{\mathrm{cr}}}{A}$$

where

P_{cr} = the Euler critical buckling load of the column: $\frac{\pi^2 EI}{(KL)^2}$

D1.L.4.1.4 Bending Stress

Allowable bending stress for tension and compression for a structural member, as given in NF-3322.1(d) is:

A. Along Major Axis:

a. For Compact Sections, tension and compression on extreme fibres of compact hot rolled or built-up members symmetrical about and loaded in the plane of their minor axes and meeting the requirements of Subsection NF shall result in a maximum bending stress:

$F_b = 0.66 \times F_y$

If meeting the requirements of this member of:

a. Width-thickness ratio of unstiffened projecting elements of the compression flange shall not exceed $65/\sqrt{F_y}$.

b. Width-thickness ratio of stiffened elements of the compression flange shall not exceed $190/\sqrt{F_y}$.

c. The depth-thickness ratio of the web shall not exceed

$d/t = (640/\sqrt{F_y})[1 - 3.74(f_a/F_y)]$ when $f_a/F_y \le 0.16$

$d/t = 257/\sqrt{F_y}$ when $f_a/F_y > 0.16$

d. The laterally unsupported length of the compression flange of members other than box-shaped members shall not exceed the value of $76b_f/\sqrt{F_y}$ nor $20{,}000/(d/A_f)F_y$.

b. For noncompact and slender elements, NF-3322.1(d)(5) and NF-3322.1(d)(3) are followed respectively.

c. For box-type flexural members, maximum bending stress is:

$F_b = 0.60 \times F_y$

B. Along Minor Axis:

a. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a) and (b), maximum tensile and compressive bending stress shall not exceed:

$F_b = 0.75 \times F_y$

b. For doubly symmetrical members (I shaped) meeting the requirements of NF-3322.1(d)(1)(a), except where $b_f/2t_f$ exceeds $65/\sqrt{F_y}$ but is less than $95/\sqrt{F_y}$, maximum tensile and compressive bending stress shall not exceed:

$F_b = F_y[1.075 - 0.005(b_f/2t_f)\sqrt{F_y}]$

D1.L.4.1.5 Combined Interaction Check

Members subjected to both axial compression and bending stresses are proportioned to satisfy

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{(1 - f_a/F'_{ex})F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a/F'_{ey})F_{by}} \le 1.0$$

and

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

when $f_a/F_a > 0.15$, otherwise

$$\frac{f_a}{F_a} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula, because this would result in a misleadingly liberal ratio. The value of the coefficient Cm is taken as 0.85 for side-sway and $0.6 - 0.4 \cdot (M1/M2)$, but not less than 0.4 for no side-sway.

Members subjected to both axial tension and bending stress are proportioned to satisfy

$$\frac{f_a}{0.60F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} \le 1.0$$

D1.L.4.1.6 Shear Stress

Allowable shear stress on the gross section [ref. NF-3322.6(e)(2)] is calculated as

$$F_v = (F_y/2.89)C_v \le 0.4F_y$$

where

C_v = $\frac{45,000k}{F_y(h/t)^2}$ when $h/b \le 0.8$

$$= \frac{190}{h/t}\sqrt{\frac{k}{F_y}} \text{ when } h/b > 0.8$$

$$k = 4.00 + 5.34/(a/h)^2 \text{ when } a/h \leq 1.0$$

$$= 5.34 + 4.00/(a/h)^2 \text{ when } a/h > 1.0$$

For actual shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as the total flange areas.

D1.L.4.2 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used namely — I-shaped section, Channel, Tee, HSS Tube, HSS Pipe, Angle, Double Angle, Double Channel section.

Member properties may also be specified using the User Table facility except for the General and Prismatic member.

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D1.L.4.3 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements for an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ value (k value in local z-axis) of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instruction (Section 5). Similarly, the `TRACK` value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the `TRACK` value must be set to 1.0.

Note: Unlike many other design codes available in STAAD.Pro (which use the `BEAM` parameter), design per ASME NF 3000 codes in STAAD.Pro is *always* performed based on forces calculated at 13 sections, including ends.

Table 107: ASME NF 3000 2001 & 2004 Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `NF3000 2001` or `NF3000 2004` Specified design code is followed for code checking purpose. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
CAN	0	Used for Deflection Check only. 0 = Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = Deflection check based on the principle that maximum deflection is of the cantilever type
CB	1.0	Bending coefficient dependent upon moment gradient 0.0 = CB is calculated itself Any other user-defined value is accepted.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
CT	0.75	Reduction Coefficient in computing effective net area of an axially loaded tension member. [Refer NF-3322.8(c)(1)(d)]
DFF	None (Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Joint of the member	Joint No. denoting starting point for calculation of "Deflection Length"
DJ2	End Joint of the member	Joint No. denoting end point for calculation of "Deflection Length"
DMAX	45 inch	Maximum allowable depth
DMIN	0.0 inch	Minimum allowable depth
FYLD	36 KSI	Yield strength of steel at temperature in current units.
FU	60 KSI	Ultimate tensile strength of steel in current units.

Parameter Name	Default Value	Description
KBK	1.0	Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit. **Note:** Ignored unless SRL is set to D.
KS	1.0	Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses. **Note:** Ignored unless SRL is set to D.
KV	1.0	Stress Limit Factor applicable to the Design Allowable Shear Stresses. **Note:** Ignored unless SRL is set to D.
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in z-axis (major).
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check
NSF	1.0	Net Section Factor for tension member.
PROFILE	None	Used in member selection. See TR. 48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.

Parameter Name	Default Value	Description
SRL	A	Service level, which defines the service level factors to use for modifying stress values for the service level conditions. **A.** Normal Conditions **B.** Upset **C.** Emergency **D.** Faulted - If any of KS, KV, or KBK parameters are not set, a warning is issued that these must be user-defined. Refer to D1.L.5. ASME NF 3000 Service Level Conditions (on page 1348) for additional information.
STIFF	Member length or depth whichever is greater	Spacing of stiffeners for plate girder design
STYPE	0.0	0.0 = Normal Steel 1.0 = Austenitic Stainless Steel
TMAIN	240 for main member 300 for "Truss" member	Slenderness limit under tension
TRACK	0.0	Controls the levels of detail to which results are reported. • Minimum detail • Intermediate detail level • Maximum detail
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the bottom flange.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression on the top flange.

Notes

1. All values are entered in the current units.

2. The parameters DMAX and DMIN are only used with the MEMBER SELECTION command.

Related Links

- *D1.L.5. ASME NF 3000 Service Level Conditions* (on page 1348)

D1.L.4.4 Code Checking and Member Selection

Both code checking and member selection options are available with the ASME NF-3000 2004 code.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D1.L.4.5 Example of 2004 Code

A cantilever beam of length 100 inch is loaded at its free end with 5 kip compressive load and a uniformly distributed load of 1 kip/inch over the whole span. The beam is assigned with B571806 steel member and is designed in accordance with ASME NF3000 2004.

The corresponding input of STAAD input editor file is shown as below:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-08
END JOB INFORMATION
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 100 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST B571806
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX -5
MEMBER LOAD
1 UNI GY -1.0 0 100
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
STYPE 1 ALL
FYLD 36 ALL
KY 0.75 ALL
```

```
KZ 0.75 ALL
FU 58 ALL
NSF 0.9 ALL
CT 0.85 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

The corresponding TRACK 2 output is as follows:

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************




 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================


      1  ST   B571806                 (AISC SECTIONS)
                            PASS      NF-3322.1(b)      0.635         1
                        5.00 C          0.00        5000.00        0.00
  |-----------------------------------------------------------------------------|
  | SLENDERNESS CHECK:    ACTUAL RATIO:  20.85 ALLOWABLE RATIO: 200.00          |
  | ALLOWABLE STRESSES:   (UNIT - KIP  INCH)                                    |
  | AXIAL: 1.20E+01  FCZ: 2.22E+01  FCY: 2.31E+01  FTZ: 2.22E+01  FTY: 2.31E+01 |
  | SHEAR: 6.28E+00                                                             |
  | ACTUAL STRESSES: (UNIT - KIP  INCH)                                         |
  | AXIAL: 1.06E-01  FBZ: 5.86E+00  FBY: 0.00E+00 SHEAR: 3.99E+00               |
  |-----------------------------------------------------------------------------|
  | SECTION PROPERTIES: (UNIT - INCH)                                           |
  | AXX:  47.00  AYY:  25.08  AZZ:  15.00  RZZ: 22.80  RYY:  3.60               |
  | SZZ:     853.77  SYY:      67.54                                            |
  |-----------------------------------------------------------------------------|
  | PARAMETER: (UNIT - KIP  INCH)                                               |
  | KL/R-Z:  3.29   KL/R-Y: 20.85   UNL:  100.0   CMZ:  1.00   CMY:  1.00       |
  | CB: 1.75   FYLD:    36.00    FU:    58.00   NET SECTION FACTOR: 0.90        |
  | CT:  0.85   STEEL TYPE:  1.0  KS:1.000  KV:1.000  KBK:1.000                 |
  |-----------------------------------------------------------------------------|
  | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -INCH)                      |
  | CLAUSE       RATIO  LOAD        FX        VY        VZ        MZ        MY  |
  | TENSION      0.005    1    5.00E+00       -         -         -         -   |
  | COMPRESSION  0.009    1    5.00E+00       -         -         -         -   |
  | COMP&BEND    0.272    1    5.00E+00       -         -    5.00E+03 0.00E+00  |
  | TEN&BEND     0.000    1    5.00E+00       -         -    5.00E+03 0.00E+00  |
  | SHEAR-Y      0.635    1        -     1.00E+02       -         -         -   |
  | SHEAR-Z      0.000    1        -          -    0.00E+00       -         -   |
  |-----------------------------------------------------------------------------|
```

Note: An asterisk following a critical load case number indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2616) for additional information.

D1.L.5. ASME NF 3000 Service Level Conditions

Service Level Conditions are basically the loading conditions for which the plant structure and its components are to be designed. The same primary load can be multiplied by different factors to signify the different service levels. Also the load combinations for various service levels are different and pre-defined by the code.

D1.L.5.1 Service Levels

The following is a short overview of each of the service levels specified by the code:

Condition	Description
A. Normal Working	
B. Upset	This situation can be termed as a short term failure or a local failure, and the repairing or modification of the structure can be done without shutting the entire plant.
C. Emergency	This situation can be termed as a major failure, and the repairing of the structure can be done only after shutting down the entire plant.
D. Faulted	This situation can be termed as devastation, and the main objective of this level is to have sufficient time for safe relocation of human life and valuable properties, and to initiate the controlled failure of the plant structure. The plant is already at an unusable state and a rare chance to repair it back into the operation.

These Service Levels are the attribute of the whole structure or the structural system. So, the existence of different Service Levels to the different parts of the structure at the same point of time is totally ruled out.

The Service Level Factors are basically few multiplying factor by which the Allowable Stress values are to be multiplied based on the Service Level. The different actions (e.g., Tension, Compression, Bending, Shear etc.) have different Service Level Factors.

However, this is to be noted, the stipulated multiplying factors for creating load combinations for Service Level B, C, and D are to be user defined in this case. The facility of creating Auto Load Combination for different Service Levels is out of the scope of this implementation. The user has to take care of this.

D1.L.5.2 Stress Level Factors

For the Member Design, as per Clause NF-3321.1, the Allowable Stresses may be increased by the Factors as per Table NF-3523(b)-1 and NF-3623(b)-1. Table NF-3523(b)-1 is applicable to Component Support Structures and Table NF-3623(b)-1 is applicable to Piping Support Structures. However, as the values are the same for the service level factors in each table, STAAD.Pro does not make any differentiation between component and piping supports.

Note: Clause NF-3321.1 also indicates that the allowable stress shall be limited to two-thirds (2/3x) the critical buckling stress. However, the critical buckling stress is not clearly defined so it is left to the user to ensure that this code requirement is met.

The values used for the stress level factors in STAAD.Pro are as follows:

Service Level	Ks	Kv	Kbk
A	1.0	1.0	1.0

Service Level	Ks	Kv	Kbk
B	1.33	1.33	1.33
C	1.50	1.50	1.50
D*	KS	KV	KBK

* It is evident from the Table NF-3523(b)-1, that there are no predefined Stress Limit Factors for Service Level D. So, for Service Level D, the Factors Ks, Kv and Kbk are to be user defined. Refer to Appendix F in the code for guidance on values to specify in the design parameters.

where

Ks Stress Limit Factor applicable to the Design Allowable Tensile and Bending Stresses.

Kv Stress Limit Factor applicable to the Design Allowable Shear Stresses

Kbk Stress Limit Factor applicable to the Design Allowable Compressive Axial and Bending stresses to determine the Buckling Limit.

The program uses the service level factors —either those specified for levels A through C or the user defined values in level D— as follows:

- The Allowable Axial Tensile Stress is to be multiplied by Ks
- The Allowable Axial Compressive Stress is to be multiplied by Kbk
- The Allowable Bending Stress is to be multiplied by Ks
- The Allowable Shear Stress is to be multiplied by Kv
- As per NF-3322.1.(e), for checking Combined Stresses as per equation 20, the value of $F'ey$ and $F'ez$ — the Euler Stress divided by the factor of safety, may also be multiplied by the appropriate Stress Limit Factor. This is also implemented. $F'e$ is to be multiplied by Kbk.

Related Links

- *D1.L.1.3 Design Parameters* (on page 1314)
- *D1.L.2.3 Design Parameters* (on page 1323)
- *D1.L.3.4 Design Parameters* (on page 1333)
- *D1.L.4.3 Design Parameters* (on page 1343)

D2. Australian Codes

D2.A. Australian Codes - Concrete Design per AS 3600 - 2001

STAAD.Pro is capable of performing concrete design based on the Australian code AS 3600-2001 *Australian Standard-Concrete Structures*.

D2.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams: Prismatic (Rectangular & Square)
- For Columns: Prismatic (Rectangular, Square, and Circular)

D2.A.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the `MEMBER PROPERTY` command. The following example shows the required input:

```
UNIT MM
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 450. ZD 250.
11 13 PR YD 350.
```

In the above input, the first set of members are rectangular (450 mm depth and 250mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 350 mm diameter. It is absolutely imperative that the user not provide the cross section area (AX) as an input.

D2.A.3 Design Parameters

The program contains a number of parameters which are needed to perform the design. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 1A.1 of this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 108: Australian Concrete Design per AS 3600 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `AUSTRALIAN` to invokes design per AS 3600 - 2001. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
CLB	40 mm	Clear cover for outermost bottom reinforcement.
CLS	40 mm	Clear cover for outermost side reinforcement.
CLT	40 mm	Clear cover for outermost top reinforcement.
DEPTH	YD	Total depth to be used for design. This value defaults to YD as provided under `MEMBER PROPERTIES`.

Parameter Name	Default Value	Description
EFACE	0	Distance from end node of beam to face of support used for shear design; used for shear and torsion calculations.
FC	30 N/mm^2	Compressive strength of concrete.
FYMAIN	400 N/mm^2	Yield Stress for main reinforcing steel. Applicable values per Table 6.2.1 of AS 3600-2001: 250 400 450 500
FYSEC	400 N/mm^2	Yield Stress for secondary reinforcing steel. Applicable values per Table 6.2.1 of AS 3600-2001: 250 400 450 500
MAXMAIN	12 mm	Maximum main reinforcement bar size.
MINMAIN	12 mm	Minimum main reinforcement bar size.
MINSEC	12 mm	Minimum secondary reinforcement bar size.
MMAG	1	Factor by which column design moments are magnified.
RATIO	4.0	Maximum percentage of longitudinal reinforcement in columns.
NSECTION	12	Number of equally spaced sections for design.
SFACE	0	Distance from start node of beam to face of support used for shear design; used for shear and torsion calculations.

Parameter Name	Default Value	Description
TRACK	0.0	For beam design: 0.0 = output consists of reinforcement details at the member start, middle, and end 1.0 = critical moments are printed in addition to `TRACK 0.0` output 2.0 = required steel for intermediate sections defined by `NSECTION` are printedin addition to `TRACK 0.0` output For column design: 0.0 = reinforcement details are printed
WIDTH	ZD	Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES.

D2.A.4 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. There are two options by which the slenderness effect can be accommodated. One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed end moments, the effect of deflections on moment and forces and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro has been written to allow the use of the first option. To perform this type of analysis, use the command `PDELTA ANALYSIS` instead of `PERFORM ANALYSIS`. The `PDELTA ANALYSIS` will accommodate the requirements of the second- order analysis described by AS 3600, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the evaluation of slenderness effects is also by an approximate method. In this method, additional moments are calculated based on empirical formula and assumptions on sidesway.

Considering all of the above information, a P-Delta analysis —as performed by STAAD— may be used for the design of concrete members. However the user must note that to take advantage of this analysis, all the combinations of loading must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the P-delta analysis based on the deflections. Also, note that the proper factored loads (like 1.5 for dead load etc.) should be provided by the user. STAAD.Pro does *not* factor the loads automatically.

D2.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13 (e.g., 0., .1, .2, .25, .3, .4, .5, .6, .7, .75, .8, .9, and 1). All of these sections are scanned to determine the design force envelopes.

D2.A.5.1 Design for Flexure

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections is designed to resist both of these critical sagging and hogging moments. Currently, design of singly reinforced sections only is permitted. If the section dimensions are inadequate as a singly reinforced section, such a message will be permitted in the output. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account the changed effective depths of sections calculated on the basis of reinforcement provided after the preliminary design. Final provisions of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per AS 3600. Although exact curtailment lengths are not mentioned explicitly in the design output (finally which will be more or less guided by the detailer taking into account of other practical consideration), user has the choice of printing reinforcements provided by STAAD at 13 equally spaced sections from which the final detailed drawing can be prepared.

D2.A.5.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design is performed at 13 equally spaced sections (0. to 1.) for the maximum shear forces amongst the active load cases and the associated torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

Example of Input Data for Beam Design:

```
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
FYSEC 415 ALL
FC 35 ALL
CLEAR 25 MEM 2 TO 6
MAXMAIN 40 MEMB 2 TO 6
TRACK 1.0 MEMB 2 TO 9
DESIGN BEAM 2 TO 9
END CONCRETE DESIGN
```

D2.A.6 Column Design

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement. The loading which yields maximum reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. By default, square and rectangular columns are designed with reinforcement distributed on each side equally. That means the total number of bars will always be a multiple of four (4). This may cause slightly conservative results in some cases. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by AS 3600 have been taken care of in the column design of STAAD.

Example of Input Data for Column Design:

```
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
FC 35 ALL
CLEAR 25 MEMB 2 TO 6
MAXMAIN 40 MEMB 2 TO 6
DESIGN COLUMN 2 TO 6
END CONCRETE DESIGN
```

D2.A.7 Slab or Wall Design

To design a slab or wall, it must be modeled using finite elements. The command specifications are in accordance with Chapter 2 and Chapter 6 of the specification.

Elements are designed for the moments Mx and My. These moments are obtained from the element force output. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement. The parameters `FYMAIN`, `FC`, `MAXMAIN`, `MINMAIN`, and `CLEAR` listed in D2.A.3 Design Parameters (on page 1351) are relevant to slab design. Other parameters mentioned in Table 1A.1 are not applicable to slab design.

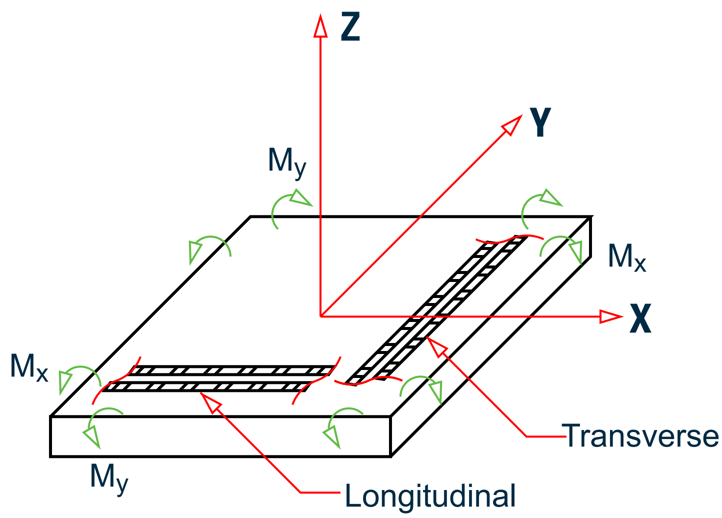


Figure 160: Element moments: Longitudinal (L) and Transverse (T)

Example of Input Data for Slab/Wall Design

```
UNIT NEWTON MMS
START CONCRETE DESIGN
CODE AUSTRALIAN
FYMAIN 415 ALL
```

```
FC 25 ALL
CLEAR 40 ALL
DESIGN ELEMENT 15 TO 20
END CONCRETE DESIGN
```

D2.B. Australian Codes - Steel Design per AS 4100 - 1998

STAAD.Pro is capable of performing steel design based on the Australian code AS 4100-1998 *Standards Australia - Steel Structural Design* , including Amendment 1 (2012).

Related Links

- *V. AS4100 1998 - Bending Capacity* (on page 3803)
- *V. AS4100 1998 - Single Angle Section in Tension* (on page 3836)

D2.B.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD implementation of AS 4100. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D2.B.1.1 Strength Limit States

Strength design capacities (ϕRu) are calculated and compared to user-defined design action effects ($S*$), so as to ensure that $S* \leq \phi Ru$ in accordance with AS 4100 3.4. Details for design capacity calculations are outlined in the sections that follow.

D2.B.1.2 Deflection Limit States

STAAD.Pro’s AS 4100 implementation does not generally check deflections. It is left to the user to check that both local member and frame deflections are within acceptable limits.

Note: Local member deflections parallel to the local member y-axis can be checked against a user-defined maximum “span / deflection” ratio. This can be performed using the `DFF`, `DJ1`, and `DJ2` design parameters, however this is only available for `MEMBER` Design. Details are provided in the sections that follow.

D2.B.1.3 Eccentric Beam Reactions

STAAD.Pro does not automatically account for minimum eccentricity distances for beam reactions being transferred to columns as per AS 4100 4.3.4. However member offsets can be used to model these eccentricities.

Refer to TR.25.1 Member Offset Specification (on page 2296) for further information on the Member Offset feature.

D2.B.1.4 Limit States Not Considered

The following limit states are not directly considered in STAAD.Pro's implementation of AS 4100.

Table 109: Limit States Not Considered in STAAD.Pro AS 4100 Design

Limit State	Code Reference
Stability	AS 4100 3.3
Serviceability	AS 4100 3.5
Brittle Fracture	AS 4100 3.7
Fire	AS 4100 3.9
Other Design Requirements	AS 4100 3.11

D2.B.1.5 Connection Design

STAAD.Pro and Bentley's RAM Connection program currently do *not* support design of connections in accordance with AS 4100. In some cases connection design may govern the size of members. Such considerations are not considered in STAAD.Pro's AS 4100 and should be checked by separately.

D2.B.1.6 Bolts and Welds

Bolt holes and welds are not generally considered in STAAD.Pro's AS 4100 member design.

Note: NSC and NSF D2.B.8 Design Parameters (on page 1364) are used to manually specify a reduction in net section area for compression or tension capacity calculations. These can be used to account for bolt hole area reductions. Further details are provided in the sections that follow.

D2.B.2 Analysis Methodology

Either the elastic or dynamic analysis methods may be used to obtain the forces and moments for design as per AS 4100 section 4.4. Analysis is done for the specified primary and repeat loading conditions. Therefore, it is your responsibility to enter all necessary loads and load combination factors for design in accordance with the AS/NZS 1170 Series or other relevant design codes. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Note: Plastic analysis and design in accordance with AS 4100 section 4.5 is not implemented in STAAD.Pro.

D2.B.2.1 Elastic Analysis

Two types of elastic analysis can be performed using STAAD.Pro in accordance with AS 4100:

i. First Order Linear, Elastic Analysis - used to perform a regular elastic stiffness analysis as per AS 4100 4.4.2.1. Refer to TR.37.1 Linear Elastic Analysis (on page 2620) for additional details on this feature.
ii. Second Order PDelta Linear, Elastic Analysis - Depending on the type of structure, a PDelta analysis may be required in order to capture second-order effects as per AS 4100 4.4.1.2. Second-order effects can be captured in STAAD.Pro by performing a PDelta second-order elastic analysis as per AS 4100 Appendix E. Refer to TR.37.2 P-Delta Analysis Options (on page 2621) for additional details on this feature.

> **Note:** Moment amplification as per AS 4100 clause 4.4.2 is not considered.

> **Tip:** In order to correctly capture second-order effects for combination load cases using a PDelta Analysis, the Repeat Load feature must be used. Second-order effects will not be correctly evaluated if the Load Combination feature is used. Load Combinations are combinations of results where Repeat Loads instruct the program to perform the analysis on the combined load actions. Refer to TR.32.11 Repeat Load Specification (on page 2595) for additional details on using Repeat Loads.

D2.B.2.2 Dynamic Analysis

Dynamic analysis may also be performed and the results combined with static analysis results. Refer TR.32.10 Dynamic Loading Specification (on page 2498) for further information on Dynamic Loading and Analysis features.

D2.B.3 Member Property Specifications

For specification of member properties, either the steel section library available in STAAD or the User Table facility may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D2.B.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members. An example of the member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Table 110: Available Australian Sections for STAAD.Pro AS 4100 Design

General Profile Type	Australian Sections	Description
I-SECTION	WB, WC	Welded beams and columns
	UB, UC	Universal beams and columns
T-SECTION	BT, CT	Tees cut from universal beams and columns

General Profile Type	Australian Sections	Description
CHANNEL	PFC	Parallel flange channels
ANGLE	EA, UA	Equal and unequal angles
TUBE	SHS, RHS	Square and rectangular hollow sections
PIPE	CHS	Circular hollow sections

Note: STAAD.Pro will not design the following section types to AS 4100: Double Profiles (D), Composite Sections (C), Top Cover Plates (TC), Bottom Cover Plates (BC), and Top & Bottom Cover Plates (TB), Double Channels (D, BA, & FR) and Double Angles (LD & SD). Refer to Section Profile Tables in the Graphical Environment for these options.

Tip: When adding and assigning sections using the built-in steel section library through the Graphical Environment, STAAD.Pro's default tables are American. To change the default tables to Australian, select **File** > **Configuration** from the STAAD.Pro Start page (no input file open). Set the Default Profile Table to Australian on the Configure Program dialog Section Profile Table.

Following are the descriptions of different types of sections.

Refer to G.6.2 Built-In Steel Section Libraries (on page 2116) for additional information.

D2.B.4.1 UB Shapes

These shapes are designated in the following way.

```
20 TO 30 TA ST UB150X14.0
36 TO 46 TA ST UB180X16.1
```

D2.B.4.2 UC Shapes

The designation for the UC shapes is similar to that for the UB shapes.

```
25 TO 35 TA ST UC100X14.8
23 56 TA ST UC310X96.8
```

D2.B.4.3 Welded Beams

Welded Beams are designated in the following way.

```
25 TO 35 TA ST WB700X115
23 56 TA ST WB1200X455
```

D2.B.4.4 Welded Columns

Welded Columns are designated in the following way.

```
25 TO 35 TA ST WC400X114
23 56 TA ST WC400X303
```

D2.B.4.5 Parallel Flange Channels

Shown below is the syntax for assigning names of channel sections.

```
1 TO 5 TA ST PFC75
6 TO 10 TA ST PFC380
```

D2.B.4.6 Double Channels

Back-to-back double channels, with or without a spacing between them, are available. The letter D in front of the section name will specify a double channel.

```
11 TA D PFC230
17 TA D C230X75X25 SP 0.5
```

In the above set of commands, member 11 is a back-to-back double channel PFC230 with no spacing in between. Member 17 is a double channel PFC300 with a spacing of 0.5 length units between the channels.

D2.B.4.7 Angles

Two types of specification may be used to describe an angle. The standard angle section is specified as follows:

```
16 20 TA ST A30X30X6
```

The above section signifies an angle with legs of length 30 mm and a leg thickness of 6 mm. This specification may be used when the local Z axis corresponds to the z-z axis specified in Chapter 2. If the local Y axis corresponds to the z-z axis, type specification "RA" (reverse angle) may be used.

```
17 21 TA RA A150X150X16
```

Note: Single angles must be specified with an “RA” (Single Angle w/Reverse Y-Z Axis) in order to be designed to AS 4100. This is to ensure that the major and minor principal axes align with the local member z and y axes respectively, similar to other section profiles.

D2.B.4.8 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by means of input of the words SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose.

```
33 35 TA SD A65X50X5 SP 0.6
37 39 TA LD A75X50X6
43 TO 47 TA LD A100X75X10 SP 0.75
```

D2.B.4.9 Tubes (Rectangular or Square Hollow Sections)

Tubes can be assigned in 2 ways. In the first method, the designation for the tube is as shown below. This method is meant for tubes whose property name is available in the steel table. In these examples, members 1 to 5 consist of a 2X2X0.5 inch size tube section, and members 6 to 10 consist of 10X5X0.1875 inch size tube section. The name is obtained as 10 times the depth, 10 times the width, and 16 times the thickness.

```
1 TO 5 TA ST TUB20202.5
6 TO 10 TA ST TUB100503.0
```

In the second method, tubes are specified by their dimensions. For example,

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a height of 8 length units, width of 6 length units, and a wall thickness of 0.5 length units. Only code checking, no member selection, will be performed for TUBE sections specified in this latter manner.

D2.B.4.10 Pipes (Circular Hollow Sections)

Pipes can be assigned in 2 ways. In the first method, the designation for the pipe is as shown below. This method is meant for pipes whose property name is available in the steel table.

```
1 TO 5 TA ST PIP180X5
6 TO 10 TA ST PIP273X6.5
```

In the second method, pipe sections may be provided by specifying the word PIPE followed by the outside and inside diameters of the section. For example,

```
1 TO 9 TA ST PIPE OD 25.0 ID 20.0
```

specifies a pipe with outside diameter of 25 length units and inside diameter of 20 length units. Only code checking, no member selection, will be performed on pipes specified in this latter manner.

D2.B.4.11 Sample File Containing Australian Shapes

```
STAAD SPACE
UNIT METER KN
JOINT COORD
1 0 0 0 11 100 0 0
MEMB INCI
1 1 2 10
UNIT CM
MEMBER PROPERTIES AUSTRALIAN
* UB SHAPES
1 TA ST UB200X25.4
* UC SHAPES
2 TA ST UC250X89.5
* CHANNELS
3 TA ST PFC125
* DOUBLE CHANNELS
4 TA D PFC200
* ANGLES
5 TA ST A30X30X6
* REVERSE ANGLES
6 TA RA A150X150X16
* DOUBLE ANGLES - SHORT LEGS BACK TO BACK
7 TA SD A65X50X5 SP 0.6
* DOUBLE ANGLES - LONG LEGS BACK TO BACK
8 TA LD A100X75X10 SP 0.75
* TUBES (RECTANGULAR OR SQUARE HOLLOW SECTIONS)
9 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
* PIPES (CIRCULAR HOLLOW SECTIONS)
10 TA ST PIPE OD 25.0 ID 20.0
PRINT MEMB PROP
FINISH
```

D2.B.5 Section Classification

The AS 4100 specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as compact, noncompact, or slender; depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD determines the section classification for the standard shapes and user specified shapes. Design is performed for all three categories of section described above.

D2.B.6 Material Properties

For specification of material properties, the user can use either:

a. built-in material constants
b. user-defined materials

Refer TR.26.2 Specifying Constants for Members and Elements (on page 2305) for further information on the Built-in Material Constants feature.

Refer TR.26.1 Define Material (on page 2303) for further information on the Define Material feature.

D2.B.6.1 Young's Modulus of Elasticity (E)

STAAD.Pro's default steel material's E value is 205,000 MPa. However AS 4100 section 1.4 states that the modulus of elasticity should be taken as 200,000 MPa. There are a number of options to change this value:

- change the steel material through the input file or GUI for each file created
- define a new steel material for each file created
- change the default STAAD.Pro metric E value in the file `C:/Windows/StaadPro20070.ini`, going to the "[Material-Metric]" section, and changing `E1=205.0e6` to `E1=200.0e6`. Restart STAAD.Pro for this to take effect.

> **Caution:** Virtualization features of Windows 7, Windows 8, and Windows 10 may require additional files to be modified. Contact Bentley Technical Support for assistance.

D2.B.7 Member Resistances

The member resistance is calculated in STAAD according to the procedures outlined in AS 4100. Calculated design capacities are compared to corresponding axial, bending moment, and shear forces determined from the STAAD.Pro analysis. These are used to report the fail or pass status for the members designed.

Two types of design checks are typically performed per AS 4100:

- Nominal section checks
- Nominal member checks

The nominal section capacity refers to the capacity of a cross-section to resists applied loads, and accounts for cross-section yielding and local buckling effects. The nominal member capacity on the other hand refers to the capacity of a member to resist applied loads, and includes checks for global member buckling effects including Euler buckling, lateral-torsional buckling, etc.

D2.B.7.1 Axial Tension

The criteria governing the capacity of tension members are based on two limit states per AS 4100 Section 7. The limit state of yielding of the gross section is intended to prevent excessive elongation of the member.

The second limit state involves fracture at the section with the minimum effective net area φN_t section axial tension capacities are calculated (Cl.7.2). Through the use of the `NSF` parameter (see D2.B.8 Design Parameters (on page 1364)), you may specify the net section area. STAAD calculates the tension capacity of a member based on these two limit states per Cl.7.1 and Cl.7.2 respectively of AS 4100. Eccentric end connections can be taken into account using the `KT` correction factor, perCl.7.3. The f_y yield stress is based on the minimum plate yield stress. Parameters `FYLD`, `FU`, and `NSF` are applicable for these calculations.

D2.B.7.2 Axial Compression

The compressive strength of members is based on limit states per AS 4100 Section 6. It is taken as the lesser of nominal section capacity and nominal member capacity. Nominal section capacity, φN_s , is a function of form factor (Cl.6.2.2), net area of the cross section, and yield stress of the material. Through the use of the `NSC` parameter (see D2.B.8 Design Parameters (on page 1364)), you may specify the net section area. Note that this parameter is different from that corresponding to tension. The program automatically calculates the form factor. The k_f form factors are calculated based on effective plate widths per Cl.6.2.4, and the f_y yield stress is based on the minimum plate yield stress.

Nominal member capacity, φN_c , is a function of nominal section capacity and member slenderness reduction factor (Cl.6.3.3). This value is calculated about both principal x and y axes. Here, you are required to supply the value of α_b (Cl.6.3.3) through the `ALB` parameter (see D2.B.8 Design Parameters (on page 1364)). The effective length for the calculation of compressive strength may be provided through the use of the parameters `KY`, `KZ`, `LY`, and `LZ` (see D2.B.8 Design Parameters (on page 1364)).

D2.B.7.3 Bending

Bending capacities are calculated to AS 4100 Section 5. The allowable bending moment of members is determined as the lesser of nominal section capacity and nominal member capacity (ref. Cl.5.1).

The nominal section moment capacity, φM_s, is calculated about both principal x and y axes and is the capacity to resist cross-section yielding or local buckling and is expressed as the product of the yield stress of the material and the effective section modulus (ref. Cl.5.2). The effective section modulus is a function of D2.B.5 Section Classification (on page 1361) (i.e., compact, noncompact, or slender) and minimum plate yield stress f_y. The nominal member capacity depends on overall flexural-torsional buckling of the member (ref.Cl.5.3).

Note: For sections where the web and flange yield stresses ($f^{y,web}$ and $f_{y.flange}$ respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements.

Member moment capacity, φM_b , is calculated about the principal x axis only (ref. Cl.5.6). Critical flange effective cross-section restraints and corresponding design segment and sub-segments are used as the basis for calculating capacities.

D2.B.7.4 Interaction of Axial Force and Bending

Combined section bending and shear capacities are calculated using the shear and bending interaction method as per Cl.5.12.3.

Note: This check is only carried out where φV_v section web shear capacities are calculated. Refer Table 1B.6-1 for details.

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. Here, the adequacy of a member is also examined against both section (ref. Cl.8.3.4) and member capacity (ref.Cl.8.4.5). These account for both in-plane and out-of-plane failures. If the summation of the left hand side of the equations, addressed by the above clauses, exceeds 1.0 or the allowable value provided using the `RATIO` parameter (see D2.B.8 Design Parameters (on page 1364)), the member is considered to have FAILed under the loading condition.

D2.B.7.5 Shear

Section web shear capacity, φV_v , is calculated per Cl.5.11, including both shear yield and shear buckling capacities. Once the capacity is obtained, the ratio of the shear force acting on the cross section to the shear capacity of the section is calculated. If any of the ratios (for both local Y & Z-axes) exceed 1.0 or the allowable value provided using the `RATIO` parameter (see D2.B.8 Design Parameters (on page 1364)), the section is considered to have failed under shear.

Table 1B.6-1 below highlights which shear capacities are calculated for different profile types.

Table 111: Section Type Shear Checks

General Profile Type	Australian Section	Shear Checks
I-SECTION (i.e., parallel to minor principal y-axis)	WB, WC, UB, UC	Calculated for web only
T-SECTION	BT, CT	
CHANNEL	PFC	
ANGLE	EA, UA	No checks performed
TUBE	SHS, RHS	Calculated parallel to both x & y principal axes
PIPE	CHS	Per AS 4100 5.11.4

Note: Only unstiffened web capacities are calculated. Stiffened webs are not considered. Bearing capacities are not considered.

D2.B.7.6 Torsion

STAAD.Pro does *not* design sections or members for torsion for AS 4100.

D2.B.7.7 Slenderness

The slenderness check is *not* used as a critical member ratio check. If a slenderness check is included and exceed unity (1.0), then it may be reported as the governing criteria. However, if it is less than unity, it will be reported but will not be used as the governing ratio.

D2.B.8 Design Parameters

The design parameters outlined in the following table are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs. The design scope indicates whether design parameters are applicable for `MEMBER` Design, `PMEMBER` Design, or both.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 112: Australian Steel Design Parameters

Parameter Name	Default Value	Design Scope	Description
CODE	-		Must be specified as AUSTRALIAN to invoke design per AS 4100 - 1998. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
ALB	2.0		Member section constant (refer cl. 6.3.3) If ALB is 2.0, it is automatically calculated based on TABLE 6.3.3(1), 6.3.3(2); otherwise the input value is used.
ALM	0.0		Moment modification factor (refer cl. 5.6.1.1) If ALM is 0.0, it is automatically calculated based cl. 5.6.1.1; otherwise the input value is used.
BEAM	0.0		0.0 = design only for end moments and those at locations specified by SECTION command. 1.0 = Perform design for moments at twelfth points along the beam.
DFF	None (Mandatory for deflection check)	Analytical members only	"Deflection Length"/ Maximum Allowable local deflection.
DJ1	Start Joint of member	Analytical members only	Joint No. denoting start point for calculation of "deflection length"
DJ2	End Joint of member	Analytical members only	Joint No. denoting end point for calculation of "deflection length"
DMAX	45.0 [in.]		Maximum allowable depth (Applicable for member selection)
DMIN	0.0 [in.]		Minimum required depth (Applicable for member selection)
FU	500.0 [MPa]		Ultimate strength of steel.
FYLD	250.0 [MPa]		Yield strength of steel.
IST	1		Steel type - 1 - SR, 2 - HR, 3 - CF, 4 - LW, 5 - HW **Note:** See p.47 of AS 4100-1998.

Parameter Name	Default Value	Design Scope	Description
KT	1.0		Correction factor for distribution of forces (refer cl. 7.2)
KY	1.0		K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.
KZ	1.0		K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
LHT	0	Physical members only	Load height position as described in Table 5.6.3(2) of AS 4100:1998 0) at Shear center 1) At top flange
LX	see desc.		Unbraced length between torsional restraints. The default is the distance between partial or full restraints which effectively prevent twist of the section about its centroid as per Sec. 8.4.4.1.2.
LY	Member Length		Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.
LZ	Member Length		Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
MAIN	0.0		Slenderness checks for compression. Checks are not explicitly required per AS 4100. A value of either 0.0 or 1.0 suppresses the slenderness ratio check. Any value greater than 1.0 is used as the limit for slenderness in compression.
NSC	1.0		Net section factor for compression members = An / Ag (refer cl. 6.2.1)
NSF	1.0		Net section factor for tension members.
PBRACE	None	Physical members only	Refer to D2.B.12 Physical Member Design (on page 1372) for details on the PBRACE parameter.
PHI	0.9		Capacity reduction factor
RATIO	1.0		Permissible ratio of actual load effect to the design strength.

Parameter Name	Default Value	Design Scope	Description
SGR	0		Steel Grade. Refer to Note c (on page 1368) below. 0) Default (see note below) 2) AS/NZS 3679.1 350 3) AS/NZS 3679.1 300 4) AS/NZS 1163 C450 5) AS/NZS 1163 C350 6) AS/NZS 1163 C250 7) AS/NZS 3678 450 8) AS/NZS 3678 400 9) AS/NZS 3678 350 10) AS/NZS 3678 WR350 11) AS/NZS 3678 300 12) AS 3597 500 13) AS 3597 600 14) AS 3597 700
SKL	1.0		A load height factor given in Table 5.6.3(2)
SKR	1.0		A lateral rotation restraint factor given in Table 5.6.3(3)
SKT	1.0		A twist restraint factor given in Table 5.6.3(1)
TMAIN	400		Slenderness limit for tension. Checks are not explicitly required per AS 4100. Any value greater than 1.0 is used as the limit for slenderness in tension.
TRACK	0		Output detail 0) report only minimum design results 1) report design strengths in addition to `TRACK 0` output 2) = provide full details of design
TSP	Member web depth		Spacing of the transverse stiffeners.
UNB	Member Length		Unsupported length in bending compression of the bottom flange for calculating moment resistance.
UNT	Member Length		Unsupported length in bending compression of the top flange for calculating moment resistance.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

D2.B.8.1 Notes

a. DFF, DJ1, and DJ2 – Deflection calculations

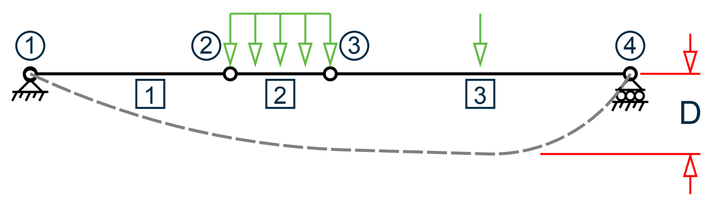
Compute Delta = $SQRT((DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2)$

Compute Length = distance between DJ1 and DJ2 or, between start node and end node, as the case may be.

Note: Deflection calculations are not applicable to PMEMBERs.

i. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.



D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

ii. If DJ1 and DJ2 are not used, "Deflection Length"will default to the member length and local deflections will be measured from original member line.

iii. It is important to note that unless a DFF value is specified, STAAD.Pro will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF.

b. LHT Parameter

If the shear force is constant within the segment, longitudinal position of the load is assumed to be at the segment end.

If there is any variation of the shear force and the load is acting downward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU) Kl is assumed to be 1.4.

If there is any variation of the shear force and the load is acting upward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU) Kl is assumed to be 1.0 as the load acting at the top flange is contributing to stabilize against local torsional buckling.

c. SGR Parameter

Rolled profiles typically use an AS/NZS 3679.1 grade (i.e., SGR 2 or SGR 3), hollow profiles typically use an AS/NZS 1163 grade (i.e., SGR 4 to SGR 6), welded profiles typically use an AS/NZS 3678 grade (i.e., SGR 7 to SGR 11), and pressure vessels typically use an AS/NZS 3597 grade (i.e., SGR 12 to SGR 14).

When an explicit grade has not been specified or the option `SGR 0` has been specified, the program determines the steel grade as follows:

Table 113: Default steel grades used for the SGR parameter

Section Type	Steel Grade Used
WB, WC, Tee section cut from WB and WC, other welded and UPT sections	AS 3678 300
UB, UC, Tee section cut from UB and UC, EA, UA and all UPT sections UB, UC, Tee section cut from UB and UC, EA, UA and all other rolled sections	AS 3679.1 300
Pipe, Tube, CHS, RHS, SHS Pipe, Tube, CHS, RHS, SHS	AS 1163 C250

Note: If a value for the FYLD parameter has been specified, then that value will be used. Otherwise, the SGR value will be used to determine the yield strength and tensile strength values for the steel. based on maximum thickness of the individual elements of the section. Only for shear capacity calculation web thickness is used. Similarly, Tensile Strength is determined either from FU parameter or from SGR parameter.

Caution: A check is introduced to see if yield stress is more than 450 MPa or not. If it is, a warning is issued and the yield stress is set to 450 MPa.

D2.B.8.2 Example

The following example uses the member design facility in STAAD.Pro. However, it is strongly recommended to use the Physical member design capabilities for AS 4100:

```
PARAMETER 1
CODE AUSTRALIAN
ALB 0.0 MEMBER ALL
ALM 1.13 MEMBER ALL
BEAM 1.0 MEMBER ALL
DFF 250.0 MEMBER ALL
DMAX 0.4 MEMBER ALL
DMIN 0.25 MEMBER ALL
FU 400.0 MEMBER ALL
FYLD 310.0 MEMBER ALL
IST 2.0 MEMBER ALL
KT 0.85 MEMBER ALL
KX 0.75 MEMBER ALL
KY 1.0 MEMBER ALL
LX 4.5 MEMBER ALL
LY 6.0 MEMBER ALL
MAIN 1.0 MEMBER ALL
NSC 0.9 MEMBER ALL
NSF 1.0 MEMBER ALL
PHI 0.9 MEMBER ALL
RATIO 0.9 MEMBER ALL
SGR 1.0 MEMBER ALL
SKT 1.0 MEMBER ALL
SKL 1.0 MEMBER ALL
SKR 1.0 MEMBER ALL
TRACK 2.0 MEMBER ALL
UNB 3.4 MEMBER ALL
```

```
UNT 6.8 MEMBER ALL
CHECK CODE MEMBER ALL
```

D2.B.9 Code Checking

The purpose of code checking is to evaluate whether the provided section properties of the members are adequate for the specified loads as per AS 4100 requirements.

Tip: The D2.B.10 Member Selection (on page 1371) facility can be used to instruct the program to select a different section if the specified section is found to be inadequate.

Code checking for an analytical member is done using forces and moments at every twelfth point along the beam. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. The extent of detail of the output can be controlled by using the `TRACK` parameter.

Note: Code checking cannot be performed on composite and prismatic sections.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Example of commands for code checking:

```
UNIT NEWTON METER
PARAMETER
CODE AUSTRALIAN
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
CHECK CODE MEMB 3 4
```

D2.B.9.1 Physical Members

For physical members (`PMEMBER`s), code checks are performed at section stations positioned at $1/12^{th}$ points along each analytical member included in the `PMEMBER`. It is up to you to determine if these locations cover critical sections for design, and adjust as necessary. The number of stations for `PMEMBER` Design cannot be altered, however the analytical members can be split so that in effect more stations are checked for a `PMEMBER`.

For each section station along a `PMEMBER`, section capacity checks are carried for design actions at that station location. Member capacity checks are also carried out for each station. For these the program searches each side of the station to find adjacent effective restraints and design forces and moments. This allows the program to determine the segment / sub-segment that the section station resides in, and then proceeds to calculate the member capacities. Enough section stations should be included to capture all segments / sub-segments for checking.

Note: When checking combined actions for the section capacities, the design actions at the section station are used. However when checking combined actions for the member capacities, the maximum forces from anywhere along the segment / sub-segment being considered are used. This is as stipulated in AS 4100 8.2.

The output reports whether the member has PASSed or FAILed the design checks, as well as the critical condition, critical load case, magnitudes of design actions for the most critical cross-section location (distance from the start joint), and complete calculations for design. The TRACK design parameter can be used to control the level of detail provided in the output.

Tip: You can use the **Utilization Ratio** tool in the graphical user interface **Postprocessing** workflow to view color-coded results.

In some cases some of the output will report “N/A” values. This occurs where a calculation does not apply to a member. For example if a member never goes into tension then no values can be reported in the tension capacity output sections.

Note: As per AS 4100 1.4, the `TRACK 2.0` detailed level of output for PMEMBER Design uses x and y subscripts to refer to major and minor principal axes respectively. These differ to STAAD.Pro local member axes, where z and y refer to major and minor principal axes.

D2.B.10 Member Selection

This process incrementally checks increasing section profile sizes until a size is found that is AS 4100 compliant, or the largest section has been checked. Only section profiles of the same type as modeled are incrementally checked, with the increasing sizes based on a least weight per unit length criteria.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table.

The design calculations for Member Selection are the same as for D2.B.9 Code Checking (on page 1370).

Tip: A Fixed Group command is also available, and can be used to force all members within a user-defined group to take the same section size based on the most critical governing design criteria for all members within that group. This is particularly useful when you want to use the Member Selection feature, but want a group of elements to have the same size. Refer to TR.50 Group Specification (on page 2680) for information on using this feature.

Note: Member Selection will change member sizes, and hence will change the structure’s stiffness matrix. In order to correctly account for this, a subsequent analysis and Code Check should be performed to ensure that the final structure is acceptable. This may need to be carried out over several iterations.

Note: Composite and prismatic sections cannot be selected.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

Example of commands for member selection:

```
UNIT NEWTON METER
PARAMETER
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
SELECT MEMB 3 4
```

D2.B.11 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. The term `CRITICAL COND` refers to the section of the AS 4100 specification which governs the design.

D2.B.12 Physical Member Design

There are two methods available in STAAD.Pro for checking members against the requirements of AS 4100:

a. Analytical member method
b. Physical member method

Herein these are referred to as `MEMBER` Design and `PMEMBER` Design respectively.

Traditionally STAAD.Pro performed code checks based on single analytical members (i.e., single members between two nodes). This implementation remains in place as shown in the example in D2.B.8 Design Parameters (on page 1364). Physical Member (`PMEMBER`) Design on the other hand allows you to group single or multiple analytical members into a single physical design member for the purposes of design to AS 4100.

`PMEMBER` Design also has additional features, including:

- automated steel grades based on section type;
- automated tensile stress (f_u) and yield stress (f_y) values based on plate thicknesses;
- automated segment / sub-segment design;
- improved detailed design calculation output; and

Thus, it is strongly recommended that `PMEMBER` Design be used, even for the design of single analytical members.

D2.B.12.1 Modeling with Physical Members

Physical Members may be grouped by either of the following methods:

- STAAD.Pro Editor - Directly specify physical members in the input file. Refer to TR.16.2 Physical Members (on page 2238) for additional information.
- Graphical Environment - Using the tools in the **Physical Member** group on the **Geometry** ribbon tab, members can be manually or automatically formed.

Note: When creating PMEMBERs for AS 4100, this must be performed in STAAD.Pro's M. Creating Model Objects (on page 653) . Do *not* use the Steel Design workflow.

D2.B.12.2 Segment and Sub-Segment Layout

For calculation of member bending capacities about the principal x-axis, the PMEMBER Design uses the concept of segment / sub-segment design. By default PMEMBERs are automatically broken up into design segments and sub-segments based on calculated effective restraints. User-defined restraints assigned using the PBRACE design parameter are checked to see if they are effective (i.e., if they are placed on the critical flange as per AS 4100 5.5). Restraints not applied to the critical flange are ineffective and hence are completely ignored.

Refer to Section 1B.7 for further information on how user-defined restraints are applied using the `PBRACE` design parameter, including available restraint types, and restraint layout rules.

Note: Segment and sub-segment layouts for PMEMBERs may change for different load cases considered for design. Some restraints may be effective for one particular load case as they are found to apply to the critical flange, however for another load case may be found not to act on the critical flange, and found to be ineffective. In other words the critical flange can change for each load case considered.

Typically the critical flange will be the compression flange, except for segments with a “U” restraint at one end, in which case it will be the tension flange (as is the case for a cantilever).

The PMEMBER Design uses the following routine to determine effective cross-section restraints for each load case considered:

i. first all user-defined restraints are checked to see if they are applied to the compression flange, with those that aren’t ignored;
ii. next a check is made to see if a “U” type restraint is found at either end of the PMEMBER. If this is the case then any adjacent “L” restraints up to the next “F”, “FR”, “P” or “PR” restraint are also ignored, regardless of whether they are placed on the critical or non-critical flange. Refer AS 4100 5.4.2.4.

The compression flange in step 1 of the routine above is calculated based on the bending moments at the locations of the restraints being considered. If the bending moment is zero at the same location as a restraint then the following method is used to determine which flange is critical at the zero moment location:

a. If the zero moment is at the end of the PMEMBER, then the compression flange is based on the bending moment at a small increment from then end;
b. If the zero moment is along the PMEMBER and is a peak value, then the compression flange is based on the bending moment at a small increment from that location;
c. If neither 1 or 2 above is valid, then the stiffer of the restraints at that location is taken. The stiffness of different restraint types from the most stiff to least stiff are taken as outlined in Table 1B.9-3.

Table 114: Assumed Order of Restraint Stiffness for Zero Moment Critical Flange

Stiffness	Restraint Type
Most Stiff	FR
↓	F
↓	PR
↓	P
↓	L
↓	U
Least Stiff	None

Once the effective restraints have been determined, the PMEMBER is divided into segments bounded by “F”, “P”, “FR”, “PR” or “U” effective restraints. These segments are then further divided into sub-segments by effective “L” restraints.

Note: Sub-segment lengths are not automatically checked to determine if they provide full lateral restraint as per AS 4100 5.3.2.4.

For design of cantilevers, the free tip should have user-defined “U” restraints applied to both top and bottom flanges.

Note: If the effective restraints for any load case consist of "U" or "L" restraints only, an error will be reported.

D2.B.12.3 Automated PMEMBER Design Calculations

The AS 4100 PMEMBER Design automates many design calculations, including those required for segment / sub-segment design.

Table 115: Automated PMEMBER AS 4100 Design Parameters and Calculations

Automated Design Calculations	PMEMBER Design Parameter	Comments
α_b compression member section constant per AS 4100 6.3.3.	ALB	
α_m moment modification factor per AS 4100 5.6.1.1.	ALM	Calculated based on moments distribution for individual segments and sub-segments.
f_u tensile strength per AS 4100 2.1.2.	FU	Based on nominal steel grade specified using SGR design parameter and section type.
f_y yield stress per AS 4100 2.1.1.	FYLD	Based on nominal steel grade specified using SGR design parameter and section type.
residual stress category for AS 4100 Table 5.2 and AS 4100 Table 6.2.4.	IST	Based on section type.
correction factor for distribution of forces in a tension member per AS 4100 7.3.	KT	Based on section type and eccentric end connection specified using EEC design parameter.
Load height position for automated calculation of the kl load height factor per AS 4100 Table 5.6.3(2).	LHT	LHT is used for automating calculation of kl load height factors for segments and sub-segments, per AS 4100 Table 5.6.3(2). See D2.B.12.4 Load Height Position (on page 1375) for details.
Segment and sub-segment layout.	PBRACE	Refer to the Segment and Sub-Segment Layout section above for details.
Nominal steel grade.	SGR	Based on section types.
k_t twist restraint factor as per AS 4100 Table 5.6.3(1).	SKT	Based on effective end restraints for each segment / sub-segment.

Automated Design Calculations	`PMEMBER` Design Parameter	Comments
k_l load height factor as per AS 4100 Table 5.6.3(2).	SKL	Based on effective end restraints for each segment / sub-segment, and LHT design parameter (refer above).
k_r lateral rotation restraint factor as per AS 4100 Table 5.6.3(3).	SKR	Based on effective end restraints for each segment / sub-segment. This is where the distinction between "F" and "FR", as well as "P" and "PR" is used.

D2.B.12.4 Load Height Position

When LHT is set to 1.0 to specify a top flange load height position, STAAD.Pro takes the top to be the positive local y-axis of the member.

Note: This may not literally be the top flange for say a column or beam with a beta angle. The local member axes can be viewed in the user interface by selecting **Beam Orientation** in the Diagrams Labels dialog (or by pressing **<Ctrl+O>**).

To automate kl using AS 4100 Table 5.6.3(2), the longitudinal position of the load also needs to be considered, i.e., as either "within segment" or "at segment end".

To determine which of these applies, the shear forces at the ends of each design segment / sub-segment is considered. If the shear force is found to have the same direction and magnitude at both ends, it is assumed that loads act at the segment end.

If on the other hand the shear force at each end is found to have different directions or magnitudes, loads are assumed to act within the segment.

Note: The above method includes an allowance for the self-weight of the member to be considered, as the self-weight always acts through the shear center.

The net sum of the end shears is also used to determine if the load is acting in the positive or negative local member y-axis direction. If LHT is set to 1.0 for top flange loading, the net sum is used to determine whether the top flange loading is acting to stabilise or destabilise the member for lateral torsional buckling. Negative local y-axis net loads act to destabilise the segments / sub-segments, whereas positive local y-axis net loads act to stabilise segments / sub-segments.

D2.B.12.5 Example

```
PARAMETER 1
CODE AUSTRALIAN
DMAX 0.4 PMEMBER 1 TO 42
DMIN 0.25 PMEMBER 1 TO 42
KX 0.75 PMEMBER 1 TO 42
KY 1.0 PMEMBER 1 TO 42
LX 4.5 PMEMBER 1 TO 42
LY 6.0 PMEMBER 1 TO 42
LHT 0.0 PMEMBER 1 TO 42
NSC 0.9 PMEMBER 1 TO 42
NSF 1.0 PMEMBER 1 TO 42
```

```
PBRACE BOTTOM 0.0 F 1.0 F PMEMBER 1 TO 42
PBRACE TOP 0.0 P 0.5 L 1.0 P PMEMBER 1 TO 42
SGR 0.0 PMEMBER 1 TO 42
TRACK 2.0 PMEMBER 1 TO 42
CHECK CODE PMEMBER 1 TO 42
```

D2.B.12.6 Physical Member Restraints Specification

The PBRACE parameter is used to specify the restraint condition along the top and bottom flange of a PMEMBER.

General Format

```
PBRACE { TOP | BOTTOM } f1 r1 f2 r2 … f52 r52 (PMEMB pmember-list)
```

fn a fraction of the PMEMBER length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.

rn one of the possible restraint conditions as follows:

Table 116: Physical Member Restraint Types

Designation, *rn*	Restraint Type	Description
F	Fully restrained	
P	Partially restrained	
L	Laterally restrained	Cannot be specified at the ends of design members.
U	Unrestrained	Can only be applied at the ends of design members, and must be applied to both flanges to be effective. **Caution:** Both top and bottom flanges can not be unrestrained at the same location (as this is unstable).
FR	Fully and rotationally restrained	
PR	Partially and rotationally restrained	
C	Continuously restrained	The flange is assumed to be continuously supported at that flange up to next restraint location. For continuously supported flange unbraced length is assumed to be zero.

Example

```
PBRACE TOP 0.85 FR 0.33 PR 0.33 PR 0.25 F 0.75 L 0.5 PR 1.0 U 0.0 U
PBRACE BOTTOM 0.75 L 0.0 U 0.25 P 0.5 L -
1.0 U PMEMB 3 7
```

Description

Refer to AS 4100 Section 5.5 for a full definition of the critical flange. Typically this will be the compression flange, except for segments with U restraint at one end, then it will be the tension flange (as is the case for cantilever portion at the end).

- when gravity loads are dominant (i.e., negative local y-axis direction), the critical flange of a segment shall be the top flange (i.e., tension).
- when upward wind loads are dominant (i.e., positive local y-axis direction), the critical flange shall be the bottom flange (i.e., tension).

Design physical members are divided into segments by “F”, “P”, “FR”, “PR” or “U” effective section restraints. Segments are further broken down into sub-segments by “L” restraints, but only if the “L” restraints are deemed to be “effective”. “L” restraints are only considered to be effective when positioned on the “critical” flange between “F”, “P”, “FR” or “FP” restraints. If an “L” restraint is positioned on the non-critical flange it shall be completely ignored. Further, if an “L” restraint is positioned between a “U” and an “F”, “P”, “FR” or “PR” restraint, it shall be ignored (regardless of whether it is on the critical or non-critical flange).

Design members must have either a F, P, FR, PR, or U restraint specified at both ends, for both flanges.

- If UNL is not specified, segment length is used as UNL and used as L in effective length calculation as per 5.6.3.
- If ALM i.e., α_m is not provided, automatic calculation of ALM is done based on moments within the segment.
- If SKR i.e., Kr is not provided, it is automatically calculated based on table 5.6.3(3) considering restraint conditions are the end of the segment. If FR or PR is found at only one of the end, Kr is assumed to be 0.85; if FR or PR is found at both the ends, 0.70 is used as Kr.
- If SKT i.e., Kt is not provided, it is automatically calculated based on Table 5.6.3(1) considering end restraints of the segment and section geometric information and segment length.
- If SKL i.e., Kl is not provided, it is automatically calculated based on Table 5.6.3(2) considering end restraints of the segment, Load Height Position parameter, LHT and shear force variation within the segment.

Notes

a. If PMEMBER list is not provided, all the PMEMBERS are restrained by same configuration.
b. It is not necessary to provide the restraint locations in sequence as the program sorts them automatically.
c. Unless specified, PMEMBER ends are assumed to be Fully Restrained (F).
d. While designing any section of the member, effective restraints are searched on each side of the section along the critical flange.
e. The types of restraints applied to the top and bottom flanges at each location determines the effective section restraints. These are outlined in the table below:

Table 117: Restraint Meanings in Critical and Noncritical Flanges

Case	Flange	Restraint on a Critical Flange	Restraint on a Non-Critical Flange	Effective Section Restraint
I		U	U	U
II	1	L	Nothing	L
	2	Nothing	L	None
III	1	P or F	Nothing or U	F

Case	Flange	Restraint on a Critical Flange	Restraint on a Non-Critical Flange	Effective Section Restraint
	2	Nothing or U	P or F	P
IV	1	PR or FR	Nothing or U	FR
	2	Nothing or U	PR or FR	PR
V	1	L, P or F	L, P, F, FR or PR	F
	2	FR or PR	L, P, F, FR or PR	FR

Note: The critical flange can change for each load case considered.

f. If a C-type restraint is defined to a flange at a particular location of the PMEMBER, program will consider that continuous restraint to be effective up to the next bracing point of the point of contraflexure whichever is nearer to the continuous restraint.

Related Links

- *M. To manually add physical member restraints* (on page 666)

D3. British Codes

D3.A. British Codes - Concrete Design per BS8110

Note: This code has been removed from the batch design. To perform design to the current BS 8110 design code, please use the D. Advanced Concrete Design (on page 1071).

D3.B. British Codes - Steel Design per BS5950:2000

STAAD.Pro is capable of performing steel design based on the British code BS 5950-1:2000 *Structural use of steelwork in building - Part 1: Code of practice for design - Rolled and welded sections*, Incorporating Corrigendum No. 1.

D3.B.1 General

The design philosophy embodied in BS5950:2000 is built around the concept of limit state design, used today in most modern steel design codes. Structures are designed and proportioned taking into consideration the limit states at which they become unfit for their intended use.

Two major categories of limit state are recognized - serviceability and ultimate. The primary considerations in ultimate limit state design are strength and stability while that in serviceability limit state is deflection. Appropriate safety factors are used so that the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation of BS5950:2000, members are proportioned to resist the design loads without exceeding the limit states of strength and stability. Accordingly, the most economic section is selected on the basis of the least weight criteria. This procedure is controlled by the designer in specification of allowable member depths, desired section type or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

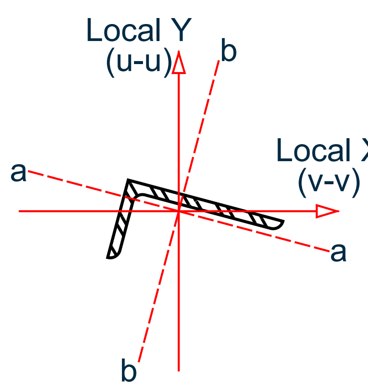
The complete B.S.C. steel tables for both hot rolled and hollow sections are built into the program for use in specifying member properties as well as for the actual design process. See D3.B.4 Built-In Steel Section Library (on page 1380) for information regarding the referencing of these sections. In addition to universal beams, columns, joists, piles, channels, tees, composite sections, beams with cover plates, pipes, tubes, and angles, there is a provision for user provided tables.

The program designs tapered I shaped beams according to Annex G of BS5950. See D3.B.13 Design of Tapered Beams (on page 1398) for a complete description.

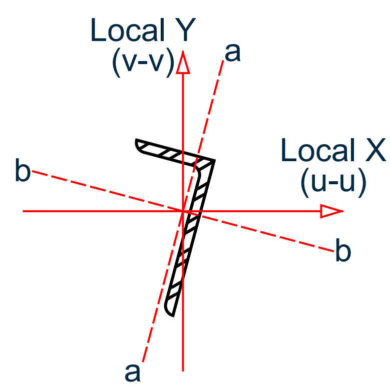
Single Angle Sections

Angle sections are unsymmetric and when using BS 5950:2000 table 25, you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, Lvv, is taken as the LVV parameter or $LY \cdot KY$, if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the LEG parameter, see section 5B.6 for more information on the LEG parameter. The effective length in the a-a axis is taken as $LY \cdot KY$ and the effective length in the b-b axis as $LZ \cdot KZ$.

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).



ST angle and USER table angles



RA angle

D3.B.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D3.B.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. Any user-defined section may be specified, except for `GENERAL` or `PRISMATIC` sections. For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D3.B.4 Built-In Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all BSI steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following are the descriptions of different types of sections available:

Refer to G.6.2 Built-In Steel Section Libraries (on page 2116) for additional information.

D3.B.4.1 Universal Beams, Columns, and Piles

All rolled universal beams, columns and pile sections are available. The following examples illustrate the designation scheme.

```
20 TO  30 TA ST UB305X165X54
33 36  TA ST UC356X406X287
100  102 106 TA ST UP305X305X186
```

D3.B.4.2 Rolled Steel Joists

Joist sections may be specified as they are listed in BSI-80 with the weight omitted. In those cases where two joists have the same specifications but different weights, the lighter section should be specified with an "A" at the end.

```
10 TO  20 TA ST JO152X127
1 2  TA ST JO127X114A
```

D3.B.4.3 Channels

All rolled steel channel sections from the BSI table have been incorporated in STAAD. The designation is similar to that of the joists. The same designation scheme as in BSI tables may be used with the weight omitted.

```
10 TO  15 TA ST CH305X102
55 57 59 61 TA ST CH178X76
```

D3.B.4.4 Double Channels

Back-to-back double channels, with or without spacing between them, are available. The letter "D" in front of the section name will specify a double channel (e.g., D CH102X51, D CH203X89, etc.)

```
51 52  53 TA D CH152X89
70 TO 80 TA D CH305X102 SP 5.
```

(specifies a double channel with a spacing of 5 length units)

Note: Face-to-face double channels can not be used in a `CHECK CODE` command.

D3.B.4.5 Tee Sections

Tee sections are not input by their actual designations, but instead by referring to the universal beam shapes from which they are cut. For example,

```
54 55  56 TA T UB254X102X22
```

(tee cut from UB254X102X22)

D3.B.4.6 Angles

All equal and unequal angles are available for analysis.Note, however, that only angles specified with an RA specification can be designed.

The standard angle section is specified as follows:

```
15 20  25 TA ST UA200X150X18
```

Note: This specification is for "standard" angles, designated by ST. In this specification, the local z-axis corresponds to the Y'-Y' axis shown in the section table. Another common practice of specifying angles assumes the local y-axis to correspond to the Y'-Y' axis. To specify angles in accordance with this convention, use the reverse angle designation, RA. Refer to G.4.2 Local Coordinate System (on page 2088) for details on the local axis systems for standard (ST) and reverse (RA) angles.

```
35 TO  45 TA RA UA200X150X18
```

D3.B.4.7 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle, either LD or SD will serve the purpose. For example,

```
14 TO  20 TA LD UA200X200X16 SP 1.5
23 27 TA SD UA80X60X6
```

"SP" denotes spacing between the individual angle sections.

Note: If the section is defined from a double angle user table, then the section properties must be defined with an 11th value which defines the radius of gyration about an individual sections' principal v-v axis (refer to TR.19.4 Double Angle (on page 2248))

D3.B.4.8 Pipes (Circular Hollow Sections)

To designate circular hollow sections from BSI tables, use PIP followed by the numerical value of diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

```
10 15  TA ST PIP213.2
```

(specifies a 21.3 mm dia. pipe with 3.2 mm wall thickness)

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

```
1 TO  9 TA ST PIPE OD 25.0 ID 20.0
```

(specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units)

Only code checking and no member selection will be performed if this type of specification is used.

D3.B.4.9 Rectangular or Square Hollow Sections (Tubes)

Designation of tubes from the BSI steel table is illustrated below:

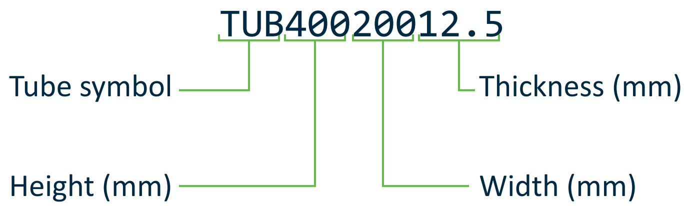


Figure 161: BSI tube nomenclature

Example:

```
15 TO 25 TA ST TUB160808.0
```

Tubes, like pipes, can also be input by their dimensions (Height, Width and Thickness) and not by any table designations.

```
6 TA  ST TUBE DT 8.0 WT 6.0 TH 0.5
(a tube  that has a height of 8, a width of 6, and a wall thickness of 0.5 length
units)
```

Note: Only code checking and no member selection is performed for TUBE sections specified this way.

D3.B.5 Member Capacities

The basic measure of capacity of a beam is taken as the plastic moment of the section. This is a significant departure from the standard practice followed in BS449, in which the limiting condition was attainment of yield stress at the extreme fibers of a given section. With the introduction of the plastic moment as the basic measure of capacity, careful consideration must be given to the influence of local buckling on moment capacity. To assist this, sections are classified as either Class 1, plastic, Class 2, compact, Class 3, semi-compact or Class 4, slender, which governs the decision whether to use the plastic or the elastic moment capacity. The section classification is a function of the geometric properties of the section. STAAD is capable of determining the section classification for both hot rolled and built up sections. In addition, for slender sections, BS5950 recommends the use of a 'stress reduction factor' to reduce the design strength. This factor is again a function of the geometry of the section and is automatically determined by STAAD for use in the design process.

D3.B.5.1 Axial Tension

In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the effective area as outlined in Section 4.6 of the code. STAAD calculates the tension capacity of a given member per this procedure, based on a user supplied net section factor (NSF-a default value of 1.0 is present but may be altered by changing the input value - see D3.B.6 Design Parameters (on page 1384)), proceeding with member selection or code check accordingly. BS5950 does not have any slenderness limitations for tension members.

D3.B.5.2 Compression

Compression members must be designed so that the compression resistance of the member is greater than the axial compressive load. Compression resistance is determined according to the compressive strength, which is a function of the slenderness of the gross section, the appropriate design strength and the relevant strut characteristics. Strut characteristics take into account the considerable influence residual rolling and welding stresses have on column behavior. Based on data collected from extensive research, it has been determined that sections such as tubes with low residual stresses and Universal Beams and Columns are of intermediate performance. It has been found that I-shaped sections are less sensitive to imperfections when constrained to fail about an axis parallel to the flanges. These research observations are incorporated in BS5950 through the use of four strut curves together with a selection of tables to indicate which curve to use for a particular case. Compression strength for a particular section is calculated in STAAD according to the procedure outlined in Annex C of BS5950 where compression strength is seen to be a function of the appropriate Robertson constant (representing Strut Curve) corresponding Perry factor, limiting slenderness of the member and appropriate design strength.

A departure from BS5950:1990, generally compression members are no longer required to be checked for slenderness limitations, however, this option can be included by specifying a MAIN parameter. Note, a slenderness limit of 50 is still applied on double angles checked as battened struts as per clause 4.7.9.

D3.B.5.3 Axially Loaded Members With Moments

In the case of axially loaded members with moments, the moment capacity of the member must be calculated about both principal axes and all axial forces must be taken into account. If the section is plastic or compact, plastic moment capacities will constitute the basic moment capacities subject to an elastic limitation. The purpose of this elastic limitation is to prevent plasticity at working load. For semi-compact or slender sections, the elastic moment is used. For plastic or compact sections with high shear loads, the plastic modulus has to be reduced to accommodate the shear loads. The STAAD implementation of BS5950 incorporates the procedure outlined in section 4.2.5 and 4.2.6 to calculate the appropriate moment capacities of the section.

For members with axial tension and moment, the interaction formula as outlined in section 4.8.2 is applied based on effective tension capacity.

For members with axial compression and moment, two principal interaction formulae must be satisfied – *Cross Section Capacity* check (4.8.3.2) and the *Member Buckling Resistance* check (4.8.3.3). Three types of approach for the member buckling resistance check have been outlined in BS5950:2000 - the simplified approach (4.8.3.3.1), the more exact approach (4.8.3.3.2) and Annex I1 for stocky members. As noted in the code, in cases where neither the major axis nor the minor axis moment approaches zero, the more exact approach may be more conservative than the simplified approach. It has been found, however, that this is not always the case and STAAD therefore performs both checks, comparing the results in order that the more appropriate criteria can be used.

Additionally the equivalent moment factors, m_x m_y and m_{yx}, can be specified by the user or calculated by the program.

Members subject to biaxial moments in the absence of both tensile and compressive axial forces are checked using the appropriate method described above with all axial forces set to zero. STAAD also carries out cross checks for compression only, which for compact/plastic sections may be more critical. If this is the case, COMPRESSION will be the critical condition reported despite the presence of moments.

D3.B.5.4 Shear

A member subjected to shear is considered adequate if the shear capacity of the section is greater than the shear load on the member. Shear capacity is calculated in STAAD using the procedure outlined in section 4.2.3, also 4.4.5 and Annex H3 if appropriate, considering the appropriate shear area for the section specified.

Since plastic moment capacity is the basic moment capacity used in BS5950, members are likely to experience relatively large deflections. This effect, coupled with lateral torsional buckling, may result in severe serviceability limit state. Hence, lateral torsional buckling must be considered carefully.

The procedure to check for lateral torsional buckling as outlined in section 4.3 has been incorporated in the STAAD implementation of BS5950. According to this procedure, for a member subjected to moments about the major axis, the 'equivalent uniform moment' on the section must be less than the lateral torsional buckling resistance moment. For calculation of the buckling resistance moment, the procedure outlined in Annex B.2 has been implemented for all sections with the exception of angles. In Annex B.2., the resistance moment is given as a function of the elastic critical moment, Perry coefficient, and limiting equivalent slenderness, which are calculated within the program; and the equivalent moment factor, m_{LT}, which is determined as a function of the loading configuration and the nature of the load (stabilizing, destabilizing, etc).

D3.B.5.6 RHS Sections - Additional Provisions

Rectangular Hollow sections are treated in accordance with S.C.I. recommendations in cases when the plastic axis is in the flange. In such cases, the following expressions are used to calculate the reduced plastic moduli:

For $n \geq 2t(D-2t)/A$

$$S_{rx} = \frac{A^2}{4(B-t)}\left(1-n\right)\left[\frac{2D(B-t)}{A} + n - 1\right]$$

For $n \geq 2t(B-2t)/A$

$$S_{ry} = \frac{A^2}{4(D-t)}\left(1-n\right)\left[\frac{2B(D-t)}{A} + n - 1\right]$$

Related Links

- *V. BS5950 2000 - Fully Restrained Simply Supported Beam* (on page 4522)
- *V. BS5950 2000 - Unrestrained Simply Supported Beam* (on page 4526)
- *V. BS5950 2000 - Beam from UB Restrained at Loading* (on page 4531)
- *V. BS5950 2000 - Beam from UC Restrained at Loading* (on page 4535)
- *V. BS5950 2000 - Pinned Column Using Non-Slender UC* (on page 4539)
- *V. BS5950 2000 - Pinned Column Using Non-Slender RHS* (on page 4542)
- *V. BS5950 2000 - Pinned Column Using Slender CHS* (on page 4544)

D3.B.6 Design Parameters

Available design parameters to be used in conjunction with BS5950 are listed in table 2B.1 along with their default values.

Note: Once a parameter is specified, its value stays at that specified number till it is specified again. This is the way STAAD works for all codes.

Table 118: British Steel Design BS5950:2000 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `BS5950` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
AD	Depth at end/2	Distance between the reference axis and the axis of restraint. See G. 2.3
BEAM	3.0	Beam divisions 0. Design only for end moments or those locations specified by the SECTION command. 1. Calculate forces and moments at 12th points along the member. Establish the location where Mz is the maximum. Use the forces and moments at that location. Clause checks at one location. 2. Same as `BEAM = 1.0` but additional checks are carried out for each end. 3. Calculate moments at 12th points along the member. Clause checks at each location including the ends of the member.
CAN	0	Deflection check method. See Note 1 below. 0. Deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1. Deflection check based on the principle that maximum deflection is of the cantilever type (see note below)
CB	1	Specifies the method used to calculate Mb. 1. Value of Mb from Clause 4.3.6 is used (default). 2. Value of Mbs from Clause 4.7.7 is used.
DFF	None(Mandatory for deflection check, TRACK 4.0)	"Deflection Length" / Maxm. allowable local deflection See Note 1d below.

Parameter Name	Default Value	Description
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length." See Note 1 below.
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length." See Note 1 below.
DMAX *	100.0cm	Maximum allowable depth
DMIN *	0.0 cm	Minimum allowable depth
ESTIFF	0.0	Clauses 4.8.3.3.1 and 4.8.3.3.2 0.0 = Fail ratio uses MIN of 4.8.3.3.1, 4.8.3.3.2. and Annex I1 checks. 1.0 = Fail ratio uses MAX of 4.8.3.3.1, 4.8.3.3.2. and Annex I1 checks.
KY	1.0	K factor value in local y - axis. Usually, this is the minor axis.
KZ	1.0	K factor value in local z - axis. Usually, this is the major axis.
LEG	0.0	Valid range from 0 – 7 and 10. The values correspond to table 25 of BS5950 for fastener conditions. See note 2 below.
LVV *	Maximum of Lyy and Lzz(Lyy is a term used by BS5950)	Used in conjunction with LEG for Lvv as per BS5950 table 25 for double angles. See note 6 below.
LY *	Member Length	Length in local y - axis (current units) to calculate (KY)(LY)/Ryy slenderness ratio.
LZ *	Member Length	Length in local z - axis (current units) to calculate (KZ)(LZ)/Rzz slenderness ratio.

Parameter Name	Default Value	Description
MLT	1.0	Equivalent moment factor for lateral torsional buckling as defined in clause 4.8.3.3.4
MX	1.0	Equivalent moment factor for major axis flexural buckling as defined in clause 4.8.3.3.4
MY	1.0	Equivalent moment factor for minor axis flexural buckling as defined in clause 4.8.3.3.4
MYX	1.0	Equivalent moment factor for minor axis lateral flexural buckling as defined in clause 4.8.3.3.4
NSF	1.0	Net section factor for tension members.
PNL *	0.0	Transverse stiffener spacing (“a” in Annex H1) 0.0 = Infinity Any other value used in the calculations.
PY *	Set according to steel grade (SGR)	Design strength of steel
MAIN	0.0	Slenderness limit for members with compression forces, effective length/ radius of gyration, for a given axis: 0.0 = Slenderness not performed. 1.0 = Main structural member (180) 2.0 = Secondary member. (250) 3.0 = Bracing etc (350)
RATIO	1.0	Permissible ratio of the actual capacities.

Parameter Name	Default Value	Description
SAME**	0.0	Controls the sections to try during a SELECT process. 0.0 = Try every section of the same type as original 1.0 = Try only those sections with a similar name as original, e.g., if the original is an HEA 100, then only HEA sections will be selected, even if there are HEM's in the same table.
SBLT	0.0	Identify Section type for section classification 0.0 = Rolled Section 1.0 = Built up Section 2.0 = Cold formed section
SWAY	none	Specifies a load case number to provide the sway loading forces in clause 4.8.3.3.4 (See additional notes)
SGR	0.0	Steel Grade per BS4360 0.0 = Grade S 275 1.0 = Grade S 355 2.0 = Grade S 460 3.0 = As per GB 1591 – 16 Mn
TB	0.0	LImit of moment capacity in Cl 4.2.5.1: 0 = Mc limit 1.5pyZ 1= Mc limit 1.2 pyZ

Parameter Name	Default Value	Description
TRACK	0.0	Output details 0.0 = Suppress all member capacity info. 1.0 = Print all member capacities. 2.0 = Print detailed design sheet. 4.0 = Deflection Check (separate check to main select / check code)
UNF	1.0	Unsupported length factor applied to the member length for the lateral torsional-buckling effective length per section 4.3.6.7 of BS5950. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL *	Member Length	Unsupported length for calculating lateral torsional-buckling resistance moment section 4.3.6.7 of BS5950. Given in current length units.
WELD	1.0 closed 2.0 open	Weld Type 1.0 = Closed sections. Welding on one side only (except for webs of wide flange and tee sections) 2.0 = Open sections. Welding on both sides (except pipes and tubes)

* current units must be considered.

**For angles, if the original section is an equal angle, then the selected section will be an equal angle and vice-versa for unequal angles.

Note: There was an NT parameter in STAAD.Pro 2005 build 1003 which is now automatically calculated during the design as it is load case dependent.

D3.B.6.1 Notes

1. CAN, DJ1, and DJ2 – Deflection

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the `CAN` parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement.

   If the `CAN` parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1, DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

   Compute Delta = SQRT((DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2)

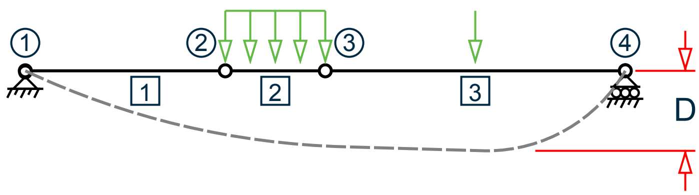
   Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

   Then, if CAN is specified a value 1, dff = L/Delta

   Ratio due to deflection = DFF/dff

b. If `CAN` = 0, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured.

   For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Thus, for all three members here, `DJ1` should be `1` and `DJ2` should be `4`.



   D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

c. If `DJ1` and `DJ2` are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

d. It is important to note that unless a `DFF` value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for `DFF`.

e. The above parameters may be used in conjunction with other available parameters for steel design.

2. `LEG` – follows the requirements of BS5950 table 28. This table concerns the fastener restraint conditions for angles, double angles, tee sections and channels for slenderness. The following values are available:

Table 119: LEG Parameter values

Clause	Bold Configuration	Leg	LEG Parameter
4.7.10.2 Single Angle	(a) - 2 bolts	short leg	1.0
		long leg	3.0
	(b) - 1 bolts	short leg	0.0

Clause	Bold Configuration	Leg	LEG Parameter
		long leg	2.0
4.7.10.3 Double Angles	(a) - 2 bolts	short leg	3.0
		long leg	7.0
	(b) - 1 bolts	short leg	2.0
		long leg	6.0
	(c) - 2 bolts	long leg	1.0
		short leg	5.0
	(d) - 1 bolts	long leg	0.0
		short leg	4.0
4.7.10.4 Channels	(a) - 2 or more rows of bolts		1.0
	(b) - 1 row of bolts		0.0
4.7.10.5 Tee Sections	(a) - 2 or more rows of bolts		1.0
	(b) - 1 row of bolts		0.0

The slenderness of single and double angle, channel and tee sections are specified in BS 5950 table 25 depending on the connection provided at the end of the member. To define the appropriate connection, a LEG parameter should be assigned to the member.

The following list indicates the value of the LEG parameter required to match the BS5950 connection definition:

Clause 4.7.10.2 Single Angle:

a. 2 Bolts: Short leg = 1.0, Long Leg = 3.0
b. 1 Bolt: Short Leg = 0.0, Long Leg = 2.0

For single angles, the slenderness is calculated for the geometric axes, a-a and b-b as well as the weak v-v axis. The effective lengths of the geometric axes are defined as:

$$La = KY \times LY$$

$$Lb = KZ \times LZ$$

The slenderness calculated for the v-v axis is then used to calculate the compression strength p_c for the weaker principal axis (z-z for ST angles or y-y for RA specified angles). The maximum slenderness of the a-a and b-b axes is used to calculate the compression strength p_c for the stronger principal axis.

Alternatively for single angles where the connection is not known or Table 25 is not appropriate, by setting the LEG parameter to 10, slenderness is calculated for the two principal axes y-y and z-z only. The LVV parameter is not used.

For double angles, the LVV parameter is available to comply with note 5 in table 25. In addition, if using double angles from user tables, (refer to TR.19.4 Double Angle (on page 2248)) an eleventh value, r_{vv}, should be supplied at the end of the ten existing values corresponding to the radius of gyration of the single angle making up the pair.

3. PY – Steel Design Strength

   The design parameter PY should only be used when a uniform design strength for an entire structure or a portion thereof is required. Otherwise the value of PY will be set according to the stipulations of BS5950 table 9 in which the design strength is seen as a function of cross sectional thickness for a particular steel grade (SGR parameter) and particular element considered. Generally speaking this option is not required and the program should be allowed to ascertain the appropriate value.

4. UNL, LY, and LZ – Relevant Effective Length

   The values supplied for UNL, LY and LZ should be real numbers greater than zero in current units of length. They are supplied along with or instead of UNF, KY and KZ (which are factors, not lengths) to define lateral torsional buckling and compression effective lengths respectively. Please note that both UNL or UNF and LY or KY values are required even though they are often the same values. The former relates to compression flange restraint for lateral torsional buckling while the latter is the unrestrained buckling length for compression checks.

5. TRACK – Control of Output Formats

   When the `TRACK` parameter is set to 0.0, 1.0, or 2.0, member capacities will be printed in design related output (code check or member selection) in kilonewtons per square meter.

   `TRACK 4.0` causes the design to carry out a deflection check, usually with a different load list to the main code check. The members that are to be checked must have the parameters `DFF`, `DJ1`, and `DJ2` set.

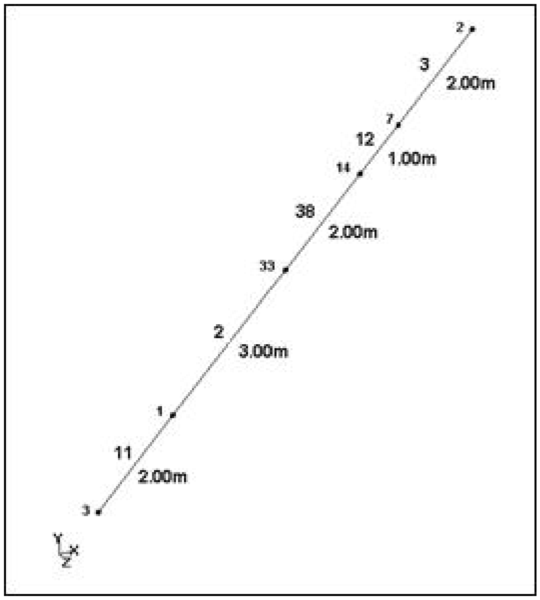
6. MX, MY, MYX, and MLT – Equivalent Moment Factors

   The values for the equivalent moment factors can either be specified directly by the user as a positive value between 0.4 and 1.0 for MX, MY and MYX and 0.44 and 1.0 for MLT.

   The program can be used to calculate the values for the equivalent moment factors by defining the design member with a GROUP command (refer to TR.16.1 Listing of Entities by Specifying Groups (on page 2235)). The nodes along the beam can then be defined as the location of restraint points with J settings.

   Additionally for the MLT parameter, the joint can be defined as having the upper flange restrained (positive local Y) with the a U setting or the lower flange restrained (negative local Y) with a L setting.

   For example, consider a series of 5 beam elements as a single continuous member as shown below:



To enable the steel design, the beam needs to be defined as a group, called MainBeam:

```
START  GROUP DEFINITION
MEMBER
_MainBeam  11 2 38 12 3
END  GROUP DEFINITION
```

Note: This can be done in the User Interface by selecting **Tools** > **Create New Group…**.

Therefore, this 5 beam member has 6 joints such that:

Joint 1 = Node 3
Joint 2 = Node 1
Joint 3 = Node 33
Joint 4 = Node 14
Joint 5 = Node 7
Joint 6 = Node 2

a. Consider MX, MY and MYX

Say that this member has been restrained in its' major axis (local Y) only at the ends. In the minor axis (local Z) it has been restrained at the ends and also at node number 33 (joint 3). For local flexural buckling, it has only been restrained at its ends. Hence:

For the major axis, local Y axis:

MX _MainBeam J1 J6

For the minor axis, local Z axis:

MY _ MainBeam J1 J3 J6

For the lateral flexural buckling, local X axis:

MYX _ MainBeam J1 J6

b. Consider MLT

Say that this member has been restrained at its' ends against lateral torsional buckling and the top flange has been restrained at node number 33 (joint 3) and only the lower flange at node number 7, (joint 5). Hence:

MLT _MainBeam J1 T3 L5 J6

To split the beam into two buckling lengths for L_y at joint 14:

MY _groupname J1 J4 J6

7. SWAY – Sway Loadcase

   This parameter is used to specify a load case that is to be treated as a sway load case in the context of clause 4.8.3.3.4. This load case would be set up to represent the $k_{am\ p}M_s$ mentioned in this clause and the steel design module would add the forces from this load case to the forces of the other load case it is designed for.

   Note that the load case specified with this parameter will not be designed as a separate load case. The following is the correct syntax for the parameter:

Parameter Name	Default Value	Description
SWAY	(load case number)	ALL MEMBER (member list) _(group name)

   Example

```
SWAY  5 MEM 1 to 10
SWAY  6 _MainBeams
```

D3.B.7 Design Operations

STAAD.Pro contains a broad set of facilities for the design of structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem.

The operations to perform a design are:

- Specify the load cases to be considered in the design; the default is all load cases.
- Specify design parameter values, if different from the default values.
- Specify whether to perform code checking or member selection along with the list of members.

These operations may be repeated by the user any number of times depending upon the design requirements.

D3.B.8 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per BS5950. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition of BS5950 code (like any of the BS5950 specifications for compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in D3.B.4 Built-In Steel Section Library (on page 1380) or any of the user defined sections as described in G.6.3 User-Provided Steel Table (on page 2117) , except profiles defined in GENERAL and ISECTION tables.

Note: PRISMATIC sections are also not acceptable steel sections for design per BS5950 in STAAD.Pro.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D3.B.9 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in D3.B.8 Code Checking (on page 1394).

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as above limitations for code checking.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D3.B.10 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

MEMBER	refers to the member number for which the design is performed.
TABLE	refers to steel section name, which has been checked against the steel code or has been selected.
RESULTS	prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
CRITICAL COND	refers to the section of the BS5950 code which governs the design.
RATIO	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
LOADING	provides the load case number, which governed the design
FX, MY, and MZ	provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.

LOCATION specifies the actual distance from the start of the member to the section where design forces govern.

TRACK If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending capacities in compression (MCY & MCZ) and reduced moment capacities (MRY & MRZ), allowable axial capacity in compression (PC) and tension (PT) and shear capacity (PV). TRACK 2.0 will produce the design results as shown in D3.B.9 Member Selection (on page 1395).

An example of each TRACK setting follows:

D3.B.10.1 Example output for TRACK 0.0

```
MEMBER     TABLE        RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
=======================================================================

      1 ST UC305X305X118 PASS     BS-4.3.6           0.769         3
                        179.66 C          0.00        334.46       0.00
```

D3.B.10.2 Example output for TRACK 1.0

```
MEMBER     TABLE        RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
=======================================================================

      1 ST UC305X305X118 PASS     BS-4.3.6           0.769         3
                        179.66 C          0.00        334.46       0.00
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     1 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  519.4 MCY=  234.3 PC=  2455.9 PT=     0.0 MB=  435.0 PV=  600.1|
 | BUCKLING CO-EFFICIENTS mLT = 1.00, mx = 1.00, my = 1.00, myx = 1.00 |
 | PZ=  3975.00    FX/PZ =   0.05       MRZ=  516.9      MRY=  234.3    |
 |----------------------------------------------------------------------|
```

D3.B.10.3 Example output for TRACK 2.0

```
MEMBER     TABLE        RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
=======================================================================

      1 ST UC305X305X118 PASS     BS-4.3.6           0.769         3
                        179.66 C          0.00        334.46       0.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  S 275
     Modulus of elasticity    =  210 kN/mm2
     Design Strength  (py)    =  265  N/mm2

  SECTION PROPERTIES (units - cm)
     Member Length =     600.00
     Gross Area =  150.00  Net Area =  127.50  Eff. Area =  150.00

                                  z-z  axis       y-y  axis
     Moment of inertia         :    27700.004        9060.001
     Plastic modulus           :     1960.000         895.000
     Elastic modulus           :     1761.526         589.460
```

```
      Effective modulus        :     1960.000         895.000
      Shear Area               :      103.471          37.740

   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
      Squash Load              :   3975.00
      Axial force/Squash load  :     0.045

                                   z-z  axis       y-y  axis
      Compression Capacity     :       3551.7         2455.9
      Moment Capacity          :        519.4          234.3
      Reduced Moment Capacity  :        516.9          234.3
      Shear Capacity           :       1645.2          600.1

   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
                                    x-x axis        y-y axis
      Slenderness              :       44.153          77.203
      Radius of gyration (cm)  :       13.589           7.772
      Effective Length         :        6.000           6.000

      LTB Moment Capacity (kNm) and LTB Length (m):   435.00,    6.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   1.00  :   mx =   1.00   :  my =   1.00   :  myx =   1.00
      Mlt =  334.46 :   Mx =  334.46  :  My =   0.00   :   My =   0.00

   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.143     3      -      85.6      -       -       -
   BS-4.3.6       0.769     3      -      85.6      -    334.5      -
   BS-4.7   (C)   0.098     1   239.7      -        -       -       -
   BS-4.8.3.2     0.647     3   179.7     85.6     0.0   334.5     0.0
   BS-4.8.3.3.1   0.842     3   179.7      -        -    334.5     0.0
   BS-4.8.3.3.2   0.842     3   179.7      -        -    334.5     0.0
   ANNEX I.1      0.714     3   179.7      -        -    334.5     0.0
    Torsion and deflections have not been considered in the design.
```

D3.B.11 Plate Girders

Sections will be considered for the Plate Girder checks (BS 5950 Section 4.4) if d/t > 70 ε for “rolled sections” or d/t >62 ε for ‘welded sections’. The parameter SBLT should be used to identify sections as rolled or welded; see the parameter list for more information.

If the plate girder has intermediate stiffeners, the spacing is set with the PNL parameter. These are then used to check against the code clauses “4.4.3.2 - Minimum web thickness for serviceability” and ‘4.4.3.3 - Minimum web thickness to avoid compression flange buckling’. The following printout is then included if a TRACK 2.0 output is selected:

```
Shear  Buckling check is required: Vb = 1070 kN : qw    = 118  N/mm2
d   = 900 mm  :   t  =  10 mm   : a  =  200 mm : pyf = 275 N/mm2
BS-4.4.3.2  status = PASS        : BS-4.4.3.3 status  = PASS
```

The section is then checked for shear buckling resistance using clause “4.4.5.2 - Simplified method” and the result is included in the ratio checks.

D3.B.12 Composite Sections

Sections that have been defined as acting compositely with a concrete flange either from a standard database section using the CM option, or from a modified user WIDE FLANGE database with the additional composite parameters, cannot be designed with BS5950:2000.

D3.B.13 Design of Tapered Beams

Sections will be checked as tapered members provided that are defined either as a Tapered I section or from a USER table.

Example using a Tapered I section:

```
UNIT CM
MEMBER PROPERTY
1 TO 5 TAPERED 100 2.5  75 25 4 25 4
```

Example using a USER table:

```
START USER TABLE
TABLE 1
UNIT CM
ISECTION
1000mm_TAPER
100 2.5 75 25 4 25 4 0  0 0
750mm_TAPER
75 2.5 50 25 4 25 4 0  0 0
END
```

You must specify the effective length of unrestrained compression flange using the parameter `UNL`.

The program compares the resistance of members with the applied load effects, in accordance with BS 5950-1:2000. Code checking is carried out for locations specified by the user via the `SECTION` command or the `BEAM` parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the `TRACK` parameter.

The beam is designed as other wide flange beams apart from the Lateral Torsional Buckling check which is replaced by the Annex G.2.2. check.

D3.B.13.1 Design Equations

A beam defined with tapered properties as defined above will be checked as a regular wide flange (e.g., UB or UC), except that the following is used in place of clause 4.3.6, the lateral torsional buckling check.

D3.B.13.2 Check Moment for Taper Members as per clause G.2.2

The following criterion is checked at each defined check position in the length of the member defined by the BEAM parameter.

$$M_{xi} \leq M_{bi} (1 - F_c/P_c)$$

where

F_c	=	the longitudinal compression at the check location
M_{bi}	=	the buckling resistance moment M_b from 4.3.6 for an equivalent slenderness λ_{TB}, see G.2.4.2, based on the appropriate modulus S, S_{eff}, Z or Z_{eff} of the cross-section at the point i considered
M_{xi}	=	the moment about the major axis acting at the point i considered

P_c	=	the compression resistance from 4.7.4 for a slenderness $\lambda_{TC \cdot y}$, see G.2.3, based on the properties of the minimum depth of cross-section within the segment length L

D3.B.13.2 G.2.3 Slenderness ITC

$\lambda_{TC} = y\lambda$

where

y	=	$\left[\frac{1 + (2a / h_s)^2}{1 + (2a / h_s)^2 + 0.05(\lambda / x)^2} \right]^{0.5}$
λ	=	L_y / r_y
a	=	the distance between the reference axis and the axis of restraint,
h_s	=	the distance between the shear centers of the flanges
L_y	=	the length of the segment
r_y	=	the radius of gyration for buckling about the minor axis
x	=	is the torsional index

D3.B.13.4 G.2.4.2 Equivalent slenderness ITB for tapered members

$\lambda_{TB} = cn_t v_t \lambda$

Where, for a two-flange haunch:

$$v_t = \left[\frac{4a / h_s}{1 + (2a / h_s)^2 + 0.05(\lambda / x)^2} \right]^{0.5}$$

where

C	=	the taper factor, see G.2.5

D3.B.13.5 G.2.5 Taper factor

For an I-section with $D \geq 1.2B$ and $x \geq 20$, the taper factor, c, is as follows:

$$c = 1 + \frac{3}{x - 9}\left(\frac{D_{max}}{D_{min}} - 1\right)^{2/3}$$

where

D_{max}	=	the maximum depth of cross-section within the length Ly, see Figure G.3
D_{min}	=	the minimum depth of cross-section within the length Ly, see Figure G.3
x	=	the torsional index of the minimum depth cross-section, see 4.3.6.8

Otherwise, c is taken as 1.0 (unity).

D3.C. British Codes - Design per BS5400

STAAD.Pro is capable of performing steel design based on the British code BS 5400:Part 3:1982 *Steel, concrete and composite bridges Part 3. Code of practice for design of steel bridges* and Amd No. 4051 and Amd No. 6488.

D3.C.1 General Comments

The British Standard, BS5400 adopts the limit state design philosophy and is applicable to steel, concrete, and composite construction. The code is in ten parts covering various aspects of bridge design. The implementation of part 3, Code of practice for design of steel bridges, in STAAD is restricted in its scope to simply supported spans. It is assumed that the depth remains constant and both construction and composite stages of steel I-Sections can be checked. The following sections describe in more detail features of the design process currently available in STAAD.

D3.C.2 Shape Limitations

The capacity of sections could be limited by local buckling if the ratio of flange outstand to thickness is large. In order to prevent this, the code sets limits to the ratio as per clause 9.3.2. In the event of exceeding these limits, the design process will terminate with reference to the clause.

D3.C.3 Section Class

Sections are further defined as compact or noncompact. In the case of compact sections, the full plastic moment capacity can be attained. In the case of noncompact sections, local buckling of elements may occur prior to reaching the full moment capacity and for this reason the extreme fibre stresses are limited to first yield. In STAAD, section types are determined as per clause 9.3.7 and the checks that follow will relate to the type of section considered.

D3.C.4 Moment Capacity

Lateral torsional buckling may occur if a member has unrestrained elements in compression. The code deals with this effect by limiting the compressive stress to a value depending on the slenderness parameter which is a modified form of the ratio Le/Ry. Le is the effective length governed by the provision of lateral restraints satisfying the requirements of clause 9.12.1. Once the allowable compressive stress is determined then the moment capacity appropriate to the section type can be calculated. STAAD takes the effective length as that provided by the user, defaulting to the length of the member during construction stage and as zero, assuming full restraint throughout, for the composite stage. The program then proceeds to calculate the allowable compressive stress based on appendix G7 from which the moment capacity is then determined.

D3.C.5 Shear Capacity

The shear capacity, as outlined in clause is a function of the limiting shear strength, l, which is dependent on the slenderness ratio. STAAD.Pro follows the iterative procedure of appendix G8 to determine the limiting shear strength of the web panel. The shear capacity is then calculated based on the formula given under clause 9.9.2.2.

D3.C.6 Design Parameters

Available design parameters to be used in conjunction with BS5400 are listed in table 2C.1. Depending on the value assigned to the `WET` parameter, you can determine the stage under consideration. For a composite design check, taking into consideration the construction stage, two separate analyses are required. In the first, member properties are non-composite and the WET parameter is set to 1.0 . In the second, member properties should be changed to composite and the `WET` parameter set to 2.0. Member properties for composite or non-composite sections should be specified from user provided tables (refer to section 5.19 of the manual for specification of user tables). Rolled sections, composite or non-composite, come under WIDE FLANGE section-type and built-up sections under ISECTION. When specifying composite properties the first parameter is assigned a negative value and four additional parameters provided giving details of the concrete section. See user table examples provided.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 120: BS5400 Design Parameters

Parameter Name	Default Value	Description
ESTIFF	0	Specify the criteria used for the design of compression members with moments. 0. Member passes if *either* Cl. 4.8.3.3.1 *or* Cl. 4.8.3.3.2 check. 1. Member passes if *both* Cl. 4.8.3.3.1 *and* Cl. 4.8.3.3.2 check.
KY	1.0	K value for bending about Y-axis. Usually this is minor axis.
KZ	1.0	K value for bending about Z-axis. Usually this is major axis.
LY	Member Length	Length to calculate slenderness ratio for bending about Y-axis, in current units of length.
LZ	Member Length	Length to calculate slenderness ratio for bending about Z-axis, in current units of length.
MAIN	1.0	Grade of concrete: 1. 30 N/mm2 2. 40 N/mm2 3. 50 N/mm2
NSF	1.0	Net section factor for tension members.
PY	*	Yield stress of steel. Set according to Design Strength of steel SGR
RATIO	1	Permissible ratio of actual to allowable stresses.

Parameter Name	Default Value	Description
SBLT	0.0	Steel 0.0 = Rolled Section 1.0 = Built up Section
SGR	0.0	Steel Grade per BS4360 0. Grade 43 1. Grade 50 2. Grade 55
TRACK	1.0	Used to control the level of detail in the output 0. Suppress all member capacities 1. Print all member capacities
UNL	Member Length	Unsupported length for calculating allowable compressive bending stress, in current units of length.
WET	0.0	Used to specify the stage of construction. 0. Wet stage with no data saved for composite stage 1. Wet stage with data saved for composite stage 2. Composite and wet stage combined 3. Composite stage only

D3.C.7 Composite Sections

The definition of composite sections has been provided for in the standard sections definition (refer to TR.20.1 Assigning Properties from Steel Tables (on page 2257) for details). This is purely for analysis and for obtaining the right section properties. It uses the American requirement of 18 times depth (CT) as the effective depth. For more control with British sections two new options are available in user provided tables.

D3.C.7.1 Wide Flange Composite

Using the standard definition of I sections in WIDE FLANGE, 4 additional values can now be provided. The first is the width of concrete to the left of center of the steel web (b1). The second is the concrete width to the right (b2). The third is the concrete depth (d1) to be considered. The last is the modular ratio. The above values are accepted in the program by adding a '-' at the first position on the first line of data. The program now awaits four extra values on line 2 as described above. If (-) is provided on the second line the program requires another 2 breadths + 1 thickness for the bottom plate.

D3.C.7.2 I Section

The same is true for ISECTION definition in user table.

D3.C.7.3 Example

```
UNIT  CM
WIDE  FLANGE
C45752
-66.5  44.98 .76 15.24 1.09 21345 645 21.3 34.185 33.223
150  150 30 10
ISECTION
PG9144
-92.05  2.15 92.05 42.05 3.66 42.05 3.66 197.9 153.9 1730
40 40  12 1
```

The larger British sections have been coded as USER TABLES under wide flange and are available on request to any existing user. Please note however that composite design is *not* available in this portion of STAAD.

D3.D. British Codes - Design per BS8007

STAAD.Pro is capable of performing concrete design based on the British code BS8007:1987 *Design of concrete structures for retaining aqueous liquids*. It is recommended that the design of the structure is carried out according to BS8110, unless modified by the recommendations given in BS8007.

The information in this section is to be used in conjunction with the BS8110. Refer to D3.A. British Codes - Concrete Design per BS8110 (on page 1378)

D3.D.1 Design Process

The design process is carried out in three stages.

1. Ultimate Limit States

   The program is structured so that ultimate design is first carried out in accordance with recommendations given in BS8110. All active design load cases are considered in turn and a tabulated output is printed showing possible reinforcement arrangements. 12, 16, and 20 mm bars are considered with possible spacings from 100,125,150,175, and 200 mm. Within these spacings, the layout providing the closest area of steel is printed under each bar size. Longitudinal and transverse moments together with critical load cases for both hogging and sagging moments are also printed. Minimum reinforcement is in any case checked and provided in each direction. Wood & Armer moments may also be included in the design.
2. Serviceability Limit States

   In the second stage, flexural crack widths under serviceability load cases are calculated. The *first* and *every other occurring* design load case is considered as a serviceability load case and crack widths are calculated based on bar sizes and spacings proposed at the ultimate limit state check.
3. Thermal crack widths

   Crack widths due to longitudinal and transverse moments are calculated directly under bars, midway between and at corners. A tabulated output indicating critical serviceability load cases and moments for top and bottom of the slab is then produced.

Finally thermal, crack width calculations are carried out. Through available parameters, the user is able to provide information on the type of slab, temperature range and crack width limits.

Surface zone depths are determined based on the type of slab and critical areas of reinforcements are calculated and printed in a tabulated form.

Four bar sizes are considered and for each, max crack spacing, Smax and crack widths are calculated for the critical reinforcements and printed under each bar size.

Maximum bar spacing to limit crack widths to the user's limit is also printed under each bar size.

D3.D.2 Design Parameters

The program contains a number of parameters which are needed to perform and control the design to BS8007.

These parameters not only act as a method to input required data for code calculations but give the Engineer control over the actual design process. Default values of commonly used values for conventional design practice have been chosen as the basis. Table 2D.1 contains a complete list of available parameters with their default values.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 121: BS8007 Design Parameters

Parameter Name	Default Value	Description
FC	30 N/mm^2	Concrete grade, in current units of length and force.
CLEAR	20 mm	Distance from the outer surface to the edge of the bar, in current units of length. This is considered the same on both surfaces.
SRA	0.0	Orthogonal reinforcement layout without considering torsional moment Mxy - slabs on -500.orthogonal reinforcement layout with Mxy used to calculate Wood & Armer moments for design. A* Skew angle considered in Wood & Armer equations. A* is any angle in degrees.
SCON	1	Parameter which indicates the type of slab ee. ground or suspended as defined in BS8007 1 = Suspended Slab 2 = Ground Slab

Parameter Name	Default Value	Description
TEMP	30°C	Temperature range to be considered in thermal crack width calculations
CRACK *	0.2 mm	Limiting thermal crack width, in current units of length.

* Provided in current unit systems

D3.D.3 Structural Model

Structural slabs that are to be designed to BS8007 must be modeled using finite elements. Refer to G.5 Finite Element Information (on page 2099) for information on the sign convention used in the program for defining elements

It is recommended to connect elements in such a way that the positive local z axis points outwards away, from the center of the container. In this manner the "Top" of elements will consistently fall on the outer surface and internal pressure loads will act in the positive direction of the local z axis.

An example of a rectangular tank is provided to demonstrate the above procedure.

Element properties are based on the thickness given under `ELEMENT PROPERTIES` command. The following example demonstrates the required input for a 300 mm slab modeled with ten elements.

```
UNIT  MM
ELEMENT  PROPERTIES
1 TO  10 THI 300.0
```

D3.D.4 Wood & Armer Moments

This is controlled by the SRA parameter. If the default value of zero is used, the design will be based on the Mx and My moments which are the direct results of STAAD analysis. The SRA parameter (Set Reinforcement Angle) can be manipulated to introduce Wood & Armer moments into the design replacing the pure Mx, My moments. These new design moments allow the Mxy moment to be considered when designing the section. Orthogonal or skew reinforcement may be considered. SRA set to -500 will assume an orthogonal layout. If however a skew is to be considered, an angle is given in degrees, measured between the local element x axis anti-clockwise (positive). The resulting Mx* and My* moments are calculated and shown in the design format.

D3.E. British Codes - Design per British Cold Formed Steel Code

STAAD.Pro is capable of performing steel design based on the British code BS 5950-5:1998 *Structural use of steelwork in building - Part 5: Code of practice for design of cold formed thin gauge sections* . The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. Cold work of forming strengthening effects have been included as an option.

D3.E.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables published in the “The Steel Construction Institute”, (*Design of Structures using Cold Formed Steel Sections*).

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Z with Lips
- Pipe
- Tube

Shape assignment may be done using the Properties - Whole Strucutre dialog (on page 2793) of the graphical user interface (GUI) or by specifying the section designation symbol in the input file.

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

D3.E.2 Design Procedure

The following two design modes are available:

D3.E.2.1 Code Checking

The program compares the resistance of members with the applied load effects, in accordance with BS 5950-5:1998. Code checking is carried out for locations specified by the user via the `SECTION` command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D3.E.2.2 Member Selection

The user may request that the program search the cold formed steel shapes database (BS standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, presents design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

The program calculates effective section properties in accordance with Section 4 of the subject code. Cross-sectional properties and overall slenderness of members are checked for compliance with:

- Clause 6.2.2, Maximum Effective Slenderness Ratio for members in Compression
- Clause 4.2, Maximum Flat Width Ratios for Elements in Compression

D3.E.3 Design Equations

D3.E.3.1 Tensile Strength

The allowable tensile strength, as calculated in STAAD as per BS5950-5, section 7 is described below.

The tensile strength, P_t of the member should be determined from clause 7.2.1

$$P_t = A_e p_y$$

where

A_e	=	the net area An determined in accordance with cl.3.5.4
p_y	=	the design strength

D3.E.3.2 Combined bending and tension

As per clause 7.3 of BS 5950-5:1998 members subjected to both axial tension and bending should be proportioned such that the following relationships are satisfied at the ultimate limit state

$$F_t/P^t + M_z/M_{cz} + M_y/M_{cy} \le 1$$

$$M_z/M_{cz} \le 1$$

and

$$M_y/M_{cy} \le 1$$

where

F_t	=	the applies tensile strength
P_t	=	the tensile capacity determined in accordance with clause 7.2.1 of the subject code
M_z,M_y,M_{cz},M_{cy}	=	as defined in clause 6.4.2 of the subject code

D3.E.3.3 Compressive Strength

The allowable Compressive strength, as calculated in STAAD as per BS5950-5, section 6 is described below

For sections symmetrical about both principal axes or closed cross-sections which are not subjected to torsional flexural buckling, the buckling resistance under axial load, Pc, may be obtained from the following equation as per clause 6.2.3 of the subject code

$$P_c = \frac{P_E P_{cs}}{\phi + \sqrt{\phi^2 - P_E P_{cs}}}$$

For sections symmetrical about a single axis and which are not subject to torsional flexural buckling, the buckling resistance under axial load, P_c, may be obtained from the following equation as per clause 6.2.4 of the subject code

$$P'_c = \frac{M_c P_c}{(M_c + P_c e_s)}$$

Where the meanings of the symbols used are indicated in the subject clauses.

D3.E.3.4 Torsional flexural buckling

Design of the members which have at least one axis of symmetry, and which are subject to torsional flexural buckling should be done according to the stipulations of the clause 6.3.2 using factored slenderness ratio $\alpha L_E/r$ in place of actual slenderness ratio while reading Table 10 for the value of Compressive strength(p_c).

where

α	=	(P_E/P_{TF}) when $P_E > P_{TF}$
α	=	1, otherwise

Where the meanings of the symbols used are indicated in the subject clause.

D3.E.3.5 Combined bending and compression

Members subjected to both axial compression and bending should be checked for local capacity and overall buckling

Local capacity check as per clause 6.4.2 of the subject code

$$F_c/P^{cs} + M_z/M_{cz} + M_y/M_{cy} \leq 1$$

D3.E.3.6 Overall buckling check as per clause 6.4.3 of the subject code

For beams not subjected to lateral buckling, the following relationship should be satisfied

$$\frac{F_c}{P_c} + \frac{M_z}{C_{bx} M_{cz}\left(1 - \frac{F_c}{P_{Ez}}\right)} + \frac{M_y}{C_{by} M_{cy}\left(1 - \frac{F_c}{P_{Ey}}\right)} \leq 1$$

For beams subjected to lateral buckling, the following relationship should be satisfied:

$$\frac{F_c}{P_c} + \frac{M_z}{M_b} + \frac{M_y}{C_{by} M_{cy}\left(1 - \frac{F_c}{P_{Ey}}\right)} \leq 1$$

where

F_c	=	the applied axial load
P_{cs}	=	the short strut capacity as per clause 6.2.3
M_z	=	the applied bending moment about z axis
M_y	=	the applied bending moment about y axis
M_{cz}	=	the moment capacity in bending about the local Z axis in the absence of F_c and M_y, as per clause 5.2.2 and 5.6
M_{cy}	=	the moment capacity in bending about the local Y axis, in the absence of F_c and M_z,as per clause 5.2.2 and 5.6
M_b	=	the lateral buckling resistance moment as per clause 5.6.2
P_{Ez}	=	the flexural buckling load in compression for bending about the local Z axis
P_{Ey}	=	the flexural buckling load in compression for bending about the local Y axis
C_{bz}, C_{by}	=	taken as unity unless their values are specified by the user

M_{cz}, M_{cy}, and M_b are calculated from clause numbers 5.2.2 and 5.6 in the manner described herein below.

For restrained beams, the applied moment based on factored loads should not be greater then the bending moment resistance of the section, M_c

$$M_{cz} = S_{zz} \text{ x } p_o$$

$$M_{cy} = S_{yy} \text{x } p_o$$

$$p_o = \left(1.13 - 0.0019\frac{D_w}{t}\sqrt{\frac{Y_s}{280}}\right)p_y$$

Where

where

M_{cz}	=	the Moment resistance of the section in z axis
M_{cz}	=	the Moment resistance of the section in z axis
p_o	=	the limiting stress for bending elements under stress gradient and should not greater then design strength p_y

For unrestrained beams the applied moment based on factored loads should not be greater than the smaller of the bending moment resistance of the section , M_c , and the buckling resistance moment of the beam, M_b

Then buckling resistance moment, M_b, may be calculated as follows

$$M_b = \frac{M_E M_y}{\phi_B + \sqrt{\phi_B^2 - M_E M_y}} \leq M_c$$

$$\varphi_B = [M_y + (1 + \eta)M_E]/2$$

where

M_Y	=	the yield moment of the section , product of design strength p_y and elastic modules of the gross section with respect to the compression flange Zc
M_E	=	the elastic lateral buckling resistance as per clause 5.6.2.2
η	=	the Perry coefficient

Please refer clause numbers 5.2.2 and 5.6 of the subject code for a detailed discussion regarding the parameters used in the above mentioned equations.

The maximum shear stress should not be greater then 0.7 ´ p_y as per clause 5.4.2

The average shear stress should not exceed the lesser of the shear yield strength, p_v or the shear buckling strength, q_{cr} as stipulated in clause 5.4.3 of the subject code.

The parameters are calculated as follows:

$$p_v = 0.6 \cdot p_y$$

$$q_{cr} = (1000 \cdot t/D)^2 \text{ N/mm}^2$$

$$P_v = A \cdot \min(p_v,\ q_{cr})$$

where

P_v	=	the shear capacity in N/mm^2
p_y	=	the design strength in N/mm^2
t	=	the web thickness in mm
D	=	the web depth in mm

For beam webs subjected to both bending and shear stresses the member should be designed to satisfy the following relationship as per the stipulations of clause 5.5.2 of the subject code

$$(F_v/P_v)^2 + (M/M_c)^2 \leq 1$$

where

F_v	=	the shear force
M	=	the bending moment acting at the same section as F_v
M_c	=	the moment capacity determined in accordance with 5.2.2

D3.E.4 Design Parameters

The design parameters outlined in Table 2E.1 are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 122: British Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description
CODE	BS5950 COLD	Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	When this parameter is set to 1.0 (default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details. For TRUSS members only start and end locations are designed.
CMZ	1.0	Coefficient of equivalent uniform bending C_b. See BS: 5950-5:1998,5.6. Used for Combined axial load and bending design.
CMY	1.0	Coefficient of equivalent uniform bending C_b. See BS: 5950-5:1998,5.6. Used for Combined axial load and bending design.
CWY	1.0	Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See BS: 5950-5:1998,3.4 0 – effect should not be included 1 – effect should be included

Parameter Name	Default Value	Description
FLX	1	Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See BS:5950-5:1998, 5.6 Values: 0 – Section subject to torsional flexural buckling 1 – Section *not* subject to torsional flexural buckling
FU	430 MPa	Ultimate tensile strength of steel in current units.
FYLD	250 MPa	Yield strength of steel in current units.
KX	1.0	Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
KY	1.0	Effective length factor for overall buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
KZ	1.0	Effective length factor for overall buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.

Parameter Name	Default Value	Description
LX	Member length	Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
LY	Member length	Effective length for overall buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LZ	Member length	Effective length for overall buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
MAIN	0	Specify the design for slenderness against the maximum slenderness as per Clause 6.2.2: 0 – Do not check slenderness ratio 1 – Check members resisting normal loads (180) 2 - Check members resisting self-weight and wind loads (250) 3 - Check members resisting reversal of stress (350)
NSF	1.0	Net section factor for tension members

Parameter Name	Default Value	Description
DMAX	2540.0 cm.	Maximum allowable depth. It is input in the current units of length.
RATIO	1.0	Permissible ratio of actual to allowable stresses
TRACK	0	This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 0 - Prints only the member number, section name, ratio, and PASS/FAIL status. 1 - Prints the design summary in addition to that printed by TRACK 1 2 - Prints member and material properties in addition to that printed by TRACK 2.

D4. Canadian Codes

D4.A. Canadian Codes - Concrete Design per CSA Standard A23.3-94

STAAD.Pro is capable of performing concrete design based on the Candadian code CSA A23.3 1994 *Design of Concrete Structures*. Given the width and depth (or diameter for circular columns) of a section, the program will calculate the required reinforcement to resist the forces and moments.

D4.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams - Prismatic (Rectangular, Square & Tee)
- For Columns - Prismatic (Rectangular, Square and Circular)
- For Slabs - 4-noded Plate Elements

D4.A.2 Member Dimensions

Concrete members that are to be designed by STAAD must have certain section properties input under the `MEMBER PROPERTIES` command. The following example demonstrates the required input:

```
UNIT  MM
MEMBER  PROPERTIES
1 3  TO 7 9 PRISM YD 450. ZD 300.
11 14  PR YD 300.
```

In the above input, the first set of members are rectangular (450mm depth and 300mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with a 300mm diameter

D4.A.3 Slenderness Effects and Analysis Considerations

STAAD.Pro provides the user with two methods of accounting for the slenderness effect in the analysis and design of concrete members. The first method is equivalent to the procedure presented in CSA Standard A23.3-94 Clause 10.13. STAAD.Pro accounts for the secondary moments, due to axial loads and deflections, when the `PDELTA ANALYSIS` command is used. After solving for the joint displacements of the structure, the program calculates the additional moments induced in the structure due to the P-Delta effect. Therefore, by performing a P-Delta analysis, member forces are calculated which will require no user modification before beginning member design. Refer to TR.37.2 P-Delta Analysis Options (on page 2621) for additional details on this analysis facility.

The second method by which STAAD allows the user to account for the slenderness effect is through user supplied moment magnification factors (see the parameter `MMAG` in D4.A.4 Design Parameters (on page 1414)). Here the user approximates the additional moment by supplying a factor by which moments will be multiplied before beginning member design. This second procedure allows slenderness to be considered in accordance with Clause 10.14 of the code.

Note: STAAD.Pro does not factor loads automatically for concrete design. All the proper factored loads must be provided by the user before the `ANALYSIS` specification.

While performing a P-Delta analysis, all load cases must be defined as primary load cases. If the effects of separate load cases are to be combined, it should be done either by using the `REPEAT LOAD` command or by specifying the load information of these individual loading cases under one single load case. Usage of the `LOAD COMBINATION` command will yield incorrect results for P-Delta Analysis in STAAD.Pro.

D4.A.4 Design Parameters

The program contains a number of parameters which are needed to perform design per CSA Standard A23.3-94. These parameters not only act as a method to input required data for code calculations but give the engineer control over the actual design process.

Default values, which are commonly used numbers in conventional design practice, have been used for simplicity. The following table contains a list of available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 123: Canadian Concrete Design CSA-A23.3-94 Parameters

Parameter Name	Default Value	Description
CLB	40	Clear cover to reinforcing bar at bottom of cross section, in mm.
CLS	40	Clear cover to reinforcing bar along the side of the cross section, in mm.

Parameter Name	Default Value	Description
CLT	40	Clear cover to reinforcing bar at top of cross section, in mm.
DEPTH	YD	Depth of the concrete member. This value defaults to YD as provided in the `MEMBER PROPERTIES`.
EFACE	0.0 (Face of Support)	Distance of face of support from end node of beam. Used for shear and torsion calculation. **Note:** Both `SFACE` & `EFACE` must be positive numbers.
FC	30	Specified compressive strength of concrete, in N/mm^2.
FYMAIN	400	Yield Stress for main reinforcing steel, in N/mm^2.
FYSEC	400	Yield Stress for secondary reinforcing steel, in N/mm^2.
MAXMAIN	7 (Number 55 bar)	Maximum main reinforcement bar size. The parameter values for metric bar sizes are: 0) 10 mm 1) 15 mm 2) 20 mm 3) 25 mm 4) 30 mm 5) 35 mm 6) 45 mm 7) 55 mm

Parameter Name	Default Value	Description
MINMAIN	0 (Number 10 bar)	Minimum main reinforcement bar size. The parameter values for metric bar sizes are: 0) 10 mm 1) 15 mm 2) 20 mm 3) 25 mm 4) 30 mm 5) 35 mm 6) 45 mm 7) 55 mm
MINSEC	0 (Number 10 bar)	Minimum secondary (stirrup) reinforcement bar size. The parameter values for metric bar sizes are: 0) 10 mm 1) 15 mm 2) 20 mm 3) 25 mm 4) 30 mm 5) 35 mm 6) 45 mm 7) 55 mm
MMAG	1.0	A factor by which the column design moments will be magnified.
NSECTION	12	Number of equally-spaced sections to be considered in finding critical moments for beam design.
REINF	0	Column comfining reinforcement type: 0) Tied column 1) Spiral reinforcement

Parameter Name	Default Value	Description
SFACE	0.0	Distance of face of support from start node of beam. Used for shear and torsion calculation. **Note:** Both SFACE & EFACE must be positive numbers.
TRACK	0	Specify the level of detail in the output: 0) Critical Moment will not be printed out with beam design report. 1) Moments will be printed.
WIDTH	ZD	Width of the concrete member. This value defaults to ZD as provided in the MEMBER PROPERTIES.

D4.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are scanned to create moment and shear envelopes, and locate critical sections. The total number of sections considered is thirteen (start, end, and 11 intermediate), unless that number is redefined with the NSECTION parameter.

D4.A.5.1 Design for Flexure

Design for flexure is performed per the rules of Chapter 10 of CSA Standard A23.3-94. Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the thirteen sections. Each of these sections are designed to resist the critical sagging and hogging moments. Currently, design of singly reinforced sections only is permitted. If the section dimensions are inadequate as a singly reinforced section, such a message will be printed in the output. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account the changed effective depths of sections calculated on the basis of reinforcement provided after the preliminary design. Final provision of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per CSA Standard A23.3-94. Although exact curtailment lengths are not mentioned explicitly in the design output (which finally will be more or less guided by the detailer taking into account other practical considerations), the user has the choice of printing reinforcements provided by STAAD at 13 equally spaced sections from which the final detailed drawing can be prepared.

The following annotations apply to the output for Beam Design.

LEVEL	Serial number of bar level which may contain one or more bar group.
HEIGHT	Height of bar level from the bottom of beam.
BAR INFOrmation	Reinforcement bar information specifying number of bars and size.

FROM	Distance from the start of the beam to the start of the rebar.
TO	Distance from the start of the beam to the end of the rebar.
ANCHOR(STA,END)	States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end (END) of the bar.

D4.A.5.2 Design for Shear and Torsion

Design for shear and torsion is performed per the rules of Chapter 11 of CSA Standard A23.3-94. Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design is performed at the start and end sections. The location along the member span for design is chosen as the effective depth + SFACE at the start, and effective depth + EFACE at the end. The load case which gives rise to the highest stirrup area for shear & torsion is chosen as the critical one. The calculations are performed assuming 2-legged stirrups will be provided. The additional longitudinal steel area required for torsion is reported.

The stirrups are assumed to be U-shaped for beams with no torsion, and closed hoops for beams subjected to torsion.

D4.A.5.3 Example of Input

Example of Input Data for Beam Design

```
UNIT  NEWTON MMS
START  CONCRETE DESIGN
CODE  CANADA
FYMAIN  415 ALL
FYSEC  415 ALL
FC 35  ALL
CLEAR  25 MEMB 2 TO 6
MAXMAIN  40 MEMB 2 TO 6
TRACK  1.0 MEMB 2 TO 9
DESIGN  BEAM 2 TO 9
END  CONCRETE DESIGN
```

D4.A.6 Column Design

Column design is performed per the rules of Chapters 7 & 8 of the CSA Standard A23.3-94. Columns are designed for axial force and biaxial moments at the ends. All active loadings are tested to calculate reinforcement. The loading which produces maximum reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and square sections, the reinforcement is always assumed to be equally distributed on each side. That means the total number of bars will always be a multiple of four (4). This may cause slightly conservative results in some cases.

Example of Input Data for Column Design

```
UNIT  NEWTON MMS
START  CONCRETE DESIGN
CODE  CANADIAN
FYMAIN  415 ALL
FC 35  ALL
CLEAR  25 MEMB 2 TO 6
MAXMAIN  40 MEMB 2 TO 6
```

```
DESIGN  COLUMN 2 TO 6
END  CONCRETE DESIGN
```

D4.A.7 Slab and Wall Design

To design a slab or wall, it must be modeled using finite elements. The commands for specifying elements are in accordance with the relevant sections of the Technical Reference Manual.

Elements are designed for the moments Mx and My using the same principles as those for beams in flexure. The width of the beam is assumed to be unity for this purpose. These moments are obtained from the element force output. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement. The effective depth is calculated assuming #10 bars are provided. The parameters `FYMAIN`, `FC`, `CLT`, and `CLB` listed in D4.A.4 Design Parameters (on page 1414) are relevant to slab design. Other parameters mentioned in Table 3A.1 are not applicable to slab design. The output consists only of area of steel required. Actual bar arrangement is not calculated because an element most likely represents just a fraction of the total slab area.

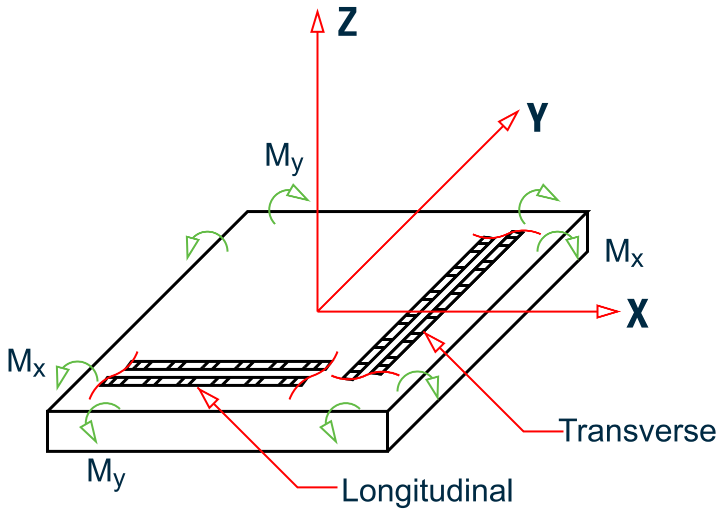


Figure 162: Element moments: Longitudinal (L) and Transverse (T)

Example of Input Data for Slab/Wall Design

```
UNIT  NEWTON MMS
START  CONCRETE DESIGN
CODE  CANADA
FYMAIN  415 ALL
FC 35  ALL
CLB  40 ALL
DESIGN  ELEMENT 15 TO 20
END  CONCRETE DESIGN
```

D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01

STAAD.Pro is capable of performing steel design based on the Canadian code CAN/CSA-S16-01 *Limit States Design of Steel Structures*.

Related Links

- *V. CSA S16-01 - Axial Tension* (on page 3846)
- *V. CSA S16-01 - Beam Shear Capacity* (on page 3850)
- *V. CSA S16-01 - Cantilever with Biaxial Loading* (on page 3852)
- *V. CSA S16-01 - Shear Capacity Combined Stresses* (on page 3858)
- *V. CSA S16-01 - Short Column Compression* (on page 3862)
- *V. CSA S16-01 - Slender Column Compression* (on page 3864)
- *V. CSA S16-01 - Wide Flange Beam Interaction Ratio 1* (on page 3867)
- *V. CSA S16-01 - Wide Flange Beam Interaction Ratio 2* (on page 3871)
- *V. CSA S16-01 - Wide Flange Beam Interaction Ratio 3* (on page 3874)

D4.B.1 General Comments

The design of structural steel members in accordance with the specification CAN/CSA S16-01 Limit States Design of Steel Structures is can be used in STAAD.Pro. This code supercedes the previous edition of the code CAN/CSA – S16.1-94.

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-states are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the probability of limits being surpassed is acceptably low.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of CAN/CSA-S16-01. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D4.B.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the specified primary and combination loading condition. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D4.B.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to the STAAD.Pro Technical Reference Manual.

D4.B.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all Canadian steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following is the description of the different types of sections available:

D4.B.4.1 Welded Wide Flanges (WW shapes)

Welded wide flange shapes listed in the CSA steel tables can be designated using the same scheme used by CSA. The following example illustrates the specification of welded wide flange shapes.

```
100 TO 150 TA ST WW400X444
34 35 TA ST WW900X347
```

D4.B.4.2 Wide Flanges (W shapes)

Designation of wide flanges in STAAD is the same as that in CSA tables. For example,

```
10 TO 75 95 TO 105 TA ST W460X106
100 TO 200 TA ST W610X101
```

D4.B.4.3 S, M, HP shapes

In addition to welded wide flanges and regular wide flanges, other I shaped sections like S, M and HP shapes are also available. The designation scheme is identical to that listed in the CSA tables. While specifying the sections, it should be remembered that the portion after the decimal point should be omitted. Thus, M310X17.6 should be specified as M310X17 and S180X22.8 should be specified as S180X22. Examples illustrating specifications of these shapes are provided below.

```
10 TO 20 BY 2 TA ST S510X98
45 TO 55 TA ST M150X6
88 90 96 TA ST HP310X79
```

D4.B.4.4 Channel Sections (C & MC shapes)

C and MC shapes are designated as shown in the following example. As in S, M and HP sections, the portion after the decimal point must be omitted in section designations. Thus, MC250X42.4 should be designated as MC250X42.

```
55 TO 90 TA ST C250X30
30 TO 45 TA ST MC200X33
```

D4.B.4.5 Double Channels

Back-to-back double channels, with or without spacing between them, are specified by preceding the section designation by the letter D. For example, a back-to-back double channel section C200X28 without any spacing in between should be specified as:

```
100 TO 120 TA D C200X28
```

If a spacing of 2.5 length units is used, the specification should be as follows:

```
100 TO 120 TA D C200X28 SP 2.5
```

Note that the specification SP after the section designation is used for providing the spacing. The spacing should always be provided in the current length unit.

D4.B.4.6 Angles

To specify angles, the angle name is preceded by the letter L. Thus, a 200X200 angle with a 25mm thickness is designated as L200X200X25. The following examples illustrate angle specifications.

```
75 TO 95 TA ST L100X100X8
33 34 35 TA ST L200X100X20
```

Note: This specification is for “standard” angles, designated by ST. In this specification, the local z-axis corresponds to the Y’-Y’ axis shown in the section table. Another common practice of specifying angles assumes the local y-axis to correspond to the Y’-Y’ axis. To specify angles in accordance with this convention, use the reverse angle designation, RA. Refer to G.4.2 Local Coordinate System (on page 2088) for details on the local axis systems for standard (ST) and reverse (RA) angles.

Refer to the following example for details.

```
10 TO 15 TA RA L55X35X4
```

D4.B.4.7 Double Angles

To specify double angles, the specification ST should be substituted with LD (for long leg back-to-back) or SD (short leg back-to-back). For equal angles, either SD or LD will serve the purpose. Spacing between angles may be provided by using the word SP followed by the value of spacing (in current length unit) after section designation.

```
25 35 45 TA LD L150X100X16
80 TO 90 TA SD L125X75X6 SP 2.5
```

The second example above describes a double angle section consisting of 125X75X6 angles with a spacing of 2.5 length units.

D4.B.4.8 Tees

Tee sections obtained by cutting W sections may be specified by using the T specification instead of ST before the name of the W shape. For example:

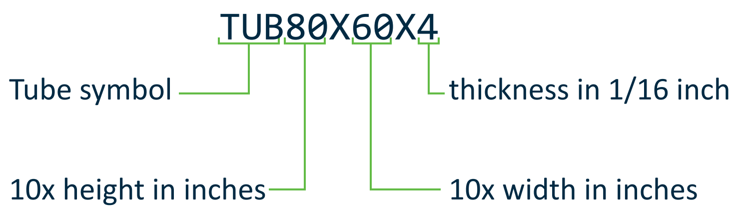
```
100 TO 120 TA T W200X42
```

will describe a T section cut from a W200X42 section.

D4.B.4.9 Rectangular Hollow Sections

These sections may be specified in two possible ways. Those sections listed in the CSA tables may be specified as follows.

```
55 TO 75 TA ST TUB80X60X4
```



In addition, any tube section may be specified by using the DT(for depth), WT(for width), and TH(for thickness) specifications.

For example:

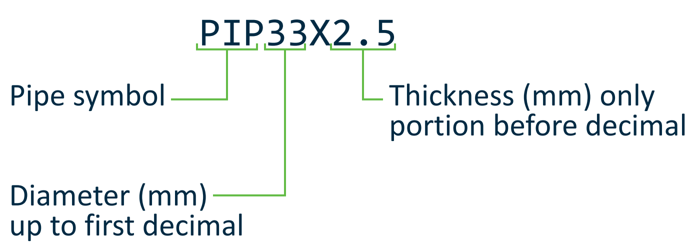
```
100 TO 200 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

will describe a tube with a depth of 8 in., width of 6 in. and a wall thickness of 0.5 inches. Note that the values of depth, width and thickness must be provided in current length unit.

D4.B.4.10 Circular Hollow Sections

Sections listed in the CSA tables may be provided as follows:

```
15 TO 25 TA ST PIP33X2.5
```



In addition to sections listed in the CSA tables, circular hollow sections may be specified by using the OD (outside diameter) and ID (inside diameter) specifications. For example:

```
70 TO 90 TA ST PIPE OD 10.0 ID 9.0
```

will describe a pipe with an outside diameter of 10 length units and inside diameter of 9.0 length units. Note that the values of outside and inside diameters must be provided in terms of current length unit.

D4.B.4.11 Example

Sample input file to demonstrate usage of Canadian shapes

```
STAAD SPACE
UNIT METER KNS
JOINT COORD
1 0 0 0 17 160 0 0
MEMBER INCIDENCES
1 1 2 16
UNIT CM
MEMBER PROPERTIES CANADIAN
*  W SHAPES
1 TA ST W250X18
*  WW SHAPES
2 TA ST WW700X185
*  S SHAPES
```

```
3 TA ST S200X27
*  M SHAPES
4 TA ST M130X28
*  HP SHAPES
5 TA ST HP310X132
*  MC CHANNELS
6 TA ST MC150X17
*  C CHANNELS
7 TA ST C180X18
* DOUBLE CHANNELS
8 TA D C250X37 SP 1.0
*  ANGLES
9 TA ST L55X35X5
*  REVERSE ANGLES
10 TA RA L90X75X5
*  DOUBLE ANGLES, LONG LEG BACK TO BACK
11 TA LD L100X90X6 SP 2.0
*  DOUBLE ANGLES, SHORT LEG BACK TO BACK
12 TA SD L125X75X6 SP 2.5
*  TUBES
13 TA ST TUB120807
*  TUBES
14 TA ST TUBE DT 16.0 WT 8.0 TH 0.8
*  PIPES
15 TA ST PIP273X6.3
*  PIPES
16 TA ST PIPE OD 16.0 ID 13.0
PRINT MEMBER PROPERTIES
FINISH
```

D4.B.5 Section Classification

The CSA specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of CAN/CSA-S16-01). This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes.

Note: The design of Class 4 sections requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or higher. Otherwise, design is performed for sections that fall into the category of Class 1,2 or 3 sections only.

D4.B.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

Note: The design of Class 4 sections requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or higher.

D4.B.6.1 Nomenclature

where

A	=	Area.
A_e	=	Effective area.
A_f	=	Area of flange.
A_w	=	Area of web.
b_e	=	Effective Flange width.
C_f	=	Compressive force in a member or component under factored load.
C_r	=	Factored compressive resistance.
C_w	=	Warping torsional constant.
C_y	=	Axial compressive load at yield stress.
D	=	Outside diameter of pipe section.
E	=	Elastic modulus of steel.
F_e	=	Elastic critical buckling stress.
F_y	=	Yield strength.
F_{ye}	=	Effective yield stress of section in compression to account for elastic local buckling.
h	=	Clear depth of web.
K	=	Effective length factor.
L	=	Length or span of member.
M_f	=	Bending moment in a member or component under factored load.
M_r	=	Factored moment resistance of a member.
M_y	=	Yield moment resistance.
S	=	Elastic section modulus.
S_e	=	Effective section modulus.
W	=	Web thickness.
λ	=	Non-dimensional slenderness parameter in column formula.
λ_{ye}	=	Effective non-dimensional slenderness parameter in column formula considering effective yield stress.
ϕ	=	Resistance factor

D4.B.6.2 Members Subject to Axial Forces

Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on these two limits states per Cl.13.2 of CAN/CSA-S16-01. Parameters `FYLD`, `FU`, and `NSF` are applicable for these calculations.

Axial Compression

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios). The effective length for the calculation of compression resistance may be provided through the use of the parameters `KT`, `KY`, `KZ`, `LT`, `LY`, and `LZ` (see Table 3B.1). Some of the aspects of the axial compression capacity calculations are :

1. For frame members not subjected to any bending, and for truss members, the axial compression capacity in general column flexural buckling is calculated from Cl.13.3.1 using the slenderness ratios for the local Y-Y and Z-Z axis. The parameters KY, LY, KZ and LZ are applicable for this.
2. For single angles, which are frame members not subjected to any bending or truss members, the axial compression capacity in general column flexural buckling and local buckling of thin legs is calculated using the rules of the AISC - LRFD code, 2^{nd} ed., 1994. The reason for this is that the Canadian code doesn't provide any clear guidelines for calculating this value. The parameters KY, LY, KZ, and LZ are applicable for this.
3. The axial compression capacity is also calculated by taking flexural-torsional buckling into account. The rules of Appendix D, page 1-109 of CAN/CSA-S16-01are used for this purpose. Parameters KT and LT may be used to provide the effective length factor and effective length value for flexural-torsional buckling. Flexural-torsional buckling capacity is computed for single channels, single angles, Tees and Double angles.
4. The variable "n" in Cl.13.3.1 is assumed as 2.24 for WWF shapes and 1.34 for all other shapes.
5. While computing the general column flexural buckling capacity of sections with axial compression + bending, the special provisions of 13.8.1(a), 13.8.1(b) and 13.8.1(c) are applied. For example, Lambda = 0 for 13.8.1(a), K=1 for 13.8.1(b), etc.)

For Class 4 members subjected to axial compression, factored compressive resistance should be determined by either of the following equations.

a. $C_r = \phi A_e F_y (1+\lambda^{2n})^{-1/n}$

where

n = 1.34

λ = $\sqrt{(F_y/F_e)}$

F_e = $(\pi^2 E)/(KL/r)^2$

Ae is calculated using reduced element widths meeting the maximum width to thickness ratio specified in Table 1.

Effective width required for the calculation of effective area Ae, for different section shapes are as follows.

- For flanges of I-section, T-section and channel section and legs of angle section

  $b_e = 200t/\sqrt{(F_y)}$
- For stem of T-section

  $b_e = 340t/\sqrt{(F_y)}$
- For flanges of HSS rectangular or Tube sections

  $b_e = 670t/\sqrt{((F_y)}$
- For circular HSS or Pipe section

  $D = 23000t/(F_y$

b. $C_r = \phi A F_{ye} (1+\lambda_{ye}{}^{2n})^{-1/n}$

where

n = 1.34

λ_{ye} = $\sqrt{(F_{ye}/F\_e)}$

F_e = $(\pi^2 E)/(KL/r)^2$

With an effective yield stress, F_{ye}, determined from the maximum width (or diameter)-to-thickness ratio meeting the limit specified in Table 1.

Following are the expressions for effective yield stress for different shaped section.

- For I-section, T-section, channel section and angle section

  $F_{ye} = 40000/(b/t)^2$

- For rectangular HSS section

  $F_{ye}= 448900/(b/t)^2$
- For circular HSS section

  $F_{ye}= 23000/(D/t)$

D4.B.6.3 Members Subject to Bending

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD with the help of the parameter UNL. If UNL is less than one tenth the member length (member length is the distance between the joints of the member), the member is treated as being continuously laterally supported. In this case, the moment resistance is computed from Clause 13.5 of the code. If UNL is greater than or equal to one tenth the member length, its value is used as the laterally unsupported length. The equations of Clause 13.6 of the code are used to arrive at the moment of resistance of laterally unsupported members. Some of the aspects of the bending capacity calculations are :

1. The weak axis bending capacity of all sections except single angles is calculated as

   For Class 1 & 2 sections, $\varphi \cdot P_y \cdot F_y$

   For Class 3 sections, $\varphi \cdot S_y \cdot F_y$

   where

φ	=	Resistance factor = 0.9
P_y	=	Plastic section modulus about the local Y axis
S_y	=	Elastic section modulus about the local Y axis
F_y	=	Yield stress of steel
2. For single angles, the bending capacities are calculated for the principal axes. The specifications of Section 5, page 6-283 of AISC-LRFD 1994, 2nd ed., are used for this purpose because the Canadian code doesn't provide any clear guidelines for calculating this value.
3. For calculating the bending capacity about the Z-Z axis of singly symmetric shapes such as Tees and Double angles, CAN/CSA-S16-01 stipulates in Clause 13.6(d), page 1-31, that a rational method, such as that given in SSRC's Guide to Stability Design Criteria of Metal Structures, be used. Instead, STAAD uses the rules of Section 2c, page 6-55 of AISC-LRFD 1994, 2nd ed.

Laterally Supported Class 4 members subjected to bending

i. When both the web and compressive flange exceed the limits for Class 3 sections, the member should be considered as failed and an error message will be thrown.
ii. When flanges meet the requirements of Class 3 but web exceeds the limits for Class 3, resisting moment shall be determined by the following equation.

$$M'_r = M_r\left[1 - 0.0005\frac{A_w}{A_f}\left(\frac{h}{w} - \frac{1,900}{\sqrt{M_f/\phi_s}}\right)\right]$$

Where Mr = factored moment resistance as determined by Clause 13.5 or 13.6 but not to exceed ϕMy = factored moment resistance for Class 3 sections = ϕMy

If axial compressive force is present in addition to the moment, modified moment resistance should be as follows.

$$M'_r = M_r\left\{1 - 0.0005\frac{A_w}{A_f}\left[\frac{h}{w} - 1,900\frac{1 - 0.65C_f/(\phi C_y)}{\sqrt{M_f/\phi_s}}\right]\right\}$$

$C_y = A \cdot F_y$

S = Elastic section modulus of steel section.

iii. For sections whose webs meet the requirements of Class 3 and whose flanges exceed the limit of Class 3, the moment resistance shall be calculated as

$$M_r = \phi \cdot S_e \cdot F_y$$

Where:

S_e = effective section modulus determined using effective flange width.

- For Rectangular HSS section, effective flange width

$$b_e = 670 \cdot t/\sqrt{(F_y)}$$

- For I-section, T-section, Channel section, effective flange width and for Angle section, effective length width

$$b_e = 200 \cdot t/\sqrt{(F_y)}$$

But shall not exceed $60 \cdot t$

Laterally Unsupported Class 4 members subjected to bending

As per clause 13.6(b) the moment resistance for class-4 section shall be calculated as follows

i. When $M_u > 0.67M_y$

$$M_r = 1.15\phi M_y\left(1 - \frac{0.28M_y}{M_u}\right)$$

M_r should not exceed $\phi S_e F_y$

ii. When $M_u \leq 0.67M_y$

$$M_r = \phi M_u$$

Where, as per clause 13.6(a),

$$M_u = (\omega_2 \pi)/L \sqrt{(EI_y GJ + (\pi E/L)^2 I_y C_w)}$$

For unbraced length subjected to end moments-

$$\omega_2 = 1.75 + 1.05k + 0.3k^2 \leq 2.5$$

When bending moment at any point within the unbraced length is larger than the larger end moment or when there is no effective lateral support for the compression flange at one of the ends of unsupported length-

$$\omega_2 = 1.0$$

k = Ratio of the smaller factored moment to the larger moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.

Se = effective section modulus determined using effective flange width.

- For Rectangular HSS section, effective flange width

$$b_e = 670t/\sqrt{(F_y)}$$

- For I-section, T-section, Channel section, effective flange width and for Angle section, effective length width

$$b_e = 200t/\sqrt{(F_y)}$$

But shall not exceed 60t.

This clause is applicable only for I shaped and Channel shaped section as there is no guide line in the code for other sections.

D4.B.6.4 Members Subject to Combined Forces

Axial compression and bending

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors. Clause 13.8 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceed 1.0 or the allowable value provided using the `RATIO` parameter (Refer to D4.B.7 Design Parameters (on page 1429)), the member is considered to have failed under the loading condition.

Axial tension and bending

Members subjected to axial tension and bending are also designed using interaction equations. Clause 13.9 of the code is used to perform these checks. The actual `RATIO` is determined as the value of the left hand side of the critical equation.

D4.B.6.5 Shear

The shear resistance of the cross section is determined using the equations of Clause 13.4 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the `RATIO` parameter (see Table 3B.1), the section is considered to have failed under shear. The code also requires that the slenderness ratio of the web be within a certain limit (See Cl.13.4.1.3, page 1-29 of CAN/CSA-S16-01). Checks for safety in shear are performed only if this value is within the allowable limit. Users may by-pass this limitation by specifying a value of 2.0 for the `MAIN` parameter.

D4.B.7 Design Parameters

The design parameters outlined in Table 3B.1 may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 124: Canadian Steel Design CSA-S16-01 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `CANADIAN 2001`. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	1.0	0.0 = design only for end moments and those at locations specified by SECTION command. 1.0 = Perform design for moments at twelfth points along the beam.
CB	1.0	Greater than 0.0 and less than 2.5 : Value of Omega_2 (Cl.13.6) to be used for calculation. Equal to 0.0 : Calculate Omega_2
CMY	1.0	1.0 = Do not calculate Omega-1 for local Y axis. 2.0 = Calculate Omega-1 for local Y axis. Used in Cl.13.8.4 of code
CMZ	1.0	1.0 = Do not calculate Omega-1 for local Z axis. 2.0 = Calculate Omega-1 for local Z axis. Used in Cl.13.8.4 of code
DFF	None(Mandatory for deflection check)	"Deflection Length"/Maxm. Allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting start point for calculation of "deflection length"
DJ2	End Joint of member	Joint No. denoting end point for calculation of "deflection length"
DMAX	45.0 in.	Maximum allowable depth (Applicable for member selection)
DMIN	0.0 in.	Minimum required depth (Applicable for member selection)
FU	345.0 MPa	Ultimate strength of steel.
FYLD	300.0 MPa	Yield strength of steel.

Parameter Name	Default Value	Description
KT	1.0	K value for flexural torsional buckling.
KY	1.0	K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.
KZ	1.0	K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
LT	Member Length	Length for flexural torsional buckling.
LY	Member Length	Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.
LZ	Member Length	Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
MAIN	0.0	0.0 = Check slenderness ratio against the limits. 1.0= Suppress the slenderness ratio check. 2.0 = Check slenderness ratio only for column buckling, not for web (See Section 3B.6, Shear)
NSF	1.0	Net section factor for tension members.
RATIO	1.0	Permissible ratio of actual load effect to the design strength.
SHEAR	1	Shear stress calculation option. 0) = compute the actual shear stress using VQ/Ib 1) = compute the actual shear stress using V/(Ay or Az)

Parameter Name	Default Value	Description
SSY	0	0 = Sway or Unbraced along local Y axis 1 = Braced along local Y axis This parameter is required to choose the proper value of U_{1y} which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength.
SSZ	0	0 = Sway or Unbraced along local Z axis 1 = Braced along local Z axis This parameter is required to choose the proper value of U_{1z} which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength.
TRACK	0.0	0.0 = Report only minimum design results. 1.0 = Report design strengths also. 2.0 = Provide full details of design.
UNB	Member Length	Unsupported length in bending compression of the bottom flange for calculating moment resistance.
UNT	Member Length	Unsupported length in bending compression of the top flange for calculating moment resistance.

D4.B.8 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per the CAN/CSA-S16-01 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1, moments are calculated at every twelfth point along the beam. When no sections are specified and the BEAM parameter is set to zero (default), design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. The extent of detail of the output can be controlled by using the TRACK parameter.

Example of commands for CODE CHECKING:

```
UNIT NEWTON METER
PARAMETER
CODE CANADIAN
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
UNL 15 MEMB 3 4
RATIO 0.9 ALL
CHECK CODE MEMB 3 4
```

D4.B.9 Member Selection

The member selection process basically involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on TUBES, PIPES or members listed as PRISMATIC.

Example of commands for MEMBER SELECTION:

```
UNIT NEWTON METER
PARAMETER
FYLD 330E6 MEMB 3 4
NSF 0.85 ALL
KY 1.2 MEMB 3 4
UNL 15 MEMB 3 4
RATIO 0.9 ALL
SELECT MEMB 3 4
```

D4.B.10 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the CAN/CSA-S16-01 specification which governed the design.

If the TRACK parameter is set to 1.0, factored member resistances will be printed. Following is a description of some of the items printed.

CR	Factored compressive resistance
TR	Factored tensile resistance
VR	Factored shear resistance
MRZ	Factored moment resistance (about z-axis)
MRY	Factored moment resistance (about y-axis)

Further details can be obtained by setting TRACK to 2.0.

CR1	CAPACITY (C_r) PER 13.8.2(a)

CR2	CAPACITY (C_r) PER 13.8.2(b)
CRZ	SEE 13.8.2(b) for uniaxial bending (called C_{RX} in that Clause)
CTORFLX	Capacity in accordance with 13.8.2(c)

D4.C. Canadian Codes - Cold Formed Steel Design per S136-94

STAAD.Pro is capable of performing steel design based on the Canadian code S136-94 *Specification for the Design of Cold-Formed Steel Structural Members*, including revisions dated May, 1995. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. For laterally supported members in bending, the Initiation of Yielding method has been used. Cold work of forming strengthening effects have been included as an option.

D4.C.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables published in the "Cold-Formed Steel Design Manual", AISI, 1996 Edition.

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Angle with Lips
- Angle without Lips
- Z with Lips
- Z without Lips
- Hat

Shape selection may be done using the member property pages of the graphical user interface (GUI) or by specifying the section designation symbol in the input file.

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

D4.C.2 Design Procedure

The following two design modes are available:

D4.C.2.1 Code Checking

The program compares the resistance of members with the applied load effects, in accordance with CSA 136. Code checking is carried out for locations specified via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. You may choose the degree of detail in the output data by setting the TRACK parameter.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D4.C.2.2 Member Selection

You may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D4.C.2.3 Code Sections Implemented

The program calculates effective section properties in accordance with Clauses 5.6.2.1 through 3 and 5.6.2.6 through 8. Cross-sectional properties and overall slenderness of members are checked for compliance with

- Clause 5.3, Maximum Effective Slenderness Ratio for members in Compression
- Clause 5.4, Maximum Flat Width Ratios for Elements in Compression
- Clause 5.5, Maximum Section Depths.

The program will check member strength in accordance with Clause 6 of the Standard as follows:

- Resistance factors listed in Clauses 6.2 (a), (b), and (e) are used, as applicable.
- Members in tension - Resistance is calculated in accordance with Clauses 6.3.1 and 6.3.2.
- Members in bending and shear

Resistance calculations are based on Clauses:

- 6.4.1 General,
- 6.4.2 and 6.4.2.1 Laterally Supported Members, compressive limit stress based on Initiation of Yielding,
- 6.4.3 Laterally Unsupported Members,
- 6.4.4 Channels and Z-Shaped Members with Unstiffened Flanges - additional limitations,
- 6.4.5 Shear in Webs,
- 6.4.6 Combined Bending and Shear in Webs.
- Members in compression

  Resistance calculations are based on Clauses:

  - 6.6.1.1, 6.6.1.2 (a) and (d), and 6.6.1.3 General,
  - 6.6.2 Sections Not Subject to Torsional-Flexural Buckling,
  - 6.6.3 Singly Symmetric Sections,
  - 6.6.4 Point-Symmetric Sections,
  - 6.6.5 Cylindrical Tubular Sections.
- Members in compression and bending

  Resistance calculations are based on Clause 6.7.1, Singly and Doubly Symmetric Sections. Input for the coefficients of uniform bending must be provided.

D4.C.3 Design Parameters

The following table contains the input parameters for specifying values of design variables and selection of design options.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 125: Canadian Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified S136. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	When this parameter is set to 1.0 (default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details). If neither the BEAM parameter nor any SECTION command is specified, STAAD will terminate the run and ask the user to provide one of those 2 commands. This rule is not enforced for TRUSS members.
CMZ	1.0	Coefficient of equivalent uniform bending w_z. See CSA 136, 6.7.2. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.
CMY	0.0	Coefficient of equivalent uniform bending w_y. See CSA 136, 6.7.2. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.
CWY	0	Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See CSA 136, 5.2. 0. effect should not be included 1. effect should be included

Parameter Name	Default Value	Description
DMAX	1000.0	Maximum depth permissible for the section during member selection. This value must be provided in the current units.
DMIN	0.0	Minimum depth required for the section during member selection. This value must be provided in the current units.
FLX	1	Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See CSA 136, 6.6.2 0. Section subject to torsional flexural buckling and restraint not provided 1. restraint provided or unnecessary
FU	450 MPa	Ultimate tensile strength of steel in current units.
FYLD	350 MPa	Yield strength of steel in current units.
KT	1.0	Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
KY	1.0	Effective length factor for overall column buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.

Parameter Name	Default Value	Description
KZ	1.0	Effective length factor for overall column buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LT	Member length	Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
LY	Member length	Effective length for overall column buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LZ	Member length	Effective length for overall column buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
NSF	1.0	Net section factor for tension members, See CSA 136, 6.3.1.
STIFF	Member length	Spacing in the longitudinal direction of shear stiffeners for stiffened flat webs. It is input in the current units of length. See section CSA 136, 6.4.5

Parameter Name	Default Value	Description
TRACK	0	This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 0. Prints only the member number, section name, ratio, and PASS/FAIL status. 1. Prints the design summary in addition to that printed by TRACK 1 2. Prints member and material properties in addition to that printed by TRACK 2.
TSA	1	Specifies whether bearing and intermediate transverse stiffeners satisfy the requirements of CSA 136, 6.5. If true, the program uses the more liberal set of interaction equations in 6.4.6. 0. stiffeners do not comply with 6.5 1. stiffeners comply with 6.5

D4.D. Canadian Codes - Timber Design per CAN/CSA-086-01

STAAD.Pro is capable of performing timber design based on the Canadian code CSA 086-01 *Wood Design Standard*.

D4.D.1 General Comments

The design philosophy of this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for the entire structure under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation, the code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD implementation of CSA086-01. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D4.D.2 Analysis Methodology

Analysis is done for the primary and combination loading conditions provided by the user. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D4.D.3 Member Property Specifications

A timber section library consisting of Sawn and Glulam timber is available for member property specification.

For specification of member properties, for Sawn timber the timber section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in timber table.

For Glulam timber, member properties can be specified using the YD (depth) and ZD (width) specifications and selecting Combination and Species specifications from the built-in table. The assignment is done with the help of the `PRISMATIC` option (Refer to TR.20.2 Prismatic Property Specification (on page 2263))

D4.D.4 Built-in Timber Section Library

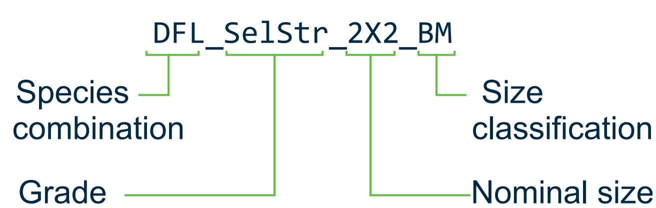
The following information is provided for use when the built-in timber tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design.

Following are the description of the different types of species combination available:

D4.D.4.1 Douglas Fir-Larch

The following example illustrates the specification of Douglas Fir-Larch species combination.

```
100 TO 150 TABLE ST DFL_SelStr_2X2_BM
```



D4.D.4.2 Hem-Fir

Designation of Hem-Fir species combination in STAAD is as follows.

```
100 TO 150 TABLE ST Hem-Fir_SelStr_2X10_BM
```

D4.D.4.3 Northern Species

Designation of Northern species combination in STAAD is as follows.

```
100 TO 150 TABLE ST Northern_SelStr_3X12_BM
```

D4.D.4.4 Spruce-Pine-Fir

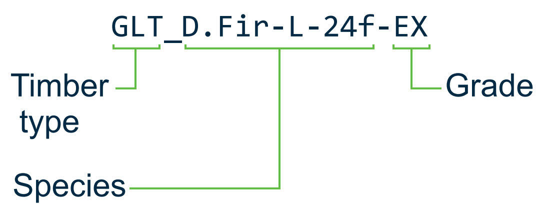
Designation of Spruce-Pine-Fir species combination in STAAD is as follows.

```
100 TO 150 TABLE ST SPF_SelStr_3X8_BM
```

D4.D.4.5 Glue Laminated timber

Designation of glue-laminated (glu-lam) timber in STAAD.Pro involves defining the material, specifying the dimensions, and associating the material with the member through the `CONSTANTS` command.

```
UNIT CM KN
DEFINE MATERIAL  START
ISOTROPIC GLT_D.Fir-L-24f-EX
E 51611.7
POISSON 0.15
DENSITY 2.5e-005
ALPHA 1.2e-011
END DEFINE MATERIAL
MEMBER PROPERTY  TIMBER CANADIAN
1 PRIS YD 12  ZD 6
CONSTANTS
MATERIAL GLT_D.Fir-L-24f-EX  MEMB 1
```



D4.D.4.6 Example

Sample input file to demonstrate usage of Canadian timber

```
STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0;
5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7  8; 8 8 5;
9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;
UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC SPF_SelStr_4X10_BM
E 1224
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY tim can
1 TO 4 9 TO 11 TABLE ST SPF_SelStr_4X10_BM
5 TO 8 12 13 TABLE ST SPF_SelStr_4X10_BM
CONSTANTS
MATERIAL SPF_SelStr_4X10_BM memb 1 TO 4 9 TO  11
MATERIAL SPF_SelStr_4X10_BM memb 5 TO 8 12 13
PRINT MEMBER PROPERTIES
FINISH
```

D4.D.5 Member Resistance

The member resistances are calculated in STAAD according to the procedures outlined in section 5 (for sawn lumber) and 6 (for Glulam) of CSA086-01.

These depend on several adjustment factors as follows:

KD	Load duration factor (Clause 4.3.2.2-CSA086-01, Table 4.3.2.2)
KH	System factor (Clause 5.4.4 and 6.4.3 and Table 5.4.4 -CSA086-01)
K_T	Treatment factor (Clause 5.4.3 and 6.4.4 -CSA086-01)
KSB	Service condition factor applicable to Bending at extreme fibre (Table 5.4.2 and 6.4.2 -CSA086-01)
KSV	Service condition factor applicable to longitudinal shear (Table 5.4.2 and 6.4.2 CSA086-01)
KSC	Service condition factor applicable to Compression parallel to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
K_SCP	Service condition factor applicable to Compression perpendicular to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
KSE	Service condition factor applicable to modulus of elasticity (Table 5.4.2 and 6.4.2 CSA086-01)
KST	Service condition factor applicable to tension parallel to the grain (Table 5.4.2 and 6.4.2 CSA086-01)
KZB	Size factor applicable to bending (Clause 5.4.5 and Table 5.4.5 -CSA086-01)
KZV	size factor applicable to shear(Clause 5.4.5 and Table 5.4.5 -CSA086-01)
KZT	size factor applicable to tension parallel to grain (Clause 5.4.5 and Table 5.4.5 -CSA086-01)
KZCP	size factor applicable to compression perpendicular to grain (Clause 5.4.5 and Table 5.4.5 -CSA086-01)
K_ZC	size factor applicable to compression parallel to grain (Clause 5.4.5 and Table 5.4.5 -CSA086-01)
CHIX	Curvature factor (Clause 6.5.6.5.2-CSA086-01)
CV	shear load coefficient (Table 6.5.7.4A- CSA086-01)
KN	Notch factor(Clause 5.5.5.4-CSA086-01)

All of these factors must be specified as input according to the classification of timber and stress grade.

Explained here is the procedure adopted in STAAD for calculating the member resistances.

D4.D.5.1 Axial Tension

i. For Sawn timber

The criterion governing the capacity of tension members is based on one limit state. The limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on this limit state per Clause 5.5.9 of CSA086-01.

ii. For Glulam timber

The design of glulam tension members differs from sawn timber since CSA 086-01 assigns different specified strength for gross and net section. The specified strength at net section is slightly higher than the strength of the gross section. Therefore, Glulam tension members are designed based on two limit states. The first one is the limit state of yielding in the gross section. The second limit state involves fracture at the section with the minimum effective net area. The net-section area may be specified by the user through the use of the parameter NSF (see Table 3B.1). STAAD calculates the tension capacity of a member based on these two limits states per Clause.6.5.11 of CSA086-01.

D4.D.5.2 Axial Compression

The compressive resistance of columns is determined based on Clause.5.5.6 and Clause.6.5.8.4 of CSA086-01. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (Kc). The effective length for the calculation of compression resistance may be provided through the use of the parameters KX, KY, KZ, LX, LY and LZ (see Table 3B.1).

D4.D.5.3 Bending

The bending resistance of Sawn members are determined based on Clause 5.5.4 of CSA086-01 and for glulam members are determined based on Clause 6.5.6.5 of CSA086-01. The allowable stress in bending is multiplied by Lateral stability factor, KL to take in account whether lateral support is provided at points of bearing to prevent lateral displacement and rotation

D4.D.5.4 Axial compression and bending

The member strength for sections subjected to axial compression and uni-axial or biaxial bending is obtained through the use of interaction equations. Clause 5.5.10 and 6.5.12 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the member is considered to have FAILed under the loading condition.

D4.D.5.5 Axial tension and bending

The member strength for sections subjected to axial tension and uniaxial or biaxial bending is obtained through the use of interaction equations. Clause 5.5.10 and 6.5.12 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the member is considered to have FAILed under the loading condition.

D4.D.5.6 Shear

The shear resistance of the cross section is determined using the equations of Clause 5.5.5 and 6.5.7.2 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see Table 3B.1), the section is considered to have failed under shear.

Related Links

- *V. CSA 086 2001 - Glulam in Compression* (on page 6232)
- *V. CSA 086 2001 - Glulam in Bending* (on page 6234)
- *V. CSA 086 2001 - Glulam in Tension* (on page 6236)
- *V. CSA 086 2001 - Beam in Compression* (on page 6238)
- *V. CSA 086 2001 - Beam in Bending* (on page 6240)
- *V. CSA 086 2001 - Beam in Tension* (on page 6242)

D4.D.6 Design Parameters

The design parameters outlined in Table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allows the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 126: Canadian Timber Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `TIMBER CANADIAN`.
CHIX	1.0	Curvature Factor for Compression [Clause 6.5.6.5.2]
CV	1.0	Shear Load Coefficient [Table 6.5.7.4A]
KD	1.0	Load Duration Factor [Clause.4.3.2, Table 4.3.2]
KH	1.0	System Factor [Clause 5.4.4/6.4.3, Table 5.4.4]
KN	1.0	Notch Factor [Clause 5.4.7.2.2]
KSB	1.0	Service Condition Factor for Bending at Extreme Fibre Applicable for bending at extreme fibre [Table 5.4.2 and 6.4.2]
KSC	1.0	Service Condition Factor for Compression, Applicable for compression parallel to grain [Table 5.4.2 and 6.4.2]
KSE	1.0	Service Condition Factor for Modulus of Elasticity, Applicable for modulus of elasticity [Table 5.4.2 and 6.4.2]

Parameter Name	Default Value	Description
KST	1.0	Service Condition Factor for Tension, Applicable for tension parallel to grain [Table 5.4.2 and 6.4.2]
KSV	1.0	Service Condition Factor for Shear, Applicable for longitudinal shear [Table 5.4.2 and 6.4.2]
KX	1.0	K value for flexural torsional buckling
KY	1.0	K value in local Y-axis, usually minor axis
KZ	1.0	K value in local Z-axis, usually major axis
KZB	1.0	Size Factor for Bending, Applicable for bending [Clause.5.4.5 and Table 5.4.5]
KZCP	1.0	Size Factor for Compression, Applicable for compression perpendicular to grain [Clause . 5.4.5 and Table 5.4.5]
KZT	1.0	Size Factor for Tension, Applicable for tension parallel to grain [Clause 5.4.5 and Table 5.4.5]
KZV	1.0	Size Factor for Shear [Clause 5.4.5 and Table 5.4.5]
K_SCP	1.0	Service Condition Factor for Compression, Applicable for compression perpendicular to grain [Clause 5.4.2 and Table 6.4.2]
K_T	1.0	Treatment Factor [Clause 5.4.3/6.4.4]

Parameter Name	Default Value	Description
K_ZC	1.0	Size Factor for Compression, Applicable for compression parallel to grain [Clause 5.4.5 and Table 5.4.5]
LX	Member length	Length for flexural torsional buckling
LY	Member length	Length in local Y axis for slenderness value KL/r
LZ	Member length	Length in local Z axis for slenderness value KL/r
NSF	1.0	Net section factor for tension members
RATIO	1.0	Permissible Ratio of Actual to Allowable Value

D4.D.7 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per the CSA086-01 requirements.

Code checking is done using forces and moments at specified sections of the members. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed.

Refer to D1.G.6 Member Design Capabilities (on page 1259) for general information on Code Checking. Refer to TR.52.2 Code Checking Specification (on page 2684) for details the specification of the Code Checking command.

```
PARAMETER
CODE TIMBER CAN
KD 0.99 ALL
KH 0.99 ALL
K_T 0.99 ALL
KSB 0.99 ALL
KSV 0.99 ALL
KSC 0.99 ALL
KSE 0.99 ALL
KST 0.99 ALL
KZB 0.99 ALL
KZV 0.99 ALL
KZT 0.99 ALL
KZCP 0.99 ALL
K_ZC 0.99 ALL
CV 0.99 ALL
KN 0.99 ALL
K_SCP 0.99 ALL
CHIX 0.99 ALL
```

```
RATIO 0.99 ALL
CHECK CODE ALL
FINISH
```

D4.D.8 Member Selection

Member selection based CSA086-2001 is *not* available.

D4.D.9 Tabulated Results of Timber Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the CSA086-01 specification, which governed the design.

Pu	Actual Load in Compression
Tu	Actual Load in Tension
Muy	Ultimate moment in y direction
Muz	Ultimate moment in z direction
V	Ultimate shear force
SLENDERNESS_Y	Actual Slenderness ratio in y direction
SLENDERNESS_Z	Actual Slenderness ratio in z direction
PY	Factored Compressive capacity in y direction
PZ	Factored Compressive capacity in z direction
T	Factored tensile capacity
MY	Factored moment of resistance in y direction
MZ	Factored moment of resistance in z direction
V	Factored shear resistance
SLENDERNESS	Allowable slenderness ratio

D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19

STAAD.Pro is capable of performing steel design based on the Canadian codes CAN/CSA-S16-09, CAN/CSA-S16-14, and CAN/CSA-S16-19 *Limit States Design of Steel Structures*.

To use the 2019 edition (default), specify the command:

```
CODE CANADIAN
```

or

```
CODE CANADIAN 2019
```

To use the 2014 edition, specify the command:

```
CODE CANADIAN 2014
```

To use the 2009 edition, specify the command:

```
CODE CANADIAN 2009
```

Related Links

- *V.CSA S16-09 - Axial Tension* (on page 3877)
- *V.CSA S16-09 - Beam Bending* (on page 3881)
- *V.CSA S16-09 - Beam Shear Capacity* (on page 3885)
- *V.CSA S16-09 - Select a Beam* (on page 3889)
- *V.CSA S16-09 - Shear Capacity Combined Stresses* (on page 3896)
- *V.CSA S16-09 - Short Column Compression* (on page 3901)
- *V.CSA S16-09 - Slender Column Compression* (on page 3905)
- *V.CSA S16-09 - Wide Flange Capacity Combined Stresses* (on page 3910)
- *V. CSA S16-14 - Axial Tension* (on page 3914)
- *V. CSA S16-14 - Beam Bending* (on page 3919)
- *V. CSA S16-14 - Beam Shear Capacity* (on page 3923)
- *V. CSA S16-14 - Select a Beam* (on page 3927)
- *V. CSA S16-14 - Shear Capacity Combined Stresses* (on page 3934)
- *V. CSA S16-14 - Short Column Compression* (on page 3939)
- *V. CSA S16-14 - Slender Column Compression* (on page 3944)
- *V. CSA S16-14 - Wide Flange Capacity Combined Stresses* (on page 3948)

D4.E.1 General Comments

The design of structural steel members in accordance with the specification CAN/CSA S16-09, S16-14, and S16-19 *Limit States Design of Steel Structures* is can be used in STAAD.Pro This code supersedes the previous editions of the CAN/CSA – S16 code.

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-states are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the probability of limits being surpassed is acceptably low.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of CAN/CSA-S16-09, S16-14, and S16-19. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D4.E.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the specified primary and combination loading condition. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D4.E.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to the STAAD.Pro Technical Reference Manual.

Hot Rolled Profiles

- I Shaped, Wide Flange Sections (including S, M, W, HP, and B Shaped)
- Tee Shaped Sections (including WT and T)
- Channel Shaped Sections (including C and MC shaped)
- Angle Shaped Sections (equal and unequal single angle)
- Tube Shaped Sections (RHS, SHS, Box and user defined tube sections)
- Circular Shaped Sections (including CHS, pipe and user defined circular pipe sections)

The following modified sections are also supported:

- I-section with cover plates

The following composite or compound sections are also supported:

- Double Angle (short leg connected and long leg connected angle sections, with and without spacing)
- Double Channel (back to back and front to front connection, with and without spacing)

Tapered Profiles

- Tapered I
- Tapered Round
- Tapered Tube

Note: Tapered members can be designed using check code method only. Member selection is not supported.

D4.E.4 Built-in Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Almost all Canadian steel sections are available for input. A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Refer to D4.B.4 Built-in Steel Section Library (on page 1421) for additional details.

D4.E.5 Section Classification

Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11 and Tables 1 & 2 of CAN/CSA-S16-09, S16-14, or S16-19). The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes.

D4.E.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

$$\phi = 0.9 \text{ and } \phi_u = 0.75$$

Torsion

There are no specific guidelines mentioned in CSA S16 for the torsion design and STAAD does *not* perform torsion check in this case.

Slenderness

The slenderness check is *not* used as a critical member ratio check. If a slenderness check is included and exceed unity (1.0), then it may be reported as the governing criteria. However, if it is less than unity, it will be reported but will not be used as the governing ratio.

D4.E.6.1 Members Subject to Axial Forces

Axial Tension

The criteria governing the capacity of tension members are based on two limit states: resistance due to yielding and resistance due to rupture. The resistance due to rupture depends on effective net section area. You may specify the net section area through the `NSF` design parameter. Additionally, the shear lag factor, U, may be entered using the `SLF` parameter. STAAD.Pro calculates the tension capacity of a member based on these two limits states per Cl.13.2 of CAN/CSA-S16-09. Design parameters `FYLD`, `FU`, `NSF`, and `SLF` (Refer to D4.E.7 Design Parameters (on page 1458)) are applicable for these calculations

i. Yielding, per Cl. 13.2(a)

$$T_r = \phi A_g F_y$$

ii. Rupture, per Cl. 13.2 (b)

$$T_r = \phi_u A_{ne} F_u$$

Note: Pin connection equations in S16-14 and 19 are not checked by the program.

Axial Compression

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios). The effective length for the calculation of compression resistance may be provided through the use of the parameters KT, KY, KZ, LT, LY, and LZ (Refer to D4.E.7 Design Parameters (on page 1458)). Some of the aspects of the axial compression capacity calculations are :

I. For doubly symmetric sections meeting the requirement of Table 1, resistance is:

Resistance due to Major axis buckling per Cl. 13.3.1.

Resistance due to Minor axis buckling per Cl. 13.3.1

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n}$$

where

n = 1.34 for hot-rolled, fabricated structural sections and hollow structural sections manufactured in accordance with CSA G40.20, Class C (cold-formed non-stress-relieved)

2.24 for doubly symmetric welded three-plate members with flange edges oxy-flame-cut and hollow structural sections manufactured in accordance with CSA G40.20, Class H (hot-formed or cold-formed stress-relieved)

Design parameters NCR and STP are used to evaluate the value of n for a member.

λ = $\sqrt{F_y / F_e}$

F_e = $\dfrac{\pi^2 E}{\left(\frac{kL}{r}\right)^2}$

Note: For CSA S16-19, the value of F_e is used per 13.3.1.1 a) for double-symmetric sections.

II. For any other section not covered under Cl. 13.3.1, the factored compressive resistance, C_r , is computed using the expression given in Cl. 13.3.1 with a value of n = 1.34 and the value of F_e taken as follows:

i. For doubly symmetric sections and axisymmetric sections, the least of F_{ex} , F_{ey} , and F_{ez} .

ii. For singly symmetric sections with the Y axis taken as the axis of symmetry, the lesser of F_{ex} and F_{eyz} where

F_{eyz} = $\dfrac{F_{ey} + F_{ez}}{2\Omega}\left[1 - \sqrt{1 - \dfrac{4F_{ey}F_{ez}\Omega}{(F_{ey} + F_{ez})^2}}\right]$

F_{ex} = $\dfrac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2}$

F_{ey} = $\dfrac{\pi^2 E}{\left(\frac{k_y L_y}{r_y}\right)^2}$

F_{ez} = $\left[\dfrac{\pi^2 E C_w}{(K_z L_z)^2} + GJ\right]\dfrac{1}{A\bar{r}_0^2}$

x_0, y_0 = the principal coordinates of the shear center with respect to the centroid of the cross section

$\bar{r}_0^2$ = $x_0^2 + y_0^2 + r_x^2 + r_y^2$

Ω = $\dfrac{r_x^2 + r_y^2}{\bar{r}_0^2}$

iii. For asymmetric sections the smallest root of:

$$(F_e - F_{ex})(F_e - F_{ey})(F_e - F_{ez}) - F_e^2(F_e - F_{ey})\left(\frac{x_0}{\square r_0}\right)^2 - F_e^2(F_e - F_{ex})\left(\frac{y_0}{\square r_0}\right)^2 = 0$$

III. For Class 4 member subjected to axial compression, the factored compressive resistance is:

$$C_r = \phi A_e F_y \left(1 - \lambda^{2n}\right)^{-1/n}$$

A_e is calculated using reduced element widths meeting the maximum width to thickness ratio specified in Table 1.

Effective width required for the calculation of effective area A_e , for different section shapes are as follows.

- For flanges of I-section, T-section and channel section and legs of angle section

$$b_e = 200t / \sqrt{F_y}$$

- For stem of T-section

$$b_e = 340t / \sqrt{F_y}$$

- For flanges of HSS rectangular or Tube sections

$$b_e = 670t / \sqrt{F_y}$$

- For circular HSS or Pipe section

$$D = 23{,}000t/(F_y$$

D4.E.6.2 Members Subject to Bending

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD.Pro through the UNT and UNB parameters (Refer to D4.E.7 Design Parameters (on page 1458)). The LAT parameter is used to specify if the member is laterally supported against lateral torsional buckling.

I. The factored moment resistance, Mr, developed by a member subjected to uniaxial bending moments about a principal axis where effectively continuous lateral support is provided to the compression flange or where the member has no tendency to buckle laterally, is calculated as:

i. For Class 1 and Class 2 sections (Cl. 13.5(a)):

$$Mr = \phi \cdot Z \cdot F_y = \phi \cdot M_p$$

ii. For Class 3 sections (Cl. 13.5(b)):

$$Mr = \phi \cdot S \cdot F_y = \phi \cdot M_y$$

> **Note:** For S16-19, the exception for single angle profiles is included per Cl. 13.5 D) I.

iii. For Class 4 sections (Cl. 13.5(c)):

$$Mr = \phi \cdot S_e \cdot F_y$$

> **Note:** For S16-19, the exceptions for circular hollow profiles per Cl. 13.5 C) IV and for single angle profiles per Cl. 13.5 d) ii) is included.

where

S_e	=	the effective section modulus determined using an effective flange width, b_e , of $670t / \sqrt{F_y}$ for flanges along two edges parallel to the direction of stress and an effective flange width, b_e of $200t / \sqrt{F_y}$ for

flanges supported along one edge parallel to the direction of stress. For flange supported along one edge, $b_e I_e / t$ shall not exceed 60.

II. For laterally unsupported members, flexural resistance is calculated as follows:

i. For doubly symmetric Class 1 and Class 2 sections (Cl 13.6(a)):

$$M_r = \begin{cases} \phi M_u \text{ when } M_u \leq 0.67 M_p \\ 1.15 \phi M_p \left(1 - \frac{0.28 M_p}{M_u}\right) \leq \phi M_p \text{ when } M_u > 0.67 M_p \end{cases}$$

where

M_u = $\frac{\omega_2 \pi}{L} \sqrt{E I_y G J + \left(\frac{\pi E}{L}\right)^2 I_y C_w}$

ω_2 = $1.75 + 1.05\kappa + 0.3\kappa^2 \leq 2.5$

κ = ratio of smaller factored moment to the larger factored moment at opposite ends of the unbraced length (positive for double curvature and negative for single curvature).

Note: The value for ω_2 can be specified using the CB parameter. Otherwise, it is calculated as indicated here.

Note: For S16-19, Cl. 13.6 H) I pertaining to cantilevers with different bracing conditions is included.

ii. For doubly symmetric Class 3 and Class 4 sections –except closed square and circular sections– and for channels:

$$M_r = \begin{cases} \phi M_u \text{ when } M_u \leq 0.67 M_y \\ 1.15 \phi M_y \left(1 - \frac{0.28 M_y}{M_u}\right) \leq \phi M_p \text{ when } M_u > 0.67 M_y \end{cases}$$

but not greater than ϕM_y for Class 3 sections and the value specified in Cl.13.5(c)(iii) for Class 4 sections.

Note: For S16-19, Cl. 13.6 H) II pertaining to cantilevers with different bracing conditions is included in accordance with Cl. 13.6(b), except for M_u and ω_2, which depend on Cl. 13.6 H) I.

iii. For singly symmetric (monosymmetric) Class 1, Class 2, or Class 3 sections and T-shape sections, lateral torsional buckling strength shall be checked separately for each flange under compression under factored loads at any point along its unbraced length:

- when $M_u > M_{yr}$:

$$M_r = \phi\left[M_p - \left(M_p - M_{yr}\right)\left(\frac{L - L_u}{L_{yr} - L_u}\right)\right] \leq \phi M_p$$

where

M_{yr} = $0.7 S_x F_y$, with S_x taken as the smaller of the two potential values

L_{yr} = length L obtained by setting $M_u = M_{yr}$

L_u = $1.1 r_t \sqrt{E / F_y} = \frac{490 r_t}{\sqrt{F_y}}$

r_t = $\frac{b_c}{\sqrt{12\left(1 + \frac{h_c w}{3 b_c t_c}\right)}}$

h_c = depth of the web in compression

b_c = width of the compression flange

t_c = thickness of the compression flange

- when $M \le M_{yr}$:

$$M_r = \phi M_u$$

where

M_u = the critical elastic moment of the unbraced section =

$$\frac{\omega_3 \pi^2 E I_y}{2L^2}\left[\beta_x + \sqrt{\beta_x^2 + 4\left(\frac{GJL^2}{\pi^2 E I_y} + \frac{C_w}{I_y}\right)}\right]$$

β_x = asymmetry parameter for singly symmetric beam =

$$0.9\left(d - t\right)\left(\frac{2I_{yc}}{I_y} - 1\right)\left[1 - \left(\frac{I_y}{I_x}\right)^2\right]$$

For CSA S16-19, the approximate formula given is only valid for cases where $I_x > 2I_y$ and where $0.1 < I_y/(I_{yc} + I_{yt}) < 0.9$ and it is not valid for T sections.

I_{yc} = moment of inertia of the compression flange about the y-axis

I_{yt} = moment of inertia of the tension flange about the y-axis

when singly symmetric beams are in single curvature,

$\omega_3 = \omega_2$ for beams with two flanges, = 1.0 for T-sections

in all other cases,

$\omega_3 = \omega_2[0.5 + 2(I_{yc}/Iy)^2]$ f, but ≤ 2.0 for T-Sections

Note: For single angles (asymmetric sections) designed per S16-19, Cl. 13.6 G (I) and (II) are used.

D4.E.6.3 Members Subject to Combined Forces

For each of the following interaction equations, the value of the `RATIO` parameter is used in lieu of 1.0 when it is specified (Refer to D4.E.7 Design Parameters (on page 1458)).

Axial compression and bending

The member strength and stability for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors (Cl. 13.8). ω_{1y} and ω_{1z} are calculated as per Cl. 13.8.5 or as specified in the `CMY` and `CMZ` design parameters, respectively.

I. For Class 1 and Class 2 sections of I-shaped members (Cl. 13.8.2):

$$\frac{C_f}{C_r} + \frac{0.85 U_{1x} M_{fx}}{M_{rx}} + \frac{\beta U_{1y} M_{fy}}{M_{ry}} \le 1.0$$

where

C_f , M_f = the maximum load effects, including stability, as specified in Cl. 8.4.

β = $0.6 + 0.4\lambda_y \le 0.85$

The capacity of the member is investigated for the following:

a. Cross sectional strength with $\beta = 0.6$, where

i. C_r as specified in Cl. 13.3 with $\lambda = 0$

   ii. M_r as specified in Cl. 13.5
   iii. U_{1x} and U_{1y} as specified in Cl. 13.8.4 but not less than 1.0. Design parameters SSY and SSZ are used to evaluate these coefficients.

b. Overall member strength, where

   i. C_r as specified in Cl. 13.3 with $K = 1$, except for uniaxial bending, in which case C_r is based on the axis of bending
   ii. M_r as specified in Cl. 13.5
   iii. U_{1x} and U_{1y} are taken as 1.0 for members in an unbraced frame, and as specified in Cl. 13.8.4 for members in a braced frame. Design parameters SSY and SSZ are used to evaluate these coefficients.

c. Lateral torsional buckling strength, when applicable, where

   i. C_r as specified in Cl. 13.3
   ii. M_{rx} as specified in Cl. 13.6
   iii. M_{ry} as specified in Cl. 13.5
   iv. U_{1x} and U_{1y} are taken as 1.0 for members in an unbraced frame, and as specified in Cl. 13.8.4 for members in a braced frame (where U_{1x} is not less than 1.0). Design parameters SSY and SSZ are used to evaluate these coefficients.

II. For Class 1 and Class 2 square or circular HSS sections per S16-19 only (Cl. 13.8.3):

- For square sections:

$$\frac{C_f}{C_r} + \frac{0.85 U_{1x} M_{fx}}{M_{rx}} + \frac{0.50 U_{1y} M_{fy}}{M_{ry}} \le 1.0$$

- For circular sections:

$$\frac{C_f}{C_r} + \frac{0.85\sqrt{(U_{1x} M_{fx})^2 + (U_{1y} M_{fy})^2}}{M_{rx}} \le 1.0$$

III. For all other cases (Cl13.8.3 per S16-09 / 14 or Cl 13.8.4 of S16-19):

$$\frac{C_f}{C_r} + \frac{U_{1x} M_{fx}}{M_{rx}} + \frac{U_{1y} M_{fy}}{M_{ry}} \le 1.0$$

The capacity of the member is investigated for the following per Cl.13.8.2:

a. Cross sectional strength
b. Overall member strength
c. Lateral torsional buckling strength,

Axial tension and bending

Members subjected to axial tension and bending must satisfy the following equation (Cl. 13.9.1):

$$\frac{T_f}{T_r} + \frac{M_f}{M_r} \le 1.0$$

where

M_r	=	the moment resistance as specified in Cl. 13.5.

Note: For I shapes with equal flanges designed to S16-19, Class 1 and 2 sections are proportioned as per Cl. 13.9.2.

Additionally, the following equations must be satisfied for laterally unsupported members (Cl. 13.9.2 of S16-09 / 14):

$$\frac{M_f}{M_r} - \frac{T_f Z}{M_r A} \le 1.0 \text{ for Class 1 and Class 2 sections}$$

$$\frac{M_f}{M_r} - \frac{T_f S}{M_r A} \le 1.0 \text{ for Class 3 and Class 4 sections}$$

where

M_r	=	the moment resistance as specified in Cl. 13.6.

The following equations must be satisfied for laterally unsupported members designed per S16-19 (Cl. 13.9.3 of S16-19):

$$\frac{M_{fx}}{M_{rx}} + \frac{M_{fy}}{M_{ry}} - \frac{T_f Z_x}{M_{rx} A} \le 1.0 \text{ for Class 1 and Class 2 sections}$$

$$\frac{M_{fx}}{M_{rx}} + \frac{M_{fy}}{M_{ry}} - \frac{T_f S_x}{M_{rx} A} \le 1.0 \text{ for Class 3 and Class 4 sections}$$

where

M_{rx}	=	the moment resistance as specified in Cl. 13.6.1.

where

M_{ry}	=	the moment resistance as specified in Cl. 13.5.

Biaxial Bending

For bending about both axis, the following equation must be satisfied (Cl. 13.8):

$$\frac{M_{fx}}{M_{rx}} + \frac{M_{fy}}{M_{ry}} \le 1.0$$

Shear and Bending

For S16-09 / 14, to resist the combined effects of shear and bending, all of the following equations must be satisfied (Cl. 14.6):

$$0.727\frac{M_f}{M_r} + 0.455\frac{V_f}{V_r} \le 1.0 \qquad \text{(S16-09 / 14)}$$

$$\frac{M_f}{M_r} \le 1.0$$

$$\frac{V_f}{V_r} \le 1.0$$

where

M_r	=	the value determined in accordance with Cl. 13.5 of Cl 13.6 as applicable
V_r	=	the value determined in accordance with Cl. 13.4

For S16-19: for beams with webs with $F_s > 0.60\ F_y$ to resist the combined effects of shear and bending, the shear resistance, V_r, is multiplied by the following reduction factor (Cl. 14.6):

$$\left[2.20 \text{ - } 1.60\frac{M_f}{M_r}\right]$$

The shear resistance is not reduced below $0.60\ \phi A_w F_y$. Also, the shear resistance is not increased due to this calculation.

D4.E.6.4 Members Subject to Shear

Factored shear resistance, V_r , developed by the web of flexural member is calculated as:

$$V_r = \phi A_w F_s$$

where

A_w = shear area

F_s is evaluated as:

I. For unstiffened webs (Cl. 13.4.1.1.(a)):

i. when $\frac{h}{w} \le \frac{1,\ 014}{\sqrt{F_y}}$, $F_s = 0.66F_y$

ii. when $\frac{1,\ 014}{\sqrt{F_y}} < \frac{h}{w} \le \frac{1,\ 435}{\sqrt{F_y}}$, $F_s = \frac{670\sqrt{F_y}}{(h\ /\ w)}$

iii. when $\frac{h}{w} > \frac{1,\ 436}{\sqrt{F_y}}$, $F_s = \frac{961,\ 200}{(h\ /\ w)^2}$

II. For stiffened webs (i.e., when the STIFF parameter is specified) (Cl. 13.4.1.1(b)):

i. when $\frac{h}{w} \le 439\sqrt{\frac{k_v}{F_y}}$, $F_s = 0.66F_y$

ii. when $439\sqrt{\frac{k_v}{F_y}} < \frac{h}{w} \le 502\sqrt{\frac{k_v}{F_y}}$, $F_s = F_{cri}$

iii. when $502\sqrt{\frac{k_v}{F_y}} < \frac{h}{w} \le 621\sqrt{\frac{k_v}{F_y}}$, $F_s = F_{cri} + k_a(0.50F_y - 0.866F_{cri})$

iv. when $621\sqrt{\frac{k_v}{F_y}} < \frac{h}{w}$, $F_s = F_{cre} + k_a(0.50F_y - 0.866F_{cre})$

where

A_e = shear buckling coefficient:

i. when $a/h < 1, k_v = 4 + \frac{5.34}{(a\ /\ h)^2}$

ii. when $a/h \ge 1, k_v = 5.34 + \frac{4}{(a\ /\ h)^2}$

a/h = stiffener aspect ration (i.e., ratio of the distance between stiffeners to web depth)

F_{cri} = $290\frac{\sqrt{F_y k_v}}{(h\ /\ w)}$

k_a = aspect coefficient = $\frac{1}{\sqrt{1 + (a\ /\ h)^2}}$

F_{cre} = $\frac{180,\ 000k_v}{(h\ /\ w)^2}$

For tubular members, the shear resistance, V_r , is calculated as:

$$V_r = 0.66\phi(A_e/2)F_y$$

where

A_e = the cross-sectional area of the tubular member

D4.E.7 Design Parameters

The design parameters outlined in the following table may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 127: Canadian Steel Design CSA-S16-09/14/19 Parameters

Parameter Name	Default Value	Description
CODE	CODE CANADIAN	Used to designate this code (default is the 2019 edition): `CODE CANADIAN (2019)` (for S16-19) `CODE CANADIAN 2014` (for S16-14) `CODE CANADIAN 2009` (for S16-09) `CODE CANADIAN 2001` (D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01 (on page 1419)) Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	Used to specify locations along member length considered for design: 0.0 = design only for end moments and those at locations specified by a `SECTION` command. 1.0 = Perform design for moments at twelfth points along the beam.

Parameter Name	Default Value	Description
CAN (S16-19 only)	0	This parameter is applicable for doubly symmetric I-shaped cantilever member as per CL 13.6 (h) subjected to transverse load only. 0 = Not a Cantilever member (default) 1 = Unbraced Cantilever Member. Both flanges unbraced. 2 = Cantilever Member braced at tension flange only 3 = Cantilever Member braced at both flanges **Notes:** **1.** Valid for CSA S16 2019 only. This parameter will be used to calculate the factored moment of resistance for a doubly symmetric I-shaped cantilever member. **2.** This parameter can be used with the LHT parameter for CL 13.6 (h) check. **3.** For deflection check, 0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. Use 1 or 2 or 3 to deflection check based on the principle that maximum deflection is of the cantilever type (see D1.B.1.2 Design Parameters (on page 1134)).
CB	0.0	Value of ω_2 (code Cl 13.6) to be used for calculations. ω_2 is calculated internally if CB is 0.0 or not provided.

Parameter Name	Default Value	Description
CMY	ω_1 calculated as per clause number 13.8.5 (a)	ω_1 in local Y direction for the member. The program calculated value of ω_1 depends on whether the member is subjected to any transverse load or not: • $\omega_1 = 0.6 - 0.4\kappa \geq 0.4$ if the member is not subjected to transverse loads between supports. Refer Clause 13.8.5 (a) • $\omega_1 = 1.0$ if the member is subjected to distributed loads or a series of point loads between supports. Refer Clause 13.8.5 (b) • $\omega_1 = 0.85$ if the member is subjected to a concentrated load or moment between supports. Refer Clause 13.8.5 (c) STAAD.Pro will calculate ω_1 based on the formula specified in 13.8.5 (a) unless you directly specify a value. You may specify any value between 0.4 to 1 for ω_1 .
CMZ	ω_1 calculated as per clause number 13.8.5 (a)	ω_1 in local Z direction for the member. The program calculated value of ω_1 depends on whether the member is subjected to any transverse load or not: • $\omega_1 = 0.6 - 0.4\kappa \geq 0.4$ if the member is not subjected to transverse loads between supports. Refer Clause 13.8.5 (a) • $\omega_1 = 1.0$ if the member is subjected to distributed loads or a series of point loads between supports. Refer Clause 13.8.5 (b) • $\omega_1 = 0.85$ if the member is subjected to a concentrated load or moment between supports. Refer Clause 13.8.5 (c) STAAD.Pro will calculate ω_1 based on the formula specified in 13.8.5 (a) unless you directly specify a value. You may specify any value between 0.4 to 1 for ω_1 .

Parameter Name	Default Value	Description
CPSACING	0.0	Spacing between connectors of built-up members required for slenderness ratio calculation per Cl. 19.1.4.
DFF	None (Mandatory for deflection check)	"Deflection Length"/Maxmimum Allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting start point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (see D1.B.1.2 Design Parameters (on page 1134))
DMAX	45.0 in.	Maximum allowable depth (Applicable for member selection)
DMIN	0.0 in.	Minimum required depth (Applicable for member selection)
FLX	0	Parameter for specifying the flexural-torsional restraint condition. 0 = Flexural-torsional restraint is not provided 1= Flexural-torsional restraint is provided along the length.
FU	345.0 MPa	Ultimate strength of steel.
FYLD	300.0 MPa	Yield strength of steel.
IMM (S16-19 only)	0	Truss Type as required for CL 13.3.2.2 and CL 13.3.2.3 as per CSA S16-19 0 = Non Truss member 1 = Planner Truss Member 2 = Space Truss Member

Parameter Name	Default Value	Description
KT	1.0	K value for flexural torsional buckling.
KY	1.0	K value for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.
KZ	1.0	K value for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
LAT	0	Specify lateral support conditions: 0 = Beam is laterally unsupported. 1 = Beam is laterally supported.
LEG	0	This parameter is meant for plain angles (clause 13.3.3.2). 0 = The angle is connected by the longer leg. 1 = The angle is connected by the shorter leg.
LHT (S16-19 only)	0	Load Height Position as required to check as per CL 13.6.(h) 0 = Load applied at the shear center 1 = Load applied at the tension side of the shear center 2 = Load applied at the compression side of the shear center **Note:** The LHT parameter should be used along with the CAN parameter.
LT	Member Length	Length for flexural torsional buckling.
LY	Member Length	Length for general column flexural buckling about the local Y-axis. Used to calculate slenderness ratio.

Parameter Name	Default Value	Description
LZ	Member Length	Length for general column flexural buckling about the local Z-axis. Used to calculate slenderness ratio.
MAIN	200	Allowable slenderness limit for compression members. A value of 1 suppresses this check. Any value greater than 1 is used as the compression slenderness check value. A value of zero (0) uses the default (i.e., slenderness limit of 200).
NCR	0	This parameter sets the *n* factor to calculate Cr value as per clause number 13.3.1 0 = 1.34 1 = 2.24 **Notes:** **1.** The default value of n is used for all sections. **2.** n = 2.24 for Hollow sections (e.g., HSST, HSSP, Pipe Tube), for doubly symmetric built-up (STP = 2) sections (e.g., W, M, S, HP sections), and for Welded Wide Flange (WWF) sections if NCR is set to 1. WWF sections are designed as built up sections. **3.** n = 1.34 for doubly symmetric hot rolled sections and for single symmetric hot rolled or built-up sections (e.g., Channel, Tee, Angle) regardless of the NCR parameter value.
NSF	1.0	Net section factor for tension members.
PROFILE	-	Used in member selection. Refer to TR.48.1 Parameter Specifications (on page 2676) for details.

Parameter Name	Default Value	Description
RATIO	1.0	Permissible ratio of actual load effect to the design strength.
SLF	1.0	The shear lag factor, U, normally taken from Cl. 12.3.3.2, combined with the `NSF` parameter to determine the rupture strength.
SNUG	1	Specify type of connection for the built-up members (Refer to Cl. 19.1.4): 0 = Welded or pretensioned bolts. 1 = Bolted snug-tight.
SSY	0	0 = Sway or Unbraced along local Y axis 1 = Braced along local Y axis This parameter is required to choose the proper value of U_{1y} which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. Refer to Axial compression and bending (on page 1454) for details.
SSZ	0	0 = Sway or Unbraced along local Z axis 1 = Braced along local Z axis This parameter is required to choose the proper value of U_{1z} which is used to determine the cross-sectional strength, overall member strength and lateral torsional buckling strength. Refer to Axial compression and bending (on page 1454) for details.
STIFF	Member length of depth of beam, whichever is lesser.	Spacing of traverse stiffeners.
STP	1	1.0 = Rolled section 2.0 = Welded built-up section

Parameter Name	Default Value	Description
TMAIN	300	Allowable slenderness limit for tension members. A value of 1 suppresses this check. Any value greater than 1 is used as the tension slenderness check value. A value of zero (0) uses the default (i.e., slenderness limit of 300).
TRACK	0.0	Design output: 0.0 = Report only minimum design results. 1.0 = Report design strengths also. 2.0 = Provide full details of design.
UNB	Member Length	Unsupported length in bending compression of the bottom flange for calculating moment resistance.
UNT	Member Length	Unsupported length in bending compression of the top flange for calculating moment resistance.

D5. European Codes

D5.A. European Codes - Concrete Design Per DD ENV 1992

Note: This code has been removed from the batch design. To perform design to the current Eurocode 2 design code, please use the D. Advanced Concrete Design (on page 1071).

D5.B. European Codes - Steel Design per Eurocode 3 [DD ENV 1993-1-1:1992]

STAAD.Pro is capable of performing steel design based on the European code EC3 DD ENV 1993-1-1:1992 *Eurocode 3: Design of steel structures Part 1.1 General rules and rules for buildings*.

Note: The DD ENV 1993-1-1:1992 code has now been officially superseded by EN 1993-1-1:2005. Hence releases of STAAD.Pro subsequent to version SS3 (20.07.08.xx) will not support this design code. The SS3 build will perform member design to this code for legacy files but has this code removed from the design codes list in the GUI. Users are advised to use the EN 1993-1-1:2005 version for Eurocode 3 design.

Tip: Design per EC3 DD ENV 1993-1-1:1992 is also available in the Steel Design mode in the Graphical User Interface.

D5.B.1 General Description

1. Selecting the applicable load cases to be considered in the design process.
2. Providing appropriate "Parameter" values if different from the default values.
3. Specify whether to perform code-checking and/or member selection.

These operations can be repeated by the user any number of times depending on the design requirements. The "Parameters" referred to above provide the user with the ability to allocate specific design properties to individual members or member groups considered in the design operation.

D5.B.1.1 Eurocode 3 DD ENV 1993-1-1:1992 (EC3 DD)

The DD ENV version of Eurocode 3, *Design of steel structures, Part 1.1 General rules and rules for buildings* (EC3 DD) provides design rules applicable to structural steel used in buildings and civil engineering works. It is based on the ultimate limit states philosophy that is common to modern standards. The objective of this method of design is to ensure that possibility of failure is reduced to a negligible level. This is achieved through application of safety factors to both the applied loads and the material properties.

The code also provides guidelines on the global methods of analysis to be used for calculating internal member forces and moments. STAAD uses the elastic method of analysis which may be used in all cases. Also there are three types of framing referred to in EC3. These are "Simple", "Continuous", and "Semi-continuous" which reflect the ability of the joints to developing moments under a specific loading condition. In STAAD only "Simple" and "Continuous" joint types can be assumed when carrying out global analysis.

D5.B.1.2 National Application Documents

Various authorities of the CEN member countries have prepared National Application Documents to be used with EC3. These documents provide alternative factors for loads and may also provide supplements to the rules in EC3.

The current version of EC3 DD implemented in STAAD adheres to the factors and rules provided in DD ENV 1993-1-1:1992 and has *not* been modified by any National Application Document.

Note: National Annex documents *are* available for EC3 BS EN 1993-1-1:2005. Refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478)

Axes convention in STAAD.Pro and Eurocode 3

By default, STAAD.Pro defines the major axis of the cross-section as Z-Z and the minor axis as Y-Y. A special case where Z-Z is the minor axis and Y-Y is the major axis is available if the `SET Z UP` command is used and is discussed in TR.5 Set Command Specification (on page 2206). The longitudinal axis of the member is defined as X and joins the start joint of the member to the end with the same positive direction.

Eurocode 3, however, defines the principal cross-section axes in reverse to that of STAAD.Pro, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

Bear this difference in mind when examining the code-check output from STAAD.Pro.

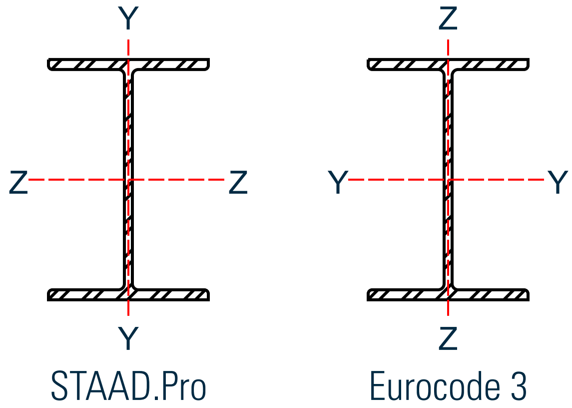


Figure 163: Axis convention in STAAD.Pro and EC3

D5.B.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D5.B.3 Material Properties and Load Factors

The characteristic yield strength of steel used in EC3 DD design is based on table 3.1 of the code. Design resistances are obtained by dividing the characteristic yield strength by the material partial safety factor Γm. The magnitude of Γm in STAAD.Pro is 1.1 which is applicable to all section types. A separate safety factor parameter named GB1 is used to check the resistance of a member to buckling and also has a default value of 1.1.

Material coefficients for steel in STAAD.Pro take the following default values unless replaced by user's numerical values provided in the input file.

Modulus of Elasticity, E = 205000 N/mm^2
Shear Modulus, G = E/2(1+ ν)
Poisson's Ratio, ν = 0.3
Unit weight, Γ = 76.8 KN/m^3

The magnitude of design loads is dependent on Γ_f, the partial safety factor for the action under consideration. In STAAD.Pro you are allowed total control in providing applicable values for the factors and their use in various load combinations.

D5.B.4 Section Classification

The occurrence of local buckling of the compression elements of a cross-section prevents the development of full section capacity. It is therefore imperative to establish this possibility prior to determining the section capacities. Cross sections are classified in accordance with their geometrical properties and the stress pattern on the compression elements. For each load case considered in the design process, STAAD determines the section class and calculates the capacities accordingly.

The EC3 DD design module in STAAD can design members with all section profiles that are of Class 1 2 or 3 as defined in section 5.3.2 of the code. However, the design of members that have a "Class 4" section profile are limited to WIDE FLANGE, TEE, SINGLE CHANNEL, SINGLE ANGLE, and RECTANGULAR HOLLOW SECTIONS. Also built-up user sections that are class 4 sections are not dealt with in the current version of EC3 design in STAAD.Pro.

Laced and battened members are not considered in the current version of EC3 DD design module in STAAD.Pro.

D5.B.5 Member Design

D5.B.5.1 Design of Beams as per DD ENV 1993-1-1:1992

EC3 DD design in STAAD.Pro considers members that are primarily in bending and/or shear as beams and performs cross section and member capacity checks in accordance with the code. The main requirement for a beam is to have sufficient cross-section resistance to the applied bending moment and shear force. The possibility of lateral-torsional buckling is also taken into consideration when the full length of the member has not been laterally restrained.

The bending capacity is primarily a function of the section type and the material yield strength and is determined according to Cl. 5.4.5 of the code. The shear capacity and the corresponding shear checks are done as per section 5.4.6 of the code.

There are four classes of cross-sections defined in EC3. Class 1 and 2 sections can both attain full capacity with the exception that the class 2 sections cannot sustain sufficient rotation required for plastic analysis of the model. Hence the full plastic section modulus is used in the design calculations. Class 3 sections, due to local buckling, cannot develop plastic moment capacity and the yield stress is limited to the extreme compression fibre of the section. The elastic section modulus is used to determine the moment capacity for class 3 sections. Class 4 sections do suffer from local buckling and explicit allowance must be made for the reduction in section properties before the moment capacity can be determined. Further, because of interaction between shear force and bending moment, the moment resistance of the cross-section may be reduced. This, however, does not occur unless the value of applied shear forces exceeds 50% of the plastic shear capacity of the section. In such cases the web is assumed to resist the applied shear force as well as contributing towards the moment resistance of the cross-section.

As mentioned in the previous section, the design of class 4 sections is limited to WIDE FLANGE, TEE, SINGLE CHANNEL, SINGLE ANGLE, and RECTANGULAR HOLLOW SECTIONS. The effective section properties are worked out as described in Cl. 5.3.5 of the code.

Beams are also checked for lateral-torsional buckling according to section 5.5.2 of the code. The buckling capacity is dependent on the section type as well as the unrestrained length, restraint conditions and type of applied loading. The lateral torsional buckling checks involves the calculation of the “Elastic critical moment”, Mcr, which is calculated in STAAD as per the method given in Annex F of the code.

In the presence of a shear force, beams are also checked for shear as per section 5.4.6 of the code. In cases where the members are subject to combined bending and shear, the combined bending and shear checks are done in STAAD as per clause 5.4.7 of the code.

D5.B.5.2 Design of Axially Loaded Members

The design of members subject to tension loads alone are performed as per Cl 5.4.3 of the code. The tension capacity is calculated based on yield strength, material factor Γm and cross-sectional area of the member with possible reduction due to bolt holes. When bolt holes need to be considered in the capacity calculations the value used for Γm is 1.2 and the yield strength is replaced with the ultimate tensile strength of the material. The tension capacity is then taken as the smaller of the full section capacity and the reduced section capacity as stated above.

The design of members subject to axial compression loads alone are performed as per Cl 5.4.4 of the code. For members with class 1 2 or 3 section profiles, the full section area is considered in calculating the section capacity. However in case of class 4 sections, the “effective cross-section” is considered to calculate the compressive strength. Also any additional moments induced in the section due to the shift of the centroidal axis of the effective section will also be taken into account as per clause 5.4.8.3 of the code. The effective section properties for class 4 sections will be worked out as given in Cl.5.3.5 of the code.

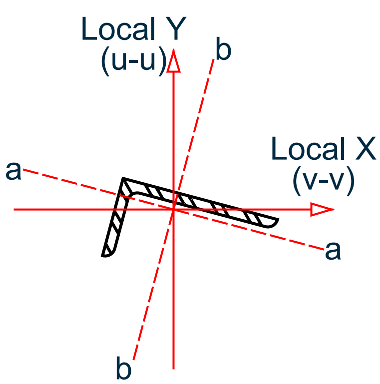
In addition to the cross section checks, buckling resistance will also be checked for such members. This is often the critical case as the buckling strength of the member is influenced by a number of factors including the section type and the unbraced length of the member. The buckling capacity is calculated as per Cl. 5.5 of the code.

DD ENV 1993-1-1:1992 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to work out the slenderness of such members. In these cases, the EC3 DD design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 and table 25 of BS 5950-1:2000 are used in the current version of the EC3 DD design module

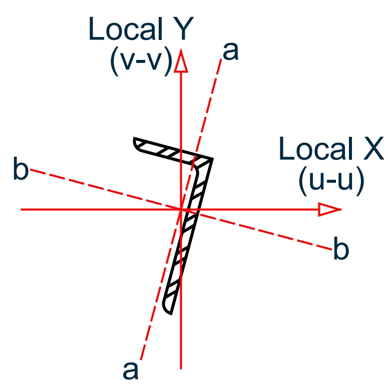
Single Angle Sections

Angle sections are unsymmetric and when using BS 5950:2000 table 25, you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, Lvv, is taken as the `LVV` parameter or $LY \cdot KY$, if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the `LEG` parameter, see section 5B.6 for more information on the `LEG` parameter. The effective length in the a-a axis is taken as $LY \cdot KY$ and the effective length in the b-b axis as $LZ \cdot KZ$.

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).



ST angle and USER table angles



RA angle

D5.B.5.3 Design of members with combined axial load and bending

The bending resistance of members could be reduced by the presence of a co-existent axial load. This is then checked against the lateral-torsional buckling resistance of the section. The EC3 DD design module in STAAD takes such a scenario into account and performs the necessary checks as per Cl. 5.4.8 of the code. Class 1 and class 2 sections are checked as per cl. 5.4.8.1 and Class 3 and Class 4 sections are checked as per clauses 5.4.8.2 and 5.4.8.3 respectively. The effective section properties for class 4 sections are worked out as given in Cl. 5.3.5 of the code.

Generally, EC3 requires checking cross-section resistance for local capacity and also checking the overall buckling capacity of the member. In the case of members subject to axial tension and bending, there is provision to take the stabilizing effect of the tension load into consideration. This is achieved by modifying the extreme compression fibre stress and calculating an effective applied moment for the section. The checks are done as per Cl. 5.5.3 of the code. In case of a combined axial compressive load and bending moment, the member will be checked as per the rules in section 5.5.4 of the code.

The presence of large shear force can also reduce the bending resistance of the section under consideration. If the shear load is large enough to cause a reduction in bending resistance, then the reduction due to shear has to be taken into account before calculating the effect of the axial load on the bending resistance of the section. If the member is subject to a combined shear, axial load and bending moment then the section capacity checks will be done as per Cl. 5.4.9 of the code.

As stated in the previous section, DD ENV 1993-1-1:1992 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to work out the slenderness of such members. In these cases, the EC3 DD design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 of BS 5950-1:2000 is used in the current version of the EC3 DD design module. Please refer to the note in section 5B.5.2 for St and RA angle specifications.

Please note that laced or battened compression members are not dealt within the current version of EC3 DD design module in STAAD.Pro.

D5.B.6 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, the user may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the new setting must be compatible with the active "unit" specification.

The following table lists all the relevant EC3 parameters together with description and default values.

Table 128: Steel Design Parameters EC3 DD

Parameter Name	Default Value	Description
CODE	Undefined	You must specify EC3 or EUROPE. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	3	Parameter to control the number of sections to checked along the length of a beam: 0. Check sections with end forces only 1. Check at location of maximum Mz along beam 2. Check sections with end forces and forces at location of `BEAM 1.0` check. 3. Check at every 1/13th point along the beam and report the maximum Refer to Note 2 below.
CAN	0	Member will be considered as a cantilever type member for deflection checks. 0 indicates that member will not be treated as a cantilever member 1 indicates that the member will be treated as a cantilever member
CMM	1.0	Indicates type of loading on member. Valid values range from 1 to 6. Refer to Table 7B.3 for more information on its use.
CMN	1.0	Indicates the level of End-Restraint. 1.0 = No fixity 0.5 = Full fixity 0.7 = One end free and other end fixed
DMAX	100.0 cm	Maximum allowable depth for the member.
DMIN	0	Minimum required depth for the member.
DFF	None (Mandatory for deflection check)	Deflection limit

Parameter Name	Default Value	Description
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length".
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length".
FU		Ultimate tensile strength of steel
GB1	1.1	Partial safety factor used in buckling checks for compression members
GM0	1.1	Corresponds to the Γ_{m0} factor in DD ENV 1993-1-1:1992
GM1	1.1	Corresponds to the Γ_{m1} factor in DD ENV 1993-1-1:1992
GM2	1.1	Corresponds to the Γ_{m2} factor in DD ENV 1993-1-1:1992
KY	1.0	K factor in local y axis.
KZ	1.0	K factor in local z axis.
LEG	0.0	Connection type Refer to Note 1 below.
LVV	Maximum of Lyy and Lzz (Lyy is a term used by BS5950)	Buckling length for angle about its principle axis
LY	Member Length	Compression length in local y axis, Slenderness ratio = (KY)*(LY)/(Ryy)
LZ	Member Length	Compression length in local z axis, Slenderness ratio = (KZ)*(LZ)/(Rzz)

Parameter Name	Default Value	Description
PLG	0	(Polish NA only) Perform additional checks per Cl. 6.3.3 0. Ignore additional PN EN checks 1. Include additional PN EN checks Refer to D5.D.8.7 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz (on page 1555)
PY	Yield Strength	The yield strength default value is set based on the default value of the "SGR" parameter.
NSF	1.0	Net tension factor for tension capacity calculation.
RATIO	1	Permissible ratio of loading to capacity.
SBLT	0.0	Indicates if the section is rolled or built-up. 0.0 = Rolled 1.0 = Built-up
SGR	0.0	Steel grade as per table 3.1 in EC3. 0.0 = Fe 360 1.0 = Fe 430 2.0 = Fe 510
TRACK	0	Controls the level of detail of output. 0 = minimum 1 = intermediate 2 = maximum 4 = perform a deflection check See note 3 below.
UNF	1.0	Unsupported buckling length as a factor of the beam length

Parameter Name	Default Value	Description
UNL	Member Length	Unrestraint length of member used in calculating the lateral-torsional resistance moment of the member.
ZIV	0.8	Specifies a reduction factor for vectoral effects to be used in axial tension checks [Cl 5.5.3(2)]

Notes

1. LEG – (Ref: Table 25 BS5950)

   The slenderness of single and double angle, channel and tee sections are specified in BS 5950 table 25 depending on the connection provided at the end of the member (Refer to D5.B.4 Section Classification (on page 1467)). To define the appropriate connection, a LEG parameter should be assigned to the member.

   The following table indicates the value of the LEG parameter required to match the BS5950 connection definition:

   Table 129: LEG Parameter values

Clause	Bold Configuration	Leg	LEG Parameter
4.7.10.2 Single Angle	(a) - 2 bolts	short leg	1.0
		long leg	3.0
	(b) - 1 bolts	short leg	0.0
		long leg	2.0
4.7.10.3 Double Angles	(a) - 2 bolts	short leg	3.0
		long leg	7.0
	(b) - 1 bolts	short leg	2.0
		long leg	6.0
	(c) - 2 bolts	long leg	1.0
		short leg	5.0
	(d) - 1 bolts	long leg	0.0
		short leg	4.0
4.7.10.4 Channels	(a) - 2 or more rows of bolts		1.0
	(b) - 1 row of bolts		0.0

Clause	Bold Configuration	Leg	LEG Parameter
4.7.10.5 Tee Sections	(a) - 2 or more rows of bolts		1.0
	(b) - 1 row of bolts		0.0

For single angles, the slenderness is calculated for the geometric axes, a-a and b-b as well as the weak v-v axis. The effective lengths of the geometric axes are defined as:

La = KY * KY

Lb = KZ * LZ

The slenderness calculated for the v-v axis is then used to calculate the compression strength pc for the weaker principal axis (z-z for ST angles or y-y for RA specified angles). The maximum slenderness of the a-a and b-b axes is used to calculate the compression strength pc for the stronger principal axis.

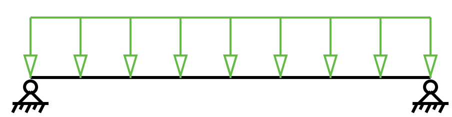
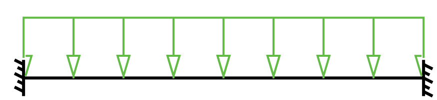
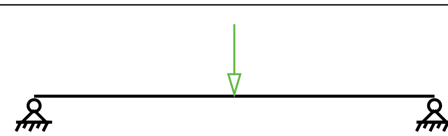
Alternatively for single angles where the connection is not known or Table 25 is not appropriate, by setting the LEG parameter to 10, slenderness is calculated for the two principal axes y-y and z-z only. The LVV parameter is not used.

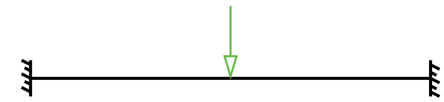
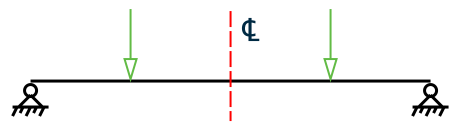
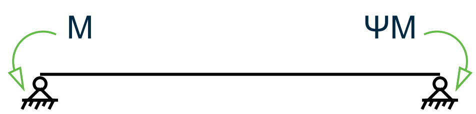
For double angles, the LVV parameter is available to comply with note 5 in table 25. In addition, if using double angles from user tables, (G.6.3 User-Provided Steel Table (on page 2117)) an eleventh value, rvv, should be supplied at the end of the ten existing values corresponding to the radius of gyration of the single angle making up the pair.

2. `BEAM`

   Ensure that this parameter is set to either 1 or 2 while performing code checking for members susceptible to Lateral - Torsional Buckling.

Table 130: Values for the CMM Parameter

CMM Value	Loading and Support Conditions
1	
2	
3	

CMM Value	Loading and Support Conditions
4	
5	
6	

3. Checking beam deflection

   With the `TRACK` parameter set to 4, the members included in a `CHECK CODE` command will be checked for the local axis deflection rather than for the stress capacity using the current `LOAD LIST`.

   If both stress capacity and deflection checks are required, then 2 parameter blocks with code checks are required, one with a TRACK 4 command and one with a TRACK 0, 1, or 2, thus:

```
LOAD LIST 1 TO 10
PARAMETER 1
CODE EN 1993
TRACK 2 ALL
CHECK CODE MEMBER 1
***************************
LOAD LIST 100 TO 110
PARAMETER 2
TRACK 4 ALL
DFF 300 MEMB 1
DJ1 1 MEMB 1
DJ2 4 MEMB 1
CODE MEMB 1
```

Note: While both sets of code checks will be reported in the output file, only the last code check results are reported in the GUI.

D5.B.7 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per DD ENV 1993-1-1:1992. Code checking is done using the forces and moments at specific sections of the members.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition ; the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in D3.B.4 Built-In Steel Section Library (on page 1380) or any of the user defined sections as described in G.6.3 User-Provided Steel Table (on page 2117) , with two exceptions; `GENERAL` and `ISECTION`. The EC3 DD design module does not consider these sections or `PRISMATIC` sections in its design process.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D5.B.8 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in D5.B.7 Code Checking (on page 1476).

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as the limitations specified in section 5.B.7(A).

D5.B.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

MEMBER	refers to the member number for which the design is performed.
TABLE	refers to steel section name, which has been checked against the steel code or has been selected.
RESULTS	prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
CRITICAL COND	refers to the clause in DD ENV 1993-1-1:1992 code which governs the design.
RATIO	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
LOADING	provides the load case number, which governed the design.
FX, MY, and MZ	provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.
LOCATION	specifies the actual distance from the start of the member to the section where design forces govern.

Note: For a `TRACK 2` output, the module will also report all the relevant clause checks that have been performed and will also indicate the critical ratio and the load case that caused the critical ratio as well as the corresponding forces that were used for the respective checks. A TRACK 2 output will also include the various design data used for the calculations such as the section modulii, section class, section capacity etc.

D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]

STAAD.Pro is capable of performing steel design based on the European code EC3 BS EN 1993-1-1:2005 *Eurocode 3: Design of steel structures Part 1.1 General rules and rules for buildings*.

Note: The implementation of EN1993-1-1:2005 includes the amendments as per CEN corrigenda of February 2006 and April 2009.

D5.C.1 General Description

The main steps in performing a design operation are:

1. Selecting the applicable load cases to be considered in the design process.
2. Providing appropriate “Parameter” values if different from the default values.
3. Specify whether to perform code-checking and/or member selection.

These operations can be repeated by the user any number of times depending on the design requirements. The “Parameters” referred to above provide the user with the ability to allocate specific design properties to individual members or member groups considered in the design operation.

D5.C.1.1 Eurocode 3 - EN 1993-1-1:2005 (EN 1993)

The EN 1993 version of Eurocode 3, *Design of steel structures, Part 1.1 General rules and rules for buildings* (EN 1993) provides design rules applicable to structural steel used in buildings and civil engineering works. It is based on the ultimate limit states philosophy that is common to modern standards. The objective of this method of design is to ensure that possibility of failure is reduced to a negligible level. This is achieved through application of safety factors to both the applied loads and the material properties.

The code also provides guidelines on the global methods of analysis to be used for calculating internal member forces and moments. STAAD uses the elastic method of analysis which may be used in all cases. Also there are three types of framing referred to in EC3. These are “Simple”, “Continuous”, and “Semi-continuous” which reflect the ability of the joints to developing moments under a specific loading condition. In STAAD only “Simple” and “Continuous” joint types can be assumed when carrying out global analysis.

D5.C.1.2 National Annex Documents

Various authorities of the CEN member countries have prepared National Annex Documents to be used with EC3. These documents provide alternative factors for loads and may also provide supplements to the rules in EC3.

The current version of EC3 (EN 1993) implemented in STAAD.Pro adheres to the factors and rules provided in EN 1993-1-1:2005. STAAD.Pro includes the following National Annexes:.

a. British National Annex [NA to BS EN 1993-1-1:2005]
b. The Dutch National Annex [NEN-EN 1993-1-1/NB] and
c. Norwegian National Annex [NS-EN 1993-1-1:2005/NA2008]
d. French National Annex [Annexe Nationale a la NF EN 1993-1-1:2005]
e. Finnish National Annex [SFS EN 1993-1-1:2005]
f. Polish National Annex [PN EN 1993-1-1:2005]

g. Singaporean National Annex [SS EN 1993-1-1:2005]
h. Belgian National Annex [NBN EN 1993-1-1 ANB:2018]
i. Malaysian National Annex [MS EN 1993-1-1]
j. German National Annex [DIN EN 1993-1-1:2005]
k. Swedish National Annex [BFS EN 1993-1-1:2005]

The choice of a particular National Annex is based on the value of a new NA parameter that is set when you specify the EN 1993 version of Eurocode 3. Refer to D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516) for a description of the NA parameter.

Axes convention in STAAD.Pro and Eurocode 3

By default, STAAD.Pro defines the major axis of the cross-section as Z-Z and the minor axis as Y-Y. A special case where Z-Z is the minor axis and Y-Y is the major axis is available if the `SET Z UP` command is used and is discussed in TR.5 Set Command Specification (on page 2206). The longitudinal axis of the member is defined as X and joins the start joint of the member to the end with the same positive direction.

Eurocode 3, however, defines the principal cross-section axes in reverse to that of STAAD.Pro, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

Bear this difference in mind when examining the code-check output from STAAD.Pro.

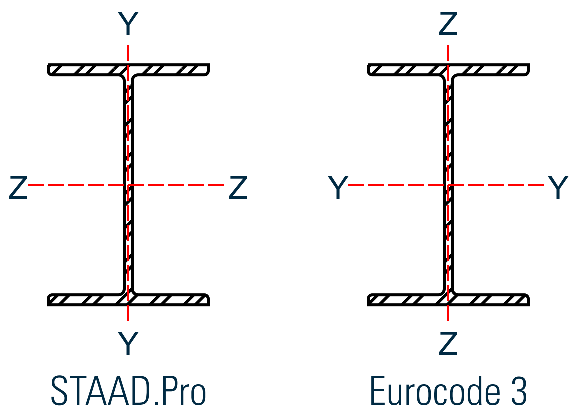


Figure 164: Axis convention in STAAD.Pro and EC3

Refer to D5.C.9 Tabulated Results of Steel Design (on page 1514) for an example of how this appears when Y is up (default).

D5.C.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations.

D5.C.3 Material Properties and Load Factors

The characteristic yield strength of steel used in EC3 (EN 1993) design is based on table 3.1 of the code. Design resistances are obtained by dividing the characteristic value of a particular resistance by the global partial safety factor for the resistance, γ_m. The magnitude of γ_m is based on Cl. 6.1 of EN 1993-1-1:2005 and can change depending on the selected National Annex.

Material coefficients for steel in STAAD.Pro take the following default values unless replaced by user's numerical values provided in the input file.

Modulus of Elasticity, E = 205,000 N/mm^2
Shear Modulus, G = E/2(1+ ν)
Poisson's Ratio, ν = 0.3
Unit weight, Γ = 76.8 KN/m^3

The magnitude of design loads is dependent on γ_f, the partial safety factor for the action under consideration. You are allowed total control in providing applicable values for the factors and their use in various load combinations.

D5.C.4 Section Classification

The occurrence of local buckling of the compression elements of a cross-section prevents the development of full section capacity. It is therefore imperative to establish this possibility prior to determining the section capacities. Cross sections are classified in accordance with their geometrical properties and the stress pattern on the compression elements. For each load case considered in the design process, the program determines the section class and calculates the capacities accordingly. It is worth noting that the section class reported in the design output corresponds to the most critical loadcase among those being considered for design.

The EC3 (EN 1993) design module in STAAD.Pro can design members with all section profiles that are of Class 1, 2, or 3 as defined in section 5.5 of the code. However, the design of members that have a Class 4 section profile are limited to:

- wide flange (with equal flanges)
- tee
- single channel
- single angle
- rectangular hollow sections
- circular hollow sections

Also built-up user sections that are class 4 sections are not dealt with in the current version of EC3 design in STAAD.Pro, unless they are defined as any of the section types given above.

The design of laced and battened members is not considered in the current version of EC3 (EN 1993) design module in STAAD.Pro. The current version also does not support the design of tapered section profiles or I-Sections with top and/or bottom plates or I sections with unequal flanges.

D5.C.5 Member Design

EN 1993-1-1:2005, together with any specified National Annex, is used for code check or member selection. However, where EN 1993 or the National Annex has *not* specified a method or values for a specific clause or parameter, STAAD.Pro uses Non-Contradictory Complementary Information (NCCI) documents as explained in the following corresponding sections.

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection.

The following sections describe the salient features of the design approach. In STAAD.Pro, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Member selection is done on the basis of selecting the most economic section on the basis of the least weight criteria. It is generally assumed that you (the engineer) will take care of the detailing requirements, such as the provision of stiffeners, and check the local effects like flange buckling, web crippling, etc.

Note: The design of class 4 (slender) sections is limited to:

- wide flange (with equal flanges)
- tee
- single channel
- single angle
- rectangular hollow sections
- circular hollow sections

The effective section properties are evaluated as described in Cl. 6.2.2.5 of the code. Refer to D5.C.4 Section Classification (on page 1480) for additional details.

Tapered member design is limited to I-sections with a tapered web, rectangular hollow sections, and circular sections. The additional design checks for bending required for these members is described in Tapered Members (on page 1484). Tapered member design is limited to EN 1993-1-1 only (i.e., no National Annex checks are considered for these members). Haunch members as described in EN 1993-1-1 are not designed in STAAD.Pro.

You are allowed complete control over the design process through the use of the parameters listed in D5.C.6 Design Parameters (on page 1502). Default values of parameters will yield reasonable results in most circumstances. However, you should control the design and verify results through the use of the design parameters.

Related Links

- *V. EC3 - Pinned column using non-slender UKC section* (on page 4059)
- *V. EC3 - Simply supported laterally unrestrained beam* (on page 4068)

D5.C.5.1 Members Subject to Axial Loads

The cross section capacity of tension only members is checked for ultimate limit state as given in Cl. 6.2.3 of the code.

Compression members will be checked for axial capacity of the cross section in addition to lateral buckling/ stability. The cross section capacity will be checked as given in section 6.2.4 of the code.

Lateral stability of a pure compression member will be checked as per the method given in Cl. 6.3 of the code. The compression member stability will be verified as:

$$\frac{N_{Ed}}{N_{b,Rd}} \leq 1.0$$

Where $N_{b,Rd}$ is the design buckling resistance given by:

$$N_{b,Rd} = \frac{\chi A f_y}{\gamma_{M1}} \text{ for Class 1, 2, or 3 cross-sections}$$

$$N_{b,Rd} = \frac{\chi A_{eff} f_y}{\gamma_{M1}} \text{ for Class 4 cross-sections}$$

Where:

χ is the reduction factor as given in section 6.3.12 of the code. The buckling curves used to evaluate the reduction factor are selected from Table 6.2 of the code based on the cross section type and the steel grade.

> **Note:** *Only* the five grades of steel given in table 6.2 will be used when selecting the buckling curve. The steel grade used for this selection is based on the SGR design input parameter (Refer to D5.C.6 Design Parameters (on page 1502)). Even if you have specified a custom yield strength (using the PY parameter), the choice of a buckling curve will be based on the value of SGR parameter.

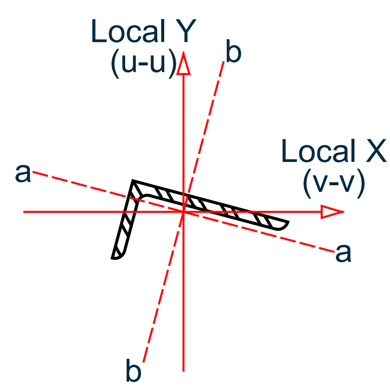
Compression members that are susceptible to torsional or torsional flexural buckling are checked for these modes of failure as well. The non-dimensional slenderness $\bar{\lambda}_T$ for these members is evaluated per Cl. 6.3.1.4 of the EN 1993 code. The maximum slenderness among the flexural buckling slenderness, torsional slenderness, and torsional-flexural slenderness is used to evaluate the reduction factor, χ, for such members. The elastic torsional buckling load, $N_{cr,T}$, and the elastic torsional-flexural buckling load, $N_{cr,TF}$, are evaluated based on the method given in the NCCI “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” (unless otherwise specified by a particular National Annex). The effective length for the members can be controlled using the KZ, KY, LZ and LY parameters. If these parameters are specified, the effective length will be calculated as KZ*LZ for length about the Z-Z axis and KY*LY for length about the Y-Y axis. By default, the effective length will be taken as the member length.

EN 1993-1-1:2005 does not specifically deal with single angle, double angles, double channels, or Tee sections and does not provide a method to evaluate the slenderness of such members. In these cases, the EC3 (EN 1993) design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 and Table 25 of BS 5950-1:2000 are used in the current version of the Eurocode 3 design module.

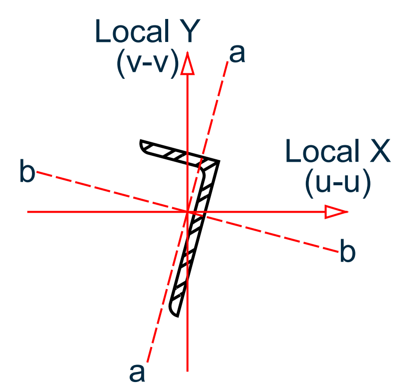
Single Angle Sections

Angle sections are unsymmetric and when using BS 5950:2000 table 25, you must consider four axes: two principal, u-u and v-v and two geometric, a-a and b-b. The effective length for the v-v axis, Lvv, is taken as the LVV parameter or $LY \cdot KY$, if not specified. The a-a and b-b axes are determined by which leg of the angle is fixed by the connection and should be specified using the LEG parameter, see section 5B.6 for more information on the LEG parameter. The effective length in the a-a axis is taken as $LY \cdot KY$ and the effective length in the b-b axis as $LZ \cdot KZ$.

The following diagram shows the axes for angles which have been defined with either an ST or RA specification and is connected by its longer leg (i.e., a-a axis is parallel to the longer leg).



ST angle and USER table angles



RA angle

D5.C.5.2 Members Subject to Bending Moments

The cross section capacity of a member subject to bending is checked as per Cl .6.2.5 of the code. The condition to be satisfied is:

$$\frac{M_{Ed}}{M_{c,Rd}} \leq 1.0$$

Where $M_{c,Rd}$ is the is the design resistance given by:

$$M_{c,Rd} = M_{pl,Rd} = \frac{W_{pl} f_y}{\gamma_{M0}}$$ for class 1 and 2 cross-sections

$$M_{c,Rd} = M_{el,Rd} = \frac{W_{el,min} f_y}{\gamma_{M0}}$$ for class 3 cross-sections

$$M_{c,Rd} = \frac{W_{eff,min} f_y}{\gamma_{M0}}$$ for class 4 cross-sections

Cross sectional bending capacity checks will be done for both major and minor axis bending moments.

Members subject to major axis bending will also be checked for lateral-torsional buckling resistance as per Section 6.3.2 of the code. The design buckling resistance moment $M_{b,Rd}$ will be calculated as:

$$M_{b,Rd} = \chi_{LT} W_y \frac{f_y}{\gamma_{M1}}$$

where

χ_{LT} = the reduction factor for lateral-torsional buckling. This reduction factor is evaluated per Cl. 6.3.2.2 or Cl 6.3.2.3 of the EN 1993 code depending on the section type. For I sections, the program will by default use Cl. 6.3.2.3 to evaluate χ_{LT} and for all other sections the program will resort to Cl 6.3.2.2. However, if a particular National Annex has been specified, the program will check if the National Annex expands on Cl.6.3.2.3 (Table 6.5) to include sections other than I sections. If so, the program will use Cl. 6.3.2.3 for the cross-section(s) included in Cl. 6.2.2.3 (or Table 6.5). For all other cases the program will use Cl. 6.3.2.2.

Note: You have the option to choose the clause to be used to calculate χ_{LT} through the `MTH` design parameter. Setting `MTH` to `0` (default value) will cause the program to choose Cl.6.3.2.3 for I Sections and Cl 6.2.3.2 for all other section types. As mentioned above, if the National Annex expands on Cl. 6.3.2.3 to include sections other than I Sections, the program will use Cl. 6.3.2.3 by default.

When using Cl. 6.3.2.3 to calculate χ_{LT}, the program will consider the correction factor kc (Table 6.6 of EN 1993-1-1:2006) based on the value of the `KC` parameter in the design input. By default the value of `KC` will be taken as 1.0. If you want the program to calculate kc, you must explicitly set the value of the `KC` parameter to zero.

Note: If the National Annex specifies a different method to calculate kc (e.g. the British, Singapore & Polish NAs), the program will use that method by default even if the KC parameter has not been explicitly set to zero. If the NA method does not deal with a specific condition while working out kc, the program will then fall back to table 6.6 of the code, thus ensuring that kc is considered for the particular NA.

The non-dimensional slenderness $\bar{\lambda}_{LT}$ (used to evaluate χ_{LT}) for both the above cases is evaluated as:

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}}$$

where

M_{cr} = the elastic critical moment for lateral-torsional buckling. EN 1993-1-1 does not however specify a method to evaluate M_{cr}. Hence, the program will make use of the method specified in Annex F of DD ENV 1993-1-1 to evaluate M_{cr} by default.

Note: The method specified in Annex F will be used only when the raw EN 1993-1-1:2005 code is used without any National Annex. If a National Annex has been specified, the calculation of M_{cr} (and $\bar{\lambda}_{LT}$) will be done based on the specific National Annex. (Refer to D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516) for specific details). If the National Annex does not specify a particular method or specify a reference document, the program will use the NCCI document SN-003a-EN-EU for doubly symmetric sections and SN030a-EN-EU for mono-symmetric sections that are symmetric about their weak axis. For all other sections types the program will use Annex F of DD ENV 1993-1-1 to calculate M_{cr}. In cases where Annex F does not provide an adequate method to evaluate M_{cr}, such as for Channel sections, the program will resort to the method as per Cl.4.3.6 of BS 5950-1:2000 to calculate the lateral-torsional buckling resistance moment ($M_{b,Rd}$) for the member.

Tapered Members

EN 1993-1-1 provides multiple methods for checking against lateral-torsional buckling in members with tapered I-shaped sections. The method given in Annex BB 3.2 of EN 1993-1-1 is used by STAAD.Pro. This method checks the unbraced length between lateral and torsional restraints against a calculated maximum length to ensure lateral-torsional stability. The tapered member is sub-divided into thirteen (13) analytical sections and bending design checks, including these for lateral-torsional buckling, are performed at each sub-section. While this approach is conservative for elastic analysis, it is necessary for plastic analysis.

The stable length between lateral restraints, L_m, is calculated as follows. This value must be greater than or equal to the design parameter LY.

$$L_m = 0.85 \frac{38 r_{zz}}{\sqrt{\frac{1}{57.4}\left(\frac{N_{\text{Ed}}}{A}\right) + \frac{1}{756 C_1^2}\left(\frac{W_{\text{pl,y}}^2}{A I_T}\right)\left(\frac{f_y}{235}\right)^2}} \quad \text{(Eqn. BB.5)}$$

where

r_{zz}	=	the radius of gyration about the major axis (notation i_z in EN 1993-1-1)
N_{Ed}	=	design value of compression force in the member
A	=	cross-sectional area of the member
$W_{pl,y}$	=	plastic section modulus of the member
I_T	=	torsional constant
f_y	=	yield strength
C_1	=	a factor depending on loading and end conditions; taken $= 1/k_c^2$, where k_c is taken from the KC parameter.

The stable length between torsional restraints, L_s, is calculated as follows. This value must be greater than or equal to the design parameter EFT.

$$L_s = 0.85 \frac{\sqrt{C_n} L_k}{c} \quad \text{(Eqn. BB.12)}$$

where

C_n = modification factor for non-linear moment gradient

$$= \frac{12 \times R_{max}}{\left[R_1 + 3R_2 + 4R_3 + 3R_4 + R_5 + 2(R_S - R_E)\right]} \leq 1.0 \quad \text{(Eqn. BB.14)}$$

R = moment ratio calculated at ends, quarter points, and mid-point of member segment between torsional restraints, calculated as:

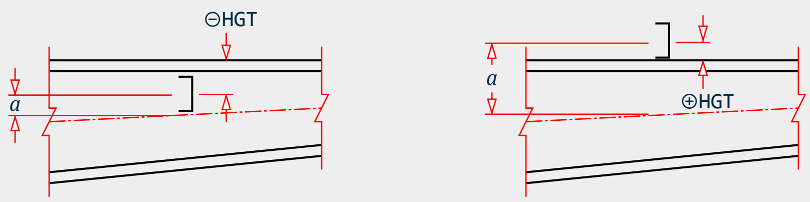
$$= \frac{M_{y,Ed} + aN_{Ed}}{f_y W_{pl,y}} \quad \text{(Eqn. BB.15)}$$

and R_1 is taken from the largest web depth.

$M_{y,Ed}$ = design bending moment about the Y axis

a = the distanced between the centroid of the member and the centroid of the restraining members (e.g., purlins).

> **Note:** This value is controlled in STAAD.Pro using the HGT parameter. To simplify the user input, the HGT parameter is specified in relation to the top of the member.



Purlin Below Top of Steel: HGT Parameter is Negative Value

Purlin Above Top of Steel: HGT Parameter is Positive Value

L_k = stable length between adjacent torsional restraints

$$= \frac{\left(5.4 + \frac{600 f_y}{E}\right)\left(\frac{h}{t_f}\right) r_{zz}}{\sqrt{5.4\left(\frac{f_y}{E}\right)\left(\frac{h}{t_f}\right)^2 - 1}} \quad \text{(Eqn. BB6)}$$

and $\frac{h}{t_f}$ is taken from the shallowest web depth.

E = modulus of elasticity

c = taper factor:

$$= 1 + \frac{3}{\left(h/t_f - 9\right)}\left(\frac{h_{max}}{h_{min}} - 1\right)^{2/3} \quad \text{(Eqn. BB.16)}$$

h = depth of segment

t_f = thickness of the flange

h_{max}, h_{min} = the maximum and minimum depth of the cross-section within the length, L_y (LY), respectively.

> **Note:** There are no provisions for lateral-torsional buckling in tapered hollow sections (i.e., tapered square or circular sections). As this is typically not a governing limit state, STAAD.Pro does not perform any such check.

Related Links

- *D5.C.6 Design Parameters* (on page 1502)

D5.C.5.3 Members Subject to Shear

The cross section capacity of a member subject to shear is checked as per Cl. 6.2.6 of the code. The condition to be satisfied is:

$$\frac{V_{Ed}}{V_{c,Rd}} \leq 1.0$$

where

$V_{c,Rd}$	=	the is the shear design resistance given by: $V_{c,Rd} = V_{pl,Rd} = \frac{A_v(f_y/\sqrt{3})}{\gamma_{M0}}$
A_v	=	the shear area and is worked out for the various section types as given in Cl. 6.2.6(3) of the code

Shear Buckling

For sections that are susceptible to shear buckling, the program will perform the shear buckling checks as given in Section 5 of EN 1993-1-5. The shear buckling checks will be done *only* for I –Sections and Channel sections. Shear stresses induced from torsional loads are taken into account while performing torsion checks.

Note: Web shear buckling is checked in STAAD.Pro V8i (SELECTseries 3) (release 20.07.08) and later.

The susceptibility of a section to shear buckling will be based on the criteria given in Cl 5.1(2) of EN 1993-1-5 as is as given as follows:

a. For unstiffened webs, if $h_w / t > 72 \times \varepsilon / \eta$, the section must be checked for shear buckling.

The design resistance is calculated as:

$$V_{b,Rd} = V_{bw,Rd} \leq \frac{\eta f_{yw} h_w t}{\sqrt{3}\gamma_{M1}}$$

$$V_{bw,Rd} = \frac{\chi_w f_{yw} h_w t}{\sqrt{3}\gamma_{M1}}$$

where

h_w	=	distance between flanges of an I Section (i.e., depth - 2x flange thickness)
t	=	thickness of the web
ε	=	$\sqrt{(235/f_y)}$, where fy is the yield stress
η	=	1.2 for steel grades up to and including S 460 and = 1.0 for other steel grades
k_τ	=	as defined in sections below
χ_w	=	the web contribution factor obtained from Table 5.1 of the EC3 code and is evaluated per the following table:

Table 131: Evaluate of χ_w

Slenderness Parameter	Rigid End Post	Non-rigid End Post
$\bar{\lambda}_w < 0.83 / \eta$	η	η
$0.83 / \eta \leq \bar{\lambda}_w < 1.08$	$0.83 / \bar{\lambda}_w$	$0.83 / \bar{\lambda}_w$

Slenderness Parameter	Rigid End Post	Non-rigid End Post
$\bar{\lambda}_w \geq 1.08$	$\frac{1.37}{0.7 + \bar{\lambda}_w}$	$0.83 / \bar{\lambda}_w$

$\bar{\lambda}_w$ = $\frac{h_w}{86.4 \times t \times \varepsilon}$

b. For stiffened webs, if $h_w / t > 31 \times E\sqrt{k_\tau / \eta}$, the section must be checked for shear buckling.

The design resistances considers tension field action of the web and flanges acting as struts in a truss model. This is calculated as:

$$V_{b,Rd} = V_{bw,Rd} + V_{bf,Rd} \leq \frac{\eta f_{yw} h_w t}{\sqrt{3}\gamma_{M1}}$$

Where:

where

$V_{bf,Rd}$ = the flange resistance per Cl.5.4 for a flange not completely utilized by bending moment

$V_{bf,Rd}$ = $\frac{h_f t_f^2 f_{yf}}{c\gamma_{M1}}\left[1 - \left(\frac{M_{Ed}}{M_{f,Rd}}\right)^2\right]$

b_f = the width of the flange which provides the least axial resistance, not to be taken greater than $15\varepsilon t_f$ on each side of the web

t_f = the thickness of the flange which provides the least axial resistance

$M_{f,Rd}$ = $M_{f,k}/\gamma_{M0}$, the moment of resistance of the cross section consisting of the effective area of the flanges only. For a typical I Section or PFD, this is evaluated as $b \cdot t_f \cdot h_w$. When an axial load, N_{Ed}, is present, the value of $M_{f,Rd}$ is reduced by multiplying by the following factor:

$$1 - \frac{N_{Ed}}{\left[\frac{(A_{f1} + A_{f2}) f_{yf}}{\gamma_{M0}}\right]}$$

A_{f1} ,A_{f2} = the areas of the top and bottom flanges, respectively

c = $a\left(0.25 + \frac{1.6 b_f t_f^2 f_{yf}}{t h_w^2 f_{yw}}\right)$

a = transverse stiffener spacing. The equation of c is likewise used to solve for a sufficient stiffener spacing in the case of demand from loads exceeding the calculated capacity for a specified stiffener spacing

The following equation must be satisfied for the web shear buckling check to pass:

$$\eta_3 = \frac{V_{Ed}}{V_{b,Rd}} \leq 1.0$$

where

V_{Ed} = the design shear force

Note: The shear forces due to any applied torsion will *not* be accounted for if the TOR parameter has been specifically set to a value of 0 (i.e., ignore torsion option).

If the stiffener spacing has not been provided (using the `STIFF` parameter), then the program assumes that the member end forms a non-rigid post (case c) and proceeds to evaluate the minimum stiffener spacing required.

D5.C.5.4 Members Subject to Torsion

Note: This feature requires STAAD.Pro V8i (SELECTseries 2) build 2007.07 or later.

General

Eurocode 3 (EN 1993-1-1:2005) gives very limited guidance for the analysis and design of torsion members. While both elastic and plastic analyses are permitted generally, the design analysis methods for torsion discussed within EC3 are primarily based on elastic methods. Also, only the first yield design resistance is specifically discussed for torsion members. Furthermore, there is no guidance on section classification nor on how to allow for the effects of local buckling on the design resistance for combined torsional effects. EC3 also does not specifically deal with members subject to combined bending and torsion and loosely states that the yield criteria (Eqn 6.1 in the code) can be used for elastic verification.

The method used by STAAD.Pro is therefore based on the SCI publication “P057: Design of members subject to combined bending and torsion”. Though this publication is based on the British standard BS 5950-1, the principles from this document are applied in the context of Eurocode 3.

Note: At the time this feature has been implemented in STAAD.Pro, SCI are in the process of updating document P057 to be in accordance with Eurocode 3. Hence this method might be subject to modifications subject to the publication of a newer version of P057. The NCCI document “SN007b-EN-EU: Torsion” will also be referenced where appropriate.

Code Basis

Torsion design in EC3 is given in Cl. 6.2.7 of EN 1993-1-1:2005. Therefore, this clause is used primarily for this implementation.

EN 1993-1-1:2005 does not deal with members subject to the combined effects of torsion and lateral torsional buckling. However, EN 1993-1-6 considers such a condition in Appendix A. Therefore, STAAD.pro uses Appendix A of EN 1993-1-6 to check for members subject to combined torsion and LTB.

The following clauses from EC3 are then considered:

- Cl. 6.2.7(1)
- Cl. 6.2.7(9)
- Cl. 6.2.7(5)
- EC-3 -6 App A

Note: STAAD.Pro does, however, use this clause (6.2.7) to report the output for all torsion checks. Also any distortional deformations and any amplification in the torsional or shear stresses due to distortions will be neglected by the program.

- Clause 6.2.7(1)

  States that for members subject to torsion, the design torsional moment T_{Ed} at each cross section should satisfy:

  $$T_{Ed} / R_{Rd} \le 1.0$$

  Where:

  T_{Rd} is the design torsional resistance of the cross section.

This is the primary condition that will need to be satisfied for members subject to torsion. The method for working out the torsional resistance T_{Rd}, for the various cases is dealt in the following sections.

- Cl. 6.2.7(9)

  States that:

  > For combined shear force and torsional moment, the plastic shear resistance accounting for torsional effects should be reduced from $V_{pl,Rd}$ to $V_{pl,T,Rd}$ and the design shear force should satisfy:
  >
  > $V_{Ed} / V_{pl,T,Rd} \leq 1.0$

  The code also gives means to evaluate $V_{pl,T,Rd}$ in equations 6.26 to 6.28. These equations, however, only deal with I/H sections, Channel sections, and structural hollow sections (RHS, SHS, CHS). Therefore, the application of Cl. 6.2.7(9) is only performed for these section profiles.

- Cl 6.2.7(5)

  States that the yield criteria given in Cl. 6.2.1(5) of EN 1993-1-1:2005 may be used for elastic verification. STAAD.Pro evaluates the stresses due to the various actions on the cross section and applies this yield criterion.

The program allows for two types of checks for members subject to torsion for EC3 design:

I. Basic Stress Check: This method is intended to be a simplified stress check for torsional effects. This method will produce the output corresponding to Cl. 6.2.7(5) of EN 1993-1-1.
II. Detailed Checks: This method will perform a full torsional analysis of the member. All four of the clause checks mentioned earlier will be performed.

The details of these checks are as described below.

You have the option to choose the method to be used for a specific member or group of members. This will be facilitated by setting the value of the `TORSION`. The `TORSION` parameter set to zero by default, which results in torsion checks only being performed if the member is subject to torsional moments (i.e., for this default setting, the program will ignore torsion checks if there is no torsional moment in the member). Setting the value of the `TORSION` parameter to three (3) will cause the program to ignore all torsional moments. The detailed output (i.e., `TRACK 2`) will indicate that torsion has been ignored for that particular member. The details of setting the values to one (1) or two (2) and the corresponding checks performed are as described below. Refer to D5.C.6 Design Parameters (on page 1502) for additional details.

Note: If the `TORSION` parameter is set to 1 or 2, the program will perform the appropriate checks even if the member is not subject to torsional moments. In such cases, the program will perform the checks with a value of zero for the torsional moment.

D5.C.5.4.1 Basic stress check
This method is used when the `TORSION` parameter is specified as one (1).

This method is intended to be a simplified stress check for torsional effects per Cl. 6.2.7(5). Any warping stresses that may develop due to the end conditions will be ignored for this option. The program will consider the forces (including torsion) at various sections along the length of the member and for each section, will calculate the resultant stress (Von Mieses) at various points on the cross section. The location and number of points checked for a cross section will depend on the cross section type and will be as described below.

The stress check will be performed using equation 6.1 of EN 1993-1-1:2005 as given below:

$$\left(\frac{\sigma_{x,Ed}}{f_y/\gamma_{M0}}\right)^2 + \left(\frac{\sigma_{z,Ed}}{f_y/\gamma_{M0}}\right)^2 - \left(\frac{\sigma_{x,Ed}}{f_y/\gamma_{M0}}\right)\left(\frac{\sigma_{z,Ed}}{f_y/\gamma_{M0}}\right) + 3\left(\frac{\tau_{Ed}}{f_y/\gamma_{M0}}\right)^2 \leq 1$$

where

$\sigma_{x,Ed}$	=	the longitudinal stress
$\sigma_{z,Ed}$	=	the transverse stress and
τ_{Ed}	=	the resultant shear stress.

Note: Since transverse stresses are very small under normal loading conditions (excluding hydrostatic forces), the term will be negligible and hence is taken as zero.

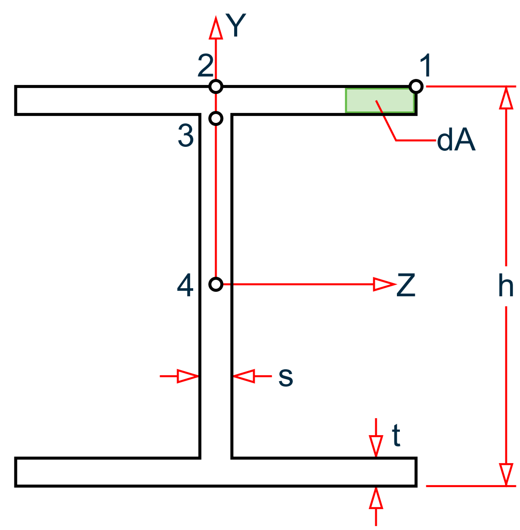
$$\sigma_{x,Ed} = \sigma_x + \sigma_{bz} + \sigma_{by} = F_x/A_x + M_z/Z_z + M_y/Z_y$$

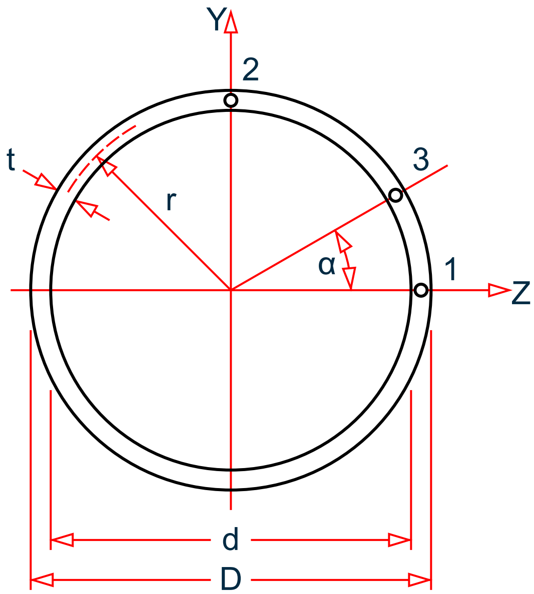
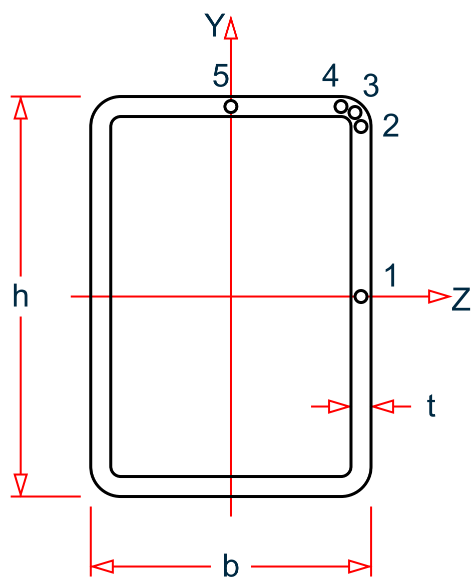
$$\tau_{Ed} = T/J \cdot t + V_y \cdot Q/(I_z \cdot t) + V_z \cdot Q/(I_y * t)$$

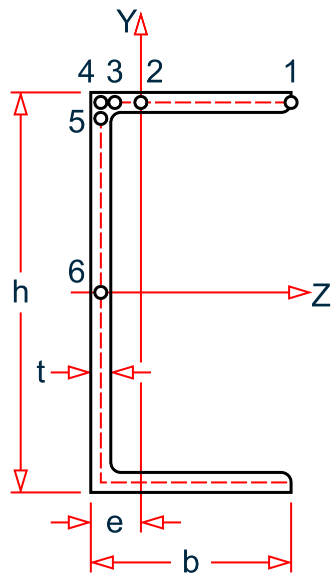
where

T	=	the torsion at the particular section along the length of the member
J	=	the torsion constant
t	=	the thickness of the web/flange
V	=	the shear force
Q	=	the statical moment about the relevant axis
I	=	the second moment of area about the relevant axis

The stress check as per equation 6.1 is performed at various stress points of a cross section as shown in figures below:

Shape	Section Sketch
Doubly symmetric wide flange profile	

Shape	Section Sketch
Pipe profiles $\alpha = \tan^{-1}(M^z/M_y)$	
Tube profiles	

Shape	Section Sketch
Channel profiles	

The resultant ratio will be reported under Cl. 6.2.7(5) in the detailed design output.

D5.C.5.4.2 Detailed stress check

This method is used when the `TORSION` parameter is specified as two (2).

This method performs a detailed torsional analysis of a member depending on the torsion loading conditions and the support conditions at the member ends. This method is based on the SCI publication P057 and includes any warping stresses (direct warping stresses and warping shear stresses) depending on the end conditions of the member. This implementation considers seven different cases of loading and end conditions as given in publication P057 – Section 6. The loading/end conditions for a member are specified by the use of the `CMT` design parameter (Refer to D5.C.6 Design Parameters (on page 1502) for parameter values and descriptions).

All the equations used to evaluate the torsional moments and associated stresses are as given in Appendix B of P057. The resultant stresses are evaluated at various sections along the length of the member and the following checks will be performed:

Clause 6.2.7(1) – Torsional resistance of the section.

In general, the torsion at any section T_{Ed} is resolved into two components, viz.

The pure torsional (St. Venant's) moment ($T_{t,Ed}$) and

The warping torsional moment($T_{w,Ed}$)

Therefore,

$$T_{Ed} = T_{t,Ed} + T_{w,Ed} = GJ\varphi' = EH\varphi''' \qquad \text{[Ref SCI pub. P057]}$$

where

φ' and φ'''	=	the first and third derivatives of twist (φ), respectively, and depend on the end conditions and loading. These are evaluated from the equations in Annex B of P057 and are based the specified `CMT` parameter.

Note: Although the equation given the NCCI document SN007b-EN-EU can be used to evaluate T_{wrd}, the NCCI does not give the eqn. to evaluate φ’’’. Therefore, Annex B of P057 is used.

The torsional resistance of the section is also considered as the sum of the pure torsion resistance and the warping torsion resistance. The pure torsion resistance ($T_{t,Rd}$) and the warping torsional resistance ($T_{w,Rd}$) are evaluated as:

For closed sections:

$$T_{t,Rd} = 2 \cdot A_c \cdot t \cdot \tau_{max}$$

where

A_c	=	the area enclosed by the mean perimeter
t	=	the max thickness
τ_{max}	=	the max. allowable shear stress = $(fy/\sqrt{3})/\Gamma m0$

For open sections (I & channel):

$$T_{t,Rd} = \tau_{max} \cdot J / t$$

where

J	=	the torsion const
t	=	the max thickness

$$T_{w,Rd} = (f_y / \Gamma_{m0}) \cdot t \cdot b^2 / 6$$

where

b	=	the width of the section
t	=	the thickness of the flange for I- sections; minimum of flange or web thickness channel sections

The check according to Cl 6.2.7(1) will then be performed to ensure that the following conditions are satisfied:

$$T_{t,Ed} / T_{t,Rd} \leq 1$$

$$T_{w,Ed} / T_{w,Rd} \leq 1$$

$$T_{Ed} / T_{Rd} \leq 1$$

Clause 6.2.7(9) – Plastic shear resistance due to torsion

STAAD.Pro checks for shear resistance of a section based on Cl. 6.2.6 for EC3 and the plastic shear resistance (in the absence of torsion) is evaluated as:

$$V_{pl,Rd} = \frac{A_v\left(f_y/\sqrt{3}\right)}{\gamma_{M0}}$$

where

A_v	=	as pre Cl.6.2.6 (3) for the various sections

When torsion is present, along with the shear force, the design shear resistance will be reduced to $V_{pl,T,Rd}$, where $V_{pl,T,Rd}$ is evaluated as follows:

i. For I or H Sections:

$$V_{pl,T,Rd} = \sqrt{1 - \frac{\tau_{t,Ed}}{1.25\left(f_y/\sqrt{3}\right)/\gamma_{M0}}}V_{pl,Rd}$$

ii. For Channel Sections:

$$V_{pl,T,Rd} = \left[\sqrt{1 - \frac{\tau_{t,Ed}}{1.25(f_y/\sqrt{3})/\gamma_{M0}}} - \frac{\tau_{w,Ed}}{(f_y/\sqrt{3})/\gamma_{M0}}\right] V_{pl,Rd}$$

iii. For Structural Hollow Sections:

$$V_{pl,T,Rd} = \left[1 - \frac{\tau_{t,Ed}}{(f_y/\sqrt{3})/\gamma_{M0}}\right] V_{pl,Rd}$$

where

$\tau_{t,Ed}$	=	the shear stress due to direct (St. Venant's) torsion
$\tau_{w,Ed}$	=	the shear stress due to warping torsion

The various shear stresses due to torsion $\tau_{t,Ed}$ and $\tau_{w,Ed}$ are evaluated as follows:

i. For Closed sections:

The shear stresses due to warping can be ignored as they will be insignificant and hence:

$\tau_{t,Ed} = T_{Ed}/(2 \cdot A_c \cdot t)$ [Ref NCCI Sn007b-EN-EU]

where

T_{Ed}	=	the applied torsion
A_c	=	the area delimited by the mean perimeter
t	=	the thickness of the cross section
$\tau_{w,Ed}$	=	0, since warping is ignored

ii. For Open sections [I, H, Channel] sections:

For I and H sections, the web will not be subject to warping stresses and therefore warping shear can be ignored ($\tau_{w,Ed}$=0).

The stress due to pure torsion is evaluated as:

$\tau_{t,Ed} = G \cdot t \cdot \varphi'$ [Ref SCI pub. P057]

where

G	=	the shear modulus
φ'	=	a function depending on the end condition and loading(T). This will be taken from section 6 and Annex B of P057.

Note: Although the maximum stress is at the thickest section of the profile, the program uses the web thickness for this clause (since the shear capacity is based on the web area) unless the load is parallel to the flanges, in which case the flange thickness is used.

For channel sections that are free to warp at the supports and, thus, are not subject to warping stresses:

The warping shear stress is evaluated as:

$\tau_{w,Ed} = E \cdot S_w \cdot \varphi''' / t$ [Ref SCI pub. P057]

where

E	=	the elastic modulus
S_w	=	the warping statistical moment
φ'	=	a function depending on the end condition and loading(T). This will be taken from section 6 and Annex B of P057.

Clause 6.2.7(5) – Check for elastic verification of yield

Eurocode 3 gives yield criterion as per eqn. 6.1 and STAAD.Pro uses the yield criterion given in EC-3. When a member is subject to combined bending and torsion, some degree of interaction occurs between the two effects. The angle of twist caused by torsion is amplified by the bending moments and will induce additional warping moments and torsional shears. Account must also be taken of the additional minor axis moments produced by the major axis moments acting through the torsional deformations, including the amplifications mentioned earlier.

For members subject to bending and torsion, the stresses are evaluated as follows:

Direct bending stress (major axis): $\sigma_{bz} = M_z \ / \ Z_z$
Direct bending stress (minor axis): $\sigma_{by} = M_y \ / \ Z_y$
Direct stress due to warping: $\sigma w = E \cdot W_{ns} \cdot \varphi''$
Direct stress due to twist (min. axis): $\sigma_{byt} = M_{yt} \ / \ Z_y$
Direct stress due to axial load (if any): $\sigma_c = P / \ A$

where

M_z	=	the major axis moment & My is the minor axis moment
φ''	=	the differential function based on twist (ref P057 Annex B. & Table 6)
W_{ns}	=	the normalized warping function
M_{yt}	=	$\varphi \cdot M_z$ (see Appendix B of P057 to evaluate φ)

Shear stresses due to torsion and/or warping is evaluated as described above for Clause 6.2.7(9).

Check for yield (capacity checks) is then done according to Eqn 6.1 of EN 1993-1-1:2005, as described for the Basic Stress Check (`TORSION` = 1):

$$\left(\frac{\sigma_{x,Ed}}{f_y / \gamma_{M0}}\right)^2 + \left(\frac{\sigma_{z,Ed}}{f_y / \gamma_{M0}}\right)^2 - \left(\frac{\sigma_{x,Ed}}{f_y / \gamma_{M0}}\right)\left(\frac{\sigma_{z,Ed}}{f_y / \gamma_{M0}}\right) + 3\left(\frac{\tau_{Ed}}{f_y / \gamma_{M0}}\right)^2 \le 1$$

Clause EC-3:6 App A – Check for combined Torsion and Lateral Torsional buckling

The interaction check due to the combined effects of bending (including lateral torsional buckling) and torsion will be checked using Annex A of EN 1993-6: 2007. Note that this interaction equation does *not* include the effects of any axial load.

Caution: At present, SCI advises that no significant work has been published for this case and work is still ongoing. So at present is advisable not to allow for torsion in a member with large axial load.

Members subject to combined bending and torsion will be checked to satisfy:

$$\frac{M_{y,ED}}{x_{LT} M_{y,RK} / \gamma_{M1}} + \frac{C_{MZ} M_{z,Ed}}{M_{z,RK} / \gamma_{M1}} + \frac{k_w k_{zw} k_a T_{w,Ed}}{T_{w,Rk} / \gamma_{M1}} \le 1$$

where

C_{mz}	=	the equivalent uniform moment factor for bending about the z-z axis, according to EN 1993-1-1 Table B.3.
k_w	=	$0.7 - \frac{0.2 T_{w,Ed}}{T_{w,Rk} / \gamma_{M1}}$
k_{zw}	=	$1 - \frac{M_{z,Ed}}{M_{z,Rk} / \gamma_{M1}}$
k_a	=	$\frac{1}{1 - M_{y,Ed} / M_{y,cr}}$
$M_{y,Ed}$ and $M_{z,Ed}$	=	the design values of the maximum moment about the y-y and z-z axis, respectively
$M_{y,Rk}$ and $M_{z,Rk}$	=	are the characteristic values of the resistance moment of the cross-section about it y-y and z-z axis, respectively, from EN 1993-1-1, Table 6.7

$M_{y,cr}$	=	the elastic critical lateral-torsional buckling moment about the y-y axis
$T_{w,Ed}$	=	the design value of the warping torsional moment
$T_{w,Rk}$	=	the characteristic value of the warping torsional resistance moment
χ_{LT}	=	the reduction factor for lateral torsional buckling according to 6.3.2 of EN 1993-1-1

Note: For all of the above checks the effective length of the member to be used for torsion can be set by using the `EFT` design parameter.

D5.C.5.5 Members Subject to Combined Forces

Members subject to Bending and Axial Force

When a member is subject to a combined axial load and a bending moment, the program evaluates a reduced moment capacity based on Cl. 6.2.9 of the code. For Class 1, 2, and 3 sections, the program evaluates the reduced moment from the equations given in Cl. 6.2.9.1 of the code. For class 4 sections, the interaction equation given by equation 6.44 are checked.

In the case of members subject to axial load and biaxial bending, the program will consider the interaction equation 6.41 of the code.

Note: By default, the program will use the values of the constants "α" and 'β' as given in the code for the different sections types. However, you can override these values using the `ALPHA` and `BETA` design parameters (Refer to D5.C.6 Design Parameters (on page 1502)).

Note: The program uses the parameter `ELB` (Refer to D5.C.6 Design Parameters (on page 1502)) to override the Cl.6.2.9 checks for combined axial load and bending case. When specfied as `1`, the program uses the more general equation 6.2 of EN 1993-1-1, instead.

Members subject to Bending, Shear, and Axial Force

When a member is subject to a combined axial load, shear force, and a bending moment, the program evaluates the reduced yield strength as given in Cl 6.2.10 (3) of the code. The reduction in the yield strength is done only when the applied shear force exceeds 50% of the design shear resistance $V_{pl,Rd}$. This reduced yield strength is then used to evaluate the reduced moment capacity of the section.

Members subject to Bending and Axial Compression

The bending resistance of members could be reduced by the presence of a co-existent axial load. This is then checked against the lateral-torsional buckling resistance of the section. The EN 1993 design module in STAAD takes such a scenario into account and performs the necessary checks as per Cl. 6.3.3 of the code.

Generally, EC3 requires checking cross-section resistance for local capacity and also checking the overall buckling capacity of the member. In the case of members subject to axial tension and bending, there is provision to take the stabilizing effect of the tension load into consideration. This is achieved by modifying the extreme compression fibre stress and calculating an effective applied moment for the section. The checks are done as per Cl. 6.2.9 of the code. In case of a combined axial compressive load and bending moment, the member is checked per the rules in section 6.3.3 of the code. The program checks to ensure that both the interaction equations 6.61 and 6.62 of the code are satisfied. The interaction factors kzz, kyy, kzy & kyz will be evaluated using Annex B of EN 1993-1-1 by default. Hence for the EN 1993-1-1 code in STAAD.Pro (without National Annexes), uses Annex B. The choice between using Annex A and Annex B will be based on the choice specified by a particular National

Annex, if used. If the National Annex itself gives a choice between Annex A and Annex B, the program uses Annex B to evaluate the interaction factors.

Note: EN 1993-1-1:2005 does not specifically deal with single angle, double angles, double channels or Tee sections and does give a method to evaluate the slenderness of such members. In these cases, the Eurocode 3 (EN 1993-1-1) design module of STAAD.Pro uses the methods specified in BS 5950-1:2000 to calculate the slenderness of these members. Cl. 4.7.10 of BS 5950-1:2000 is used in the current version of the EC3 design module. See D5.C.5.1 Members Subject to Axial Loads (on page 1481) for ST and RA angle specifications.

Note: Laced or battened compression members are *not* dealt within the current version of EC3 (EN 1993) design module in STAAD.Pro.

D5.C.5.6 Design of Slender pipe sections to EN 1993-1-6

The design of Slender CHS sections is performed per EN 1993-1-6:2007 (hereafter, EC3-6). EC3-6 does not specify additional or modified safety factors. Therefore, the program uses the default safety factors from EN 1993-1-1.

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters.

EC3-6 deals with four types of ultimate limits states: plastic limit state, cyclic capacity limit state, buckling limit state, and fatigue. The following are considered by STAAD.Pro:

- LS1 – Plastic limit state: Deals with the condition when the capacity of the structure is exhausted by yielding of the material.
- LS3 – Buckling Limit state: Deals with the condition in which the structure (or shell) develops large displacements normal to the shell surface, caused by loss of stability under compressive and/or shear membrane stresses.

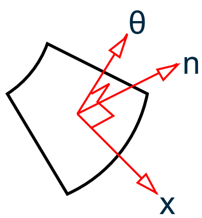
The limit state verification is made based on the “Stress design” method described in EC3-6. The stress design approach takes into account three categories of stresses:

- Primary stresses: Stresses that are generated for the member to be in equilibrium with the direct imposed loads.
- Secondary stresses: Those that are generated for internal compatibility or for compatibility at supports due to imposed loads or displacements (e.g., temperature, settlement etc.)
- Local stresses: Local stresses generated due to cyclic loading (or fatigue).

Only the primary stresses are considered the program. The primary stresses considered are those generated due to axial loads, bending, shear and /or a combination of these conditions.

Note: In the context of slender pipe section design for the Eurocode 3 module, the secondary and local stresses can be neglected since the loads and corresponding stresses dealt with in the design engine are largely direct and shear stresses.

The local axis coordinate system for a CHS is defined as:

	**circumferential**	around the circumference of the circular cross section (θ)
	meridional	along the length of the member (x)
	normal	perpendicular to the tangential plane formed by the circumferential and meridional directions (n)

and the corresponding membrane stresses will follow the convention given below:

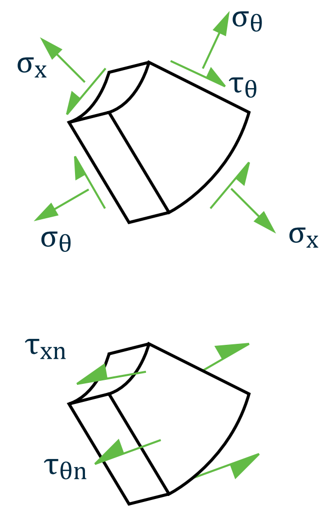


Figure 165: Nomenclature for membrane and transverse stresses in Slender CHS sections

Stress Design

Stress checks are made based on the "Stress design" method as per Section 8.5 of the code. This section deals with the buckling strength of the member (LS3). The principle is to evaluate the membrane stresses due to the applied loads and then compare that to the buckling strength, which is evaluated giving due consideration for local buckling effects.

The membrane stresses are evaluated as given in Annex A of the code. The pipe section is considered as an unstiffened cylindrical shell.

i. Meridional Stresses:

a. Axial load

$$F_x = 2 \cdot \pi \cdot r \cdot P_x$$

$$\sigma_x = -F_x/(2 \cdot \pi \cdot r \cdot t)$$

b. Axial stress from bending

$$M = \pi \cdot r^2 \cdot P_{x,max}$$

$$\sigma_x = \pm M/(\pi^2 \cdot r \cdot t)$$

ii. Shear Stress:

a. Transverse force, V

$$V = \pi \cdot r \cdot P_{\theta,max}$$

$$\tau_{max} = \pm V/(\pi \cdot r \cdot t)$$

b. Shear from torsional moment, M

$$M_t = 2\pi \cdot r^2 \cdot P_\theta$$

$$\tau = M_t/(2\pi^2 \cdot r^2 \cdot t)$$

Where:

r is the radius of the middle surface of the shell wall.
t is the wall thickness of the cylinder

Calculation of Axial Buckling Stress

The buckling strength of A slender pipe section is evaluated using the method given in section 8.5.2 ofEC3-6. The design buckling stresses (buckling resistance) are calculated separately for axial, circumferential, and shear. The circumferential stresses are ignored in STAAD.Pro.

The naming convention and the coordinate axis used will be as given in the following diagram:

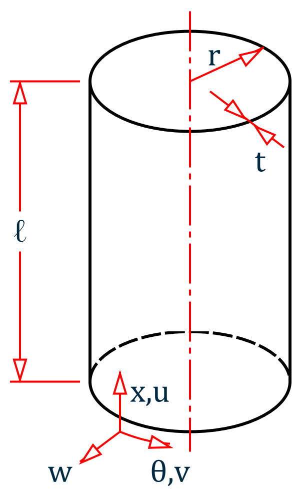


Figure 166: Naming convention and coordinate system used for the buckling stress of a slender CSH section

The axial buckling resistance is given by:

$$\sigma_{x,Rd} = \sigma_{x,Rk}/\gamma_{M1}$$

Note: Γ_{M1} will have the same default value of 1.0 as in EN 1993-1-1.

$\sigma_{x,Rk}$ is the characteristic buckling strength given by:

$$\sigma_{x,Rk} = X_x \cdot f_{yk}$$

Where:

χ_x is the meridional buckling reduction factor. χ_x is evaluated per Section 8.5.2(4) of EC3-6 and is determined as a function of the relative shell slenderness given by:

$$\lambda_x = \sqrt{\frac{f_{yk}}{\sigma_{x,cr}}}$$

Where:

$\sigma_{x,cr}$ is the elastic buckling critical stress.

Once the relative slenderness is evaluated, the reduction factor is calculated as follows:

$\chi = 1$ when $\bar{\lambda} \le \bar{\lambda}_0$

$\chi = 1 - \beta\left(\frac{\bar{\lambda} - \bar{\lambda}_0}{\bar{\lambda}_P - \bar{\lambda}_0}\right)^{\eta}$ when $\bar{\lambda}_0 < \bar{\lambda} < \bar{\lambda}_P$

$\chi = \alpha / \bar{\lambda}^2$ when $\bar{\lambda}_P \le \bar{\lambda}$

where

$\bar{\lambda}_P$ = the plastic limit for slenderness given by:

$$\sqrt{\frac{\alpha}{1 - \beta}}$$

The meridional buckling parameters the factors α and β are evaluated per section D.1.2.2 of EC3-6.

Note: A "Normal" fabrication quality will be assumed when evaluating the fabrication quality parameter as given in table D.2 of the code, unless the fabrication quality is set using the FAB design parameter. Refer to D5.C.6 Design Parameters (on page 1502)

The elastic critical buckling stress, $\sigma_{x,cr}$ and the factors α and β are evaluated per Annex D of EC3-6. The details are as given below:

The CHS section is classified based on the following criteria:

CHS Length Classification	Criteria
Short	$\omega \le 1.7$
Medium	$1.7 < \omega \le 0.5 \cdot r/t$
Long	$\omega > 0.5 \cdot r/t$

Where:

$$\omega = \frac{l}{\sqrt{rt}}$$

The elastic critical buckling critical stress is evaluated as:

$$\sigma_{x,Rcr} = 0.605 \cdot E \cdot C_x \cdot (t/r)$$

Where:

C_x is a factor dependent upon the CHS length classification as described in section D.1.2.1 of EC-3-6.
For a long cylinder, there are two separate methods that can be used to evaluate the C_x factor: Eqns D.9/10 and Eqn D.12. Initially the program evaluates C_x based on the maximum from equations D.9 and D.10. However, for long cylinders that satisfy the conditions in equation D.11, the program will also work out Cx based on equation D.12 and then choose the minimum obtained from D.12 and D.9/10.

Calculation of Shear Buckling Stress

The shear buckling resistance is given by:

$$\tau_{x\theta,Rd} = \tau_{x\theta,Rk}/\gamma_{M1}$$

Note: γ_{M1} will have the same default value of 1.0 as in EN 1993-1-1.

$\tau_{x\theta,Rk}$ is the characteristic buckling shear strength given by:

$$\tau_{x\theta,Rk} = X_\theta \cdot f_{yk}$$

Where:

χ_θ is the shear buckling reduction factor. χ_θ will be worked out as given in section 8.5.2(4) of En 1993-1-6 and is determined as a function of the relative shell slenderness given by:

$$\lambda_\theta = \sqrt{\frac{f_{yk}}{\tau_{x\theta,cr}}}$$

Where:

$\tau_{x\theta,Rk}$ is the elastic buckling critical stress.

The reduction factor, χ_θ, is then evaluated as described for the axial buckling stress, based on the same $\bar{\lambda}_p$, α, and β parameters given in Annex D of EC3-6.

The CHS section is classified based on the following criteria:

CHS Length Classification	**Criteria**
Short	$\omega \le 10$
Medium	$10 < \omega \le 8.7 \cdot r/t$
Long	$\omega > 8.7 \cdot r/t$

Where:

$$\omega = \frac{l}{\sqrt{rt}}$$

The elastic critical buckling critical stress is evaluated as:

$$\tau_{x\theta,Rcr} = 0.75 E C_\tau \sqrt{\frac{1}{\omega}} \left(\frac{t}{r}\right)$$

Where:

C_τ is a factor dependent upon whether the CHS length classification as described in section D.1.4.1 of EC-3-6. A “Normal” fabrication quality will be assumed when working out the fabrication quality parameter as given in table D.6 of the code, unless the fabrication quality is set using the FAB design parameter.

Buckling Strength Verification

The buckling strength verification will be performed so as to satisfy the following conditions:

For axial stresses:

$$\sigma_{x,Ed} \le \sigma_{x,Rd}$$

For shear stresses:

$$\tau_{x\theta,Ed} \leq \tau_{x\theta,Rd}$$

For a combined case of axial and shear stresses acting together, an interaction check will be done according to equation 8.19 of the code as below:

$$\left(\frac{\sigma_{x,Ed}}{\sigma_{x,Rd}}\right)^{k_x} + \left(\frac{\tau_{x\theta,Ed}}{\tau_{x\theta,Rd}}\right)^{k_\tau} \leq 1$$

Where:

k_x and k_τ are the interaction factors as given in section D.1.6 of EN 1993-1-6:

$$k_x = 1.25 + 0.75 \cdot \chi_x$$

$$k_\tau = 1.75 + 0.25 \cdot \chi_\tau$$

D5.C.6 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, you may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the n setting must be compatible with the active “unit” specification.

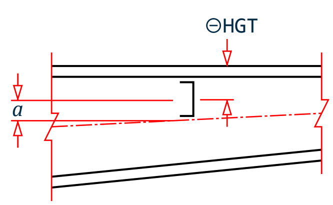
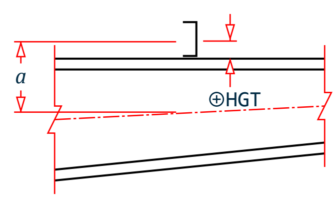
The follow table lists all the relevant EC3 parameters together with description and default values.

Table 132: Steel Design Parameters EC3 EN

Parameter Name	Default Value	Description
CODE	-	Must be specified as `EN 1993-1-1:2005` to invoke design per Eurocode 3:2005 (EN 1993). Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
ALH	0.5	The ratio of the distance of the point torque (from the start of the member) to the length of the member. The default value of 0.5 represents torque acting at the mid-span of a symmetrically loaded member. Values can range from 0 to 1.
ALPHA	1.0	Used to input a user defined value for the α factor in equation 6.41 for combined bending and axial force checks.

Parameter Name	Default Value	Description
BEAM	3	Parameter to control the number of sections to checked along the length of a beam: 1. Check at location of maximum Mz along beam 2. Check sections with end forces and forces at location of `BEAM 1.0` check. 3. Check at every 1/13th point along the beam and report the maximum
BETA	1.0	Used to input a user defined value for the β factor in equation 6.41 for combined bending and axial force checks.
C1	1.132	Corresponds to the C1 factor to be used to calculate Elastic critical moment M_{cr} as per Clause 6.3.2.2
C2	0.459	Corresponds to the C2 factor to be used to calculate Elastic critical moment M_{cr} as per Clause 6.3.2.2
C3	0	Corresponds to the C3 factor to be used to calculate Elastic critical moment M_{cr} as per Clause 6.3.2.2
CAN	0	Member will be considered as a cantilever type member for deflection checks. 0. indicates that member will not be treated as a cantilever member 1. indicates that the member will be treated as a cantilever member
CMM	1.0	Indicates type of loading and support conditions on member. Used to calculate the C1, C2, and C3 factors to be used in the M_{cr} calculations. Can take a value from 1 to 8. Refer to Note 2 (below) for more information on its use.
CMN	1.0	Indicates the level of End-Restraint. 1.0 = No fixity 0.5 = Full fixity 0.7 = One end free and other end fixed
CMT	1	Used to indicate the loading and support condition for torsion (ref. SCI publication P-057). Can take a value of 1-7. The values correspond to the various cases defined in section 6 and App. B of SCI-P-057. Refer to Note 4 (below) for more information

Parameter Name	Default Value	Description
DFF	360	"Deflection Length" / Max.. allowable local deflection See Note 1d below. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length". See Note 1 below.
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length". See Note 1 below.
DMAX	100.0 cm	Maximum allowable depth for the member.
DMIN	0	Minimum required depth for the member.
EFT	Member Length	The distance between the adjacent torsional restraints. Effective length for torsion, lateral-torsional buckling check of tapered members per Annex BB 3.2.2. A value of 0 defaults to the member length.
ELB	0	Used to specify the method for combined axial load + bending checks 0. Uses Cl. 6.2.9 of EN 1993-1-1:2005 1. Uses Cl. 6.2.1(7) - Eqn. 6.2 of EN 1993-1-1:2005
ESTIFF	0	(For use with the Dutch NA only) Method for checking columns forming part of (non)/buttressed framework: 0. Checks per Cl 12.3.1.2.3 of NEN 6770: Section 1 1. Checks per Cl 12.3.1.2.3 of NEN 6770: Section 2 Refer to D5.D.3.8 Clause 6.33 – Uniform members in bending and axial compression (on page 1526) for additional description on this parameter.
FAB	3	Used to specify the fabrication class to be used to check for slender (Class 4) CHS/pipe sections (EN 1993-1-6:2007) 1. Class A – Excellent 2. Class B – High 3. Class C – Normal
FU	0	Ultimate tensile strength of steel.
GM0	1.0	Corresponds to the γ_{m0} factor in EN 1993-1-1:2005
GM1	1.0	Corresponds to the γ_{m1} factor in EN 1993-1-1:2005

Parameter Name	Default Value	Description
GM2	1.25	Corresponds to the γ_{m2} factor in EN 1993-1-1:2005
GST	0	Used to specify the section type to be used for designing a "General Section" from the user table. The member will be considered as the specified type with the user defined properties. The available options and corresponding values are as below: 0. I-Section 1. Single Channel 2. Rectangular Hollow Section 3. Circular Hollow Section 4. Angle Section 5. Tee Section **Note:** This parameter will be ignored if it has been assigned to any section other than a General Section.
HGT	0.0	Distance from the top of the tapered web section to the center of gravity of purlin. • positive values when purlin is above the top flange • negative values when purlin is below the top flange  Purlin Below Top of Steel: HGT Parameter is Negative Value  Purlin Above Top of Steel: HGT Parameter is Positive Value Used for the lateral-torsional buckling calculations for web-tapered members.
KC	1.0	Corresponds to the correction factor as per Table 6.6 of EN 1993-1-1:2005. Program will calculate kc automatically if this parameter is set to `0`. **Note:** For the British, Singapore, & Polish NAs, kc will be calculated as given in the NA by default.
KY	1.0	K factor in local y axis. Used to calculate the effective length for slenderness and buckling calculations.
KZ	1.0	K factor in local z axis. Used to calculate the effective length for slenderness and buckling calculations.

Parameter Name	Default Value	Description
LEG	0	Slenderness values for angles as determined from BS 5950-2000 Table 25. Refer to D3.B.6 Design Parameters (on page 1384)
LVV	Max. value of Lyy	Leg length for Lvv (length about v-v- axis of single angle section), as per Lyy. Used for slenderness calculations.
LY	Member Length	Compression length in local y axis between lateral restraints $$\text{Slenderness ratio} = \texttt{KY}\times\texttt{LY} / \text{Ryy}$$ Distance between lateral restraints used for tapered I sections. Used for the lateral-torsional buckling calculations for web-tapered members. The default is the member's length.
LZ	Member Length	Compression length in local z axis between lateral restraints $$\text{Slenderness ratio} = \texttt{KZ}\times\texttt{LZ} / \text{Rzz}$$
MTH	0	Used to select the clause to be used to calculate the LTB reduction factor, χ_{LT}. The available options and corresponding values are as below: 0. Use default method based on section type (default) 1. Use Cl.6.3.2.2 2. Use Cl.6.3.2.3 By default, the program will use Cl 6.3.2.3 for rolled & built-up I-sections and Cl. 6.3.2.2 for all other sections. If, however, the specified National Annex expands on Cl. 6.3.2.3 to include other section types (e.g., the UK NA), the program will use Cl. 6.3.2.3 by default for that particular section type. Refer to D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516) for additional details on NA documents.
MU	0	The ratio of the moment due to the transverse force to the maximum moment M. To be used with `CMM` values of 7 and 8. See Note 2 (below). **Note:** This parameter is only applicable to the French NA & Belgian NA.
NA	0	Choice of National Annex to be used for EC3 design. Refer to D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005] (on page 1516) for values allowed for this parameter. (Refer to D5.C.1 General Description (on page 1478) for more information)
NSF	1.0	Net tension factor for tension capacity calculation.

Parameter Name	Default Value	Description
PLG	0	To be used to determine whether to include the additional interaction checks as per CL. NA.20(2) and NA.20(3) of the D5.D.8 Polish National Annex to EC3 (on page 1550). **Note:** This parameter is only applicable to the Polish NA.
PY	Yield Strength	The yield strength default value is set based on the default value of the SGR parameter.
RATIO	1	Permissible ratio of loading to capacity.
SBLT	0.0	Indicates if the section is rolled or built-up. 0.0 = Rolled 1.0 = Built-up 2.0 = Cold-formed (uses the appropriate buckling curve from Table 6.2)

Parameter Name	Default Value	Description
SGR	0	Steel grade as in Table 3.1: EN 1993-1-1: 2005: 0.0 - indicates S 235 grade steel - EN10025-2 1.0 - indicates S 275 grade steel 2.0 - indicates S 355 grade steel 3.0 - indicates S 450 grade steel 4.0 - indicates S 275 N/NL grade steel - EN10025-3 5.0 - indicates S 355 N/NL grade steel 6.0 - indicates S 420 N/NL grade steel 7.0 - indicates S 460 N/NL grade steel 8.0 - indicates S 275 M/ML grade steel - EN10025-4 9.0 - indicates S 355 M/ML grade steel 10.0 - indicates S 420 M/ML grade steel 11.0 – indicates S 460 M/ML grade steel 12.0 – indicates S 235 W grade steel - EN10025-5 13.0 – indicates S 355 W grade steel 14.0 – indicates S 460 Q/QL/QL1 grade steel - EN10025-6 15.0 – indicates S 235 H grade steel - EN10210-1 16.0 – indicates S 275 H grade steel 17.0 – indicates S 355 H grade steel 18.0 – indicates S 275 NH/NLH grade steel 19.0 – indicates S 355 NH/NLH grade steel 20.0 – indicates S 420 NH/NLH grade steel 21.0 – indicates S 460 NH/NLH grade steel 22.0 – indicates S 235 H grade steel - EN10219-1 23.0 – indicates S 275 H grade steel 24.0 – indicates S 355 H grade steel 25.0 – indicates S 275 NH/NLH grade steel 26.0 – indicates S 355 NH/NLH grade steel 27.0 – indicates S 460 NH/NLH grade steel 28.0 – indicates S 275 MH/MLH grade steel 29.0 – indicates S 355 MH/MLH grade steel 30.0 – indicates S 420 MH/MLH grade steel 31.0 – indicates S 460 MH/MLH grade steel **Note:** As EN 1993-1-1:2005 does not provide a buckling curve in table 6.2 for grade S 450 steel (in Table 3.1 of EN 1993-1-1:2005), the program will use the same buckling curves as for grade S 460 when calculating the buckling resistance as per clause 6.3.
STIFF	Member Length or depth of beam, whichever is lesser	Distance between transverse stiffener plates, used to prevent web shear buckling. If not specified or if a value of 0 is provided, the program will assume the web is unstiffened.

Parameter Name	Default Value	Description
TOM	0	Total torsion for design used for torsion checks. Can be used to override the total torsional moment to be used for member design.
TORSION	0	Method to be used for a specific member or group of members: 0. Perform basic torsion checks if member is subject to torsion. 1. Perform basic stress check (Ignore warping effects). 2. Perform detailed checks (including warping effects). 3. Ignore all torsion checks **Note:** For options 1 or 2, the program will perform the torsion related checked even if torsional moment is absent and will use a value of zero for the torsional moment.
TRACK	0	Specify level of detail in output. 0. Summary of results only. 1. Summary with member capacities. 2. Detailed results. 4. Deflection check results only.
UNF	1	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.
ZG	+Section Depth/2	Distance of transverse load from shear center. Used to calculate M_{cr}. **Note:** For Tee sections, ZG will have a default value of (+Flange thickness/2)

Notes

1. CAN, DJ1, and DJ2 – Deflection

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement..

If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1, DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

Compute Delta = SQRT((DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2)

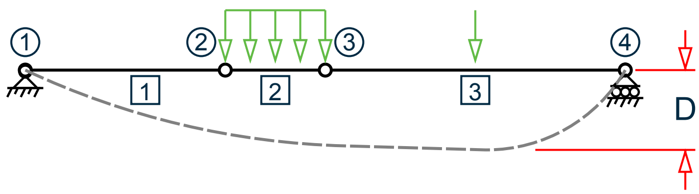
Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

Then, if CAN is specified a value 1, dff = L/Delta

Ratio due to deflection = DFF/dff

b. If `CAN` = 0, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Thus, for all three members here, `DJ1` should be `1` and `DJ2` should be `4`.



D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

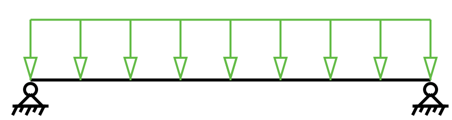
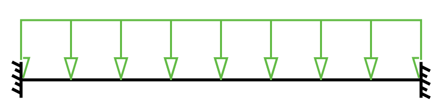
c. If `DJ1` and `DJ2` are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

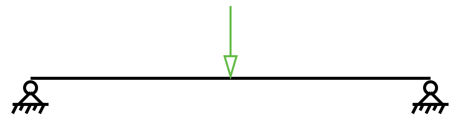
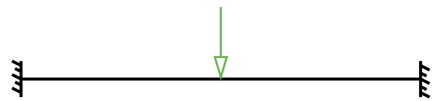
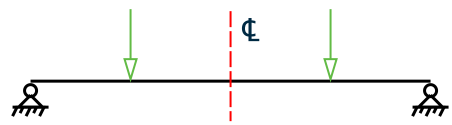
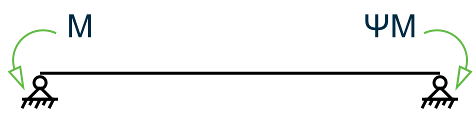
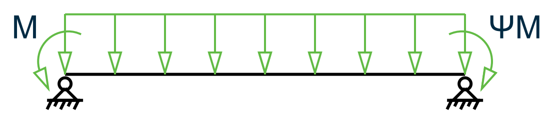
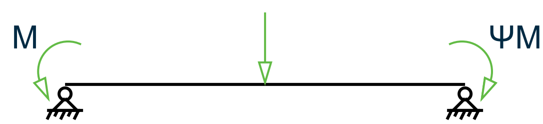
d. If a serviceability load envelope is specified under “LOAD LIST ENV” before a parameter block, the default value of DFF 360 is used for deflection calculations. If DFF 0 is specified for any member, then the deflection check is skipped for that member.

e. The above parameters may be used in conjunction with other available parameters for steel design.

2. The values of `CMM` for various loading and support conditions are as given below:

Table 133: Values for the CMM Parameter

CMM Value	Loading and Support Conditions
1	
2	

CMM Value	Loading and Support Conditions
3	
4	
5	
6	
7	 varying end moments and uniform loading
8	 varying end moments and central point load

3. Checking beam deflection

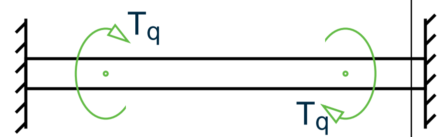
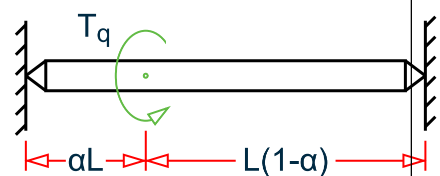
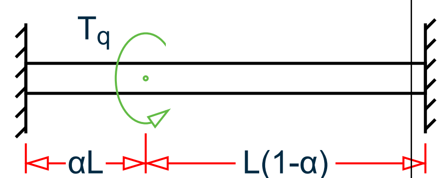
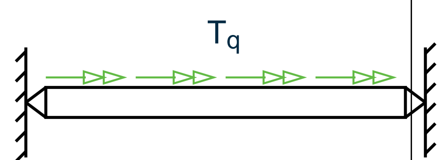
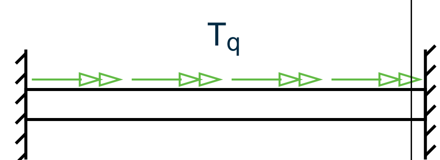
   With the `TRACK` parameter set to 4, the members included in a `BEAM CHECK` command will be checked for the local axis deflection rather than for the stress capacity using the current `LOAD LIST`.

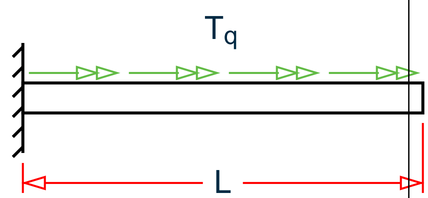
   If `LOAD LIST ENV` is used, then serviceability envelope must be present to check for deflections. If only strength load cases are given in `LOAD LIST ENV`, only strength checks will be performed and `TRACK 4` is treated as `TRACK 2`.

   Note: While both sets of code checks will be reported in the output file, only the last code check results are reported in the STAAD.Pro graphical interface.

4. The values of `CMT` for various loading and support conditions are as given below:

Table 134: Loading and Support Conditions represented by CMT Parameter Values

CMT Value	Description	Diagram
1	(Default) : Concentrated Torque at Ends. Ends Torsion fixed and Warping fixed	
2	Concentrated Torque along length of member. Ends Torsion fixed and Warping free	
3	Concentrated Torque along length of member. Ends Torsion fixed and Warping fixed	
4	Uniform Torque in member. Ends Torsion fixed and Warping free	
5	Uniform Torque in member. Ends Torsion fixed and Warping fixed	
6	Concentrated Torque in cantilever. End Torsion fixed and Warping fixed	

CMT Value	Description	Diagram
7	Uniform Torque in cantilever. End Torsion fixed and Warping fixed	

Note: For CMT = 2 and CMT = 3, you have the option of specifying the distance at which the concentrated torque acts, measured from the start of the member. This can be done by using the ALH design parameter. The ALH parameter indicates the ratio of the distance of the point torque (from the start of the member) to the length of the member. This parameter will have a default value of 0.5 (i.e., the torque acts at the center of the span) and will accept values ranging from 0 to 1.

Related Links

- *D5.C.5.2 Members Subject to Bending Moments* (on page 1482)
- *AD.2007-11.3.9 Eurocode 3 Steel Grades* (on page 159)
- *AD.2007-03.2.5 Updated and Additional Standard Steel Grades in Eurocode 3* (on page 265)

D5.C.7 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate. The adequacy is checked as per EN 1993-1-1:2005 and a corresponding National Annex (if specified). Code checking is done using the forces and moments at specific sections of the members.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks; the critical condition; the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member of forces in the member where the critical condition occurs).

Code checking can be done with any type of steel section listed in D3.B.4 Built-In Steel Section Library (on page 1380) or any of the user defined sections as described in G.6.3 User-Provided Steel Table (on page 2117) , with the exception of ISECTION. ISECTION has been currently excluded since the option of Tapered section design is currently not supported in the EC3 module. The EC3 (EN 1993) design module does not consider these sections or PRISMATIC sections in its design process.

Note: Checks for slender sections to EN 1993-1-1 are limited to I-SECTIONS, TEE, SINGLE CHANNEL, SINGLE ANGLE and CIRCULAR & RECTANGULAR HOLLOW SECTIONS.

Code checking for GENERAL sections can be also done using the EN1993 module. The program will design GENERAL sections as I sections by default. However, you are given the option to choose a “section type” to be considered while designing the member. Refer to the description of the GST design parameter in D5.C.6 Design Parameters (on page 1502) for details.

D5.C.8 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e., the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters DMAX and DMIN, which limits the maximum and minimum depth of the members.

Member selection can be performed with all the types of steel sections with the same limitations as defined in D5.C.7 Code Checking (on page 1513).

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection cannot be performed on members whose section properties are input as prismatic or as the limitations specified in D5.C.7 Code Checking (on page 1513).

D5.C.9 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

MEMBER	refers to the member number for which the design is performed.
TABLE	refers to steel section name, which has been checked against the steel code or has been selected.
RESULTS	prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
CRITICAL COND	refers to the clause in EN 1993-1-1:2005 code which governs the design.
RATIO	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
LOADING	provides the load case number, which governed the design.
FX, MY, and MZ	provide the axial force, moment in local Y-axis and the moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.
LOCATION	specifies the actual distance from the start of the member to the section where design forces govern.

Note: For a TRACK 2 output, the module will also report all the relevant clause checks that have been performed and will also indicate the critical ratio and the load case that caused the critical ratio as well as the corresponding forces that were used for the respective checks. A TRACK 2 output will also include the various design data used for the calculations such as the section modulii, section class, section capacity etc.

If an NA parameter (other than 0) has been specified and if the particular National Annex requires additional checks outside those specified in EN 1993-1-1:2005 (e.g., The Dutch National Annex), the respective NA clauses and any associated code clauses will be listed along with the critical ratios and the forces that were used for these clause checks.

Documentation notes appear in red.

Note: The results and output follow the axis convention as described in Section 7C.1.3

```
Code title & version                         STAAD.PRO CODE CHECKING - BS EN
1993-1-1:2005
                                          ********************************************
National Annex used, if any                          NATIONAL ANNEX - NA to BS EN
```

```
1993-1-1:2005
            Design engine version PROGRAM CODE REVISION V1.9 BS_EC3_2005/1
  ALL UNITS ARE - KN   METE (UNLESS OTHERWISE NOTED)
  MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
 =======================================================================
Member number, section profile & table       1 ST   HD320X127    (EUROPEAN
SECTIONS)Design status, critical code clause, & critical
ratio                           PASS     EC-6.3.3-662       0.045         1Section
forces & critical section location                       25.00 C          5.00
-10.00       0.00
=======================================================================
   MATERIAL DATA
      Grade of steel            =  USER
      Modulus of elasticity     =  205 kN/mm2
      Design Strength  (py)     =  275  N/mm2

   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30

"z-axis" here refers to bending about Z-Z (when Y is Up), where as
```

[D5.C.1 General Description](#) (on page 1478).

```
                                      z-axis          y-axis
      Moment of inertia         :    30820.004       9239.001
      Plastic modulus           :     2149.000        939.100
      Elastic modulus           :     1926.250        615.933
      Shear Area                :       81.998         51.728
      Radius of gyration        :       13.823          7.568
      Effective Length          :      500.000        500.000

    DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
Section class as per Table 5.2      Section Class            :   CLASS 1Max. cross
section capacity (A · fy/GM0      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006

Partial safety factors used      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10

                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4435.8          4435.8
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3


    BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  591.0
Factor C1 used in Mcr calculations and End restraint factor (corresponds to the CMN
design parameters      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000

      Elastic Critical Moment for LTB,               Mcr    =  1541.5
      Critical Load For Torsional Buckling,          NcrT   = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
```

```
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE NOTED)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================

    CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

Max. ratio, loadcase, & section forces for each clause check
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.3.1.1     0.008     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-661   0.035     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-662   0.045     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.2 LTB   0.017     1    25.0      0.0     0.0   -10.0     5.0
    Torsion and deflections have not been considered in the design.
                        _________________________


    ************** END OF TABULATED RESULT OF DESIGN **************
```

D5.D. European Codes - National Annexes to Eurocode 3 [EN 1993-1-1:2005]

A number of countries that have signed up to the replace their current steel design standards with the Eurocode, EN 1993-1-1:2005, known commonly as Eurocode 3, have published their National Annex documents. These documents make small changes to the base document and STAAD.Pro has been updated to incorporate some of these National Annex documents.

The parameter NA sets the default material gamma factors and any additional changes outlined in the country specific National Annex such as specific equations or methods. These are described for each National Annex document in the following sections.

The output file printout has been updated to indicate which National Annex (if any) has been used in a code check / select process (For all TRACK settings).

D5.D.1 General Format

The format of the EN 1993-1-1:2005 National Annex is as follows:

```
CODE EN 1993
```

```
NA  f1
```

```
{code  parameters}
```

Refer to D5.C.6 Design Parameters (on page 1502) D5.C.6 Design Parameters (on page 1502)

Where:

`f1` represents the number designation for a specific country's National Annex:

Table 135: Table 5B1.2(B) - Numerical Code for Eurocode National Annex

NA Value	Country
0	None — Uses the base EN 1993-1-1:2005 code, with no national annex changes or additions. The default values specified in En 1993-1-1:2005 will be used for the partial safety factors and various parameter values where applicable (default).
1	United Kingdom (D5.D.5 UK National Annex to EC3 (on page 1528)) — Uses the BS EN 1993-1-1:2005 version of Eurocode 3 along with the UK National Annex.
2	Netherlands (D5.D.3 Dutch National Annex to EC3 (on page 1518)) — Uses the NEN EN 1993-1-1:2005 version of the code. The Dutch National Annex [NEN-EN 1993-1-1/NB] has been added in this module. Please note that the Dutch National requires additional checks as per NEN 6770 and NEN 6771 which will also be performed during design checks with this parameter value
3	Norway (D5.D.4 Norwegian National Annex to EC3 (on page 1527)) — Uses the NS-EN 1993-1-1:2005 version of the code. The Norwegian National Annexe [NS-EN 1993-1-1:2005/Na 2008] has been added to this implementation.
4	France (D5.D.6 French National Annex to EC3 (on page 1537)) — Uses the Annexe Nationale a la NF EN 1993-1-1:2005 version of the code along with the French National Annex..
5	Finland (D5.D.7 Finnish National Annex to EC3 (on page 1543)) - Uses the SFS EN 1993-1-1:2005 version of Eurocode 3 along with the Finnish National Annex.
6	Poland (D5.D.8 Polish National Annex to EC3 (on page 1550)) - Uses the PN EN 1993-1-1:2005 version of Eurocode 3 along with the Polish National Annex.
7	Singapore (D5.D.9 Singaporean National Annex to EC3 (on page 1556)) - Uses the SS EN 1993-1-1:2005 version of Eurocode 3 along with the Singaporean National Annex.

NA Value	Country
8	Belgium (D5.D.10 Belgian National Annex to EC3 (on page 1563)) - Uses the NBN EN 1993-1-1:2005 version of Eurocode 3 along with the Belgian National Annex.
9	Malaysian (D5.D.11 Malaysian National Annex to EC3 (on page 1570)) - Uses the MS EN 1993-1-1:2005 version of Eurocode 3 along with the Malaysian National Annex.
10	German (D5.D.12 German National Annex to EC3 (on page 1575)) - Uses the DIN EN 1993-1-1:2005 version of Eurocode 3 along with the German National Annex.
11	Swedish (D5.D.13 Swedish National Annex to EC3 (on page 1579)) - Uses the BFS EN 1993-1-1:2005 version of Eurocode 3 along with the Swedish National Annex.

D5.D.2 Specifying the design engine to use a national annex

Use the following procedure to include additional check specified by a National Annex:

Batch steel design is performedin the **Analytical Modeling** workflow.

1. On the **Analysis and Design** ribbon tab, select **Steel** in the **Design** gallery.
   The **Steel Design - Whole Structure** dialog box opens.
2. In the **Current Code** drop-down menu, select **EN 1993-1-1:2005**.
3. Click **Define Parameters...**.
   The **Design Parameters** dialog box opens.
4. Select the **NA** parameter in the list box.
5. Select the option corresponding to the National Annex document you want to use .
6. Click **Add**.

   This will insert the following commands into the STAAD input file:

   ```
   CODE EN 1993-1-1:2005
   ```

   ```
   NA n
   ```

Refer to D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for additional information on steel design per EC3.

A design performed to the new Eurocode 3 National Annex is displayed in the output file (`*.ANL`) with the following header, in addition to the base EC3 output.

D5.D.3 Dutch National Annex to EC3

Adds values from the Dutch National Annex —titled NEN-EN 1993-1-1:2006+C1:2006/NB:2007— for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the Dutch National Annex, NEN-EN 1993-1-1:2006+C1:2006/NB:2007, (hereafter referred to as D-NA) are described in the following sections.

Refer to the [D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005]] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the D-NA.

Note: Clause 6.3.2.4 deals with a simplified assessment method for beams. STAAD.Pro only uses the more accurate method (6.3.2.2 and 6.3.2.3 in EC-3) and therefore this section is ignored.

The local axis convention in the Dutch codes is: Y – major axis & Z – minor axis (as opposed to the convention followed in STAAD.Pro).

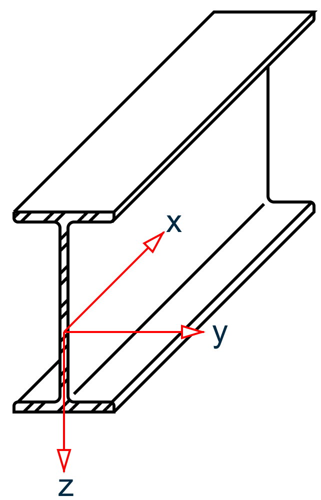


Figure 167: Local axis convention used in the Dutch NA to EC-3

D5.D.3.2 Clause 6.1 – General

The partial safety factors will use the following values:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
- Resistance of members to instability, $\gamma_{M1} = 1.0$
- Resistance of cross sections to tension, $\gamma_{M2} = 1.25$

The design function in STAAD.Pro sets these values as the default values for the D-NA (`NA 3` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to [D5.C.6 Design Parameters] (on page 1502)

D5.D.3.3 Clause 6.2.8 – Bending and shear

The D-NA requires the implementation of causes 11.3.1.1 and 11.3.1.3 of NEN 6770.

Clause 11.3.1.1 (NEN 6770): Class 1 and Class 2 I-section profiles

Class 1 and class 2 I section profiles must satisfy the interaction formulae given in tables 10 & 11 of NEN 6770.

Table 10 Provides interaction checks for bending about the major axis (All necessary terms and formulae are described below):

1. If $V_{z;s;d} \leq 0.5 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot \alpha_1 \cdot N_{pl;d}$, check equation 11.3.1
2. If $V_{z;s;d} \leq 0.5 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot \alpha_1 \cdot N_{pl;d}$, check equation 11.3.2
3. If $V_{z;s;d} > 0.5 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot \alpha_2 \cdot N_{v;u;d}$, check equation 11.3-3
4. If $V_{z;s;d} > 0.5 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot \alpha_2 \cdot N_{v;u;d}$, check equation 11.3-4

where

$V_{z;s;d}$	=	Actual Shear force in the section along Z- axis
$V_{z;pl;d}$	=	Shear capacity of section along Z - axis
		$= A_w \cdot f_{y;d} / \sqrt{3}$
$f_{y;d}$	=	yield stress
A_w	=	$A - 2 (bf - tw - 2r) tf$

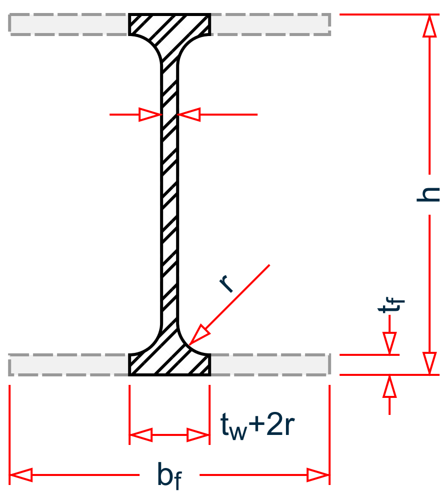


Figure 168: Definition of A_w

$N_{s;d}$	=	Axial force in the section
$N_{pl;d}$	=	Axial capacity of section = $A \cdot f_{y;d}$
$M_{y;s;d}$	=	Bending moment about major axis
$M_{y;pl;d}$	=	Plastic moment capacity of section = $f_{y;d} \cdot W_{y;pl}$
$W_{y;pl}$	=	Plastic section modulus
α_1	=	$min(A\text{-}2bfx\ tf)/A\ ,\ 0.5)$ - used in tables 10 & 11
α_2	=	see eqn 11.3-10- used in tables 10 & 11
$M_{v;y;ud}$	=	see eqn 11.3.12
$N_{;v;u;d}$	=	see eqn 11.3-13

Table 11: Provides interaction formulae for bending about the minor axis

1. If $V_{y;s;d} \leq 0.25 \cdot V_{y;pl;d}$ and $N_{s;d} \leq 1.0 \cdot \alpha_1 \cdot N_{pl;d}$ check equation 11.3-5
2. If $V_{y;s;d} \leq 0.25 \cdot V_{y;pl;d}$ and $N_{s;d} > 1.0 \cdot \alpha_1 \cdot N_{pl;d}$ check equation 11.3-6
3. If$V_{y;s;d} > 0.25 \cdot V_{y;pl;d}$ and $N_{s;d} \leq 1.0 \cdot \alpha_1 \cdot N_{v;u;d}$ check equation 11.3-7
4. If $V_{y;s;d} > 0.25 \cdot V_{y;pl;d}$ and $N_{s;d} > 1.0 \cdot \alpha_1 \cdot N_{v;u;d}$ check equation 11.3-8

where

$V_{y;s;d}$	=	Actual Shear force in the section along Y-axis
$V_{y;pl;d}$	=	Shear capacity of section along Y-axis

$$2bt_f \frac{f_{y;d}}{\sqrt{3}}$$

$M_{v;z;u;d}$	=	$q \cdot M_{z;pld} = q \cdot f_{y;d} \cdot W_{pl;z;d}$
$W_{pl;z;d}$	=	plastic section modulus about minor axis) & q as per eqn 11.3-14
$N_{v;u;d}$	=	$N_{pl;d} - 2 \cdot (1 - q) \cdot b_f \cdot t_f \cdot f_{y;d}$

Clause 11.3.1.3 (NEN 6770) : Class 1 and Class 2 Square and rectangular hollow sections

This clause requires class 1 and class 2 square and rectangular tube profiles to satisfy the interaction equations in Table 13.

1. If $V_{z;s;d} \leq 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_3 \cdot N_{pl;d}$ check equation 11.3.22
2. If $V_{z;s;d} \leq 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot a_3 \cdot N_{pl;d}$ check equation 11.3.23
3. If $V_{z;s;d} > 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} \leq 0.5 \cdot a_4 \cdot N_{v;u;d}$ check equation 11.3-24
4. If $V_{z;s;d} > 0.25 \cdot V_{z;pl;d}$ and $N_{s;d} > 0.5 \cdot a_4 \cdot N_{v;u;d}$ check equation 11.3-25

where

$V_{z;s;d}$	=	Actual Shear force in the section along Z-axis
$V_{z;pl;d}$	=	Shear capacity of section along Z-axis
b	=	breadth of section
h	=	height of section
A	=	area of section
$V_{z;pl;d}$	=	$V_{z;cl;d} = \frac{h}{b+h} A \frac{f_{y;d}}{\sqrt{3}}$
a_3	=	$min\{(A - 2 \cdot b \cdot t)/A \text{ or } 0.5\}$
a_4	=	from equation 11.3.27

=

=

D5.D.3.4 Clause 6.2.10 – Bending shear and axial force

Requires the implementation of clauses 11.3.1.1 to 11.3.1.3 and 11.3.2.1 to 11.3.2.3 of NEN 6770 and clause 11.3 of NEN 6771

Clause 11.3.1.1 (NEN 6770) and Clause 11.3.1.3 (NEN 6770)

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519)

Clause 11.3.1.2 (NEN 6770): Class 1 and class 2 circular hollow (CHS) profiles

Class 1 and class 2 sections with circular hollow profiles should satisfy the interaction equations given in table 12.

- Check #1 – If Vz;s;d ≤ 0.25 Vz;pl;d check equation 11.3.17
- Check #2 – If Vz;s;d > 0.25 Vz;pl;d check equation 11.3.18.

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519) of this document for equations to derive Vz;s;d

Vz;pl;d = Shear capacity of CHS sections

$$V_{pl;d} = 2\frac{A}{\pi}\frac{f_{y;d}}{\sqrt{3}}$$

See equations 11.3-19 and 11.3-20 to evaluate Mv;y;u;d and N;v;u;d.

To check for these conditions about the y axis, substitute the index “z” in the above equations with ‘y’ (should be the same of CHS sections).

Clause 11.3.2 (NEN 6770)

Section 11.3.2 in general deals with Biaxial bending with axial force and shear. The general condition to be satisfied in this case is given by equation 11.3-31 of NEN 6770

$$\beta_0\left(\frac{M_{y;s;d}}{M_{N;V;y;u;d}}\right)^{\alpha_1} + \beta_1\left(\frac{M_{z;s;d}}{M_{N;V;z;u;d}}\right)^{\alpha_2} \le 1$$

Clause 11.3.2.1 : Class 1 and class2 I-sections with biaxial bending + shear + axial force

The formula to evaluate M;N;v;y;u;d and M;N;v;z;u;d are to be taken from tables 14 and 15 of NEN 6770 respectively.

Checks for table 14:

1. Check #1 – If Vz;s;d ≤ 0.5 Vz;pl;d and Ns;d ≤ 0.5 x a1 x Npl;d use equation 11.3.32
2. Check #2 – If Vz;s;d ≤ 0.5 Vz;pl;d and Ns;d > 0.5 x a1 x Npl;d use equation 11.3.33
3. Check #3 – If Vz;s;d > 0.5 Vz;pl;d and Ns;d ≤ 0.5 x a2 x Nv;u;d use equation 11.3-34
4. Check #4 – If Vz;s;d > 0.5 Vz;pl;d and Ns;d > 0.5 x a2 x Nv;u;d use equation 11.3-35

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519) for equations to evaluate Vz;s;d, My;pl;d, Npl;d, Mv;y;ud, N;v;u;d, a1 ,a2 and Vz;pl;d.

Checks for table 15:

1. Check #1 – If Vy;s;d ≤ 0.25 Vy;pl;d and Ns;d ≤ 1.0 x a1 x Npl;d use equation 11.3.36
2. Check #2 – If Vy;s;d ≤ 0.25 Vy;pl;d and Ns;d > 1.0 x a1 x Npl;d use equation 11.3.37
3. Check #3 – If Vy;s;d > 0.25 Vy;pl;d and Ns;d ≤ 1.0 x a1 x Nv;u;d check equation 11.3-38
4. Check #4 – If Vy;s;d > 0.25 Vy;pl;d and Ns;d > 1.0 x a1 x Nv;u;d check equation 11.3-39

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519) for equations to evaluate Vy;s;d, Mz;pl;d, Npl;d, Mv;z;ud, N;v;u;d, a1 ,a2 and Vy;pl;d.

See table 16 for α1, α1, β0 and β1 use in tables 14 and 15.

Clause 11.3.2.2 : Class 1 and Class 2 Circular hollow tubes

The formula to evaluate M;N;v;y;u;d and M;N;v;z;u;d (to be used in equation 11-3-31, see description of clause 11.3.2 above) are to be taken from table 17 of NEN 6770.

1. Check #1 – If Vz;s;d ≤ 0.25 Vz;pl;d use equation 11.3.44
2. Check #2 – If Vz;s;d > 0.25 Vz;pl;d use equation 11.3.45.

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519) for equations to evaluate Vz;pl;d, My;pl;d, and Npl;d use in equations 11.3.44 & 11.3.45.

For values to be used for α1, α2, β1 and β2 in this case refer to table 18 of NEN 6770.

Clause 11.3.2.3 : Class 1 and class2 Rectangular and square hollow tubes

The formula to evaluate M;N;v;y;u;d and M;N;v;z;u;d (to be used in equation 11-3-31, see description of clause 11.3.2 above) are to be taken from table 19 of NEN 6770.

1. Check #1 – If Vz;s;d ≤ 0.25 Vz;pl;d and Ns;d ≤ 0.5 x a3 x Npl;d use equation 11.3-48
2. Check #2 – If Vz;s;d ≤ 0.25 Vz;pl;d and Ns;d > 0.5 x a3 x Npl;d use equation 11.3.49
3. Check #3 – If Vz;s;d > 0.25 Vz;pl;d and Ns;d ≤ 0.5 x a4 x Nv;u;d use equation 11.3-50

4. Check #4 – If Vz;s;d > 0.25 Vz;pl;d and Ns;d > 0.5 x a4 x Nv;u;d check equation 11.3-51

Refer to D5.D.3.3 Clause 6.2.8 – Bending and shear (on page 1519) for equations to evaluate Vz;pl;d, My;pl;d, Npl;d, Mv;y;ud, N;v;u;d, a3, a4 and Vz;pl;d to be used in the above equations. For values to be used for α1, α2, β1 and β2 in this case refer to table 20 of NEN 6770.

To check for these conditions about the y axis, substitute the index "z" in the above equations with 'y'.

Clause 11.3 (NEN 6771)

In general, this section deals with Biaxial bending with axial force and shear for class 3 and class 4 sections.

Check for class 3 sections: For class 3 sections use the method in section 11.3 NEN 6770. For class 3 sections the methods and equations discussed above can be used with the "plastic section modulus" being substituted with the 'elastic modulus'.

Check for class 4 sections: Class 4 sections can be treated as class 3 sections if the effective section properties are used as given in clause 10.2.4.2.3 of NEN 6771. Working out the effective section properties for slender sections has already been done in STAAD.Pro.

For I- section profiles and tubular sections, the following cases are checked:

1. If M;y;s;d / MN;y;f;u;d ≤ 1 check equation 11.2-7 (given below)

   $$V_{z;s;d}/V_{z;u;d} \le 1$$

   Where

   $V_{z;s;d}$ is the shear for in the Z direction
   $V_{z;u;d}$ is the shear capacity in the Z direction for ultimate limit state.

   For an I section,

   $$V_{z;u;d} = \frac{2}{3} A_{w;et} \frac{f_{y;d}}{\sqrt{3}}$$

   Where

   Aw,ef = effective web area as given in section 10.2.4.2.3.
   MN;y;f;u;d is the moment capacity about the Y axis for the effective section. = (fy·W,eff)
2. If M;y;s;d / MN;y;f;u;d > 1 and M;y;s;d / M;y;f;u;d ≤1 check equation 11.2-13 (given below):

   $$\frac{M_{y;s;d}}{M_{N;y;f;u;d} + \left(M_{N;y;u;d} - M_{N;y;f;u;d}\right)\left\{1 - \left(\frac{2V_{z;s;d}}{V_{z;u;d}} - 1\right)^2\right\}} \le 1$$

D5.D.3.5 Clause 6.3 – Buckling resistance of members

The D-NA introduces a new clause 6.3.0, which in turns requires the checks as per clauses 12.1.2.2, 12.13.2 and 12.1.4.2 of NEN 6771 to be applied.

Clause 12.1.2.2 (NEN 6771)

This clause in NEN 6771 determines the relative torsional slenderness and is given as:

$$\lambda_{\theta,rel} = \sqrt{\frac{N_{c;u;d}}{F_{E;\theta}}}$$

Where:

$$N_{c;u;d} = A \cdot f_{y;d}$$

A = area of section
$f_{y;d}$ = the yield stress
$F_{E;\theta}$ is the Euler-torsion formula

This value of slenderness is to be used to calculate the modification factors used in section 6.3 of EC-3.

Clause 12.1.3.2 (NEN 6771)

This clause works out the relative torsional-flexural buckling slenderness for compression members. The relative torsional-flexural buckling slenderness is given as:

$$\lambda_{tk,rel} = \sqrt{\frac{N_{c;u;d}}{F_{E;tk}}}$$

Where

$$N_{c;u;d} = A \cdot f_{y;d}$$

A = area of section
$f_{y;d}$ = yield stress
$F_{E;tk}$ is the Euler torsional buckling strength

Clause 12.1.4.2 (NEN 6771)

Buckling lengths of rotationally restrained bars with intermediate spring supports.

Note: STAAD.Pro does not allow for these end conditions, specifically. The effective length factors may be used to accommodate this requirement.

D5.D.3.6 Clause 6.3.1.3 – Slenderness for flexural buckling

The Dutch NA requires the implementation of clause 12.1.1.3 and 12.1.5.3.2 of NEN 6770 and clause 12.1.1.3 of NEN 6771.

Clause 12.1.1.3 (NEN 6770)

This clause gives the equations to evaluate the effective lengths for various support conditions. STAAD.Pro uses the effective length factor “K” which allows the user to set/modify the effective lengths for a member.

Clause 12.1.5.3.2 (NEN 6770)

This clause gives methods to evaluate the buckling length of lattice sections. We do not deal with latticed section in the current version of STAAD.Pro. In any case the buckling length can be adjusted using the “K” factor.

Clause 12.1.1.3 (NEN 6771)

This clause again deals with working out the effective lengths of prismatic and non-prismatic rods. Again, the “K” factor in the current implementation of STAAD.Pro is adequate to cater for adjusting the effective lengths as necessary.

D5.D.3.6 Clause 6.3.1.4 – Slenderness for torsional and torsional-flexural buckling

The D-NA requires the implementation of clauses 12.1.2 and 12.1.3 of NEN 6770

Clause 12.1.2 (NEN 6770): Torsional stability

IPE, HEA, HEB & HEM sections and pipe sections do not need to be checked for torsional instability.

If torsional checks need to be performed, they should be done according to 12.1.2 of NEN 6771.

Clause 12.1.2 (NEN 6771)

This clause gives the condition to check for torsion instability. The condition being:

$$\frac{N_{c;s;d}}{\omega_\theta N_{c;u;d}} \le 1$$

Where:

$N_{c;s;d}$ = the applied axial load
$N_{C;u;d}$ = the axial capacity = A · f_y.

$$\omega_\theta = \frac{\sigma_{\theta;d}}{f_{u;d}}$$

Clause 12.1.3 (NEN 6770): Torsional flexural stability

Doubly symmetric sections need not be checked for torsional flexural instability. However, for I sections that have rigid supports that is not along the axis of the section and any other sections will need to be checked as per clause 12.1.3 of NEN 6771.

Clause 12.1.3 (NEN 6771)

This clause gives the condition to check for torsional flexural instability. The condition being:

$$\frac{N_{c;s;d}}{\omega_{t;k} N_{c;u;d}} \le 1$$

Where:

$N_{c;s;d}$ and $N_{c;u;d}$ as in clause 12.1.2 above.

D5.D.3.7 Clauses 6.3.2.2 and 6.3.2.3 – Lateral torsional buckling curves

Clause 6.3.2.2 – Lateral torsional buckling curves - general

The D-NA states that the values for the imperfection factor, αLT, to be used in equation 6.56 of EC-3 are to be obtained from sTable 6.3 of EC-3. These are the values used by STAAD.Pro.

Clause 6.3.2.3 – Lateral torsional buckling curves for rolled sections or equivalent welded sections

The D-NA states that:

1. The values for the:
   - Imperfection factor αLT0 = 0.4 (used in equation 6.57 of EC-3)
   - B = 0.75 (used in equation 6.57 of EC-3)

   These are the default values used by the program.
2. The buckling curves shall be selected as per Table 6.5.
3. The reduction factor, f, is given by

   $$F = 1 – 0.5(1 - kc)[1 - 2x\ (\lambda LT\ -0.8)^2].$$

   kc is a correction factor for moment distribution determined from Table 6.6. This value can be specified or calculated by the program using the KC parameter. Refer to D5.C.6 Design Parameters (on page 1502)

The current implementation of STAAD.Pro conservatively uses a value of f = 1.0.

D5.D.3.8 Clause 6.33 – Uniform members in bending and axial compression

The D-NA recommends the use of the method in Annex B of EC-3 to determine the values of kyy, kyz, kzy and kzz to be used in 6.3.3 (EC-3) checks. STAAD.Pro uses the method in Annex B.

Clause 12.3.1.2.3 (NEN 6770): Rotation/bending capacity

The Dutch NA also requires additional checks as per clause 12.3.1.2.3 of NEN 6770.

The checks given in this clause deals with additional checks for columns that form part of a buttressed or non-butressed framework. The program uses the `ESTIFF` parameter with two different values to identify the framework type:

Table 136: Framework parameter `ESTIFF` values for the Dutch NA

`ESTIFF` value	Description
0	(default) Column part of a buttressed framework. Selecting this value will internally perform the checks as per section 1 of clause 12.3.1.2.3
1	Column is *not* part of a buttressed framework. Selecting this value will internally perform the checks as per section 2 of clause 12.3.1.2.3

These checks are described below:

1. For columns in buttressed frameworks the buckling length is to be taken based on either

- the system length or
- the distance between adjacent lateral supports

The following conditions should also be satisfied:

If $Nc;s;d / Npl;d < 0.15$, no additional checks are required

If $Nc;s;d / Npl;d \geq 0.15$ and the steel grade is S235 or S 275 then

$$\frac{N_{c;s;d}}{N_{pl;d}} + \frac{\lambda_y}{120} \leq 1$$

Where:

$N_{c;s;d}$ is the axial load in the section
$N_{pl;d}$ = Axial capacity of section = $A \cdot f_{y;d}$
λ_y = Slenderness of the section about the major axis (Y-axis)

If $Nc;s;d / Npl;d \geq 0.15$ and the steel grade is S355 then

$$\frac{N_{c;s;d}}{N_{pl;d}} + \frac{\lambda_y}{100} \leq 1$$

Where:

$N_{c;s;d}$ = the axial load in the section
$N_{pl;d}$ = Axial capacity of section = $A \cdot f_{y;d}$
λ_y = Slenderness of the section about the major axis (Y-axis)

2. For columns that are not part of buttressed frameworks the following additional checks need to be done:

   If $Nc;s;d / Npl;d < 0.15$, no additional checks are required

   If $Nc;s;d / Npl;d \geq 0.15$ and the steel grade is S235 or S 275 then

$$\frac{N_{c;s;d}}{N_{pl;d}} + \frac{\lambda_y}{100} \leq 1$$

   Where:

   $N_{c;s;d}$ = the axial load in the section and
   $N_{pl;d}$ = Axial capacity of section = $A \cdot f_{y;d}$
   λ_y = Slenderness of the section about the major axis (Y-axis)

   If $Nc;s;d / Npl;d \geq 0.15$ and the steel grade is S355 then

$$\frac{N_{c;s;d}}{N_{pl;d}} + \frac{\lambda_y}{80} \leq 1$$

D5.D.4 Norwegian National Annex to EC3

Adds values from the Norwegian National Annex —titled NS-EN 1993-1-1:2005/NA 2008— for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the Norwegian National Annex are:

D5.D.4.1 Clause 6.1(1) – General: Partial Safety Factors for buildings

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values:

- Resistance of cross-sections - γ_{M0} = 1.05
- Resistance of members to instability - γ_{M1} = 1.05
- Resistance of cross sections to tension - γ_{M2} = 1.25

The design function in STAAD.Pro sets these values as the default values for the Norwegian-NA (`NA 3` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Refer to the basic code (EC3) for a description of these clauses. The sections below refer to the corresponding clauses in the Norwegian -NA.

D5.D.5 UK National Annex to EC3

Adds values from the UK National Annex - titled NA to BS EN 1993-1-1:2005 - for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the UK-NA.

The following clauses are not implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	The UK NA specifies the value of λc0 for I, H channel or box section to be used in equation 6.59 of BS EN 1993-1-1:2005 as 0.4. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the UK National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	The value of the modification factor kfl to be used in equation 6.60 of BS EN 1993-1-1. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the UK National Annex.

The clauses/sections in EN 1993-1-1:2005 that have been dealt with in the UK National Annex (hereafter referred to as the UK-NA) are:

D5.D.5.1 Clause 6.1(1) – General: Partial Safety Factors for buildings

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the UK National Annex:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = 1.1

The design function in STAAD.Pro sets these values as the default values for the UK-NA (`NA 1` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Caution: The `GB1` parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence, any legacy files that use `GB1` parameter will indicate an error message and the user will need to substitute `GB1` with `GM1` in line with EN 1993-1-1:2005.

D5.D.5.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The UK-NA recommends the use of Table 6.3 and 6.4 of BS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the “Elastic Critical Buckling Moment”, Mcr. The UK National Annex does not specify a particular method to calculate M_{cr}. Hence the calculation of M_{cr} has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling:

This document provides a method to calculate “Mcr” specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method in the proposed implementation.

The equation to evaluate M_{cr} is given in the NCCI as:

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(kL)^2} \left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(kL)^2 G I_t}{\pi^2 E I_s} + (C_2 z_s)^2} - C_2 z_s \right]$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in the tables below:

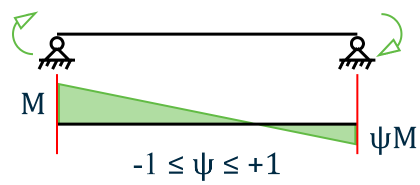
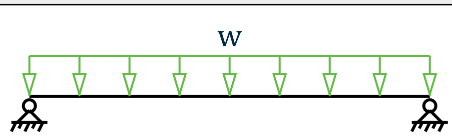
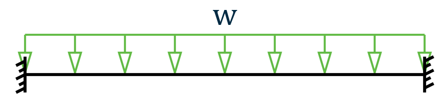
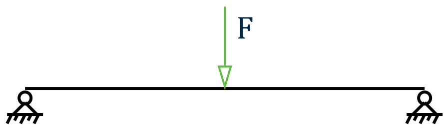
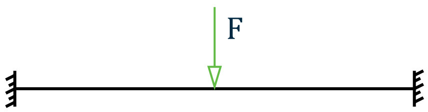


Table 137: Values of C_1 for end moment loading (for k=1)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05
-0,50	2,33
-0,75	2,57

Table 138: Values of factors C_1 and C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2
		1.127	0.454

Loading and support conditions	Bending moment diagram	C_1	C_2
		2.578	1.554
		1.348	0.630
		1.683	1.645

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

The implementation of EC3 in STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter. The first two loading conditions mentioned above and its variants can be dealt with by using the existing values of the `CMM` parameter (i.e., 1 to 6). Hence the appropriate values from this NCCI will be used for "C1" and 'C2' coefficients depending on the value of `CMM` specified. The default value of `CMM` is 1, which considers the member as a pin ended member with UDL along its span. The user will also have the option to specify specific values for C_1 and C_2 using the `C1` and `C2` parameters in the design input mode. Refer to D5.C.6 Design Parameters (on page 1502)

However, for cases with end moments and transverse loading, the NCCI provides graphs to evaluate the C1 and C2 coefficients. It does not however, provide a set of equations for these graphs. However the "end moments and transverse loading" condition cannot be currently specified in the design input. Hence this implementation will introduce two new values for the CMM parameter viz.

`CMM 7`: Member with varying end moments and uniform loading.

`CMM 8`: Member with varying end moments and central point load.

For these two conditions, the UK National Annex (nor the NCCI) does not provide equations to evaluate C1 and C2. Hence in STAAD.Pro the user will have to use the new "C1" & 'C2' parameters to input the required values for C1 & C2 to be used in calculating Mcr. For values of 7 or 8 for the CMM parameter, the program will issue a warning if C1 and C2 have not been specified.

Note: If the NA parameter has not been specified, the program obtains the values of C1 and C2 from Annex F of DD ENV version of 1993-1-1:1992.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (Mcr) for uniform mono symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for "Tee-Sections" will be worked out using the method in this NCCI.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections. In any case, the actual LTB capacity will still be worked out as per BS 5950-1 as in the current EC3 implementation.

The equation to evaluate M_{cr} for mono symmetric sections is given as :

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(k_x L)^2}\left[\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_x} + (C_2 z_e - C_3 z_1)^2} - \left(C_2 z_e - C_3 z_1\right)\right]$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. This implementation will consider C_1, C_2, and C_3as given in the tables below:

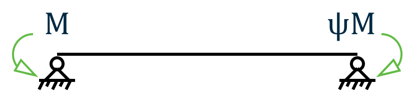
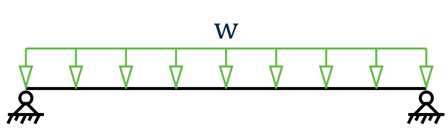
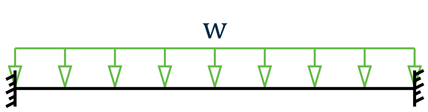
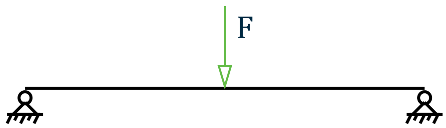
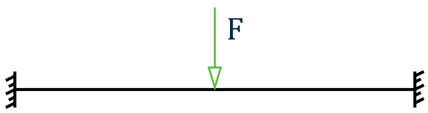


Table 139: Values of C_1 and C_2 for end moment loading (for $k_c = 1$)

Ψ	C_1	C_2
+1.00	1.00	1.00
+0.75	1.14	0.99
+0.50	1.31	0.99
+0.25	1.52	0.98
0.00	1.77	0.94
-0.25	2.05	0.85
-0.50	2.33	0.68
-0.75	2.57	0.38
-1.00	2.55	0.00

Table 140: Values of factors C_1, C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2	C_3
		1.13	0.45	0.52
		2.57	1.55	0.75

Loading and support conditions	**Bending moment diagram**	**C_1**	**C_2**	**C_3**
		1.35	0.63	1.73
		1.68	1.64	2.64

The CMM parameter (see section (i) above) specified during design input will determine the values of C1, C2 and C3. The default value of CMM is 1, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the "end moments and transverse loading" condition. The user however can use the new C1, C2 and C3 parameters to input the required values for C1, C2 and C3 to be used in calculating Mcr. As described in section (i) above, the user must use C1, C2 and C3 parameters along with CMM values of 7 and 8.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). The current implementation of EC3 in STAAD takes into account of the end conditions using the CMN parameter. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints, i.e., CMN = 1.0. For members with partial or end fixities (ie, CMN = 0.5 or CMN = 0.7), the proposed implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) the proposed implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term "zg" in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of 'zg' is considered positive if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The user will be allowed to modify this value by using the new 'ZG' parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term 'zg' in the equation will have a value of zero.

D5.D.5.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The UK-NA specifies different values for the $\lambda_{LT,0}$ and β factors to be used in equation 6.57 of BS EN 1993-1-1 for rolled and equivalent welded sections. The current implementation in STAAD.pro does not differentiate between rolled and welded sections and uses the default values in BS EN 1993-1-1 for λLT,0 and β. The values specified in the UK-NA are:

- For rolled sections and hot-rolled & cold formed hollow sections:

  $\lambda_{LT,0} = 0.4$

  $\beta = 0.75$
- For welded sections:

  $\lambda_{LT,0} = 0.2$

$\beta = 1.00$

The current implementation of STAAD.Pro uses the buckling curves based on Table 6.5 of BS EN 1993-1-1:2005. The UK-NA specifies different limits and buckling curves to be used in this clause as given below:

Table 141: Buckling curves to use with BS EN 1993-1-1:2005

Cross Section	Limits	Buckling Curve
Rolled doubly symmetric I and H sections and hot-finished hollow sections	h/b ≤ 2	b
	2.0 < h/b ≤ 3.1	c
	h/b > 3.1	d
Angles (for moments in the major principle plane)		d
All other hot-rolled sections		d
Welded, doubly symmetric sections and cold-formed hollow sections	h/b ≤ 2	c
	2.0 < h/b ≤ 3.1	d

This table again does not specify which buckling curve is to be used in case of welded doubly symmetric sections with h/b ≥ 3.1 and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

D5.D.5.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per UK NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of BS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...". Hence in the implementation of EC3 (and the UK Annex) in STAAD.Pro, by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the UK National Annex states that Table 6.5 in BS EN 1993-1-1:2005 should be replaced with the table given in the NA (See section 4.3 of this document). Hence for all cases dealt with by the table in the UK NA, this implementation will choose the buckling curves from the UK National Annex. For any case that is not dealt with by the table in the UK NA, the program will use the method given in Cl. 6.3.2.2 of BS EN 1993-1-1:2005.

Hence for the following cross sections the program will use the Table in the UK NA for choosing a buckling curve for LTB checks (when the UK NA has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section

- Welded doubly symmetric sections with h/b < 3.1

For the following cross sections, the program will use Cl. 6.3.2.3 of BS EN 1993-1-1:2005 to evaluate χLT

- Welded I & H Sections with h/b ≥ 3.1.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χLT .

In any case the Elastic critical moment "Mcr" (used to evaluate the non dimensional slenderness) will be worked out as given in section 4.2 of this document. Since the UK National Annex uses the NCCIs mentioned in the sections above, this implementation will only consider end restraint conditions corresponding to the CMN parameter=1.0 (See section 4.2 above). For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.5.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The UK NA specifies the use of eqn. 6.58 of BS EN 1993-1-1:2005 to evaluate the modification factor, f, for the LTB reduction factor χ_{LT}. To evaluate the modification factor BS EN 1993-1-1:2005 uses a correction factor kc given by Table 6.6 in the code.

The UK-NA however, specifies that the correction factor, kc, is to be obtained as below:

$Kc = 1 / \sqrt{C1}$, where C1 is to be obtained from the NCCI documents given in section 4.2 of this document. The NCCI document SN003a-EN-EU specifies the values of C1 to be used in table 3.1 as shown below. This proposed implementation will allow for the reduction factor based on the UK-NA.

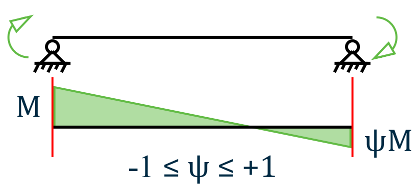
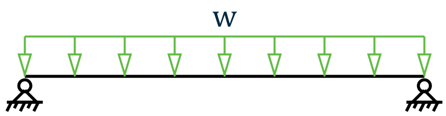
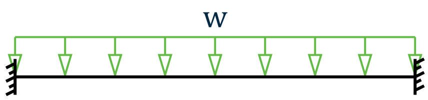
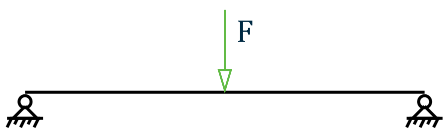
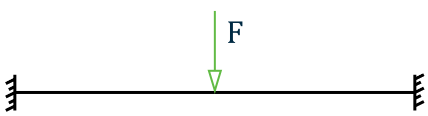


Table 142: Values of C_1 for end moment loading (for k=1)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05

ψ	C_1
-0,50	2,33
-0,75	2,57

Table 143: Values of factors C_1 and C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2
		1.127	0.454
		2.578	1.554
		1.348	0.630
		1.683	1.645

These values are for an end restraint factor of k=1 (i.e., `CMN 1.0`). Hence for all other values of `CMN` (i.e., 0.7 or 0.5) this implementation will use the values of C1 from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for kc. However the user can also input a custom value of kc by setting the design parameter `KC` to the desired value. The user can also get the program to calculate the value of kc automatically by setting the value of the `KC` parameter in the design input to 0. This will cause the program to evaluate a value of C1 corresponding to the end conditions and the Bending moment of the member and in turn calculate kc as given in the NA. To evaluate C1, the program will use the NCCI documents mentioned in section 4.2 of this document.

D5.D.5.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The UK-NA recommends that the method in Annex A or Annex B of BS EN 1993-1-1:2005 can be used to calculate the interaction factors for Cl. 6.3.3 checks in the case of doubly symmetric sections. The proposed implementation will hence use equations in Annex B of BS EN 1993-1-1:2005 to calculate these interaction factors for doubly symmetric sections. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B.

However for non-doubly symmetric sections, the UK NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the UK NA). The UK NA requires additional checks to be done to check for the maximum allowable values of λ and X to be used in equations 6.61 and 6.62 of BS EN 1993-1-1:2005.

As per the UK NA, for non-doubly symmetric sections, the slenderness about the weak axis (λy in STAAD) and the corresponding reduction factor χy should be taken as the values from the highest values of slenderness (λ) among the flexural buckling slenderness (λy), torsional slenderness (λ_T) and torsional-flexural slenderness (λ_{TF}) as given in Clauses 6.3.1.3 and 6.3.1.4 of BS EN 1993-1-1:2005. Hence for non-doubly symmetric sections the program will calculate the critical non-dimensional slenderness as:

λ_y = the maximum of either λ from Cl. 6.3.1.3 or λ_T from Cl. 6.3.1.4

$$\lambda_T = \sqrt{\frac{A f_y}{N_{cr}}}$$

Where:

N_{cr} = min (N_{CrT}, N_{crTF}).

The UK NA or EC3 does not however specify a method to evaluate NCrT or NcrTF. Hence this implementation will use the method specified in the NCCI document "SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes" to calculate these. See section 4.9 below for details.

Note: The UK National Annex or EC3 does not deal with angle sections in specific and hence this implementation will use the method used in the current EC3 implementation to deal with slenderness of angle sections. In the current implementation this is done as per cl 4.7.10 of BS 5950. This proposed implementation will still use the same method for single and double angle sections to evaluate the slenderness.

Clause NA 3.2 of the UK NA also requires that "Where the section is not an I Section or a hollow section and is a class1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause". Hence for all Class 1 or Class 2 cross sections that are NOT I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.

D5.D.5.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of BS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. BS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of BS EN 1993-1-1:2005).

The NCCI document "SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes" provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the UK NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.6 French National Annex to EC3

Adds values from the French National Annex - titled *Annexe Nationale a la NF EN 1993-1-1:2005* - for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the French National Annex.
Clause 6.3.2.4(2)B – Modification factor, kfl	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the French National Annex.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the French-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the French National Annex (hereafter referred to as FR-NA) and that are relevant to the proposed implementation are:

Related Links

- *V. EC3 French NA - Channel Section with Conc Load* (on page 4111)
- *V. EC3 French NA - Column with Axial Load* (on page 4114)
- *V. EC3 French NA - I Section with Conc Load* (on page 4121)
- *V. EC3 French NA - I Section with UDL* (on page 4126)
- *V. EC3 French NA - Tee Section* (on page 4130)
- *V. EC3 French NA - Varying End Mom CMM8* (on page 4133)

D5.D.6.1 Clause 3.2.1(1) - Material Properties

The material strengths (i.e., - steel grade strengths) to be used with NF EN 1993-1-1 are given in Table 3.1 of the code. The French National Annex however, specifies a separate table (Table 3.1 NF) for the yield and tensile strengths of steel grades. This new table replaces Table 3.1 in NF EN 1993-1-1:2005. Table 3.1 NF excludes steel grades from standards EN 10210-1 and EN 10219-1 that are given in EC-3.

STAAD.Pro uses the steel grades and values from the table given in the National Annex (i.e., - Table 3.1 NF). Table 3.1 NF is similar to table 3.1 in EC3, apart from the f_u values for S 355 and S355 W grade steel.

Table 144: Material strengths specified for use with the NF-NA

Standard and grade of steel		Nominal thickness, t, of the element (mm)			
		t 40 mm		40 mm < t ≤ 80 mm	
		f_y (N/mm^2)	f_u (N/mm^2)	f_y (N/mm^2)	f_u (N/mm^2)
EN 10025-2	S 235	235	360	215	360
	S 275	275	430	255	410
	S 355	355	490	335	470
	S 450	440	550	410	550

Standard and grade of steel		Nominal thickness, t, of the element (mm)			
		t 40 mm		40 mm < t ≤ 80 mm	
		f_y (N/mm^2)	f_u (N/mm^2)	f_y (N/mm^2)	f_u (N/mm^2)
EN 10025-3	S 275 N/NL	275	390	255	370
	S 355 N/NL	355	490	335	470
	S 420 N/NL	420	520	390	520
	S 460 N/NL	460	540	430	540
EN 10025-4	S 275 M/ML	275	370	255	360
	S 355 M/ML	355	470	335	450
	S 420 M/ML	420	520	390	500
	S 460 M/ML	460	540	430	530
EN 10025-5	S 235 W	235	360	215	340
	S 355 W	355	490	335	490
EN 10025-6	S 460 Q/QL/QL 1	460	570	440	550

If you specify a steel grade that is not given in the Annex Table 3.1 (NF) but is present in Table 3.1 of EN 1993-1-1:2005, the program uses the values from Table 3.1 of EN 1993-1-1:2005. The appropriate yield strength (f_y) used is shown in the design output file.

D5.D.6.2 Clause 6.1(1) – General

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the French National Annex:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = 1.25

The design function in STAAD.Pro sets these values as the default values for the NF-NA (`NA 4` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

> **Caution:** The GB1 parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005. Hence, any legacy files that use GB1 parameter will indicate an error message and the user will need to substitute GB1 with GM1 in line with EN 1993-1-1:2005.

D5.D.6.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The French NA recommends the use of Table 6.3 and 6.4 of NF EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the "Elastic Critical Buckling Moment", M_{cr}. The French NA gives a method to evaluate M_{cr} in its "Annex MCR". This implementation will make use of this method to evaluate Mcr. Annex MCR however deals with the calculation of Mcr for doubly symmetric sections. Hence this implementation will use this method only for doubly symmetric sections. For mono symmetric sections that are symmetric about the minor axis (i.e Tee sections) this implementation will use the method from the NCCI document SN030a-EN-EU as given in the section below. For any other type of section that is not dealt with by the Annex, this implementation will use the method and tables given in Annex F of DD ENV 1993-1-1:1992.

Annex MCR

This document provides a method to calculate M_{cr} specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method in this implementation.

The equation to evaluate M_{cr} is given as:

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(kL)^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(kL)^2 G I_t}{\pi^2 E I_s} + (C_2 z_s)^2} - C_2 z_s\right]$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions. The NCCI provides values for C_1 and C_2 for the different cases as given in Table1 and Table 2 of the Annex. Table 1 deals with the condition of a simply supported member with end moments and the value of C_1 is determined by the end moment ratio (Refer to the NA for details). Clause 3.2 of the National Annex however gives a formula to evaluate C_1 as:

$$C_1 = \frac{1}{\sqrt{0.325 + 0.423\psi + 0.252\psi^2}}$$

This formula however does not match the values given in Table 1 of the NA. Hence this implementation will use the values of C1 from Table 1 if the end moment ration (ψ) is exactly equal to the values of ψ in the table. For all other cases this implementation will calculate the value of C1 from equation (6) in the Annex.

The value of C2 will be determined from Table 2 of the Annex based on the loading and end conditions (i.e the CMM parameter in STAAD.Pro).

You also have the option to specify specific values for C_1 and C_2 using the C1 and C2 parameters in the design input mode. Refer to D5.C.6 Design Parameters (on page 1502)

The French NA considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading: use CMM 7 or CMM 8 for this condition.

The load to moment ratio (μ) will then be used in the calculations will then be used to calculate C1 and C2 as given in section 3.5 of Annex MCR (See Annex MCR in the NA for details).

The parameter MU may be specified when using CMM = 7 or 8 to specify the load to moment ratio (µ) to be used in the calculations. For the French National Annex if CMM = 7 or 8 is been specified, you must also either specify a value for MU or values for the C1 and C2 parameters directly.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (Mcr) for uniform mono symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(k_x L)^2}\left[\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_x} + (C_2 z_e - C_3 z_1)^2} - \left(C_2 z_e - C_3 z_1\right)\right]$$

The factors C_1, C_2, and $C_3$are dependent on the end conditions and loading criteria. This implementation will consider C_1, C_2, and C_3as given in the tables below:

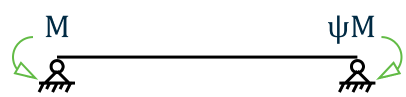
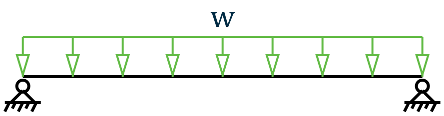
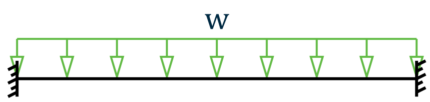
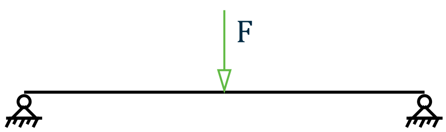
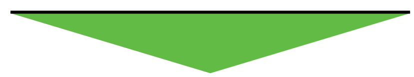
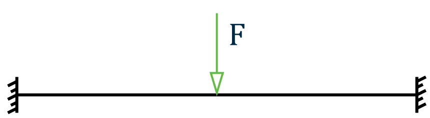


Table 145: Values of C_1 and C_2 for end moment loading (for $k_c = 1$)

Ψ	C_1	C_2
+1.00	1.00	1.00
+0.75	1.14	0.99
+0.50	1.31	0.99
+0.25	1.52	0.98
0.00	1.77	0.94
-0.25	2.05	0.85
-0.50	2.33	0.68
-0.75	2.57	0.37
-1.00	2.55	0.00

Table 146: Values of factors C_1, C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2	C_3
		1.13	0.45	0.52
		2.57	1.55	0.75
		1.35	0.63	1.73
		1.68	1.64	2.64

The `CMM` parameter specified during design input will determine the values of C1, C2 and C3. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the "end moments and transverse loading" condition. The user however can use the new "C1", 'C2' and 'C3' parameters to input the required values for C1, C2 and C3 to be used in calculating Mcr.

Note: If "MU" as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. The current implementation of EC3 in STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Also, the NCCI document and Annex MCR of the FR-NA assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member(k = kw=1 .i.e., CMN parameter =1.0). Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints. For members with partial or end fixities (ie, CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term "zg" in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of 'zg' is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the `ZG` parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term "zg" in the equation will have a value of zero.

Note: There is a separate method specified in the NCCI document "SN006a-EN-EU" to calculate Mcr for cantilever beams. Again this document does not give any specific formulae to evaluate the coefficients. Hence, this has not been implemented in STAAD.Pro.

D5.D.6.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The FR-NA provides equations to evaluate the $\lambda_{LT,0}$ and α_{LT} factors given in clause 6.3.2.3

For rolled doubly symmetric sections use:

$$\bar{\lambda}_{LT,0} = 0.2 + 0.1\frac{b}{h}$$

$$\alpha_{LT} = 0.4 - 0.2\frac{b}{h}\bar{\lambda}^2_{LT} \geq 0$$

Note: Since EN 1993-1-1:2005 limits the value of $\lambda_{LT,0}$ to 0.4, STAAD.Pro limits $\lambda_{LT,0}$ to a maximum value of 0.4.

For welded doubly symmetric sections use:

$$\bar{\lambda}_{LT,0} = 0.3\frac{b}{h}$$

$$\alpha_{LT} = 0.5 - 0.25\frac{b}{h}\bar{\lambda}^2_{LT} \geq 0$$

For other sections:

$$\lambda_{LT,0} = 0.2$$

$$\alpha_{LT} = 0.76$$

And for all sections, β = 1.0

These equations and factors are then applied to equation 6.57 of NF EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor χ_{LT}.

D5.D.6.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The French NA specifies that the modification factor is to be obtained as per the default method given in EC-3. Hence this implementation will use the existing functionality to evaluate the correction factor kc to be used in the modification factor f.

The program uses a default value of 1.0 for kc. However the user can also input a custom value of kc by setting the design parameter KC to the desired value. You may instruct the program to calculate the value of kc automatically by setting the value of the KC parameter in the design input to `0`. This will cause the program to evaluate kc from Table 6.6 of NF EN 1993-1-1:2005. This will correspond to the end conditions and the bending moment of the member (i.e., the value of `CMM` parameter specified).

For `CMM = 7`, the program will choose the value of kc to be either 0.90 or 0.91 based on the end moment ratio.

For `CMM = 8`, the program will choose the value of kc to be either 0.77 or 0.82 based on the end moment ratio.

An additional check will also be performed as given below:

$$\chi_{LT,\mathrm{mod}} \leq \frac{1}{\bar{\lambda}^2_{LT}}$$

The French Annex specifies that the modification factor is applicable only to members that are free to rotate on plan (i.e., `CMN 1.0`). Hence for all other values of `CMN`, this implementation will ignore “f” and hence will use $\chi_{LT,mod} = \chi_{LT}$.

D5.D.6.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The French NA recommends the use of equations in Annex A of NF EN 1993-1-1:2005 to calculate these interaction factors. STAAD.pro uses the method in Annex B for design per EC3 (without National Annex). Therefore, the method in Annex A has been added into the program.

Note: The NA mentions that this method can be extended to singly symmetric I-Sections (symmetric about the minor axis) if the elastic properties are used instead of the plastic properties. However, since STAAD does not have a provision to specify such sections, this case will *not* be considered for this implementation.

The NA also mentions that torsional flexural buckling needs to be taken into account in case of mono symmetric sections. This is taken into account based on the method given in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes”. Refer to D5.D.6.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling (on page 1543)

The NA also recommends a lower limit as given below for the term $C_{mi,0}$ in Table A.2 of Annex A:

$$C_{mi,0} \geq 1 - \frac{N_{Ed}}{N_{cr,i}}$$

D5.D.6.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of NF EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. NF EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of NF EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the French NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.7 Finnish National Annex to EC3

Adds values from the Finnish National Annex - titled *National Annex to Standard SFS-EN 1993-1-1* - for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Finnish National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Finnish National Annex.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the Finnish-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Finnish National Annex (hereafter referred to as SFS-NA) and that are relevant to the proposed implementation are:

D5.D.7.1 Clause 3.2.1(1) - Material Properties

The material strengths (i.e., steel grade strengths) to be used with SFS-EN 1993-1-1 are given in Table 3.1 of the code. These steel grade values are specified using the `SGR` parameter (Refer to D5.C.6 Design Parameters (on page 1502)).

The Finnish National Annex states in Cl. 3.1(2) that, apart from the steel grades specified in Table 3.1 of SFS EN 1993-1-1, the following steel grades can also be used:

- Steel grades S315MC, S355MC, S420MC and S460MC according to SFS-EN 10149-2
- Steel grades S260NC, S315NC, S355NC and S420NC according to SFS-EN 10149-3

These grades of steel can be specified by using the `PY` (Yield Strength) and `FU` (Ultimate Strength) parameters in STAAD.Pro. Set these parameters to the respective values as given in SFS-EN 10149-2/3 for the steel grades specified above. The choice of the buckling curve to be used is based on the value of the `SGR` parameter specified. The output will include the appropriate yield strength used for design.

D5.D.7.2 Clause 6.1(1) – General

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Finnish National Annex:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = 1.25

The design function in STAAD.Pro sets these values as the default values for the SFS-NA (`NA 5` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

Caution: The `GB1` parameter that is being used for compression checks in builds preceding this release (STAAD.Pro 2007 build 06) has been removed as this parameter is no longer required in EN 1993-1-1:2005.

Hence, any legacy files that use `GB1` parameter will indicate an error message and the user will need to substitute `GB1` with `GM1` in line with EN 1993-1-1:2005.

D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The Finnish NA recommends the use of Table 6.3 and 6.4 of SFS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the "Elastic Critical Buckling Moment", M_{cr}. The Finnish National Annex does not specify a particular method to calculate M_{cr}. Hence the calculation of M_{cr} has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling:

This document provides a method to calculate M_{cr} specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate M_{cr} is given in the NCCI as:

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(kL)^2} \left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(kL)^2 G I_t}{\pi^2 E I_s} + (C_2 z_s)^2} - C_2 z_s \right]$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in the tables below:

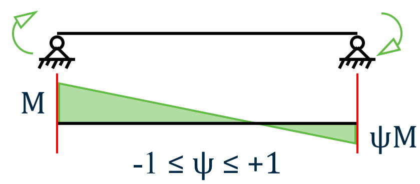


Table 147: Values of C_1 for end moment loading (for k=1)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05
-0,50	2,33
-0,75	2,57

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (M_{cr}) for uniform mono symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for "Tee-Sections" will be worked out using the method in this NCCI.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as :

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(k_x L)^2} \left[\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_x} + (C_2 z_e - C_3 z_1)^2} - \left(C_2 z_e - C_3 z_1\right) \right]$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. This implementation will consider C_1, C_2, and C_3 as given in the tables below:

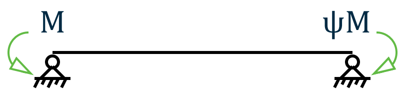
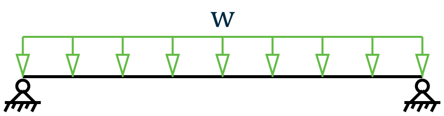
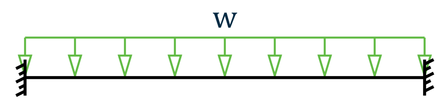
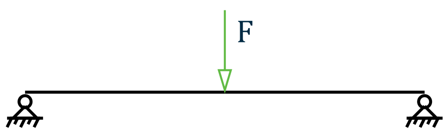
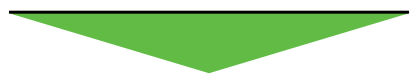
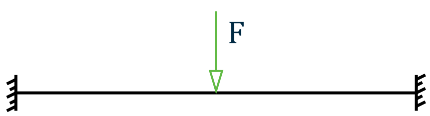


Table 148: Values of C_1 and C_2 for end moment loading (for $k_c = 1$)

Ψ	C_1	C_2
+1.00	1.00	1.00
+0.75	1.14	0.99
+0.50	1.31	0.99
+0.25	1.52	0.98
0.00	1.77	0.94
-0.25	2.05	0.85
-0.50	2.33	0.68
-0.75	2.57	0.37
-1.00	2.55	0.00

Table 149: Values of factors C_1, C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2	C_3
		1.13	0.45	0.52
		2.57	1.55	0.75
		1.35	0.63	1.73
		1.68	1.64	2.64

The CMM parameter specified during design input will determine the values of C_1, C_2, and C_3. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the "end moments and transverse loading" condition. You can use the C1, C2, and C3 parameters to input the required values for C_1, C_2, and C_3 to be used in calculating M_{cr}.

Note: If MU as well as C1, C2, and C3 have been specified, the program will ignore MU and use the user input values of C1, C2, and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = k_w = 1.0$). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of $K = k_w = 1$ is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

For the term z_j, please refer to Annex E of NBN NA 2018.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.7.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The Finnish-NA provides the values for the terms $\lambda_{LT,0}$ and β factors given in clause 6.3.2.3(1) as follows:

For rolled doubly symmetric sections and hollow sections, use:

$\lambda_{LT,0}$ =0.4 and β = 0.75

For welded doubly symmetric sections and hollow sections use:

$\lambda_{LT,0}$ = 0.2 and β = 1.0

The Finnish NA specifies the following limits for choosing the buckling curves:

Table 150: Selection of lateral torsional buckling curve for cross sections using equation (6.57)

Cross-section (constant cross-section)	Limits	Buckling Curve
Rolled double symmetric I- and H-sections and hot finished hollow sections.	h/b ≤ 2 2 < h/b <3.1	b c
Welded double symmetric I-section and H- sections and cold-formed hollow sections	h/b ≤ 2 2 < h/b < 3.1	c d

The NA says that for all other cases the rules given in Cl 6.3.2.2 should be used. Hence even for rolled or welded doubly symmetric sections with h/b ratio ≥ 3.1, this implementation will resort to checks as per clause 6.3.2.2.

These equations and factors are then applied to equation 6.57 of SFS-EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor χ_{LT}.

D5.D.7.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Finnish NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of SFS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...". Hence in the implementation of EC3 (and the Finnish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the Finnish National Annex gives equations to evaluate the imperfection factors to be used for various section types (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB

checks (on page 1545)). Hence for all cases dealt with by the equations in the Finnish NA, this implementation will use Cl 6.3.2.3 to evaluate χ_{LT}.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χ_{LT} .

In any case, the elastic critical moment, Mcr, (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the `CMN` parameter=1.0 (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1545)) will be considered. For all other cases of the `CMN` parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., `NA` parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., `SBLT` parameter = 1.0 in design input).

D5.D.7.6 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

STAAD.Pro uses the value of the modification factor f = 1.0 as given in the Finnish NA.

D5.D.7.7 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The Finnish NA recommends the use of equations in Annex A or Annex B of SFS-EN 1993-1-1 to calculate these interaction factors. STAAD.Pro uses the method in Annex B by default. This implementation of the Finnish NA will also use Annex B for Cl.6.3.3 checks.

D5.D.7.8 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of SFS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. SFS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SFS EN 1993-1-1:2005).

The NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the Finnish NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.8 Polish National Annex to EC3

Adds values from the Polish National Annex - titled *National Annex to Standard NA FOR PN-EN 1993-1-1:2006* - for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Polish National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Polish National Annex.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the Polish-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Polish National Annex, NA for PN-EN 1993-1-1:2006, (hereafter referred to as PN-NA) and that are relevant to the proposed implementation are:

Related Links

- *V. EC3 Polish NA - Column with Axial Load* (on page 4167)
- *V. EC3 Polish NA - I Section with Conc Load* (on page 4174)
- *V. EC3 Polish NA - I Section with UDL* (on page 4178)

D5.D.8.1 Clause 3.2.1(1) - Material Properties

The material strengths (i.e., steel grade strengths) to be used with PN-EN 1993-1-1 are given in Table 3.1 of the code. The Polish National Annex states in Cl. 3.1(2) that the steel grades to be used will be based on Table 3.1 of PN EN 1993-1-1. These steel grade values are specified using the `SGR` parameter (Refer to D5.C.6 Design Parameters (on page 1502)).

D5.D.8.2 Clause 6.1(1) – General

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Polish National Annex:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = minimum of 1.1 or 0.9 x f_u/f_y

where

f_u	=	the ultimate steel strength
f_y	=	the yield strength of steel

The design function in STAAD.Pro sets these values as the default values for the PN-NA (`NA 6` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

D5.D.8.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The Polish NA recommends the use of Table 6.3 and 6.4 of PN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the “Elastic Critical Buckling Moment”, M_{cr}. The Polish National Annex does not specify a particular method to calculate M_{cr}. Hence the calculation of M_{cr} has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling

This document provides a method to calculate M_{cr} specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate M_{cr} is given in the NCCI as:

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(kL)^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(kL)^2 G I_t}{\pi^2 E I_s} + (C_2 z_s)^2} - C_2 z_s\right]$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in the tables below:

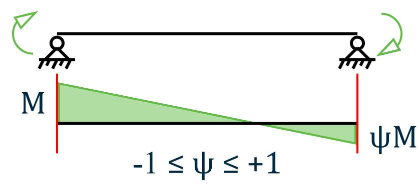


Table 151: Values of C_1 for end moment loading (for k=1)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05
-0,50	2,33
-0,75	2,57

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression

This document provides a method to evaluate the elastic critical moment (M_{cr}) for uniform mono symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for “Tee-Sections” will be worked out using the method in this NCCI.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as :

$$M_{cr} = C_1 \frac{\pi^2 E I_s}{(k_x L)^2}\left[\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I_s} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_x} + (C_2 z_e - C_3 z_1)^2} - \left(C_2 z_e - C_3 z_1\right)\right]$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. This implementation will consider C_1, C_2, and C_3 as given in the tables below:

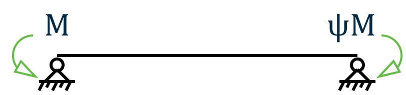
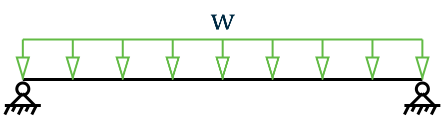
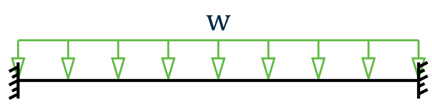
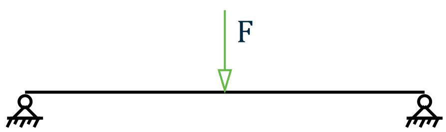
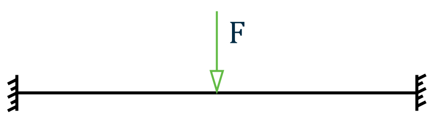


Table 152: Values of C_1 and C_2 for end moment loading (for $k_c = 1$)

Ψ	C_1	C_2
+1.00	1.00	1.00
+0.75	1.14	0.99
+0.50	1.31	0.99
+0.25	1.52	0.98
0.00	1.77	0.94
-0.25	2.05	0.85
-0.50	2.33	0.68
-0.75	2.57	0.38
-1.00	2.55	0.00

Table 153: Values of factors C_1, C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2	C_3
		1.13	0.45	0.52
		2.57	1.55	0.75
		1.35	0.63	1.73
		1.68	1.64	2.64

The CMM parameter specified during design input will determine the values of C_1, C_2, and C_3. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You can use the C1, C2, and C3 parameters to input the required values for C_1, C_2, and C_3 to be used in calculating M_{cr}.

Note: If MU as well as C1, C2, and C3 have been specified, the program will ignore MU and use the user input values of C1, C2, and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). Other values of CMN (i.e., CMN = 0.5 or CMN = 0.7) are *not* applicable to the Polish NA.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term “zg” in the equation to calculate Mcr refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of ‘zg’ is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term “zg” in the equation will have a value of zero.

Note: There is a separate method specified in the NCCI document "SN006a-EN-EU" to calculate Mcr for cantilever beams. Again this document does not give any specific formulae to evaluate the coefficients. Hence, this has not been implemented in STAAD.Pro.

D5.D.8.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The Polish-NA provides the values for the terms $\lambda_{LT,0}$ and β factors given in clause 6.3.2.3(1) as follows:

For all sections, use:

$\lambda_{LT,0}$ =0.4 and β = 0.75

The Polish NA specifies the use of uses table 6.5 to work out the buckling curves for use in Cl. 6.3.2.3. Hence table 6.5 in PN-EN 1993-1-1 will be used for this.

These equations and factors are then applied to equation 6.57 of PN-EN 1993-1-1 to evaluate the Lateral Torsional Buckling reduction factor χ_{LT}.

D5.D.8.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Polish NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of PN EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...". Hence in the implementation of EC3 (and the Polish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the Finnish National Annex gives equations to evaluate the imperfection factors to be used for various section types (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1545)). Hence for all cases dealt with by the equations in the Finnish NA, this implementation will use Cl 6.3.2.3 to evaluate χ_{LT}.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χ_{LT} .

In any case, the elastic critical moment, M_{cr}, (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (Refer to D5.D.7.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1545)) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.8.6 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

STAAD.Pro uses the value of the modification factor, *f*, as per eqn 6.58 of PN-EN 1993-1-1. The correction factor, k_c, will be evaluated as:

$$k_c = \sqrt{C_{mLT}}$$

where

C_{mLT}	=	the equivalent uniform moment factor from table B.3 of PN-EN 1993-1-1. C_{mLT} is evaluated based on the end conditions of the member and the shape of the bending moment diagram. However, if the KC parameter is used, then the program will use the specified value.

D5.D.8.7 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The Polish NA recommends the equations in Annex B of PN-EN 1993-1-1 to calculate these interaction factors. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B by default. The proposed implementation of the Polish NA will also use Annex B for Cl.6.3.3 checks.

The Polish NA also gives two additional simplified checks. This implementation will provide for these additional checks as well. However as they are intended as optional checks, by default, the program will not perform these checks. However, the user can invoke these checks by using the PLG parameter. Refer to D5.C.6 Design Parameters (on page 1502)

If the value of the PLG parameter is set to 1, the following two checks will be performed as per Cl. NA.20.(2) and NA.20(3) respectively:

- Cl. NA.20.(2): The following condition will be checked

  $$n/\chi \text{ and } + C_{my}\, m_y/\chi_{LT} + C\ mz\ m \text{ with} \leq 1\text{-}\ \Delta_0\ (I = y \text{ or } z)$$

  where

n	=	N_{Ed}/N_{Rd}
m_y	=	$max\ M_{y,Ed}\ (+\ \Delta\ M_{y,\ Ed})/M_{y,\ Rd};\ m_z = max\ M_{,Z\ Ed}\ (+\ \Delta\ M_{,\ Ed})/M_{Z\ Rd,}$
χ and	=	buckling factor
χ_{LT}	=	LTB factor
C_m	=	moment factor from table B 3 of PN EN 1993-1-1
Δ_0	=	correction factor (estimation of maximum reduction) and will be worked out as:

  Δ_0 = 0,1 + 0,2 (w_i – 1), przy czym $w_i = W_{pl,i}/W_{el,i}$, or
  Δ_0 = 0,1 – in case of class 3 and 4 sections.

- Cl. NA.20.(3): This condition will only be checked for circular hollow sections.

  $$n/\chi_i + [(k_{ii}\ m_i)^2 + (C_{mj}\ m_j)^2]\ 1/2 \leq 1\ (i,j = y,z)$$

  where

k	=	the interaction factor from table B.1 of PN-EN 1993-1-1
n, m, and Cmj	=	as previously described

If the PLG parameter has been set to 1, the maximum among the following ratios will be taken as being critical for Cl 6.3.3:

6.3.3: Eqn6.61
6.3.3: Eqn6.62
NA.20(2) and

NA.20(3)

If however `PLG` has been set to `0` (or not specified at all), the program will ignore the last two checks in the list above.

D5.D.8.8 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of PN EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. PN EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,TF}$ and $N_{cr,T}$ (refer 6.3.14 of PN EN 1993-1-1:2005).

The NCCI document "SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes" provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the Polish NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.9 Singaporean National Annex to EC3

Adds values from the Singaporean National Annex - titled *National Annex to Standard SS-EN 1993-1-1* - for use with Eurocode 3, or EN 1993-1-1:2010. The NA document makes small changes to the base document.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the Singaporean-NA.

The following clauses are not implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	The SINGAPORE NA specifies the value of λc0 for I, H channel or box section to be used in equation 6.59 of SS EN 1993-1-1:2010 as 0.4. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Singaporean National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	The value of the modification factor kfl to be used in equation 6.60 of SS EN 1993-1-1. However, STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Singaporean National Annex.

The clauses/sections in EN 1993-1-1:2010 (hereafter referred to as EC-3) that have been dealt with in the Singaporean National Annex (hereafter referred to as SS-NA) and that are relevant to the proposed implementation are:

Related Links

- *V. EC3 Singapore NA - Channel Section with Conc Load* (on page 4181)

D5.D.9.1 Clause 6.1 – General

The partial safety factors will use the following values:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = 1.1

The design function in STAAD.Pro sets these values as the default values for the SS-NA (`NA 7` is specified)..

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters have been specified by the user as “0”, STAAD.Pro will ignore the specified value and use the default values as given above.

D5.D.9.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The Singaporean NA recommends the use of Table 6.3 and 6.4 of NF EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor X_{LT}, requires the calculation of the “Elastic Critical Buckling Moment”, Mcr. The Singaporean National Annex does not specify a particular method to calculate Mcr. Hence the calculation of Mcr has been based on the following NCCI documents:

SN003a-EN-EU – Elastic critical moment for Lateral torsional Buckling

This document provides a method to calculate “Mcr” specifically for doubly symmetric sections only. Hence only doubly symmetric sections will be considered for this method. The equation to evaluate Mcr is given in the NCCI as:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

C1 and C2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C1 and C2 for the different cases as given in the tables below:

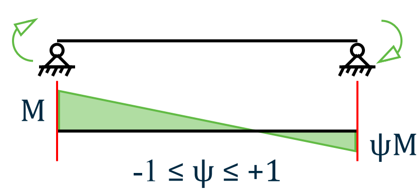


Table 154: Values of C_1 for end moment loading (for k=1)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05
-0,50	2,33
-0,75	2,57

This NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter.

SN030a-EN-EU – Mono-symmetrical uniform members under bending and axial compression:

This document provides a method to evaluate the elastic critical moment (Mcr) for uniform mono symmetric sections that are symmetric about the weak axis. Hence, the elastic critical moment for "Tee-Sections" will be evaluated using the method in this NCCI.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as :

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{(k_x L)^2} \left\{ \sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_z} + (C_2 z_g - C_3 z_1)^2} - C_2 z_g - C_3 z_1 \right\}$$

The factors C1, C2 and C3 are dependent on the end conditions and loading criteria. This implementation will consider C1, C2 and C3 as given in the tables below:

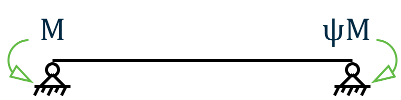
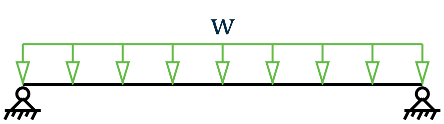
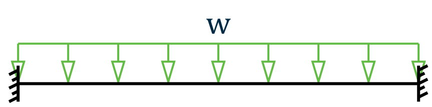
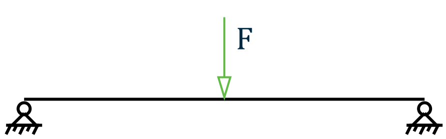
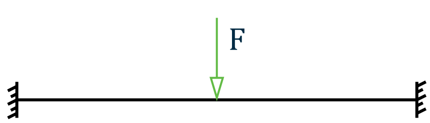


Table 155: Values of C_1 and C_2 for end moment loading (for $k_c = 1$)

Ψ	C_1	C_2
+1.00	1.00	1.00
+0.75	1.14	0.99
+0.50	1.31	0.99
+0.25	1.52	0.98
0.00	1.77	0.94
-0.25	2.05	0.85
-0.50	2.33	0.68
-0.75	2.57	0.37
-1.00	2.55	0.00

Table 156: Values of factors C_1, C_2, and C_3 for cases with transverse loading (for $k_c = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2	C_3
		1.13	0.45	0.52
		2.57	1.55	0.75
		1.35	0.63	1.73
		1.68	1.64	2.64

The CMM parameter specified during design input will determine the values of C1, C2 and C3. The default value of CMM is 0, which considers the member as a pin ended member with UDL along its span. This NCCI does not however consider the “end moments and transverse loading” condition. The user however can use the new “C1”, ‘C2’ and ‘C3’ parameters to input the required values for C1, C2 and C3 to be used in calculating Mcr.

Note: If "MU" as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = k_w = 1.0$). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of $K = k_w = 1$ is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

For the term z_j, please refer to Annex E of NBN NA 2018.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.9.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The Singaporean NA specifies different values for the $\lambda_{LT,0}$ and β factors to be used in equation 6.57 of SS EN 1993-1-1 for rolled and equivalent welded sections. STAAD.Pro does not differentiate between rolled and welded sections and uses the default values in SS EN 1993-1-1 for $\lambda_{LT,0}$ and β. The values specified in the Singapore NA are:

- For rolled sections and hot-rolled & cold formed hollow sections:

  $\lambda_{LT,0} = 0.4$

  $\beta = 0.75$

- For welded sections:

  $\lambda_{LT,0} = 0.2$

  $\beta = 1.00$

STAAD.Pro uses the buckling curves based on Table 6.5 of SS EN 1993-1-1:2005. The Singaporean-NA provides the values for the terms λ_{LT0} and β factors given in clause 6.3.2.3(1) as follows:

Table 157: Buckling curves to use with SS-EN 1993-1-1:2005

Cross Section	Limits	Buckling Curve
Rolled doubly symmetric I and H sections and hot-finished hollow sections	h/b ≤ 2	b
	2.0 < h/b ≤ 3.1	c
	h/b > 3.1	d

Cross Section	Limits	Buckling Curve
Angles (for moments in the major principle plane)		d
All other hot-rolled sections		d
Welded, doubly symmetric sections and cold-formed hollow sections	h/b ≤ 2	c
	2.0 < h/b ≤ 3.1	d

Note: This table does not specify which buckling curve is to be used in case of welded doubly symmetric sections with h/b ≥ 3.1 and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

D5.D.9.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Singaporean NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005) both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of SS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...”. Hence in the implementation of EC3 (and the Singaporean Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the Singaporean National Annex states that Table 6.5 in SS EN 1993-1-1:2005 should be replaced with the table given in the NA (See section 4.3 of this document). Hence for all cases dealt with by the table in the Singaporean NA, this implementation will choose the buckling curves from the Singaporean National Annex. For any case that is not dealt with by the table in the Singaporean NA, the program will use the method given in Cl. 6.3.2.2 of SS EN 1993-1-1:2005.

For the following cross sections, the program will use the Table in the Singaporean NA for choosing a buckling curve for LTB checks (when the SS EN has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section
- Welded doubly symmetric sections with h/b < 3.1

For the following cross sections, the program will use Cl. 6.3.2.3 of SS EN 1993-1-1:2005 to evaluate χ_{LT}

- Welded I & H Sections with h/b ≥ 3.1.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χ_{LT} .

In any case, the elastic critical moment, M_{cr}, (used to evaluate the non dimensional slenderness) will be evaluated as given above. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the `CMN` parameter=1.0 (See section above) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.9.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The Singaporean NA specifies the use of Equation 6.58 of SS EN 1993-1-1:2005 to evaluate the modification factor “f” for the LTB reduction factor χ_{LT}. To evaluate the modification factor SS EN 1993-1-1:2005 uses a correction factor “kc” given by Table 6.6 in the code.

The Singaporean-NA however, specifies that the correction factor “kc” is to be obtained as below:

$$K_c = 1 / \sqrt{C_1}$$

Where:

> C_1 is to be obtained from the NCCI documents as previously described (Refer to D5.D.9.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks (on page 1557)). The NCCI document SN003a-EN-EU specifies the values of C1 to be used in table 3.1 as shown below. The current implementation does not account for the K_c factor and conservatively uses a reduction factor equal to 1. The program allows for the reduction factor based on the Singaporean-NA.

These values are for an end restraint factor of k = 1 (i.e., design parameter `CMN` = 1.0). Hence for all other values of `CMN` (i.e., 0.7 or 0.5) this implementation will use the values of C_1 from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for K_c. However, you can also input a custom value of K_c by setting the design parameter `KC` to the desired value. If the `KC` parameter in the design input is set to 0, then the program will automatically calculate its value. This will cause the program to evaluate a value of C_1 corresponding to the end conditions and the Bending moment of the member and in turn calculate K_c as given in the NA. To evaluate C_1, the program will use the NCCI documents as previously described.

D5.D.9.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The Singaporean NA recommends the methods in either Annex A or Annex B of SS-EN 1993-1-1 to calculate these interaction factors. The current implementation of EC3 BS in STAAD.pro uses the method in Annex B by default. The proposed implementation of the Singaporean NA will also use Annex B for Cl.6.3.3 checks.

However for non-doubly symmetric sections, the Singaporean NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the Singaporean NA). The Singaporean NA requires additional checks to be done to check for the maximum allowable values of λ and X to be used in equations 6.61 and 6.62 of SS EN 1993-1-1:2005.

As per the Singaporean NA, for non-doubly symmetric sections, the slenderness about the weak axis (λy in STAAD) and the corresponding reduction factor χy should be taken as the values from the highest values of slenderness (λ) among the flexural buckling slenderness (λy), torsional slenderness (λT) and torsional-flexural slenderness (λTF) as given in Clauses 6.3.1.3 and 6.3.1.4 of SS EN 1993-1-1:2005. Hence for non-doubly symmetric sections the program will calculate the critical non-dimensional slenderness as:

λ_y = the maximum of either λ from Cl 6.3.1.3 or λ_T from Cl 6.3.1.4

Where:

$$\lambda_T = \sqrt{\frac{A \cdot f_y}{N_{cr}}}$$

$N_{cr} = \min(N_{CrT}, N_{crTF})$.

The Singaporean NA or EC3 does not, however, specify a method to evaluate N_{CrT} or N_{crTF}. Therefore, the program uses the method specified in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” to calculate these. Refer to D5.D.9.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling (on page 1563).

Note: The Singaporean National Annex or EC3 does not deal with angle sections in specific and hence this implementation will use the method used in the current EC3 implementation to deal with slenderness of angle sections. In the current implementation this is done as per cl 4.7.10 of BS 5950. This proposed implementation will still use the same method for single and double angle sections to evaluate the slenderness.

Clause NA 3.2 of the Singaporean NA also requires that “Where the section is not an I Section or a hollow section and is a class1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause”. Hence, for all Class 1 or Class 2 cross sections that are *not* I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.

D5.D.9.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of SS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness parameter, λ_T, to be used for torsional and torsional-flexural buckling checks. The SS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SS EN 1993-1-1:2005). Therefore, the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,T,F}$ and $N_{cr,T}$ factors and hence will to be included in this implementation of the Singaporean NA.

The program will only consider Channel Sections and Tee- sections while working out the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.10 Belgian National Annex to EC3

Adds values from the Belgian National Annex—titled *National Annex to Standard NBN-EN 1993-1-1 ANB: 2018*—for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Belgian National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Belgian National Annex.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the NBN-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Belgian National Annex (hereafter referred to as NBN-NA) and that are relevant to the proposed implementation are:

Related Links

- *V. EC3 Belgian NA - Channel Section with Conc Load* (on page 4085)
- *V. EC3 Belgian NA - Column with Axial Load* (on page 4088)
- *V. EC3 Belgian NA - I Section with Conc Load* (on page 4096)
- *V. EC3 Belgian NA - I Section with UDL* (on page 4099)
- *V. EC3 Belgian NA - Tee Section* (on page 4104)
- *V. EC3 Belgian NA - Varying End Moments* (on page 4108)

D5.D.10.1 Clause 6.1(1) – General

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Belgian National Annex:

- Resistance of cross-sections, $\gamma_{M0} = 1.0$
- Resistance of members to instability, $\gamma_{M1} = 1.0$
- Resistance of cross sections to tension, $\gamma_{M2} = 1.25$

The design function in STAAD.Pro sets these values as the default values for the NBN-NA (`NA 8` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user-specified value (i.e., 0) and use the default values as given above.

D5.D.10.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The NBN-NA recommends the use of Table 6.3 and 6.4 of EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the Elastic Critical Buckling Moment, M_{cr}. The NBN-NA gives a method to calculate M_{cr} in Annex E, which is used by STAAD.Pro. Annex E, however, only deals with the calculation of M_{cr} for doubly symmetric sections and mono symmetric sections that are symmetric about the minor axis (i.e, Tee sections). For any other type of section that is not dealt with by Annex E, STAAD.Pro uses the method and tables given in Annex F of DD ENV 1993-1-1:1992:

Doubly symmetric sections

Annex D of NBN-NA provides equation used to calculate M_{cr} specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

C_1 & C_2 are factors that depend on the end conditions and the loading conditions. The Annex provides values for C_1 & C_2 for the different cases as given in Table1 and Table 2 of the Annex. Table 1 deals with the condition of a simply supported member with end moments and the value of C_1 is determined by the end moment ratio (Refer to the NA for details). Clause 3.2 of the National Annex however gives a formula to calculate C_1 as:

$$C_1 = 1.77 - 1.04\psi + 0.27\psi^2 \leq 2.60$$

The value of C_2 is determined based on the Table 2 of the Annex, based on the loading and end conditions as specified using the `CMM` parameter.

This NBN-NA considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter.

Mono-symmetric sections with symmetry about their weak axis

Annex D of NBN-NA also provides a method to evaluate the elastic critical moment, M_{cr}, for uniform mono symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for Tee-Sections is evaluated using the method in this Annex.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro does not have a means to specify/identify a mono-symmetric built-up section. Hence, this implementation will use this method only for Tee-sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{(k_x L)^2} \left\{ \sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I} + \frac{(k_x L)^2 GI_T}{\pi^2 EI_z} + (C_2 z_g - C_3 z_j)^2} - C_2 z_g - C_3 z_j \right\}$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. This implementation will consider C_1, C_2 and C_3 as given in the tables below:

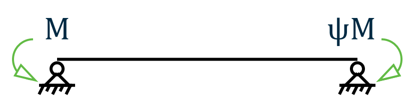


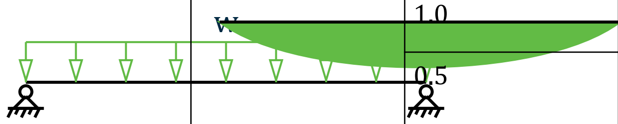
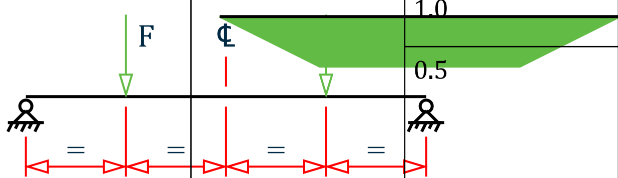
Figure 169: End moments and support conditions

Table 158: Critical moment coefficients for singly symmetric sections with end moments

Bending moment diagram	k_z	Value of coefficients		
		C_1	C_3	
			$\psi_f \leq 0$	$\psi_f > 0$
$\psi = +1$	1.0	1.00	1.000	
	0.5	1.05	1.019	
$\psi = +3/4$	1.0	1.14	1.000	
	0.5	1.19	1.017	
$\psi = +1/2$	1.0	1.31	1.000	
	0.5	1.37	1.000	
$\psi = +1/4$	1.0	1.52	1.000	
	0.5	1.60	1.000	
$\psi = 0$	1.0	1.77	1.000	
	0.5	1.86	1.000	
$\psi = -1/4$	1.0	2.06	1.000	0.850
	0.5	2.15	1.000	0.650
$\psi = -1/2$	1.0	2.35	1.000	$1.3 - 1.2\psi_f$
	0.5	2.42	0.950	$0.77 - \psi_f$
$\psi = -3/4$	1.0	2.60	1.000	$0.55 - \psi_f$
	0.5	2.45	0.850	$0.35 - \psi_f$
$\psi = -1$	1.0	2.60	$-\psi_f$	$-\psi_f$
	0.5	2.45	$0.125 - 0.7\psi_f$	$-0.125 - 0.7\psi_f$

Note: According to Section 3(1): $C_2 z_g = 0$

Table 159: Value of coefficients

Load and support conditions	Bending moment diagram	k_z	Value of coefficients		
			C_1	C_2	C_3
		1.0	1.12	0.45	0.525
		0.5	0.97	0.36	0.478
		1.0	1.35	0.59	0.411
		0.5	1.05	0.48	0.338
		1.0	1.04	0.42	0.562
		0.5	0.95	0.31	0.539

The CMM parameter specified during design input will determine the values of C_1, C_2, and C_3. The default value of CMM is 0, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the "end moments and transverse loading" condition. You can use the parameter MU to describe the moment distribution for cases where the end moments vary (i.e., CMM 7 or CMM 8). Alternatively, you can use the C1, C2, and C3 parameters to input the required values for C_1, C_2, and C_3 to be used in calculating M_{cr}.

Note: If MU as well as C1, C2, and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member ($k = k_w = 1.0$). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of $K = k_w = 1$ is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

For the term z_j, please refer to Annex E of NBN NA 2018.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The NBN-NA recommends the use of the values specified in EN 1993-1-1 for the LTB factors λ_{LT0} and β. However it gives two different sets of values for λ_{LT0} and β based on two different conditions as give below:

1. If M_{cr} is determined by considering the properties of the gross cross section and the lateral restraints, the following values are used:

   λ_{LT0} =0.2 and $\beta = 1.0$
2. If M_{cr} is determined by ignoring the lateral restraints, the following values are used:

   λ_{LT0} =0.4 and $\beta = 0.75$

The program evaluates which factors to use based on the CMN parameter. If CMN = 1.0 (default) or = 0.7, then the program assumes the restraints are ignored and the second set of values is used for λ_{LT0} and β. If CMN = 0.5, then the first set of λ_{LT0} and β values is used.

These factors are then applied to equation 6.57 of NBN-EN to evaluate the Lateral Torsional Buckling reduction factor χ_{LT}.

D5.D.10.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Belgium NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005) both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of NBN-EN 1993-1-1:2018.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for “Rolled I Sections”, “Welded I Sections” and “Any other sections”. Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with “Rolled I Sections” and “Welded I Sections”.

Cl. 6.3.2.2 states “Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...”. Hence in the implementation of EC3 (and the Belgian Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the Belgian National Annex gives equations to evaluate the imperfection factors to be used for various section types. (Refer to D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section (on page 1568)). Hence for all cases dealt with by the equations in the NBN-NA, this implementation will use Cl 6.3.2.3 to evaluate χ_{LT}.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χ_{LT} .

In any case, the elastic critical moment, Mcr, (used to evaluate the non dimensional slenderness) will be evaluated as given above. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (Refer to D5.D.10.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section (on page 1568)) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

You can override the default behavior and specify the clause that is to be used for LTB checks. This can be specified using the `MTH` design parameter (Refer to D5.C.6 Design Parameters (on page 1502)).

> **Note:** If a National Annex has not been specified (i.e., `NA` parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of NBN-EN 1993-1-1:2018. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., `SBLT` parameter = 1.0 in design input).

D5.D.10.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The Belgian NA specifies that the modification factor is to be obtained as per the default method given in EC-3. Hence the proposed implementation will use the existing functionality to work out the correction factor “kc” to be used in the modification factor f.

The program uses a default value of 1.0 for “kc”. However, you can also input a custom value of “kc” by setting the design parameter `KC` to the desired value. You can also get the program to calculate the value of “kc” automatically by setting the value of the `KC` parameter in the design input to 0. This will cause the program to work out “kc” from table 6.6 of NBN EN 1993-1-1:2018. This will correspond to the end conditions and the bending moment of the member (i.e., the value of `CMM` parameter specified).

- For `CMM` = 7 the program will choose the value of “kc” to be either 0.90 or 0.91 based on the end moment ratio.
- For `CMM` = 8 the program will choose the value of “kc” to be either 0.77 or 0.82 based on the end moment ratio.

An additional check will also be performed as given below:

$$\chi_{LT,\mathrm{mod}} \leq \frac{1}{\lambda_{LT}^{2}}$$

D5.D.10.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The NBN-NA recommends the equations in Annex A of NBN-EN 1993-1-1 to calculate these interaction factors.

The NA also mentions that torsional flexural buckling needs to be taken into account in case of mono symmetric sections. Torsional flexural buckling will need to be taken into account based on the method given in the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes”. See section below for details.

The NA also recommends a lower limit as given below for the term $C_{mi,0}$ in table A.2 of Annex A:

$$C_{mi,0} \geq 1 - \frac{N_{Ed}}{N_{cr,i}}$$

D5.D.10.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of NBN-EN 1993-1-1:2018 are to be used to calculate the non-dimensional slenderness parameter, λ_T, to be used for torsional and torsional-flexural buckling checks. The NBN-EN 1993-1-1:2018 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SS EN 1993-1-1:2005). Therefore, the NCCI document “SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes” provides methods to calculate the $N_{cr,T,F}$ and $N_{cr,T}$ factors and hence will to be included in this implementation of the Belgian NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

The program will only consider channel sections and Tee- sections while working out the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

D5.D.11 Malaysian National Annex to EC3

Adds values from the Malaysian National Annex—titled *National Annex to Standard MS-EN 1993-1-1*—for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Malaysian National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the Malaysian National Annex.

Note: Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the MS-NA.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that have been dealt with in the Malaysian National Annex (hereafter referred to as MS-NA) and that are relevant to the proposed implementation are:

D5.D.11.1 Clause 6.1(1) – General: Partial Safety Factors for buildings

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are γ_{M0}, γ_{M1}, and γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Malaysian National Annex:

- Resistance of cross-sections, γ_{M0} = 1.0
- Resistance of members to instability, γ_{M1} = 1.0
- Resistance of cross sections to tension, γ_{M2} = 1.1

The design function in STAAD.Pro sets these values as the default values for the MS-NA (`NA 9` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

D5.D.11.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The MS-NA recommends the use of Table 6.3 and 6.4 of MS EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the Elastic Critical Buckling Moment, M_{cr}. The MS-NA does not specify a particular method to calculate Mcr. Hence the calculation of Mcr has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU *NCCI: Elastic critical moment for lateral torsional buckling* provides equation used to calculate M_{cr} specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2}\left\{\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_S} + \frac{(kL)^2 GI_t}{\pi^2 EI_S} + (C_2 Z_g)^2} - C_2 Z_g\right\}$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter. The values of C_1 and C_2 may also be directly specified using the `C1` and `C2` parameters, respectively (required for `CMM = 7` or `CMM = 8`).

Mono-symmetric sections with symmetry about their weak axis

Annex D of MS-NA also provides a method to evaluate the elastic critical moment, M_{cr}, for uniform mono symmetric sections that are symmetric about the weak axis. Hence for this implementation the elastic critical moment for Tee-Sections is evaluated using the method in this Annex.

Note: Though this method could also be applicable to mono-symmetric built-up sections, STAAD.Pro currently does not have a means to specify/identify a mono-symmetric built-up section. Hence this implementation will use this method only for Tee-Sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{(k_x L)^2}\left\{\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I} + \frac{(k_x L)^2 GI_T}{\pi^2 EI_z} + (C_2 z_g - C_3 z_1)^2} - C_2 z_g - C_3 z_1\right)\right\}$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. The program considers C_1, C_2, and C_3 as given in the tables 4.1 and 4.2 of the NCCI, based on the `CMM` parameter.

The default value of `CMM = 0`, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You use the `C1`, `C2` and `C3` parameters to input the required values for C_1, C_2, and C_3, respectively, to be used in calculating M_{cr}.

Note: If "MU" as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Note: When `CMM = 7` or `CMM = 8`, the values for `C1`, `C2` and `C3` parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the `CMN` parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the `ZG` parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.11.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The MS-NA specifies different values for the $\lambda_{LT,0}$ and β factors to be used in equation 6.57 of MS EN 1993-1-1 for rolled and equivalent welded sections. STAAD.Pro does not differentiate between rolled and welded sections and uses the default values in MS EN 1993-1-1 for $\lambda_{LT,0}$ and β. The values specified in the MS-NA are:

1. For rolled sections and hot-rolled & cold formed hollow sections:

   $\lambda_{LT,0}$ = 0.4 and β = 0.75
2. For welded sections:

   $\lambda_{LT,0}$ = 0.2 and β = 1.00

STAAD.Pro uses the buckling curves based on Table 6.5 of MS EN 1993-1-1:2005, based on different limits. This table again does not specify which buckling curve is to be used in case of welded doubly symmetric sections with $h/b \geq 3.1$ and welded non-doubly symmetric sections. Hence for these cases the new implementation will still use the method specified in the base code as per clause 6.3.2.2(2).

D5.D.11.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Malaysian NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of MS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...". Hence in the implementation of EC3 (and the MS NA) in STAAD.Pro, by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Cl. 6.3.2.3 in the MS NA states that Table 6.5 in MS EN 1993-1-1:2005 should be replaced with the table given in the NA (Refer to D5.D.11.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section (on page 1572)). Hence for all cases dealt with by the table in the MS NA, this implementation will choose the buckling curves from the MS NA. For any case that is not dealt with by the table in the MS NA, the program will use the method given in Cl. 6.3.2.2 of MS EN 1993-1-1:2005.

Hence for the following cross sections the program will use the Table in the MS NA for choosing a buckling curve for LTB checks (when the MS NA has been specified):

- Rolled doubly symmetric I & H Sections
- Rolled doubly symmetric hollow sections (SHS, RHS, CHS)
- Angle Sections
- Any other rolled section
- Welded doubly symmetric sections with h/b < 3.1

For the following cross sections, the program will use Cl. 6.3.2.3 of MS EN 1993-1-1:2005 to evaluate χ_{LT}

- Welded I & H Sections with $h/b \geq 3.1$.

For any other type of cross section that is not dealt with by the National Annex or Cl.6.3.2.3, the program will use Cl. 6.3.2.2 to evaluate χ_{LT} .

In any case the Elastic critical moment "Mcr" (used to evaluate the non dimensional slenderness) will be evaluated as described in D5.D.11.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1571). Since the MS NA uses the NCCI documents mentioned in the sections above, this implementation will only consider end restraint conditions corresponding to the `CMN` parameter=1.0. For all other cases of the `CMN` parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

> **Note:** If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of MS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., `SBLT` parameter = 1.0 in design input).

D5.D.11.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The MS NA specifies the use of eqn. 6.58 of MS EN 1993-1-1:2005 to evaluate the modification factor, *f*, for the LTB reduction factor χ_{LT}. To evaluate the modification factor MS EN 1993-1-1:2005 uses a correction factor, kc, given by Table 6.6 in the code.

The program does not calculate the kc factor and conservatively uses a reduction factor equal to 1. The proposed implementation will allow for the reduction factor based on the MS NA.

These values are for an end restraint factor of k = 1 (i.e., `CMN` = 1.0). Hence for all other values of `CMN` (i.e., 0.7 or 0.5), the program uses the values of C1 from DD ENV 1993-1-1:1992 Annex F.

You can also manually specify a value for kc by setting the design parameter, `KC`, to the desired value. The user can also get the program to calculate the value of kc automatically by setting the value of the `KC` parameter in the design input to 0. This will cause the program to evaluate a value of C_1 corresponding to the end conditions and the Bending moment of the member and in turn calculate k_c as given in the NA. To evaluate C_1, the program will use the NCCI documents (Refer to D5.D.11.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1571)).

Note that for the MS NA, the program will attempt to evaluate k_c by default using the equation in NA,

$$kc = 1/\sqrt{C_1}$$

where C1 will be the value used for the M_{cr} calculations.

If k_c evaluates to be greater than 1.0, the program will then evaluate kc as per Table 6.6 of EN 1993-1-1:2005.

D5.D.11.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The MS NA recommends that the method in Annex A or Annex B of MS EN 1993-1-1:2005 can be used to calculate the interaction factors for Cl. 6.3.3 checks in the case of doubly symmetric sections. STAAD.Pro uses the equations in Annex B of MS EN 1993-1-1:2005 to calculate these interaction factors for doubly symmetric sections..

However, for non-doubly symmetric sections, the MS NA gives the option of using Annex B with some modifications as given in the NA. (Cl. NA-3.2 of the MS NA). The MS NA requires additional checks to be done to check for the maximum allowable values of λ and X to be used in equations 6.61 and 6.62 of MS EN 1993-1-1:2005.

As per the MS NA, for non-doubly symmetric sections, the slenderness about the weak axis (λ_y in STAAD.Pro) and the corresponding reduction factor χ_y should be taken as the values from the highest values of slenderness (λ) among the flexural buckling slenderness (λ_y), torsional slenderness (λ_T) and torsional-flexural slenderness (λ_{TF}) as given in Clauses 6.3.1.3 and 6.3.1.4 of MS EN 1993-1-1:2005. Hence for non-doubly symmetric sections the program will calculate the critical non-dimensional slenderness as:

$$\lambda_y = \max \begin{cases} \lambda \text{per Cl. 6.3.1.3} \\ \lambda_T \text{per Cl. 6.3.1.4} \end{cases}$$

where

$$\lambda_T = \sqrt{\frac{A \cdot f_y}{N_{cr}}}$$

$$N_{cr} = \min\left(N_{crT}, N_{crTF}\right)$$

The MS NA or EC3 does not, however, specify a method to evaluate N_{crT} or N_{crTF}. Hence, the program uses the method specified in the NCCI document *SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes* to calculate these. Refer to D5.D.11.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling (on page 1575) for details.

Note: The MS NA or EC3 does not deal with angle sections specifically and therefore STAAD.Pro uses the method described in the EC3 implementation to deal with slenderness of angle sections. This is done as per cl 4.7.10 of BS 5950.

Clause NA 3.2 of the MS NA also requires that “Where the section is not an I Section or a hollow section and is a class1 or class 2 section, it will be treated as a class 3 section for the purposes of this clause”. Hence for all Class 1 or Class 2 cross sections that are *not* I, H, SHS, RHS or CHS sections, the elastic properties will be used for the purposes of 6.3.3 checks.

D5.D.11.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of MS-EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness parameter, λ_T, to be used for torsional and torsional-flexural buckling checks. The MS-EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of SS EN 1993-1-1:2005). Therefore, the NCCI document *SN001a-EN-EU: Critical axial load for torsional and flexural torsional buckling modes* provides methods to calculate the $N_{cr,T,F}$ and $N_{cr,T}$ factors and hence will to be included in this implementation of the MS NA.

The program will only consider Channel Sections and Tee- sections when evaluating the critical torsional and Flexural Torsional buckling loads as per Cl 6.3.1.4.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.12 German National Annex to EC3

Adds values from the German National Annex—titled DIN EN 1993-1-1:2005—for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the German National Annex (hereafter referred to as DE-NA) are described in the following sections.

Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the DE-NA.

The following clauses are *not* implemented in STAAD.Pro:

Clause 6.3.2.4(1) B – Slenderness for flexural buckling	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the German National Annex.
Clause 6.3.2.4(2)B – Modification factor "kfl"	STAAD.Pro does not use this clause for design per EC-3. Therefore, this clause is ignored for the German National Annex.

D5.D.12.1 Clause 6.1(1) – General: Partial Safety Factors for buildings

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are Γ_{M0}, Γ_{M1}, and Γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the German National Annex:

- Resistance of cross-sections, Γ_{M0} = 1.0
- Resistance of members to instability, Γ_{M1} = 1.10

- Resistance of cross sections to tension, Γ_{M2} = 1.25

The design function in STAAD.Pro sets these values as the default values for the DE-NA (`NA 10` is specified).

Note: You can change these values through the `GM0`, `GM1`, & `GM2` design parameters. Refer to D5.C.6 Design Parameters (on page 1502)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

D5.D.12.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The DE-NA recommends the use of Table 6.3 and 6.4 of DIN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the Elastic Critical Buckling Moment, M_{cr}. The DE-NA does not specify a particular method to calculate Mcr. Hence the calculation of Mcr has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU *NCCI: Elastic critical moment for lateral torsional buckling* provides equation used to calculate M_{cr} specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2}\left\{\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_S} + \frac{(kL)^2 GI_t}{\pi^2 EI_S} + (C_2 Z_g)^2} - C_2 Z_g\right\}$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter. The values of C_1 and C_2 may also be directly specified using the `C1` and `C2` parameters, respectively (required for `CMM = 7` or `CMM = 8`).

Mono-symmetric sections with symmetry about their weak axis

Annex D of DE-NA also provides a method to evaluate the elastic critical moment, M_{cr}, for uniform mono-symmetric sections that are symmetric about the weak axis. STAAD.Pro uses this method for the evaluating the elastic critical moment for Tee sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{(k_x L)^2}\left\{\sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I} + \frac{(k_x L)^2 GI_T}{\pi^2 EI_z} + (C_2 z_g - C_3 z_1)^2} - C_2 z_g - C_3 z_1\right\}$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. The program considers C_1, C_2, and C_3 as given in the tables 4.1 and 4.2 of the NCCI, based on the `CMM` parameter.

The default value of `CMM = 0`, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You use the C1, C2 and C3 parameters to input the required values for C_1, C_2, and C_3, respectively, to be used in calculating M_{cr}.

Note: If "MU" as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Note: When CMM = 7 or CMM = 8, the values for C1, C2 and C3 parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the CMN parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the ZG parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.12.3 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The DE-NA specifies that the default values in Table 6.5 for the $\lambda_{LT,0}$ and β factors given in clause 6.3.2.3(1) as follows:

For all sections, use:

$\lambda_{LT,0}$ =0.4 and β = 0.75

D5.D.12.4 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per German NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of DIN EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from...". Hence in the implementation of EC3 (and the German Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Note: The MTH design parameter can be used to control the choice of the clause used to calculate χ_{LT}.

In any case, the elastic critical moment, Mcr, (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the CMN parameter=1.0 (Refer to D5.D.12.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1576)) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of DIN EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.12.5 Clause 6.3.2.3(2) – Modification factor, f, for LTB checks

The DE-NA specifies the use of Equation 6.58 of DIN EN 1993-1-1:2005 to evaluate the modification factor “f” for the LTB reduction factor χ_{LT}. To evaluate the modification factor SS EN 1993-1-1:2005 uses a correction factor “kc” given by Table 6.6 in the code.

The DE-NA however, specifies that the correction factor “kc” is to be obtained as below:

$$K_c = 1/\sqrt{C_1}$$

Where:

C_1 is to be obtained from the NCCI documents as previously described (Refer to D5.D.12.2 Clause 6.3.2.2 – Elastic critical moment and imperfection factors for LTB checks (on page 1576)). The NCCI document SN003a-EN-EU specifies the values of C1 to be used in table 3.1 as shown below. The current implementation does not account for the K_c factor and conservatively uses a reduction factor equal to 1. The program allows for the reduction factor based on the DE-NA.

These values are for an end restraint factor of k = 1 (i.e., design parameter CMN = 1.0). Hence for all other values of CMN (i.e., 0.7 or 0.5) this implementation will use the values of C_1 from DD ENV 1993-1-1:1992 Annex F.

The program will use a default value of 1.0 for K_c. However, you can also input a custom value of K_c by setting the design parameter KC to the desired value. If the KC parameter in the design input is set to 0, then the program will automatically calculate its value. This will cause the program to evaluate a value of C_1 corresponding to the end conditions and the Bending moment of the member and in turn calculate K_c as given in the NA. To evaluate C_1, the program will use the NCCI documents as previously described.

D5.D.12.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The DE-NA recommends the use of equations in Annex A or Annex B of DIN-EN 1993-1-1 to calculate these interaction factors. STAAD.Pro uses the method in Annex B by default. Thus the program uses Annex B for Cl. 6.3.3 checks.

The DE-NA or EC3 do not deal with angle sections in specific and thus the program uses the method per cl 4.7.10 of BS 5950 for single and double angle sections to evaluate the slenderness.

D5.D.12.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of DIN EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. DIN EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of DIN EN 1993-1-1:2005).

The NCCI document SN001a-EN-EU: *Critical axial load for torsional and flexural torsional buckling modes* provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the DE-NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.D.13 Swedish National Annex to EC3

Adds values from the Swedish National Annex—titled BFS EN 1993-1-1:2005—for use with Eurocode 3, or EN 1993-1-1:2005. The NA document makes small changes to the base document.

The clauses/sections in EN 1993-1-1:2005 (hereafter referred to as EC-3) that require additional clauses from the Swedish National Annex (hereafter referred to as SW-NA) are described in the following sections.

Refer to the D5.C. European Codes - Steel Design to Eurocode 3 [EN 1993-1-1:2005] (on page 1478) for a description of these clauses. The sections below refer to the corresponding clauses in the SW-NA.

D5.D.13.1 Clause 3.2.(2) - Steel Grades

The Swedish NA allows the use of custom steel grades as given in Table E-1 of the National Annex. These grades of steel can be specified by using the `PY` (Yield Strength) and `FU` (Ultimate Strength) parameters in STAAD.Pro.

D5.D.13.2 Clause 6.1(1) – General: Partial Safety Factors for Buildings

EN 1993-1-1:2005 specifies the use of the partial safety factors to be used in for design as given in Cl. 6.1 of the code. These factors are Γ_{M0}, Γ_{M1}, and Γ_{M2}. EN 1993 provides default values for these factors. However, any National Annex is allowed to override these default values.

The partial safety factors will use the following values for the Swedish National Annex:

- Resistance of cross-sections, Γ_{M0} = 1.0
- Resistance of members to instability, Γ_{M1} = 1.0
- Resistance of cross sections to tension, Γ_{M2} = $0.9\ x\ f_u/f_y$, but not less than 1.1

Where:

f_u is the ultimate steel strength
f_y is the yield strength of steel

The design function in STAAD.Pro sets these values as the default values for the SW-NA (`NA 10` is specified).

Note: You can change these values through the GM0 and GM1 design parameters. Refer to D5.C.6 Design Parameters (on page 1502) The value of GM2 (Γ_{M2}) is calculated based on the steel grade values specified. Refer to D5.D.13.1 Clause 3.2.(2) - Steel Grades (on page 1579)

Note: If any of these parameters are specified as `0`, STAAD.Pro will ignore the user specified value (i.e., 0) and use the default values as given above.

D5.D.13.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks

The SW-NA recommends the use of Table 6.3 and 6.4 of DIN EN 1993-1-1:2005 to calculate the imperfection factors for Lateral Torsional Buckling (LTB) checks.

The calculation of the LTB reduction factor χ_{LT}, requires the calculation of the Elastic Critical Buckling Moment, M_{cr}. The SW-NA does not specify a particular method to calculate Mcr. Hence the calculation of Mcr has been based on the following NCCI documents:

Doubly symmetric sections

SN003a-EN-EU *NCCI: Elastic critical moment for lateral torsional buckling* provides equation used to calculate M_{cr} specifically for doubly symmetric sections:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_S} + \frac{(kL)^2 G I_t}{\pi^2 E I_S} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

C_1 and C_2 are factors that depend on the end conditions and the loading conditions of the member. The NCCI provides values for C_1 and C_2 for the different cases as given in Table 3.1 and Table 3.2.

The NCCI considers three separate loading conditions:

- Members with end moments
- Members with transverse loading
- Members with end moments and transverse loading.

STAAD.Pro accounts for the loading condition and the bending moment diagram through the `CMM` parameter. The values of C_1 and C_2 may also be directly specified using the `C1` and `C2` parameters, respectively (required for `CMM = 7` or `CMM = 8`).

Mono-symmetric sections with symmetry about their weak axis

Annex D of SW-NA also provides a method to evaluate the elastic critical moment, M_{cr}, for uniform mono-symmetric sections that are symmetric about the weak axis. STAAD.Pro uses this method for the evaluating the elastic critical moment for Tee sections.

The equation to evaluate M_{cr} for mono symmetric sections is given as:

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{(k_x L)^2} \left\{ \sqrt{\left(\frac{k_x}{k_w}\right)^2 \frac{I_w}{I} + \frac{(k_x L)^2 G I_T}{\pi^2 E I_z} + (C_2 z_g - C_3 z_1)^2} - C_2 z_g - C_3 z_1 \right\}$$

The factors C_1, C_2, and C_3 are dependent on the end conditions and loading criteria. The program considers C_1, C_2, and C_3 as given in the tables 4.1 and 4.2 of the NCCI, based on the `CMM` parameter.

The default value of `CMM = 0`, which considers the member as a pin ended member with uniformly distributed load (UDL) along its span. This NCCI does not however consider the “end moments and transverse loading” condition. You use the `C1`, `C2` and `C3` parameters to input the required values for C_1, C_2, and C_3, respectively, to be used in calculating M_{cr}.

Note: If "MU" as well as C1, C2 and C3 have been specified, the program will ignore MU and use the user input values of C1, C2 and C3. STAAD.Pro obtains these values from Annex F of DD ENV version of 1993-1-1:1992.

Note: When `CMM = 7` or `CMM = 8`, the values for `C1`, `C2` and `C3` parameters must be manually specified.

Both the NCCI documents mentioned above assume that the member under consideration is free to rotate on plan and that there are no warping restraints for the member (k = kw = 1.0). STAAD.Pro takes into account of the end conditions using the `CMN` parameter for EC3. A value of K = kw =1 is indicated by a value of CMN = 1.0 in the design input. Hence the above methods will be used only for members which are free to rotate on plan and which have no warping restraints (i.e., CMN = 1.0). For members with partial or end fixities (i.e., CMN = 0.5 or CMN = 0.7), this implementation will fall back on to the method and coefficients in DD ENV 1993-1-1:1992 – Annex F.

For all cases that are not dealt with by the National Annex (or the NCCI documents) this implementation will use the method as per the DD ENV 1993-1-1:1992 code.

The term z_g in the equation to calculate M_{cr} refers to the distance between the point of application of load on the cross section in relation to the shear center of the cross section. The value of z_g is considered positive, if the load acts towards the shear center and is negative if it acts away from the shear center. By default, the program will assume that the load acts towards the shear center at a distance equal to (Depth of section/2) from the shear center. The use will be allowed to modify this value by using the `ZG` parameter. Specifying a value of ZG = 0 in the design input would indicate that the load acts exactly at the shear center of the section so that the term z_g in the equation will have a value of zero.

Note: The program does not consider the case of cantilevers.

D5.D.13.4 Clause 6.3.2.3(1) – LTB for rolled sections or equivalent welded section

The SW-NA specifies that the following values for the $\lambda_{LT,0}$ and β factors:

For all sections, use:

$\lambda_{LT,0}$ =0.4 and β = 0.75

D5.D.13.5 Clauses 6.3.2.2 and 6.3.2.3 — Calculation of LTB Reduction factor, χ_{LT} as per Swedish NA

Clauses 6.3.2.2 and 6.3.2.3 (EN 1993-1-1:2005), both give equations to evaluate the LTB reduction factor χ_{LT} to be used in eqn. 6.55 of BFS EN 1993-1-1:2005.

Cl. 6.3.2.2 uses tables 6.3 and 6.4 to choose the buckling curve and the imperfection factors to be used for calculating χ_{LT}. Table 6.4 specifies the choice of buckling curves for "Rolled I Sections", "Welded I Sections" and "Any other sections". Cl 6.3.2.3 on the other hand uses tables 6.5 and 6.3 to choose the buckling curves and imperfection factors. Table 6.5 however only deals with "Rolled I Sections" and "Welded I Sections".

Cl. 6.3.2.2 states "Unless otherwise specified, see 6.3.2.3, for bending members of constant cross section the value of χ_{LT} should be determined from…". Hence in the implementation of EC3 (and the Swedish Annex) in STAAD.Pro: by default the program will consider clause Cl. 6.3.2.3 to evaluate χ_{LT}. For any case that is not dealt with by Cl. 6.3.2.3, the program will consider Cl. 6.3.2.2 to evaluate χ_{LT}.

Note: The `MTH` design parameter can be used to control the choice of the clause used to calculate χ_{LT}.

In any case, the elastic critical moment, Mcr, (used to evaluate the non dimensional slenderness) will be evaluated as previously given. Since this implementation uses the NCCIs mentioned in the sections above, only end restraint conditions corresponding to the `CMN` parameter=1.0 (Refer to D5.D.13.3 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1580)) will be considered. For all other cases of the CMN parameter values, this implementation will use the method specified in Annex F of DD ENV 1993-1-1:1992.

Note: If a National Annex has not been specified (i.e., NA parameter in the design input = 0), the program will use Cl. 6.3.2.3 only in the case of Rolled or welded I & H Sections. For all other cases, the program will use Cl. 6.3.2.2 of BFS EN 1993-1-1:2005. Also, I sections with plates will be treated as built-up sections only if the section has been explicitly specified as a built-up section (i.e., SBLT parameter = 1.0 in design input).

D5.D.13.6 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz

The SW-NA recommends the use of equations in Annex A of EN 1993-1-1:2005 to calculate these interaction factors, which are used by STAAD.Pro when the Swedish NA is selected.

The SW-NA or EC3 do not deal with angle sections in specific and thus the program uses the method per cl 4.7.10 of BS 5950 for single and double angle sections to evaluate the slenderness.

D5.D.13.7 Clause 6.3.1.4 - Slenderness for torsional and torsional-flexural buckling

Equations 6.52 and 6.53 of BFS EN 1993-1-1:2005 are to be used to calculate the non-dimensional slenderness λ_T, to be used for torsional and torsional-flexural buckling checks. BFS EN 1993-1-1:2005 does not provide equations to calculate the elastic critical loads $N_{cr,T,F}$ and $N_{cr,T}$ (refer 6.3.14 of BFS EN 1993-1-1:2005).

The NCCI document SN001a-EN-EU: *Critical axial load for torsional and flexural torsional buckling modes* provides methods to calculate the $N_{cr,TF}$ and $N_{cr,T}$ factors and therefore these methods are used to evaluate the elastic critical loads for the SW-NA.

The critical axial load for torsional buckling is evaluated as:

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$
i_y and i_z	=	the radius of gyration about the Y-Y (weak axis) and Z-Z (strong axis) respectively

The critical axial load for torsional-flexural buckling is evaluated as:

$$N_{cr,TF} = \frac{i_o^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_o^2}}\right]$$

For details on these equations, refer to the NCCI document SN001a-EN-EU.

D5.E. European Codes - Timber Design Per EC 5: Part 1-1

STAAD.Pro is capable of performing timber design based on the European code EC5 Part 1-1 *Eurocode 5: Design of timber structures - Part 1.1: General-Common rules and rules for buildings*.

D5.E.1 General Comments

Principles of Limit States Design of Timber Structures are used as specified in the code.

Design per EC5 is limited to the prismatic, rectangular shapes only. There is no Eurocode-specific timber section database or library consisting of pre-defined shapes for analysis or for design. The feature of member selection is thus not applicable to this code.

The design philosophy of this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all timber structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the STAAD.Pro implementation of EC 5. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

Axes convention in STAAD and EC5

STAAD.Pro defines the major axis of the cross-section as zz and the minor axis as yy. The longitudinal axis of the member is defined as x and joins the start joint of the member to the end with the same positive direction.

EC5, however, defines the principal cross-section axes in reverse to that of STAAD.Pro, but the longitudinal axis is defined in the same way. Both of these axes definitions follow the orthogonal right hand rule.

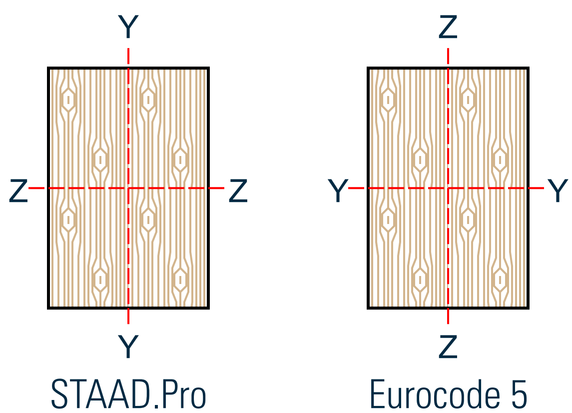


Figure 170: Axis conventions per STAAD and Eurocode 5

Determination of Factors

A. Kmod – Modification factor taking into account of Load-duration (LDC) and Moisture-content (Service Class - SCL). Reference Table 3.1 of EC-5-2004.

For “Solid Timber”, the values are incorporated in the program.

B. γ_m – Partial factor for Material Property values. Reference Table 2.3 of EC-5-2004.

For “Solid Timber”, the value of $\gamma_m = 1.3$ is incorporated in the program.

C. Kh – Size Factor.

For members, subjected to tension, whose maximum c/s dimension is less than the reference width in tension the characteristic strength in tension (ft0k) is to be increased by the factor Kh.

For members, subjected to bending, whose depth is less than reference depth in bending, the characteristic strength in bending (fmk) is to be increased by the factor Kh.

As per clause 3.2(3) of EC 5- 2004, for rectangular solid timber with a characteristic timber density $\rho_k \leq 700$ kg/m^3 the reference depth in bending or the reference width (maximum cross-sectional dimension) is 150 mm.

The value of Kh = Minimum of {(150/h) 0.2 and 1.3} for such solid timber is incorporated in the software. Please refer clause numbers 3.3 and 3.4 for the value of Kh for Glued laminated timber and Laminated veneer lumber respectively.

D. KC90 – Factor taking into account the load configuration, possibility of splitting and degree of compressive deformation.

For members, subjected to compression, perpendicular to the direction of grain alignment, this factor should be taken into account. Default value of 1 is used in STAAD.Pro. You may override the value. Please refer clause 6.1.5 of EC-5-2004 in this regard.

E. Km – Factor considering re-distribution of bending stress in cross section.

For members, subjected to bending, this factor is taken into account for stress checking. For rectangular section the value of Km is 0.7, and this value is incorporated in STAAD.Pro. You may override the value. Please refer clause 6.1.6 of EC-5-2004 in this regard.

F. Kshape – Factor depending on shape of cross section.

For members, subjected to torsional force, design torsional stress should be less than equal design shear strength multiplied by the factor Kshape. This factor is determined by STAAD.Pro internally using the guidelines of clause 6.1.8 of EC-5-2004.

D5.E.2 Analysis Methodology

Table 160: EC5 Nomenclature

Symbol	Description
S_{t0d}	Design tensile stress parallel (at zero degree) to grain alignment.
S_{t90d}	Design tensile stress perpendicular (at 90 degrees) to grain alignment.
S_{c0d}	Design compressive stress parallel to grain alignment.
S_{c90d}	Design compressive stress perpendicular to grain alignment.
S_{mzd}	Design bending stress about zz axis.
S_{myd}	Design bending stress about yy axis.
S_{vd}	Design shear stress.
S_{tor_d}	Design torsional stress.

Symbol	Description
F_{t0d}	Design tensile strength - parallel to the grain alignment.
F_{t90d}	Design tensile strength - perpendicular to the grain alignment.
F_{c0d}	Design compressive strength - parallel to the grain alignment.
F_{c90d}	Design compressive strength - perpendicular to the grain alignment.
F_{mzd}	Design bending strength - about zz-axis.
F_{myd}	Design bending strength - about yy-axis.
F_{vd}	Design shear strength about yy axis.
RATIO	Permissible ratio of stresses as input using the `RATIO` parameter. The default value is 1.
l_z ,$l_{rel,z}$	Slenderness ratios corresponding to bending about zz axis.
l_y,$l_{rel,y}$	Slenderness ratios corresponding to bending about yy axis.
$E_{0,05}$	Fifth percentile value of modulus of elasticity parallel to grain.
$G_{0,05}$	Fifth percentile value of shear modulus parallel to grain.
I_z	Second moment of area about the strong z-axis.
I_y	Second moment of area about the weak y-axis.
I_{tor}	Torsional moment of inertia.
f_{mk}	Characteristic bending strength.
b, h	Width and depth of beam.

Equations for Characteristic Values of Timber Species as per Annex-A of EN 338:2003

The following equations were used to determine the characteristic values:

For a particular Timber Strength Class (TSC), the following characteristic strength values are required to compute the other related characteristic values.

i. Bending Strength – $f_{m,k}$
ii. Mean Modulus of Elasticity in bending – $E_{0, mean}$
iii. Density - ρ_k

Sl No.	Property	Symbol	Wood Type	
			Softwood (C)	Hardwood (D)
1.	Tensile Strength parallel to grain	$f_{t,0,k}$	0.6 * $f_{m,k}$	
2.	Tensile Strength perpendicular to grain	$f_{t,90,k}$	Minimum of {0.6 and (0.0015*r_k)}	
3.	Compressive Strength parallel to grain	$f_{c,0,k}$	5 * $(f_{m,k})^{0.45}$	
4.	Compressive Strength perpendicular to grain	$f_{c,90,k}$	0.007*r_k	0.0015*r_k
5.	Shear Strength	$f_{v,k}$	Minimum of {3.8 and (0.2*$f_{m,k}^{\ 0.8}$)}	
6.	Modulus of Elasticity parallel to grain	$E_{0,05}$	0.67* $E_{0,mean}$	0.84* $E_{0,mean}$
7.	Mean Modulus of Elasticity perpendicular to grain	$E_{90,mean}$	$E_{0,mean}$ /30	$E_{0,mean}$ /15
8.	Mean Shear Modulus	G_{mean}	$E_{0,mean}$ /16	
9.	Shear Modulus	$G_{0,05}$	$E_{0,05}$ /16	

The values of the characteristic strengths computed using the above equations, may differ with the tabulated values in Table-1 of EN 338:2003. However, in all such cases, the values obtained from the provided equations are treated as actual and is used by the program, as the values of Table-1 are based on these equations.

Design values of Characteristic Strength

As per clause 2.4.1, Design values of a strength property shall be calculated as:

$$X_d = K\ mod \cdot (X_k/\gamma_m)$$

Where:

X_d is design value of strength property
X_k characteristic value of strength property
γ_m is partial factor for material properties.

The member resistance in timber structure is calculated in STAAD according to the procedures outlined in EC5. This depends on several factors such as cross sectional properties, different load and material factors, timber strength class, load duration class, service class and so on. The methodology adopted in STAAD for calculating the member resistance is explained here.

Check for Tension stresses

If the direction of applied axial tension is parallel to the direction of timber grain alignment, the following formula should be checked per Equation 6.1 of EC-5 2004:

$$S_{t0d}/F_{t0d} \leq$$

If the direction of applied axial tension is perpendicular to the direction of timber grain alignment, the following formula should be checked:

$$S_{t90d}/F_{t90d} \leq$$

Check for Compression stresses

If the direction of applied axial compression is parallel to the direction of timber grain alignment, the following formula should be checked per Equation 6.2 of EC-5 2004:

$$S_{c0d}/F_{c0d} \leq$$

If the direction of applied axial compression is perpendicular to the direction of timber grain alignment, the following formula should be checked per Equation 6.3 of EC-5 2004:

$$S_{t0d}/(F_{t0d} \cdot Kc90) \leq$$

Check for Bending stresses

If members are under bending stresses, the following conditions should be satisfied per Equations 6.11 and 6.12 of EC-5 2004.

Note: In STAAD z-z axis is the strong axis.

$$(S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \leq$$

$$Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \leq$$

Check for Shear stresses

Horizontal stresses are calculated and checked against allowable values per Equation 6.13 of EC-5 2004:

$$S_{vd}/F_{vd} \leq$$

Check for Torsional stresses

Members subjected to torsional stress should satisfy Equation 6.14 of EC-5 2004:

$$S_{tor\_d}/(Kshape \cdot F_{tor\_d}) \leq$$

Check for combined Bending and Axial tension

Members subjected to combined action of bending and axial tension stress should satisfy Equations 6.17 and 6.18 of EC-5 2004:

Note: In STAAD z-z axis is the strong axis.

$$(S_{t0d}/F_{t0d}) + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \le$$

$$(S_{t0d}/F_{t0d}) + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \le$$

Check for combined Bending and axial Compression

If members are subjected to bending and axial compression stress, Equations 6.19 and 6.20 of EC-5 2004 should be satisfied:

Note: In STAAD z-z axis is the strong axis.

$$(S_{c0d}/F_{c0d})^2 + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \le$$

$$(S_{c0d}/F_{c0d})^2 + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \le$$

Stability check

A. Column Stability check

The relative slenderness ratios should be calculated per Equations 6.21 and 6.22 of EC-5 2004.

Note: In STAAD z-z axis is the strong axis.

$$\lambda_{rel,z} = \lambda_z/\pi \cdot (S_{c0k}/E_{0,05})^{1/2}$$

$$\lambda_{rel,y} = \lambda_y/\pi \cdot (S_{c0k}/E_{0,05})^{1/2}$$

If both $\lambda_{rel,z}$ and $\lambda_{rel,y}$ are less than or equal to 0.3 the following conditions should be satisfied:

$$(S_{c0d}/F_{c0d})^2 + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \le$$

$$(S_{c0d}/F_{c0d})^2 + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \le$$

In other cases, the conditions in Equations 6.23 and 6.24 of EC-5 2004 should be satisfied.

Note: In STAAD z-z axis is the strong axis.

$$S_{c0d}/(Kcz \cdot F_{c0d}) + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \le$$

$$S_{c0d}/(Kcy \cdot F_{c0d}) + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \le$$

Where (Equations 6.25 through 6.28 of EC-5 2004):

$$Kcz = 1/\{K_z + [(K_z)^2 - (\lambda_{rel,z})^2]^{1/2}\}$$

$$Kcy = 1/\{K_y + [(K_{zy})^2 - (\lambda_{rel,y})^2]^{1/2}\}$$

$$Kz = 0.5 \cdot [1 + \beta_c \cdot (\lambda_{rel,z} - 0.3) + (\lambda_{rel,z})^2]$$

$$Ky = 0.5 \cdot [1 + \beta_c \cdot (\lambda_{rel,y} - 0.3) + (\lambda_{rel,y})^2]$$

The value of β_c incorporated in the software is the one for solid timber (i.e., 0.2).

B. Beam Stability check

If members are subjected to only a moment about the strong axis z, the stresses should satisfy Equation 6.33 of EC-5 2004:

$$S_{mzd}/(Kcrit \cdot F_{mzd}) \le$$

Where a combination of moment about the strong z-axis and compressive force exists, the stresses should satisfy Equation 6.35 of EC-5 2004 (ref. to Equations 6.32 and 6.34 of the same):

$$[S_{mzd}/(Kcrit \cdot F_{mzd})]^2 + S_{c0d}/(Kcz \cdot F_{c0d}) \leq$$

Where:

$Kcrit = 1.0$ when $\lambda_{rel,m} \leq 0.75$
$Kcrit = 1.56 - 0.75 \lambda_{rel,m}$ when $0.75 < \lambda_{rel,m} \leq 1.4$
$Kcrit = 1/(\lambda_{rel,m})^2$ when $1.4 < \lambda_{rel,m}$
$\lambda_{rel,m} = (f_{mk}/S_{m,crit})^{1/2}$

For hardwood, use Equation 6.30 of EC-5 2004:

$$S_{m,crit} = \pi \cdot (E_{0,05} \cdot I_y \cdot G_{0,05} \cdot I_{tor})^{1/2}/(l_{ef} \cdot W_z)$$

For softwood, use Equation 6.31 of EC-5 2004:

$$S_{m,crit} = 0.78 \cdot b^2 \cdot E_{0,05}/(h \cdot l_{ef})$$

Related Links

- *V. EC5 - Timber Column* (on page 6244)
- *V. EC5 - Timber Column with Bending* (on page 6247)

D5.E.3 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Depending on the model being designed, the user may have to change some or all of the parameter default values. Some parameters are unit dependent and when altered, the new setting must be compatible with the active "unit" specification.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 161: Timber Design EC 5: Part 1-1 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `TIMBER EC5` Design Code to follow. See section TR.52.1 Timber Design Parameter Specifications (on page 2683) .
ALPHA	0.0	Angle of inclination of load to the grain alignment. (Ref. Cl.6.1.1, Cl. 6.1.2, Cl.6.1.3, Cl.6.1.4) 0.0 = Load parallel to grain 90.0 = Load Perpendicular to grain

Parameter Name	Default Value	Description
DFF	None	"Deflection Length" / Max. Allowable Net Final Local Deflection. In this case, deflection check will be performed, if both the parameters SERV and DFF are present with specific values. For appropriate range of values, please refer Cl.7.2 (Table 7.2)
DJ1		Start node number for a physical member under consideration for Deflection Check.
DJ2		End node number for a physical member under consideration for Deflection Check.
KC90	1.0	Factor taking into account the load configuration, possibility of splitting and degree of compressive deformation. (Ref. Cl.6.1.5-(2)) • Range: 1.0 ≤ KC90 ≤ 4.0 • Other than the default value, user may specify any value within the range, depending on load-position, load-dispersion, contact length at support locations etc.
KLEF	1.0 (Member Length)	Effective Length Factor to check Lateral Torsional Buckling (Ref. Table 6.1). Factor multiplied by the span of the beam and depends on the support conditions and load configurations. The user will put the appropriate value from the Table 6.1. Required only for `MTYP` value of 1 (Beam).
KY	1.0 (Member Length)	Effective Length Factor for Local-y-axis. (Ref. Cl.6.3.2), for the computation of the relative slenderness ratios.

Parameter Name	Default Value	Description
KZ	1.0 (Member Length)	Effective Length Factor for Local-z-axis. (Ref. Cl.6.3.2), for the computation of the relative slenderness ratios.
LDC	1	Load Duration Class (Ref. Cl. 2.3.1.2), required to get the K-MOD value from Table – 3.1. 1.0 = Permanent action 2.0 = Long term action 3.0 = Medium term action 4.0 = Short term action 5.0 = Instantaneous action
MTYP	0	Member Type: Beam/Column. (Ref. Cl.6.3.2, Cl.6.3.3) 0.0 = Not defined; both clauses are checked (Default) 1.0 = Beam Member 2.0 = Column Member This information is required to find which stability check will be performed as per the Cl 6.3 according to the Member Type.
RATIO	1.0	Permissible ratio of actual to allowable value.
SCL	3	Service Class (Ref. Cl.2.3.1.3) 1.0 = Class 1, Moisture content ≤ 12% 2.0 = Class 2, Moisture content ≤ 20% 3.0 = Class 3, Moisture content > 20%

Parameter Name	Default Value	Description
TRACK	0	Degree/Level of Details of design output results. 1.0 = Print the design output at the minimal detail level 2.0 = Print the design output at the intermediate detail level 3.0 = Print the design output that the maximum detail level
TSC	6 (C24)	Timber Strength Class (Ref. Reference EN338 – 2003) • Softwood: 1 = C14, 2 = C16, 3 = C18, 4 = C20, 5 = C22, 6 = C24, 7 = C27, 8 = C30, 9 = C35, 10 = C40, 11 = C45, 12 = C50. • Hardwood: 13 = D30, 14 = D35, 15 = D40, 16 = D50, 17 = D60, 18 = D70. This TSC definition will calculate the corresponding characteristic strength values using the equations as given in BS-EN-338, Annex - A.

D6. French Codes

D6.A. French Codes - Steel Design per CM66-1977 (French)

STAAD.Pro is capable of performing steel design based on the French code CM66, 1977 edition *Centre Technique Industriel de la Construction Metallique* (*Industrial Technical Center of Metal Construction*) publication entitled *Design Rules for Structural Steelwork* .

D6.A.1 General Comments

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned according to the limit states of which they would become unfit for their intended use. Two major categories of limit-states are recognized: ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability; that in serviceability is deflection. Appropriate load and resistance factors are used so that uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria, as augmented by the designer in specification of allowable member depths, desired section type, or other related parameters. The code checking portion of the program verifies that code requirements for each selected section are met and also identifies the governing criteria.

The next few sections describe the salient features of STAAD.Pro implementation of "Design Rules for Structural Steelwork." A detailed description of the design process, along with its underlying concepts and assumptions, is available in the specification document.

D6.A.2 Basis of Methodology

The *Design Rules for Structural Steelwork* (Revision 80) permits the usage of elastic analysis. Thus, in STAAD.Pro, linear elastic analysis method is used to obtain the forces and moments in the members. However, strength and stability considerations are based on the principles of plastic behavior. Axial compression buckling and lateral torsional buckling are taken into consideration for calculation of axial compression resistance and flexural resistance of members. Slenderness calculations are made and overall geometric stability is checked for all members.

D6.A.3 Member Capacities

The member strengths are calculated in STAAD.Pro according to the procedures outlined in section 4 of this specification. Note that the program automatically considers co-existence of axial force, shear and bending in calculating section capacities.

For axial tension capacity, procedures of section 4.2 are followed. For axial compression capacity, formulas of section 5.3 are used.

Moment capacities about both axes are calculated using the procedures of sections 4.5 and 4.6. Lateral torsional buckling is considered in calculating ultimate twisting moment per section 5.22 of the specification. The parameter `UNL` (see D6.A.5 Design Parameters (on page 1593)) must be used to specify the unsupported length of the compression flange for a laterally unsupported member. Note that this length is also referred to as twisting length.

D6.A.4 Combined Axial Force and Bending

The procedures of sections 4.55 and 5.32 are implemented for interaction of axial forces and bending. Appropriate interaction equations are used and the governing criterion is determined.

D6.A.5 Design Parameters

The design parameters outlined in the following table may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program, thus allowing the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected as frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 162: French Steel Design Parameters

Parameter Name	Default Value	Description
CODE	FRENCH	Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	0.0	0.0 = design only for end moments and those at locations specified by `SECTION` command. 1.0 = calculate moments at tenth points long the beam, and use maximum Mz for design.
C1	1.0	Parameter used in clause 5.21 in the calculation of M(D), the critical twisting moment and as shown in CM 66 Addendum 80, table 5, usual range from 0.71 to 4.10
C2	1.0	Parameter used in clause 5.21 in the calculation of M(D), the critical twisting moment and as shown in CM 66 Addendum 80, table 5, usual range from 0.0 to 1.56
DFF	None (Mandatory for deflection check)	"Deflection Length" divided by the Maximum allowable local deflection
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length" (See Note 1)
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See Note 1)
DMAX	100.0 cm.	Maximum allowable depth (used in member selection).
DMIN	0.0 cm.	Minimum allowable depth (used in member selection).
FYLD	250.0 MPa	Yield strength of steel.
KY	1.0	K value for axial compression buckling about local Y-axis. Usually, this is the minor axis.

Parameter Name	Default Value	Description
KZ	1.0	K value for axial compression buckling about local Z-axis. Usually, this is the major axis.
LY	Member Length	Length to calculate slenderness ratio about Y-axis for axial compression.
LZ	Member Length	Length to calculate slenderness ratio about Z-axis for axial compression.
NSF	1.0	Net section factor for tension members.
RATIO	1.0	Permissible ratio of actual load effect and design strength.
SAME	0.0	Controls the sections to try during a SELECT process. 0.0 = Try every section of the same type as original (see note a (on page 1596) below) 1.0 = Try only those sections with a similar name as original, e.g., if the original is an HEA 100, then only HEA sections will be selected, even if there are HEM's in the same table.
TRACK	0.0	0.0 = Suppress printing of all design strengths. 1.0 = Print all design strengths.
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.

Parameter Name	Default Value	Description
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.

a. For angles, if the original section is an equal angle, then the selected section will be an equal angle and vice versa for unequal angles.

D6.A.6 Code Checking and Member Selection

Both code checking and member selection options are available in the STAAD.Pro implementation of CM 66 (Revn. 80).

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D6.A.7 Tabulated Results of Steel Design

Results of code checking and member selection are presented in the output file in a tabular format.

Note: `COND CRITIQUE` refers to the section of the CM 66 (Revn. 80) specification which governed the design.

If the `TRACK` parameter is set to 1.0, calculated member capacities will be printed. The following is a detailed description of printed items:

PC	Member Compression Capacity
TR	Member Tension Capacity
MUZ	Member Moment Capacity (about z-axis)
MUY	Member Moment Capacity (about y-axis)
VPZ	Member Shear Capacity (z-axis)
VPY	Member Shear Capacity (y-axis)

STAAD.Pro contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.

These operations may be repeated by the user any number of times depending upon the design requirements.

STAAD.Pro supports steel design of wide flange, S, M, HP shapes, angle, double angle, channel, double channel, beams with cover plate, composite beams and code checking of prismatic properties.

Sample Input data for Steel Design:

```
UNIT  METER
PARAMETER
CODE  FRENCH
NSF  0.85 ALL
UNL  10.0 MEMBER 7
KY  1.2 MEMBER 3 4
RATIO  0.9 ALL
TRACK  1.0 ALL
CHECK  CODE ALL
```

D6.A.8 Built-in French Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members.

An example of the member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained by using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

D6.A.8.1 IPE Shapes

These shapes are designated in the following way.

```
10 15  TA ST IPE140
20 TO  30 TA ST IPEA120
33 36  TO 46 BY 2 TA ST IPER180
```

D6.A.8.2 HE shapes

HE shapes are specified as follows.

```
3 5  TA ST HEA120A
7 10  TA ST HEM140
13 14  TA ST HEB100
```

D6.A.8.3 IPN Shapes

The designation for the IPN shapes is similar to that for the IPE shapes.

```
25 TO  35 TA ST IPN200
23 56  TA ST IPN380
```

D6.A.8.4 T Shapes

Tee sections are not input by their actual designations, but instead by referring to the I beam shapes from which they are cut. For example,

```
1 5  TA T IPE140
2 8  TA T HEM120
```

D6.A.8.5 U Channels

Shown below is the syntax for assigning 4 different names of channel sections.

```
1 TO  5 TA ST UAP100
6 TO  10 TA ST UPN220
11 TO  15 TA ST UPN240A
16 TO  20 TA ST UAP250A
```

D6.A.8.6 Double U Channels

Back to back double channels, with or without a spacing between them, are available. The letter D in front of the section name will specify a double channel.

```
11 TA  D UAP150
17 TA  D UAP250A SP 0.5
```

In the above set of commands, member 11 is a back-to-back double channel UAP150 with no spacing in between. Member 17 is a double channel UAP250A with a spacing of 0.5 length units between the channels.

D6.A.8.7 Angles

Two types of specification may be used to describe an angle. The standard angle section is specified as follows:

```
16 20  TA ST L30X30X2.7
```

The above section signifies an angle with legs of length 30mm and a leg thickness of 2.7mm. This specification may be used when the local Z axis corresponds to the z-z axis specified in Chapter 2. If the local Y axis corresponds to the z-z axis, type specification "RA" (reverse angle) should be used instead of ST.

```
17 21  TA RA L25X25X4
22 24  TA RA L100X100X6.5
```

Note that if the leg thickness is a round number such as 4.0, only the number 4 appears in the section name, the decimal part is not part of the section name.

D6.A.8.8 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by means of input of the words SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose.

```
33 35  TA SD L30X20X4 SP 0.6
37 39  TA LD L80X40X6
43 TO  47 TA LD L80X80X6.5 SP 0.75
```

D6.A.8.9 Tubes (Rectangular or Square Hollow Sections)

Section names of tubes, just like angles, consist of the depth, width and wall thickness as shown below.

```
64 78  TA ST TUB50252.7
66 73  TA ST TUB2001008.0
```

Members 64 and 78 are tubes with a depth of 50mm, width of 25mm and a wall thickness of 2.7mm. Members 66 and 73 are tubes with a depth of 200mm, width of 100mm and a wall thickness of 8.0mm. Unlike angles, the ". 0" in the thickness is part of the section name.

Tubes can also be input by their dimensions instead of by their table designations. For example,

```
6 TA  ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a depth of 8 length units, width of 6 length units, and a wall thickness of 0.5 length units. Only code checking, no member selection, will be performed for TUBE sections specified in this way.

D6.A.8.10 Pipes (Circular Hollow Sections)

To designate circular hollow sections, use PIP followed by numerical value of the diameter and thickness of the section in mm omitting the decimal portion of the value provided for the diameter. The following example illustrates the designation.

```
8 TO  28 TA ST PIP422.6
3 64  78 TA ST PIP21912.5
```

Members 8 to 28 are pipes 42.4mm in dia, having a wall thickness of 2.6mm. Members 3, 64 and 78 are pipes 219.1mm in dia, having a wall thickness of 12.5mm.

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

```
1 TO  9 TA ST PIPE OD 25.0 ID 20.0
```

specifies a pipe with outside dia. of 25 length units and inside dia. of 20 length units. Only code checking, no member selection will be performed if this type of specification is used.

D6.A.8.11 Example

Sample file containing French shapes:

```
STAAD  SPACE
UNIT  METER KN
JOINT  COORD
1 0  0 0 15 140 0 0
MEMB  INCI
1 1  2 14
UNIT  CM
MEMBER  PROPERTIES FRENCH
* IPE  SHAPES
1 TA  ST IPEA120
* IPN  SHAPES
2 TA  ST IPN380
*HE  SHAPES
3 TA  ST HEA200
* T  SHAPES
4 TA  T HEM120
* U  CHANNELS
5 TA  ST UAP100
* DOUBLE  U CHANNELS
6 TA  D UAP150 SP 0.5
* ANGLES
7 TA  ST L30X30X2.7
* REVERSE  ANGLES
8 TA  RA L25X25X4
* DOUBLE  ANGLES - SHORT LEGS BACK
* TO  BACK
9 TA  SD L30X20X4 SP 0.25
* DOUBLE  ANGLES - LONG LEGS BACK
* TO  BACK
10 TA  LD L80X40X6 SP 0.75
* TUBES  (RECTANGULAR OR SQUARE
* HOLLOW  SECTIONS)
```

```
11 TA  ST TUB50252.7
* TUBES  (RECTANGULAR OR SQUARE
* HOLLOW  SECTIONS)
12 TA  ST TUBE DT 8.0 WT 6.0 TH 0.5
* PIPES  (CIRCULAR HOLLOW SECTIONS)
13 TA  ST PIP422.6
* PIPES  (CIRCULAR HOLLOW SECTIONS)
14 TA  ST PIPE OD 25.0 ID 20.0
PRINT  MEMB PROP
FINISH
```

D7. German Codes

D7.A. German Codes - Steel Design per DIN 18800 Code

STAAD.Pro is capable of performing concrete design based on the German code DIN 18800, Parts 1 & 2: *Stahlbauten - Teil 1: Bemessung und Konstruktion* (*Steel structures - Part 1: Design and construction*) and *Stahlbauten - Teil 2: Stabilitätsfälle - Knicken von Stäben und Stabwerken* (*Steel structures - Part 2: Analysis of safety against buckling of linear members and frames*)

D7.A.1 General

This section presents some general statements regarding the implementation of the DIN code. The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceeding the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. It is recommended that you use the following steps in performing the steel design:

1. Specify the geometry and loads and perform the analysis.
2. Specify the design parameter values if different from the default values.
3. Specify whether to perform code checking or member selection.

D7.A.2 Analysis Methodology

The elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions you provide. You are allowed complete flexibility in providing loading specifications and in using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D7.A.3 Member Property Specifications

For specification of member properties of standard German steel sections, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D7.A.4 Built-in German Steel Section Library

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, these properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members during the analysis. An example of member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

Refer to G.6.2 Built-In Steel Section Libraries (on page 2116) for additional information.

D7.A.4.1 IPE Shapes

These shapes are designated in the following way:

```
20 TO 30 TA ST IPEA120
33 36 TO 46 BY 2 TA ST IPER140
```

D7.A.4.2 HE Shapes

The designation for HE shapes is similar to that for IPE shapes.

```
25 TO 35 TA ST HEB300
23 56 TA ST HEA160
```

D7.A.4.3 I Shapes

I shapes are identified by the depth of the section. The following example illustrates the designation.

```
14 15 TA ST I200
```

(indicates an I-section with 200mm depth)

D7.A.4.4 T Shapes

Tee sections are not input by their actual designations, but instead by referring to the I beam shapes from which they are cut. For example,

```
1 5 TA T HEA220
2 8 TA T IPE120
```

D7.A.4.5 U Channels

The example below provides the command for identifying two channel sections. The former (U70X40) has a depth of 70mm and a flange width of 40mm. The latter (U260) has a depth of 260mm.

```
11 TA D U70X40
27 TA D U260
```

D7.A.4.6 Double Channels

Back-to-back double channels, with or without spacing between them, are available. The letter “D” in front of the section name will specify a double channel, e.g., D U180. The spacing between the double channels is provided following the expression “SP”.

```
11 TA D U180
27 TA D U280 SP 0.5
```

(Indicates 2 channels back-to-back spaced at 0.5 length units)

D7.A.4.7 Angles

Two types of specifications may be used to describe an angle. The standard angle section is specified as follows:

```
16 20 TA ST L20X20X2.5
```

The above section signifies an angle with legs of length 20mm and a leg thickness of 2.5mm. The above specification may be used when the local z-axis corresponds to the Z-Z axis specified in Chapter 2. If the local y-axis corresponds to the Z-Z axis, type specification "RA" (reverse angle) may be used.

```
17 21 TA RA L40X20X5
```

D7.A.4.8 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by using the word SD or LD, respectively, in front of the angle size. In case of an equal angle, either SD or LD will serve the purpose. Spacing between the angles is provided by using the word SP and the spacing value following the section name.

```
14 TO 20 TA SD L40X20X4 SP 0.5
21 TO 27 TA LD L40X20X4 SP 0.5
```

D7.A.4.9 Pipes (Circular Hollow Sections)

To designate circular hollow sections, use `PIP` followed by numerical value of the diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

```
8 TO 28 TA ST PIP602.9
* (60.3mm dia, 2.9mm wall thickness)
3 64 67 TA ST PIP40612.5
*(406.4mm dia, 12.5mm wall thickness)
```

Circular hollow sections may also be provided by specifying the outside and inside diameters of the section. For example,

```
1 TO 9 TA ST PIPE OD 25.0 ID 20.0
```

specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units. Only code checking and no member selection will be performed if this type of specification is used.

D7.A.4.10 Tubes (Rectangular or Square Hollow Sections)

Tube names are input by their dimensions. For example,

```
15 TO 25 TA ST TUB100603.6
```

is the specification for a tube having sides of 100mm x 60mm and the wall thickness of 3.6mm.

Tubes, like pipes can also be input by their dimensions (Height, Width and Thickness) instead of by their table designations.

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5 in current length units. Only code checking and no member selection will be performed for TUBE sections specified this way.

D7.A.4.11 Example

Sample input file containing German shapes:

```
STAAD SPACE
UNIT METER KN
JOINT COORDINATES
1 0 0 0 15 140 0 0
MEMBER INCIDENCES
1 1 2 14
UNIT CM
MEMBER PROPERTIES GERMAN
* IPE SHAPES
1 TA ST IPEA120
* HE SHAPES
2 TA ST HEB300
* I SHAPES
3 TA ST I200
* T SHAPES
4 TA T HEA220
* U CHANNELS
5 TA ST U70X40
* DOUBLE U CHANNELS
6 TA D U260
* ANGLES
7 TA ST L20X20X2.5
* REVERSE ANGLES
8 TA RA L40X20X5
* DOUBLE ANGLES - LONG LEGS BACK TO BACK
9 TA LD L40X20X4 SP 0.5
* DOUBLE ANGLES - SHORT LEGS BACK TO BACK
10 TA SD L40X20X4 SP 0.5
* PIPES
11 TA ST PIP602.9
* PIPES
12 TA ST PIPE OD 25.0 ID 20.0
* TUBES
13 TA ST TUB100603.6
* TUBES
14 TA ST TUBE DT 8.0 WT 6.0 WT 0.5
*
PRINT MEMBER PROPERTIES
FINISH
```

D7.A.5 Member Capacities

The allowable stresses used in the implementation are based on DIN 18800 (Part 1) - Section 7. The procedures of DIN 18800 Part 2 are used for stability analysis. The basic measure of member capacities are the allowable stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD for calculating such capacities.

D7.A.5.1 Checks for Axial Tension

In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the member area. STAAD calculates the tension capacity of a given member based on a user supplied net section factor (NSF -a default value of 1.0 is present but may be altered by changing the input value, see D7.A.7 Design Parameters (on page 1604)) and proceeds with member selection or code checking.

D7.A.5.2 Checks for Axial Compression

The compression capacity for members in compression is determined according to the procedure of DIN 18800-Part 2. Compressive resistance is a function of the slenderness of the cross-section (Kl/r ratio) and the user may control the slenderness value by modifying parameters such as KY, LY, KZ and LZ.

D7.A.5.3 Checks for Bending and Shear

The bending compressive and tensile capacities are dependent on such factors as length of outstanding legs, thickness of flanges, unsupported length of the compression flange (UNL, defaults to member length) etc. Shear capacities are a function of web depth, web thickness etc. Users may use a value of 1.0 or 2.0 for the TRACK parameter to obtain a listing of the bending and shear capacities.

D7.A.6 Combined Loading

For members experiencing combined loading (axial force, bending, and shear), applicable interaction formulas are checked at different locations of the member for all modeled loading situations. Members subjected to axial force and bending are checked using the criteria of DIN 18800 (Part 1) - Section 6.1.6. In addition, for members with axial loads and bending, the criteria of DIN 18800(Part 2) - Sections 3.4 and 3.5 are used.

D7.A.7 Design Parameters

You are allowed complete control over the design process through the use of parameters described in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 163: German Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as DIN18800. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	0.0	Number of sections to be checked per member: 0. Design only for end sections. 1. Check at location of maximum MZ along member. 2. Check ends plus location of beam 1.0 check. 3. Check at every 1/13th of the member length and report the maximum.
CB	0	Beam coefficient n, defined in Table 9: If Cb = 0, program will use n = 2.5 for rolled sections and 2.0 for welded sections.
CMM	1.0	Moment factor, Zeta, defined in Table 10: 1. fixed ended member with constant moment, Zeta = 1.0 2. pin ended member with UDL, Zeta = 1.12 3. pin ended member with central point load, Zeta = 1.35 4. fixed ended member, Zeta calculated from end moments.
DMAX	1.0 m	Maximum allowable depth during member selection
DMIN	0.0 m	Minimum required depth during member selection
KY	1.0	K value in local y-axis. Usually, this is the minor axis.
KZ	1.0	K value in local z-axis. Usually, this is the major axis.
LY	Member Length	Length in local y-axis to calculate slenderness ratio.
LZ	Member Length	Length in local z-axis to calculate slenderness ratio.
PY	240 N/sq.mm	Strength of steel.

Parameter Name	Default Value	Description
NSF	1.0	Net section factor for tension members.
RATIO	1.0	Permissible ratio of actual to allowable stresses
SAME	0.0	Control of sections to try during a SELECT process: 0. Try every section of the same type as the original. 1. Try only those with a similar name.
SBLT	0	Specify section as either rolled or built-up: 0. Rolled 1. Built-up
SGR	0.0	Grade of steel: 0. St 37-2 1. St 52-3 2. St E 355
TRACK	0.0	Level of detail in output file: 0. Output summary of results 1. Output summary of results plus member capacities 2. Output detailed results
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.

D7.A.9 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate to carry the forces transmitted to it by the loads on the structure. The adequacy is checked per the DIN requirements.

Code checking is done using forces and moments at specified sections of the members. If the `BEAM` parameter for a member is set to 1, moments are calculated at every twelfth point along the beam, and the maximum moment about the major axis is used. When no sections are specified and the `BEAM` parameter is set to zero (default), design will be based on member start and end forces. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from start joint) and magnitudes of the governing forces and moments are also printed.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D7.A.9 Member Selection

The member selection process basically involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on TUBES, PIPES, or members listed as PRISMATIC.

Sample Input data for Steel Design

```
UNIT METER
PARAMETER
CODE GERMAN
NSF 0.85 ALL
UNL 10.0 MEMBER 7
KY 1.2 MEMBER 3 4
RATIO 0.9 ALL
TRACK 1.0 ALL
CHECK CODE ALL
```

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D8. Indian Codes

D8.A. Indian Codes - Steel Design per IS 800 - 2007

STAAD.Pro is capable of performing steel design based on the Indian code IS 800-2007 *General construction in steel - Code of practice*.

D8.A.1 General Comments

For steel design, STAAD.Pro compares the actual design forces with the capacities as defined by the Indian Standard Code. The IS 800: 2007 Code is used as the basis of this design.

A brief description of some of the major capacities is described herein.

The following commands should be used to initiate design per Limit State Method of this code:

```
PARAMETER n
```

```
CODE IS800 LSD
```

The following commands should be used to initiate design per Working Stress Method of this code:

```
PARAMETER n
```

```
CODE IS800 WSD
```

Where:

> *n* = optional integer (i.e., - 1, 2) which signifies the numerical order of parameter command block (if multiple blocks are specified).

D8.A.2 Design Process

The design process follows the following design checks.

When a design is performed, the output file reports the maximum utilization ratio from all of these checks.

Table 2 of IS800:2007 contains the various limit of “limiting width to thickness ratio” for various section types. This provision is not applicable for solid rod design.

Design for Work Stress Method

The scope of the implementation for WSD method is limited to the following range of sections – from CL 11.1 to 11.5.3 of the IS800:2007 specification.

Related Links

- *V. IS 800 2007 LSD - Angle - Flexural Torsional Buckling* (on page 4206)
- *V. IS 800 2007 LSD - Angle - Tension with Block Shear* (on page 4213)
- *V. IS 800 2007 LSD - I Section - High Shear* (on page 4228)
- *V. IS 800 2007 LSD - I Section - with LTB* (on page 4238)
- *V. IS 800 2007 LSD - I Section - without LTB* (on page 4248)
- *V. IS 800 2007 LSD - I Section with Cover Plate* (on page 4257)
- *V. IS 800 2007 LSD - Pipe - Tension and Bending* (on page 4269)
- *V. IS 800 2007 - LSD Rod Compression and Bending* (on page 4276)
- *V. IS 800 2007 - WSD Rod Compression and Bending* (on page 4284)

D8.A.2.1 Slenderness

As per Section 3.8 Table 3, the slenderness ratio (KL/r) of compression members shall not exceed 180, and the slenderness ratio (L/r) of tension members shall not exceed 400.

You can edit the default values through `MAIN` and `TMAIN` parameters, as defined in D8.A.4 Design Parameters (on page 1617).

D8.A.2.2 Section Classification

The IS 800: 2007 specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion.

Steel sections are classified as Plastic, Compact, Semi-Compact, or Slender element sections depending upon their local buckling characteristics.

This classification is a function of the geometric properties of the section as well as nature of the load applied to the member. The design procedures are different depending on the section class.

STAAD.Pro is capable of determining the section classification for the standard shapes and design the section for the critical load case accordingly. The Section Classification is done as per section 3.7 of IS 800:2007 and Table B2, for Outstanding and Internal Elements of a section.

For the criteria for being included in those classes, refer to section 3.7.2-(a) – (d) of the code.

Slender Sections

STAAD.Pro is capable of designing I-Sections with slender webs for IS 800:2007.

The IS:800-2007 code does not provide any clear guidelines about what method should be adopted for the design of slender section. The “Flange Only” methodology is used where it is assumed that flexure is taken by the flanges alone and the web will resist shear with adequate shear buckling resistance. This method requires that the flanges be non-slender elements (i.e., on the web is a slender element) to qualify for a valid section for design. If any of the flange elements become slender, the design will not be performed and a warning message is displayed in the output.

D8.A.2.3 Tension

Limit State Method

The criteria governing the capacity of Tension members are based on:

- Design Strength due to Yielding in Gross Section
- Design Strength due to Rupture of Critical Section
- Design Strength due to Block Shear (does not apply to solid rods)

STAAD.Pro calculates the tension capacity of a given member based on these three limit states.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section 6.2 of the code.

The Design strength, involving rupture at the section with the net effective area, is evaluated as per section 6.3 of the code. Here, the number of bolts in the connection may be specified through the use of the design parameter ALPHA.

The Design strength, involving block shear at an end connection, is evaluated as per section 6.4 of the code.

Note: Block shear is *not* checked by default. This is only checked for members for which parameters AVG, AVN, ATG, and ATN are specified.

The Net Section Area may be specified through the use of the parameter NSF.

Working Stress Method

The criteria governing the allowable stress from tension in members are based on Section 11.2.1 of the code:

- Yielding of Gross Section - to prevent excessive elongation of the member due to material yielding.

- Rupture of Net Section - to prevent rupture of the net effective section area. The number of bolts in the connection may be specified through the use of the design parameter ALPHA. The code parameter, γ_{M1}, is taken as 1.25 per Table 5, Clause 5.4.1 of the code.
- Block Shear — to prevent block shearing at the end connection. The code parameters,, γ_{M0} and γ_{M1}, are taken as 1.10 and 1.25, respectively, per Table 5, Clause 5.4.1 of the code.

> **Note:** Block shear is *not* checked by default. This is only checked for members for which parameters AVG, AVN, ATG, and ATN are specified.

These criteria are dependent on the steel material yield stress parameter, FYLD, and ultimate tensile strength parameter, FU.

D8.A.2.4 Compression

The design capacity of the section against compressive force, the guiding phenomenon is the flexural buckling.

Limit State Method

The buckling strength of the member is affected by residual stress, initial bow and accidental eccentricities of load.

To account for all these factors, the strength of the members subjected to axial compression is defined by buckling class a, b, c or d as per clause 7.1.2.2 and Table 7 of IS 800:2007.

Imperfection factor, obtained from buckling class, and Euler's Buckling Stress ultimately govern compressive force capacity of the section as per clause 7.1.2 of IS 800:2007.

The buckling class of a solid rod section is determined per Table 10 of the specification.

Working Stress Method

The actual compressive stress is given by:

$$f_c = FX/A_e$$

where:

A_e = The effective section area as per Clause 7.3.2 of the code. This is equal to the gross cross sectional area, AX, for any non-slender (plastic, compact, or semi-compact) D8.A.2.2 Section Classification (on page 1609). In the case of slender sections, this is limited to value of Ae as described below.

The permissive compressive stress is calculated by first determining the Buckling Class of the section per Table 10 of the code and α_{YY} & α_{ZZ} based on Table 7.

$$F_{ac} = 0.6 \cdot F_{cd}$$

where:

F_{cd} = the minimum of the values of Fcd calculated for the local Y and Z axis.

$$F_{cd} = (/\gamma_{mo})/\ [\varphi + (\varphi^2 + \lambda^2]$$

λ = the non-dimensional slenderness factor is evaluated for each local Y and Z axis.

$$\lambda = (/F_{cc})^{1/2}$$

$$\varphi = 0.5[\ 1 + a(\lambda - 0.2) + \lambda^2]$$

F_{cc} = the Euler Buckling Stress.

$$F_{cc} = \pi^2 \cdot E/(Kl/r)^2$$

K = the effective length factor for bending about either the local Y or Z axis, as provided in the `KY` and `KZ` parameters, respectively.
r = radius of gyration about the local Y or Z axis for the section.
`FYLD` = The yield strength of steel specified in the `FYLD` parameter.

Slender Sections

For member with slender section under axial compression, design compressive strength should be calculated on area ignoring depth thickness ratio of web in excess of the class 3 (semi-compact) limit.

Refer to clause 7.3.2 and Table 2 of IS 800:2007, (corresponding to "Internal Element of Compression Flange")

$$A_e = A_g - (d/t_w - 42\varepsilon) \cdot t_w^{\ 2}$$

where:

A_e = Effective area of section.
A_g = Gross area of section.
d = Depth of web.
t_w = thickness of web.

Flexural-Torsional Buckling for Single Angles

In the case of single angles (ST and RA), a axial compression load does not pass through the member centroid. In practical applications, the load must pass through one of the legs. The deviation between the load and the centroid can be considerable, and thus the effect of flexural-torsional buckling must be considered.

The parameters of `ANG`, `FXTY`, and `NBL` are used to control these calculations.

The calculations for the flexural-torsional buckling strength of a single angle member in compression are performed per section 7.5.1.2 of the code (either limit state method or working stress method are applicable).

D8.A.2.5 Shear

The design capacities of the section against shear force in major- and minor-axis directions are evaluated as per section 8.4 of the code, taking care of the following phenomena:

- Nominal Plastic Shear Resistance
- Resistance to Shear Buckling

Shear area of the sections are calculated as per sec. 8.4.1.1. The full cross-sectional area is considered as the shear area for a solid rod section.

Nominal plastic shear resistance is calculated as per sec. 8.4.1.

Among shear buckling design methods, Simple post-critical method is adopted as per sec. 8.4.2.2(a).

Working Stress Design

The actual shear stress is determined about the major and minor axes, respectively:

$$\tau_{bY} = F_Y / A_Y$$

$$\tau_{bZ} = F_Z / A_Z$$

The permissible shear stress is determined as:

a. When subjected to pure shear:

$\tau_{ab} = 0.40 \cdot F_y$

b. When subjected to shear buckling:

$\tau_{ab} = 0.70 \cdot V_n \cdot A_v$

where

V_n	=	Nominal Shear Strength as per Clause 8.4.2.2.(a) $= V_{cr} = \tau_b \cdot A_v$
A_v	=	AY or AZ, whichever is appropriate, with reference to Clause 8.4.1.1.

Shear buckling must be checked when $(d/t_w) > 67 \cdot \epsilon_w$ for webs without stiffener or $(d/t_w) > 67 \cdot \epsilon_w \cdot \sqrt{(K_v/5.35)}$ for webs with stiffeners.

where

d	=	Clear Depth of Web between Flanges.
t_w	=	Thickness of Web.
ϵ_w	=	$\sqrt{(250 / F_y)}$
F_y	=	Yield Strength of Web, specified using the FYLD parameter.
K_v	=	Shear Buckling Coefficient: = 5.35, when transverse stiffeners are provided only at supports. $= 4.0 + 5.35 / (c/d)^2$ for $(c/d) < 1.0$ $= 5.35 + 4.0 / (c/d)^2$ for $(c/d) \geq 1.0$
c	=	Spacing of Transverse Stiffeners
μ	=	Poisson's Ratio
τ_b	=	Shear Stress corresponding to Web-buckling: $= F_y / \sqrt{3}$, when, $\lambda_w \leq 0.8$ $= (1 - 0.8 \cdot (\lambda_w - 0.8)) \cdot (F_y / \sqrt{3})$ when, $0.8 < \lambda_w < 1.2$ $= F_y / (\sqrt{3} \cdot \lambda_w^{\ 2})$ when, $\lambda_w \geq 1.2$
$\tau_{cr,e}$	=	The Elastic Critical Shear Stress of the Web

$\tau_{cr,e} = (K_v \cdot \pi^2 \cdot E) / (12 \cdot (1 - \mu^2) \cdot (d/t_w)^2)$

where

λ_w	=	Non-dimensional Web Slenderness Ratio for Shear Buckling Stress.
λ_w	=	$[F_y / (\sqrt{3} \cdot \tau_{cr,e})]^{1/2}$

Slender Sections

Slender sections should be verified against shear buckling resistance if $d/t_w > 67 \cdot \varepsilon$ for web without stiffeners or if it exceeds $67 \cdot \varepsilon \cdot \sqrt{(K_v/5.35)}$ for a web with stiffeners.

Design methods for resistance to shear buckling are described in clause 8.4.2.2 of IS:800-2007 code.

$V_n = V_{cr}$

where

V_{cr}	=	shear force corresponding to web buckling $= A_v \cdot \tau_b$
τ_b	=	shear stress corresponding to web buckling, determined as follows:

i. When $\lambda_w \leq 0.8$

$\tau_b = f_{yw}/\sqrt{3}$

ii. When $0.8 < \lambda_w < 1.2$

$\tau_b = [1 - 0.8(\lambda_w - 0.8)](f_{yw}/\sqrt{3})$

iii. When $\lambda_w \geq 1.2$

$\tau_b = f_{yw}/((\sqrt{3}\, \lambda_w^{\ 2}))$

λ_w = non-dimensional web slenderness ratio or shear buckling stress, given by:

$= [f_{yw}/(\sqrt{3}\, \tau_{cr,e})]^{1/2}$

$\tau_{cr,e}$ = elastic critical shear stress of the web

$= (k_v \cdot \pi^2 \cdot E)/[12 \cdot (1 - \mu^2)(d/t_w)^2]$

μ = Poisson's ratio

K_v = 5.35 when transverse stiffeners are provided only at supports

$= 4.0 + 5.35/(c/d)^2$ for $c/d < 1.0$

$= 5.35 + 4.0/(c/d)^2$ for $c/d \geq 1.0$

c = spacing of transverse stiffeners

d = depth of the web

D8.A.2.6 Bending

The design bending moment capacity of a section is primarily dependent on whether the member is laterally supported or unsupported.

You can control the lateral support condition of the member by the use of LAT parameter. The type of member (i.e., cantilever, simply supported, or general) is specified using the CAN parameter.

If the member is laterally supported, then the design strength is calculated as per the provisions of the section 8.2.1 of IS 800:2007, based on the following factors:

- Whether section with webs susceptible to shear buckling before yielding
- Ratio of shear force to design shear strength
- Section classification

If the member is laterally unsupported, then the design strength is calculated as per the provisions of the section 8.2.2 of IS 800:2007, based on the following factors:

- Lateral Torsional Buckling
- Section Classification

Laterally unsupported sections of a solid rod are considered as laterally supported as mentioned in Cl. 8.2.2(b). The plastic moment of inertia, Z_p, is calculated as $D^3/6$.

Working Stress Design

Actual bending stress values are given by, about major (Z) and minor (Y) axes, respectively:

$f_{bcz} = M_z/Z_{ecz}$

$f_{btz} = M_z/Z_{etz}$

$f_{bcy} = M_y/Z_{ecy}$

$f_{bty} = M_y/Z_{ety}$

The permissible bending stress is given as follows:

a. For laterally supported beams:

$F_{abc} = F_{abt} = 0.66 \cdot F_y$ for Plastic or Compact sections
$F_{abc} = F_{abt} = 0.60 \cdot F_y$ for Semi-compact sections

where

F_y	=	Yield strength of steel, indicated by the FYLD parameter.

b. For laterally unsupported beams:

i. About the major axis:

$f_{abcz} = 0.60 \cdot M_d / Z_{ecz}$

$f_{abtz} = 0.60 \cdot M_d / Z_{etz}$

where

M_d	=	Design Bending Strength as per Clause 8.2.2 $= \beta_b \cdot Z_{pz} \cdot f_{bd}$
f_{bd}	=	$\chi_{LT} \cdot F_y / \gamma_{mo}$
Z_{ez}	=	Elastic Section Modulus of the Section
Z_{pz}	=	Plastic Section Modulus of the Section
α_{LT}	=	0.21 for Rolled Steel Section and 0.49 for Welded Steel Section
β_b	=	1.0 for Plastic and Compact Section or Z_{ez}/Z_{pz} for Semi-Compact Section
λ_{LT}	=	Non-dimensional slenderness ratio
λ_{LT}	=	$(\beta_b \cdot Z_{pz} \cdot F_y / M_{cr})^{1/2} \leq (1.2 \cdot Z_{ez} \cdot F_y / M_{cr})^{1/2}$
ϕ_{LT}	=	$0.5 \cdot (1 + \alpha_{LT} \cdot (\lambda_{LT} - 0.2) + \lambda_{LT}{}^2)$
χ_{LT}	=	The Bending Stress Reduction Factor to account for Lateral Torsional Buckling
χ_{LTZ}	=	$\dfrac{1}{\phi_{LTZ} + \sqrt{\phi_{LTZ}^2 - \lambda_{LTZ}^2}}$
Z_{ecz}	=	Elastic Section Modulus of the section about Major Axis for the compression side
Z_{etz}	=	Elastic Section Modulus of the section about Major Axis for the tension side
M_{cr}	=	$M_{cr} = \sqrt{\dfrac{\pi^2 E I_y}{L_{LT}^2}\left(G I_t + \dfrac{\pi^2 E I_w}{L_{LT}^2}\right)}$
I_y	=	Moment of inertia about the minor axis
L_{LT}	=	Effective length for lateral torsional buckling as determined using either the KX or LX parameters
I_t	=	Torsional constant of the section
I_t	=	Warping constant of the section
G	=	Shear modulus of the material

ii. About the minor axis, the permissible bending stress is calculated as for a laterally supported section.

Slender Sections

For member with slender section subjected to bending, moment is taken by flanges alone. Design bending strength should be calculated with effective elastic modulus disregarding the contribution of web of the section.

$Z_{ez} = 2 \cdot [B_f \cdot t_f{}^3/12 + (B_f \cdot t_f) \cdot (D/2 - t_f/2)^2)] / (0.5 \cdot D)$

$Z_{ey} = 2 \cdot (B_f \cdot t_f{}^3/12) / (0.5 \cdot B_f)$

Where:

where

Z_{ez}	=	Elastic Section modulus about major principal axis
Z_{ey}	=	Elastic Section modulus about minor principal axis
B_f	=	Width of flange
T_f	=	thickness of flange
D	=	Overall depth of section

The Moment Capacity will be $M_d = Z_e \cdot f_y / \gamma_{m0}$ for “Laterally Supported” condition.

The Moment Capacity will be $M_d = Z_e \cdot f_{bd} / \gamma_{m0}$ for “Laterally Un-Supported” condition.

Where, f_{bd} is defined in clause 8.2.2 of IS:800-2007 (described in previous Working Stress Design section).

Note: Slender section can only attain elastic moment capacity and cannot reach to plastic moment capacity.

D8.A.2.7 Combined Interaction Check

Members subjected to various forces – axial, shear, moment, torsion - are checked against combined interaction check.

Limit State Method

This interaction check is done taking care of two aspects:

- Section strength
- Overall member strength

Section strength interaction ratio is calculated as per sec. 9.3.1 of the code.

Overall Member Strength interaction ratio is calculated as per sec. 9.3.2, taking care of the design parameters `PSI`, `CMX`, `CMY`, and `CMZ`.

Working Stress Method

The following interactions are considered:

a. Combined Bending and Shear — No reduction in allowable stresses for the interaction of bending and shear is considered.

b. Combined Axial Compression and Bending — The following formulas are intended to require member stability:

$$f_c/f_{acy} + 0.6 \cdot K_y(C_{my}f_{bcy}/f_{abcy}) + K_{LT}f_{bcz}/f_{abcz} \leq 1.0$$

$$f_c/f_{acz} + 0.6 \cdot K_y(C_{my}f_{bcy}/f_{abcy}) + K_z f_{bcz}/f_{abcz} \leq 1.0$$

$$f_c/(0.6f_y) + f_{bcy}/f_{abcy} + f_{bcz}/f_{abcz} \leq 1.0$$

where

f_c	=	Actual axial compressive stress.
f_{acy}, f_{acz}	=	Allowable compressive stress, governed by buckling, about the local Y and Z axis, respectively.
f_{bcy}, f_{bcz}	=	Actual bending compressive stress about minor and major axes, respectively.
f_{abcy}, f_{abcz}	=	Allowable bending compressive stress about minor and major axes, respectively.

$$K_y = 1 + (\lambda_y - 0.2) \cdot n_y \leq 1 + 0.8 \cdot n_y$$

$$K_z = 1 + (\lambda_z - 0.2) \cdot n_z \leq 1 + 0.8 \cdot n_z$$

$$K_{LT} = 1 - 0.1 \cdot \lambda_{LT} \cdot n_y/(C_{mLT} - 0.25) \geq 0.1 \cdot n_y/(C_{mLT} - 0.25)$$

c. Combined Axial Tension and Bending — The following formulas are intended to require member stability:

$$f_t/f_{at} + f_{bty}/f_{abty} + f_{btz}/f_{abtz} \leq 1.0$$

where

f_t	=	Actual axial tensile stress.
f_{at}	=	Allowable axial tensile stress.
f_{bty}, f_{btz}	=	Actual bending tensile stress about minor and major axes, respectively.
f_{abty}, f_{abtz}	=	Allowable bending tensile stress about minor and major axes, respectively.

Solid Rod Design

The a_1 and a_2 values are not explicitly available for circular rod section in the IS800:2007 specification. However, there is a design note mentioned for Eq6-41 in the EN 1993-1-1:2005 specification regarding this. Thus, these values are taken as 1.0 by the program.

The IS800:2007 does not provide guidance for the M_{ndy} and M_{ndz} calculation for circular solid rod section. The formula mentioned in the CL 9.3.1.2(e) has been followed by the program for solid circular sections.

D8.A.2.8 Minimum Web Thickness

The minimum web thickness is checked against the serviceability requirements in 8.6.1.1 and the compression flange buckling requirement of 8.6.1.2.

If these sections are not satisfied, then a warning message is included in the output, including the clause which did not meet the web thickness requirements.

This check applies to the following sections:

- Sections with a stiffened web (i.e., connected to flanges along both longitudinal edges):
  - wide flange (rolled, UPT, tapered)
  - I section with cover plates
  - double I-section side-by-side
  - channel
  - double channel (back-to-back and front-to-front)
  - tube, HSS rectangle
- Sections with unstiffened web (i.e., connected to flange along only one longitudinal edge):
  - tee
  - single angle
  - double angle
  - star angle

D8.A.3 Member Property Specification

For specification of member properties, the specified steel section available in Steel Section Library of STAAD.Pro may be used.

The following section types may be used for design:

- I-shapes
- channel

- tee
- HSS tube
- HSS pipe
- angle
- double angle (including “star angle” arrangement; see below)
- double channel
- solid rod (from section database or solid prismatic circular sections which use steel material)

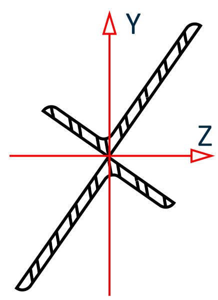
Member properties may also be specified using TR.19 User Steel Table Specification (on page 2241) except for the General and Prismatic member types (other than prismatic circular sections used as steel rods indicated above).

For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

Note: Refer to D8.B.12 Built-in Indian Steel Table (on page 1634) for details on using Indian steel sections per the ISI steel tables.

D8.A.3.1 Star Angle Arrangements

The design of double angles defined as “star angle” sections (i.e., a heel-to-heel alignment using the SA parameter; see section TR.20.1 Assigning Properties from Steel Tables (on page 2257)) per IS 800:2007.



Notes:

- Members using this section must be axial only (i.e., use `TRUSS` specification).
- It is assumed that the star angle arrangement is treated as a welded shape (i.e., two ISA welded at heels). Plated shapes are not accounted for in the program.
- The members are checked for tension and compression limit states for the cross-sectional properties calculated about the principal axes. Other limit states such as bending, shear, and interaction are not considered.

D8.A.4 Design Parameters

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are listed in the following table.

Table 164: Indian Steel Design IS 800:2007 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `IS800 LSD` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
ALPHA	0.8	A factor, based on the end-connection type, controlling the Rupture Strength of the Net Section, as per Section 6.3.3: 0.6) For one or two bolts 0.7) For three bolts 0.8) For four or more bolts 0.9) For threaded rods (per Section 6.3.2)
ANG	0	The location of the angle section through which the axial load passes by. It will only work if the section is single angle. The options are: 0) Centroid. 1) any of the legs. If the value is zero, it means the member is loaded through centroid. In that case Flexural-torsional buckling will not be checked.
ATG	None (Mandatory for Block Shear check)	Minimum Gross Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force.
ATN	None (Mandatory for Block Shear check)	Minimum Net Area in Tension from the bolt hole to the toe of the angle, end bolt line, perpendicular to the line of the force.
AVG	None (Mandatory for Block Shear check)	Minimum Gross Area in shear along bolt line parallel to external force.
AVN	None (Mandatory for Block Shear check)	Minimum Net Area in shear along bolt line parallel to external force.
BEAM	1	0) design at ends and those locations specified by the `SECTION` command. 1) design at ends and at every 1/12th point along member length (default).

Parameter Name	Default Value	Description
CAN	0.0	Beam Type, used for both deflection checks and the limit of bending strength in clause 8.2.1.2: 0) General Member, no limit used in clause 8.2.1.2, member deflection taken as the distance from the displaced end points of the member (DJ1, DJ2) 1) Cantilever Member, Md limited in clause 8.2.1.2, member deflection taken as the cantilever deflection as defined D1.B.1.2 Design Parameters (on page 1134) . 2) Simply Supported Member, Md limited in clause 8.2.1.2. The member deflection is taken as the same as a general member.
CMX	0.9	Equivalent uniform moment factor for Lateral Torsional Buckling(as per Table 18, section 9.3.2.2)
CMY CMZ	0.9	Cm value in local Y & Z axes, as per Section 9.3.2.2.
DFF	None(Mandatory for deflection check)	“Deflection Length” / Maximum allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of “Deflection Length”.
DJ2	End Joint of member	Joint No. denoting end point for calculation of “Deflection Length”.
DMAX	1000 in.	Maximum allowable depth.
DMIN	0.0 in.	Minimum allowable depth.
FU	420 MPA	Ultimate Tensile Strength of Steel in current units.
FXTY	0	The fixity of the connection between gusset plate and each end of member. Any value between 0 and 1 is allowed where 1 = Fixed and 0 = Hinged. Default is hinged. A value between 0 and 1 means the level of fixity that is between hinged and fixed. This value will be used to calculate the coefficients k1, k2 and k3 of table 12 by linear interpolation between their specified values for the two extreme (hinged and fixed) conditions.

Parameter Name	Default Value	Description
FYLD	250 MPA	Yield Strength of Steel in current units.
IMM	0	(LSD method only) Member classification for seismic detailing per Section 12 provisions: 0) Beam Member (default) 1) Column Member 2) Bracing (secondary member) See note b below.
KX	1.0	Effective Length Factor for Lateral Torsional Buckling (as per Table-15, Section 8.3.1)
KY	1.0	K value in local Y-axis. Usually, the Minor Axis.
KZ	1.0	K value in local Z-axis. Usually, the Major Axis.
LAT	0	Specifies lateral support of beam, as per Section 8.2.1 and 8.2.2, respectively: 0) Beam is laterally unsupported 1) Beam is laterally supported
LST	0	Defines the number of longitudinal stiffeners used: 0) No longitudinal stiffener 1) Longitudinal stiffener is provided at 0.2D of web from the compression flange 2) Longitudinal stiffeners are provided at 0.2D and 0.5D of the web from the compression flange
LX	Member Length	Effective Length for Lateral Torsional Buckling (as per Table-15, Section 8.3.1)
LY	Member Length	Length to calculate Slenderness Ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in Z-axis (Major).
MAIN	180	Allowable Slenderness Limit for Compression Member (as per Section 3.8) 0 = the default value will be used -1 = the slenderness check will not be performed (any other negative value will be ignored

Parameter Name	Default Value	Description
NBL	1	The number of bolts connecting each end of member with the gusset plate. The options are : 0) one bolt. 1) two or more bolts or weld.
NSF	1.0	Net Section Factor for Tension Member.
PROFILE	None	Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2676) for details.
PSI	1.0	Ratio of the Moments at the ends of the laterally unsupported length of the beam, as per Section 9.3.2.1: 0.8) where Factored Applied Moment and Tension can vary independently 1.0) For any other case
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SEISMIC	0	(LSD method only) Specifies if the detailing per Section 12 provisions is performed: 0) will *not* perform the detailing for the earthquake load (by default) 1) will perform the detailing for the earthquake load See Note a below.
STP	1	Specifies the section type per Table 2 and Table 10: 1) Hot rolled section 2) Welded section

Parameter Name	Default Value	Description
SSY	0	(LSD method only) Seismic bracing condition along the local Y axis (see note c below): 0- Unbraced along the local Y axis 1- Ordinary Concentrically Braced Frames (OCBF) in the local Y direction (R=4) applicable for X bracing only 2- Special Concentrically Braced Frame (SCBF) in the local Y direction (R=4.5) applicable for column or X bracing only 3- Ordinary Moment Frame (OMF) in the local Y direction (R=4) 4- Special Moment Frame (SMF) in the local Y direction (R=5)
SSZ	0	(LSD method only) Seismic bracing condition along the local Z axis (see note c below): 0- Unbraced along the local Z axis 1- Ordinary Concentrically Braced Frames (OCBF) in the local Z direction (R=4) applicable for X bracing only 2- Special Concentrically Braced Frame (SCBF) in the local Z direction (R=4.5) applicable for column or X bracing only 3- Ordinary Moment Frame (OMF) in the local Z direction (R=4) 4- Special Moment Frame (SMF) in the local Z direction (R=5)
TMAIN	400	Allowable Slenderness Limit for Tension Member (as per Section 3.8) 0 = the default value will be used -1 = the slenderness check will not be performed (any other negative value will be ignored
TRACK	0	Controls the levels of detail to which results are reported. 0) Minimum detail 1) Intermediate detail level 2) Maximum detail
TSP	0	Spacing of transverse stiffeners.

Parameter Name	Default Value	Description
TST	0	Used to control transverse stiffeners in design: 0) No Transverse Stiffener is provided 1) Transverse Stiffener is provided

Notes

a. The program will detail members as per the Section 12 provisions if the design parameter `SEISMIC` is set to 1. It is also recommended that a seismic load using an IS 1893 load definition is in the model. Seismic definitions using other seismic codes are not supported.

b. By default, all the members are considered as main members (i.e., `IMM` = 0). The scope for the seismic detailing depends upon the framing type. The program cannot determine the member or bracing type.

   For example, the detailing guidelines for OCBF framing are applicable for X type of bracing only. Therefore, you must assign the appropriate member type (`IMM` = 2) for the bracing members.

c. The response reduction factor, R, will be determined from the parameters `SSY` and `SSZ`. If this R differs from the that defined in the seismic load list, the program issues a warning.

D8.A.5 Code Checking and Member Selection

Both code checking and member sSelection options are available for the IS 800: 2007 code.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D8.A.5.1 Example 1

Commands for code checking:

```
UNIT NEWTON METER
PARAMETER 1
CODE IS800 LSD
ALPHA 0.7 ALL
CAN 1 MEMB 2
PSI 0.8 MEMB 2
TMAIN 350 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
```

D8.A.5.2 Example 2

Commands for member selection:

```
UNIT NEWTON METER
PARAMETER 1
CODE IS800 LSD
MAIN 160 MEMB 7
KY 0.8 MEMB 7
KZ 0.9 MEMB 7
```

```
FYLD 350 ALL
SELECT ALL
```

D8.A.6 Amendment 1 - January 2012

The various sections of Amendment-1 to the IS:800-2007 Code, dated January 2012, are implemented in STAAD.Pro as applicable.

Please refer the link below for the details of the list and their implementation status in STAAD.Pro.

http://communities.bentley.com/products/structural/structural_analysis___design/w/structural_analysis_and_design__wiki/29602.addendum-1-to-is-800-2007

D8.A.7 Seismic Detailing per Section 12

Seismic detailing of steel members per portions of Section 12 of IS 800-2007 can be performed by STAAD.Pro for LSD design.

The program will check seismic detailing requirements when the D8.A.4 Design Parameters (on page 1617) `SEISMIC` (design detailing for earthquake load) has been specified as 1. It is also recommended that an IS 1893 seismic load definition is included in the model.

Additionally, a set of automatic load combinations is included per IS 800 (in the IS 456 / IS 800 code selection) for IS 800 Clause 12.2.3 which is intended for use with these checks. These are:

1.2×DL + 0.5×LL + 2.5×EL
1.2×DL + 0.5×LL - 2.5×EL
0.9×DL + 2.5×EL
0.9×DL - 2.5×EL

Multiple earthquake load (EL) cases will result in multiples of the different load combinations, as each combination will use only one type of earthquake load in one direction.

Note: To limit the design checks to only the pertinent seismic load combinations per Cl. 12.2.3, a `LOAD LIST` may be used. Otherwise, the checks for Section 12 will be done for all load cases and combinations.

Among the various seismic criteria present in the section 12 of IS800:2007 only compression and tension strength and slenderness are considered as a failure criterion. Detailed output of the seismic checks are provided only for `TRACK 2` output. Other `TRACK` options will only report a seismic provision if it is the critical clause for member checks.

Limitations

Note: This is only available for design using the LFD method. It is not applicable to design using the ASD method.

As the design checks are performed on analytical members (rather than physical members), there are no means by which to ensure provisions of Cl. 12.5.1.2 (i.e., the “strong column, weak beam” design philosophy) are met. That is left to the engineer. Similarly, code checks which require comparison of one member's strength to another are not possible in design checks under STAAD.Pro.

Applicable Code Sections

The following portions of Section 12 have been implemented in STAAD.Pro:

- 12.2.1 - the R values are derived from the specified frame type. If this differs from the R values manually specified in the seismic load definition, a warning is presented in the output. If the assigned R in the local Z and local Y differs, the program will consider the minimum R.
- 12.2.3 - the IS 800 load combinations should be manually added or included from automatically generated load combinations
- 12.3 - the frame type is specified by use of the `SSY` and `SSZ` parameters
- 12.5.1.1
- Ordinary concentrically braced frames
  - 12.7.1.1 - limits the use of OCBFs in seismic zones IV and V, as well as in seismic zone III when the importance factor is greater than one
  - 12.7.1.2 - the bracing type used in the model should be consistent with the `SSY` and `SSZ` parameters specified. Members used as bracing should also be specified using the `IMM` parameter.
  - 12.7.2.1
  - 12.7.2.2
  - 12.7.2.4
  - 12.7.2.6
- Special concentrically braced frames
  - 12.8.1.1
  - 12.8.1.2 - the bracing type used in the model should be consistent with the `SSY` and `SSZ` parameters specified. Members used as bracing should also be specified using the `IMM` parameter.
  - 12.8.2.1
  - 12.8.2.2
  - 12.8.2.3
  - 12.8.2.5
  - 12.8.2.7
  - 12.8.4.1
- 12.10.1.1 - Ordinary moment frames shall not be used in seismic zones IV and V, as well as in seismic zone III when the importance factor is greater than one
- Special moment frames
  - 12.11.1 - checks the material grades assigned (i.e, the `FYLD` and `FU` parameter values assigned) are equal to or higher than grade E250B.
  - 12.11.1.1
  - 12.11.3.1

D8.B. Indian Codes - Steel Design per IS 800 - 1984

STAAD.Pro is capable of performing steel design based on the Indian code IS 800 - 1984 *General construction in steel - Code of practice*.

Note: The IS 800 - 1984 is not appropriate for the design of hollow pipe or tube sections. The design of such members should be done using D8.A. Indian Codes - Steel Design per IS 800 - 2007 (on page 1607).

D8.B.1 Design Operations

STAAD.Pro contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.
- Specify whether to perform member selection by optimization.

These operations may be repeated by the user any number of times depending upon the design requirements. The entire ISI steel section table is supported. D8.B.13 Column With Lacings And Battens (on page 1638) describes the specification of steel sections.

D8.B.2 General Comments

This section presents some general statements regarding the implementation of Indian Standard code of practice (IS:800-1984) for structural steel design in STAAD.Pro. The design philosophy and procedural logistics for member selection and code checking are based upon the principles of allowable stress design.

Two major failure modes are recognized:

i. failure by overstressing
ii. failure by stability considerations.

The flowing sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economic section is selected on the basis of least weight criteria. The code checking part of the program checks stability and strength requirements and reports the critical loading condition and the governing code criteria.

It is generally assumed that the engineer will take care of the detailing requirements like provision of stiffeners and check the local effects such as flange buckling and web crippling.

There are no special provisions specifically for double-I sections in IS 800-1984. Therefore, these sections are designed in STAAD.Pro as other general sections using the section property values of the double-I section specified.

D8.B.3 Allowable Stresses

The member design and code checking in STAAD.Pro are based upon the allowable stress design method as per IS:800 (1984). It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions.

It is not practical to describe every aspect of IS:800 here. This section, however, will discuss the salient features of the allowable stresses specified by IS:800 as implemented in STAAD.Pro. Appropriate sections of IS:800 will be referenced during the discussion of various types of allowable stresses.

D8.B.3.1 Axial Stress

Tensile Stress

The allowable tensile stress, as calculated in STAAD.Pro as per IS:800 is described below.

The permissible stress in axial tension, σ_{at} in MPa on the net effective area of the sections shall not exceed

$$\sigma_{at} = 0.6\ f_y$$

where

f_y = minimum yield stress of steel in MPa

Compressive Stress

Allowable compressive stress on the gross section of axially loaded compression members shall not exceed $0.6 f_y$ nor the permissible stress s_{ac} calculated based on the following equation (per Clause: 5.1.1):

$$\sigma_{ac} = 0.6\{(f_{cc} \cdot f_y)/[(f_{cc})^n + (f_y)^n]^{1/n}\}$$

where

σ_{ac}	=	Permissible stress in axial compression, in MPa
f_y	=	Yield stress of steel, in MPa
f_{cc}	=	Elastic critical stress in compression = $\pi^2 E/\lambda^2$
E	=	Modulus of elasticity of steel, 2 X 10^5 MPa
λ	=	l/r = Slenderness ratio of the member, ratio of the effective length to appropriate radius of gyration
n	=	A factor assumed as 1.4.

D8.B.3.2 Bending Stress

The allowable bending stress in a member subjected to bending is calculated based on the following formula.

$$\sigma_{bt} \text{ or } \sigma_{bc} = 0.66 f_y \qquad \text{(Clause: 6.2.1)}$$

where

σ_{bt}	=	Bending stress in tension
σ_{bc}	=	Bending stress in compression
f_y	=	Yield stress of steel, in MPa

For an I-beam or channel with equal flanges bent about the axis of maximum strength (z-z axis), the maximum bending compressive stress on the extreme fibre calculated on the effective section shall not exceed the values of maximum permissible bending compressive stress. The maximum permissible bending compressive stress shall be obtained by the following formula: (Clause: 6.2.2)

$$\sigma_{bc} = 0.66\frac{f_{cb} f_y}{\left[(f_{cb})^n + (f_y)^n\right]^{1/n}} \qquad \text{(Clause 6.2.3)}$$

where

f_y	=	Yield stress of steel, in Mpa
n	=	A factor assumed as 1.4.
f_{cb}	=	Elastic critical stress in bending, calculated by the following formula:
		$= k_1\left(X + k_2 Y\right)\frac{c_2}{c_1}$
X	=	$Y\sqrt{1 + \frac{1}{20}\frac{\pi^2}{r_y D}}$ in MPa
Y	=	$\frac{26.5(10)^5}{(1/r_y)^2}$
k_1	=	a coefficient to allow for reduction in thickness or breadth of flanges between points of effective lateral restraint and depends on y, the ratio of the total area of both flanges at the point of least bending moment to the corresponding area at the point of greatest bending moment between such points of restraint.

k_2	=	a coefficient to allow for the inequality of flanges, and depends on w, the ratio of the moment of inertia of the compression flange alone to that of the sum of the moment of the flanges each calculated about its own axis parallel to the y-yaxis of the girder, at the point of maximum bending moment.
l	=	effective length of compression flange
r_y	=	radius of gyration of the section about its axis of minimum strength (y-y axis)
T	=	mean thickness of the compression flange, is equal to the area of horizontal portion of flange divided by width.
D	=	overall depth of beam
c_1 ,c_2	=	respectively the lesser and greater distances from the section neutral axis to the extreme fibres.

D8.B.3.3 Shear Stress

Allowable shear stress calculations are based on Section 6.4 of IS:800.

For shear on the web, the gross section taken into consideration consist of the product of the total depth and the web thickness. For shear parallel to the flanges, the gross section is taken as 2/3 times the total flange area.

D8.B.3.4 Combined Stress

Members subjected to both axial and bending stresses are proportioned accordingly to section 7 of IS:800. All members subject to bending and axial compression are required to satisfy the equation of Section 7.1.1.(a) for intermediate points, and equation of Section 7.1.1.(b) for support points.

For combined axial tension and bending the equation of Section 7.1.2. is required to be satisfied.

Cm coefficients are calculated according to the specifications of Section 7.1.3. information regarding occurrence of sidesway can be provided through the use of parameters `SSY` and `SSZ`. In the absence of any user-provided information, sidesway will be assumed.

D8.B.4 Design Parameters

In the STAAD.Pro implementation of IS:800, you are allowed complete control of the design process through the use of design parameters. Available design parameters to be used in conjunction with IS:800 are listed in the table below along with their default values and applicable restrictions.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 165: Indian Steel Design IS 800:1984 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `INDIAN` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	1.0	0.0) design only for end moments and those at locations specified by the SECTION command. 1.0) calculate section forces at twelfth points along the beam, design at each intermediate location and report the critical location where ratio is maximum.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maxm. allowable local deflection. See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See Note 1 (on page 1631))
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See Note 1 (on page 1631))
DMAX	100.0 cm.	Maximum allowable depth.
DMIN	0.0 cm.	Minimum allowable depth.
FYLD	250 MPa (36.25 ksi)	Yield strength of steel.
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length in local y-axis to calculate slenderness ratio.
LZ	Member Length	Same as above except in local z-axis (major).

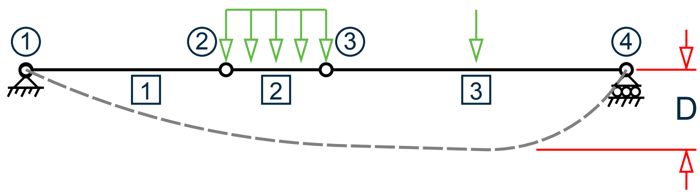
Parameter Name	Default Value	Description
MAIN	180 (Comp. Memb.)	Allowable Kl/r for slenderness calculations for compression members.
NSF	1.0	Net section factor for tension members.
PROFILE	-	Used to search for the lightest section for the profile(s) specified for member selection. See TR.48.1 Parameter Specifications (on page 2676) for details.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SSY	0.0	0.0) Sidesway in local y-axis. 1.0) No sidesway
SSZ	0.0	Same as above except in local z-axis.
TMAIN	400 (Tension Memb)	Allowable Kl/r for slenderness calculations for tension members.
TRACK	0.0	0.0) Suppress critical member stresses 1.0) Print all critical member stresses 2.0) Print expanded output. If there is deflection check it will also print the governing load case number for deflection check whenever critical condition for design is not DEFLECTION.
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.

Parameter Name	Default Value	Description
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.

Notes

a. "Deflection Length" is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different. A straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Thus, for all three members here, `DJ1` should be `1` and `DJ2` should be `4`.



D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

b. If `DJ1` and `DJ2` are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.

c. The above parameters may be used in conjunction with other available parameters for steel design.

D8.B.5 Stability Requirements

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. Section 3.7 of IS:800 summarizes the maximum slenderness ratios for different types of members.

In the STAAD.Pro implementation of IS:800, the appropriate maximum slenderness ratio can be provided for each member. If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

D8.B.6 Truss Members

A truss member is capable of carrying only axial forces. So in design no time is wasted in calculating bending or shear stresses, thus reducing design time considerably. Therefore, if there is any truss member in an analysis (like bracing or strut, etc.), it is wise to declare it as a truss member rather than as a regular frame member with both ends pinned.

D8.B.7 Deflection Check

This feature allows you to consider deflection as a criteria in the CODE CHECK and MEMBER SELECTION processes.

The deflection check may be controlled using three parameters which are described in D8.B.4 Design Parameters (on page 1628).

Note: Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

D8.B.8 Code Checking

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS:800 (1984) requirements.

Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

The code checking output labels the members as “PASS” or “FAIL”. In addition, the critical condition (applicable IS:800 clause no.), governing load case, location (distance from the start) and magnitudes of the governing forces and moments are also printed out.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D8.B.9 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, that is, the lightest section, which satisfies the applicable code requirements.

The section selected will be of the same type (I-Section, Channel etc.) as originally specified by the user. Member selection may be performed with all types of steel sections listed in D8.B.12 Built-in Indian Steel Table (on page 1634) and user-provided tables. Selection of members whose properties are originally provided from user-specified table is limited to sections in the user-provided table.

Member selection cannot be performed on members whose cross sectional properties are specified as PRISMATIC.

The process of MEMBER SELECTION may be controlled using the parameters listed in D8.B.4 Design Parameters (on page 1628). The parameters DMAX and DMIN may be used to specify member depth constraints for selection. If PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D8.B.10 Member Selection By Optimization

Steel section selection of the entire structure may be optimized. The optimization method utilizes a state-of-the -art numerical technique which requires automatic multiple analysis.

You may start without a specifically designated section. However, the section profile type (`BEAM`, `COLUMN`, `CHANNEL`, `ANGLE`) must be specified using the `ASSIGN` command (see TR.20.5 Assign Profile Specification (on page 2268)). The optimization is based on member stiffness contributions and corresponding force distributions. An optimum member size is determined through successive analysis/design iterations. This method requires substantial computer time and hence should be used with caution.

Refer to TR.49.2 Member Selection by Optimization (on page 2679) for additional details.

D8.B.11 Tabulated Results of Steel Design

For code checking or member selection, the program produces the result in a tabulated fashion. The items in the output table are explained as follows:

MEMBER	the member number for which the design is performed
TABLE	the INDIAN steel section name which has been checked against the steel code or has been selected.
RESULT	prints whether the member has PASSED or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark in front of the member number.
CRITICAL COND	the section of the IS:800 code which governs the design.
RATIO	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
LOADING	provides the load case number which governs the design.
FX, MY, and MZ	provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX,MY and MZ are printed since they are the ones which are of interest, in most cases.
LOCATION	specifies the actual distance from the start of the member to the section where design forces govern.

Note: If the parameter TRACK is set to `1.0`, the program will block out part of the table and will print allowable bending stresses in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV). When the parameter TRACK is set to 2.0 for all members parameter code values are as shown in the following example.

```
                       STAAD.PRO CODE CHECKING - (          IS-800)   v1.0
                       ********************************************


   |---------------------------------------------------------------------------|
   |                                                   Y         PROPERTIES    |
   |*************                                      |         IN  CM  UNIT  |
   |            *   |============================|  ===|===      ------------  |
   |MEMBER   7  *   |  INDIAN SECTIONS           |     |         AX =   85.0   |
   |            *   | ST  ISWB400                |     |    --Z  AY =   34.4   |
   |DESIGN CODE *   |                            |     |         AZ =   34.7   |
   |  IS-800    *   ===============================  ===|===      SY =  138.8   |
   |            *                                                SZ = 1171.3   |
   |            *   |<---LENGTH (ME=     3.00 --->|                RY =    4.0   |
   |*************                                                RZ =   16.6   |
   |                                                                           |
   |                112.1( KN-METR)                                            |
```

```
|PARAMETER        |L1                                         STRESSES     |
|IN   NEWT  MM    |                                           IN   NEWT  MM|
|---------------  +                                           -------------|
| KL/R-Y=   74.2  |                                           FA  =  150.0 |
| KL/R-Z=   18.1  +                                           fa  =    1.0 |
| UNL   = 3000.0  |                                           FCZ =  139.9 |
| C     =  400.0  +                                           FTZ =  165.0 |
| CMY   =   0.60  |                                           FCY =  165.0 |
| CMZ   =   0.40  +                                           FTY =  165.0 |
| FYLD  =  249.9  |                                       L3  fbz =   95.7 |
| NSF   =    0.9  +---+---+---+---+---+---+---+---+---+---|   fby =    0.0 |
| DFF   =    0.0  90.5                                        FV  =  100.0 |
| dff   =    0.0              ABSOLUTE MZ ENVELOPE            fv  =   17.1 |
|                                (WITH LOAD NO.)                           |
|                                                                          |
|                      MAX FORCE/ MOMENT SUMMARY ( KN-METR)                |
|                      -------------------------                           |
|                                                                          |
|                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
|                                                                          |
|      VALUE     -23.9        60.6        0.0         0.0       112.1      |
|   LOCATION       0.0         3.0        0.0         0.0         0.0      |
|    LOADING         3           1          0           0           1      |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                        DESIGN SUMMARY ( KN-METR)                       *|
|*                        --------------                                  *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/            *|
|         FX                MY            MZ          LOCATION             |
|        ======================================================            |
|          PASS       7.1.2 BEND C     0.684             1                 |
|          7.39 T          0.0       -112.1            0.00                |
|*                                                                        *|
|**************************************************************************|
```

D8.B.12 Built-in Indian Steel Table

This is an important feature of the program since the program will read section properties of a steel member directly from the ISI steel tables (as published in ISI-800). These properties are stored in memory corresponding to the section designation (e.g., ISMB250, etc.). If called for, the properties are also used for member design. Since the shear areas are built in to these tables, shear deformation is always considered for these members.

Almost all ISI steel tables are available for input. A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.

D8.B.12.1 Rolled Steel Beams (ISJB, ISLB, ISMB, and ISHB)

All rolled steel beam sections are available the way they are designated in the ISI handbook (e.g., ISJB225, ISWB400, etc.)

```
20 TO 30 TABLE ST ISLB325
```

Note: In case of two identical beams, the heavier beam is designated with an "A" on the end (e.g., ISHB400 A, etc.).

```
1 TO 5 TABLE ST ISHB400A
```

D8.B.12.2 Rolled Steel Channels (ISJC, ISLC and ISMC)

All these shapes are available as listed in ISI section handbook. Designation of the channels are per the scheme used by ISI.

```
10 TO 20 BY 2 TABLE ST ISMC125
12 TABLE ST ISLC300
```

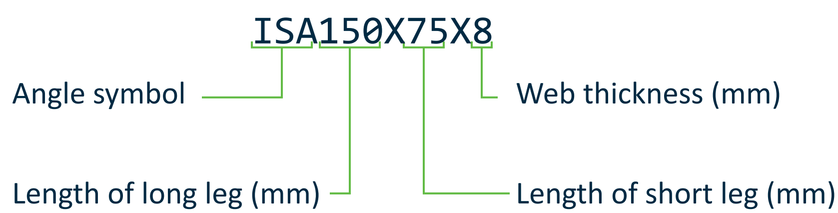
D8.B.12.3 Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel (e.g., D ISJC125, D ISMC75, etc.).

```
21 22 24 TABLE D ISLC225
```

D8.B.12.4 Rolled Steel Angles

Both rolled steel equal angles and unequal angles are available for use in the STAAD implementation of ISI steel tables. The following example with explanations will be helpful in understanding the input procedure:



At present there is no standard way to define the local y and z axes for an angle section. The standard section has local axis system. The standard angle is specified as:

```
51 52 53 TABLE ST ISA60X60X6
```

This specification has the local z-axis (i.e., the minor axis corresponding to the V-V axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD.Pro provides for this convention by accepting the command:

```
54 55 56 TABLE RA ISA50X30X6
```

Tip: "RA" denotes reverse angle

D8.B.12.5 Double Angles

Short leg back-to-back or long leg back-to-back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose. For example,

```
14 TO 20 TABLE LD ISA50X30X5 SP 1.5
23 27 TABLE SD ISA75X50X6
```

D8.B.12.6 Rolled Tees (ISHT, ISST, ISLT and ISJT)

All the rolled tee sections are available for input as they are specified in the ISI handbook. The following example illustrates the designated method.

```
1 2 5 8 TABLE ST ISNT100
67 68 TABLE ST ISST250
```

D8.B.12.7 Pipes (Circular Hollow Sections)

To designate circular hollow sections from ISI tables, use PIP followed by the numerical value of diameter and thickness of the section in mm omitting the decimal section of the value provided for diameter. The following example will illustrate the designation.

```
10 15 TABLE ST PIP 213.2
```

specifies a 213 mm dia. pipe with 3.2 mm wall thickness

Circular pipe sections can also be specified by providing the outside and inside diameters of the section. For example,

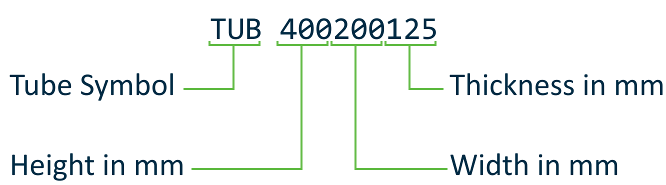
```
1 TO 9 TABLE ST PIPE OD 25.0ID 20.0
```

specifies a pipe with outside dia. of 25 and inside dia. of 20 in current length units

Only code checking and no member selection will be performed if this type of specification is used.

D8.B.12.8 Tubes (Rectangular or Square Hollow Sections)

Designation of tubes from the ISI steel table is illustrated below.



For example,

```
15 TO 25 TABLE ST TUB 160808
```

Tubes, like pipes, can also be input by their dimensions (Height, Width and Thickness) and not by any table designations.

```
6 TABLE ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5.

Note: Only code checking and no member selection is performed for TUBE sections specified this way.

D8.B.12.9 Plate And Angle Girders (With Flange Plates)

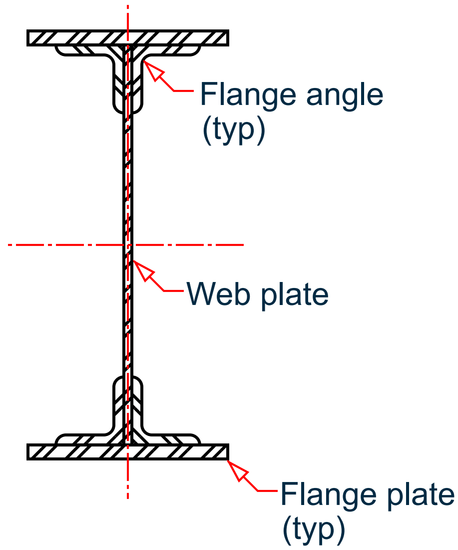
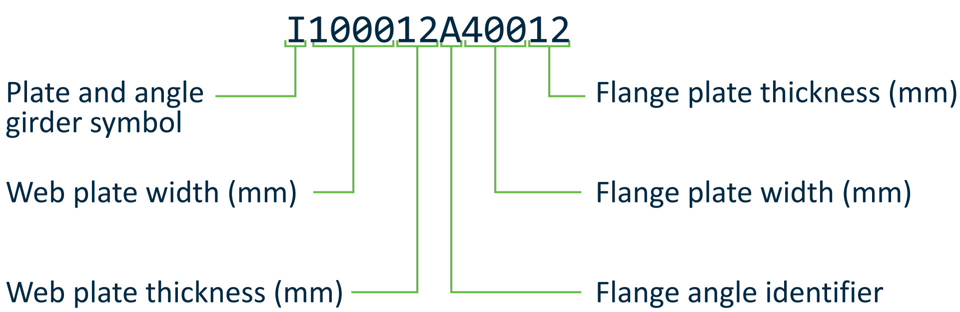


Figure 171: Plate and Angle girder

Plate and angle grinders (with flange plates) are available as listed in Table XX of the ISI section handbook. The input of these composite sections are given as indicated in the following diagram. The web height and thickness are the first numeric entries, followed by a letter identifier used to indicate the angle size used, the followed by the flange width and thickness entries.



Flange angle key

Angle Identifier	**Angle Size Used (A × B × t)**
A	150×150×18
B	200×100×15
C	200×150×18
D	200×200×18

D8.B.12.10 Single Joist with Channels and Plates on the Flanges to be Used as Girders

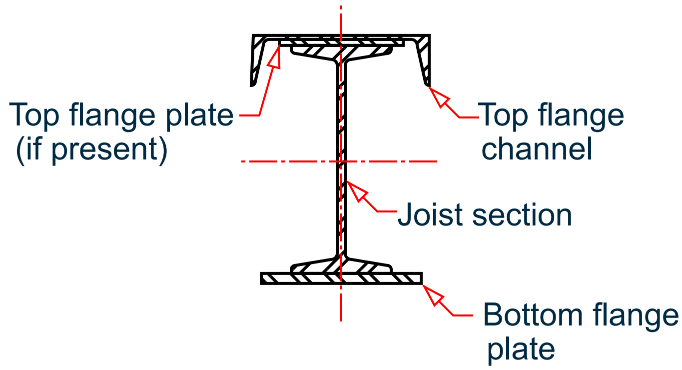
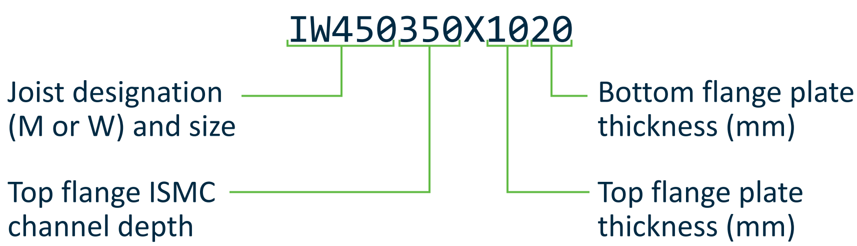


Figure 172: Single Joist with Channel and Plates on the Flanges

All single joist with channel and plates on the flanges to be used as girders are available as listed in Table XV of the ISI section handbook. The input of these composite sections are given as indicated in the following diagram. The following example with explanations will be helpful in understanding the input procedure.



The plate widths are specific to the combination of joist and channel, so they are not directly specified in the section size. To specify a section *without* a top flange plate (to flange channel only), then a zero may be specified for the top flange thickness value. For example, a `IM600400X016` indicates an ISMB 600 joist with an ISMC 400 top flange channel (no plate) and a 320×16 mm bottom plate.

Note: The table lists two different ISWB600 sections. The heavier ISWB600 has been omitted, since the lighter ISWB600 is more efficient.

D8.B.13 Column With Lacings And Battens

For columns with large loads it is desirable to build rolled sections at a distance and inter-connect them. The joining of element sections is done by two ways:

a. Lacing
b. Batten

Double channel sections (back-to-back and face-to-face) can be joined either by lacing or by batten plates having riveted or welded connection.

The following table gives the parameters that are required for Lacing or batten design. These parameters will have to be provided in unit NEW MMS along with parameters defined in D8.B.4 Design Parameters (on page 1628).

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 166: Parameters used in Indian Lacing or Batten steel member design.

Parameter Name	Default Value	Description
CTYPE	1	Type of joining 1) implies single lacing with riveted connection 2) implies double lacing with riveted connection 3) implies single lacing with welded connection 4) implies double lacing with welded connection 5) implies batten with riveted connection 6) implies batten with welded connection
COG	0.0 mm	Center of gravity of the channel. This parameter is used when member properties are defined through user provided table using GENERAL option.
DBL	20 mm	Nominal diameter of rivet
DCFR	0.0	Used when member properties are defined through user provided table using GENERAL option. 0) double channel back-to-back. 1) double channel face-to-face.
EDIST	32 mm (riveted connection) 25 mm (welded connection)	Edge Distance.
FVB	100 N/mm^2	Allowable shear stress in rivet
FYB	300 N/mm^2	Allowable bearing stress in rivet
SPA	0.0 mm	Spacing between double channels. This parameter is used when member properties are defined through user provided table using GENERAL option.

Parameter Name	Default Value	Description
THETA	50 degree	Angle of inclination of lacing bars. It should lie between 40 degree and 70 degree.
WMIN	6 mm	Minimum thickness of weld
WSTR	108 N/mm^2	Allowable welding stress

D8.C. Indian Codes - Cold Formed Steel Design per IS 801 - 1975

STAAD.Pro is capable of performing steel design based on the Indian code IS 801 1975 *Code of practice for use of cold-formed light gauge steel structural members in general building construction*, reaffirmed in 2010. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations. Cold work of forming strengthening effects has been included as an option.

D8.C.1 Cross-Sectional Properties

You specify the geometry of the cross-section by selecting one of the section shape designations from the Gross Section Property Tables from IS:811-1987 (*Specification for cold formed light gauge structural steel sections*).

The AISI Design Manual has been followed to calculate section properties (e.g., Cw) which are not available in the Indian Standard and Manual.

Checks per IS 801 are supported for the following profile shapes:

- Angle with lips
- Angle
- Channel with Lips
- Channel (or eave strut)
- Zee or zee purlin
- Zee with lips
- Hat
- Tube
- Pipe

The properties listed in the tables are gross section properties. STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

D8.C.2 Design Procedure

The design procedure according to IS 801 is based on an allowable stress method. In essence, the principle is to determine the maximum allowable design stress for a given section profile. The section properties that are used to calculate the allowable stress are based on the gross section properties or the “effective” section properties, based on the load being applied and the check being considered.

The design procedure varies slightly from profile to profile, based on the characteristics of the elements that form the profile shape. IS 801 classifies the elements (flat plates) that form the section profile into the following element types:

- stiffened element – an element that is supported along two opposite edges either by another element or a stiffening element such as a lip
- unstiffened element – an element that is supported by another element only along one edge
- multiple stiffened element – an element that is supported in between web(s) or stiffened element(s) by intermediate stiffeners that are parallel to the direction of stress

The various checks for a section profile and the design methods adopted depend on the type of the elements that form the profile. The following clauses from IS 801 are implemented and are used as appropriate:

- Cl 5.2.1
  - Cl.5.2.1.1
- Cl.5.2.3
- Cl.5.2.4
- Cl.6.1
  - Cl.6.1.1.1
- Cl.6.2
- Cl.6.3
- Cl.6.4
  - Cl.6.4.1
  - Cl.6.4.2
  - Cl.6.4.3
- Cl.6.6
  - Cl.6.6.1.1
  - Cl.6.6.1.2
  - Cl.6.6.1.3
- Cl.6.7
  - Cl.6.7.1
  - Cl.6.7.2
  - Cl.6.7.3
- Cl.6.8

Note: Deflection checks based on the effective section properties are not performed.

D8.D.2.1 Maximum allowable section segment dimensions

Checks will be done to verify the maximum allowable flat width ratios for the flange segments forming section profile as per Cl.5.2.3 and for the web segments as per Cl.5.2.4. Note that Cl.5.2.3(d) will not be considered as this can vary based on the amount of curling.

D8.D.2.2 Maximum member slenderness

Member slenderness checks will be performed for members subject to compression. The maximum allowable slenderness for a compression member will be limited to 200 as per Cl.6.3.3.

Slenderness for a member about both the z & y local axes will be calculated. If any of these exceeds the limiting value of 200, the program will consider that as a failure criterion. Note that members subject to tension will not have any slenderness checks performed.

D8.D.2.3 Members subject to tension

The following check will be performed for all members in tension:

$$\text{Tensile stress ratio} = F_{actual} / F$$

where

F_{actual}	=	the applied stress on the member. = Applied Tension / A
A	=	the net cross section area of the profile

Note: If the parameter list includes the `NSF` parameter, the net area will be calculated as $A \times NSF$, where A is the gross area of the section. If NSF has not been specified, a default value of 1.0 will be used.

The maximum allowable design stress for members in tension will be calculated as:

$$F = 0.6 \times F_y$$

where

F_y	=	the minimum yield strength of the section.

If the increase in steel strength due to cold work forming is to be considered (ref. `CWY` param), then the design stress will be calculated as:

$$F = 0.6 \times F_{ya}$$

where

F_{ya}	=	the average yield point of the full section and will be calculated as: $Fya = C \times Fyc + (1 - C)\ Fyt.$ (refer to Cl.6.6.1.a for the description of the terms used.)

Note: The FY and FU values are expected to be user inputs. The default value of FY will be taken as 3600 kg/ cm^2(Table 2 of IS:801) and FU will be taken as 4588 kg/cm^2.

D8.D.2.4 Members subject to compression

The following check will be performed for all members in compression:

$$\text{Compression check ratio} = F_{actual} / F_{cap}$$

where

F_{actual}	=	the applied axial stress on the member
F_{cap}	=	the allowable compressive stress for the member

The compression capacity will be calculated as follows:

- For profile shapes with stiffened elements alone and not susceptible to Flexural torsional buckling (Tube):

  The maximum allowable design stress will be calculated as:

  $$F = 0.6 \times Fy$$

  where

F_y	=	the minimum yield strength of the section.

  Note: Allowance will be made for cold forming strength as required.

  Note that if the effect of cold work forming is to be considered, then the enhanced strength Fya will have to be determined based on the value of the factor Q. Hence to determine the factor Q, the program would use the design stress (= $0.6 \times F_y$). This factor Q will then be used to determine the enhanced strength Fya, which will then be used for the design checks.

This design stress, F, will be used to calculate the effective area of the section. The procedure to calculate the effective area involves calculating the effective width of each element that forms the profile. The effective width of each element will be calculated as described below:

- For elements without intermediate stiffeners:

  The effective width of such elements will be calculated as per the equations in Cl.5.2.1.1 (for closed square and rectangular tubes). The value of 'f" used will be the maximum allowable design stress F as in section 3.2.1 above.

  > **Note:** The current implementation does not allow for sections to have multiple stiffened elements as per Cl.5.2.1.2. Hence Cl.5.2.1.2 is *not* be considered for calculating the effective width of elements.

  The maximum allowable compression stress for the member, Fc, will then be determined based on the equations as per Cl.6.6.1.1. Note that the factor Q will be based on the effective area determined using the design stress F as mentioned above.
- For profile shapes with stiffened elements alone (Pipe sections):

  The maximum allowable compressive stress for pipes will be determined as per cl.6.8:

  For sections with D/t < 232000 / Fy, F = 0.6 × Fy

  For section with 232000 / Fy < D/t < 914000 / Fy, F = 46540 / (D/t) + 0.399 Fy

  Section with D/t > 914000 / Fy will not be designed.

  A check will also be done to make sure that the compressive stress < Fa1 as determined by 6.6.1.1. for a value of Q=1.
- For profile shapes with un-stiffened elements alone (Angle sections):

  The maximum allowable design stress Fc will be calculated as per the equations in Cl.6.2 (a), 6.2(b), 6.2(c) or 6.2 (d) as appropriate, based on the plate width-to-thickness ratio of each element.

  This design stress, Fc, will be used to calculate the factor Q when checking for flexural/flexural torsional buckling failure modes. Note that since this section is subject to torsional buckling, the factor Q, to be used to calculate the allowable compressive stress Fa1 as per Cl. 6.6.1.2/3, will be calculated as the ratio between the Fc for the element with the largest w/t ratio to the basic design stress as per Cl.6.1 & 6.2.
- For profile shapes with both stiffened and unstiffened elements and /or susceptible to Flexural torsional buckling (Channel, Zee, Hat, Channels with Lips, Zee with Lips, Angle with lip sections sections):

  These sections are subject to torsional-flexural buckling. The factor Q will be determined as per Cl.6.6.1.1 (3) as:

  $$Q = Qs \times Qa$$

  Qs will be calculated as the ratio between the Fc for the element with the largest w/t ratio to the basic design stress as per Cl.6.1 & 6.2.

  Qa will be calculated for the stiffened elements as per Cl.6.6.1.1 (1) but with Fa taken as the stress used to calculate Qs.

  If Q =1.0, the allowable stress Fa2 will be calculated as per Cl.6.6.1.2. If however the factor Q < 1.0, the allowable stress Fa2 will be calculated as per Cl.6.6.1.2, but with the term Q being replaced with Q.Fy.

D.2.5 Members subject to bending

The following check will be performed for all members in bending:

$$\text{Bending check ratio} = M_{actual} / M_{cap}$$

where

M_{actual}	=	the applied bending stress on the member
M_{cap}	=	the allowable bending stress of the member

This check will be done separately for bending about both the Z & Y axes.

The bending capacity of the section will be based on the maximum allowable bending stress, F_b. The maximum allowable stress in a member subject to bending shall not exceed the following:

1. The maximum allowable stress for the extreme tension fiber as given in Cl.6.1

   $$F_{bt\_allowable} = F = 0.6 \times F_y$$

   where

F_y	=	the minimum yield strength of the section

2. The maximum allowable stress in the extreme compression fiber as given in Cl.6.2

   This will be calculated based on the w/t ratio of the compression element as per Cl.6.2(a), Cl.6.2(b), Cl.6.2(c), or Cl.6.2(d) as applicable. Note that these clauses will be applicable only for unstiffened elements in compression. For stiffened elements F_b will be taken as given in Cl.6.1
3. The maximum allowable lateral buckling stress as given in C.l6.3 (a) or Cl.6.3 (b) as appropriate for the section profile.

   This clause will only be applicable for sections that are subject to bending about their major axis. For sections subject to minor axis bending this check will be ignored.

   The moment of inertia terms, Iyc and the section modulus term Zxc in the equations will be based on the entire section. The value of the moment factor Cb, will be based on the end moments of the analytical member being designed. If the member is subject to axial loads along with the bending moment, the value of Cb will be taken as 1.0
4. The maximum allowable bending stress in the webs of a given section (I, C, Hat, Tube , Z), Fbw shall be calculated as per Cl6.4.2 of the code.

For pipe sections:

The maximum allowable compressive stress for pipes will be determined as per cl.6.8:

- For sections with $D/t < 232000 / F_y$, $F = 0.6 \times F_y$
- For sections with $232000 / F_y < D/t < 914000 / F_y$, $F = 46540 / (D/t) + 0.399 F_y$
- Sections with $D/t > 914000 / F_y$ will not be designed.

D.2.6 Members subject to shear

The applied shear stress will be calculated based on the calculated shear area of the given section profile. The shear area for the various shapes will be taken as follows:

- Angle with Lips
  - For shear along the Y axis: $(D\text{-}t)\times t + 2.0\times(Lip\ Length\text{-}t)\times t$
  - For shear along the X axis: $(B\text{-}t)\times t$
- Angle
  - For shear along the Y axis: $(D\text{-}t)\times t$

- For shear along the X axis: $(B\text{-}t)\times t$
- Channel with Lips
  - For shear along the Y axis: $(D\text{-}2t)\times t + 2.0\times(Lip\ Length\text{-}t)\times t$
  - For shear along the X axis: $2.0\times B\times t$
- Channel or Eave Strut
  - For shear along the Y axis: $(D\text{-}2t)\times t$
  - For shear along the X axis: $2.0\times B\times t$
- Zee or Zee Purlin
  - For shear along the Y axis: $(D\text{-}2t)\times t$
  - For shear along the X axis: $2.0\times B\times t$
- Zee with Lips
  - For shear along the Y axis: $(D\text{-}2t)\times t + 2.0\times(Lip\ Length\text{-}t)\times t$
  - For shear along the X axis: $2.0\times B\times t$
- Hat
  - For shear along the Y axis: $2.0 \times (D\text{-}t)\times t$
  - For shear along the X axis: $B\times t + 2.0\times(Lip\ Length\text{-}t)\times t$
- Tube
  - For shear along the Y axis: $2.0\times(D\text{-}2t)\times t$
  - For shear along the X axis: $2.0\times(B\text{-}2t)\times t$
- Pipe
  - For shear along the Y axis: $0.5 \times 2\times\pi\times r\times t$
  - For shear along the X axis: $0.5 \times 2\times\pi\times r\times t$

For shear checks along the local Y-axis, the maximum allowable shear stress, Fv, in the web of a section (I, C, Hat, Tube, Z) will be calculated as per Cl.6.4.1 based on the h/t ratio of the web element.

The code does not explicitly mention about shear checks along the horizontal (Z-axis). Hence, the program takes the maximum allowable shear stress along Z as $0.4\times Fy$.

D.2.7 Members subject to combined axial, bending, and shear

- Checks for members subject to combined axial compression and bending:

  The combined stress checks will be based on whether the member is susceptible to torsional buckling mode or not. The implementation will consider all loads as being applied through the shear center.

  - Members not susceptible to torsional buckling

    For members that are not susceptible to torsional buckling, both the interaction equations as per Cl.6.7.1 will be checked as below:

    $$fa/ Fa1 + (Cmx \times fbx) / [(1- fa /F'ex)\times Fbx] + (Cmy \times fby) / [(1- fa /F'ey)\times Fby] \le 1.0$$

    $$fa/ Fa0 + fbx/ Fb1x + fby/ Fb1y \le 1.0$$

    where

    fa = P/ A

    with A = the full cross section area

fbx	=	maximum applied bending stress about the x -axis. Note that fb for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.
fby	=	maximum applied bending stress about the y -axis. Note that fb for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.
Cm	=	a moment factor based on the ratio of the end moments and defined as $0.6- 0.4 \times M1/M2 \geq 0.4$. M1 & M2 are the smaller & larger end moments respectively, about the relevant axis.
Fa1	=	allowable compressive stress as per Cl.6.6.1.1
F'e	=	$12 \times \pi^2 \times E\ /\ [23 \times (KL/r)^2]$ Note that F'e will be calculated for the respective axes.
Fa0	=	is the allowable compressive stress from Cl.6.6.1.1 using L = 0.
Fb1	=	maximum bending stress without any lateral buckling as per Cl.6.1 and Cl.6.2 as appropriate)

- Members susceptible to torsional buckling:

  For members that are susceptible to torsional buckling, both the interaction equations as per Cl.6.7.2 (a) will be checked as below:

  1. Sections that have the factor Q = 1.0:

     The following two checks will be performed for all section profiles:

     $$\frac{f_a}{F_{a1}} + \frac{f_{b1} \times C_m}{F_{b1}\left(1 - f_a / F'_e\right)} \leq 1.0 \text{ and}$$

     $$\frac{f_a}{F_{a0}} + \frac{f_{b1}}{F_{b1}} \leq 1.0$$

     where

fa	=	P/ A with A = the full cross section area
fb1	=	maximum applied bending stress about the relevant axis. Note that fb for a section with stiffened compression element will be based on the effective width and the corresponding effective section modulus.
Cm	=	a moment factor based on the ratio of the end moments and defined as $0.6- 0.4 \times M1/M2 \geq 0.4$. M1 & M2 are the smaller & larger end moments respectively, about the relevant axis. Note that the end moments used will be the ones at the ends of the analytical member being designed.
Fa1	=	allowable compressive stress as per Cl.6.6.1.1
F'e	=	$\frac{12\pi^2 E}{23(KL\ /r)^2}$ Note that F'_e will be calculated for the respective axes.
Fa0	=	the allowable compressive stress from Cl.6.6.1.1 using L = 0.
Fb1	=	maximum bending stress without any lateral buckling as per Cl.6.1 and Cl.6.2 as appropriate)

     Since the loads are being taken as being applied through the shear centre, the checks as per Cl.6.7.2 (b), Cl.6.7.2 (c) & Cl.6.7.2 (d) will not be performed.

  2. Sections that have the factor Q < 1.0:

For members with a Q factor less than 1.0, the interaction checks as per Cl.6.7.2 (a) will be performed, but with the term Fy being replaced with $Q \times Fy$.

- Checks for members subject to combined shear and bending:

  Only web elements of sections will be checked for the effects of combined bending & shear forces. The checks will be done as per Cl.6.4.3. The following check will be performed:

$$\sqrt{\left(\frac{f_{bw}}{F_{bw}}\right)^2 + \left(\frac{f_v}{F_v}\right)^2} \leq 1.0$$

where

fbw	=	the applied bending stress at junction of flange & web
Fbw	=	$36560000 / (h/t)^2 \; kgf/cm^2$
fv	=	the applied shear stress
Fv	=	allowable shear stress as per Cl.6.4.1, but not limited to $0.4 \times Fy$.

Related Links

- *V. IS 801-Beam with axial and major axis bending* (on page 4293)
- *V. IS 801-Column with axial and major axis bending* (on page 4301)

D8.C.3 Code Checking and Member Selection

The following two design modes are available:

D8.D.3.1 Code Checking

The program compares the resistance of members with the applied load effects, in accordance with IS:801-1975. Code checking is carried out for locations specified using the `BEAM` parameter. The results are presented in a form of a PASS/FAIL identifier and a `RATIO` of load effect to resistance for each member checked. You may choose the degree of detail in the output data by setting the `TRACK` parameter.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D8.D.3.2 Member Selection

You may request that the program search the cold formed steel shapes database (standard sections) for an alternative section to pass the code check and meet the least weight criterion. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, presents design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D8.C.4 Design Parameters

The following table contains the input parameters for specifying values of design variables and selection of design options.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 167: IS 801: Indian cold formed steel design parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `IS801` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	When this parameter is set to 0) the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (see TR.41 Section Specification (on page 2665) for details) For TRUSS members only start and end locations are designed. 1) the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member.
CB	0	Bending coefficient Cb used for bending checks. By default (value of 0), this is calculated by the program.
CMY	calculated	Coefficient Cm as per Cl. 6.7 for bending about the member Y axis. See IS:801-1975, 6.7. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.
CMZ	calculated	Coefficient Cm as per Cl. 6.7 for bending about the member Z axis. See IS:801-1975, 6.7. Used for Combined axial load and bending design. Values range from 0.4 to 1.0.
CWY	0	Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See IS:801-1975, 6.1.1 0) effect should not be included 1) effect should be included

Parameter Name	Default Value	Description
FLX	0	Specifies whether the member has torsional-flexural buckling restraints that will in turn determine whether the member is susceptible to flexural-torsional bucking mode. See IS:801-1975, 6.6.1 0) No Torsional restraints provided and the section subject to torsional flexural buckling 1) Torsional restraints provided and the section is not subject to torsional flexural buckling
FU	450 MPa (4588.72 kg/cm^2)	Ultimate tensile strength of steel in current units.
FYLD	353.04 MPa (3600.0 kg/cm^2)	Yield strength of steel in current units.
KX	1.0	Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
KY	1.0	Effective length factor for overall buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
KZ	1.0	Effective length factor for overall buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LX	Member length	Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.

Parameter Name	Default Value	Description
LY	Member length	Effective length for overall buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LZ	Member length	Effective length for overall buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a member completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
NSF	1.0	Net section factor for tension members. The net area is calculated as `NSF`×*gross area*.
RATIO	1.0	Permissible ratio of actual to allowable stresses
TRACK	0	This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 0) Prints only the member number, section name, ratio, and PASS/FAIL status. 1) Prints the design summary including allowable stress in addition to that printed by `TRACK 0`. 2) Prints member, material properties, and stress check table in addition to that printed by `TRACK 1`.
TSA	0	Specifies whether transverse web stiffeners have been provided to check for the requirements of IS: 801-1975, 5.2.4. 0) Web stiffeners not provided 1) Web stiffeners provided
UNL	Member length	Unsupported length (in current units) for calculating the allowable bending stress.

D8.C.5 Design Results

Track 2 Output

```
                        STAAD.Pro CODE CHECKING - ( IS:801 )   v3.0
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|------------------------------------------------------------------------------
```

```
|
Most critical result and corresponding forces.|  MEMBER:     1  SECTION:
60CS40X4              LEN:    3.000   LOC:   0.000   |
| STATUS: FAIL    RATIO:   2.148              REF: 6.3 LTB     LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Fx:(C)        0.000       Fy:         3.749         Fz:       0.000
|
|  Mx:           0.000       My:         0.000         Mz:       2.247
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:                                           (Unit:   CM)
|
|  Ag:         5.82000       Az:     3.20000           Ay:     3.36000
|
|  Cz:         1.75000       Cy:     3.00000
|
|  Iz:        28.20000       Iy:    12.30000            J:     0.29600
|
|  Sz:         9.40000       Sy:     5.46000
|
|  Rz:         2.20122       Ry:     1.45375           Cw:   162.00003
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:                                                 (Unit: MPa)
|
|  Fy:   250.018        Fu:   449.993      E: 203404.356      G:  77968.401
|
| Cold-formed strength used for compression and bending checks respectively.
Fya(compression):   325.621             Fya(bending):   387.521            |
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:     3.000      Lz:     3.000 Ly:     3.000 Lb    3.000
|
Input parameters specified. Note Cb of 0 indicates that this value will be
program-calculated.| DESIGN
PARAMETERS:                                                        |
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|  Actual:       0.000      Allowable:       0.000      Ratio:   0.000
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
7
| CHECKS:                            |      Stresses       |
|
```

```
|              | Loc. | Demand |  L/C |  Actual  |  Allow   |Ratio | Ref CL
|
|              |(MET) |(KN-MET)|      | (MPa)    | (MPa)    |      |
|
|--------------|------|--------|------|----------|----------|------|----------
|
| Tension      |  -   |   -    |  -   |    -     | 195.372 |  -   | 6.1
|
| Compression  |  -   |   -    |  -   |    -     |  24.609 |  -   | 6.6.1.1
|
Bending stress checks at the critical compression and tension fibers for
bending about Z.| BendZComp    | 0.000|    2.25|     1|  239.014 |  232.512 |
1.028| 6.3       |
| BendZTens    | 0.000|    2.25|     1|  239.014 |  232.512 | 1.028| 6.3
|
| BendUnbraced | 0.000|    2.25|     1|  239.014 |  111.292 | 2.148| 6.3 LTB
|
| BendYComp    |  -   |   -    |  -   |    -     |  232.512 |  -   | 6.3
|
| BendYTens    |  -   |   -    |  -   |    -     |  232.512 |  -   | 6.3
|
Bending stress checks at the web-flange junction.| Bend Web     | 0.000|
2.25|     1|  207.145 |  232.512 | 0.891| 6.4.2    |
| Shear Z      |  -   |   -    |  -   |    -     |  100.007 |  -   | 6.4.1
|
| Shear Y      | 0.000|    3.75|     1|   11.158 |  100.007 | 0.112| 6.4.1
|
| Axial+Bend   | 0.000|   -    |     1|    -     |     -    | 1.028| 6.7.1(T)
|
| Bend+Shear   | 0.000|   -    |     1|    -     |     -    | 0.025| 6.4.3
|
|-----------------------------------------------------------------------------
|
Effective section area used for compression checks and effective modulii used
for bending checks.| Effective Section Properties:
(cm)                                            |
|  Ae:   5.820 SzTop:   9.400 SzBot:   9.400 SyLeft:   7.029 SyRight:   5.467
|
|-----------------------------------------------------------------------------
|
```

D8.D. Indian Codes - Steel Design per IS 802 - 1995

STAAD.Pro is capable of performing steel design based on the Indian code IS 802 1995 *Use of Structural Steel in Overhead Transmission Line Towers - Code of Practice*.

D8.D.1 General Comments

This section presents some general statements regarding the implementation of Indian Standard code of practice (IS:802-1995 – Part 1) for structural steel design for overhead transmission line towers in STAAD. The design philosophy and procedural logistics for member selection and code checking are based upon the principles of allowable stress design.

Two major failure modes are recognized: failure by overstressing, and failure by stability considerations. The flowing sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economic section is selected on the basis of least weight criteria. The code checking part of the program checks stability and strength requirements and reports the critical loading condition and the governing code criteria.

D8.D.2 Allowable Stresses

The member design and code checking in STAAD.Pro are based upon the allowable stress design method as per IS:802 (1995). It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions.

This section discusses the salient features of the allowable stresses specified by IS:802 and implemented in STAAD.Pro.

8D.D.2.1 Axial Stress

Tensile Stress

The allowable tensile stress, as calculated in STAAD.Pro as per IS:802 is described below.

The estimated tensile stresses on the net effective sectional area in various members, multiplied by the appropriate factor of safety shall not exceed minimum guaranteed yield stress of the material.

Thus, the permissible stress in axial tension, σ_{at} in MPa on the net effective area of the sections shall not exceed

$$\sigma_{at} = F_y$$

where

F_y = minimum yield stress of steel in Mpa

Compressive Stress

The estimated compressive stresses in various members multiplied by the appropriate factor of safety shall not exceed the value given by the formulae described below.

I. Condition: when $(b/t) \le [\,(b/t)_{lim} = 210/\sqrt{F_y}]$:

  i. When $KL/r \le C_c$, the allowable compressive stress is (in N/mm^2)

$$F_a = F_y\{1 - 0.5[(KL/r)/C_c]^2\}$$

  ii. When $KL/r > C_c$, the allowable compressive stress is (in N/mm^2)

$$F_a = \pi^2 E/(KL/r)^2$$

II. Condition: when $(b/t)_{lim} < (b/t) \le 378/\sqrt{F_y}$:

The equations in condition 1 shall be used, substituting for F_y the value F_{cr} given by:

$$F_{cr} = F_y[1.677 - 0.677 \cdot (b/t)/(b/t)_{lim}]$$

III. Condition: when $(b/t) > 378/\sqrt{F_y}$

The equations in condition 1 shall be used, substituting for F_y the value F_{cr} given by:

$$F_{cr} = 65{,}550/(b/t)$$

where

F_a = allowable unit stress in compression, Mpa
F_y = minimum guaranteed yield stress of the material, Mpa
K = restraint factor
L = unbraced length of the compression member, cm

R	=	appropriate radius of gyration, cm
E	=	modulus of elasticity, N/mm^2
KL/r	=	largest effective slenderness ratio of any unbraced segment of the member
b	=	distance from edge of the fillet to the extreme fiber, mm
t	=	thickness of flange, mm

Note: The maximum permissible value of b/t for any type of steel shall not exceed 25.

8D.D.3 Stability Requirements

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. The following are the default values used in STAAD.Pro.

8D.D.3.1 Compression Member

Table 168: Slenderness ratio limits of compression members

Type of Member	Slenderness Limit
Leg Members, ground wire peak member and lower members of cross arms in compression	120
Other members carrying computed stress	200
Redundant members and those carrying nominal stresses	250

Slenderness ratios of compression members are determined as follows:

Table 169: Compression slenderness ratio calculation depending on ELA parameter

ELA Value	Type of Member	Calculation of KL/r
1	Leg sections or joint members bolted at connections in both faces	L/r
2	Members with concentric loading at both ends of the unsupported panel with values of L/r up to and including 120	L/r
3	Member with concentric loading at one end and normal eccentricities at the other end of the unsupported panel for value of L/r up to and including 120	30 + 0.75L/r

ELA Value	Type of Member	Calculation of KL/r
4	Members with normal framing eccentricities at both ends of the unsupported panel for values of L/r up to and including 120	60 + 0.5L/r
5	Member unrestrained against rotation at both ends of the unsupported panel for value of L/r from 120 to 200	L/r
6	Members partially restrained against rotation at one end of the unsupported panel for values of L/r over 120 and up to and including 225	28.6 + 0.762L/r
7	Members partially restrained against rotation at both ends of the unsupported panel for values of L/r over 120 and up to and including 250	46.2 + 0.615L/r

If the value for ELA specified for any particular member results in the L/r ratio to fall outside of the specified range, then STAAD.Pro ignores this input and evaluates the slenderness ratio using the KL/r formula.

8D.D.3.2 Tension Members

Slenderness ratio KL/r of a member carrying axial tension only, shall not exceed 400.

8D.D.4 Minimum Thickness Requirement

As per Clause7.1 of IS: 802-1995 minimum thickness of different tower members shall be as follows:

Members	Minimum Thickness (mm)	
	Galvanized	Painted
Leg Members, ground wire peak member and lower members of cross arms in compression	5	6
Other members	4	5

8D.D.5 Code Checking

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS:802 (1995) requirements. Axial forces at two ends of the members are utilized for the code checking calculations.

Note: Shear and bending forces are *not* considered in design per IS:802 as this code has no provisions for either. If a member design per IS:802 is specified for a member without a TRUSS specification within a model that is not a STAAD TRUSS, then a warning is given in the analysis results file that only axial forces are considered.

The code checking output labels the members as passed or failed. In addition, the critical condition, governing load case, location (distance from the start) and magnitudes of the governing forces are also printed out.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

8D.D.5.1 Design Steps

The following are the steps used by the program in member design:

1. Thickness of the member (maximum of web and flange thicknesses) is checked against minimum allowable thickness, depending upon whether the member is painted or galvanized.
2. If the minimum thickness criterion is fulfilled, the program determines whether the member is under compression or tension for the load case under consideration. Depending upon whether the member is under tension or compression the slenderness ratio of the member is calculated. This calculated ratio is checked against allowable slenderness ratio.
3. If the slenderness criterion is fulfilled check against allowable stress is performed. Allowable axial and tensile stresses are calculated. If the member is under tension and there is no user defined net section factor (NSF), the net section factor is calculated by the program itself (Refer to 8D.D.9 Calculation of Net Section Factor (on page 1660)). Actual axial stress in the member is calculated. The ratio for actual stress to allowable stress, if less than 1.0 or user defined value, the member has passed the check.
4. Number of bolts required for the critical load case is calculated.

8D.D.6 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, that is, the lightest section, which satisfies the applicable code requirements. The section selected will be of the same type (either angle or channel) as originally specified by the user. Member selection may be performed with all angle or channel sections and user-provided tables. Selection of members, whose properties are originally provided from user specified table, will be limited to sections in the user provided table.

The process of MEMBER SELECTION may be controlled using the parameters listed in 8D.D.8 Design Parameters (on page 1658). It may be noted that the parameters DMAX and DMIN may be used to specify member depth constraints for selection. If PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR.49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

8D.D.6.1 Member Selection by Optimization

Steel section selection of the entire structure may be optimized. The optimization method utilizes a state-of-the-art numerical technique which requires automatic multiple analysis. The optimization is based on member stiffness contributions and corresponding force distributions.

An optimum member size is determined through successive analysis/design iterations. This method requires substantial computer time and hence should be used with caution.

Refer to TR.49.2 Member Selection by Optimization (on page 2679) for details.

8D.D.7 Tabulated Results of Steel Design

An example of a `TRACK 2.0` output for a compression member is shown here:

```
                       STAAD.PRO CODE CHECKING - (         IS-802)   v1.0
                       ********************************************
 |--------------------------------------------------------------------------|
 |                                                     Y        PROPERTIES  |
 |*************                                        |        IN  CM  UNIT |
 |            *  |============================|   ==| |==      ------------ |
 |MEMBER   8  *  |  INDIAN SECTIONS           |     | |        AX =   17.0  |
 |            *  | ST  ISA125x95x8            |     | |   --Z  AY =    6.7  |
 |DESIGN CODE *  |                            |     | |        AZ =    5.1  |
 |  IS-802    *  ===============================  ==| |==      SY =   38.8  |
 |            *                                                SZ =   16.6  |
 |            *  |<---LENGTH (ME=     1.80 --->|               RY =    4.4  |
 |*************                                                RZ =    2.0  |
 |                                                                          |
 |                                                                          |
 |PARAMETER                   BOLTING                          STRESSES     |
 |IN   NEWT  MM                                                IN   NEWT  MM|
 |---------------             -------------                    -------------|
 | L/R-Y =   40.5             BOLT DIA = 12 MM                 FA  =  188.4 |
 | L/R-Z =   87.9             BOLT CAP =  24.66 KN             fa  =   80.7 |
 | KL/R  =   87.9             #  BOLT  =  6                    FYB =  436.0 |
 | FYLD  =  250.0                                              FVB =  218.0 |
 | GALVA =    0.0                                                           |
 | C     =    1.0                                                           |
 | LEG   =    1.0                                                           |
 | ELA   =    1.0                                                           |
 | NSF   =    1.0                                                           |
 |                                                                          |
 |**************************************************************************|
 |*                                                                        *|
 |*                         DESIGN SUMMARY ( KN-METR)                      *|
 |*                         --------------                                 *|
 |*                                                                        *|
 |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/           *|
 |         FX               MY             MZ            LOCATION            |
 |        ======================================================            |
 |          PASS        COMPRESSION      0.428              1               |
 |        137.13 C          0.0          0.0              0.00              |
 |*                                                                        *|
 |**************************************************************************|
 |                                                                          |
 |--------------------------------------------------------------------------|
```

8D.D.8 Design Parameters

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 170: Indian Steel Design IS 802 Parameters

Parameter Name	Default Value	Description
CNSF	0.0	This parameter indicates whether user has defined the net section factor or the program will calculate it. 0) Use specified NSF value 1) Net section factor will be calculated.
DANGLE	0.0	This parameter indicates how the pair of angles are connected to each other. This is required to find whether the angle is in single or double shear and the net section factor. 0) Double angle placed back-to-back and connected to each side of a gusset plate 1) Pair of angle placed back-to-back connected by only one leg of each angle to the same side of a gusset plate
DBL	12 mm	Diameter of bolt for calculation of number of bolts and net section factor.
DMAX	100.0 cm.	Maximum allowable depth.
DMIN	0.0 cm.	Minimum allowable depth.
ELA	1.0	This parameter indicates what type of end conditions is to be used. Refer to 8D.D.3 Stability Requirements (on page 1654).
FVB	218 MPA	Allowable shear stress in bolt
FYB	436 MPA	Allowable bearing stress in bolt

Parameter Name	Default Value	Description
FYLD	250 MPA	Yield Strength of steel
GUSSET	5 mm	Thickness of gusset plate. Minimum of the thicknesses of the gusset plate and the leg is used for calculation of the capacity of bolt in bearing
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LEG	1.0	This parameter is meant for plain angles. 0) The angle is connected by shorter leg 1) The angle is connected by longer leg
LY	Member Length	Unbraced length in local z-axis to calculate slenderness ratio.
LZ	Member Length	Unbraced length in local z-axis to calculate slenderness ratio.

Parameter Name	Default Value	Description
MAIN	1.0	Type of member to find allowable Kl/r for slenderness calculations for members. 1) Leg, Ground wire peak and lower members of cross arms in compression (KL/r = 120) 2) Members carrying computed stress (KL/r = 200) 3) Redundant members and members carrying nominal stresses (KL/r = 250) 4) Tension members (KL/r = 400) 10) Do not perform KL/r check Any value greater than 10.0 indicates user defined allowable KL/r ratio. For this case KY and KZ values are must to find actual KL/r ratio of the member.
NSF	1.0	Net section factor for tension members
NHL	0.0 mm	Deduction for holes. Default value is one bolt width plus 1.5 mm. If the area of holes cut by any straight, diagonal or zigzag line across the member is different from the default value, this parameter is to be defined.
TRACK	0.0	Level of output detail: 0) Suppress critical member stresses 1) Print all critical member stresses 2) Print expanded output. 9) Print design calculations along with expanded output (not available in GUI input).

8D.D.9 Calculation of Net Section Factor

The procedure for calculating the net section factor for an angle section is as follows:

- For a channel section, net section factor is taken to be 1.0.
- For an angle section, it is the ratio of the net effective area, A_{net}, to the gross area, where:

a. Single angle connected by only one leg

$$A_{net} = A_1 + A_2 \times K_1$$

where

A_1	=	net cross-sectional area of the connected leg
A_2	=	gross cross-sectional area of the unconnected leg
K_1	=	$3 \cdot A_1 / (3 \cdot A_1 + A_2)$

The area of a leg of an angle = Thickness of angle × (length of leg – 0.5× thickness of leg)

b. Pair of angles placed back-to-back connected by only one leg of each angle to the same side of a gusset plate

$$A_{net} = A_1 + A_2 \cdot K_1$$

where

A_1	=	net cross-sectional area of the connected leg
A_2	=	gross cross-sectional area of the unconnected leg
K_1	=	$5 \cdot A_1 / (5 \cdot A_1 + A_2)$

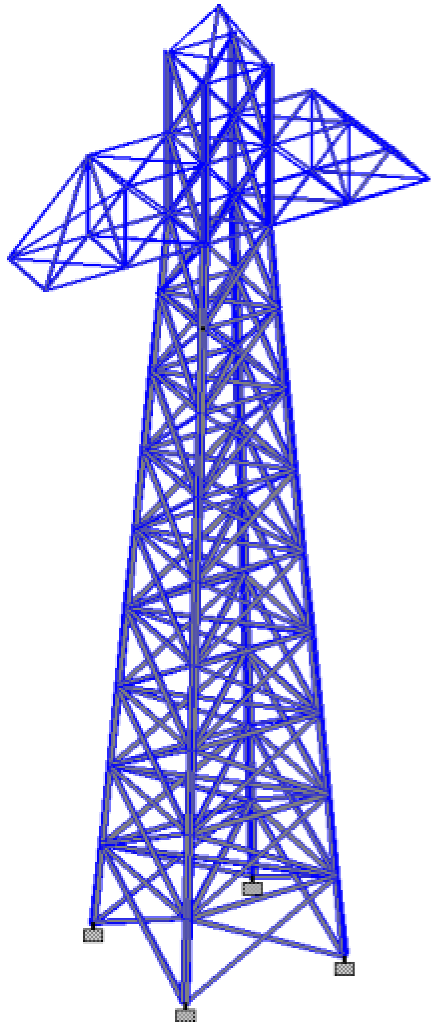
The area of a leg of an angle = Thickness of angle × (length of leg – 0.5× thickness of leg)

c. Double angles placed back-to-back and connected to each side of a gusset plate

A_{net} = gross area minus the deduction for holes

8D.D.10 Example Problem

A transmission line tower is subjected to different loading conditions. Design some members as per IS-802 and show detailed calculation steps for the critical loading condition.



Given

The members are all considered as axial-only members (i.e., truss members).

Note: The model is specified as a `STAAD TRUSS`, which indicates that members have no bending. Refer to G.2 Types of Structures (on page 2085) for details.

End Condition = Members with normal framing eccentricities at both ends of the unsupported panel for values of L/r up to and including 120

Diameter of the bolt = 16 mm

Thickness of the gusset plate = 8 mm

Net Section Factor is to be calculated.

STAAD Input File

Note: This example uses the command `STAAD TRUSS`. If this model had used `STAAD SPACE` (i.e., a space frame rather than a truss model), a warning would be presented since the IS 802 code does not have provisions for non-truss members.

```
STAAD TRUSS
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1  3 0 3; 2 1.2 27 1.2; 3 2.8 3 2.8; 4 2.6 6 2.6; 5 2.4 9 2.4;
6 2.2 12 2.2; 7  2 15 2; 8 1.8 18 1.8; 9 1.6 21 1.6; 10 1.4 24 1.4;
11 -3 0 3; 12 -1.2  27 1.2; 13  -2.8 3 2.8; 14 -2.6 6 2.6; 15 -2.4 9 2.4;
16 -2.2 12 2.2; 17 -2 15 2; 18  -1.8 18 1.8; 19 -1.6 21 1.6; 20 -1.4 24 1.4;
21 3 0 -3; 22 1.2 27 -1.2; 23  2.8 3 -2.8; 24 2.6 6 -2.6; 25 2.4 9 -2.4;
26 2.2 12 -2.2; 27 2 15 -2; 28  1.8 18 -1.8; 29 1.6 21 -1.6; 30 1.4 24 -1.4;
31 -3 0 -3; 32 -1.2 27 -1.2; 33  -2.8 3 -2.8; 34 -2.6 6 -2.6; 35 -2.4 9 -2.4;
36 -2.2 12 -2.2; 37 -2 15  -2; 38  -1.8 18 -1.8; 39 -1.6 21 -1.6;
40 -1.4 24 -1.4; 41 1.2 30 1.2; 42  -1.2 30 1.2; 43 1.2 30 -1.2;
44 -1.2 30 -1.2; 45 4.2 27 1.2; 46 7.2 27  1.2; 47  4.2 30 1.2; 48 4.2 27 -1.2;
49 7.2 27 -1.2; 50 4.2 30 -1.2; 51 -4.2 27  1.2; 52  -7.2 27 1.2;
53 -4.2 30 1.2; 54 -4.2 27 -1.2; 55 -7.2 27 -1.2; 56 -4.2 30 -1.2;
57 1.2 33 1.2; 58 -1.2 33 1.2; 59 1.2 33 -1.2; 60  -1.2 33 -1.2; 61 0 35 0;
MEMBER  INCIDENCES
1 1 3; 2 3 4; 3 4 5; 4 5 6; 5 6 7; 6 7 8; 7 8 9; 8 9 10; 9 10 2; 10 11 13;
11 13 14; 12 14 15; 13 15 16; 14 16 17; 15 17 18; 16 18 19; 17 19 20; 18 20 12;
19 13 3; 20 14 4; 21 15 5; 22 16 6; 23 17 7; 24 18 8; 25 19 9; 26 20 10;
27 12 2; 28 11 3; 29 1 13; 30 13 4; 31 3 14; 32 14 5; 33 15 4; 34 15 6;
35 16 5; 36 16 7; 37 17 6; 38 17 8; 39 18 7; 40 18 9; 41 19 8; 42 19 10;
43 20 9; 44 20 2; 45 12 10; 46 21 23; 47 23 24; 48 24 25; 49 25 26; 50 26  27;
51  27 28; 52 28 29; 53 29 30; 54 30 22; 55 3 23; 56 4 24; 57 5 25; 58 6 26;
59 7 27; 60 8 28; 61 9 29; 62 10 30; 63 2 22; 64 1 23; 65 21 3; 66 3 24;
67 23 4; 68 4 25; 69 5 24; 70 5 26; 71 6 25; 72 6 27; 73 7 26; 74 7 28;
75 8 27; 76 8 29; 77 9 28; 78 9 30; 79 10 29; 80 10 22; 81 2 30; 82 31 33;
83 33 34; 84 34 35; 85 35 36; 86 36 37; 87 37 38; 88 38 39; 89 39 40; 90 40 32;
91 23 33; 92 24 34; 93 25 35; 94 26 36; 95 27 37; 96 28 38; 97 29 39; 98 30 40;
99 22 32; 100 21 33; 101 31 23; 102 23 34; 103 33 24; 104 24 35; 105 25 34;
106 25 36; 107 26 35; 108 26 37; 109 27 36; 110 27 38; 111 28 37; 112 28 39;
113 29 38; 114 29 40; 115 30 39; 116 30 32; 117 22 40; 118 33 13; 119 34 14;
120 35 15; 121 36 16; 122 37 17; 123 38 18; 124 39 19; 125 40 20; 126 32 12;
127 31 13; 128 11 33; 129 33 14; 130 13 34; 131 34 15; 132 35 14; 133 35 16;
134 36 15; 135 36 17; 136 37 16; 137 37 18; 138 38 17; 139 38 19; 140 39 18;
141 39 20; 142 40 19; 143 40 12; 144 32 20; 145 32 44; 146 12 42; 147 2 41;
148 22 43; 149 42 41; 150 41 43; 151 43 44; 152 44 42; 153 12 41; 154 42 2;
155 22 41; 156 43 2; 157 43 32; 158 44 22; 159 12 44; 160 32 42; 161 41 47;
162 47 45; 163 45 2; 164 47 46; 165 46 45; 166 41 45; 167 43 50; 168 50 48;
169 48 22; 170 50 49; 171 49 48; 172 43 48; 173 47 50; 174 46 49; 175 45 48;
176 41 50; 177 50 46; 178 43 47; 179 47 49; 180 22 50; 181 2 47; 182 22 45;
183 2 48; 184 47 48; 185 50 45; 186 45 49; 187 48 46; 188 42 53; 189 53 51;
190 51 12; 191 53 52; 192 52 51; 193 42 51; 194 44 56; 195 56 54; 196 54 32;
197 56 55; 198 55 54; 199 44 54; 200 53 56; 201 52 55; 202 51 54; 203 42 56;
204 56 52; 205 44 53; 206 53 55; 207 32 56; 208 12 53; 209 32 51; 210 12 54;
211 53 54; 212 56 51; 213 51 55; 214 54 52; 215 44 60; 216 42 58; 217 41 57;
218 43 59; 219 60 59; 220 59 57; 221 57 58; 222 58 60; 223 44 58; 224 42 60;
225 42 57; 226 41 58; 227 44 59; 228 43 60; 229 43 57; 230 41 59; 231 60 57;
232 59 58; 235 33 3; 236 13 23; 237 34 4; 238 14 24; 239 35 5; 240 15 25;
241 36 6; 242 16 26; 243 37 7; 244 17 27; 245 38 8; 246 18 28; 247 39 9;
248 19 29; 249 40 10; 250 20 30; 251 32 2; 252 22 12; 253 44 41; 254 43 42;
```

```
255 60 61; 256 58 61; 257 57 61; 258 59 61;
MEMBER PROPERTY INDIAN
1 TO 18 46 TO 54 82 TO 90 145 TO 148 215 TO 218 TA LD ISA200X150X18 SP 0.01
19 TO 26 28 TO 45 55 TO 62 64 TO 81 91 TO 98 100 TO 125 127 TO 144 155 156 -
159 160 223 224 229 230 235 TO 250 TA ST ISA150X150X10
27 63 99 126 149 TO 154 157 158 161 TO 214 219 TO 222 225 TO 228 231 232 251 -
252 TO 258 TA ST ISA80X50X6
CONSTANTS
E 2.05e+008 ALL
POISSON 0.3 ALL
DENSITY 76.8195 ALL
ALPHA 6.5e-006 ALL
SUPPORTS
1 11 21 31 FIXED
UNIT METER KG
LOAD 1 VERT
SELFWEIGHT Y -1
JOINT LOAD
61 FX 732
46 49 52 55 FX 153
61 FX 1280 FY -1016 FZ 160
46 49 52 55 FX 9006 FY -7844 FZ 1968
2 12 22 32 FX 4503 FY -3937 FZ 1968
LOAD 2 GWBC
SELFWEIGHT Y -1
JOINT LOAD
61 FX 549
46 49 52 55 FX 1148
61 FX 515 FY -762 FZ 2342
46 49 52 55 FX 6755 FY -5906
2 12 22 32 FX 3378 FY -2953
LOAD 3 LEFT PCBC
SELFWEIGHT Y -1
JOINT LOAD
61 FX 549
46 49 52 55 FX 1148
61 FX 960 FY -762
46 49 FX 6755 FY -5906
52 55 FX 4211 FY -4551 FZ 13293
2 12 22 32 FX 3378 FY -2953
LOAD 4 RIGHT PCBC
SELFWEIGHT Y -1
JOINT LOAD
61 FX 549
46 49 52 55 FX 1148
61 FX 960 FY -762
52 55 FX 6755 FY -5906
46 49 FX 4211 FY -4551 FZ 13293
2 12 22 32 FX 3378 FY -2953
PERFORM ANALYSIS
UNIT NEW MMS
PARAMETER
CODE IS802
LY 2800 MEMB 28
LZ 2800 MEMB 28
MAIN 1.0 MEMB 1
ELA 4 MEMB 1
CNSF 1.0 MEMB 28
```

```
DBL 16 ALL
GUSSET 8 ALL
TRACK 2 ALL
CHECK CODE MEMB 1 28
FINISH
```

Output

A portion of the output for the TRACK 2 member code check follows:

```
                       STAAD.PRO CODE CHECKING - (          IS-802)   v1.0
                       ********************************************


  |--------------------------------------------------------------------------|
  |                                                   Y        PROPERTIES    |
  |*************                                      |        IN  CM  UNIT  |
  |            *  |=============================|   ==|==      ------------  |
  |MEMBER   1  *  |  INDIAN SECTIONS            |     ||       AX =  120.0   |
  |            *  | LD  ISA200X150X18           |     ||   --Z AY =   48.0   |
  |DESIGN CODE *  |                             |     ||       AZ =   36.0   |
  |  IS-802    *  |-----------------------------|     ||       SY =  297.3   |
  |            *                                               SZ =  350.6   |
  |            *  |<---LENGTH (ME=     3.01 --->|              RY =    6.2   |
  |*************                                               RZ =    6.3   |
  |                                                                          |
  |                                                                          |
  |PARAMETER                  BOLTING                          STRESSES      |
  |IN   NEWT  MM                                               IN   NEWT  MM|
  |---------------            -------------                    -------------|
  | L/R-Y =   48.6            BOLT DIA = 16 MM                 FA  =  195.1  |
  | L/R-Z =   47.7            BOLT CAP =  55.81 KN             fa  =  145.2  |
  | KL/R  =   84.3            #  BOLT  = 32                    FYB =  436.0  |
  | FYLD  =  250.0                                             FVB =  218.0  |
  | GALVA =    0.0                                                           |
  | C     =    1.0                                                           |
  | LEG   =    1.0                                                           |
  | ELA   =    4.0                                                           |
  | NSF   =    1.0                                                           |
  |                                                                          |
  |*************************************************************************|
  |*                                                                        *|
  |*                         DESIGN SUMMARY ( KN-METR)                      *|
  |*                         --------------                                 *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/           *|
  |         FX               MY             MZ          LOCATION             |
  |        ===================================================              |
  |          PASS        COMPRESSION      0.744              1               |
  |       1742.26 C           0.0         0.0             0.00               |
  |*                                                                        *|
  |*************************************************************************|
  |                                                                          |
  |--------------------------------------------------------------------------|
   STAAD TRUSS                                                -- PAGE NO.    5

  DETAILS OF CALCULATION
  ----------------------
```

```
CHECK FOR MINIMUM THICKNESS
---------------------------

TYPE : PAINTED

MIN. ALLOWABLE THICKNESS : 6.0 MM

ACTUAL THICKNESS : 18.0 MM

RESULT : PASS

CHECK FOR SLENDERNESS RATIO
---------------------------

VALUE OF L/r :  48.63

EQN. USED TO FIND KL/r : 60.0 + 0.5*L/r

ACTUAL VALUE OF KL/r :  84.31

ALLOWABLE KL/r : 120.00

RESULT : PASS

CALCULATION OF ALLOWABLE STRESS
---------------------------------

CRITICAL CONDITION : COMPRESSION

Cc : sqrt(2*3.14159265*3.14159265*E :  127.24

b : LENGTH OF LEG - WEB THICKNESS - ROOT RADIUS
  : 200.0 - 18.0 -  15.0 : 167.0 MM

(b/t)lim : 210/sqrt(fy)     : 13.28

(b/t)cal :  9.28

(b/t)cal <= (b/t)lim AND KL/r <= Cc

ALLOWABLE AXIAL COMP. STRESS : (1-0.5*(KL/r/Cc)*(KL/r/Cc))*fy   :  195.07 MPA

CHECK AGAINST PERMISSIBLE STRESS
--------------------------------

LOAD NO. :      1

DESIGN AXIAL FORCE : 1742259.75 N

ACTUAL AXIAL COMP. STRESS :1742259.75 / 12000.0 :  145.19 MPA

RESULT : PASS
 STAAD TRUSS                                             -- PAGE NO.    6


BOLTING
-------
```

```
BOLT DIA : 16 MM

SHEARING CAP :  87.66 KN

BEARING CAP :  55.81 KN

BOLT CAP :  55.81 KN

NO. OF BOLTS REQD. :  32
 STAAD TRUSS                                              -- PAGE NO.    7


                    STAAD.PRO CODE CHECKING - (          IS-802)   v1.0
                    ********************************************


|--------------------------------------------------------------------------|
|                                                       Y        PROPERTIES |
|*************                                          |        IN  CM  UNIT |
|            *  |=============================|   ==| |==     ------------ |
|MEMBER  28  *  |  INDIAN SECTIONS            |     | |       AX =   29.2 |
|            *  | ST  ISA150X150X10           |     | |  --Z  AY =   10.0 |
|DESIGN CODE *  |                             |     | |       AZ =   10.0 |
|  IS-802    *  ===============================   ==| |==     SY =   95.7 |
|            *                                                 SZ =   44.8 |
|            *  |<---LENGTH (ME=     6.53 --->|                 RY =    5.9 |
|*************                                                 RZ =    3.0 |
|                                                                          |
|                                                                          |
|PARAMETER                   BOLTING                           STRESSES    |
|IN   NEWT  MM                                                 IN   NEWT  MM|
|---------------             -------------                     -------------|
| L/R-Y =   47.5             BOLT DIA = 16 MM                  FA  =  249.9 |
| L/R-Z =   94.0             BOLT CAP =  43.83 KN              fa  =   48.5 |
| KL/R  =   94.0             #  BOLT  =  3                     FYB =  436.0 |
| FYLD  =  250.0                                               FVB =  218.0 |
| GALVA =    0.0                                                            |
| C     =    1.0                                                            |
| LEG   =    1.0                                                            |
| ELA   =    1.0                                                            |
| NSF   =    0.8                                                            |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                         DESIGN SUMMARY ( KN-METR)                      *|
|*                         --------------                                 *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX                MY           MZ          LOCATION             |
|        =====================================================            |
|          PASS        TENSION          0.194            3                |
|        112.86 T           0.0          0.0           6.53               |
|*                                                                        *|
|**************************************************************************|
|                                                                          |
|--------------------------------------------------------------------------|
 STAAD TRUSS                                              -- PAGE NO.    8
```

```
DETAILS OF CALCULATION
----------------------

CHECK FOR MINIMUM THICKNESS
---------------------------

TYPE : PAINTED

MIN. ALLOWABLE THICKNESS : 6.0 MM

ACTUAL THICKNESS : 10.0 MM

RESULT : PASS

CHECK FOR SLENDERNESS RATIO
---------------------------

VALUE OF L/r :  93.96

EQN. USED TO FIND KL/r : K*L/r

ACTUAL VALUE OF KL/r :  93.96

ALLOWABLE KL/r : 400.00

RESULT : PASS

CALCULATION OF ALLOWABLE STRESS
---------------------------------

CRITICAL CONDITION : TENSION

ALLOWABLE AXIAL TENSILE STRESS :  249.94 MPA

CHECK AGAINST PERMISSIBLE STRESS
--------------------------------

LOAD NO. :      3

DESIGN AXIAL FORCE :  112855.91 N

ACTUAL AXIAL TENSILE STRESS : 112855.91 / ( 2920.0*0.797 ) :   48.51 MPA

RESULT : PASS

BOLTING
-------

BOLT DIA : 16 MM

SHEARING CAP :  43.83 KN

BEARING CAP :  55.81 KN

BOLT CAP :  43.83 KN

NO. OF BOLTS REQD. :   3
```

Note: Were you to perform an analysis with the structure type changed to `STAAD SPACE`, then the following warning would be given for each member design:

```
WARNING : THE MEMBER      1 IS NOT A TRUSS MEMBER.
          THIS MEMBER HAS BEEN DESIGNED CONSIDERING AXIAL FORCE ONLY
```

D8.E. Indian Codes - Concrete Design per IS 456

STAAD.Pro is capable of performing concrete design based on the Indian code IS 456 2000 *Code of Practice for Plain and Reinforced Concrete*.

Related Links

- *D8.E. Indian Codes - Concrete Design per IS 456* (on page 1669)

D8.E.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams — Prismatic (Rectangular & Square), T-Beams, and L-shapes
- For Columns — Prismatic (Rectangular, Square, and Circular)

D8.E.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the `MEMBER PROPERTY` command. The following example shows the required input:

```
UNIT  MM
MEMBER  PROPERTY
1 3  TO 7 9 PRISM YD 450. ZD 250.
11 13  PR YD 350.
14 TO  16 PRIS YD 400. ZD 750. YB 300. ZB 200.
```

In the above input, the first set of members are rectangular (450 mm depth and 250mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 350 mm diameter. The third set numbers in the above example represents a T-shape with 750 mm flange width, 200 width, 400 mm overall depth, and 100 mm flange depth (See section 6.20.2).

Note: The program will determine whether the section is rectangular, flanged, or circular and the beam or column design.

D8.E.3 Design Parameters

The program contains a number of parameters which are needed to perform design as per IS:456 2000. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. The following table contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as millimeter and Newton before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 171: Indian Concrete Design IS456 2000 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `INDIAN`. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BRACING	0.0	Column bracing condition: correspond to the terms “Braced”and “Unbraced” described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000. 0) The column is braced about both axes. 1) The column is unbraced about major axis. 2) The column is unbraced about minor axis. 3) The column is unbraced about both axis.
CLB	25 mm	The clear cover for the outermost bottom reinforcement in beams. **Tip:** If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, beams with 25 mm clear and 8 mm ties should use `CLB 33`.
CLEAR	30 mm (beams) 40 mm (columns)	The clear distance between to the main member reinforcement. **Tip:** If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, columns with 35 mm clear and 8 mm ties should use `CLEAR 43`.

Parameter Name	Default Value	Description
CLT	25 mm	The clear cover for the outermost top reinforcement in beams. **Tip:** If you want to specify the clear distance to ties or stirrups, include the full diameter of those in this value. For example, beams with 25 mm clear and 8 mm ties should use `CLT 33`.
DEPTH	YD	Total depth to be used for design. This value defaults to `YD` as provided under `MEMBER PROPERTIES`.
EFACE	0.0	Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member. **Note:** Both `SFACE` and `EFACE` are input as positive numbers.
ELZ	1.0	Ratio of effective length to actual length of column about major axis. See Note b below.
ELY	1.0	Ratio of effective length to actual length of column about minor axis. See Note b below.

Parameter Name	Default Value	Description
ENSH	0.0	Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000. • ENSH = 1.0 means ordinary shear check to be performed (no enhancement of shear strength at sections close to support) • For ENSH = a positive value(say x), shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note) • For ENSH = a negative value(say –y), shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts.(Refer note) If default value (0.0) is used the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (<2d) close to support otherwise ordinary shear check will be performed.
FC	30 N/mm^2	Concrete Yield Stress.
FYMAIN	415 N/mm^2	Yield Stress for main reinforcing steel.
FYSEC	415 N/mm^2	Yield Stress for secondary reinforcing steel.
MINMAIN	10 mm	Minimum main reinforcement bar size.
MAXMAIN	60 mm	Maximum main reinforcement bar size.
MINSEC	8 mm	Minimum secondary reinforcement bar size.

Parameter Name	Default Value	Description
MAXSEC	12 mm	Maximum secondary reinforcement bar size.
METHOD	0	Consider minimum eccentricity one axis at a time: 0) Do not consider 1) Consider as per Cl 25.4 of IS 456:2000
MFACE	0	Design beam for flexure at any point along the length of the beam as specified by SFACE and EFACE parameters. 0) (Off) Do not design at sections 1) (On) Design at specified sections from start and end of the members assigned. **Note:** If SFACE and EFACE sections are *not* previously defined, then the output will present a warning and no flexure design will be performed (i.e., the MFACE parameter is ignored).
RATIO	4.0	Maximum percentage of longitudinal reinforcement in columns.
REINF	0.0	Tied column. A value of 1.0 will mean spiral reinforcement.
RENSH	0.0	Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts. (Refer note)

Parameter Name	Default Value	Description
RFACE	4	2) Two faced distribution about major axis. 3) Two faced distribution about minor axis. 4) Longitudinal reinforcement in column is arranged equally along 4 faces.
SFACE	0.0	Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.
SPSMAIN	25 mm	Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm.
TORSION	0.0	0) torsion to be considered in beam design. 1) torsion to be neglected in beam design.

Parameter Name	Default Value	Description
TRACK	0	Beam Design: 0) output consists of reinforcement details at START, MIDDLE, and END. 1) critical moments are printed in addition to `TRACK 0.0` output. 2) required steel for intermediate sections defined by `NSECTION` are printed in addition to `TRACK 1.0` output. Column Design: 0) reinforcement details are printed. 1) column interaction analysis results are printed in addition to `TRACK 0.0` output. 2) a schematic interaction diagram and intermediate interaction values are printed in addition to `TRACK 1.0` output.
ULY	1.0	Ratio of unsupported length to actual length of column about minor axis. See Note c below.
ULZ	1.0	Ratio of unsupported length to actual length of column about major axis. See Note c below.
WIDTH	ZD	Width to be used for design. This value defaults to `ZD` as provided under `MEMBER PROPERTIES`.

Notes

a. You may specify reinforcing bar combinations through the `BAR COMBINATION` command. Refer to D8.E.7 Bar Combination (on page 1681) for details.

b. `ELY` and `ELZ` parameters are used to calculate effective length of column to find whether it is a short or long column. Please refer CL 25.1.2 of IS456:2000.

In CL 25.1.2 of IS456:2000, you will find two term, l_{ex} and l_{ey}, which STAAD.Pro calculates as:

- l_{ex} = `ELZ` multiplied by the member length (distance between the two nodes of the member)
- l_{ey} = `ELY` multiplied by the member length (distance between the two nodes of the member)

For the term "D" in CL 25.1.2 of IS456:2000, STAAD.Pro uses the `YD` dimension of the column.

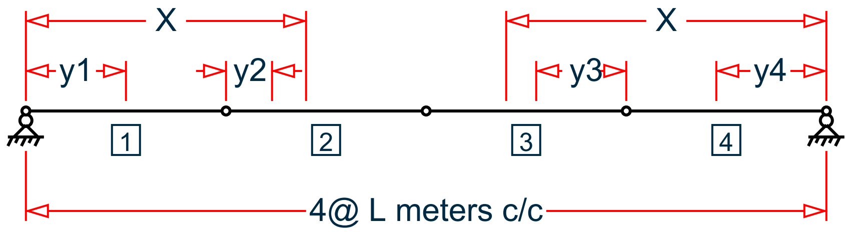
For the term "b" in CL 25.1.2 of IS456:2000, STAAD.Pro uses the `ZD` dimension of the column.

c. `ULY` and `ULZ` parameters are used to calculate unsupported length of column to find minimum eccentricity. Please refer CL 25.4 of IS456:2000.

In CL 25.4 of IS456:2000, you will find an expression "unsupported length of column". This term is calculated as

- `ULZ` multiplied by the member length for the Z axis
- `ULY` multiplied by the member length for the Y axis

d. The value of the `ENSH` parameter (other than 0.0 and 1.0) is used only when the span of a beam is subdivided into two or more parts. When this condition occurs, the `RENSH` parameter is also to be used.



The span of the beam is subdivided four parts, each of length L meter. The shear strength will be enhanced up to X meter from both supports. The input should be the following:

Steps:

a. `ENSH L MEMB 1 =>` Shear strength will be enhanced throughout the length of the member 1, positive sign indicates length measured from start of the member
b. `ENSH (X-L) MEMB 2` => Shear strength will be enhanced up to a length (X-L) of the member 2, length measured from the start of the member
c. `ENSH –L MEMB 4` => Shear strength will be enhanced throughout the length of the member 4, negative sign indicates length measured from end of the member
d. `ENSH –(X-L) MEMB 3` => Shear strength will be enhanced up to a length (X-L) of the member 3, length measured from the end of the member
e. `RENSH L MEMB 2 3` => Nearest support lies at a distance L from both the members 2 and 3.
f. `DESIGN BEAM 1 TO 4=>` This will enhance the shear strength up to length X from both ends of the beam consisting of members 1 to 4 and gives spacing accordingly.

At section = y1 from start of member 1 av = y1

At section = y2 from the start of member 2 av = y2+L

At section = y3 from the end of member 3 av = y3+L

At section = y4 from end of member 4 av = y4

where tc, enhanced = 2dtc/av

At section 0.0, av becomes zero. Thus enhanced shear strength will become infinity. However for any section shear stress cannot exceed tc, max. Hence enhanced shear strength is limited to a maximum value of tc, max.

D8.E.4 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The IS:456 code specifies two options by which the slenderness effect can be accommodated (Clause 39.7). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed end moments, the effect of deflections on moment and forces and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD.Pro uses of the first option. To perform this type of analysis, use the command `PDELTA ANALYSIS` instead of `PERFORM ANALYSIS`. The P-Delta analysis will accommodate all requirements of the second- order analysis described by IS:456, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, additional moments are calculated based on empirical formula and assumptions on sidesway (Clause 39.7.1 and 39.7.1.1,IS: 456 - 2000). The rules of Clause 39.7.1 have been implemented in STAAD.Pro. They will be checked if the `ELY` and `ELZ` parameters are specified.

Considering all these information, a P-Delta analysis, as performed by STAAD.Pro may be used for the design of concrete members.

Note: To take advantage of this analysis, all the combinations of loading must be provided as primary load cases and <u>not</u> as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments (i.e., analysis results), whereas a primary load case is revised during the P-delta analysis based on the deflections. Loads can be combined prior to analysis using the `REPEAT LOAD` command.

Note: You must specify the appropriate load factors (e.g., 1.5 for dead load, etc.) as STAAD.Pro does not factor the loads automatically.

D8.E.5 Beam Design

Beams are designed for flexure, shear and torsion. If required the effect the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The design is performed at 13 evenly spaced points along the length of the beam, including start and end points (i.e., 1/12th points or at ends of 12 equal length segments). All of these sections are scanned to determine the design force envelopes.

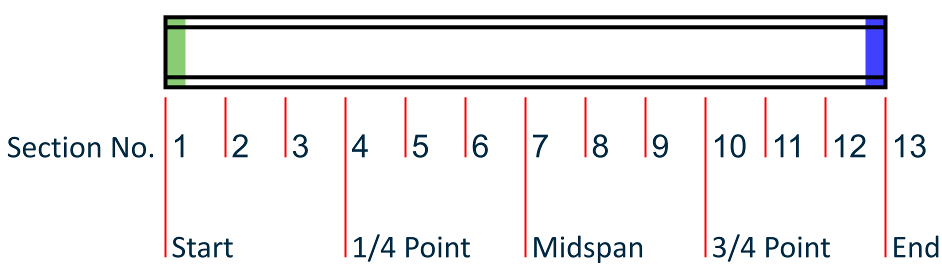


Figure 173: The sections used for design along the length of a member

D8.E.5.1 Design for Flexure

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections is designed to resist both of these critical sagging and hogging moments. Where ever the rectangular section is inadequate as singly reinforced section, doubly reinforced section is tried. However, presently the flanged section is designed only as singly reinforced section under sagging moment. It may also be noted all flanged sections are automatically designed as rectangular section under hogging moment as the flange of the beam is ineffective under hogging moment. Flexural design of beams is performed in two passes. In the first pass, effective depths of the sections are determined with the assumption of single layer of assumed reinforcement and reinforcement requirements are calculated. After the preliminary design, reinforcing bars are chosen from the internal database in single or multiple layers. The entire flexure design is performed again in a second pass taking into account of the changed effective depths of sections calculated on the basis of reinforcement provide after the preliminary design. Final provisions of flexural reinforcements are made then. Efforts have been made to meet the guideline for the curtailment of reinforcements as per IS:456-2000 (Clause 26.2.3). Although exact curtailment lengths are not mentioned explicitly in the design output (finally which will be more or less guided by the detailer taking into account of other practical consideration), user has the choice of printing reinforcements provided by STAAD.Pro at 11 equally spaced sections from which the final detail drawing can be prepared.

Once you have specified `SFACE` and `EFACE` parameters to indicate sections, the `MFACE` parameter can be used to design for flexure at any point along the length of the beam, in addition to the equally spaced sections normally used.

D8.E.5.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear design are performed at 11 equally spaced sections (0.0 to 1.0) for the maximum shear forces amongst the active load cases and the associated torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by the program. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

As per Clause 40.5 of IS:456-2000 shear strength of sections (< 2d where d is the effective depth) close to support has been enhanced, subjected to a maximum value of τ_{cmax}.

D8.E.5.3 Beam Design Output

The default design output of the beam contains flexural and shear reinforcement provided at 5 equally spaced (0, .25, .5, .75 and 1.) sections along the length of the beam. User has option to get a more detail output. All beam design outputs are given in IS units. An example of rectangular beam design output with `TRACK 2.0` output is presented below:

```
                  B E A M  N O.      1   D E S I G N  R E S U L T S

         M20                  Fe415 (Main)            Fe250 (Sec.)

       LENGTH:  6400.0 mm      SIZE:   300.0 mm X  400.0 mm   COVER: 25.0 mm


                          DESIGN LOAD SUMMARY (KN MET)
   ----------------------------------------------------------------------------
   SECTION |FLEXURE  (Maxm. Sagging/Hogging moments)|          SHEAR
   (in mm) |     P        MZ        MX   Load Case  |    VY       MX  Load Case
   ----------------------------------------------------------------------------
       0.0 |     0.00      0.00      0.00     1     |   60.61     0.00      1
```

```
                |    0.00      0.00      0.00     1      |
        533.3   |    0.00     29.63      0.00     1      |   50.51      0.00       1
                |    0.00      0.00      0.00     1      |
       1066.7   |    0.00     53.88      0.00     1      |   40.41      0.00       1
                |    0.00      0.00      0.00     1      |
       1600.0   |    0.00     72.73      0.00     1      |   30.31      0.00       1
                |    0.00      0.00      0.00     1      |
       2133.3   |    0.00     86.20      0.00     1      |   20.20      0.00       1
                |    0.00      0.00      0.00     1      |
       2666.7   |    0.00     94.28      0.00     1      |   10.10      0.00       1
                |    0.00      0.00      0.00     1      |
       3200.0   |    0.00     96.98      0.00     1      |    0.00      0.00       1
                |    0.00      0.00      0.00     1      |
       3733.3   |    0.00     94.28      0.00     1      |  -10.10      0.00       1
                |    0.00      0.00      0.00     1      |
       4266.7   |    0.00     86.20      0.00     1      |  -20.20      0.00       1
                |    0.00      0.00      0.00     1      |
       4800.0   |    0.00     72.73      0.00     1      |  -30.31      0.00       1
                |    0.00      0.00      0.00     1      |
       5333.3   |    0.00     53.88      0.00     1      |  -40.41      0.00       1
                |    0.00      0.00      0.00     1      |
       5866.7   |    0.00     29.63      0.00     1      |  -50.51      0.00       1
                |    0.00      0.00      0.00     1      |
       6400.0   |    0.00      0.00      0.00     1      |  -60.61      0.00       1
                |    0.00      0.00      0.00     1      |
   ----------------------------------------------------------------------------


                       SUMMARY OF REINF. AREA (Sq.mm)

   ----------------------------------------------------------------------------
   SECTION  |          TOP            |         BOTTOM          |   STIRRUPS
   (in mm)  |  Reqd./Provided reinf.  |  Reqd./Provided reinf.  |  (2 legged)
   ----------------------------------------------------------------------------
       0.0  |     0.00/ 402.12( 2-16í )|     0.00/ 981.75( 2-25í )|  8í  @ 180 mm
     533.3  |     0.00/ 402.12( 2-16í )|   237.32/1472.62( 3-25í )|  8í  @ 180 mm
    1066.7  |     0.00/ 402.12( 2-16í )|   450.84/1472.62( 3-25í )|  8í  @ 180 mm
    1600.0  |     0.00/ 402.12( 2-16í )|   632.82/1472.62( 3-25í )|  8í  @ 180 mm
    2133.3  |     0.00/ 402.12( 2-16í )|   773.83/1472.62( 3-25í )|  8í  @ 180 mm
    2666.7  |     0.00/ 402.12( 2-16í )|   863.91/1472.62( 3-25í )|  8í  @ 180 mm
    3200.0  |     0.00/ 402.12( 2-16í )|   894.99/1472.62( 3-25í )|  8í  @ 180 mm
    3733.3  |     0.00/ 402.12( 2-16í )|   863.91/1472.62( 3-25í )|  8í  @ 180 mm
    4266.7  |     0.00/ 402.12( 2-16í )|   773.83/1472.62( 3-25í )|  8í  @ 180 mm
    4800.0  |     0.00/ 402.12( 2-16í )|   632.82/1472.62( 3-25í )|  8í  @ 180 mm
    5333.3  |     0.00/ 402.12( 2-16í )|   450.84/1472.62( 3-25í )|  8í  @ 180 mm
    5866.7  |     0.00/ 402.12( 2-16í )|   237.32/1472.62( 3-25í )|  8í  @ 180 mm
    6400.0  |     0.00/ 402.12( 2-16í )|     0.00/ 981.75( 2-25í )|  8í  @ 180 mm
   ----------------------------------------------------------------------------
```

Related Links

- *V. IS456 2000-Doubly Reinforced Rectangular Beam* (on page 6146)
- *V. IS456 2000-Singly Reinforced Rectangular Beam* (on page 6152)
- *V. IS456 2000-Singly Reinforced Square Beam* (on page 6157)

D8.E.6 Column Design

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement.

The loading which yields maximum reinforcement is called the critical load. Column design is done for square, rectangular, and circular sections. By default, square and rectangular columns and designed with reinforcement distributed on each side equally for the sections under biaxial moments and with reinforcement distributed equally in two faces for sections under uniaxial moment. You may change the default arrangement of the reinforcement with the help of the parameter `RFACE` (see D8.E.3 Design Parameters (on page 1669)). Depending upon the member lengths, section dimensions, and effective length coefficients specified, STAAD.Pro automatically determines the criterion (short or long) of the column design. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 have been taken care of in the column design of STAAD.Pro. Default clear spacing between main reinforcing bars is taken to be 25 mm while arrangement of longitudinal bars.

Default column design output (`TRACK 0.0`) contains the reinforcement provided by STAAD.Pro and the capacity of the section. With the option `TRACK 1.0`, the output contains intermediate results such as the design forces, effective length coefficients, additional moments etc. All design output is given in SI units. An example of a `TRACK 2.0` output follows:

```
            C O L U M N   N O.     1  D E S I G N   R E S U L T S

         M20                   Fe415 (Main)              Fe250 (Sec.)

   LENGTH:  3000.0 mm   CROSS SECTION:  400.0 mm X  600.0 mm  COVER: 40.0 mm

   ** GUIDING LOAD CASE:    1 END JOINT:     1  SHORT COLUMN

   DESIGN FORCES (KNS-MET)
   -----------------------
   DESIGN AXIAL FORCE (Pu)            :    2000.00

                                           About Z            About Y
   INITIAL MOMENTS                    :     160.00             120.00
   MOMENTS DUE TO MINIMUM ECC.        :      52.00              40.00

   SLENDERNESS RATIOS                 :     -                  -
   MOMENTS DUE TO SLENDERNESS EFFECT  :     -                  -
   MOMENT REDUCTION FACTORS           :     -                  -
   ADDITION MOMENTS (Maz and May)     :     -                  -

   TOTAL DESIGN MOMENTS               :     160.00             120.00

   REQD. STEEL AREA   :     3587.44 Sq.mm.
   REQD. CONCRETE AREA:   236412.56 Sq.mm.
   MAIN REINFORCEMENT : Provide  32 - 12 dia. (1.51%,   3619.11 Sq.mm.)
                        (Equally distributed)
   TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

   SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
   ----------------------------------------------------------
   Puz :   3244.31   Muz1 :    269.59   Muy1 :    168.42

   INTERACTION RATIO: 0.98 (as per Cl. 39.6, IS456:2000)

   SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
```

```
----------------------------------------------------------
WORST LOAD CASE:     1
END JOINT:     1 Puz :   3253.88   Muz :    271.48   Muy :    170.09   IR: 0.96
============================================================================
```

Related Links

- *V. IS456 2000-Axially Loaded Rectangular Column* (on page 6136)
- *V. IS456 2000-Axially Loaded Square Column* (on page 6139)

D8.E.7 Bar Combination

Initially the program selects only one bar to calculate the number of bars required and area of steel provided at each section along the length of the beam. You may use the `BAR COMBINATION` command to specify two bar diameters to calculate a combination of each bar to be provided at each section. The syntax for bar combination is given below.

```
START BAR COMBINATION
MD1 <bar diameter>  MEMB  <member list>
MD2 <bar diameter>  MEMB  <member list>
END BAR COMBINATION
```

Note: The bar sizes should be specified in the order of increasing size (i.e., `MD2` bar diameter should be greater than `MD1` bar diameter).

The beam length is divided into three parts, two at its ends and one at span. “Ld” gives the development length to be provided at the two ends of each section.

The typical output for bar combination is shown below:

```
                        OUTPUT FOR BAR COMBINATION


----------------------------------------------------------------------------
           |     M A I N          R E I N F O R C E M E N T
|

----------------------------------------------------------------------------
   SECTION |      0.0- 1600.0     |     1600.0- 4800.0     |     4800.0- 6400.0
|
           |         mm           |           mm           |           mm
|

----------------------------------------------------------------------------
   TOP     |        2-16í         |         2-16í          |         2-16í
|
           |    in 1 layer(s)     |    in  1 layer(s)      |    in  1 layer(s)
|
   Ast Reqd|           0.00       |           0.00         |           0.00
|
       Prov|         402.29       |         402.29         |         402.29
|
   Ld (mm) |         752.2        |         1175.3         |         752.2
|
```

```
-----------------------------------------------------------------------------
   BOTTOM  |       4-16í          |    2-16í   +  2-25í  |         4-16í
|
           |    in 1 layer(s)     |    in  1 layer(s)    |     in  1 layer(s)
|
   Ast Reqd|        632.82        |         894.99       |         632.82
|
       Prov|        804.57        |        1384.43       |         804.57
|
   Ld (mm) |        752.2         |        1175.3        |         752.2
|

-----------------------------------------------------------------------------

=============================================================================
```

D8.E.8 Element Design

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for SX, SY, SXY, SQX, SQY, or MXY. Also, design is not performed at any other point on the surface of the element.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement (Refer to D8.E.1 Section Types for Concrete Design (on page 1669)). The parameters `FYMAIN`, `FC`, and `CLEAR` listed in D8.E.3 Design Parameters (on page 1669) are relevant to slab design.

Other parameters mentioned in the design parameters table are *not* applicable to slab design.

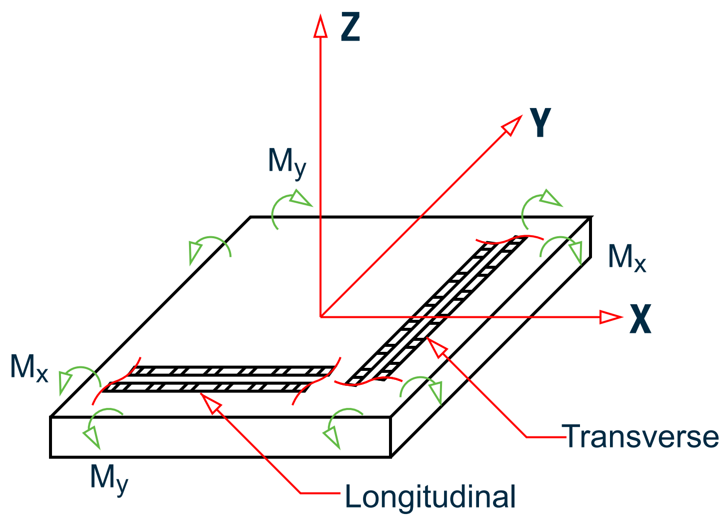


Figure 174: Sign convention of loaded plate element

Example element design input

Element design per the IS 456 code

```
UNIT MMS
START CONCRETE DESIGN
CODE INDIA
```

```
FYMAIN 415 ALL
MAXMAIN 60 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

Example element design output

```
   ELEMENT DESIGN SUMMARY
   ----------------------

   ELEMENT     LONG. REINF      MOM-X /LOAD     TRANS. REINF      MOM-Y /LOAD
               (SQ.MM/ME)     (KN-M/M)          (SQ.MM/ME)      (KN-M/M)

      47 TOP :       156.         0.00 /    0        156.          0.00 /    0
         BOTT:       351.       -15.83 /    3        448.        -20.00 /    3
```

D8.E.9 Wall Design in accordance with IS 456-2000

Note: Shearwall design has been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow it but its use is not recommended.

D8.F. Indian Codes - Concrete Design per IS 13920-2016

STAAD.Pro is capable of performing concrete design based on the 2016 edition of the Indian code IS 13920 *Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces - Code of Practice*. Designs per IS 13920-2016 satisfy all provisions of IS 456-2000 and IS 13920-2016 for beams and columns (Refer to D8.E. Indian Codes - Concrete Design per IS 456 (on page 1669)).

The IS:13920 code provides the design and detailing requirements of reinforced concrete (RC) members to resist lateral effects of earthquake shaking. This gives the RC members sufficient strength and ductility to resist severe earthquake shaking without collapse.

Related Links

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

D8.F.1 Scope

The following reinforced concrete members that can be designed by the latest revision are the following:

a. Reinforced concrete beams (excluding deep beams)
b. Reinforced concrete columns (see note b (on page 1684) for details)

The following types of cross sections for concrete members can be designed:

- For beams:

  prismatic (rectangular and square)
  T-beams (singly reinforced only)
- For columns:

  prismatic: rectangular, square, and circular

Note: IS13920 does not deal with plate design. Plate design is to be performed per IS:456:2000.

Notes

a. The clauses that are already implemented for IS-13920:1993 revision and remains unchanged in revision 2016 are not discussed in these design notes.
b. Clause 7.2 of IS:13920-2016 is checked. Please refer to Relative Strengths of Beams and Columns at a Joint (on page 1686) for additional details.
c. Implementation of Clause 9.0 of IS:13920-2016 for design of beam-column joint for distortional shear is currently excluded.
d. The provisions for the spacing of for special confining reinforcement per Amendment 1 to IS:13920-2016 are not implemented. The design of this reinforcement is designed to the initial IS:13920-2016 edition.
e. STAAD.Pro only designs singly reinforced T beam (flanged) sections.

D8.F.2 Beam Design

Beams are designed for flexure, shear, and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13. All of these sections are scanned to determine the design force envelopes.

For design to be performed as per IS:13920 2016 the width of the member shall not be less than 200 mm (Clause 6.1.2). Also the member shall preferably have a width-to depth ratio of more than 0.3 (Clause 6.1.1).

The factored axial stress on the member should not exceed $0.08fck$ (Clause 6.1) for all active load cases. If it exceeds allowable axial stress no design will be performed.

Cl 6.1 of IS:13920-2016 is applicable for beams resisting earthquake induced effects. The program performs a load type check where it determines if the design load contains earthquake or dynamic load or not. If yes, it will perform check against Cl 6.1. Otherwise, the program will ignore the design load for check against this code provision.

D8.F.2.1 Design for Flexure

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS13920 2016:

1. The minimum grade of concrete shall preferably be M20. (Clause 5.2)
2. Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3.1)
3. The minimum tension steel ratio on any face, at any section, is given by (Clause 6.2.1b)

$$\rho_{min} = 0.24fck/fy$$

   The maximum steel ratio on any face, at any section, is given by (Clause 6.2.2)

$$\rho_{max} = 0.025$$

4. The longitudinal steel on bottom at a joint face must be at least equal to half the steel at its top at the same section. (Clause 6.2.3)
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint. (Clause 6.2.4)
6. Beams shall have at least two 12 mm diameter bars each at the top and bottom faces.

D8.F.2.2 Design for Shear

The shear force to be resisted by vertical hoops is guided by the Clause 6.3.3 of IS 13920:2016 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if the `PLASTIC` parameter is used (refer to D8.F.4 Design Parameters (on page 1686)).

Shear reinforcement is calculated to resist both shear forces and torsional moments. The procedure is the same as that used for IS 456.

The following criteria are satisfied while performing design for shear as per Cl. 6.3.5 of IS-13920 2016:

- The spacing of vertical hoops over a length of 2d at either end of the beam shall not exceed
  - **a.** d/4
  - **b.** 6 times the diameter of the smallest longitudinal bars
  - **c.** 100 mm

  The spacing calculated from above, if less than that calculated from IS 456 considerations, is provided.
- While calculating shear force reinforcement the provision of Clause 6.3.4c (i.e. neglecting design shear capacity of concrete of RC section) is taken into consideration. The spacing of links is provided in accordance with clause 6.3.5.2 of IS-13920 2016.

Related Links
- *V. IS13920 2016-Singly Reinforced Rectangular Beam* (on page 6163)
- *V. IS13920 2016-Doubly Reinforced Rectangular Beam* (on page 6171)

D8.F.3 Column Design

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces as per Clause 7.3.4. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 are performed by the column design of STAAD.Pro. However following clauses have been satisfied to incorporate provisions of IS 13920 2016:

- The minimum grade of concrete shall preferably be M20. (Clause 5.2)
- Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3.1)
- The minimum dimension of column member shall not be less than:
  - **a.** 20 times the diameter of the largest longitudinal reinforcement in the beam passing through or anchoring into the column at the joint, or
  - **b.** 300 mm (Clause 7.1.1)
- The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.45. (Clause 7.1.2)
- The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided. (Clause 7.4.2d)
- The minimum diameter of the rectangular hoop is 8 mm when diameter of the longitudinal bar is less than or equal to 32 mm, and 10 mm when diameter of longitudinal bar is more than 32 mm. (Clause 7.4.2a)
- Special confining reinforcement shall be provided over a length l_o from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length l_o shall not be less than:
  - **a.** larger lateral dimension of the member at the section where yielding occurs,
  - **b.** 1/6 of clear span of the member,
  - **c.** 450 mm (Clause 8.1a)
- The spacing of hoops used as special confining reinforcement shall not exceed:
  - **a.** ¼ of minimum member dimension,
  - **b.** 6 times diameter of the smallest longitudinal bars, or

c. 100 mm (Clause 8.1b)

- The area of cross-section of hoops provided are checked against the provisions for minimum area of cross-section of the bar forming rectangular, circular, or spiral hoops, to be used as special confining reinforcement. (Clause 8.1c)

Cl 7.1 of IS:13920-2016 is applicable for columns resisting earthquake induced effects. The program will check whether the design load is gravity load case (as defined by the `GLD` parameter) or earthquake load. If yes, the program will perform this code check. Otherwise, the program will ignore the design load case for check against this code provision.

Note: In previous releases, the design performed checks against this clause for *all* types of design load cases, irrespective of whether they contain earthquake load or not. This logic is now automated to include only earthquake or gravity load case.

Relative Strengths of Beams and Columns at a Joint

Clause 7.2 of IS:13920-2016 checks the relative strength of beams and columns at a joint to evaluate if any column at a joint is acting as a gravity column or is a part of lateral load resisting system. After the concrete design is completed per IS 13920-2016, then this relative strength check is automatically performed at joints in which *all* connecting beams and columns were designed. Thus, no new commands are required for this check to be performed so long as all concrete members at a joint are designed. Per Cl 7.2, the sum of all column moment resistances at a joint shall be greater that 1.4× the sum of all beam moment resistances at the same point.

Note: STAAD.Pro does not have designate columns as gravity load structural elements. Rather, the program considers all columns as lateral load resisting. If this check fails at any joint, the column dimensions, beam dimension, or both should be changed to make the column suitable for the lateral load resisting system.

It is recommend that the design of all concrete beams and columns be performed in one concrete block (a "concrete block" is the information contained in the `START CONCRETE DESIGN … END CONCRETE DESIGN` input section) so this check can be performed. If multiple concrete blocks are defined, all previous relative strength check result will be deleted since multiple design of beams and column designs may change the relative strength of beams and columns at a joint. The strength check performed after the last concrete block will be considered valid and is available as result.

The output file will issue a note regarding the results of this check. A detailed output of the checks can be included in the output using the `TRACK 4` parameter in the concrete design. The detailed output of the joint relative strength checks are also included in the *`filename`*`_13920.txt` file (which is generated regardless of the `TRACK` parameter used or check status).

Note: The program assumes that all beams connected to a column are horiztonal and parallel to global X and Z axes. If beams do not meet this requirement a warning message is issued in the analysis output.

Related Links

- *V. IS13920 2016-Relative Strength Check* (on page 6178)

D8.F.4 Design Parameters

The program contains a number of parameters that are needed to perform design as per IS 13920 2016. It accepts all parameters that are needed to perform design as per IS:456. Over and above it has some other parameters that are required only when designed is performed as per IS:13920 2016. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. The following table contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 172: Indian Concrete Design IS 13920 2016 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `IS13920 2016` or `ISH13920` Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BRACING	0.0	Beam Design 1.0 = the effect of axial force will be taken into account for beam design. Column Design: Correspond to the terms "Braced" and "Unbraced" described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000. 1.0 = the column is unbraced about major axis. 2.0 = the column is unbraced about minor axis. 3.0 = the column is unbraced about both axis.
DEPTH	YD	Total depth to be used for design. This value defaults to YD (depth of section in Y direction) as provided under `MEMBER PROPERTIES`.

Parameter Name	Default Value	Description
CLB	25 mm	Clear cover to stirrups at bottom for beams. **Note:** If both CLEAR and CLB are defined for beams, CLB is considered in design. Concrete cover to main reinforcement at bottom of beam will be calculated from CLB.
CLEAR	>30 mm 40 mm	Concrete cover to main reinforcement bars for beams Concrete cover to main longitudinal bars for columns **Notes:** 1. This is the clear cover to the outermost main reinforcing bar. It is *not* the clear cover for the stirrups or the tie bars. 2. If the CLB parameter is not defined, then the CLEAR parameter will be used by default.
CLT	25 mm	Clear cover to stirrups at top for beams. **Note:** If both CLEAR and CLT are defined for beams, CLT is considered in design. Concrete cover to main reinforcement at top of beam will be calculated from CLT.

Parameter Name	Default Value	Description
COMBINE	0.0	Default value means there will be no member combination. 1.0 = no printout of sectional force and critical load for combined member in the output. 2.0 = printout of sectional force for combined member in the output. 3.0 = printout of both sectional force and critical load for combined member in the output. *** **Note:** Refer description of COMBINE parameter at the end of the parameter table.
EFACE	0.0	Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member. **Note:** Both `SFACE` and `EFACE` are input as positive numbers. Not valid along with `COMBINE` parameter. *
ELY	1.0	Ratio of effective length to actual length of column about minor axis.
ELZ	1.0	Ratio of effective length to actual length of column about major axis.

Parameter Name	Default Value	Description
ENSH	0.0	Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000. 1.0 = ordinary shear check to be performed (no enhancement of shear strength at sections close to support) a positive value(say x) = shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1) a negative value(say –y) = shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts.(Refer note after Table 8A.1) 0.0 = the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (<2d) close to support otherwise ordinary shear check will be performed.
EUDL	None	Equivalent u.d.l on span of the beam. This load value must be the unfactored load on span. During design the load value is multiplied by a factor 1.2. If no u.d.l is defined factored shear force due to gravity load on span will be taken as zero. No elastic or plastic moment will be calculated. Shear design will be performed based on analysis result. (Refer note)
FYMAIN	415 N/mm^2	Yield Stress for main reinforcing steel.
FYSEC	415 N/mm^2	Yield Stress for secondary reinforcing steel.

Parameter Name	Default Value	Description
FC	Fcu or 30 N/mm^2	Concrete Yield Stress. The FC value will be taken from Material Definition where Compressive Strength (Fcu) is defined. If the input is not available, the design will consider 30 MPa as default value.
GLD	None	Gravity load number to be considered for calculating equivalent u.d.l on span of the beam, in case no EUDL is mentioned in the input. This load case can be any static load case containing **MEMBER LOAD** on the beam which includes **UNI**, **CON**, **LIN** and **TRAP** member loading. **CMOM** member loading is considered only when it is specified in local direction. **FLOOR LOAD** is also considered. The load can be primary or combination load. For combination load only load numbers included in load combination is considered. The load factors are ignored. Internally the unfactored load is multiplied by a factor 1.2 during design. If both EUDL and GLD parameters are mentioned in the input mentioned EUDL will be considered in design **Note:** No dynamic (Response spectrum, 1893, Time History) and moving load cases are considered. CMOM member loading in global direction is not considered. UMOM member loading is not considered.

Parameter Name	Default Value	Description
HLINK	Spacing of longitudinal bars measured to the outer face	Longer dimension of the rectangular confining hoop measured to its outer face. It shall not exceed 300 mm as per Cl. 7.4.8. If the HLINK value as provided in the input file does not satisfy the clause the value will be internally assumed as the default one. This parameter is valid for rectangular column.
IPLM	0.0	Default value calculates elastic/ plastic hogging and sagging moments of resistance of beam at its ends. 1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at start node of beam. This implies no support exists at start node. -1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at start node of beam. . This implies support exists at start node. 2.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at end node of beam. This implies no support exists at end node. -2.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at end node of beam. . This implies support exists at end node. ** **Note:** Not valid along with COMBINE parameter. **

Parameter Name	Default Value	Description
IMB	0.0	Default value calculates elastic/ plastic hogging and sagging moments of resistance of beam at its ends. 1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at both ends of beam. This implies no support exist at either end of the member. -1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at both ends of beam. This implies support exist at both ends of the member.** **Note:** Not valid along with `COMBINE` parameter. **
MINMAIN	10 mm	Minimum main reinforcement bar size.
MAXMAIN	60 mm	Maximum main reinforcement bar size.
MINSEC	8 mm	Minimum secondary reinforcement bar size.
MAXSEC	12 mm	Maximum secondary reinforcement bar size.
PLASTIC	0.0	Default value calculates elastic hogging and sagging moments of resistance of beam at its ends. 1.0 = plastic hogging and sagging moments of resistance of beam to be calculated at its ends.
RATIO	4.0	Maximum percentage of longitudinal reinforcement in columns.

Parameter Name	Default Value	Description
REINF	0.0	0.0 = Tied column (default) 1.0 = spiral reinforcement
RENSH	0.0	Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts. Refer to Notes (on page 1675) for IS456 Design Parameters.
RFACE	4.0	4.0 = longitudinal reinforcement in column is arranged equally along four faces. 2.0 invokes two faced distribution about major axis. 3.0 invokes two faced distribution about minor axis.
SFACE	0.0	Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.* **Note:** Both SFACE and EFACE are input as positive numbers.*
SPSMAIN	25 mm	Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm.
TORISION	0.0	0.0 = torsion to be considered in beam design. 1.0 = torsion to be neglected in beam design.

Parameter Name	Default Value	Description
TRACK	0.0	Beam Design: 0) output consists of reinforcement details at START, MIDDLE and END. 1) critical moments are printed in addition to `TRACK 0` output. 2) required steel for intermediate sections defined by `NSECTION` are printed in addition to `TRACK 1` output. 4) prints the output for joint strength check per Cl. 7.2 of IS 13920-2016 Column Design: 0) reinforcement details are printed. 1) column interaction analysis results are printed in addition to `TRACK 0` output. 2) a schematic interaction diagram and intermediate interaction values are printed in addition to `TRACK 1` output. 4) prints the output for joint strength check per Cl. 7.2 of IS 13920-2016 9) details of section capacity about major and minor axes are printed in addition to `TRACK 1` output
ULY	1.0	Ratio of unsupported length to actual length of column about minor axis.
ULZ	1.0	Ratio of unsupported length to actual length of column about major axis.
WIDTH	ZD	Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES.

Bar combination has been introduced for detailing. Please refer D8.G.6 Bar Combination (on page 1712) for details.

* `EFACE` and `SFACE` command is not valid for member combination.

** `IPLM` and `IMB` commands are not valid for member combination. These commands are ignored for members forming physical member.

*** The purpose of `COMBINE` command is the following:

1. If a beam spanning between two supports is subdivided into many sub-beams this parameter will combine them into one member. It can also be used to combine members to form one continuous beam spanning over more than two supports.
2. When two or more members are combined during design plastic or elastic moments will be calculated at the column supports. At all the intermediate nodes (if any) this calculation will be ignored.

   Note: Please note that the program only recognizes column at right angle to the beam. Inclined column support is ignored.
3. It will calculate sectional forces at 13 sections along the length of the combined member.
4. It will calculate critical loads (similar to that of Design Load Summary) for all active load cases during design. Beams will be combined only when `DESIGN BEAM` command is issued.

The following lines should be satisfied during combination of members:

1. Members to be combined should have same sectional properties if any single span between two column supports of a continuous beam is subdivided into several members.
2. Members to be combined should have same constants (E, Poi ratio, alpha, density, and beta angle)
3. Members to be combined should lie in one straight line.
4. Members to be combined should be continuous.
5. Vertical members (i.e., columns) cannot be combined.
6. Same member cannot be used more than once to form two different combined members.
7. The maximum number of members that can be combined into one member is 299.

Note: Sectional forces and critical load for combined member output will only be available when all the members combined are successfully designed in both flexure and shear.

`ENSH` and `RENSH` parameters will have to be provided (as and when necessary) even if physical member has been formed.

D8.F.3.1 Example

The following lines show a standard example for design to be performed in IS 13920 2016.

```
STAAD SPACE
UNIT METER MTON
JOINT COORDINATES
… … …
MEMBER INCIDENCES
… … …
MEMBER PROPERTY INDIAN
… … …
CONSTANTS
… … …
SUPPORTS
… … …
DEFINE 1893 LOAD
ZONE 0.05 I 1 K 1 B 1
SELFWEIGHT
```

```
JOINT WEIGHT
… … …
LOAD 1 SEISMIC LOAD IN X DIR
1893 LOAD X 1
LOAD 2 SEISMIC LOAD IN Z DIR
1893 LOAD Z 1
LOAD 3 DL
MEMBER LOAD
…… UNI GY -5
LOAD 4 LL
MEMBER LOAD
…… UNI GY -3
LOAD COMB 5 1.5(DL+LL)
3 1.5 4 1.5
LOAD COMB 6 1.2(DL+LL+SLX)
1 1.2 3 1.2 4 1.2
LOAD COMB 7 1.2(DL+LL-SLX)
1 1.2 3 1.2 4 -1.2
LOAD COMB 8 1.2(DL+LL+SLZ)
2 1.2 3 1.2 4 1.2
LOAD COMB 9 1.2(DL+LL-SLZ)
2 1.2 3 1.2 4 -1.2
PDELTA ANALYSIS
LOAD LIST 5 TO 9
START CONCRETE DESIGN
CODE IS13920 2016
UNIT MMS NEWTON
FYMAIN 415 ALL
FC 20 ALL
MINMAIN 12 ALL
MAXMAIN 25 ALL
TRACK 2.0 ALL
*** Unfactored gravity load on members 110 to 112 is 8 t/m (DL+LL) i.e. 78.46
N/mm
EUDL 78.46 MEMB 110 TO 112
** Members to be combined into one physical member
COMBINE 3.0 MEMB 110 TO 112
*** Plastic moment considered
PLASTIC 1.0 MEMB 110 TO 112
DESIGN BEAM 110 TO 112
DESIGN COLUMN ………
END CONCRETE DESIGN
FINISH
```

D8.G. Indian Codes - Concrete Design per IS 13920-1993

STAAD.Pro is capable of performing concrete design based on the 1993 edition of the Indian code IS 13920 *Code of Practice for Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces*. Designs per IS 13920-1993 satisfy all provisions of IS 456 - 2000 and IS 13920-1993 for beams and columns (Refer to D8.E. Indian Codes - Concrete Design per IS 456 (on page 1669)).

D8.G.1 Design Operations

Earthquake motion often induces force large enough to cause inelastic deformations in the structure. If the structure is brittle, sudden failure could occur. But if the structure is made to behave ductile, it will be able to sustain the earthquake effects better with some deflection larger than the yield deflection by absorption of energy. Therefore ductility is also required as an essential element for safety from sudden collapse during severe shocks.

D8.G.2 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed.

- For Beams: Prismatic (Rectangular & Square) and T-shape
- For Columns : Prismatic (Rectangular, Square, and Circular)

D8.G.3 Design Parameters

The program contains a number of parameters that are needed to perform design as per IS 13920 1993. It accepts all parameters that are needed to perform design as per IS:456. Over and above it has some other parameters that are required only when designed is performed as per IS:13920 1993. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. The following table contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 173: Indian Concrete Design IS 13920 1993 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `IS13920 1993` Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).

Parameter Name	Default Value	Description
BRACING	0.0	Beam Design 1.0 = the effect of axial force will be taken into account for beam design. Column Design: Correspond to the terms "Braced" and "Unbraced" described in Notes 1, 2, and 3 of Clause 39.7.1 of IS456:2000. 1.0 = the column is unbraced about major axis. 2.0 = the column is unbraced about minor axis. 3.0 = the column is unbraced about both axis.
DEPTH	YD	Total depth to be used for design. This value defaults to YD (depth of section in Y direction) as provided under `MEMBER PROPERTIES`.
CLEAR	25 mm 40 mm	For beam members. For column members **Note:** This is the clear cover to the outermost main reinforcing bar. It is *not* the clear cover for the stirrups or the tie bars.
COMBINE	0.0	Default value means there will be no member combination. 1.0 = no printout of sectional force and critical load for combined member in the output. 2.0 = printout of sectional force for combined member in the output. 3.0 = printout of both sectional force and critical load for combined member in the output. ***

Parameter Name	Default Value	Description
EFACE	0.0	Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member. **Note:** Both SFACE and EFACE are input as positive numbers.*
ELZ	1.0	Ratio of effective length to actual length of column about major axis.
ELY	1.0	Ratio of effective length to actual length of column about minor axis.
ENSH	0.0	Perform shear check against enhanced shear strength as per Cl. 40.5 of IS456:2000. 1.0 = ordinary shear check to be performed (no enhancement of shear strength at sections close to support) a positive value(say x) = shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. (Refer note after Table 8A.1) a negative value(say –y) = shear strength will be enhanced up to a distance y from the end of the member. This is used only when a span of a beam is subdivided into two or more parts.(Refer note after Table 8A.1) 0.0 = the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (<2d) close to support otherwise ordinary shear check will be performed.

Parameter Name	Default Value	Description
EUDL	None	Equivalent u.d.l on span of the beam. This load value must be the unfactored load on span. During design the load value is multiplied by a factor 1.2. If no u.d.l is defined factored shear force due to gravity load on span will be taken as zero. No elastic or plastic moment will be calculated. Shear design will be performed based on analysis result. (Refer note)
FYMAIN	415 N/mm^2	Yield Stress for main reinforcing steel.
FYSEC	415 N/mm^2	Yield Stress for secondary reinforcing steel.
FC	30 N/mm^2	Concrete Yield Stress.

Parameter Name	Default Value	Description
GLD	None	Gravity load number to be considered for calculating equivalent u.d.l on span of the beam, in case no EUDL is mentioned in the input. This load case can be any static load case containing **MEMBER LOAD** on the beam which includes **UNI**, **CON**, **LIN** and **TRAP** member loading. **CMOM** member loading is considered only when it is specified in local direction. **FLOOR LOAD** is also considered. The load can be primary or combination load. For combination load only load numbers included in load combination is considered. The load factors are ignored. Internally the unfactored load is multiplied by a factor 1.2 during design. If both `EUDL` and `GLD` parameters are mentioned in the input mentioned EUDL will be considered in design **Note:** No dynamic (Response spectrum, 1893, Time History) and moving load cases are considered. `CMOM` member loading in global direction is not considered. `UMOM` member loading is not considered.
HLINK	Spacing of longitudinal bars measured to the outer face	Longer dimension of the rectangular confining hoop measured to its outer face. It shall not exceed 300 mm as per Cl. 7.4.8. If the `HLINK` value as provided in the input file does not satisfy the clause the value will be internally assumed as the default one. This parameter is valid for rectangular column.

Parameter Name	Default Value	Description
IPLM	0.0	Default value calculates elastic/ plastic hogging and sagging moments of resistance of beam at its ends. 1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at start node of beam. This implies no support exists at start node. -1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at start node of beam. . This implies support exists at start node. 2.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at end node of beam. This implies no support exists at end node. -2.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at end node of beam. . This implies support exists at end node. **
IMB	0.0	Default value calculates elastic/ plastic hogging and sagging moments of resistance of beam at its ends. 1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be ignored at both ends of beam. This implies no support exist at either end of the member. -1.0 = calculation of elastic/ plastic hogging and sagging moments of resistance of beam to be considered at both ends of beam. This implies support exist at both ends of the member.**

Parameter Name	Default Value	Description
MINMAIN	10 mm	Minimum main reinforcement bar size.
MAXMAIN	60 mm	Maximum main reinforcement bar size.
MINSEC	8 mm	Minimum secondary reinforcement bar size.
MAXSEC	12 mm	Maximum secondary reinforcement bar size.
PLASTIC	0.0	Default value calculates elastic hogging and sagging moments of resistance of beam at its ends. 1.0 = plastic hogging and sagging moments of resistance of beam to be calculated at its ends.
RATIO	4.0	Maximum percentage of longitudinal reinforcement in columns.
REINF	0.0	0.0 = Tied column (default) 1.0 = spiral reinforcement
RENSH	0.0	Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts. Refer to Notes (on page 1675) for IS456 Design Parameters.
RFACE	4.0	4.0 = longitudinal reinforcement in column is arranged equally along four faces. 2.0 invokes two faced distribution about major axis. 3.0 invokes two faced distribution about minor axis.

Parameter Name	Default Value	Description
SFACE	0.0	Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.* **Note:** Both SFACE and EFACE are input as positive numbers.*
SPSMAIN	25 mm	Minimum clear distance between main reinforcing bars in beam and column. For column center to center distance between main bars cannot exceed 300 mm.
TORISION	0.0	0.0 = torsion to be considered in beam design. 1.0 = torsion to be neglected in beam design.
TRACK	0.0	Beam Design: 0) output consists of reinforcement details at START, MIDDLE and END. 1) critical moments are printed in addition to TRACK 0 output. 2) required steel for intermediate sections defined by NSECTION are printed in addition to TRACK 1 output. Column Design: 0) reinforcement details are printed. 1) column interaction analysis results are printed in addition to TRACK 0 output. 2) a schematic interaction diagram and intermediate interaction values are printed in addition to TRACK 1 output.

Parameter Name	Default Value	Description
ULY	1.0	Ratio of unsupported length to actual length of column about minor axis.
ULZ	1.0	Ratio of unsupported length to actual length of column about major axis.
WIDTH	ZD	Width to be used for design. This value defaults to ZD as provided under MEMBER PROPERTIES.

Bar combination has been introduced for detailing. Please refer D8.G.6 Bar Combination (on page 1712) for details.

* `EFACE` and `SFACE` command is not valid for member combination.

** `IPLM` and `IMB` commands are not valid for member combination. These commands are ignored for members forming physical member.

*** The purpose of `COMBINE` command is the following:

1. If a beam spanning between two supports is subdivided into many sub-beams this parameter will combine them into one member. It can also be used to combine members to form one continuous beam spanning over more than two supports.
2. When two or more members are combined during design plastic or elastic moments will be calculated at the column supports. At all the intermediate nodes (if any) this calculation will be ignored.

   Note: Please note that the program only recognizes column at right angle to the beam. Inclined column support is ignored.
3. It will calculate sectional forces at 13 sections along the length of the combined member.
4. It will calculate critical loads (similar to that of Design Load Summary) for all active load cases during design. Beams will be combined only when `DESIGN BEAM` command is issued.

The following lines should be satisfied during combination of members:

1. Members to be combined should have same sectional properties if any single span between two column supports of a continuous beam is subdivided into several members.
2. Members to be combined should have same constants (E, Poi ratio, alpha, density, and beta angle)
3. Members to be combined should lie in one straight line.
4. Members to be combined should be continuous.
5. Vertical members (i.e., columns) cannot be combined.
6. Same member cannot be used more than once to form two different combined members.
7. The maximum number of members that can be combined into one member is 299.

Note: Sectional forces and critical load for combined member output will only be available when all the members combined are successfully designed in both flexure and shear.

`ENSH` and `RENSH` parameters will have to be provided (as and when necessary) even if physical member has been formed.

D8.B.3.1 Example

The following lines show a standard example for design to be performed in IS 13920 1993.

```
STAAD SPACE
UNIT METER MTON
JOINT COORDINATES
…
MEMBER INCIDENCES
…
MEMBER PROPERTY INDIAN
…
CONSTANTS
…
SUPPORTS
…
DEFINE 1893 LOAD
ZONE 0.05 I 1 K 1 B 1
SELFWEIGHT
JOINT WEIGHT
…
LOAD 1 SEISMIC LOAD IN X DIR
1893 LOAD X 1
LOAD 2 SEISMIC LOAD IN Z DIR
1893 LOAD Z 1
LOAD 3 DL
MEMBER LOAD
…… UNI GY -5
LOAD 4 LL
MEMBER LOAD
……. UNI GY -3
LOAD COMB 5 1.5(DL+LL)
3 1.5 4 1.5
LOAD COMB 6 1.2(DL+LL+SLX)
1 1.2 3 1.2 4 1.2
LOAD COMB 7 1.2(DL+LL-SLX)
1 1.2 3 1.2 4 -1.2
LOAD COMB 8 1.2(DL+LL+SLZ)
2 1.2 3 1.2 4 1.2
LOAD COMB 9 1.2(DL+LL-SLZ)
2 1.2 3 1.2 4 -1.2
PDELTA ANALYSIS
LOAD LIST 5 TO 9
START CONCRETE DESIGN
CODE IS13920 1993
UNIT MMS NEWTON
FYMAIN 415 ALL
FC 20 ALL
MINMAIN 12 ALL
MAXMAIN 25 ALL
TRACK 2.0 ALL
*** Unfactored gravity load on members 110 to 112 is 8 t/m (DL+LL) i.e.,
78.46 New/mm
EUDL 78.46 MEMB 110 TO 112
** Members to be combined into one physical member
COMBINE 3.0 MEMB 110 TO 112
*** Plastic moment considered
PLASTIC 1.0 MEMB 110 TO 112
```

```
DESIGN BEAM 110 TO 112
DESIGN COLUMN …
END CONCRETE DESIGN
FINISH
```

D8.G.4 Beam Design

Beams are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. The total number of sections considered is 13. All of these sections are scanned to determine the design force envelopes.

For design to be performed as per IS:13920 the width of the member shall not be less than 200 mm(Clause 6.1.3). Also the member shall preferably have a width-to depth ratio of more than 0.3 (Clause 6.1.2).

The factored axial stress on the member should not exceed 0.1fck (Clause 6.1.1) for all active load cases. If it exceeds allowable axial stress no design will be performed.

D8.G.4.1 Design for Flexure

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS-13920:

1. The minimum grade of concrete shall preferably be M20. (Clause 5.2)
2. Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3)
3. The minimum tension steel ratio on any face, at any section, is given by (Clause 6.2.1b)

   $$\rho_{min} = 0.24fck/fy$$

   The maximum steel ratio on any face, at any section, is given by (Clause 6.2.2)

   $$\rho_{max} = 0.025$$
4. The positive steel ratio at a joint face must be at least equal to half the negative steel at that face. (Clause 6.2.3)
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint. (Clause 6.2.4)

D8.G.4.2 Design for Shear

The shear force to be resisted by vertical hoops is guided by the Clause 6.3.3 of IS 13920:1993 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if PLASTIC parameter is mentioned in the input file. (Refer Table 8A1.1)

Shear reinforcement is calculated to resist both shear forces and torsional moments. Procedure is same as that of IS 456.

The following criteria are satisfied while performing design for shear as per Cl. 6.3.5 of IS-13920:

The spacing of vertical hoops over a length of 2d at either end of the beam shall not exceed

a. d/4
b. 8 times the diameter of the longitudinal bars

In no case this spacing is less than 100 mm.

The spacing calculated from above, if less than that calculated from IS 456 consideration is provided.

D8.G.4.3 Beam Design Output

The default design output of the beam contains flexural and shear reinforcement provided at 5 equally spaced sections along the length of the beam. User has option to get a more detail output. All beam design outputs are given in IS units. An example of rectangular beam design output with the `TRACK 2.0` is presented below:

```
                  B E A M  N O.      1   D E S I G N  R E S U L T S

         M20                   Fe415 (Main)             Fe250 (Sec.)

       LENGTH:  6400.0 mm      SIZE:   300.0 mm X  400.0 mm   COVER: 25.0 mm


                          DESIGN LOAD SUMMARY (KN MET)
   -----------------------------------------------------------------------------
   SECTION |FLEXURE  (Maxm. Sagging/Hogging moments)|          SHEAR
   (in mm) |      P         MZ         MX   Load Case |    VY        MX  Load Case
   -----------------------------------------------------------------------------
       0.0 |     0.00      0.00      0.00     1      |   60.61     0.00      1
           |     0.00      0.00      0.00     1      |
     533.3 |     0.00     29.63      0.00     1      |   50.51     0.00      1
           |     0.00      0.00      0.00     1      |
    1066.7 |     0.00     53.88      0.00     1      |   40.41     0.00      1
           |     0.00      0.00      0.00     1      |
    1600.0 |     0.00     72.73      0.00     1      |   30.31     0.00      1
           |     0.00      0.00      0.00     1      |
    2133.3 |     0.00     86.20      0.00     1      |   20.20     0.00      1
           |     0.00      0.00      0.00     1      |
    2666.7 |     0.00     94.28      0.00     1      |   10.10     0.00      1
           |     0.00      0.00      0.00     1      |
    3200.0 |     0.00     96.98      0.00     1      |    0.00     0.00      1
           |     0.00      0.00      0.00     1      |
    3733.3 |     0.00     94.28      0.00     1      |  -10.10     0.00      1
           |     0.00      0.00      0.00     1      |
    4266.7 |     0.00     86.20      0.00     1      |  -20.20     0.00      1
           |     0.00      0.00      0.00     1      |
    4800.0 |     0.00     72.73      0.00     1      |  -30.31     0.00      1
           |     0.00      0.00      0.00     1      |
    5333.3 |     0.00     53.88      0.00     1      |  -40.41     0.00      1
           |     0.00      0.00      0.00     1      |
    5866.7 |     0.00     29.63      0.00     1      |  -50.51     0.00      1
           |     0.00      0.00      0.00     1      |
    6400.0 |     0.00      0.00      0.00     1      |  -60.61     0.00      1
           |     0.00      0.00      0.00     1      |
   *** DESIGN SHEAR FORCE AT SECTION     0.0 IS    60.61 KN.
                                             - CLAUSE 6.3.3 OF IS-13920
   *** DESIGN SHEAR FORCE AT SECTION  6400.0 IS    60.61 KN.
                                             - CLAUSE 6.3.3 OF IS-13920
   NOTE :

   MOMENT OF RESISTANCE IS CALCULATED BASED ON THE AREA OF STEEL PROVIDED.
   IF AREA OF STEEL PROVIDED IS MUCH HIGHER COMPARED TO AREA OF STEEL
   REQUIRED MOMENT OF RESISTANCE WILL INCREASE WHICH MAY INCREASE DESIGN
   SHEAR FORCE.
   -----------------------------------------------------------------------------
    STAAD SPACE                                              -- PAGE NO.    7
```

```
      0.0 |    0.00/ 402.12( 2-16í )|    0.00/ 981.75( 2-25í )|  8í  @ 100 mm
    533.3 |    0.00/ 402.12( 2-16í )|  281.26/1472.62( 3-25í )|  8í  @ 180 mm
   1066.7 |    0.00/ 402.12( 2-16í )|  450.84/1472.62( 3-25í )|  8í  @ 180 mm
   1600.0 |    0.00/ 402.12( 2-16í )|  632.82/1472.62( 3-25í )|  8í  @ 180 mm
   2133.3 |    0.00/ 402.12( 2-16í )|  773.83/1472.62( 3-25í )|  8í  @ 180 mm
   2666.7 |    0.00/ 402.12( 2-16í )|  863.91/1472.62( 3-25í )|  8í  @ 180 mm
   3200.0 |    0.00/ 402.12( 2-16í )|  894.99/1472.62( 3-25í )|  8í  @ 180 mm
   3733.3 |    0.00/ 402.12( 2-16í )|  863.91/1472.62( 3-25í )|  8í  @ 180 mm
   4266.7 |    0.00/ 402.12( 2-16í )|  773.83/1472.62( 3-25í )|  8í  @ 180 mm
   4800.0 |    0.00/ 402.12( 2-16í )|  632.82/1472.62( 3-25í )|  8í  @ 180 mm
   5333.3 |    0.00/ 402.12( 2-16í )|  450.84/1472.62( 3-25í )|  8í  @ 180 mm
   5866.7 |    0.00/ 402.12( 2-16í )|  281.26/1472.62( 3-25í )|  8í  @ 180 mm
   6400.0 |    0.00/ 402.12( 2-16í )|    0.00/ 981.75( 2-25í )|  8í  @ 100 mm
  ---------------------------------------------------------------------------
```

D8.G.5 Column Design

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces as per Clause 7.3.4. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS:456 have been taken care of in the column design of STAAD.Pro. However following clauses have been satisfied to incorporate provisions of IS 13920:

- The minimum grade of concrete shall preferably be M20. (Clause 5.2)
- Steel reinforcements of grade Fe415 or less only shall be used. (Clause 5.3)
- The minimum dimension of column member shall not be less than 200 mm. For columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300 mm. (Clause 7.1.2)
- The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.4. (Clause 7.1.3)
- The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided. (Clause 7.3.3)
- Special confining reinforcement shall be provided over a length l_0 from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length l_0 shall not be less than a) larger lateral dimension of the member at the section where yielding occurs, b) 1/6 of clear span of the member, and c) 450 mm. (Clause 7.4.1)
- The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75 mm nor more than 100 mm. (Clause 7.4.6)
- The area of cross-section of hoops provided are checked against the provisions for minimum area of cross-section of the bar forming rectangular, circular or spiral hoops, to be used as special confining reinforcement. (Clause 7.4.7 and 7.4.8)

D8.G.5.1 Column Design Output

Default column design output (`TRACK 0.0`) contains the reinforcement provided by STAAD.Pro and the capacity of the section. With the option `TRACK 1.0`, the output contains intermediate results such as the design forces, effective length coefficients, additional moments etc. All design output is given in SI units. An example of a column design output (with option `TRACK 1.0`) is given below.

```
============================================================================
        C O  L U M N   N O.     3  D E S I  G N   R E S U L T S
     M20
                Fe415 (Main)
            Fe415 (Sec.)
LENGTH:  3000.0 mm   CROSS SECTION:
            350.0 mm X  400.0 mm  COVER: 40.0 mm
```

```
** GUIDING LOAD CASE:   5 END JOINT:
            2  SHORT COLUMN
DESIGN FORCES (KNS-MET)
-----------------------
DESIGN AXIAL FORCE (Pu)
            :   226.7
About Z
            About Y
INITIAL MOMENTS
                :    0.64
            146.28
MOMENTS DUE TO MINIMUM ECC.
            :    4.53            4.53
SLENDERNESS RATIOS
                :     -
            -
MOMENTS DUE TO SLENDERNESS EFFECT   :
                -
            -
MOMENT REDUCTION FACTORS
                :     -
            -
ADDITION MOMENTS (Maz and May)
                :     -
            -
TOTAL DESIGN MOMENTS
                :    4.53
            146.28
** GUIDING LOAD CASE:   5

            Along Z         Along Y
DESIGN SHEAR FORCES
                :   43.31
            76.08
REQD. STEEL AREA   :   3313.56  Sq.mm.
MAIN REINFORCEMENT : Provide  12 - 20 dia.
            (2.69%,   3769.91 Sq.mm.)

            (Equally distributed)
CONFINING REINFORCEMENT  : Provide 10 mm dia.
            rectangular ties @  85 mm c/c
        over  a length   500.0 mm from each joint face towards
                           midspan  as per Cl. 7.4.6 of IS-13920.
TIE REINFORCEMENT
            : Provide 10 mm dia. rectangular ties @ 175 mm c/c
SECTION CAPACITY (KNS-MET)
--------------------------
Puz :   2261.52   Muz1 :
            178.71   Muy1 :    150.75
INTERACTION RATIO: 1.00 (as per Cl. 39.6, IS456:2000)
============================================================================
********************END OF COLUMN DESIGN RESULTS********************
```

D8.G.6 Bar Combination

Initially the program selects only one bar to calculate the number of bars required and area of steel provided at each section along the length of the beam. You may use the `BAR COMBINATION` command to specify two bar diameters to calculate a combination of each bar to be provided at each section. The syntax for bar combination is given below.

```
START BAR COMBINATION
MD1 <bar diameter>  MEMB  <member list>
MD2 <bar diameter>  MEMB  <member list>
END BAR COMBINATION
```

Note: The bar sizes should be specified in the order of increasing size (i.e., `MD2` bar diameter should be greater than `MD1` bar diameter).

The beam length is divided into three parts, two at its ends and one at span. Ld gives the development length to be provided at the two ends of each section.

The typical output for bar combination is shown below:

```
                         OUTPUT FOR BAR COMBINATION


-----------------------------------------------------------------------------
           |     M A I N          R E I N F O R C E M E N T
|

-----------------------------------------------------------------------------
   SECTION |     0.0- 1600.0     |     1600.0- 4800.0     |     4800.0- 6400.0
|
           |         mm          |           mm           |           mm
|

-----------------------------------------------------------------------------
   TOP     |       2-16í         |        2-16í           |        2-16í
|
           |   in 1 layer(s)     |    in  1 layer(s)      |    in  1 layer(s)
|
   Ast Reqd|          0.00       |          0.00          |          0.00
|
       Prov|       402.29        |        402.29          |        402.29
|
   Ld (mm) |       752.2         |       1175.3           |        752.2
|

-----------------------------------------------------------------------------
   BOTTOM  |       4-16í         |    2-16í   +  2-25í    |        4-16í
|
           |   in 1 layer(s)     |    in  1 layer(s)      |    in  1 layer(s)
|
   Ast Reqd|       632.82        |        894.99          |        632.82
|
       Prov|       804.57        |       1384.43          |        804.57
|
   Ld (mm) |       752.2         |       1175.3           |        752.2
```

```
|
--------------------------------------------------------------------------
==========================================================================
```

D9. Japanese Codes

D9.A. Japanese Codes - Concrete Design per 1991 AIJ

STAAD.Pro is capable of performing concrete design based on the Japan code AIJ 1991 *Architectural Institute of Japan Standards for Structural Calculation of Steel Reinforced Concrete Structures*. Design for a member involves calculation of the amount of reinforcement required for the member. Calculations are based on the user specified properties and the member forces obtained from the analysis. In addition, the details regarding placement of the reinforcement on the cross section are also reported in the output.

D9.A.1 Section Types for Concrete Design

The following types of cross sections for concrete members can be designed:

- For Beams — Prismatic (Rectangular and Square)
- For Columns — Prismatic (Rectangular, Square, and Circular)

D9.A.2 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the `MEMBER PROPERTY` command. These are the D (YD) and b (ZD) dimensions for rectangular or square cross sections and the D (YD) for circular cross sections.

The following is an example the required input:

```
UNIT MM
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 450. ZD 250.
11 13 PR YD 350.
```

In the above input, the first set of members are rectangular (450 mm depth and 250 mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with a 350 mm diameter.

Caution: It is absolutely imperative that you do *not* provide the cross section area (AX) as an input.

D9.A.3 Slenderness Effects and Analysis Considerations

Slenderness effects are extremely important in designing compression members. Slenderness effects result in additional forces being exerted on the column over and above those obtained from the elastic analysis. There are two options by which the slenderness effects can be accommodated.

The first option is to compute the secondary moments through an exact analysis. Secondary moments are caused by the interaction of the axial loads and the relative end displacements of a member. The axial loads and joint displacements are first determined from an elastic stiffness analysis and the secondary moments are then evaluated.

The second option is to approximately magnify the moments from the elastic analysis and design the column for the magnified moment. It is assumed that the magnified moment is equivalent to the total moment comprised of the sum of primary and secondary moments.

STAAD provides facilities to design according to both of the above methods. To utilize the first method, the command `PDELTA ANALYSIS` must be used instead of `PERFORM ANALYSIS` in the input file. The user must note that to take advantage of this analysis, all the combinations of loading must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the P-delta analysis based on the deflections. Also, note that the proper factored loads (like 1.5 for dead load etc.) should be provided by the user. STAAD does not factor the loads automatically. The second method mentioned above is utilized by providing the magnification factor as a concrete design parameter (See the parameter `MMAG` in D9.A.7 Design Parameters (on page 1716)). The column is designed for the axial load and total of primary and secondary biaxial moments if the first method is used and for the axial load and magnified biaxial moments if the second method is used.

D9.A.4 Beam Design

Beams are designed for flexure, shear, and torsion. The program considers 12 equally spaced divisions of the beam member. However this number can be redefined by `NSECTION` parameter. All these sections are designed for flexure, shear and torsion for all load cases. The results include design results for most critical load case.

Example

```
UNIT  KG CM
START  CONCRETE DESIGN
CODE  JAPAN
FYMAIN  SRR295 ALL
FYSEC  SRR295 ALL
FC  350 ALL
CLEAR  2.5 MEM 2 TO 6
TRACK  1.0 MEMB 2 TO 9
DESIGN  BEAM 2 TO 9
END  CONCRETE DESIGN
```

D9.A.4.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of section properties provided by the user. Program first try to design the section for g = 0 and pt = balanced reinforcement ratio. If allowable moment is lower than the actual moment program increases g value for same pt and checks the satisfactory conditions. If conditions are not satisfied this procedure continues until g reaches to 1.0 and then pt value is increased keeping g = 1.0. This procedure continues until pt reaches to its maximum value(2 %). But if the allowable moment for pt = maximum value and g = 1.0 is lower than the actual moment the program gives message that the section fails.

This program automatically calculates the Bar size and no. of bars needed to design the section. It arranges the bar in layers as per the requirements and recalculate the effective depth and redesign the sections for this effective depth.

Notes

a. Beams are designed for MZ only. The moment MY is not considered in flexure design
b. MMAG parameter can be used to increase design moment
c. 1.4 cm. is added to the clear cover to take stirrup size into consideration for flexure design.

d. STAAD beam design procedure is based on the local practice and considering the fact that Japan is a high seismic zone area.

D9.A.4.2 Design for Shear

The Design Shear value, Q_D, is evaluated for the beam. The update effective depth is used to then calculate the allowable shear stress. The allowable shear stress of concrete, f_s, is automatically calculated from design load type (permanent or temporary) and given density of concrete. The program then calculates the required bar size, a_w, and spacing of stirrups. The reinforcement ratio for the stirrup, p_w, is calculated for design Bar size and stirrup pitch and all the necessary checking is done.

For seismic loading it is needed to increase shear force ≥ 1.5 times the actual value and this can be done utilizing the Design Shear Modification factor, k (`SMAG` parameter) without changing the Design Moment.

Notes:

a. Stirrups are always assumed to be 2-legged
b. Governing density to determine Light weight or Normal Weight Concrete is 2.3 kg/sq. cm

D9.A.4.3 Design for Torsion

Torsion design for beam is optional. If the `TORSION` parameter value is 1.0, the program will design the assigned beam(s) for torsion. The program first checks whether extra reinforcement is needed for torsion or not. If additional reinforcement is needed, this additional pt is added to flexure pt and additional Pw is added to shear design Pw.

D9.A.5 Column Design

Columns are designed for axial force, MZ moment, MY moment, and shear force. Both the ends of the members are designed for all the load cases and the loading which produces largest amount of reinforcement is called as critical load. If Track 0 or Track 1 is used, design results will be printed for critical load only. But if Track 2 is used, you can get detailed design results of that member. The value of Pt needed for minimum axial force, maximum axial force, maximum MZ, maximum MY among all the load cases for both the ends will be printed. If the `MMAG` parameter is used, the column moments will be multiplied by that value. If the `SMAG` parameter is used, column shear force will be multiplied by that value.

Column design is done for Rectangular, Square and Circular sections. For rectangular and square sections Pt value is calculated separately for MZ and MY, while for circular sections Pg value is calculated for MZ and MY separately.

Column design for biaxial moments is optional. If the `BIAXIAL` parameter value is 1.0, the program will design the column for biaxial moments. Otherwise column design is always uniaxial.

Steps involved:

1. Depending on the axial force zone is determined for Pt = 0.0 .
2. If the column is in "zone A", design is performed by increasing Pt and checking allowable load for that known Pt and known actual eccentricity of the column.
3. If the column is in "zone B" or in "zone C", xn is calculated for given P and Pt and checking is done for allowable moment, if allowable moment is less than the actual moment, program increases Pt and this procedure continues until the column design conditions are satisfied or the column fails as the required Pt is higher than Pt maximum value.
4. If the column is in tension, design is done by considering allowable tensile stress of steel only.
5. If biaxial design is requested program solve the following interaction equation

6. where, a = 1.0+1.66666666 ´ (ratio-0.2), ratio = P/Pcap & 1.0 £ a £ 2.0, Mycap, Mzcap & Pcap represents section capacity
7. If the interaction equation is not satisfied program increases Pt and calculates Pcap, Mycap and Mzcap and solve the interaction equation again and this process continues until the eqn. is satisfied or the column fails as Pt exceeds its maximum limit.
8. If biaxial design is not requested program assumes that interaction equation is satisfied (if uniaxial design is performed successfully).
9. If the interaction equation is satisfied program determines bar size and calculates no. of bars and details output is written.

D9.A.5.1 Example

```
UNIT KGS CMS
START CONCRETE DESIGN
CODE JAPAN
FYMAIN SRR295 ALL
FC 210 ALL
CLEAR 2.5 MEMB 2 TO 6
DESIGN COLUMN 2 TO 6
END CONCRETE DESIGN
```

D9.A.6 Slab and Wall Design

To design a slab or a wall, it must first be modeled using finite elements and analyzed. The command specifications are in accordance with Chapter 2 and Chapter 6 of the Technical Reference Manual.

Elements are designed for the moments Mx and My. These moments are obtained from the element force output (see Chapter 2 of the Technical Reference Manual). The reinforcement required to resist the Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist the My moment is denoted as transverse reinforcement.

The longitudinal bar is the layer closest to the exterior face of the slab or wall. The following parameters are those applicable to slab and wall design:

1. `FYMAIN` — Yield stress for reinforcing steel - transverse and longitudinal.
2. `FC`— Concrete grade
3. `CLEAR` — Distance from the outer surface of the element to the edge of the bar. This is considered the same on both top and bottom surfaces of the element.
4. `MINMAIN` — Minimum required size of longitudinal/transverse reinforcing bar

The other parameters shown in D9.A.7 Design Parameters (on page 1716) are not applicable to slab or wall design.

D9.A.7 Design Parameters

The program contains a number of parameters which are needed to perform the design. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 10A.1 contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as centimeters and Kilograms before performing the concrete design.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 174: Japanese Concrete Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as JAPAN. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BIAXIAL	0.0	Value to define biaxial or uniaxial design type for Column 0. uniaxial design only 1. design for biaxial moments
CLEAR	3.0 cm (beam) 4.0 cm (Column)	Clear cover for Beam or clear side cover for column.
DEPTH	YD	Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.
EFACE	0.0	Face of support location at end of beam. (Note: Both SFACE & EFACE are input as positive numbers).
FC	210 Kg/cm^2	Compressive Strength of Concrete.
FYMAIN	SR235	Steel grade. Acceptable values for steel grade and their associated yield stress values are shown in the following table. Program automatically calculates yield stress value depending on design load type (permanent or temporary).
FYSEC	SR235	Same as FYMAIN except this is for secondary steel.
LONG	0.0	Value to define design load type 0. Permanent Loading 1. Temporary Loading

Parameter Name	Default Value	Description
MAXMAIN	41.0 cm	Maximum main reinforcement bar size
MAXSEC	41.0 cm	Maximum secondary reinforcement bar size.
MINMAIN	10 mm	Minimum main reinforcement bar size.
MINSEC	10 mm	Minimum secondary reinforcement bar size.
MMAG	1.0	Design moment magnification factor
NSECTION	12	Number of equally-spaced sections to be considered in finding critical moments for beam design.
REINF	0.0	Tied Column. A value of 1.0 will mean spiral.
SFACE	0.0	Face of support location at start of beam.
SMAG	1.0	Design shear magnification factor
TORSION	0.0	Value to request for torsion design for beam 0. torsion design not needed 1. torsion design needed
TRACK	0.0	Beam Design: 0. Critical section design results. 1. Five section design results & design forces. 2. 12 section design results & design forces. Column Design: 1. Detail design results for critical load case only. 2. Design results for minimum P, maximum P, maximum MZ and maximum MY among all load cases for both ends.

Parameter Name	Default Value	Description
WIDTH	ZD	Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES.

Table 175: Table of permissible Steel Grades and associated Yield Stresses for FYMAIN and FYSEC parameters

Steel Grade	Long Term Loading		Short Term Loading	
	Tension & Compression	Shear Reinforcement	Tension & Compression	Shear Reinforcement
SR235 SRR235 SDR235	1600	1600	2400	2400
SR295 SRR295	1600	2000	3000	3000
SD295A SD295B SDR295	2000	2000	3000	3000
SDR345 SD345	2200 (2000)	2000	3500	3500
SD390	2200 (2000)	2000	4000	4000

D9.B. Japanese Codes - Steel Design per 2005 AIJ

STAAD.Pro is capable of performing steel design based on the Japanese code AIJ 2005 *Design Standard for Steel Structures*.

D9.B.1 General

This section presents some general statements regarding the implementation of the "Architectural Institute of Japan" (AIJ) specifications for structural steel design (2005 edition) in STAAD. The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceedance of the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. Users are recommended to adopt the following steps in performing the steel design:

- Specify the geometry and loads and perform the analysis.
- Specify the design parameter values if different from the default values.
- Specify whether to perform code checking or member selection.

The method for calculating allowable bending stress was updated for the AIJ 2005 from the AIJ 2002 code. All other allowable limit states, analysis and design methods, etc., remain unchanged. Refer to the D9.C. Japanese Codes - Steel Design per 2002 AIJ (on page 1728) for additional details.

D9.B.2 Member Capacities

Member design and code checking per AIJ 2005 are based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. The basic measure of member capacities are the allowable stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD.Pro for calculating such capacities.

D9.B.2.1 Design Capabilities

All types of available shapes like H-Shape, I-Shape, L-Shapes, CHANNEL, PIPE, TUBE, etc. can be used as member property and STAAD.Pro will automatically adopt the design procedure for that particular shape if Steel Design is requested. STEEL TABLE available within STAAD.Pro or UPTABLE facility can be used for member property.

D9.B.2.2 Methodology

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as required by AIJ specifications. The design procedure consist of following three steps.

1. Calculation of sectional properties

   The program extracts section properties cross sectional area, A, moment of inertia about Y and Z axes, I_{yy} and I_{zz}, and the St. Venant torsional constant, J, from the built-in steel tables. The program then calculates the elastic section moduli, Z_z and Z_y, torsional section modulus, Z_x, and radii of gyration, i_y and i_z, using the appropriate formulas.
2. Calculation of actual and allowable stresses

   Program calculates actual and allowable stresses by following methods:

   i. Axial Stress:

      Actual tensile stresses,

      $$F_T = \text{force} / (A \times)$$

      where

      NSF = Net Section Factor for tension input as a design parameter

      Actual compressive stress , $F_C = force\ /\ A$

      Allowable tensile stress, f_t

      = FYLD / 1.5 (For Permanent Case)
      = FYLD (For Temporary Case)

      where

FYLD = Yield stress input as a design parameter

Allowable compressive stress, f_c

$$f_c = \begin{cases} \dfrac{\left[1 - 0.4\left(\frac{\lambda}{\Lambda}\right)^2\right]F}{\nu} & when \quad \lambda \le \Lambda \\ \dfrac{0.277F}{\left(\frac{\lambda}{\Lambda}\right)^2} & when \quad \lambda > \Lambda \end{cases}$$

$= f_c \; x \; 1.5$ (for Temporary case)

where

Λ = $\sqrt{\dfrac{\pi^2 E}{0.6F}}$

ν = $\dfrac{3}{2} + \dfrac{2}{3}\left(\dfrac{\lambda}{\Lambda}\right)^2$

λ = maximum slenderness, considering both principal axis

E = Modulus of elasticity of steel (Young's Modulus)

Actual torsional stress, $f_t = torsion \; / \; Z_x$

where

Z_x = J / max(t_f, t_w)

t_f = flange thickness

t_w = web thickness

ii. Bending Stress:

Actual bending stress for My for compression:

$(F_{bcy}) = M_y / Z_{cy}$

Actual bending stress for Mz for compression

$(F_{bcz}) = M_z / Z_{cz}$

Actual bending stress for My for tension

$(F_{bty}) = M_y / Z_{ty}$

Actual bending stress for Mz for tension

$(F_{btz}) = M_z / Z_{tz}$

where

Z_{cy}, Z_{cz} = elastic section modulus for compression due to bending about the y and z axes, respectively

Z_{ty}, Z_{tz} = elastic section modulus for tension due to bending about the y and z axes, respectively

Allowable bending stress for M_y

$(f_{bcy}) = f_t$

Allowable bending stress for M_z

When $\lambda_b \le {}_p\lambda_b$, $f_b = F/\nu$

When ${}_p\lambda_b < \lambda_b \le {}_e\lambda_b$,

$$f_b = \frac{F\left(1 - 0.4\frac{\lambda_b - {}_p\lambda_b}{{}_e\lambda_b - {}_p\lambda_b}\right)}{\nu}$$

When ${}_e\lambda_b < \lambda_b$,

$$f_b = \frac{1}{\lambda_b^2}\frac{F}{2.17}$$

where

λ_b = $\sqrt{M_y / M_e}$

${}_e\lambda_b$ = $1 / \sqrt{0.6}$

ν = $\frac{3}{2} + \frac{2}{3}\left(\frac{\lambda_b}{{}_e\lambda_b}\right)^2$

M_e = $C\sqrt{\frac{\pi^4 EI_y EI_w}{I_b^4} + \frac{\pi^2 EI_y GJ}{I_b^2}}$

${}_p\lambda_b$ = $0.6 + 0.3\left(\frac{M2}{M1}\right)$

or taken as the value of PLB if not 0

C = $1.75 + 1.05\,(M2 / M1) + 0.3\,(M2 / M1)^2 \le 2.3$

M1 = the larger of end moments about the major axis

M2 = the smaller of end moments about the major axi

Note: $M2/M1$ will be +ve for double curvature and -ve for single curvature.

For Temporary case, $f_{bcz} = 1.5\ x\ (f_{bcz}$ for Permanent case)

where

f_t = Allowable bending stress for M_y, f_{bty}

f_{bcz} = Allowable bending stress for M_z, f_{btz}

Note: The parameter CB can be used to specify a value for C directly.

iii. Shear Stress

Actual shear stresses are calculated by the following formula:

$$Q_y = F_y / A_{ww}$$

where

A_{ww} = web shear area = depth times web thickness

$$Q_z = F_z / A_{ff}$$

where

A_{ff} = flange shear area = 2/3 times total flange area

Allowable shear stress:

Permanent Loads: $f_s = (F_y/\sqrt{(3)})/\ 1.5$

Temporary Loads: $f_s = F_y\ /\ \sqrt{(3)}$

where

F_y = yield strength of steel, specified by the FYLD parameter.

3. Checking design requirements:

User provided RATIO value (default 1.0) is used for checking design requirements:

The following conditions are checked to meet the AIJ specifications. For all the conditions calculated value should not be more than the value of RATIO. If for any condition value exceeds RATIO, program gives the message that the section fails.

Conditions:

i. Axial tensile stress ratio = F_T / f_t
ii. Axial compressive stress ratio = F_C / f_c
iii. Combined compression & bending compressive ratio = F_C / f_c+F_{bcz}/f_{bcz}+F_{bcy}/f_{bcy}
iv. Combined compression & bending tensile ratio = $(F_{btz}+F_{bty}-F_C)$ / f_t
v. Combined tension & bending tensile ratio = $(F_T+F_{btz}+F_{bty})$ / f_t
vi. Combined tension & bending compressive ratio = $F_{bcz}/f_{bcz}+F_{bcy}/f_{bcy}- F_T/f_t$
vii. Shear stress ratio in Y = q_y / f_s
viii. Shear stress ratio in Z = q_z / f_s
ix. von Mises stress ratio (if the D9.B.4 Von Mises Stresses Check (on page 1727) were set to be checked) = $f_m/(k \cdot f_t)$

Note: All other member capacities (axial tension, axial compression, and shear) are calculated as for AIJ 2002. Refer to D9.C.5 Member Capacities (on page 1733)

Related Links

- *V.AIJ 2005 Check for MBG parameter* (on page 4329)
- *V.AIJ 2005 UPT Channel* (on page 4343)
- *V.AIJ 2005 UPT Double Angle* (on page 4350)
- *V.AIJ 2005 UPT General* (on page 4356)
- *V.AIJ 2005 UPT I* (on page 4362)
- *V.AIJ 2005 UPT Tee* (on page 4368)

D9.B.3 Design Parameters

You are allowed complete control over the design process through the use of parameters in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 176: 2005 Japanese Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as JAPANESE 2005 to invoke the AIJ 2005. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	1.0	Locations of design: 0.0) design only for end moments or those at locations specified by the `SECTION` command. 1.0) calculate moments at twelfth points along the beam.
CAN	0	Specifies the method used for deflection checks 0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see note a (on page 1726))
CB	0	C value from the AIJ code. Refer to D9.B.2 Member Capacities (on page 1720) - Bending Stress for how C is calculated and applied. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See note b (on page 1726))
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See note b (on page 1726))
DMAX	100 cm	Maximum allowable depth for member.
DMIN	0.0 cm	Minimum allowable depth for member.
FYLD	235 MPA	Yield strength of steel in Megapascal.
KY	1.0	K value in local y-axis. Usually, this is the minor axis.
KZ	1.0	K value in local z-axis. Usually, this is the major axis.
LY	Member Length	Length in local y-axis to calculate slenderness ratio.
LZ	Member Length	Same as above except in z-axis
MAIN	200	Allowable Slenderness Limit for Compression Member 0.0) check for slenderness using default value 1.0) suppress compression slenderness check Any value greater than 1 = Allowable KL/r in compression (up to 250)

Parameter Name	Default Value	Description
MBG	0	Specifies how to calculate the section modulus about the Z-Z axis for H-shape, I-shape, and channel sections when performing major axis bending checks: 0) Consider the flanges and the web 1) Consider only the flanges; the web is ignored for the calculation of the section modulus.
MISES	1	Von Mises check options: 0) Do not perform von Mises check. 1) Standard AIJ calculation. The direct stress σ_x is determined by calculating the stress at each of the corners of the section using signed forces and the appropriate elastic modulus. The magnitude of the maximum stress is used. 2) τ_y excludes torsion stresses. The same as option 1, but in the calculation of the shear stress, the torsional moment is excluded. 3) σ_x based on absolute forces, The direct stress σ_x is calculated using the absolute value of the force at the section divided by the minimum of the elastic section moduli for each axis. 4) σ_x based on absolute forces and τ_y excludes torsion stress. Same as option 3, but excluding the torsional moment when calculating the shear stress. For more details, refer to D9.B.4 Von Mises Stresses Check (on page 1727).
NSF	1.0	Net section factor for tension members.
PLB	0	Plastic critical slenderness ratio. If this is 0 (the default value), it will be calculated according to AIJ 2005 eqn. 5.12 or 5.14. Any other entered value will be used as the value of p-lambda-b.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SLF	1	Slender section design option: 0) Slender sections are not designed - an error message will be presented that design of these sections is not supported 1) Slender sections designed with unreduced profile (i.e., full profile) - an error message will be presented that this may be an unconservative design

Parameter Name	Default Value	Description
TMAIN	400	Allowable Slenderness Limit for Tension Member 0.0) check for slenderness using default value 1.0) suppress slenderness check Any value greater than 1 = Allowable KL/r in tension.
TRACK	0.0	Level of output detail: 0) Produce design summary only 1) Produce intermediate detailed output 2) Produce maximum detailed output
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.

D9.B.3.1 Notes

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement..

If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at DJ2 or the end node of the member.

Compute Delta = $\sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}$

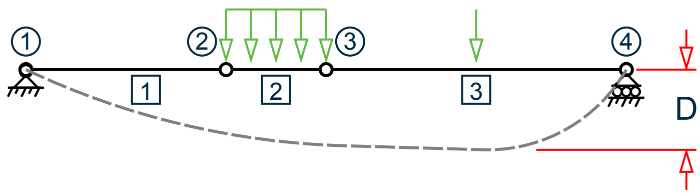
Compute Length = distance between DJ1 and DJ2 or, between start node and end node, as the case may be.

Then, if CAN is specified a value 1, dff = L/Delta

Ratio due to deflection = DFF/dff

b. If CAN = 0, the “Deflection Length” is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Thus, for all three members here, DJ1 should be 1 and DJ2 should be 4.



D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

c. If DJ1 and DJ2 are not used, “Deflection Length” will default to the member length and local deflections will be measured from original member line.

d. The above parameters may be used in conjunction with other available parameters for steel design.

D9.B.4 Von Mises Stresses Check

The von Mises stress equation shown below, which is modified for beam elements based on the corresponding equation in AIJ steel design code (both 2002 and 2005 editions of AIJ), indicates that the left-hand side in the equation should be less than unity. These checks are performed at locations indicated by the BEAM parameter.

Note: As with other design checks, the unity check value can be modified by use of the RATIO parameter.

The von Mises stresses are evaluated and checked per AIJ clause 5.16 as follows:

$$\frac{\sqrt{\sigma_x^2 + 3\tau_{xy}^2}}{k \times f_t} < 1.0$$

where

σ_x = Longitudinal stress in beam element. The following equation is used when the MISES parameter is set to 1 or 2. This is performed multiple times, once in each corner with the appropriate sign of the moment and value of elastic modulus. The largest stress is then used.

$$= \frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}$$

When the MISES parameter is set to 3 or 4, then the longitudinal stress is calculated once using the smalles elastic modulus for each axis as follows.

$$= \left|\frac{F_x}{A_x}\right| + \left|\frac{M_y}{Z_y}\right| + \left|\frac{M_z}{Z_z}\right|$$

F_x = axial force

M_y = bending moment about y-axis

M_z = bending moment about z-axis

A_x = cross-sectional area

Z_y = section modulus about y-axis

Z_z = section modulus about z-axis

τ_{xy} = shear stress in the beam. When the MISES parameter is set to 1 or 3, this includes torsion stresses:

$$= \left| \frac{M_x}{Z_x} \right| + \sqrt{\left| \frac{F_y}{A_y} \right|^2 + \left| \frac{F_z}{A_z} \right|^2}$$

When the `MISES` parameter is set to 2 or 4, the torsion stresses are excluded:

$$= \sqrt{\left| \frac{F_y}{A_y} \right|^2 + \left| \frac{F_z}{A_z} \right|^2}$$

M_x	=	torsional moment.
F_y	=	shear stress in the y direction
F_z	=	shear stress in the z direction
Z_x	=	torsional section modulus
D_x	=	depth of the member
I_x	=	torsional constant
A_y	=	effective shear area in the y direction
A_z	=	effective shear area in the z direction
f_t	=	allowable tensile stress
k	=	loading duration factor as specified by the `TMP` parameter:

1.0 for permanent
1.5 for temporary

In the STRESSES output category, stress value of (numerator of the von Mises stress equation) is output as the value of fm. Along with slenderness ratios, stresses, and deflections, von Mises stress equation is checked. When its left-hand side yields the maximum ratio value, it is printed as RATIO and "VON MISES" is printed as CRITICAL COND.

Related Links

- *V.AIJ 2005 Check for MISES parameter* (on page 4335)
- *V.AIJ 2005 Check for MISES parameter 2* (on page 4339)

D9.C. Japanese Codes - Steel Design per 2002 AIJ

STAAD.Pro is capable of performing steel design based on the Japanese code AIJ 2002 *Design Standard for Steel Structures*.

D9.C.1 General

The design philosophy and procedural logistics are based on the principles of elastic analysis and allowable stress design. Facilities are available for member selection as well as code checking. Two major failure modes are recognized: failure by overstressing and failure by stability considerations. The following sections describe the salient features of the design approach.

Members are proportioned to resist the design loads without exceedance of the allowable stresses or capacities and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements and the stability criteria. Users are recommended to adopt the following steps in performing the steel design:

- Specify the geometry and loads and perform the analysis.
- Specify the design parameter values if different from the default values.
- Specify whether to perform code checking or member selection.

D9.C.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and in using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

D9.C.3 Member Property Specifications

For specification of member properties of standard Japanese steel shapes, the steel section library available in STAAD may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Members properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D9.C.4 Built-in Japanese Steel Section Library

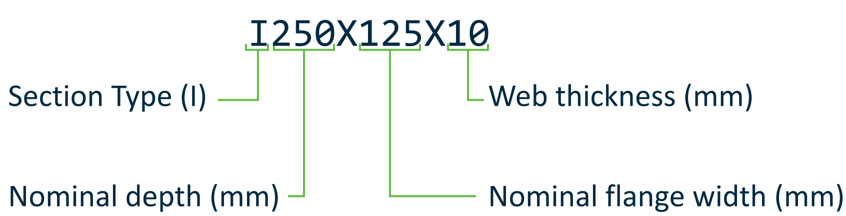
The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, these properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered for these members during the analysis. An example of member property specification in an input file is provided at the end of this section.

A complete listing of the sections available in the built-in steel section library may be obtained using the tools of the graphical user interface.

Following are the descriptions of different types of sections.

D9.C.4.1 I shapes

I shapes are specified in the following way:

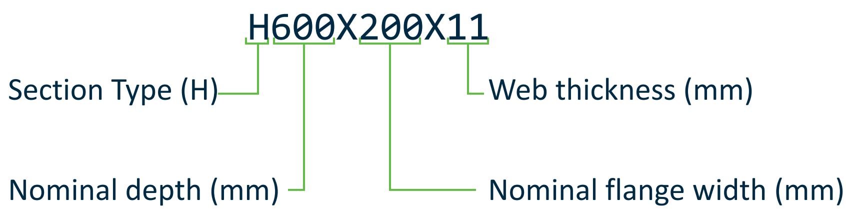


Note: While specifying the web thickness, the portion after the decimal point should be excluded.

```
1 TO 9 TA ST I300X150X11
12 TO 15 TA ST I350X150X9
```

D9.C.4.2 H shapes

H shapes are specified as follows:

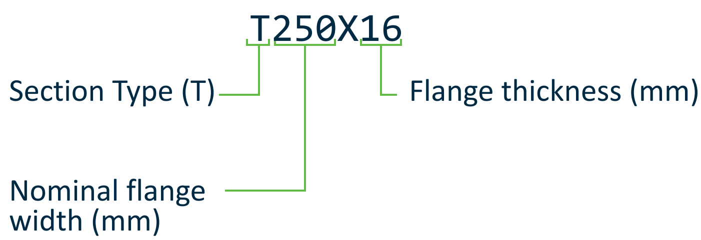


Note: While specifying the web thickness, the portion after the decimal point should be excluded.

```
1 TO 8 TA ST H200X100X4
13 TO 17 TA ST H350X350X12
```

D9.C.4.3 T shapes

T shapes are specified as follows:

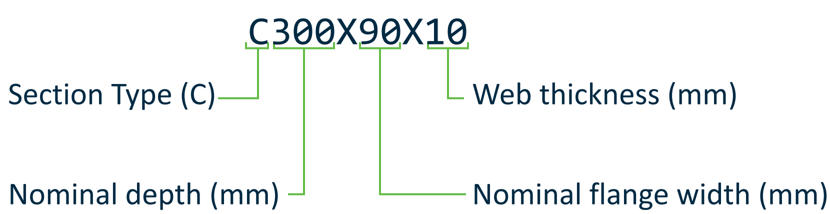


Note: While specifying the web thickness, the portion after the decimal point should be excluded.

```
20 TO 25 TA ST T250X19
```

D9.C.4.4 Channels

Channel sections are specified as follows.




```
25 TO 34 TA ST C125X65X6
46 TO 49 TA ST C200X90X8
```

D9.C.4.5 Double Channels

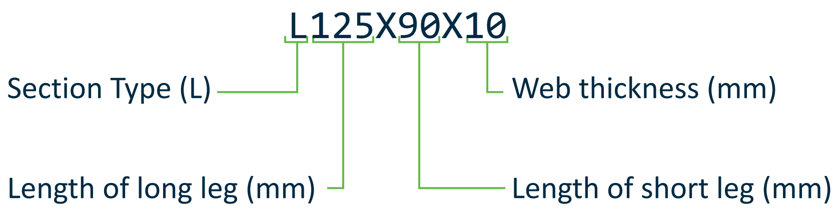
Back to back double channels, with or without a spacing in between them, are available. The letter D in front of the section name is used to specify a double channel. Front-to-front double channels are similarly added by adding FR in front of the section name.

```
17 TO 27 TA D C300X90X10
45 TO 76 TA D C250X90X11 SP 2.0
28 TO 30 TA FR C200X90X8 SP 2.5
```

In the above commands, members 17 to 27 are a back-to-back double channels C300X90X10 with no spacing in between. Members 45 to 76 are a double channels C250X90X11 with a spacing of 2 length units. Members 28 to 30 are front-to-front double channels C200X90X8 with a spacing of 2.5 length units.

D9.C.4.6 Angles

Two types of specification may be used to describe an angle. The standard angle specification is as follows.



The letter L (signifying that the section is an angle) is followed by the length of the legs and then the thickness of the leg, all in millimeters. The word ST signifies that the section is a standard angle meaning that the major principal axis coincides with the local YY axis specified in G.4.2 Local Coordinate System (on page 2088) .

```
1 4 TA ST L150X90X9
```

If the minor principal axis coincides with the local YY axis specified in Chapter 2 of the User's Manual, the word RA (Reverse Angle) should be used instead of ST as shown below.

```

7 TO 23 TA RA L90X75X9
```

D9.C.4.7 Double angles

Short leg back-to-back and long leg back-to-back double angles may be specified by using the words SD or LD in front of the angle size. In the case of an equal angle, either SD or LD will serve the purpose. The spacing between the angles may be specified by using the word SP after the angle size followed by the value of the spacing.

```
8 TO 25 TA SD L100X65X7 SP 2.0
36 TO 45 TA LD L300X90X11 SP 3.0
```

The first example indicates a short legs back-to-back double angle comprised of 100X65X7 angles separated by 2 length units. The latter is a long legs back-to-back double angle comprised of 300X90X11 angles separated by 3 length units.

D9.C.4.8 Tubes

Tube names are input by their dimensions. For example,

```
6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5
```

is a tube that has a height of 8 length units, width of 6 length units and a wall thickness of 0.5 length units. Only code checking, no member selection can be performed on TUBE sections.

D9.C.4.10 Pipes (General Pipe sections)

Circular hollow sections defined by JIS G3444:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* as general pipe sections are specified as shown in the following example.

```
1 TO 9 TA ST PIPE PIP267.4x7.0
```

specifies a pipe with outside diameter of 267.0 mm and a thickness of 7.0 mm. Only code checking, no member selection, can be performed on PIPE sections.

D9.C.4.11 Circular Hollow sections

Circular hollow sections defined by JIS G3475:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* as Architectural pipe sections are specified as shown in the following example.

```
1 TO 9 TA ST PIPE CHS660.4x16
```

specifies a pipe with outside diameter of 660.4 mm and a thickness of 16.0 mm. Only code checking, no member selection, can be performed on CHS sections.

D9.C.4.12 Rectangular Hollow sections

Rectangular hollow sections defined by JIS G3466:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* are specified as shown in the following example.

```
1 TO 9 TA ST PIPE RHS200x100x12
```

specifies a tube with a depth of 200 mm, a width of 100 mm, and a thickness of 12 mm. Only code checking, no member selection, can be performed on RHS sections.

D9.C.4.13 Square Hollow sections

Square hollow sections defined by JIS G3466:2005 *Design Standard for Steel Structures - Based on Allowable Stress Concept* are specified as shown in the following example.

```
1 TO 9 TA ST PIPE SHS200xs00x12
```

specifies a square tube with a width of 200 mm and a thickness of 12 mm. Only code checking, no member selection, can be performed on SHS sections.

D9.C.4.14 Example

The following is a sample input file containing Japanese shapes.

```
STAAD  SPACE
UNIT  KIP FEET
JOINT  COORD
1  0 0 0 12 11 0 0
MEMB  INCIDENCE
1  1 2 11
UNIT  INCH
MEMBER  PROPERTY JAPANESE
*  H-SHAPE
1  TA ST H200X100X4
*  I SHAPE
2  TA ST I250X125X10
*  T SHAPE
3  TA ST T200X19
*  CHANNEL
4  TA ST C125X65X6
*  DOUBLE CHANNEL
5  TA D C200X90X8
*  REGULAR ANGLE
6  TA ST L100X75X7
*  REVERSE ANGLE
7  TA RA L90X75X9
```

```
*  DOUBLE ANGLE - LONG LEG BACK TO BACK
8  TA LD L125X75X7 SP 2.0
*  DOUBLE ANGLE - SHORT LEG BACK TO BACK
9  TA SD L300X90X11 SP 1.5
*  TUBE
10  TA ST TUBE DT 3.0 WT 2.5 TH 0.25
*  PIPE
11  TA ST PIPE OD 3.0 ID 2.5
PRINT  MEMBER PROPERTIES
FINISH
```

D9.C.5 Member Capacities

Member design and code checking per AIJ 2002 are based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. The basic measure of member capacities are the allowable stresses on the member under various conditions of applied loading such as allowable tensile stress, allowable compressive stress etc. These depend on several factors such as cross sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Explained here is the procedure adopted in STAAD for calculating such capacities.

D9.C.5.1 Design Capabilities

All types of available shapes such as H-Shape, I-Shape, L-Shapes, channel, pipe, tube, etc. can be used as member property and STAAD.Pro will automatically adopt the design procedure for that particular shape if Steel Design is requested. The steel tables available within STAAD.Pro or user provided table (UPTABLE) can be used for member properties.

D9.C.5.2 Methodology

For steel design, STAAD.Pro compares the actual stresses with the allowable stresses as required by AIJ specifications. The design procedure consist of following three steps.

1. Calculation of sectional properties

   The program extracts section properties cross sectional area, A, the moment of inertia about Y and Z axes, I_{yy} and I_{zz}, and the St. Venant torsional constant, J, from the built-in steel tables. The program then calculates the elastic section moduli, Z_z and Z_y, torsional section modulus, Z_x, and radii of gyration, i_y and i_z, using the appropriate formulas.

   For I shapes, H shapes, and channel sections, the program calculates the radius of gyration needed for bending using following formula:

   $$i = \sqrt{I_i / A_i}$$

2. Calculation of actual and allowable stresses

   Allowable stresses for structural steel under permanent loading shall be determined on the basis of the values of F given in the following table.

Table 177: Table: Values of F (N/mm²)

	Steel for Construction Structures		Steel for General Structures	Steel for Welded Structures					
Thickness	**SN400** **SNR400** **STKN400**	**SN490** **SNR490** **STKN490**	**SS400** **STK400** **STKR400** **SSC400** **SWH400**	**SS490**	**SS540**	**SM400** **SMA400**	**SM490** **SM490 Y** **SMA490** **STKR490** **STK490**	**SM520**	**SM570**
t≤ 40	235	325	235	275	375	235	325	355	400
40< t ≤ 100	215	295	215	255	-	215	295*	335	400

* F = 325 N/mm² when t > 75mm

Note: In checking members for temporary loading be the combination of stresses described in Chap.3, allowable stresses specified in this chapter may be increases by 50%

Program calculates actual and allowable stresses by following methods:

i. Axial Stress:

Actual tensile stresses,

F_T = force / (A ×)

where

`NSF` = Net Section Factor for tension input as a design parameter

Actual compressive stress , $F_C = force\ /\ A$

Allowable tensile stress, f_t

= `FYLD` / 1.5 (For Permanent Case)
= `FYLD` (For Temporary Case)

where

`FYLD` = Yield stress input as a design parameter

Allowable compressive stress, f_c

$$f_c = \begin{cases} \dfrac{\left[1 - 0.4\left(\dfrac{\lambda}{\Lambda}\right)^2\right]F}{\nu} when \lambda \le \Lambda \\ \dfrac{0.277F}{\left(\dfrac{\lambda}{\Lambda}\right)^2} when \lambda > \Lambda \end{cases}$$

$= f_c \times 1.5$ (for Temporary case)

where

Λ = $\sqrt{\frac{\pi^2 E}{0.6F}}$

ν = $\frac{3}{2} + \frac{2}{3}\left(\frac{\lambda}{\Lambda}\right)^2$

Actual torsional stress, $f_t = torsion\ /\ Z_x$

where

Z_x = J / max(t_f, t_w)

t_f = flange thickness

t_w = web thickness

ii. Bending Stress:

Actual bending stress for My for compression

$(F_{bcy}) = M_y / Z_{cy}$

Actual bending stress for Mz for compression

$(F_{bcz}) = M_z / Z_{cz}$

Actual bending stress for My for tension

$(F_{bty}) = M_y / Z_{ty}$

Actual bending stress for Mz for tension

$(F_{btz}) = M_z / Z_{tz}$

where

Z_{cy}, Z_{cz} = elastic section modulus for compression due to bending about the y and z axes, respectively

Z_{ty}, Z_{tz} = elastic section modulus for tension due to bending about the y and z axes, respectively

Note: The web is ignored in the calculation of Z_z for H-shape, I-shape, and channel sections when the MBG parameter = 1.

Allowable bending stress for M_y

$(f_{bcy}) = f_t$

Allowable bending stress for M_z

$(f_{bcz}) = \{ 1 - .4 \times (lb / i)^2 / (C \lambda^2)\}$ ft max

$= 89{,}000/ (lb \times h / A_f)$

For Temporary case, f_{bcz} = 1.5 x (f_{bcz} for Permanent case)

where

C = $1.75 - 1.05\ (M2\ /\ M1) + 0.3\ (M2\ /\ M1)^2$

Allowable bending stress for M_y, $f_{bty} = f_t$

Allowable bending stress for M_z, $f_{btz} = f_{bcz}$

Note: The parameter CB can be used to specify a value for C directly.

iii. Shear Stress

Actual shear stresses are calculated by the following formula:

$q_y = Q_y / A_{ww}$

where

A_{ww} = web shear area = product of depth and web thickness

$q_z = Q_z / A_{ff}$

where

A_{ff} = flange shear area = 2/3 times total flange area

f_s = Allowable shear stress, $F_s / 1.5$, $F_s = F / \sqrt{(3)}$

3. Checking design requirements:

   User provided RATIO value (default 1.0) is used for checking design requirements

   The following conditions are checked to meet the AIJ specifications. For all the conditions calculated value should not be more than the value of RATIO. If for any condition value exceeds RATIO, the program gives the message that the section fails.

   Conditions:

   i. Axial tensile stress ratio = F_T / f_t
   ii. Axial compressive stress ratio = F_C / f_c
   iii. Combined compression & bending compressive ratio = $F_C / f_c + F_{bcz}/f_{bcz} + F_{bcy}/f_{bcy}$
   iv. Combined compression & bending tensile ratio = $(F_{btz} + F_{bty} - F_C) / f_t$
   v. Combined tension & bending tensile ratio = $(F_T + F_{btz} + F_{bty}) / f_t$
   vi. Combined tension & bending compressive ratio = $F_{bcz}/f_{bcz} + F_{bcy}/f_{bcy} - F_T/f_t$
   vii. Shear stress ratio in Y = q_y / f_s
   viii. Shear stress ratio in Z = q_z / f_s
   ix. von Mises stress ratio (if the D9.C.10 Von Mises Stresses Check (on page 1742) were set to be checked) = $f_m/(k \cdot f_t)$

D9.C.5.4 Allowable stress for Axial Tension

Allowable axial stress in tension is calculated per section 5.1 (1) of the AIJ code. In members with axial tension, the tensile load must not exceed the tension capacity of the member. The tension capacity of the member is calculated on the basis of the member area. STAAD calculates the tension capacity of a given member based on a user supplied net section factor (NSF-a default value of 1.0 is present but may be altered by changing the input value, see Table 8B.1) and proceeds with member selection or code checking.

D9.C.5.5 Allowable stress for Axial Compression

The allowable stress for members in compression is determined according to the procedure of section 5.1 (3). Compressive resistance is a function of the slenderness of the cross-section (Kl/r ratio) and the user may control the slenderness value by modifying parameters such as KY, LY, KZ and LZ. In the absence of user provided values for effective length, the actual member length will be used. The slenderness ratios are checked against the permissible values specified in Chapter 11 of the AIJ code.

D9.C.5.6 Allowable stress for Bending

The permissible bending compressive and tensile stresses are dependent on such factors as length of outstanding legs, thickness of flanges, unsupported length of the compression flange (UNL, defaults to member length) etc. The allowable stresses in bending (compressive and tensile) are calculated as per the criteria of Clause 5.1 (4) of the code.

D9.C.5.7 Allowable stress for Shear

Shear capacities are a function of web depth, web thickness etc. The allowable stresses in shear are computed according to Clause 5.1 (2) of the code.

D9.C.6 Combined Loading

For members experiencing combined loading (axial force, bending and shear), applicable interaction formulas are checked at different locations of the member for all modeled loading situations. Members subjected to axial tension and bending are checked using the criteria of clause 6.2. For members with axial compression and bending, the criteria of clause 6.1 is used.

D9.C.7 Design Parameters

You are allowed complete control over the design process through the use of parameters in the following table. These parameters communicate design decisions from the engineer to the program. The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of the situation, some or all of these parameter values may have to be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 178: 2002 Japanese Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `JAPANESE 2002` to invoke the AIJ 2002. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1.0	Locations of design: 0.0) design only for end moments or those at locations specified by the `SECTION` command. 1.0) calculate moments at twelfth points along the beam.
CAN	0	Specifies the method used for deflection checks 0) deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1) deflection check based on the principle that maximum deflection is of the cantilever type (see note a (on page 1740))
CB	0	C value from the AIJ code. Refer to D9.C.5 Member Capacities (on page 1733) Bending Stress for how C is calculated and applied. Use 0.0 to direct the program to calculated Cb. Any other value be used in lieu of the program calculated value.

Parameter Name	Default Value	Description
DFF	None(Mandatory for deflection check)	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See note b (on page 1740))
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See note b (on page 1740))
DMAX	100 cm	Maximum allowable depth for member.
DMIN	0.0 cm	Minimum allowable depth for member.
KY	1.0	K value in local y-axis. Usually, this is the minor axis.
KZ	1.0	K value in local z-axis. Usually, this is the major axis.
LY	Member Length	Length in local y-axis to calculate slenderness ratio.
LZ	Member Length	Same as above except in z-axis
FYLD	235 MPA	Yield strength of steel in Megapascal.
MAIN	200	Allowable Slenderness Limit for Compression Member 0.0) check for slenderness using default value 1.0) suppress compression slenderness check Any value greater than 1 = Allowable KL/r in compression (up to 250)
MBG	0	Specifies how to calculate the section modulus about the Z-Z axis for H-shape, I-shape, and channel sections when performing major axis bending checks: 0) Consider the flanges and the web 1) Consider only the flanges; the web is ignored for the calculation of the section modulus.

Parameter Name	Default Value	Description
MISES	1	Von Mises check options: 0) Do not perform von Mises check. 1) Standard AIJ calculation. The direct stress σ_x is determined by calculating the stress at each of the corners of the section using signed forces and the appropriate elastic modulus. The magnitude of the maximum stress is used. 2) τ_y excludes torsion stresses. The same as option 1, but in the calculation of the shear stress, the torsional moment is excluded. 3) σ_x based on absolute forces, The direct stress σ_x is calculated using the absolute value of the force at the section divided by the minimum of the elastic section moduli for each axis. 4) σ_x based on absolute forces and τ_y excludes torsion stress. Same as option 3, but excluding the torsional moment when calculating the shear stress. For more details, refer to D9.C.10 Von Mises Stresses Check (on page 1742).
NSF	1.0	Net section factor for tension members.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
SLF	1	Slender section design option: 0) Slender sections are not designed - an error message will be presented that design of these sections is not supported 1) Slender sections designed with unreduced profile (i.e., full profile) - an error message will be presented that this may be an unconservative design
TMAIN	400	Allowable Slenderness Limit for Tension Member 0.0) check for slenderness using default value 1.0) suppress slenderness check Any value greater than 1 = Allowable KL/r in tension.
TRACK	0.0	Level of output detail: 0) Produce design summary only 1) Produce intermediate detailed output 2) Produce maximum detailed output

Parameter Name	Default Value	Description
UNF	1.0	Unsupported length provided as a fraction of actual member length used for lateral-torsional buckling calculation. **Note:** If both UNF and UNL parameters are specified, the effective length used is UNF×UNL.
UNL	Member Length	Unsupported length for calculating allowable bending stress. Used for the lateral-torsional buckling calculation. Value should be in the current units of length.
YNG	0	Method for evaluating Young's modulus, E, for equation 5.8: 0) Use equation 5.8 from *Design Standard for Steel Structures* 1) Use equation SSB-1.10 of JSME

D9.C.7.1 Notes

a. When performing the deflection check, you can choose between two methods. The first method, defined by a value `0` for the `CAN` parameter, is based on the local displacement. Refer to TR.44 Printing Section Displacements for Members (on page 2672) for details on local displacement..

If the `CAN` parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by `DJ1` (or in the absence of `DJ1`, the start node of the member). Similarly, (DX2, DY2, DZ2) represent the deflection values at `DJ2` or the end node of the member.

Compute Delta = $\sqrt{(DX2 - DX1)^2 + (DY2 - DY1)^2 + (DZ2 - DZ1)^2}$

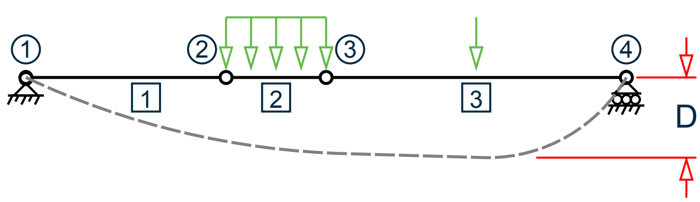
Compute Length = distance between `DJ1` and `DJ2` or, between start node and end node, as the case may be.

Then, if `CAN` is specified a value 1, dff = L/Delta

Ratio due to deflection = `DFF`/dff

b. If `CAN` = 0, the “Deflection Length” is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. A straight line joining `DJ1` and `DJ2` is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Thus, for all three members here, `DJ1` should be `1` and `DJ2` should be `4`.



D = Maximum local deflection for members 1, 2, and 3.

```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

c. If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.
d. The above parameters may be used in conjunction with other available parameters for steel design.

D9.C.8 Code Checking

The purpose of code checking is to check whether the provided section properties of the members are adequate to carry the forces transmitted to it by the loads on the structure. The adequacy is checked per the AIJ requirements.

Code checking is done using forces and moments at specified sections of the members. If the `BEAM` parameter for a member is set to 1, moments are calculated at every twelfth point along the beam, and the maximum moment about the major axis is used. When no sections are specified and the `BEAM` parameter is set to zero (default), design will be based on the forces at the start and end joints of the member. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from start joint) and magnitudes of the governing forces and moments are also printed.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Related Links

- *V.AIJ 2002 Check for MBG parameter* (on page 4319)

D9.C.9 Member Selection

The member selection process basically involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments obtained from the most recent analysis. The section selected will be of the same type as that specified initially. For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table.

Note: Member selection cannot be performed on members listed as PRISMATIC or for user-defined sections.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

Sample Input data for Steel Design

```
UNIT  METER
PARAMETER
CODE  JAPANESE 2002
NSF  0.85 ALL
UNL  10.0 MEMBER 7
KY 1.2  MEMBER 3 4
RATIO  0.9 ALL
TRACK  1.0 ALL
```

```
CHECK  CODE ALL
SELECT  ALL
```

D9.C.10 Von Mises Stresses Check

The von Mises stress equation shown below, which is modified for beam elements based on the corresponding equation in AIJ steel design code (both 2002 and 2005 editions of AIJ), indicates that the left-hand side in the equation should be less than unity. These checks are performed at locations indicated by the `BEAM` parameter.

Note: As with other design checks, the unity check value can be modified by use of the `RATIO` parameter.

The von Mises stresses are evaluated and checked per AIJ clause 5.16 as follows:

$$\frac{\sqrt{\sigma_x^2 + 3\tau_{xy}^2}}{k \times f_t} < 1.0$$

where

σ_x = Longitudinal stress in beam element. The following equation is used when the `MISES` parameter is set to 1 or 2. This is performed multiple times, once in each corner with the appropriate sign of the moment and value of elastic modulus. The largest stress is then used.

$$= \frac{F_x}{A_x} + \frac{M_y}{Z_y} + \frac{M_z}{Z_z}$$

When the `MISES` parameter is set to 3 or 4, then the longitudinal stress is calculated once using the smalles elastic modulus for each axis as follows.

$$= \left| \frac{F_x}{A_x} \right| + \left| \frac{M_y}{Z_y} \right| + \left| \frac{M_z}{Z_z} \right|$$

F_x = axial force

M_y = bending moment about y-axis

M_z = bending moment about z-axis

A_x = cross-sectional area

Z_y = section modulus about y-axis

Z_z = section modulus about z-axis

τ_{xy} = shear stress in the beam. When the `MISES` parameter is set to 1 or 3, this includes torsion stresses:

$$= \left| \frac{M_x}{Z_x} \right| + \sqrt{\left| \frac{F_y}{A_y} \right|^2 + \left| \frac{F_z}{A_z} \right|^2}$$

When the `MISES` parameter is set to 2 or 4, the torsion stresses are excluded:

$$= \sqrt{\left| \frac{F_y}{A_y} \right|^2 + \left| \frac{F_z}{A_z} \right|^2}$$

M_x = torsional moment.

F_y = shear stress in the y direction

F_z = shear stress in the z direction

Z_x = torsional section modulus

D_x = depth of the member

I_x	=	torsional constant
A_y	=	effective shear area in the y direction
A_z	=	effective shear area in the z direction
f_t	=	allowable tensile stress
k	=	loading duration factor as specified by the TMP parameter:

1.0 for permanent

1.5 for temporary

In the STRESSES output category, stress value of (numerator of the von Mises stress equation) is output as the value of fm. Along with slenderness ratios, stresses, and deflections, von Mises stress equation is checked. When its left-hand side yields the maximum ratio value, it is printed as RATIO and "VON MISES" is printed as CRITICAL COND.

Related Links

- *V.AIJ 2002 Check for MISES parameter* (on page 4325)
- *V.AIJ 2002 Check for MISES parameter* (on page 4325)

D10. Mexican Codes

D10.A. Mexican Codes - Concrete Design per MEX NTC 1987

STAAD.Pro is capable of performing concrete design based on the Mexican code NTC 1987 *Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto* (*Complementary Technical Norms for Design and Construction of Concrete Structures*).

Design of members per NTC 1987 requires the *STAAD Latin American Design Codes* SELECT Code Pack.

D10.A.1 Design Operations

STAAD.Pro has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current: Complementary Technical Standards for the Design and Construction of Concrete Structures – Nov. 1987. (Normas Técnicas Complementarias para Diseño y construcción de Estructuras de Concreto) of the Mexican Construction Code for the Federal District –Aug. 1993 (Reglamento de Construcciones para el Distrito Federal).

D10.A.2 Section Types for Concrete Design

The following types of cross sections can be defined for concrete design.

- Columns — Prismatic (Rectangular, Square, and Circular)
- Beams — Prismatic (Rectangular & Square), Trapezoidal, and T-shapes
- Walls — Finite element with a specified thickness

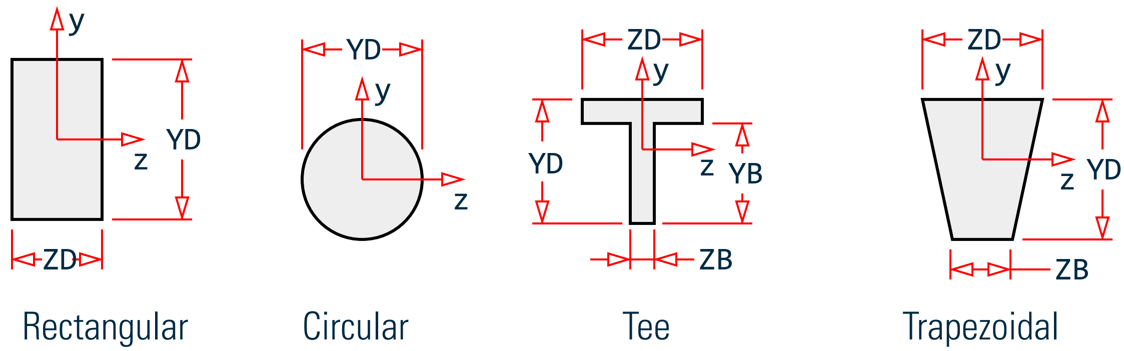


Figure 175: Concrete shape nomenclature for beams and columns

D10.A.3 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the `MEMBER PROPERTY` command. The following example shows the required input:

```
UNIT CM
MEMBER PROPERTY
13 TO 79 PRISM YD 40. ZD 20. IZ 53333 IY 13333
11 13 PR YD 20.
14 TO 16 PRIS YD 24. ZD 48. YB 18. ZB 12.
17 TO 19 PR YD 24. ZD  18. ZB 12.
```

In the above input, the first set of members are rectangular (40 cm depth and 20 cm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 20 cm diameter. Note that no area (AX) is provided for these members. For concrete design, this property must not be provided. If shear areas and moments of inertias are not provided, the program calculates these values from YD and ZD. Notice that in the above example the IZ and IY values provided are actually 50% of the values calculated using YD and ZD. This is a conventional practice which takes into consideration revised section parameters due to cracking of section.

Note that the third and the fourth set of members in the above example represent a T-shape and a TRAPEZOIDAL shape respectively. Depending on the properties (YD, ZD, YB, ZB, etc.) provided, the program will determine whether the section is rectangular, trapezoidal or T-shaped and the BEAM design will be done accordingly.

D10.A.4 Design Parameters

The program contains a number of parameters which are needed to perform design by the Mexican code. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 3.1 is a complete list of the available parameters and their default values.

The manual describes the commands required to provide these parameters in the input file. For example, the values of `SFACE` and `EFACE` (parameters that are used in shear design), the distances of the face of supports from the end nodes of a beam, are assigned values of zero by default but may be changed depending on the actual situation. Similarly, beams and columns are designed for moments directly obtained from the analyses without any magnification. The factors `MMY` and `MMZ` may be used for magnification of column moments. For beams, the user may generate load cases which contain loads magnified by the appropriate load factors.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 179: Mexican Concrete Design Parameters

Parameter Name	Default Value	Parameters
CODE	-	Must be specified as MEXICAN. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BTP	2	Bar type to use: 0. IMPERIAL (No 3 to 18) 1. METRIC (4.2 to 60mm) 2. MEXICAN (No 2 to 18)
CCL	1	Concrete class according to 1.4.1d) to define Modulus of Elasticity 1. Class 1 Concrete 2. Class 2 Concrete
CFB	FALSE	Cold formed Bar classification to define development multipliers according to table 3.1 NTC • FALSE - Not cold formed bar • TRUE - Cold formed bar
CLB	3 cm	Clear cover for bottom reinforcement
CLS	3 cm	Clear cover for side reinforcement
CLT	3 cm	Clear cover for top reinforcement
DAG	2 cm	Maximum diameter of aggregate, in current units.
DCP	TRUE	Beam Loads and reactions in direct compression Cl-2.1.5.a.I 2nd paragraph • FALSE - Loads applied indirectly • TRUE - Direct compression

Parameter Name	Default Value	Parameters
DEPTH	YD	Depth of concrete member, in current units. This value defaults to YD as provided under MEMBER PROPERTIES.
DIM	TRUE	• FALSE: Not precautions taken - Section reduction to section 1.5 NTC Concrete • TRUE: Precautions are taken to assure dimensions
DSD	TRUE	Ductile frames in accordance with Section 5 of the code. Some design conditions are considered (not including, for the time being, geometric or confinement ones) • FALSE - Non-Ductile frames • TRUE - Ductile Frames
EFACE	0	Face to support location of end of beam. If specified, for shear force at start is computed at a distance of EFACE+d from the start joint of the member. Positive number.
EXP	FALSE	Exposition to soil or weather to define cover and min Steel reinforcement • FALSE - Not exposed to soil or weather • TRUE - Exposed to soil or weather
FC	200 Kg/cm^2	Compressive Strength of Concrete
FYMAIN	4,200 Kg/cm^2	Yield Stress for main reinforcing steel
FYSEC	4,200 Kg/cm^2	Yield Stress for secondary (stirrup) reinforcing steel
LSS	0	Part of the longitudinal steel considered to reduce shear. 0 (zero) is conservative. Value between 1 and 0.

Parameter Name	Default Value	Parameters
LTC	FALSE	Light Concrete to define development multipliers according to table 3.1 NTC • FALSE - Regular concrete • TRUE - Lightweight concrete
MAXMAIN	12	Maximum main reinforcement bar size (Number 2 -18)
MINMAIN	2.5	Minimum main reinforcement bar size (Number 2 -18)
MINSEC	2.5	Minimum secondary reinforcement bar size (Number 2 -18)
MMY	1.0	Moment magnification factor for columns, about My.
MMZ	1.0	Moment magnification factor for columns, about Mz.
MOE	198,000 Kg/cm^2	Concrete modulus of elasticiy.
NSECTION	12	Number of equally-spaced sections to be considered in finding critical moments for beam design
PHI	90 degrees	Stirrups angle with the axis of the element
PSS	TRUE	Slab beared perimeter. To calculate min steel required according to 2.1.2
REINF	0	Tied Column. A value of 1 will mean spiral.
SFACE	0	Face to support location of start of beam. If specified, for shear force at start is computed at a distance of SFACE+d from the start joint of the member. Positive number
TEQ	FALSE	Beam needed for torsional equilibrium Cl.2.1.6a) 2nd paragraph • FALSE - No • TRUE - Yes

Parameter Name	Default Value	Parameters
TRACK	0	Beam Design 0. Critical Moment will not be printed out with beam design report. 1. Will mean a print out. 2. Will print out required steel areas for all intermediate sections specified by `NSECTION`. Column Design 0. Will print out detailed design results. 1. Will mean a print out column interation analysis results in addition to TRACK 0 output. 2. will print out a schematic interaction diagram and intermediate interaction values in addition to all of the above.
WIDTH	ZD	Width of concrete member, in current units. This value defaults to `ZD` as provided under `MEMBER PROPERTIES`

* These values must be provided in the current unit system being used.

Note: When using metric bars for design, provide values for these parameters in actual ‘mm‘ units instead of the bar number. The following metric bar sizes are available: 4.2mm, 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm and 60 mm.

D10.A.5 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an `NSECTION` parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

D10.A.5.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of the section properties provided by the user. If the section dimensions are inadequate to carry the applied load, that is if the required reinforcement is greater than the maximum allowable for the cross section, the program reports that beam fails in maximum reinforcement. Rectangular sections are also designed with compression reinforcement.

Effective depth is chosen as Total depth - (Clear cover + diameter of stirrup + half the dia. of main reinforcement), and a trial value is obtained by adopting proper bar sizes for the stirrups and main reinforcements. The relevant clauses in Sections 1.5, 1.6, 2.1.1-2-5, 3.10 and 5.2.2 of NTC Concrete are utilized to obtain the actual amount of steel required as well as the maximum allowable and minimum required steel. These values are reported as ROW, ROWMX and ROWMN in the output and can be printed using the parameter TRACK 1.0 (see D10.A.4 Design Parameters (on page 1744)). In addition, the maximum, minimum and actual bar spacing are also printed.

It is important to note that beams are designed for flexural moment MZ only. The moment MY is not considered in the flexural design.

D10.A.5.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear forces are calculated at a distance (d+SFACE) and (d+EFACE) away from the end nodes of the beam. SFACE and EFACE have default values of zero unless provided under parameters (see D10.A.4 Design Parameters (on page 1744)). Note that the value of the effective depth "d" used for this purpose is the update value and accounts for the actual c.g. of the main reinforcement calculated under flexural design. Clauses 2.1.5-6 and 5.2.4 of NTC Concrete are used to calculate the reinforcement for shear forces and torsional moments. Based on the total stirrup reinforcement required, the size of bars, the spacing, the number of bars and the distance over which they are provided are calculated. Stirrups due to geometric conditions are assumed to be 2-legged, due to design conditions could be 2 or 4-legged.

D10.A.5.3 Design for Anchorage

In the output for flexural design, the anchorage details are also provided. At any particular level, the START and END coordinates of the layout of the main reinforcement is described along with the information whether anchorage in the form of a hook or continuation is required or not at these START and END points. Note that the coordinates of these START and END points are obtained after taking into account the anchorage requirements. Anchorage length is calculated on the basis of the Clauses described in Section 3.1 of NTC concrete. In case the program selects 2 different diameters for the main or compression reinforcement, only the anchorage for the largest diameter is analyzed.

D10.A.5.4 Output

Level	Serial number of bar level which may contain one or more bar group
Height	Height of bar level from the bottom of the beam
Bar Info	Reinforcement bar information specifying number of bars and bar size
From	Distance from the start of the beam to the start of the reinforcement bar
To	Distance from the start of the beam to the end of the reinforcement bar
Anchor (STA/ END)	States whether anchorage, either hook or continuation, is needed at the start (STA) or at the end (END).
Row	Actually required flexural reinforcement (As/bd) where b = width of cross section (ZD for a rectangular or square section) and d = effective depth of cross section (YD minus the distance from extreme tension fiber to the centroid of main reinforcement).
ROWMN	Minimum required flexural reinforcement (Amin/bd)
ROWMX	Maximum required flexural reinforcement (Amax/bd)
Spacing	Distance between centers of adjacent bars of main reinforcement
Vu	Factored shear force at section

Vc	Nominal shear strength provided by concrete
Vs	Nominal shear strength provided by shear reinforcement
Tu	Factored torsional moment at section
Tc	Nominal torsional moment strength provided by concrete
Ts	Nominal torsional moment strength provided by torsion reinforcement

D10.A.5.5 Example Output for Beam Design

```
  =====================================================================

         BEAM  NO.     2 DESIGN RESULTS - FLEXURE

 PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES,DDF

  LEN -  6000.00(mm)  FY -   412.  FC -   20.  SIZE -  253.75 X  253.75(mm)

  LEVEL    HEIGHT     BAR INFO       FROM          TO           ANCHOR
            (mm)                     (mm)         (mm)         STA  END
  _____________________________________________________________________

    1         42.     5 - 2.MM      2468.         6000.        NO   YES
    2        212.     5 - 2.MM         0.         2782.        YES  NO


         B E A M  N O.     2 D E S I G N  R E S U L T S - SHEAR

   AT START SUPPORT - Vu=    5.63 KN  Vc=    0.00 KN  Vs=    0.00 KN
 Tu=      0.09 Kn Me  Tc=      0.00 Kn Me  Ts=      0.00 Kn Me  LOAD     1
                      STIRRUPS ARE NOT REQUIRED.
   AT END   SUPPORT - Vu=    5.63 KN  Vc=    0.00 KN  Vs=    0.00 KN
 Tu=      0.09 Kn Me  Tc=      0.00 Kn Me  Ts=      0.00 Kn Me  LOAD     1
                      STIRRUPS ARE NOT REQUIRED.
```

D10.A.6 Column Design

Columns design in STAAD.Pro per the Mexican code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAGx & -MMAGy parameters are specified, the column moments are multiplied by the corresponding MMAG value to arrive at the ultimate moments on the column. Minimum eccentricity conditions to be satisfied according to section 2.1.3.a are checked.

Method used: Bresler Load Contour Method

Known Values: Pu, Muy, Muz, B, D, Clear cover, Fc, Fy

Ultimate Strain for concrete : 0.003

Steps involved:

1. Assume some reinforcement. Minimum reinforcement (1% for ductile design or according to section 4.2.2) is a good amount to start with.

2. Find an approximate arrangement of bars for the assumed reinforcement.
3. Calculate PNMAX = Po, where Po is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed PNMAX. If PNMAX is less than the axial force Pu/FR, (FR is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 6% (or 4% for ductile design), the column cannot be designed with its current dimensions.
4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as MYCAP and MZCAP respectively.
5. Solve the Interaction Bresler equation:

   $$(M_{ny}/M_{ycap})^{\alpha} + (M_{nz}/M_{zcap})^{\alpha}$$

   Where α = 1.24. If the column is subjected to uniaxial moment: α = 1
6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 6% or 4% respectively) and steps 2 to 6 are repeated.

By the moment to check shear and torsion for columns the sections have to be checked as beams and the most strict of both shear and torsion reinforcement adopted.

D10.A.7 Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

- P0 = Maximum allowable pure axial load on the column (moment zero).
- Pnmax = Maximum allowable axial load on the column.
- P_bal = Axial load capacity of balanced strain condition.
- M_bal = Uniaxial moment capacity of balanced strain condition.
- E_bal = M_bal / P_bal = Eccentricity of balanced strain condition.
- M0 = Moment capacity at zero axial load.
- P_tens = Maximum permissible tensile load on the column.
- Des. Pn = Pu/FR where FR is the Strength Reduction Factor and Pu is the axial load for the critical load case.
- Des.Mnx = Mux*MMAGx/FR where FR is the Strength Reduction Factor and Mu is the bending moment for the appropriate axis for the critical load case.
- Mu = $\sqrt{}(Mux.Mmagx)^2 + (Muy.Mmagy)^2$
- e/h = (Mn/Pn)/h where h is the length of the column

D10.A.8 Column Design Output

The next table illustrates different levels of the column design output.

The output is generated without any TRACK specification (or `TRACK 0`):

```
   ====================================================================

        COLUMN  NO.     1  DESIGN PER MEX NTC-87 - AXIAL + BENDING



   FY - 411.9  FC -  19.6 MPa     SQRE SIZE   30.0 x  30.0 (mm) TIED

           AREA OF STEEL REQUIRED =1422.857

   BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
   ----------------------------------------------------------

    12 - NUMBER  4           1.693        1      END     0.700
   (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
```

`TRACK 1` generates the following additional output:

```
       COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)

    --------------------------------------------------------
       P0          Pn max        P-bal.       M-bal.      e-bal.(mm)
       1807.71      1807.71       472.28        89.02      18.8
       M0          P-tens.       Des.Pn      'Des.Mn     e/h
         66.71      -627.70       106.12        35.18    0.074
    -----------------------------------------------------------------------

       COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)

    --------------------------------------------------------
       P0          Pn max        P-bal.       M-bal.      e-bal.(mm)
       1807.71      1807.71       472.28        89.02      18.8
       M0          P-tens.       Des.Pn      'Des.Mn     e/h
         66.71      -627.70       106.12        44.63    0.093
    -----------------------------------------------------------------------
```

`TRACK 2` generates the interaction diagram output in addition to all the above:

```
                            Pn          Mn          Pn          Mn       (@
Z )
            |                1668.66       12.60       834.33       76.51
          P0 |*              1529.60       29.22       695.27       81.63
            | *              1390.55       43.18       556.22       86.29
      Pn,max|__*             1251.49       54.53       417.16       87.72
            |   *            1112.44       63.31       278.11       83.38
      Pn    |    *            973.38       70.55       139.05       76.67
     NOMINAL|     *     Pn          Mn          Pn          Mn       (@
Y )
       AXIAL|      *         1668.66       12.60       834.33       76.51
 COMPRESSION|       *        1529.60       29.22       695.27       81.63
          Pb|-------*Mb      1390.55       43.18       556.22       86.29
            |      *         1251.49       54.53       417.16       87.72
```

```
___________|____*_______     1112.44        63.31        278.11        83.38
           | * M0    Mn,      973.38        70.55        139.05        76.67
           |*    BENDING
     P-tens|*     MOMENT
           |
```

D10.A.9 Slab Design

Slabs are designed per Mexican NTC specifications. To design a slab, it must be modeled using finite elements.

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for FX, FY, FXY, MXY. Also, design is not performed at any other point on the surface of the element. Shear is checked with Q.

A typical example of element design output is shown below. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement. The parameters FYMAIN, FC, CLB, CLS, CLT, DIM, and EXP listed in D10.A.4 Design Parameters (on page 1744) are relevant to slab design. Other parameters mentioned are not used in slab design.

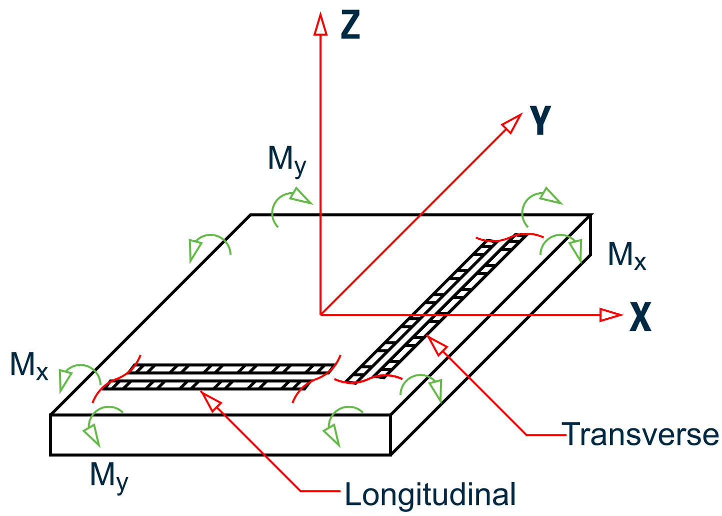


Figure 176: Element moments: Longitudinal (L) and Transverse (T)

D10.A.9.1 Example Output for Element Design

```
   ELEMENT DESIGN SUMMARY
   ----------------------

   ELEMENT     LONG. REINF       MOM-X /LOAD    TRANS. REINF      MOM-Y /LOAD
               (SQ.MM/MM)        (KN-MM/MM)      (SQ.MM/MM)       (KN-MM/MM)

     47 TOP : Longitudinal direction - Only minimum steel required.
     47 TOP : Transverse direction   - Only minimum steel required.
     47 TOP :     0.205          0.00 /    0       0.205          0.00 /    0
        BOTT:     0.254         10.44 /    1       0.362         13.35 /    1

     47 SHEAR CAPACITY     57.06 KN ***PASS*** FOR LOAD CASE    3


          ***** INDICATES REINFORCEMENT EXCEEDS MAXIMUM
```

```
**************************END OF ELEMENT DESIGN**************************
```

D10.B. Mexican Codes - Steel Design per NTC 1987

STAAD.Pro is capable of performing steel design based on the Mexican code NTC 1987 *(Normas Técnicas Complementarias para Diseño y construcción de Estructuras Metálicas)* (*Complementary Technical Standards for the Design and Construction of Steel Structures – Dec. 1987*) or the *Reglamento de Construcciones para el Distrito Federal* (*Mexican Construction Code for the Federal District* –Aug. 1993).

Design of members per NTC 1987 requires the *STAAD Latin American Design Codes* SELECT Code Pack.

D10.B.1 General

The design philosophy considered is that of the Load Cases and Resistance Method or Limit States Design usually known as Load and Resistance Factor Design (LRFD).

Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized--ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD.Pro implementation of the Mexican Standards for steel structures, members are proportioned to resist the design loads without exceeding the limit states of strength, and stability. It allows to check deformation to verify serviceability.

Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that main code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the Mexican specifications as implemented in STAAD.Pro steel design. A brief description of the fundamental concepts is presented here.

D10.B.2 Limit States Design Fundamentals

The primary objective of the Limit States Design Specification is to provide a uniform reliability for all steel structures under various loading conditions.

The Limit States Design Method uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible.

The method may be summarized by the inequality

$$Y_i\ Q_i \leq R_n\ FR$$

On the left side of the inequality, the required strength is the summation of the various load effects, Q_i, multiplied by their respective load factors, Y_i. The design strength, on the right side, is the nominal strength or resistance, R_n, multiplied by a resistance factor, FR.

In the STAAD implementation of the Mexican Standards, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

D10.B.3 Member End Forces and Moments

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. the following figures show the member end actions with their directions. Refer to G.18 Member End Forces (on page 2187) .

D10.B.4 Section Classification

The Limit States Design specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact (type 2), noncompact (type 3), or slender element (type 4), sections depending upon their local buckling characteristics, besides sections type 1 are able for plastic design. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and design accordingly.

D10.B.5 Member in Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter `NSF` (D10.B.11 Design Parameters (on page 1761)), that always refers to the gross section. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

In addition to the tension resistance criterion, the user defines if tension members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D10.B.6 Axial Compression

The column strength equations take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD.Pro according to the procedure outlined in Section 3.2 of the NTC. For slender elements, the procedure described in Section 2.3.6.NTC is also used.

The procedures of Section 3.2 of the Commentaries, design helps and examples of the Complementary Technical Standards for the Design and Construction of Steel Structures (de los Comentarios, ayudas de diseño y ejemplos de las Normas Técnicas Complementarias para el Diseño y Construcción de Estructuras Metálicas, DDF (Comentarios - Julio 1993) were implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD.Pro immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

D10.B.7 Flexural Design Strength

In the Limit States Design Method, the flexural design strength of a member is determined mainly by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section.

The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length Lu and flexural resistance Mr are functions of the section geometry and are calculated as per the procedure of Section 3.3.2 of the NTC.

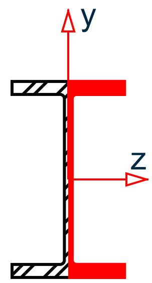
The purpose of bending coefficient Cb is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter `CB` or CBy (D10.B.11 Design Parameters (on page 1761)) or may be calculated by the program (according to the American LRFD specification) if CB is specified as 0.0. In the absence of the parameter `CB`, a default value of 1.0 will be used.

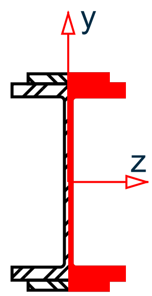
To specify laterally unsupported length, either or both of the parameters `UNL` and `UNF` (see D10.B.11 Design Parameters (on page 1761)) can be used.

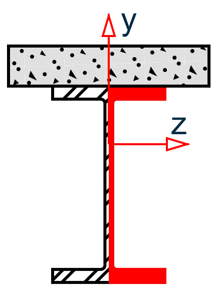
It is taken into account the reduction of flexural resistance due to slender web according to section 4.5.8 of the NTC

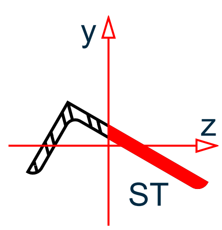
For the sections where the web and flange are slender the American LRDF specification was used.

Stress Zones Due to Bending









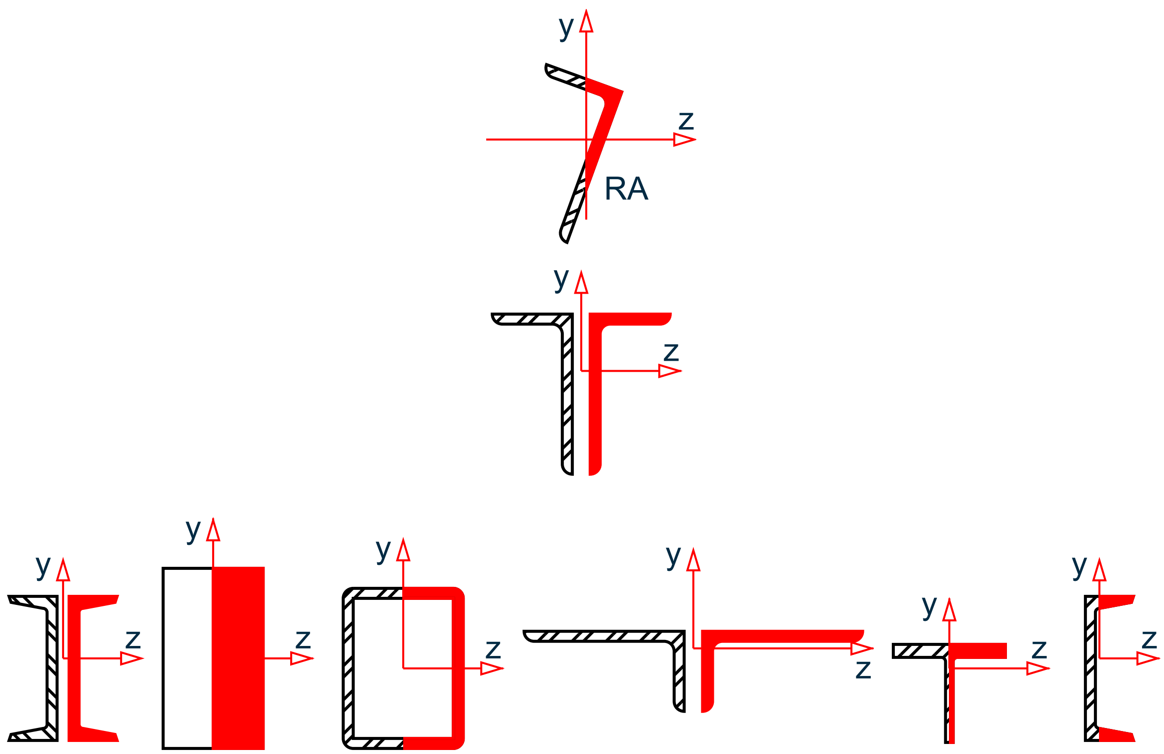
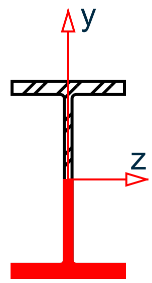
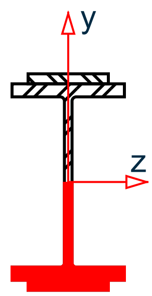
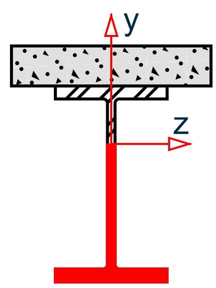


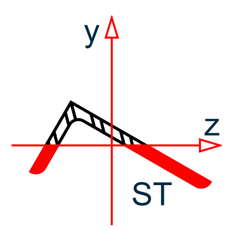
Figure 177: Stress zones due to bending about the Y axis (MY) for various section types

> **Note:** Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension


y
z


y
z


y
z


y
z
ST

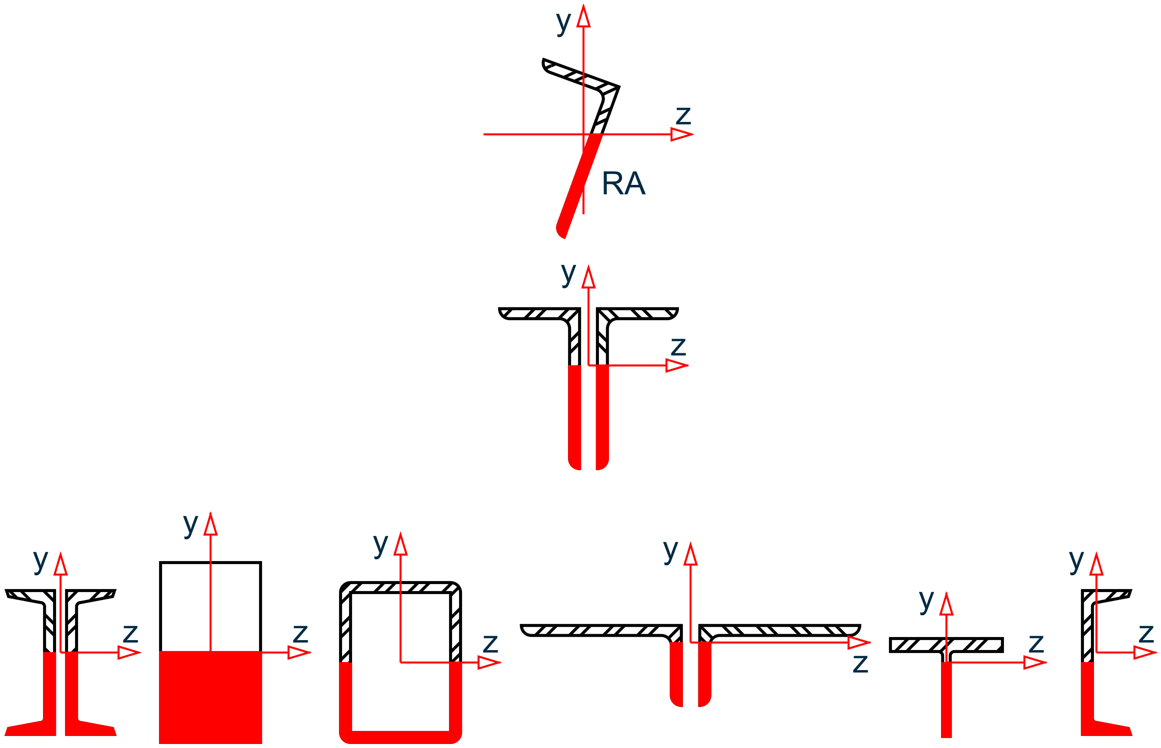


Figure 178: Stress zones due to bending about the Z axis (MZ) for various section types

Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension.

D10.B.8 Design for Shear

The procedure of Sect. 3.3.3 of the NTC is used in STAAD to design for shear forces in members. Besides combined bending and shear is checked according to section 3.3.4 of the NTC, considering also the limits for stiffeners of the web according to sections 4.5.6/7 of the NTC. Shear in wide flanges and channel sections is resisted by the area of the web/s..

D10.B.9 Combined Compression Axial Force and Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas of the Section 3.4 of the NTC. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

It is considered that the frames are part of structures that have shear walls or rigid elements so that the lateral displacements of a floor could be disregarded. The program has included formulas to include structures with lateral displacements in the future considering for B2 the columns individually and not the complete floor analysis.

It is taken into account if the elements have transverse loads and if the ends are angularly restrained.

D10.B.10 Combined Tension Axial Force and Bending

Based on Section 3.5 4 of the NTC.

D10.B.11 Design Parameters

Design per Mexican Standards is requested by using the CODE. Other applicable parameters are summarized in Table 13B.1 below. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters DMAX and DMIN may only be used for member selection only.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 180: Design Parameters According to Mexican NTC Standards - Steel

Parameter Name	Default Value	Description
CODE	-	Must be specified as MEXICAN. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	0	0: Design at ends and those locations specified by SECTION command. 1: Design at ends and at every y cada 1/12th point along member length
CB	1	Coefficient C defined per section 3.3.2.2. If Cb is set to 0.0 it will be calculated by the program according to LRFD USA (CbMex=1/CbUSA). Any other value will be directly used in the design.
CMB	1	Cfactor for combined forces when there are transverse loads in the members. Section 3.4.3.3.ii of NTC CMB 1.0 = Members ends are restricted angularly. CMB 0.85 = Members ends are not restricted angularly.

Parameter Name	Default Value	Description
DFF	None(Mandatory for deflection check, TRACK 4.0)	"Deflection Length" / Maxm. allowable local deflection See Note 1 below.
DJ1	Start Jointof member	Joint No. denoting starting point for calculation of "Deflection Length ." See Note 1 below.
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length." See Note 1 below.
DMAX	114 cm	Maximum allowable depth
DMIN	0.0 cm	Minimum allowable depth
DSD	T	Perform the ductile seismic design in accordance with Section 11 (True or False). Main design conditions are considered (not including, at the moment, geometric ones)
FU	4,230 Kg/cm^2	Ultimate tensile strength of steel
FYLD	2,530 kg/cm^2	Minimum Yield strength of steel
IMM	0	Main or secondary member for the purpose of checking slenderness 0. Main member 1. Secondary and wind trusses
INO	0	Curve Definition according to NTC. 3.2.2.1a, defined for I shapes or tubes 0. n=1.4, laminated I shapes, tubes or built up with 3 or 4 welded plates obtained from wider plates cuts with oxygen. 1. n=1, I shapes, tubes or built up with 3 or 4 welded plates

Parameter Name	Default Value	Description
IRR	0	Variable defined for the whole structure indicating if it is regular or irregular according to section 3.4 of the NTC. Columns that are part of regular structures Columns that are part of irregular structures
KX	1.0	Effective length factor for flexural-torsional buckling
KY	1.0	Effective length factor for local Y axis- Usually minor axis
KZ	1.0	Effective length factor for local Z axis- Usually major axis
LDR	T	Defines if the structure has elements to bear the wind load (shear walls, wind trusses, or bracing rigid elements) that restrict lateral displacements and allow to disregard slenderness effects. (True or False)
LX	Member length	Length for determining flexural-torsional buckling
LY	Member length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member length	Length to calculate slenderness ratio for buckling about local Z axis.
NSF	1	Net section factor for tension members
RATIO	1.0	Permissible ratio of actual load effect and design strength
STIFF	Longer of Member length or depth	Spacing of stiffeners for beams for shear design

Parameter Name	Default Value	Description
TRACK	0	Controls the level of detail in output 0. = Suppress all design strengths 1. = Print all design strengths 2. = Print expanded design output
UNB	Member length	Unsupported length (L) of the bottom* flange for calculating flexural strength . Will be used only if compression is in the bottom flange. See Note 2 below.
UNT	Member length	Unsupported length (L) of the top* flange for calculating flexural strength . Will be used only if compression is in the top flange. See Note 2 below.

D10.B.11.1 Notes

1. For deflection check, parameters `DFF`, `DJ1`, and `DJ2` from D3.B.6 Design Parameters (on page 1384) may be used. All requirements remain the same.
2. Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if `SET Z UP` is used).

D10.B.12 Code Checking and Member Selection

Both code checking and member selection options are available in STAAD Mexican Standards implementation.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D10.B.13 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format.

CRITICAL COND refers to the section of the Mexican NTC which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

D11. New Zealand Codes

D11.A. New Zealand Codes - Steel Design per NZS 3404-1997

STAAD.Pro is capable of performing steel design based on the New Zealand code NZS 3404-1997 *New Zealand Standard for Steel Structures*, Parts 1 & 2, including Amendments 1 & 2.

D11.A.1 Member Property Specifications

For specification of member properties, either the steel section library available in STAAD.Pro or the User Table facility may be used.

The following shapes are supported for design per NC3404-1997:

I Section (rolled and welded)
Tee cut from rolled I-section
Single Channel
Single Angle
Pipe / CHS
Tube / RHS / SHS
Tapered I

Note: Using any shape not listed will result in an error message in the output and the section will *not* be designed.

D11.A.2 Material Properties

Any material with modulus of elasticity between 137,895 MPa (20,000 ksi) – 344738 MPa (50,000 ksi). You can design members with either built-in material constants or user-defined materials.

Note: If any material with modulus of elasticity value out of that range is assigned to member, it will be designed but an additional warning message will be produced.

Refer to TR.26.1 Define Material (on page 2303) for further information on the Define Material feature.

Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) for further information on the Built-in Material Constants feature.

Yield stress, f_y , and ultimate stress, f_u , are calculated based on steel grade table. The design parameters `FYLD`, `FU`, and `SGR` are used to directl specify these values.

D11.A.3 Section Classification

The slenderness limit for each element is calculated using table 5.2 and 6.2.4 for elements under uniform compressive stress and varying compressive stress (bending stress).

The `IST` design parameter related to section classification calculations.

D11.A.4 Member Resistance

The member resistance is calculated in STAAD according to the procedures outlined in NZS 3404-1997. Calculated design capacities are compared to corresponding axial, bending moment, and shear forces determined from the STAAD.Pro analysis. These are used to report the fail or pass status for the members designed.

Slenderness

The slenderness check is *not* used as a critical member ratio check. If a slenderness check is included and exceed unity (1.0), then it may be reported as the governing criteria. However, if it is less than unity, it will be reported but will not be used as the governing ratio.

D11.A.4.1 Slenderness

For calculating member slenderness, the length of unbraced segment for compression is considered.

For analytical member design, the `MAIN` and `TMAIN` parameters are used.

D11.A.4.2 Bending

Bending capacities are calculated per NZS3404 Section 5. The allowable bending moment of members is determined as the lesser of nominal section capacity and nominal member capacity (ref. Cl.5.1).

D11.A.4.2.1 Bending Section Strength

The nominal section moment capacity, ϕM_s , is calculated about both principal x and y axes and is the capacity to resist cross-section yielding or local buckling and is expressed as the product of the yield stress of the material and the effective section modulus (ref. Cl.5.2.1). The effective section modulus is a function of section type (i.e., compact, non-compact, or slender) and minimum plate yield stress fy. The nominal member capacity depends on overall flexural-torsional buckling of the member (ref.Cl.5.3).

D11.A.4.2.2 Bending Member Strength

For sections where the web and flange yield stresses ($f_{y,web}$ and $f_{y.flange}$ respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements.

For sections where the web and flange yield stresses (fy,web and fy.flange respectively) are different, the lower of the two yield stresses is applied to both the web and flange to determine the slenderness of these elements. Member moment capacity, φM_b , is calculated about the principal x axis only (ref. Cl.5.6). Critical flange effective cross-section restraints and corresponding design segment and sub-segments are used as the basis for calculating capacities.

The `ALM` design parameter is used to specify a_m (refer cl. 5.6.1.1).

The `SKR`, `SKT`, `SKL`, `UNT`, `UNB`, and `PBRACE` design parameters are also used for bending checks.

D11.A.4.3 Shear

Section web shear capacity, φV_v , is calculated per Cl.5.11, including both shear yield and shear buckling capacities. Once the capacity is obtained, the ratio of the shear force acting on the cross section to the shear capacity of the section is calculated. If any of the ratios (for both local Y & Z-axes) exceed 1.0 or the allowable value provided using the `RATIO` parameter, the section is considered to have failed under shear.

The following table highlights which shear capacities are calculated for different profile types.

General Profile Type	Shear Checks
I-Section (i.e., parallel to minor principal axis)	Calculated for web only

General Profile Type	Shear Checks
T-Section	
Channel	
Angle	
Tube	Calculated parallel to both z & y principal axes
Pipe	Per NZS3404 5.11.4

Only unstiffened web capacities are calculated. Stiffened webs are not considered. Bearing capacities are not considered.

The TSP design parameter is used in shear capacity calculations.

D11.A.4.4 Compression

The compressive strength of members is based on limit states per NZS3404 Section 6. It is taken as the lesser of nominal section capacity and nominal member capacity.

D11.A.4.4.1 Compression Section Strength

Nominal section capacity, φN_s , is a function of form factor (Cl.6.2.2), net area of the cross section, and yield stress of the material. Through the use of the NSC parameter, you may specify the net section area. Note that this parameter is different from that corresponding to tension. The program automatically calculates the form factor. The

$$\mathrm{k_f}$$

form factors are calculated based on effective plate widths per Cl.6.2.4, and the f_y yield stress is based on the minimum plate yield stress.

The NSC (net section factor for compression) design parameter is used for compression sectoin strength checks.

D11.A.4.4.2 Compression Bending Strength

Nominal member capacity, φN_c , is a function of nominal section capacity and member slenderness reduction factor (Cl.6.3.3). This value is calculated about both principal x and y axes. Here, you are required to supply the value of a_b (Cl.6.3.3) through the ALB parameter. The effective length for the calculation of compressive strength may be provided through the use of the parameters KY, KZ, LY, and LZ.

The PBCRES and LHT design parameters are also used for this check.

D11.A.4.5 Tension

The criteria governing the capacity of tension members are based on two limit states per NZS3404 Section 7.

The limit state of yielding of the gross section is intended to prevent excessive elongation of the member.

The second limit state involves fracture at the section with the minimum effective net area φN_t section axial tension capacities are calculated (Cl.7.2). Through the use of the NSF parameter, you may specify the net section area. STAAD.Pro calculates the tension capacity of a member based on these two limit states per Cl.7.1 and Cl.7.2 respectively of NSZ3404. Eccentric end connections can be taken into account using the KT correction factor, per

Cl.7.3. The f_y yield stress is based on the minimum plate yield stress. Parameters FYLD, FU, and NSF are applicable for these calculations.

D11.A.4.6 Combined Forces

D11.A.4.6.1 Combined Section Strength

Combined section bending and shear capacities are calculated using the shear and bending interaction method as per Cl.5.12.2.

D11.A.4.6.2 Combined Member Strength

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. Here, the adequacy of a member is also examined against both section (ref. Cl.8.3.4) and member capacity (ref.Cl.8.4.5). These account for both in-plane and out-of-plane failures. If the summation of the left hand side of the equations, addressed by the above clauses, exceeds 1.0 or the allowable value provided using the RATIO parameter, the member is considered to have failed under the loading condition.

D11.A.4.7 Torsion

STAAD.Pro does *not* design sections or members for torsion for NZS3404.

D11.A.4.8 Seismic Provisions

The program performs the following checks per the seismic provisions in section 12 of the code.

1. Minimum specified yield stress (table 12.4)
2. Maximum ratio of (fy / fu) (table 12.4)

   The member seismic category is specified using the DUCT design parameter.
3. Fabrication requirement (Sec.12.4.1.2) - Category 1 or 2 members shall be hot-rolled or fabricated by welding from hot-rolled plate, except that category 2 members may be cold-formed, provided that adequate ductility capacity of the member and its connections is established by experimental testing or rational design.
4. Element slenderness (sec.12.5.1.1) - The elements of category 1, 2, 3 and 4 shall comply with the plate-element slenderness limitations presented in Table 12.5.
5. Section symmetry requirement (sec.12.5.2) – The yielding regions of category 1 or 2 members shall be doubly symmetric sections. The yielding regions of category 3 members shall be doubly or singly symmetric sections.
6. Web slenderness of beam (sec.12.7.2.1) - The web thickness within the yielding region of a beam shall be not less than $(d1/82)\ (fy\ /\ 250)$ for a category 1 or 2 member or less than $(d1/101)\ (fy\ /\ 250)$ for a category 3 member.
7. Limit on axial force (sec.12.8.3.1) - The ratio of design axial force, N*, to design section capacity, ϕN_s , (refer to 6.2) shall not exceed the values given in (a) through (c) below:

   a. The general limit given in table 12.8.1.
   b. In addition to (a), for category 1, 2 and 3 column members, excluding brace members of concentrically and eccentrically braced frames, the following limitation on design axial compression shall apply, unless waived according to 12.8.3.2

$$N^* \le \phi N_s \left(\frac{1 + \beta_m - \sqrt{N_s / N_{oL}}}{1 + \beta_m + \sqrt{N_s / N_{oL}}} \right) \qquad \text{Eq. 12.8.3.1}$$

c. When the slenderness ratio for the member web exceeds that given in table 12.8.2 for the appropriate member category, the design axial force generated by gravity loading alone, Ng^*, shall comply with the axial force limitation equation given therein.

8. Shear-bending interaction (12.10.3.1) - When a capacity design procedure is not used, at yielding regions in category 1 or 2 beams forming part of a seismic-resisting system, when designing for load combinations including earthquake loads the nominal web shear capacity of the beams shall be taken as 80% of that calculated from 5.11.4.1 and the interaction of shear and bending moment shall satisfy:

$$M^* \leq \phi M_{sv}$$

where

$$M_{sv} = \begin{cases} M_s & \text{for } V^* \leq 0.6\phi V_w \\ M_s\left(1.38 - \frac{V^*}{1.6V_w}\right) & \text{for } 0.6\phi V_w \leq V^* \leq 0.8\phi V_w \end{cases}$$

The ratio will be calculated as – $M^* / \phi M_{sv}$

D11.A.5 Member Design

There are two methods available in STAAD.Pro for checking members against the requirements of NZS 3404:

Analytical member method (referred to as `MEMBER` design), and
Physical member method (referred to as `PMEMBER` design)

Traditionally STAAD.Pro performed code checks based on single analytical members (i.e., single members between two nodes). This implementation assumes that the design inputs (except forces and section properties) remain same throughout the length of the member. Physical member design on the other hand allows you to group single or multiple analytical members into a single physical design member for the purposes of design to NZS 3404. Thus the some of the design inputs related to member strength (unbraced lengths etc.) varies along the length of the member.

D11.A.5.1 Analytical Member Design

Thirteen equidistant cross-sections including two end sections are selected from an analytical member. For each cross-section, the member resistance checks are evaluated for each load case. The design inputs (except forces and section properties) are treated same for throughout the length of the member.

For analytical member design, by default the member is divided at 13 evenly spaced points along the member length.

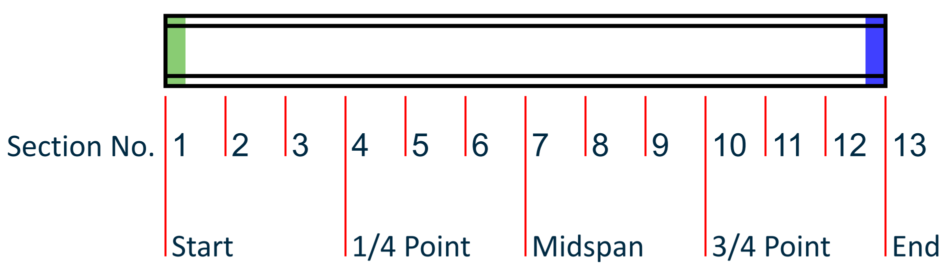


Figure 179: The default sections for design when `BEAM 1.0` is used

D11.A.5.1.1 Automated MEMBER Design Calculations

Automated Design Calculations	PMEMBER Design Parameter	Comments
α_b compression member section constant per NZS3404 6.3.3.	n/a	Calculated from table 6.3.3.
α_m moment modification factor per NZS3404 5.6.1.1 or Table 5.6.2 depending on if the segment is restrained at on/both ends.	ALM	Calculated based on moments distribution for individual segments and sub-segments.
f_u tensile strength per NZS3404 2.1.2 and AS4100 table 2.1.	FU	Based on nominal steel grade specified using SGR design parameter and element thickness.
f_y yield stress per NZS3404 2.1.1 and AS4100 table 2.1.	FYLD	Based on nominal steel grade specified using SGR design parameter and element thickness.
Residual stress category for NZS3404 Table 5.2 and 6.2.4.	IST	Based on section type.

Related Links

- *V.NZS3404 1997-Angle section compact* (on page 4374)
- *V.NZS3404 1997-Angle section Non compact* (on page 4384)
- *V.NZS3404 1997-Channel Section* (on page 4393)
- *V.NZS3404 1997-I section* (on page 4402)
- *V.NZS3404 1997-RHS Section* (on page 4408)
- *V.NZS3404 1997-Simply Supported Beam With Overhang* (on page 4420)
- *V.NZS3404 1997-Tube Section Compact* (on page 4431)
- *V.NZS3404 1997-Tube Section Non Compact* (on page 4438)
- *V. NZS3404 1997-UB Section* (on page 4448)
- *V.NZS3404 1997-Unequal Angle Section* (on page 4459)

D11.A.5.2 Physical Member Design

In case of analytical member design, all the design inputs are controlled by user input which is one-time for a whole analytical member. Specially the different unbraced lengths (UNT, UNB, LZ, LY), unbraced length factors (Kz, Ky) are considered constant throughout the length of the analytical member. But in case of a physical member, these values tend not to be constant throughout the length. The real essence of physical member design in NZS3404:1997 is the consideration of different values of unbraced lengths and unbraced length factors in different locations of same physical member to replicate the actual condition. This is achieved by auto calculation of these values from the user-provided physical bracings

In case of Physical member design in NZS3404:1997, the program does not divide the physical member into analytical member segments of equal length. Instead, each consisting analytical member is divided into 13 sections. Hence the physical member is considered as a member as a whole with “13n” sections for design checks, where n = number of component analytical members in the physical member under consideration. Clearly, in most of the cases, the distance between two sections will not be constant throughout the length of the member. In order to prevent overlapping design sections at the ends of adjoining componenet analytical members, a small gap of 0.001x the member length is used from the common joint to the design section at these locations. Doing so allows the program to capture changes in internal forces, particularly shear, at these joints within the physical member.

Additionally, PMEMBER automates:

steel grades based on section type, and
tensile stress, f_u , and yield stress, f_y , values based on plate thicknesses

Therefore it is recommend that PMEMBER design be used, even when a physical member consists of only one component analytical member.

D11.A.5.2.1 Modeling with Physical Members

You can use the physical modeling tools available in the user interface **Analytical Modeling** and **Steel Design** workflows to form physical members.

You can also define physical members using the `DEFINE PMEMBER` command. Refer to TR.16.2 Physical Members (on page 2238) for details.

D11.A.5.2.2 Section Profiles

When a physical member is checked against the code (`CODE CHECK`) in NZS3404-1997, all the component analytical members within a single physical member must have the same section assigned. If not, a code check will *not* be made and an error is displayed in the output.

When a physical member profile is selected (`SELECT`) in NZS3404-1997, all the component analytical members within a single physical member must be assigned from the same profile table. For example, if one analytical member is assigned an Australian UB150X18.0, then the other analytical members must be assigned any profile from the Austrialin UB table.

D11.A.5.2.3 Calculation for Compression Buckling

For calculation of member compression buckling capacities about the principal axis, the PMEMBER Design uses the concept of unbraced segments against compression (Lz, Ly). The unbraced segments against compression buckling are evaluated using the user defined compression bracings (PBCRES) separately for ZZ and YY axis. The types of restraints that can be provided at any location of the PMEMBER using PBCRES are: U(unbraced), T(translational), R(rotational), and TR(translational-rotational). Compression buckling restraints automate following design calculations for each segment for buckling against both the principle axis:

- Unbraced length of a segment for compression buckling – the distance between two consecutive T/R/TR type restraint. The program calculates it only when LZ or LY are not assigned.
- Effective length factor, k_e – according to figure 4.8.3.2 of NZS3404:1997 specification using type of restraint (T/R/TR) at the ends of a segment/sub-segment. The program calculates it only when KZ and KY are not assigned.

If you do not assign any compression buckling restraint to the physical member, the program will assume T type restraints at the member ends. But if a member is cantilever, then the program will keep the free end of the member unrestrained. Any user-provided input against member end restraints will override the values assumed by program.

Any value, other than 1.0, given to KZ and KY parameter will replace the automatically calculated effective length factors for the entire PMEMBER.

Any value other than 0.0 (default), given to LZ or LY parameter will replace the automatically calculated unbraced lengths for compression buckling of the entire PMEMBER.

D11.A.5.2.4 Calculation for Flexural/Flange Buckling

For calculation of member bending capacities about the principal x-axis, the PMEMBER Design uses the concept of unbraced segments against flange buckling (UNT, UNB). The unbraced segments against flange buckling are evaluated using the user defined compression bracings (PBRACE) separately for TOP and BOTTOM flange. User-defined flange restraints assigned using the PBRACE design parameter are checked to see if they are effective (i.e., if they are placed on the critical flange as per NZS3404 5.5). Flange restraints not applied to the critical flange are ineffective and hence are completely ignored.

Segment layouts for PMEMBERs may change for different load cases considered for design. Some restraints may be effective for one particular load case as they are found to apply to the critical flange, however for another load case may be found not to act on the critical flange, and found to be ineffective. In other words, the critical flange can change for each load case considered.

Typically, the critical flange will be the compression flange, except for segments with a “U” restraint at one end, in which case it will be the tension flange (as is the case for a cantilever).

Design unbraced flange segments are evaluated by “F”, “P”, “L”, “FR”, “PR”, or “U” effective section restraints. L restraints are considered only if are deemed to be “effective.” L restraints are only considered to be effective when positioned on the “critical” flange between F, P, FR, or FP restraints. If an L restraint is positioned on the non-critical flange it is ignored. Further, if an L restraint is positioned between a U and an F, P, FR, or FP restraint, it is ignored (regardless of whether it is on the critical or non-critical flange).

The PMEMBER Design uses the following steps to determine effective cross-section restraints for each load case considered:

i. first all user-defined restraints are checked to see if they are applied to the compression flange, with those that are not ignored

ii. next a check is made to see if a “U” type restraint is found at either end of the PMEMBER. If this is the case then any adjacent “L” restraints up to the next “F”, “FR”, “P” or “PR” restraint are also ignored, regardless of whether they are placed on the critical or non-critical flange. Refer NZS3404 5.4.2.3.

The compression flange in step i) of the routine above is calculated based on the bending moments at the locations of the restraints being considered.

Restraint Type	Definition	Stiffness	Description
FR	Fully and rotationally restrained	Most stiff	
F	Fully restrained	↓	
PR	Partially and rotationally restrained	↓	
P	Partially restrained	↓	
L	Laterally restrained	↓	Cannot be specified at the ends of design members.

Restraint Type	Definition	Stiffness	Description
U	Unrestrained	↓	Can only be applied at the ends of design members, and must be applied to both flanges to be effective. Both top and bottom flanges cannot be unrestrained at the same location (as this is unstable).
None		Least stiff	
C	Continuous restrained up to next restraint location		The flange is assumed to be continuously supported at that flange up to next restraint location. For continuously supported flange unbraced length is assumed to be zero.

Segment lengths are not automatically checked to determine if they provide full lateral restraint as per NZS3404 5.3.2.4.

For design of cantilevers, the free tip should have user-defined “U” restraints applied to both top and bottom flanges.

If the effective flange restraints for any load case consist of “U” or “L” restraints only, an error will be reported.

Hence flange restraints automate following design calculations for each unbraced flange segment for bending about both the principle axis:

- Unbraced length of top flange in bending compression – only when no value is assigned to `UNT`
- Unbraced length of bottom flange in bending compression – only when no value is assigned to `UNB`
- Moment modification factor, am, per NZS3404 5.6.1.1 – only when no value is assigned to `ALM`
- Load height factor, kl, given in Table 5.6.3(2) - only when no value is assigned to `SKL`
- Lateral rotation restraint factor, kr, given in Table 5.6.3(3) - only when no value is assigned to `SKR`
- Twist restraint factor, kt, given in Table 5.6.3(1) - only when no value is assigned to `SKT`

Notes:

a. If PMEMBER list is not provided, all the PMEMBERS are restrained by same configuration
b. It is not necessary to provide the restraint locations in sequence as the program sorts them automatically
c. Unless specified, PMEMBER ends are assumed to be Fully Restrained (F)
d. While designing any section of the member, effective restraints are searched on each side of the section along the critical flange
e. The types of restraints applied to the top and bottom flanges at each location determines the effective section restraints. These are outlined in the table below:

Case	Flange	Restraint on a Critical Flange	Restraint on a Non-Critical Flange	Effective Section Restraint
I		U	U	U
II	1	L	Nothing	L
	2	Nothing	L	None
III	1	P or F	Nothing or U	F
	2	Nothing or U	P or F	P
IV	1	PR or FR	Nothing or U	FR
	2	Nothing or U	PR or FR	PR
V	1	L, P or F	L, P, F, FR or PR	F
	2	FR or PR	L, P, F, FR or PR	FR

D11.A.5.2.5 Automated PMEMBER Design Calculations

The NZS3404 PMEMBER Design automates many design calculations, including those required for segment / sub-segment design.

Automated Design Calculations	PMEMBER Design Parameter	Comments
ab compression member section constant per NZS3404 6.3.3.	Not applicable	Calculated from table 6.3.3.
am moment modification factor per NZS3404 5.6.1.1 or Table 5.6.2 depending on if the segment is restrained at on/both ends.	ALM	Calculated based on moments distribution for individual segments and sub-segments.
f_u tensile strength per NZS3404 2.1.2 and AS4100 table 2.1.	FU	Based on nominal steel grade specified using SGR design parameter and element thickness.
fy yield stress per NZS3404 2.1.1 and AS4100 table 2.1.	FYLD	Based on nominal steel grade specified using SGR design parameter and element thickness.
Residual stress category for NZS3404 Table 5.2 and 6.2.4.	IST	Based on section type.
Segment and sub-segment layout for flange unbraced length.	PBRACE	Refer to the Segment or Sub-Segment for Flexural/Flange Buckling for details.
Unbraced length of top and bottom flange for flange buckling.	UNT, UNB	

Automated Design Calculations	PMEMBER Design Parameter	Comments
Segment layout for compression buckling unbraced length.	PBCRES	Refer to the Segment for Compression Buckling for details.
Effective length factor (ke) for compression buckling	KZ, KY	
Unbrace length for compression buckling	LZ, LY	
k_t twist restraint factor as per NZS3404 Table 5.6.3(1).	SKT	Based on effective end restraints for each segment / sub-segment.
k_l load height factor as per NZS3404 Table 5.6.3(2).	SKL, LHT	Based on effective end restraints for each segment / sub-segment, and LHT design parameter.
k_r lateral rotation restraint factor as per NZS3404 Table 5.6.3(3).	SKR	Based on effective end restraints for each segment / sub-segment. This is where the distinction between "F" and "FR", as well as "P" and "PR" is used.

D11.A.5.2.6 Load Height Position

When LHT is set to 1.0 to specify a top flange load height position, STAAD.Pro takes the top to be the positive local y-axis of the member.

Tip: This may not literally be the top flange for say a column or beam with a beta angle. The local member axes can be viewed in the user interface by selecting **Beam Orientation** in the **Diagrams Labels** dialog (or press **<Ctrl+O>**).

To automate kl using NZS3404 Table 5.6.3(2), the longitudinal position of the load also needs to be considered, i.e., as either "within segment" or "at segment end."

To determine which of these applies, the shear forces at the ends of each design segment / sub-segment is considered. If the shear force is found to have the same direction and magnitude at both ends, it is assumed that loads act at the segment end.

If on the other hand, the shear force at each end is found to have different directions or magnitudes, loads are assumed to act within the segment.

Tip: The above method includes an allowance for the self-weight of the member to be considered, as the self-weight always acts through the shear center.

The net sum of the end shears is also used to determine if the load is acting in the positive or negative local member y-axis direction. If LHT is set to 1.0 for top flange loading, the net sum is used to determine whether the top flange loading is acting to stabilize or destabilize the member for lateral torsional buckling. Negative local y-axis net loads act to destabilize the segments / sub-segments, whereas positive local y-axis net loads act to stabilize segments / sub-segments.

D11.A.6 Design Parameters

he design parameters outlined in the following table are used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs. The design scope indicates whether design parameters are applicable for MEMBER Design, PMEMBER Design, or both.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 181: New Zealand Steel Design Parameters

Parameter Name	Default Value	Design Scope	Description
CODE	-		Must be specified as NZS3404 1997 to invoke design per NZS 3404-1997. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
ALM	1.0		Moment modification factor (refer cl. 5.6.1.1) If ALM is 0.0 or not specified, it is automatically calculated based cl.5.6.1.1; otherwise the input value is used.
BEAM	1.0		Design locations: 0 = design only for end moments and those at locations specified by SECTION command. 1 = Perform design for moments at twelfth points along the beam.

Parameter Name	Default Value	Design Scope	Description
DFF	None	Analytical members only	"Deflection Length"/ Maximum Allowable local deflection. See DFF, DJ1, and DJ2 Parameters (on page 1781) See TR.40 Load Envelope (on page 2663) for deflection checks using serviceability load envelopes.
DJ1	Start Joint of member	Analytical members only	Joint No. denoting start point for calculation of "deflection length"
DJ2	End Joint of member	Analytical members only	Joint No. denoting end point for calculation of "deflection length"
DMAX	1000.0 mm		Maximum allowable depth (Applicable for member selection)
DMIN	0.0 mm		Minimum required depth (Applicable for member selection)
DUCT	0		Seismic category of member (sec. 12.2.5): 0 = Non-seismic (section 12 will not be checked) 1 = High ductile 2 = Limited ductile 3 = Nominal ductile 4 = Non-ductile
FU	500 MPa.		Ultimate strength of steel. If not specified, ultimate strength is calculated from SGR as per SGR Parameter (on page 1783).

Parameter Name	Default Value	Design Scope	Description
FYLD	248.0 MPa		Yield strength of steel. If not specified, yield strength is calculated from SGR as per SGR Parameter (on page 1783).
GLD	blank		Gravity load case number. By default, no load case will be selected as gravity load case.
IST	Refer to IST Parameter (on page 1782).		Residual stress category: 1 - SR 2 - HR 3 - CF 4 - LW 5 - HW If not specified, IST is evaluated depending upon shape. (See IST Parameter (on page 1782) for details)
KT	1.0		Correction factor for distribution of forces (refer cl. 7.2)
KY	1.0		K value for general column flexural buckling about the local Y/Z-axis. Used to calculate slenderness ratio. If specified, overrides the value calculated by program from restraint definition (PBCRES).
KZ	1.0		
LHT	0	Physical members only	Load height position as described in Table 5.6.3(2) of NZS3404:1997 0 = at Shear center 1 = At top flange (See LHT Parameter (on page 1782))

Parameter Name	Default Value	Design Scope	Description
LX	Member Length		The distance between partial or full restraints which effectively prevent twist of the section about its centroid. (sec. 8.4.4.1.2)
LY			Length for general column flexural buckling about the local Y/Z-axis. Used to calculate slenderness ratio. If specified, overrides the value calculated by program from restraint definition (PBCRES).
LZ			
MAIN	180.0		A value less than or equal to 1.0 suppresses the slenderness ratio check. checks are not explicitly required per NZS3404. Any value greater than 1.0 is used as the limit for slenderness in compression.
NSC	1.0		Net section factor for compression members = An / Ag (refer cl. 6.2.1)
NSF	1.0		Net section factor for tension members.
PBRACE	None	Physical members only	Refer to PBRACE Parameter (on page 1784) for details on the PBRACE parameter.
PBCRES	None		Refer to PBCRES Parameter (on page 1784) for details on the PBCRES parameter.
RATIO	1.0		Permissible ratio of actual load effect to the design strength.

Parameter Name	Default Value	Design Scope	Description
SGR	0		Steel Grade. 0 = Default (see note) 1 = AS 3679.1 - 350 2 = AS 3679.1 - 300 (default for standard rolled sections) 3 = AS 3679.1 - 250 4 = AS 1163 - C450 5 = AS 1163 - C350 6 = AS 1163 - C250 (default for tube sections) 7 = AS 3678 - 450 8 = AS 3678 - 400 9 = AS 3678 - 350 10 = AS 3678 - WR350 11 = AS 3678 - 300 (default for welded profiles and UPT) **Note:** Refere to SGR Parameter (on page 1783)
SKL	1.0		A load height factor given in Table 5.6.3(2). If not specified or specified as 0.0, will be automatically calculated.
SKR	1.0		A lateral rotation restraint factor given in Table 5.6.3(3). If not specified or specified as 0.0, will be automatically calculated.
SKT	1.0		A twist restraint factor given in Table 5.6.3(1). If not specified or specified as 0.0, will be automatically calculated.

Parameter Name	Default Value	Design Scope	Description
TMAIN	400.0		A value less than or equal to 1.0 suppresses the slenderness ratio check. checks are not explicitly required per NZS3404. Any value greater than 1.0 is used as the limit for slenderness in tension.
TRACK	0		Output detail 0 = report only minimum design results 1 = report design strengths in addition to TRACK 0 output 2 = provide full details of design
TSP	0.0		Spacing between transverse stiffener provided in the web. If any value less than or equal to 0.0 is specified, clear depth of web will be used.
UNB	Member Length		Unsupported length in bending compression of the bottom and top flange respectively for calculating moment resistance. If not equal to member length, overrides the value calculated by program from restraint definition (PBRACE).
UNT	Member Length		

DFF, DJ1, and DJ2 Parameters

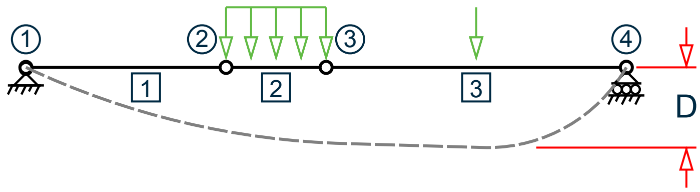
(Analytical members only) Deflection calculations are not applicable to PMEMBERs.

Compute Delta = $\sqrt{(\mathtt{DX2} - \mathtt{DX1})^2 + (\mathtt{DY2} - \mathtt{DY1})^2 + (\mathtt{DZ2} - \mathtt{DZ1})^2}$

Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

- A straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured.

For example, refer to the figure below where a beam has been modeled using four joints and three members. The "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters `DJ1` and `DJ2` should be used to model this situation. Thus, for all three members here, `DJ1` should be 1 and DJ2 should be 4.




```
PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL
```

D = Maximum local deflection for members 1, 2, and 3.

- If `DJ1` and `DJ2` are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.
- It is important to note that unless a `DFF` value is specified, STAAD.Pro will not perform a deflection check. This is in accordance with the fact that there is no default value for `DFF`.

IST Parameter

`IST` parameter is used to specify the residual stress category referred in table 5.2 and 6.2.4.

If `IST` is specified, then that value is used. But if not specified then residual stress category automated from shape of the section according to Note 1 of table 5.2.

Profile	Residual stress Category
Australian UB, UC, Angle, Channel, Tee, SHS, RHS, CHS	HR-hot-rolled or hot-finished
Australian cold formed shapes	CF-cold formed
Any other Australian shape	HW-heavily welded longitudinally
Any shape from any country other than Australia	

LHT Parameter

If the shear force is constant within the segment, longitudinal position of the load is assumed to be at the segment end.

If there is any variation of the shear force and the load is acting downward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU), Kl is taken to be 1.4.

If there is any variation of the shear force and the load is acting upward determined from shear force variation and load height parameter indicates the load is acting on top flange (flange at the positive local y axis) and restraints at the end of the segment is not FU (FRU) or PU (PRU) Kl is taken to be 1.0 as the load acting at the top flange is contributing to stabilize against local torsional buckling.

SGR Parameter

NZS3404 defines the values of steel grades that are used as either normal steel or high grade steel. The following table explains the material values used when either option is specified for a particular shape:

SGR Value	Steel Grade Used	Description
0	Default	AS/NZS 3679.1 300 for rolled sections, AS 1163 C250 for hollow sections (Pipe, Tube, HSS), AS 3678 300 for welded sections (WB, WC, UPT, Tapered)
1	AS/NZS 3679.1 350	
2	AS/NZS 3679.1 300	
3	AS/NZS 3679.1 250	
4	AS 1163 C450	
5	AS 1163 C350	
6	AS 1163 C250	
7	AS 3678 450	
8	AS 3678 400	
9	AS 3678 350	
10	AS 3678 WR350	
11	AS 3678 300	

If a value for the FYLD parameter has been specified, then that value will be used. Otherwise, the SGR value will be used to determine the yield strength and tensile strength values for the steel. based on maximum thickness of the individual elements of the section. Only for shear capacity calculation web thickness is used. Similarly, Tensile Strength is determined either from FU parameter or from SGR parameter.

A check is introduced to see if yield stress is less than 100 MPa or more than 450 MPa or not. If less than 100 MPa, a warning is issued but the yield stress will not be modified. If more than 450 MPa, then also a warning is issued and the yield stress is set to 450 MPa.

Example of Member Design

The following example uses the Member design facility in STAAD.Pro. However, it is strongly recommended to use the Physical member design capabilities for NZS3404:

```
PARAMETER 1
CODE NZS3404 1997
ALB 0.0 MEMBER ALL
ALM 1.13 MEMBER ALL
BEAM 1.0 MEMBER ALL
```

```
DFF 250.0 MEMBER ALL
DMAX 0.4 MEMBER ALL
DMIN 0.25 MEMBER ALL
FU 400.0 MEMBER ALL
FYLD 310.0 MEMBER ALL
IST 2.0 MEMBER ALL
KT 0.85 MEMBER ALL
KX 0.75 MEMBER ALL
KY 1.0 MEMBER ALL
LX 4.5 MEMBER ALL
LY 6.0 MEMBER ALL
MAIN 1.0 MEMBER ALL
NSC 0.9 MEMBER ALL
NSF 1.0 MEMBER ALL
PHI 0.9 MEMBER ALL
RATIO 0.9 MEMBER ALL
SGR 2.0 MEMBER ALL
SKT 1.0 MEMBER ALL
SKL 1.0 MEMBER ALL
SKR 1.0 MEMBER ALL
TRACK 2.0 MEMBER ALL
UNB 3.4 MEMBER ALL
UNT 6.8 MEMBER ALL
CHECK CODE MEMBER ALL
```

PBRACE Parameter

`PBRACE {TOP | BOTTOM} f1 r1 f2 r2 … f52 r52 (PMEMB` *`pmember-list`*`)`

where:

f*n* a fraction of the PMEMBER length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.

r*n* one of the possible restraint condition:

U
F
P
L
FR
PR
C

PBCRES Parameter

`PBCRES {ZZ | YY} f1 r1 f2 r2 … f52 r52 (PMEMB` *`pmember-list`*`)`

where:

f*n* a fraction of the PMEMBER length where restraint condition is being specified. This value is any ratio between 0.0 and 1.0.

r*n* one of the possible restraint condition:

U
T
R
TR

Example of PMEMBER Design

```
PARAMETER 1
CODE NZS3404 1997
DMAX 0.4 PMEMBER 20 TO 25
DMIN 0.25 PMEMBER 20 TO 25
KX 0.75 PMEMBER 20 TO 25
KY 1.0 PMEMBER 20 TO 25
LX 4.5 PMEMBER 20 TO 25
LY 6.0 PMEMBER 20 TO 25
LHT 0.0 PMEMBER 20 TO 25
NSC 0.9 PMEMBER 20 TO 25
NSF 1.0 PMEMBER 20 TO 25
PBRACE BOTTOM 0.0 F 1.0 F PMEMBER 20 TO 25
PBRACE TOP 0.0 P 0.5 L 1.0 P PMEMBER 20 TO 25
PBCRES ZZ 0.0 TR 0.3 R 1.0 U
PBCRES YY 0.0 TR 1.0 U
SGR 0.0 PMEMBER 20 TO 25
TRACK 2.0 PMEMBER 20 TO 25
CHECK CODE PMEMBER 20 TO 25
```

Note: Parameters for PMEMBER design must be listed using member numbers. The `ALL` list option may not be used for PMEMBERs.

D11.A.7 Output Format

Results of code checking and member selection are presented in a tabular format. The term `CRITICAL COND` refers to the section of the NZS3404 specification which governs the design.

D11.A.7.1 TRACK 0

```
 |--------------------------------------------------------------------------------|
|* PMemberPMEMBER for physical member, MEMBER for analytical member Number:
1                                                         |
|  Member Section:  ST   PFC250          (AISC SECTIONS)                        |
|  Status: FAIL  Ratio:   4.381  Critical Load Case:     1  Location:      0.42  |
|  Critical Condition: Cl.5.1                                                   |
|  Critical Design Forces:  (Unit: KN    METE)                                  |
|      FX:       4.448E+00 C     FY:       0.000E+00      FZ:      43.563E+00    |
|      MX:       0.000E+00       MY:       0.000E+00      MZ:       0.000E+00    |
|--------------------------------------------------------------------------------|
```

D11.A.7.2 TRACK 1

```
 |--------------------------------------------------------------------------------|
|* PMemberPMEMBER for physical member, MEMBER for analytical member Number:
1                                                         |
|  Member Section:  ST   PFC250          (AISC SECTIONS)                        |
```

```
|  Status: FAIL  Ratio:   4.381  Critical Load Case:     1  Location:       0.42   |
|  Critical Condition: Cl.5.1                                                      |
|  Critical Design Forces:  (Unit: KN    METE)                                     |
|      FX:       4.448E+00 C     FY:       0.000E+00      FZ:      43.563E+00       |
|      MX:       0.000E+00       MY:       0.000E+00      MZ:       0.000E+00       |
|----------------------------------------------------------------------------------|
| φMsz  =     113.670E+00 KNm     φMsy  =      24.010E+00 KNm     [Cl.5.1    ]      |
| φMbz  =      43.939E+00 KNm                                     [Cl.5.1    ]      |
| φVvy  =     324.000E+00 KNm     φVvz  =     437.400E+00 KNm     [Cl.5.12.3 ]      |
| φNs   =       1.220E+03 KN                                      [Cl.6.1    ]      |
| φNcz  =     601.788E+00 KN      φNcy  =      74.742E+00 KN      [Cl.6.1    ]      |
| φNt   =       1.220E+03 KN                                      [Cl.7.1    ]      |
| φMrz  =     113.670E+00 KNm     φMry  =      24.010E+00 KNm     [Cl.8.3.2  ]      |
| φMiz  =     113.670E+00 KNm     φMiy  =      24.010E+00 KNm     [Cl.5.3.2.4]      |
| φMozc =       0.000E+00 KNm     φMozt =       0.000E+00 KNm     [          ]      |
| φMcz  =       0.000E+00 KNm                                     [          ]      |
| φMtz  =       0.000E+00 KNm                                     [          ]      |
|----------------------------------------------------------------------------------|
```

D11.A.7.3 TRACK 2

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
**************************************************


MEMBER DESIGN OUTPUT FOR PMEMBERPMEMBER for physical member, MEMBER for analytical
member     1
DESIGN Notes
------------

1. (*) next to a Load Case number signifies that a P-Delta analysis has not been
performed for
that particular Load Case; i.e. analysis does not include second-order effects.
2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
3. (#) next to Young's modulus E indicates that its value is not 200000 MPa as per
NZS3404 1.4.

DESIGN SUMMARY
--------------

Designation: ST   PFC250                  (AISC SECTIONS)
Governing Load Case:     1*
Governing Criteria: Cl.5.1
Governing Ratio:   4.381 *(FAIL)
Governing Location:   4.500 m from Start.

SECTION PROPERTIES
------------------

d:       250.0000 mm   bf:        90.0000 mm
tf:        15.0000 mm   tw:         8.0000 mm
Ag:      4520.0000 mm2   J:   238.0000E+03 mm4             Iw:    34.8250E+09 mm6
Iz:    45.1000E+06 mm4  Sz:   421.0000E+03 mm3 (plastic)   Zz:   360.8000E+03 mm3
(elastic)
rz:    99.8893E+00 mm
Iy:     3.6400E+06 mm4  Sy:   107.0000E+03 mm3 (plastic)   Zy:    59.2834E+03 mm3
(elastic)
ry:    28.3780E+00 mm
```

```
STAAD SPACE                                              -- PAGE NO.    9


MATERIAL PROPERTIES
-------------------

Material Standard        :  AS/NZS 3679.1
Nominal Grade            :  300
Residual Stress Category :  HR (Hot-rolled)
E (#)       : 204999.984 MPa         [NZS3404 1.4]
G           :  80000.000 MPa         [NZS3404 1.4]
fy, flange  :    300.000 MPa         [NZS3404 Table 2.1]
fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
fu          :    440.000 MPa         [NZS3404 Table 2.1]

BENDING
-------
Section Bending Capacity (about Z-axis)

Critical Load Case :     1*
Critical Ratio     :   1.694
Critical Location  :   4.500 m from Start.
Mz* =  -192.5156E+00 KNm
Section Slenderness: Compact
Zez =   421.0000E+03 mm3
φMsz =   113.6700E+00 KNm               [NZS3404 Cl.5.1    ]

Section Bending Capacity (about Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
My* =     0.0000E+00 KNm
Section Slenderness: Compact
Zey =    88.9251E+03 mm3
φMsy =    24.0098E+00 KNm               [NZS3404 Cl.5.1    ]

Member Bending Capacity

Critical Load Case :     1*
Critical Ratio     :   4.381
Critical Location  :   4.500 m from Start.
Crtiical Flange Segment:
Location (Type):   0.00 m(F )-  9.00 m(F )
Mz* =  -192.5156E+00 KNm
kt   =      1.00                        [NZS3404 Table 5.6.3(1)]
kl   =      1.00                        [NZS3404 Table 5.6.3(2)]
kr   =      1.00                        [NZS3404 Table 5.6.3(3)]
le   =      9.00 m                      [NZS3404 5.6.3]
αm   =     1.384                        [NZS3404 5.6.1.1.1(b)(iii)]
Mo   =    42.2526E+00 KNm               [NZS3404 5.6.1.1.1(d)]
αsz  =     0.279                        [NZS3404 5.6.1.1.1(c)]
φMbz =    43.9390E+00 KNm (<= φMsz)     [NZS3404 5.6.1.1.1(a)]
STAAD SPACE                                              -- PAGE NO.   10


SHEAR
-----
```

```
Section Shear Capacity (along Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.134
Critical Location  :   0.000 m from Start.
Vy*  =    43.5625E-03 KN
φVvy =   324.0000E-03 KN                 [NZS3404 5.11.2]

Section Shear Capacity (along Z-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
Vz*  =     0.0000E+00 KN
φVvz =   437.4000E-03 KN                 [NZS3404 5.11.2]
STAAD SPACE                                                -- PAGE NO.   11


AXIAL
-----
Section Compression Capacity

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Ae    =     4.5200E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
kf    =     1.000                        [AS 4100 6.2.2]
An    =     4.5200E+03 mm2
φNs   =     1.2204E+03 KN                [NZS3404 6.2.1]

Member Compression Capacity (about Z-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Unbraced Segment:
Location (Type):   0.00 m(U )-  9.00 m(U )
Lez   =      9.00 m
αb    =      0.50                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
λn,z  =    98.699                        [NZS3404 6.3.3]
λ,z   =   107.400                        [NZS3404 6.3.3]
ε,z   =     0.959                        [NZS3404 6.3.3]
αc,z  =     0.493                        [NZS3404 6.3.3]
φNcz  = 0.6018E+3 KN                     [NZS3404 6.3.3]

Member Compression Capacity (about Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
Unbraced Segment:
Location (Type):   0.00 m(U )-  9.00 m(U )
Ley   =      9.00 m
λn,y  =   347.417                        [NZS3404 6.3.3]
λ,y   =   350.403                        [NZS3404 6.3.3]
```

```
ε,y   =     0.569                       [NZS3404 6.3.3]
αc,y  =     0.061                       [NZS3404 6.3.3]
φNcy  = 0.7474E+2 KN                    [NZS3404 6.3.3]

Section Tension Capacity

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
N*    =     0.0000E+00 KN
kt    =      0.00                       [User defined]
An    =     4.5200E+03 mm2
φNt   =     1.2204E+03 KN               [NZS3404 7.2]
STAAD SPACE                                                 -- PAGE NO.   12


COMBINED BENDING AND AXIAL
------------------------
Section Combined Capacity (about Z-axis)

Critical Load Case :     1*
Critical Ratio     :   1.694
Critical Location  :   4.500 m from Start.
φMrz  =   113.6700E+00 KNm              [NZS3404 8.3.2]

Section Combined Capacity (about Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
φMry  =    24.0098E+00 KNm              [NZS3404 8.3.3]

Section Combined Capacity (Biaxial)

Critical Load Case :     1*
Critical Ratio     :   1.694
Critical Location  :   4.500 m from Start.
γ     =     1.400                        [NZS3404 8.3.4]

Member In-plane Capacity (about Z-axis)

Critical Load Case :     1*
Critical Ratio     :   1.694
Critical Location  :   4.500 m from Start.
φMiz  =   113.6700E+00 KNm              [NZS3404 8.4.2]

Member In-plane Capacity (about Y-axis)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
φMiy  =    24.0098E+00 KNm              [NZS3404 8.4.2]

Member Out-of-plane Capacity (Tension)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
```

```
αbc   =      0.00
φNoz  =     0.0000E+00 KN               [NZS3404 8.4.4.1.2]
φMoz,t=     0.0000E+00 KNm              [NZS3404 8.4.4.1]

Member Out-of-plane Capacity (Compression)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.
φMoz,c=     0.0000E+00 KNm              [NZS3404 8.4.4.2]

Member Biaxial Capacity (Tension)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.

Member Biaxial Capacity (Compression)

Critical Load Case :     1*
Critical Ratio     :   0.000
Critical Location  :   0.000 m from Start.

********************************************************************************
```

D12. Norwegian Codes

D12.A. Norwegian Codes - Steel Design per NS 3472 / NPD

STAAD.Pro is capable of performing steel design based on the Norwegian code NS 3472 *Steel structures. Design rules* (3rd Edition) and NPD 1993 *Veiledning om utforming, beregning og dimensjonering av stalkonstruksjoner. Sist enderet 1.* (*Guidance on the design, calculation and dimensioning of figures constructions. Revision 1*).

Design of members per NS 3472 / NPD requires the *STAAD ECC Super Code* SELECT Code Pack.

D12.A.1 General Notes

This user manual presents a description of the design basis, parameters and theory applied to STAAD.Pro for performing code checks according to NS 3472 ref. [1] and NPD ref. [5]. The code checks include:

- stability check (buckling)
- lateral buckling check
- yield check (von Mises)
- stability check including local plate buckling of un-stiffened pipe walls according to NPD

The code check is available for the following cross-section types:

- wide flange profiles (HEA, HEB, IPE etc.)
- pipe (OD xx ID xx)
- tube (RHS, HUP)
- channel
- angle type (only RA)
- rectangular massive box (prismatic)

- user table (wide flange, I-sections, tapered I, tube, channel and RA angle)

The code check is *not* available for the following cross-section types:

- Double angles
- Tapered tubes
- Prismatic sections with too few section parameters defined
- Other sections that are not in the "available" list above

Please note the following:

- NS 3472 and NPD code checking covered in this document are available through two separate STAAD.Pro Code check packages.
- This document is not a lecture in use of NS 3472 or NPD. This document explains how, and which parts of, the Norwegian steel codes that have been implemented in STAAD.Pro.
- When L-sections are used, the Code Check requires RA angle definition.
- Weld design is not included in the Norwegian code checks.
- The prismatic section defined in the code check (rectangular massive box) is not identical to the general prismatic profile defined in the STAAD.Pro analysis package.

EDR does not accept any liability for loss or damage from or in consequence for use of the program.

D12.A.1.1 Nomenclature

NS - refers to NS 3472 ref. [1]

NS2 - refers to NS 3472 ref. [6]

NPD - refers to NPD94 ref. [5]

D12.A.1.2 References

1. NS 3472 3.utg. 2001

Prosjektering av stålkonstruksjoner

Beregning og dimensjonering

2. STAAD.Pro Technical Reference Manual, Release 2002

3. NS 3472 1.utg. 1973

Prosjektering av stålkonstruksjoner

Beregning og dimensjonering

4. Roark &Young`s 5th edition

5. NPD utg. 1994

Veiledning om utforming, beregning og dimensjonering av stålkonstruksjoner. Sist

endret 1. oktober 1993.

6. NS 3472 2.utg.1984

Prosjektering av stålkonstruksjoner

Beregning og dimensjonering

D12.A.2 Basis for Code Checking

This section presents general information regarding the implementation of the Norwegian codes of practice for structural steel design. This manual describes the procedures and theory used for both NS and NPD.

In general NS is used for all cross sections and shapes listed in section 1 of this manual. An exception is the treatment and check of pipe members in framed structures. NS does not give specific details about the treatment of pipes. Section 3.4 explains how this is adopted when NS is selected for code checking.

The NPD however have a more thorough check of pipe members, and consider the effect of local buckling of the pipe wall in conjunction with the stability check. In addition, the NPD code gives joint capacity formulae for brace to chord connections for pipe members.

The design philosophy and procedural logistics are based on the principles of elastic analysis and ultimate limit state design. Two major failure modes are recognized:

- failure by overstressing
- failure by stability considerations

The following sections describe the salient features of the design approach. Members are proportioned to resist the design loads without exceeding the characteristic stresses or capacities and the most economic section is selected on the basis of the least weight criteria. It is generally assumed that the user will take care of the detailing requirements like the provision of stiffeners and check the local effects like flange buckling, web crippling, etc.

You are allowed complete control over the design process through the use of the parameters listed in Table 2.1. Default values of parameters will yield reasonable results in most circumstances. However, the user should control the design and verify results through the use of the design parameters.

D12.A.2.1 Calculation of Forces and Bending Moments

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary loading conditions and combinations provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary load combinations.

D12..A.2.2 Members with Axial Forces

For tension only members, axial tension capacity is checked for the ultimate limit stress. For compression members, axial compression capacity is checked in addition to lateral buckling and ultimate limit stress. The largest slenderness ratio (λ) shall not be greater than 250 according to NS 11.7 Stability is checked as per the procedure of NS 12.3. The buckling curves of NS fig. 3 have been incorporated into the STAAD.Pro code check. The coefficient α (as per NS Table 10) can be specified in both directions through the use of parameters CY and CZ. In the absence of parameters CY and /or CZ, default a- value will be according to NS table 11.

D12.A.2.3 Members with Axial Force and Bending Moments

For compression members with bending, interaction formulae of NS table 12.3.4.2 are checked for appropriate loading situation. All compression capacities are calculated per the procedure of NS 12.3.

The equivalent moment factor β is calculated using the procedure of NS table 12. Two different approaches are used depending upon whether the members can sway or not. Conditions for sidesway and transverse loading can be specified through the use of parameters SSY and SSZ. For members that cannot sway, without transverse loading, coefficients b are calculated and proper dimensioning moments are used in the interaction formulae.

D12.A.2.4 Lateral Buckling

Lateral torsional buckling is checked as per the procedure of NS 12.3.4. The procedure for calculation of ideal buckling moment for sections with two axis of symmetry has been implemented. The coefficient can be provided by the user through the use of parameter CB. In the absence of CB, a value of 1.0 will be used. Torsional properties for cross sections (torsional constant and warping constant) are calculated using formulae from NS 3472. This results in slightly conservative estimates of torsional parameters. The program will automatically select the maximum moment in cases where Mvd is less than Mzd.

D12.A.2.5 Von Mises Yield Criterion

Combined effect of axial, bending, horizontal/vertical shear and torsional shear stress is calculated at 13 sections on a member and up to 9 critical points at a section. The worst stress value is checked against yield stress divided by appropriate material factor. The von Mises calculates as:

$$\sigma_j = \sqrt{(\sigma_x + \sigma_{by} + \sigma_{bz})^2 + 3(\tau_x + \tau_y + \tau_z)^2} \le \frac{f_y}{\gamma_m}$$

D12.A.2.6 Material Factor and Nominal Stresses

The design resistances are obtained by dividing the characteristic material strength by the material factor.

NS 3472

The material factor default value is 1.10. Other values may be input with the MF parameter. The nominal stresses should satisfy

$$\sigma_j \le \frac{f_y}{\gamma_m} = f_d$$

NPD

The general requirement is according to NPD 3.1.1. For stability the NPD 3.1.1 and 3.1.3 requires that the structural coefficient is considered.

$$S_d \le f_{kd} = \frac{f_k}{\gamma_m \cdot \gamma_{mk}(S_d)}$$

Where:

S_d = reference stress or load effect resultant
f_k = characteristic capacity
f_{kd} = design capacity
γ_m = material coefficient
γ_{mk} = structural coefficient

γ_m is default set to 1.10.

γ_{mk} shall be equal to 1.0 for frames. For pipe members γ_{mk} is a function of the reduced slenderness. In the STAAD.Pro implemented NPD code this is calculated automatically.

D12.A.2.7 Code Checking According to NPD

The following parts of Chapter 3 in the NPD guidelines have been implemented.

a. Control of nominal stresses. (NPD 3.1.2).

b. Buckling of pipe members in braced frames, including interaction with local shell buckling (NPD 3.2.2, 3.2.3).

c. Buckling of un-stiffened closed cylindrical shells, including interaction with overall column buckling (NPD 3.4.4, 3.4.6, 3.4.7 and 3.4.9).

d. Joint capacity check for gap as well as for overlap joints (NPD 3.5.2).

Check b) provides the unity check based on the beam-column buckling interaction formulae in NPD 3.2.2. The interaction between global and local buckling due to axial load and hydrostatic pressure is accounted for through computation of an axial characteristic capacity to replace the yield stress inn the beam-column buckling formulae.

Note: Check b) handles members subjected to axial loads, bending moments and hydrostatic pressure. In other words, check b) assumes that stresses resulting from shear and torsion are of minor importance, e.g., in jacket braces.

Check c) provides the unity check based on the stability requirement for un-stiffened cylindrical shells subjected to axial compression or tension, bending, circumferential compression or tension, torsion or shear. The unity check refers to the interaction formulae in NPD 3.4.4.1. The stability requirement is given in NPD 3.4.7.

Norwegian Codes - Steel Design per NS 3472 / NPD

STAAD.Pro performs a stability check on aluminum alloys according to buckling curve in ECCS (European recommendation for aluminum ally structures 1978). It is possible to select heat-treated or non heat-treated alloy from the parameter list in the STAAD.Pro input file.

For heat-treated use CY = CZ = 0.1590, and for non heat-treated use CY = CZ = 0.2420.

Tracks 1.0 and 9.0 print buckling curve H for heat-treated, and buckling curve N for non neat-treated. The yield check is the same as for steel.

D12.A.3 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Table 182: Design Parameters for Norwegian Steel design code

Parameter Name	Default Value	Description	Reference
CODE	none	Must be specified as either NS3472 for NS or NPD for NPD (NOR may also be used for both). Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).	

Parameter Name	Default Value	Description	Reference
BEAM	0.0	Parameter `BEAM 1.0 ALL` tells the program to calculate von Mises at 13 sections along each member, and up to 8 points at each section. (Depending on what kind of shape is used.) **Note:** Must be set to 1.0	Sec. NS 12.2.2
BY	1.0	Buckling length coefficient, β for weak axis buckling (y-y) (NOTE: BY > 0.0)	Fig. NS 3 Sec. NS 12.3
BZ	1.0	Buckling length coefficient, β, for strong axis buckling (z-z) (NOTE: BZ > 0.0)	Fig. NS 3 Sec. NS 12.3
CB	1.0	Lateral buckling coefficient, Y. Used to calculate the ideal buckling moments, M_{vi}	Sec. NS2 A5.5.2 Fig. NS2 A5.5.2a)-e)
CMY	1.0	Water depth in meters for hydrostatic pressure calculation for pipe members	Valid for the NPD code only
CMZ	0.49	α_{LT} for sections in connection with lateral buckling	Sec. NS 12.3.4 Fig. NS 6.
CY CZ	Default see NS 3472	Buckling curve coefficient, a about local z-axis (strong axis). Represent the a, a0, b, c, d curve.	Fig. NS 3 Sec. NS 12.2 NS Table 11
DMAX	100.0 [cm]	Maximum allowable depth of steel section.	
DMIN	0.0 [cm]	Minimum allowable depth of steel section.	
FYLD	235	Yield strength of steel, fy (St37) [N/mm^2]	Tab. NS 3

Parameter Name	Default Value	Description	Reference
MF	1.1 (NS3472) 1.15 (NPD)	Material factor / Resistance factor, γ_m	Sec. NS 10.4.2 Sec. NPD 3.1
RATIO	1.0	Permissible ratio of the actual to allowable stresses.	Sec. NS 12.3.4.2
SSY	0.0	0.0 = No sidesway. β calculated. > 0.0 = Sidesway in local y-axis weak axis β_M=SSY	Sec. NS 12.3.4 Tab. NS 12 Sec. NPD 3.2.1.4
SSZ	0.0	0.0 = No sidesway. β calculated. > 0.0 = Sidesway in local y-axis weak axis β_M	Sec. NS 12.3.4 Tab. NS 12 Sec NPD 3.2.1.4
TRACK	0.0	Controls the level of detail in the output: 0.0 = Suppress critical member stresses. 1.0 = Print all critical member stresses, i.e., DESIGN VALUES 2.0 = Print von Mises stresses. 9.0 = Large output, 1 page for each member. Refer to D12.A.8 Tabulated Results (on page 1823) for complete list of available TRACKs and print examples.	
UNL	Member length	Effective length for lateral buckling calculations (specify buckling length). Distance between fork supports or between effective side supports for the beam	Sec. NS 12.3

The parameter CMY will, when given with negative value, define an inside pressure in pipe members. The pressure corresponds to given water depth in meters.

The parameter CB defines the φ value with respect to calculation of the ideal lateral buckling moment for single symmetric wide flange profiles, ref. NS app. 5.2.2.

D12.A.3.1 Example

Note: This is a partial example containing only the information pertaining to the Norwegian steel design code; used at the end of the input file.

```
* Code check according to NS3472
PARAMETERS
CODE NS3472
BEAM 1.0 ALL
FYLD 340 ALL
MF 1.10 ALL
CY 0.49 MEMB 1
CZ 0.49 MEMB 1
BY 0.9 MEMB 1
BZ 0.7 MEMB 1
SSY 1.1 MEMB 1
SSZ 1.3 MEMB 1
CB 0.9 MEMB 1
RATIO 1.0 ALL
TRACK 9.0 ALL
UNIT KNS METER
LOAD LIST 1
CHECK CODE MEMB 1
FINISH
```

D12.A.4 Stability Check According to NS 3472

The stability check is based on the assumption that both ends of the member are structural nodes. Buckling lengths and results for member with joints between the structural nodes have to be evaluated in each separate case.

Effects from local buckling or external hydrostatic pressure on pipes and tubes are not included.

The general stability criteria is: (ref. NS 12.3)

D12.A.4.1 Buckling

$$n_{max} + k^{z} \times m_{z} + k_{y} \times m_{y} \leq 1$$

D12.A.4.2 Lateral Buckling

$$\frac{n}{x_y} + k_{LT}\frac{m_z}{x_{LT}} + k_y m_y \leq 1$$

Where:

$i = z,y$

$n_{max} = n/\chi_{min}$

$n = N_f/N_d$

$\chi_{min} = \min(\chi_z,\chi_y)$

$\chi_i = N_{kd,i}/N_d$

$$k_i = 1 - \mu_i \frac{n}{\chi_i \gamma_m} \le 1.5$$

$\mu_i = {}_i(2 \cdot \beta_{Mi} - 4) \le 0.9$

β_{Mi} ref. NS Tab. 12

$$k_{LT} = 1 - \mu_{LT} \frac{n}{\chi_y \gamma_m} \le 1.0$$

$\mu_{LT} = 0.15({}_y \cdot \beta_M - 1) \le 0.9$

${}_i = \lambda_i/\lambda_1$

$\lambda_i = L_{ki}/i_i$

$$\lambda_i = \pi \sqrt{\frac{E}{f_y}}$$

$$\chi_i = \frac{1}{\phi + \sqrt{\phi^2 - \bar{\lambda}^2}}$$

$\varphi = 0.5[1 + \alpha(- 0.2) + {}^2]$

α ref. NS Tab 10 & 11

$$\chi_{LT} = \frac{1}{\phi_{LT} + \sqrt{\phi_{LT}^2 - \bar{\lambda}_{LT}^2}}$$

$$\phi_{LT} = 0.5\left[1 + \alpha\left(\bar{\lambda}_{LT} - 0.4\right) + \bar{\lambda}_{LT}^2\right]$$

α ref. NS sec. 12.3.4.1

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_z f_z}{M_{cr}}}$$

$M_{cr} = \psi \cdot M_{vio}$

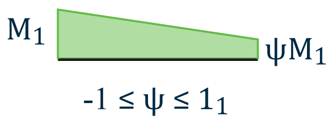
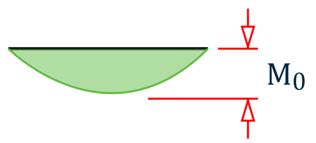
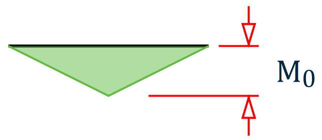
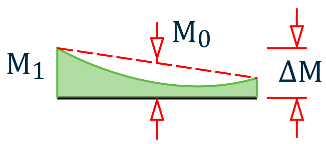
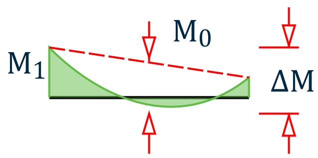
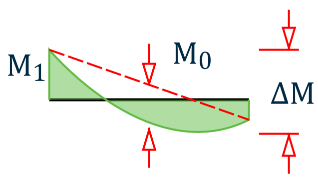
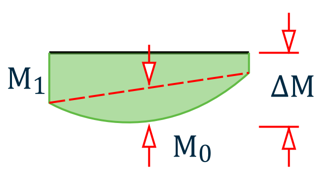
ψ ref. NS2 A5.5.2 Sect. a - d

$$M_{vio} = \frac{\pi}{L} \sqrt{E I_z G I_T} \sqrt{1 + \frac{\pi^2}{L^2} \frac{E C_w}{G I_T}}$$

D12.A.4.3 Determination of β_z and β_y

The equivalent moment factor β (for z and y) is calculated dependent on moment distributions as shown in the following table:

Table 183: β for different moment distributions

Moment diagram	β_M (β_{LT})
	$\beta_{M\,\psi} = 1.8 - 0.7\psi$
 	$\beta_{M\,0} = 1.3$ $\beta_{M\,0} = 1.4$
   	$\beta_M = \beta_{M,\psi} + \frac{M_0}{\Delta M}\left(\beta_{M,0} - \beta_{M,\psi}\right)$ $M_0 = \lvert M_{max} \rvert$ due to transverse load only $\Delta M = \lvert M_{max} \rvert$ if the moment has the same sign $\Delta M = \lvert M_{max} \rvert + \lvert M_{min} \rvert$ if the moment changes sign

The user can override the calculated factor with the following parameters:

β_y=SSY

β_z=SSZ

D12.A.4.4 Lateral buckling

The Ideal lateral buckling moment is calculated according to NS2 A5.5.2

$$M_{vi} = \psi M_{vio} = \psi L \ 95 \frac{E}{L} \sqrt{I_y I_x} \sqrt{1 + \frac{\pi^2}{L^2} \frac{2.6 C_w}{I_x}}$$

concern double symmetric cross sections where y is given in NS fig. A5.5.2, (input parameter CB), L = member length for lateral buckling (input parameter UNL), Cw and Ix , see section 5.

For single symmetric cross sections, the ideal lateral buckling moment is

$$M_{vix} = \phi \frac{\pi^2 E I_y}{L^2} \sqrt{\left(\frac{5a}{\pi^2} + \frac{r_x}{3} - y_s\right)^2 + C^2} - \left(\frac{5a}{\pi^2} + \frac{r_x}{3} - y_s\right)$$

Where:

$$C^2 = \frac{C_w + 0.039 L^2 I_T}{I_y}$$

α = distance from profile CoG to point where the load is acting, assumed to be on top flange.

The φ parameter (ref NS fig. A5.5.2.g) is controlled by the input parameter CB.

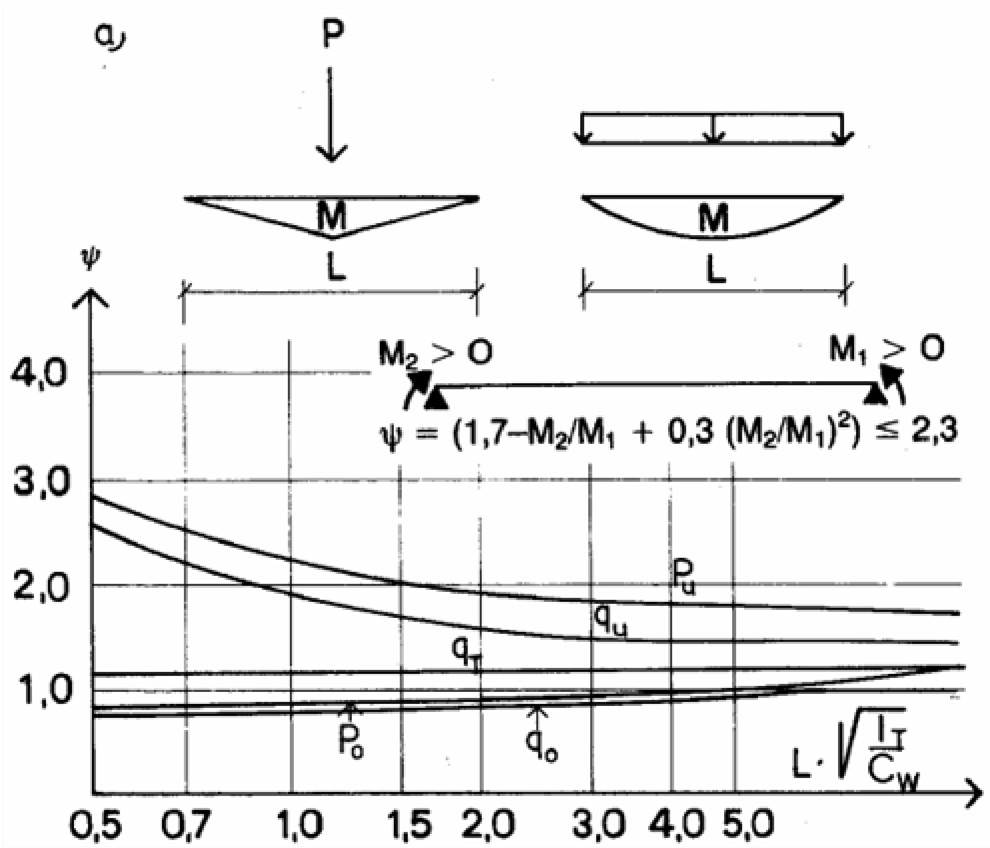


Figure 180: ψ-coefficients for a simple span beam

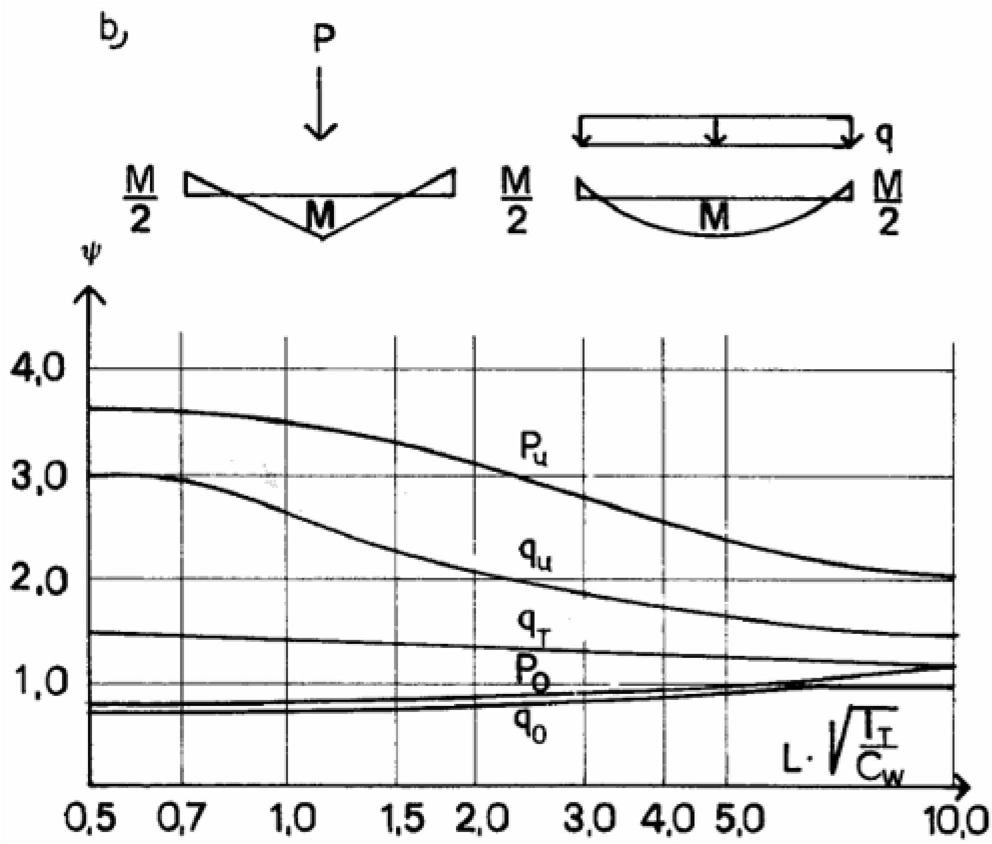


Figure 181: ψ-coefficients for a partially restrained beam

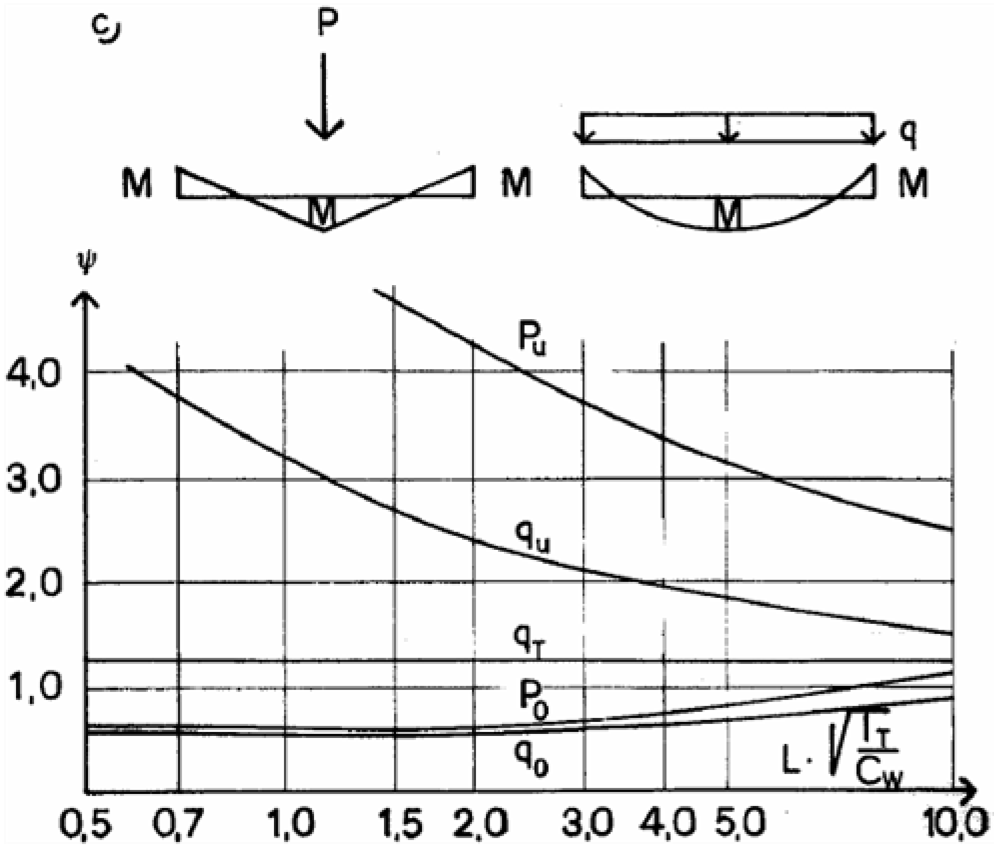


Figure 182: ψ-coefficients for a fully restrained beam

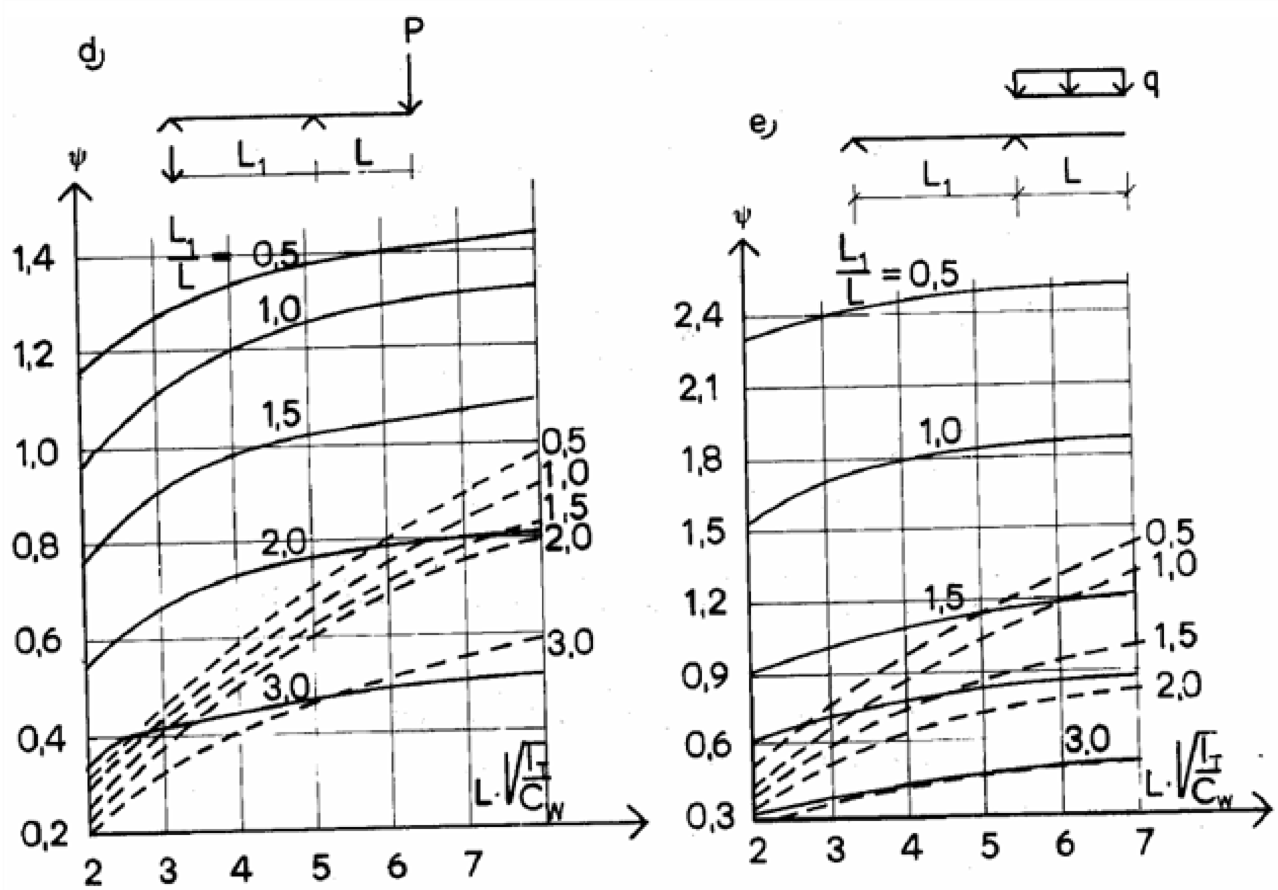


Figure 183: ψ-coefficients for the cantilevered beam with single loads and distributed loads. Dashed curves apply load on the surface.

D12.A.4.5 Stability check of pipe members

The stability criteria applied for members with pipe cross section is:

$$IR = \frac{N}{N_{kd}} + \sqrt{\left[\frac{\overline{M}_z}{M_d\left(1 - \frac{N}{N_{Ezd}}\right)}\right]^2 + \left[\frac{\overline{M}_y}{M_d\left(1 - \frac{N}{N_{Eyd}}\right)}\right]^2} \leq 1.0$$

Where:

$$\frac{N}{N_{kd}} = \max\left(\frac{N}{N_{kzd}}, \frac{N}{N_{kyd}}\right)$$

$\overline{M}_z$ and $\overline{M}_z$ are given in NS 5.4.2.

For the print output option `TRACK 9.0` $K_E \equiv 1.0$ and $M_{vd} \equiv M_d$

D12.A.4.6 Angle profiles type RA (reverse angle)

The axial contribution to the total interaction ratio is checked according to the modified EECS-method, see NS A5.4.

The stability criterion is:

$$IR = \frac{N}{N_{kd}} + \frac{\overline{M}_y}{M_{yd}\left(1 - \frac{N}{N_{Eyd}}\right)} + \frac{\overline{M}_z}{M_{zd}\left(1 - \frac{N}{N_{Ezd}}\right)} \leq 1.0$$

Where:

$$\frac{N}{N_{kd}} = \max\left(\frac{N}{N_{kzd}}, \frac{N}{N_{kyd}}\right)$$

N_{kyd} and N_{kzd} are found from NS 3472 fig. 5.4.la C-curve for y- and z-axis, respectively.

For $\bar{\lambda} \leq \sqrt{(2)}$

$$_{eff} = 0.60 + 0.57$$

For $\bar{\lambda} > \sqrt{(2)}$

$$_{eff} =$$

Where:

$$\bar{\lambda} = \frac{\lambda_k}{\pi}\sqrt{\frac{f_y}{E}}$$

$$\lambda_k = l_k/i$$

$$i = \sqrt{I / A}$$

Possible lateral buckling effects and torsional buckling (NS A5.4.5) is *not* included in the code check. This has to be evaluated by the user separately.

D12.A.4.7 Stability check of members with tapered section

Stability of members with tapered cross section is calculated as described in section 3.1. The cross section properties used in the formulae are calculated based on the average profile height. (i.e., I_z, I_y values are taken from the middle of the member.)

D12.A.4.8 Lateral buckling for tension members

When compressive stress caused by large bending moment about strong axis is greater than tension stress from axial tension force, lateral buckling is considered as defined below.

$$\sigma_a = N/A \text{ (+ tension, - compression)}$$

$$\sigma_{bz} = \pm M_z/W_z$$

$$M_{warp} = | \sigma_a + \sigma_b | W_z \text{ for } \sigma_a + \sigma_b < 0 \text{ (compression)}$$

$$IR = M_{warp}/M_{vd} + M_{y,max}/M_{yd} \leq 1.0$$

D12.A.5 Stability Check According to NPD

D12.A.5.1 Buckling of pipe members

Tubular beam-columns subjected to compression and lateral loading or end moments shall be designed in accordance with NPD 3.2.2

$$\sigma_c \gamma_{mk} + B\sigma_b^* + \sqrt{(B_z \sigma_{bz})^2 + (B_y \sigma_{by})^2} \le \frac{f_y}{\gamma_m}$$

Where:

σ_c = N/A = axial compressive stress
ν_{mk} = structural coefficient
B = bending amplification factor = 1/ (1 - μ), B is taken as the larger of B_z and B_y
B_z = bending amplification factor about the Z-axis
B_y = bending amplification factor about the Y-axis
$\mu = \sigma_c / f_E$
$f_E = \frac{\pi^2 E}{l_k^2} i^2$
$i = \sqrt{I / A}$
$\sigma_b^* = \sigma_c \left(\frac{f_y}{f_k} - 1 \right) \left(1 - \frac{f_k}{\gamma_m f_E} \right)$
l_k = kl
k = effective length factor
f_k = characteristic buckling capacity according to NS fig. 5.4.1a, curve A.

D12.A.5.2 Interaction with local buckling, NPD 3.2.3

If the below conditions are not satisfied, the yield strength will be replaced with characteristic buckling stress given in NPD 3.4.

a. members subjected to axial compression and external pressure

$$\frac{d}{t} \le 0.5 \sqrt{\frac{E}{f_y}}$$

b. members subjected to axial compression only

$$\frac{d}{t} \le 0.1 \frac{E}{f_y}$$

D12.A.5.3 Calculation of buckling resistance of cylinders

The characteristic buckling resistance is defined in accordance with NPD 3.4.4

$$f_k = \frac{f_y}{\sqrt{1 + \lambda^4}}$$

Where:

$$\lambda^2 = \frac{f_y}{\sigma_j} \left(\frac{\sigma_{ao}}{f_{ea}} + \frac{\sigma_{b0}}{f_{eb}} + \frac{\sigma_{p0}}{f_{ep}} + \frac{\tau}{f_{e\tau}} \right)$$

$$\sigma_j = \sqrt{(\sigma_a + \sigma_b)^2 - (\sigma_a + \sigma_b)\sigma_p + \sigma_p^2 + 3\tau^2}$$

$\sigma_a \ge 0$ when $\sigma_{a0} = 0$
$\sigma_a < 0$ when $\sigma_{a0} = -\sigma_a$
$\sigma_b \ge 0$ when $\sigma_{b0} = 0$

$\sigma_b < 0$ when $\sigma_{b0} = -\sigma_b$
$\sigma_p \geq 0$ when $\sigma_{p0} = 0$
$\sigma_p < 0$ when $\sigma_{p0} = \sigma_p$
σ_a = design axial stress in the shell due to axial forces (tension positive)
σ_b = design bending stress in the shell due to global bending moment (tension positive)
$\sigma_p = \sigma_\Theta$ = design circumferential stress in the shell due to external pressure (tension positive)
τ_S = design shear stress in the shell due to torsional moments and shear force.
f_{ea}, f_{eb}, f_{ep} and f_{et} are the elastic buckling resistances of curved panels or circular cylindrical shells subjected to axial compression forces, global bending moments, lateral pressure, and torsional moments and/or shear forces respectively.

D12.A.5.4 Elastic buckling resistance for un-stiffened, closed cylinders

The elastic buckling resistance for un-stiffened closed cylinders according to NPD 3.4.6 is:

$$f_e = k \frac{\pi^2 E}{12\left(1 - \nu^2\right)} \left(\frac{t}{l}\right)^2$$

where k is a buckling coefficient dependent on loading condition, aspect ratio, curvature, boundary conditions, and geometrical imperfections. The buckling coefficient is:

$$k = \psi \sqrt{1 + \left(\frac{p\xi}{\psi}\right)^2}$$

The values of ψ, ζ, and p are given in Table 4.1 for the most important loading cases.

Table 184: Table 4.1 Buckling coefficients for un-stiffened cylindrical shells

	ψ	ζ	p
Axial or Bending stress	1	$0.702\ Z$	$0.5\left(1 + \frac{r}{150t}\right)^{-0.5}$
Torsion and shear force	5.34	$0.856\ Z0.75$	0.6
Lateral pressure	4	$1.04\ Z0.5$	"
Hyrdostatic pressure	2	$1.04\ Z\ 0.5$	"

The curvature parameter is defined by

$$Z = \frac{1^2}{rt}\sqrt{1 - \nu^2}$$

For long shells the elastic buckling resistance against shear stresses is independent of shell length. For cases with:

$$\frac{1}{r} > 3.85\sqrt{\frac{r}{t}}$$

the elastic buckling resistance may be taken as:

$$f_{ep} = 0.25E\left(\frac{t}{r}\right)^2$$

D12.A.5.5 Stability requirements

The stability requirement for curved panels and un-stiffened cylindrical shells subjected to axial compression or tension, bending, circumferential compression or tension, torsion or shear is given by NPD 3.4.7:

$$\sigma_j < f_{kd}$$

where the design buckling resistance is

$$f_{kd} = \frac{f_k}{\gamma_m \gamma_{mk}}$$

D12.A.5.6 Column buckling, NPD 3.4.9

For long cylindrical shells it is possible that interaction between shell buckling and overall column buckling may occur because second-order effects of axial compression alter the stress distribution as compared to that calculated from linear theory. It is necessary to take this effect into account in the shell buckling analysis when the reduced slenderness of the cylinder as a column exceeds 0,2 according to NPD 3.4.4.1.

$\sigma_b$shall be increased by an additional compressive stress which may be taken as:

$$\Delta\sigma = B\sigma_a\left(\frac{f_y}{f_k} - 1\right)\left(1 - \frac{f_k}{f_e}\right) + \left(B - 1\right)\sigma_b$$

Where:

$$\bar{B} = \frac{1}{1 - \mu}$$

$$\bar{\lambda} = \sqrt{f_y / f_e}$$

$$f_e = \frac{\pi^2 E}{\lambda^2}$$

λ = slenderness of the cylinder as a column.

B, σ_a, σ_b, and μ are calculated in accordance with NPD 3.2.2.

D12.A.6 Yield Check

The yield check is performed at member ends and at 11 equally spaced intermediate sections along the member length.

At each section the following forces are applied:

- F_x max. axial force along member
- F_y actual shear in local y-direction at section
- F_z actual shear in local z-direction at section
- M_x max. torsional moment along member
- M_y actual bending about local y-axis at section
- M_z actual bending about local z-axis at section

For all profiles other than angle sections absolute values of the stresses are used. For

double symmetric profiles there will always be one stress point.

The stresses are calculated in several stress points at each member section. At each stress point the von Mises stress is checked as follows:

$$\sigma_j = \sqrt{\sigma_{tot}^2 + \sigma_p^2 - \sigma_{tot} \cdot \sigma_p + 3(\tau_x + \tau_y + \tau_z)^2} \le \frac{f_y}{\gamma_m}$$

Where:

$$\sigma_{tot} = | \sigma_x + \sigma_{by} + \sigma_{bz} |$$

σ_p stress from hydrostatic pressure.

D12.A.6.1 Double symmetric wide flange profile

The von Mises stress is checked at four stress points as shown in figure below.

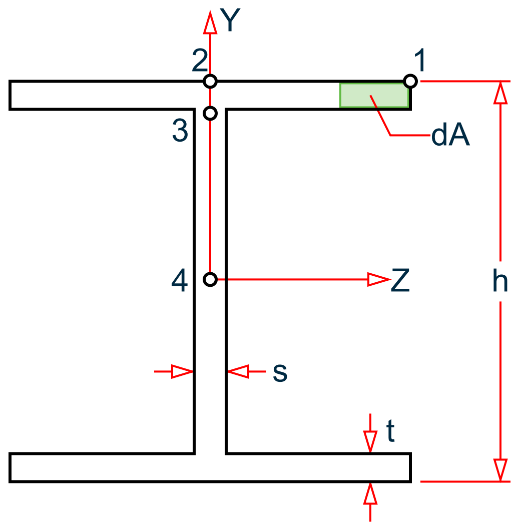


Figure 184: Stress points checked for a wide flange section

Section Properties

A_x , I_x , I_y, and I_z are taken from STAAD.Pro database
$A_y = h \times s$ Applied in STAAD.Pro print option PRINT MEMBER STRESSES

$$A_z = (2/3) \cdot b \cdot t \cdot 2$$

$$\tau_y = F_y/A_y$$

$$\tau_z = F_z/A_z$$

A_y and A_z are not used in the code check

$C_w = \frac{(h - t)^2 b^3 t}{24}$ ref. NS app. C3

$$T_y = dA \times z$$

$$T_z = dA \times y$$

Stress calculation

General stresses are calculated as:

$$\sigma = \sigma_x + \sigma_{by} + \sigma_{bz} = \frac{F_x}{A_x} + \frac{M_y}{I_y}z + \frac{M_z}{I_z}y$$

$$\tau = \tau_x + \tau_y + \tau_z = \frac{M_x}{I_x}c + \frac{V_y T_z}{I_z t} + \frac{V_z T_y}{I_y t}$$

Where the component stresses are calculated as shown in the following table:

Table 185: Stress calculations at selected stress points for a wide flange section

$h_1 = ½(h - 2 t)$

$h_2 = ½ (h - t)$

<table>
<tr><th>Point No</th><th>σ_x</th><th>σ_{by}</th><th>σ_{bz}</th><th>τ_x</th><th>τ_y</th><th>τ_z</th></tr>
<tr><td>1</td><td rowspan="4">$\frac{F_x}{A_x}$</td><td>$\frac{M_y}{I_y}\frac{b}{2}$</td><td rowspan="2">$\frac{M_z}{I_z}\frac{b}{2}$</td><td rowspan="2">$\frac{M_x}{I_x}t$</td><td>0</td><td>0</td></tr>
<tr><td>2</td><td>0</td><td>$\frac{F_y}{I_z}\frac{bth_2}{2t}$</td><td>$\frac{F_z}{I_y}\frac{tb^2}{8t}$</td></tr>
<tr><td>3</td><td>0</td><td>$\frac{M_z}{I_z}h_1$</td><td rowspan="2">$\frac{M_x}{I_x}s$</td><td>$\frac{F_y}{I_z}\frac{bth_2}{s}$</td><td>0</td></tr>
<tr><td>4</td><td>0</td><td>0</td><td>$\frac{F_y}{I_z}\frac{\left(bth_2 + 0.5h_1^2 s\right)}{s}$</td><td>0</td></tr>
</table>

In general wide flange profiles are not suitable for large torsional moments. The reported torsional stresses are indicative only. For members with major torsional stresses a separate evaluation has to be carried out. Actual torsional stress distribution is largely dependent on surface curvature at stress point and warping resistance.

D12.A.6.2 Single symmetric wide flange profile and tapered section

The von Mises stress is checked at nine stress points as shown in figure below.

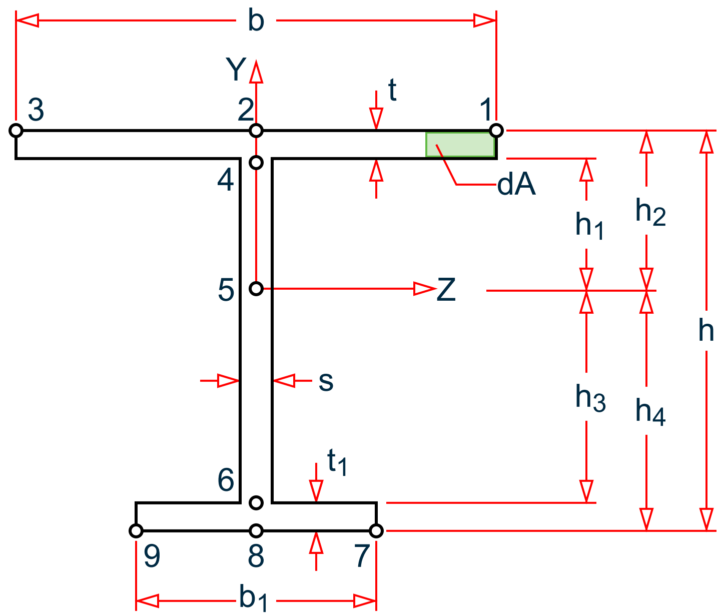


Figure 185: Stress points checked for a singly symmetric wide flange section

Section properties

A_x, I_x , I_y, and I_z are taken from STAAD.Pro database, except for tapered sections where these values are calculated for each section checked. (i.e., Iz, Iy values are taken from the middle of the member.)

$$A_z = 2/3(b \cdot t + b_1 \cdot t_1)$$

$$C_w = \frac{b^3 t \cdot b_1^3 t_1 (h - t/2 - t_1/2)^2}{12\left(b^3 t + b_1^3 t_1\right)}$$ ref. NS app. C3

Refer to D12.A.6.1 Double symmetric wide flange profile (on page 1808) for equations used in section property calculations.

Stress calculation

Refer to D12.A.6.1 Double symmetric wide flange profile (on page 1808) for equations used in general stress calculations.

Where the component stresses are calculated as shown in the following table:

Table 186: Stress calculations at selected stress points for a singly symmetric wide flange section

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$-\frac{M_y}{I_y}\frac{b}{2}$	$\frac{M_z}{I_z}h_2$	$\frac{M_x}{I_x}t$	0	0
2		0			$\frac{F_y}{I_z}\frac{bt(h_1 + t/2)}{2t}$	$\frac{F_z}{I_y}\frac{tb^2}{8t}$

Point No	**σ_x**	**σ_{by}**	**σ_{bz}**	**τ_x**	**τ_y**	**τ_z**
3		$\frac{M_y}{I_y}\frac{b}{2}$			0	0
4		0	$\frac{M_z}{I_z}h_1$	$\frac{M_x}{I_x}s$	$\frac{F_y}{I_z}\frac{bt(h_1+t/2)}{s}$	0
5		0	0		$\frac{F_y}{I_z}\frac{bt(h_1+t/2)+0.5h_1^2 s}{s}$	0
6		0	$-\frac{M_z}{I_z}h_3$		$\frac{F_y}{I_z}\frac{b_1t_1(h_3+t_1/2)}{s}$	0
7		$-\frac{M_y}{I_y}\frac{b_1}{2}$	$-\frac{M_z}{I_z}h_4$	$\frac{M_x}{I_x}t$	0	0
8		0			$\frac{F_y}{I_z}\frac{b_1t_1(h_3+t_1/2)}{2t_1}$	$\frac{F_z}{I_y}\frac{t_1b^2}{8t_1}$
9		$\frac{M_y}{I_y}\frac{b_1}{2}$			0	0

In general wide flange profiles are not suitable for large torsional moments. The reported torsional stresses are indicative only. For members with major torsional stresses a separate evaluation has to be carried out. Actual torsional stress distribution is largely dependent on surface curvature at stress point and warping resistance.

D12.A.6.3 Pipe profile

The von Mises stress is checked in three stress points as shown in figure below.

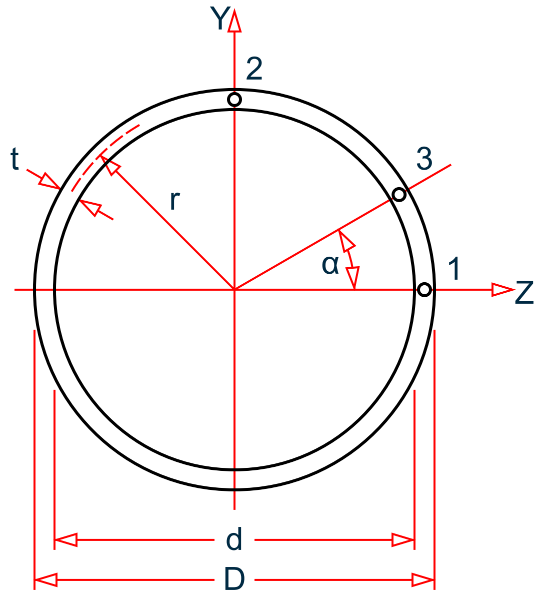


Figure 186: Stress points for a pipe section

Section properties

$d = D - 2t$
$r = 0.5 (D-t)$
$a = \tan^{-1} M_z/M_y$
$Ax = \pi/4 (D^2 - d^2)$
$A_y = A_z = 0.5A_x$
$I_x = 2I_z = \pi/32 (D^4 - d^4)$
$I_y = I_z = \pi/64 (D^4 - d^4)$

Note: In STAAD.Pro, slightly different values are used for A_y , A_z and I_x , however this has insignificant influence on the force distribution.

$A_Y = A_z = 0.6A_x$
$I_x = 2\pi R^3 t$

Stress calculation at selected stress points

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$\frac{M_y}{I_y}r$	0	$\frac{M_T}{I_x}r$	$\frac{F_y}{0.5A_x}$	0
2		0	$\frac{M_y}{I_y}r$		0	$\frac{F_y}{0.5A_x}$
3		σ_b	σ_b		τ	τ

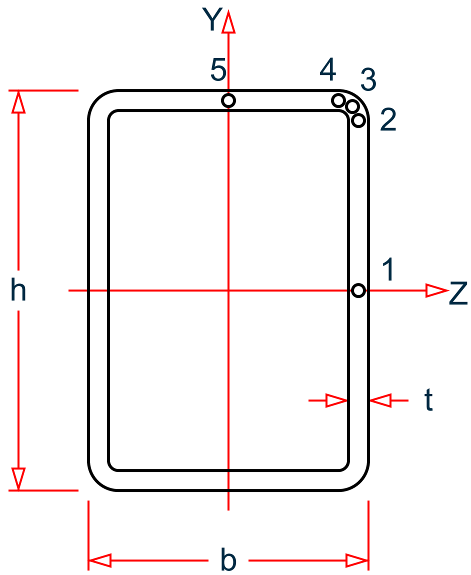
where

$$\sigma_b = \frac{\sqrt{M_y^2 + M_z^2}}{I_z} r$$

$$\tau = \frac{\sqrt{F_y^2 + F_z^2}}{0.5A_x}$$

D12.A.6.4 Tube profile

Tube sections are rectangular or quadratic hollow uniform profiles. Critical stress is checked at 5 locations as shown in figure below.



Section Properties

A_x , I_x , I_y , and I_z are taken from the STAAD.Pro database.

where

Ay = $2ht$, similar as for wide flange profiles, see Section 5.2

Az = $2(2/3)bt$; (Ay and Az are not used in code checks)

Cw = $\frac{b^2h^2t(h-b)^2}{24(h+b)}$ ref. NS app. C3

Stress calculation at selected stress points

$h_1 = h/2 - t$

$h_2 = h/2 - t/2$

$b_1 = b/2 - t$

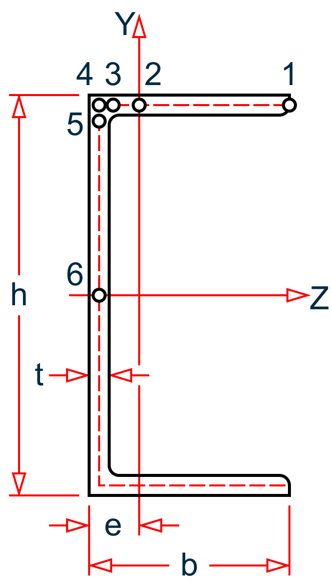
$b_2 = b/2 - t/2$

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$\frac{M_y}{I_y} b_2$	0	$\frac{M_x(h-t)(b-t)}{I_x(h+b-2t)}$	$\frac{F_y}{I_z} \frac{bth_2 + th_1^2}{2t}$	0
2		$\frac{M_y}{I_y} b_2$	$\frac{M_z}{I_z} h_1$		$\frac{F_y}{I_z} \frac{bth_2}{2t}$	$\frac{F_z}{I_y} \frac{2h_1tb_2}{2t}$
3		$\frac{M_y}{I_y} b_2$	$\frac{M_z}{I_z} h_2$		$\frac{F_y}{I_z} \frac{2b_2th_2}{2t}$	$\frac{F_z}{I_y} \frac{2h_2tb_2}{2t}$
4		$\frac{M_y}{I_y} b_1$	$\frac{M_z}{I_z} h_2$		$\frac{F_y}{I_z} \frac{2b_1th_2}{2t}$	$\frac{F_z}{I_y} \frac{htb_2}{2t}$
5		0	$\frac{M_z}{I_z} h_2$		0	$\frac{F_z}{I_y} \frac{htb_2 + tb_1^2}{2t}$

The general stress formulation is given in sec. 5.2.

D12.A.6.5 Channel profile

For channel profiles the von Mises stress is checked at 6 locations as shown in the figure below.



Cross section properties
where

A_x , S_y , S_z , I_x , I_y , and I_z	=	taken from the STAAD.Pro database
A_y	=	$2ht$, similar as for wide flange profiles
A_z	=	$2\times(2/3)bt$ (Ay and Az are not used in code checks)
e	=	$b\times(I_y/S_y)$
x	=	h-t

y	=	$b\text{-}t/2$
C_w	=	$\frac{x^2 y^3 t}{12} \frac{(2xt + 3yt)}{(xt + 6yt)}$, ref.[4] tab. 21, case 1

Stress calculations at selected stress points

$h_1 = h/2 - t$

$h_2 = h/2 - t/2$

$b_1 = e - t$

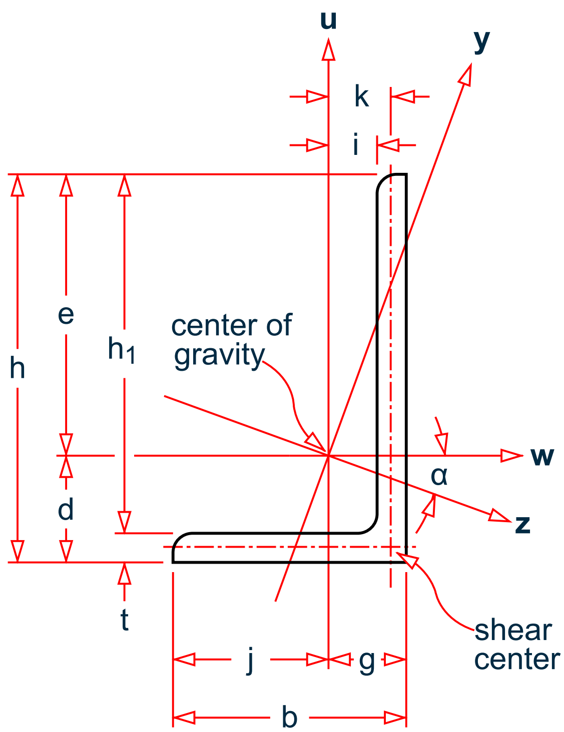
$b_2 = e - t/2$

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$\frac{M_y}{I_y}\left(b - e\right)$	$\frac{M_z}{I_z}h_2$	$\frac{M_x}{I_x}t$	0	0
2		0	$\frac{M_z}{I_z}h_2$	$\frac{M_x}{I_x}t$	$\frac{F_y}{I_z}\frac{(b - e)th_2}{t}$	$\frac{F_z}{I_y}\frac{0.5(b - e)^2 t}{t}$
3		$\frac{M_y}{I_y}b_1$	$\frac{M_z}{I_z}h_2$	$\frac{M_x}{I_x}t$	$\frac{F_y}{I_z}\frac{(b - t)th_2}{t}$	$\frac{F_z}{I_y}\frac{(b - t)t[0.5(b + t) - e]}{t}$
4		$\frac{M_y}{I_y}b_2$	$\frac{M_z}{I_z}h_2$	$\frac{M_x}{I_x}t$	$\frac{F_y}{I_z}\frac{(b - 0.5t)th_2}{t}$	$\frac{F_z}{I_y}\frac{\left(b - \frac{t}{2}\right)t\left[0.5\left(b + \frac{t}{2}\right) - e\right]}{t}$
5		$\frac{M_y}{I_y}b_2$	$\frac{M_z}{I_z}h_1$	$\frac{M_x}{I_x}t$	$\frac{F_y}{I_z}\frac{bth_2}{t}$	$\frac{F_z}{I_y}\frac{bt\left(\frac{b}{2} - e\right)}{t}$
6		$\frac{M_y}{I_y}b_2$	0	$\frac{M_x}{I_x}t$	$\frac{F_y}{I_z}\frac{bth_2 + 0.5th_1^2}{t}$	0

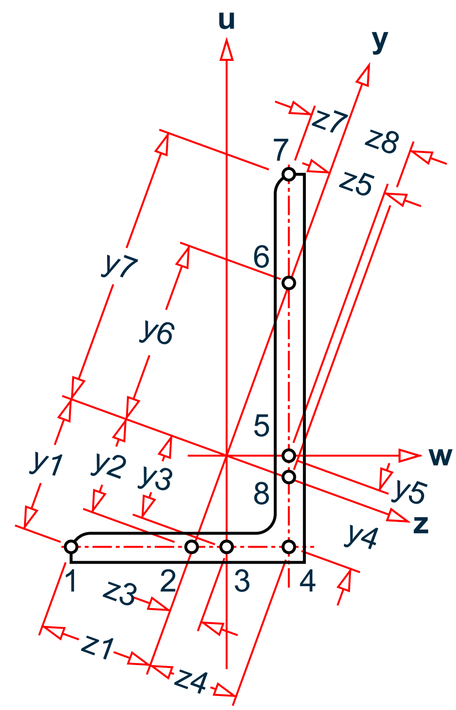
The general stress formulation is given in sec. 5.2.f

D12.A.6.6 Angle profile type RA (reverse angle)

For angle profiles the von Mises check is checked at 8 stress points as shown in figure below.



Section Dimensions



Stress Point Locations

Axes y and z are principal axes.

Axes u and w are local axes.

Cross section properties
where

A_x , I_x , I_y , and I_z	=	taken from the STAAD.Pro database
A_y	=	$\frac{2}{3}ht$, applied in STAAD.Pro print option `PRINT MEMBER STRESSES`
A_z	=	$\frac{2}{3}bt$ (Az is not used in code checks)
τ_y	=	$\frac{F_x}{A_y}$
τ_z	=	$\frac{F_x}{A_z}$
h_2	=	$0.5h_1 = t$
f	=	$h_1 e$
d	=	$\frac{th_1 h_2 + 0.5t^2 b}{A_x}$
g	=	$\frac{t^2 h_1 + tb^2}{2A_x}$

I_u	=	$\frac{h_1 t^3}{12} + htk^2 + \frac{tb^3}{12} + tb\left(\frac{b}{2} - g\right)^2$
I_w	=	$\frac{th_1^3}{12} + h_1 t(h_2 - d)^2 + \frac{bt^3}{12} + bt\left(d - \frac{t}{2}\right)^2$
I_{wu}	=	$\frac{\left(d - \frac{1}{2}\right)t}{2}\left(g^2 - j^2\right) - \frac{kt}{2}\left(e^2 - f^2\right)$
a	=	$0.5\tan^{-1}\left(\frac{2I_{uw}}{I_u + I_w}\right)$

Section forces

The section forces from the STAAD.Pro analysis are about the principle axis y and z.

The second moment of area (Ty L TZ):

$T_y = A\ Z$

$T_z = A\ Y$

Stress calculation at selected stress points

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$-\frac{M_y Z1}{I_y}$	$-\frac{M_z Y1}{I_z}$	$\frac{M_x}{I_x}t$	0	0
2		0	$-\frac{M_z Y2}{I_z}$		$\frac{F_y T_z}{I_z t}$	$\frac{F_z T_y}{I_y t}$
3		$\frac{M_y Z3}{I_y}$	$-\frac{M_z Y3}{I_z}$			
4		$\frac{M_y Z4}{I_y}$	$-\frac{M_z Y4}{I_z}$			
5		$\frac{M_y Z5}{I_y}$	$\frac{M_z Y5}{I_z}$			
6		0	$\frac{M_z Y6}{I_z}$			
7		$-\frac{M_y Z7}{I_y}$	$\frac{M_z Y7}{I_z}$			
8		$\frac{M_y Z8}{I_y}$	0		0	0

An additional torsional moment is calculated based on:

$M_T = F_y\ Z4$

$$M_T = F_z \, Y4$$

This torsion moment is included in M_x if F_y and F_Z exist.

Beta-rotation of equal & unequal legged angles

Note: The order of the joint numbers in the member incidence command specifies the direction of the local x-axis.

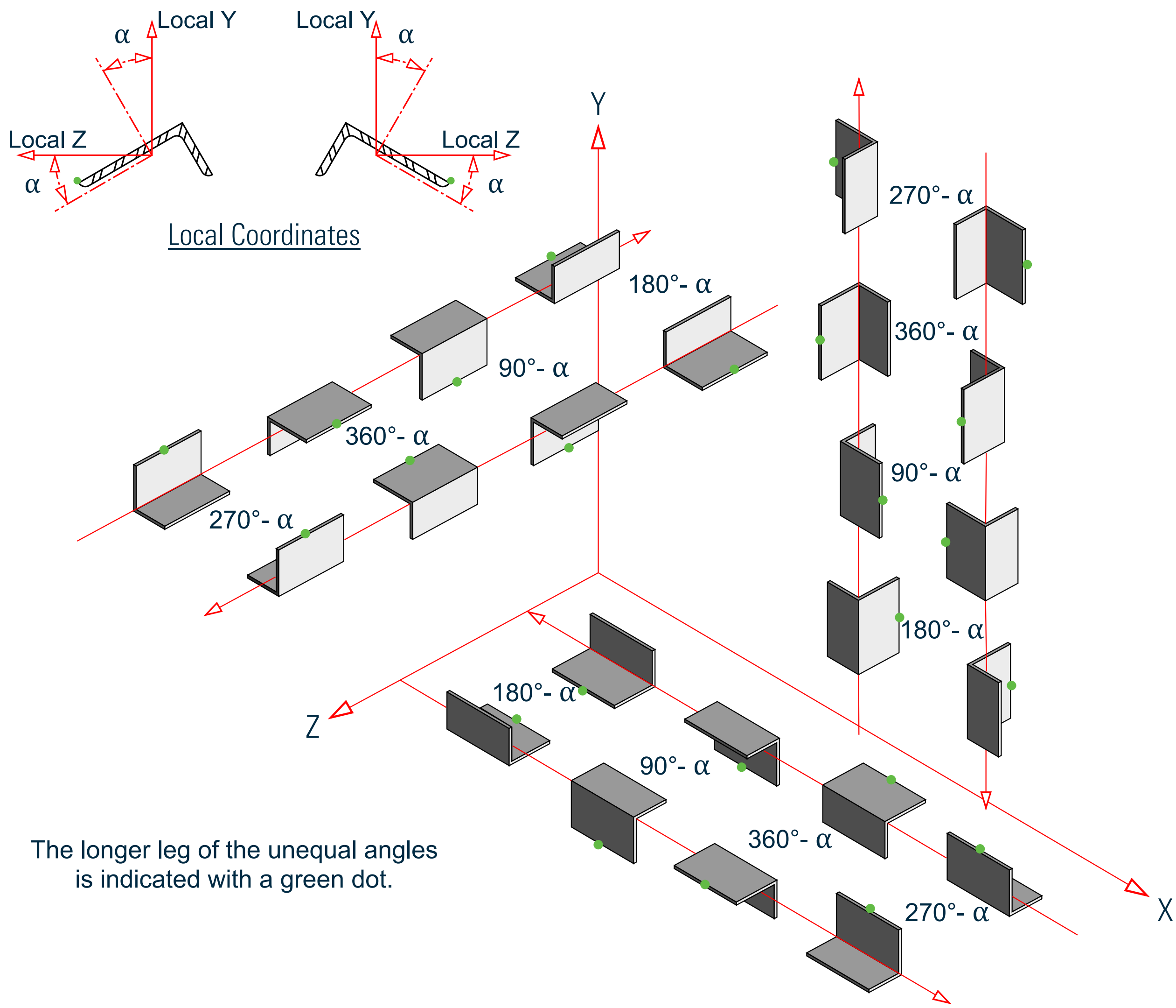
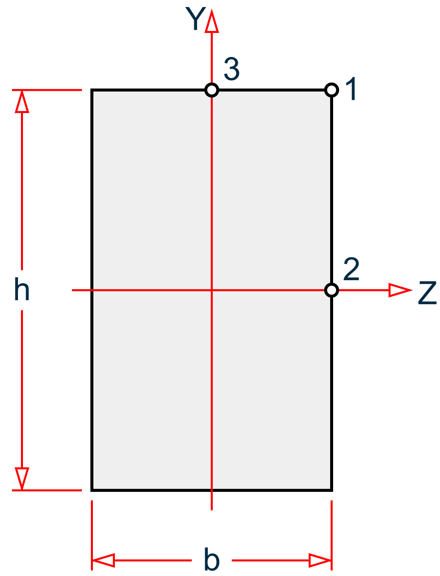


Figure 187: Beta rotation of angles

D12.A.6.7 Rectangular massive box (prismatic)

The prismatic section is assumed to be a rectangular massive box and the von Mises stress is checked at 3 locations as shown in figure below.

Note: Code check of the general purpose prismatic cross section is not available.



Note: Note that “b” may not be much greater than “h”. If that is the case, define the member with h > b and beta angle 90° instead.

Section Properties

where

A_x , A_y , A_z , I_x , I_y , and I_z , b, h	=	are user-specified. Refer to TR.20.2 Prismatic Property Specification (on page 2263).
b	=	ZD
h	=	YD
C_w	=	$\frac{1}{24}\frac{b}{2}\frac{(h-b)^2}{h+b}h^2 b^2$, ref.NS app C3.

General Stress Calculation

$$\sigma = \sigma_x + \sigma_{by} + \sigma_{bz} = \frac{F_x}{A_x} + \frac{M_y}{I_y}z + \frac{M_z}{I_z}y$$

$$\tau = \tau_x + \tau_y + \tau_z = \tau_{x,\max}\left(\frac{c}{b}\right)^2 + \frac{V_y}{A_y} + \frac{V_z}{A_z}$$

$$\tau_{x,\max} = \frac{M_x(1.5h + 0.9b)}{0.5h^2 b^2}$$

ref. [4] tab. 20, case 4 at midpoint the largest (i.e, point 2)

Stress calculation at selected stress points

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
1	$\frac{F_x}{A_x}$	$\frac{M_y}{I_y}\frac{b}{2}$	$\frac{M_z}{I_z}\frac{h}{2}$	$\tau_{x,max}\frac{b^2}{b^2+h^2}$	0	0

Point No	σ_x	σ_{by}	σ_{bz}	τ_x	τ_y	τ_z
2		$\frac{M_y}{I_y}\frac{b}{2}$	0	$\tau_{x,max}$	$\frac{F_y}{A_y}$	0
3		0	$\frac{M_z}{I_z}\frac{h}{2}$	$\tau_{x,max}\frac{b^2}{h^2}$	0	$\frac{F_z}{A_z}$

D12.A.7 Tubular Joint Check, NPD 3.5

For pipe members, punching shear capacity is checked in accordance with the NPD sections 3.5.1 to 3.5.2, except 3.5.2.4. The chord is defined as the member with the greater diameter in the joint. If the diameters are the same the program selects the member with the greater thickness of the two. The chord members must be collinear by 5 degrees.

The punching shear run sequence is performed in two steps. The program will first identify all tubular joints and classify them as T type joints (TRACK99). The joints to be checked will be listed in a file specified in the CODE NPD parameter list, below called GEOM1. This file is used as input in the second run. The file is an editable ACSII file saved under the file name given in the CODE NPD parameter. The TRACK parameter is then set to 98 which directs the program to read from the file GEOM1 file and use it as input to the second run, i.e., the joint capacity checking. The program will check the capacity for both chord members entering the joint. The local y and z moments will be transformed into the plane defined by the joint itself and the far end joints of the brace and chord, defined as in- and out-of plane moments.

The ASCII file should be edited to reflect the correct classification of the joints, gap, can or stub dimensions, yield stress and other geometric options if required. The program will not change the brace or chord definition if this is changed or modified in the input file GEOM1. See Appendix A page xx for GEOM1 example file.

Joint classification parameters in the file GEOM1 are:

KO K joint overlapped
KG K joint with gap
TY T or Y joint
X X joint

Input example for the classification run.

```
*CLASSIFICATION OF JOINTS, TRACK 99
UNITS MM NEWTON
PARAMETER
CODE NPD GEOM1
FYLD 350 ALL
TRACK 99 ALL
BEAM 1.0 ALL
CHECK CODE ALL
```

Static strength of tubular joints

The basic consideration is the chord strength. The required chord wall thickness shall be

determined when the other dimensions are given.

The following symbols are used:

T = Cord wall thickness
t = Brace wall thickness
R = Outer radius of chord
r = Outer radius of brace
Θ = Angel between chord and considered brace
D = Outer diameter of chord
d = Outer diameter of brace
a = Gap (clear distance) between considered brace and nearest load-carrying brace measured along chord outer surface
ß = r/R
g = R/T
g = a/D
f_y = Yield stress
Q_f = Factor
Q_g = See table 6.1
Q_u = See table 6.1
$Q_ß d$ = See table 6.1
N = Design axial force in brace
M_{IP} = Design in-plane bending moment in brace
M_{OP} = Design out-of plane bending moment in brace
N_k = Characteristic axial load capacity of brace (as governed by the chord strength)
M_{OPk} = Characteristic out-of-plane bending moment capacity of brace (as governed by the chord strength)
σ_{ax} = Design axial stress in chord
σ_{IP} = Design in-plane bending stress in chord
σ_{OP} = Design out-of-plane bending stress in chord

This section gives design formulae for simple tubular joints without overlap and without gussets, diaphragms or stiffeners. Tubular joints in a space frame structure shall satisfy:

$$N \le N_k / \gamma_m$$

Where:

$$N_k = Q_u Q_f \frac{f_y T^2}{\sin \Theta}$$

Q_u is given in Table 6.1 and Q_f is a factor to account for the nominal longitudinal stress in the chord.

$$Q_f = 1.0 - 0.03\gamma A^2$$

$$A^2 = \frac{\sigma_{ax}^2 + \sigma_{IP}^2 + \sigma_{OP}^2}{0.64 f_y^2}$$

Table 187: Values for Q_u

Type of joint and geometry	Type of load in brace member		
	Axial	**In-plane bending**	**Out-of-plane bending**
T and Y	2.5 + 19β	5.0√(γ)β	3.2/(1-0.81β)
X	(2.7 + 13β)Q_β		
K	0.90(2+21β)Q_β		

For $\beta > 0.6$, $Q_\beta = 0.3/[\beta(1 - 0.833\beta)]$

For $\beta \leq 0.6$, $Q_\beta = 1.0$

For $\gamma \leq 20$, $Q_g = 1.8 - 0.la/T$

For $\gamma > 20$, $Q_g = 1.8 - 4g$

but in no case shall Q_g be taken as less than 1.0.

When β ≥ 0.9, Q_f is set to 1.0. This is also applicable for moment loading. For cases with tension in the chord, Qf is set to 1.0. This is also applicable for moment loading.

The brace end moments shall be accounted for in the following cases:

- **a.** Out-of-plane bending moment when β > 0.85
- **b.** When the brace acts as a cantilever
- **c.** When the rotational stiffness of the connection is considered in the determination of effective buckling length, and / or the structural coefficient γ_{mk} = 1.00 for the beam-column design of the brace or chord. See Section 3.1.3.

The characteristic capacity of the brace subjected to in-plane bending moment shall be determined by:

$$M_{IPk} = Q_u Q_f \frac{d f_y T^2}{\sin \Theta}$$

Where Q_u is given in Table 6.1 and

$$Q_f = 1.0 - 0.045\gamma A^2$$

The characteristic capacity of the brace subjected to out-of-plane bending moment shall be determined by:

$$M_{OPk} = Q_u Q_f \frac{d f_y T^2}{\sin \Theta}$$

Where Q_u is given in Table 6.1 and

$$Q_f = 1.0 - 0.021\gamma A^2$$

For combined axial and bending loads in the brace, the following interaction equation should be satisfied:

$$\frac{N}{N_k} + \left(\frac{M_{IP}}{M_{IPk}}\right)^2 + \frac{M_{OP}}{M_{OPk}} \leq \frac{1}{\gamma_m}$$

For overlapping tubular joints without gussets, diaphragms, or stiffeners, the total load component normal to the chord, NN, shall not exceed

$$N_N = \frac{N_k}{\gamma_m}\frac{l_1}{l}\sin\Theta + \frac{2 f_y t_w l_2}{\sqrt{3}\gamma_m}$$

where (see NPD fig. 3.10)

l_l = circumference for that portion of the brace in contact with the chord (actual length)
l = circumference of brace contact with chord, neglecting presence of overlap
N_k = characteristic axial load capacity of brace
t_w = the lesser of the throat thickness of the overlapping weld or the thickness t of the thinner brace
l_2 = length as shown in NPD fig. 3.10

The above formula for the capacity of overlapping joints is valid only for K joints, where compression in a brace is essentially balanced by tension in brace(s) in the same side of the joint.

D12.A.8 Tabulated Results

This section presents a table with the various TRACKs available with respect to print out from the code check. Example prints and explanation to the information / heading given on the print out is given in Appendix A.

Table 188: Available TRACK parameter values

TRACK no.	Description
0	Brief print of member utilizations (2 lines for each member) sorted with highest utilized members first
1	Based on TRACK 3 with additional information regarding stability factors and capacities
2	Simple print of stresses, including von Mises stress
3	Brief print of member utilizations (two lines for each member)
9	Comprehensive print with detailed information about member and member utilization(one page for each member)
99	Used in connection with tubular joint check according to NPD. This TRACK identifies tubular joints to be checked and classifies all members entering the joint as T connection
98	Used in connection with tubular joint check according to NPD. This TRACK performs the joint capacity check
49	Prints member end forces for members entering each joint (at the end of the member connected to the joint)
31	Prints maximum and minimum member end forces (axial force defines max and min) at member end 1

TRACK no.	Description
32	Prints maximum and minimum member end forces (axial force defines max and min) at member end 2

D12.A.8.1 Output for member design

Output example for TRACK 0.0

Symbol	Description	Unit
MEMB	Member number	kN
FX	Axial force in the member (T = tension, C = compression)	kN·m
MYs	Start moment about the y-axis	kN·m
MYm	Mid moment about the y-axis	kN·m
MYe	End moment about the y-axis	kN·m
MYb	Buckling moment about the y-axis	kN·m
RATIO	Interaction ratio	
LOAD	The critical load case number	
TABLE	Section type (HE, IPE, TUBE, etc.)	
MZs	Start moment about z-axis	kN·m
MZm	Mid moment about the z-axis	kN·m
MZe	End moment about the z-axis	kN·m
MZb	Buckling moment about z-axis	kN·m
COND	Critical condition	
DIST	Distance from the start of the member to the critical section	m

Note: Myb and Mzb are the design moments used for max unity ratio.

```
                           NS3472 (VERSION 06002)
                           UNITS ARE  KN   AND   METE

MEMB           FX              MYs       MYm       MYe       MYb   RATIO  LOAD
          TABLE                MZs       MZm       MZe       MZb   COND   DIST
===============================================================================
      1       12.80 C         0.0       0.0       0.0       0.0  5.08       1
      FAIL  PIPS40                 (AISC SECTIONS)
                             31.9     -15.9     -36.2      36.2 STAB    10.00
```

```
     4       24.20 C         0.0       0.0       0.0       0.0  1.30       1
     FAIL  PIPS40                    (AISC SECTIONS)
                            -0.2      -1.4      -2.9       2.9 STAB    14.14
     3       26.31 C         0.0       0.0       0.0       0.0  0.78       1
           PIPS40                    (AISC SECTIONS)
                             5.1       1.3       2.5       5.1 STAB     0.00
     2        4.02 C         0.0       0.0       0.0       0.0  0.58       1
           PIPD60                    (AISC SECTIONS)
                            36.4     -38.1      -6.8      38.9 STAB     5.83
     5        5.02 T         0.0       0.0       0.0       0.0  0.34       1
           PIPS40                    (AISC SECTIONS)
                             3.6      -1.8       1.7       2.8 VMIS     0.00
```

Output example for TRACK 1.0

Symbol	Description	Unit
CURVE St	Buckling curve about the strong axis	
CURVE Wk	Buckling curve about the weak axis	
Beta Z	Buckling length factor about z-axis	
Beta Y	Buckling length factor about y-axis	
FYLD	Allowable yield strength	N/mm^2
Betamz	Equivalent moment factor β_m about z-axis	
Betamy	Equivalent moment factor β_m about y-axis	
Fak Z	Factor k according to 12.3.4.2 about the z-axis	
Fak Y	Factor k according to 12.3.4.2 about the y-axis	
MYD	Moment capacity about the y axis	kN·m
MZD	Moment capacity about the z axis	kN·m
MVD	Lateral buckling moment	kN·m
IR1	Interaction ratio for buckling without lateral buckling (Cl. 12.3.4.2)	
IR2	Interaction ratio for buckling with lateral buckling (Cl. 12.3.4.2)	

Symbol	Description	Unit
VON MISES	Interaction ratio for von Mises	

```
                           NS3472 (VERSION 06002)
                           UNITS ARE  KN   AND   METE

 MEMB         FX              MYs       MYm       MYe       MYb   RATIO  LOAD
           TABLE              MZs       MZm       MZe       MZb   COND   DIST
===============================================================================
      1      12.80 C          0.0       0.0       0.0       0.0  5.08       1
      FAIL  PIPS40                   (AISC SECTIONS)
                             31.9     -15.9     -36.2      36.2 STAB    10.00
 |-----------------------------------------------------------------------------|
 | CURVE St A  Wk A  Beta Z  1.00      Beta Y  1.00      FYLD= 235. N/MM2  |
 | Betamz=1.295      Betamy=1.000      FakZ=1.500        FakY=1.500        |
 | MYD =.112E+2 KNM  MZD =.112E+2 KNM  MVD =.112E+2 KNM                    |
 | IR1       = 5.076 IR2      = 5.076  VON MISES = 3.251                   |
 |-----------------------------------------------------------------------------|
      2       4.02 C          0.0       0.0       0.0       0.0  0.58       1
            PIPD60                   (AISC SECTIONS)
                             36.4     -38.1      -6.8      38.9 STAB     5.83
 |-----------------------------------------------------------------------------|
 | CURVE St A  Wk A  Beta Z  1.00      Beta Y  1.00      FYLD= 235. N/MM2  |
 | Betamz=1.377      Betamy=1.000      FakZ=1.021        FakY=1.033        |
 | MYD =.701E+2 KNM  MZD =.701E+2 KNM  MVD =.701E+2 KNM                    |
 | IR1       = 0.575 IR2      = 0.575  VON MISES = 0.557                   |
 |-----------------------------------------------------------------------------|
      3      26.31 C          0.0       0.0       0.0       0.0  0.78       1
            PIPS40                   (AISC SECTIONS)
                              5.1       1.3       2.5       5.1 STAB     0.00
 |-----------------------------------------------------------------------------|
 | CURVE St A  Wk A  Beta Z  1.00      Beta Y  1.00      FYLD= 235. N/MM2  |
 | Betamz=2.152      Betamy=1.000      FakZ=0.602        FakY=1.500        |
 | MYD =.112E+2 KNM  MZD =.112E+2 KNM  MVD =.112E+2 KNM                    |
 | IR1       = 0.784 IR2      = 0.784  VON MISES = 0.510                   |
 |-----------------------------------------------------------------------------|
      4      24.20 C          0.0       0.0       0.0       0.0  1.30       1
      FAIL  PIPS40                   (AISC SECTIONS)
                             -0.2      -1.4      -2.9       2.9 STAB    14.14
 |-----------------------------------------------------------------------------|
 | CURVE St A  Wk A  Beta Z  1.00      Beta Y  1.00      FYLD= 235. N/MM2  |
 | Betamz=1.510      Betamy=1.000      FakZ=1.500        FakY=1.500        |
 | MYD =.112E+2 KNM  MZD =.112E+2 KNM  MVD =.112E+2 KNM                    |
 | IR1       = 1.304 IR2      = 1.304  VON MISES = 0.310                   |
 |-----------------------------------------------------------------------------|
```

Output example for TRACK 2.0

```
                              NS3472 (VERSION 06001)
                              UNITS ARE  mm AND    N

 MEMB        Sx              Sby       Sbz       Stot      Spmx    Svm      LOAD
           TABLE             Ty        Tz        Tto       Spmn    POINT    DIST
===========================================================================
   111       10.87 T        242.0        .0     252.9        .0  257.90    11
           PIP 600X15        28.9        .0        .3        .0    3     3.46
   112       13.70 T         66.0        .0      79.7        .0   82.70    13
           PIP 600X15        12.4        .0        .3        .0    3     2.83
   113         .42 T         78.7        .0      79.1        .0   80.18    11
           PIP 600X15         7.4        .0        .2        .0    3     3.46
   114       13.28 C        118.6        .0     131.9        .0  135.52    11
           PIP 600X15        17.7        .0        .3        .0    3     2.83
   115       68.71 C         18.8        .0      87.5        .0   87.66    11
           PIP 200X8          1.0        .0       2.3        .0    3      .00
   116       66.13 T         37.9        .0     104.0        .0  105.91    11
           PIP 200X8          1.1        .0      10.3        .0    3     5.13
   117       63.29 T         47.5        .0     110.8        .0  110.84    11
           PIP 200X8          1.7        .0        .4        .0    3      .00
   118       85.64 C         39.8        .0     125.4        .0  125.43    11
           PIP 200X8          1.2        .0        .2        .0    3     6.20
   119       90.54 C         21.4        .0     111.9        .0  111.96    14
           PIP 200X8          1.0        .0        .6        .0    3      .00
   120       94.89 T         24.7        .0     119.6        .0  119.61    14
           PIP 200X8          1.2        .0        .3        .0    3     5.23
   121       79.98 C         43.7        .0     123.6        .0  123.68    11
           PIP 200X8          1.3        .0        .2        .0    3     6.20
```

Symbol	Description	Unit
MEMB	Member number	
Sx	Axial stress in the member (T = tension, C = compression)	N/mm^2
Sby	Stress from moment about y-axis	N/mm^2
Sbz	Stress from moment about z-axis	N/mm^2
Stot	Sum of Sx + Sby + Sbz	N/mm^2
Spmx	Not used	
Spmn	Not used	
Svm	von Mises stress	N/mm^2
Ty	Stress from shear force in the y direction	N/mm^2
Tz	Stress from shear force in the z direction	N/mm^2

Symbol	Description	Unit
Tto	Total shear stress used in von Mises calculation	N/mm^2
TABLE	Section type (HE, IPE, TUBE, etc.)	
POINT	Location in cross section with maximum von Mises stress	
LOAD	Governing load condition	
DIST	Distance from the start of the member to the critical section	m

Note: Do not use `TRACK 2.0` in connection with `SELECT OPTIMIZED` or `SELECT MEMBER / ALL` commands.

Output example for TRACK 3

```
                              NS3472 (VERSION 06001)
                              UNITS ARE  KNS  AND   METE

 MEMB         FX                 MYs       MYm        MYe        MYb   RATIO  LOAD
              TABLE              MZs       MZm        MZe        MZb   COND   DIST
==================================================================================
   111       299.78 T          -13.2     -17.5       21.8       18.4   .86      11
              PIP  600X15     -426.9     262.5     -951.8      400.3    VMIS  3.46
   112       377.68 T          -13.8     -11.8        9.8       12.2   .28      13
              PIP  600X15      224.1     -17.7      259.5      103.8    VMIS  2.83
   113        11.61 T          -18.4     -16.4       14.3       16.8   .27      11
              PIP  600X15      -43.3     132.9     -309.1      168.2    LATB  3.46
   114       366.02 C          -13.2     -11.3        9.3       11.6   .45      11
              PIP  600X15     -224.8     120.9     -466.5      190.0    VMIS  2.83
   115       331.57 C           -4.1        .3       -2.0        1.7   .33      11
              PIP  200X8         -.5       1.0       -2.5        1.3    STAB   .00
   116       319.14 T           -1.9       -.6       -3.3        1.4   .35      11
              PIP  200X8         -.4       3.7       -7.8        4.5    VMIS  5.13
   117       305.44 T             .0       -.3         .5         .3   .37      11
              PIP  200X8        10.6       -.8        9.9        5.3    VMIS   .00
   118       413.31 C             .1       -.3         .6         .3   .50      11
              PIP  200X8         4.4      -1.1        8.8        4.6    STAB  6.20
   119       436.94 C           -4.5        .2       -2.1        1.9   .41      14
              PIP  200X8        -1.6        .0       -1.7         .7    STAB   .00
   120       457.92 T           -3.7       -.7       -5.1        2.0   .40      14
              PIP  200X8         1.7       -.2        2.1         .8    VMIS  5.23
   121       385.99 C            -.2       -.1         .0         .1   .48      11
              PIP  200X8         6.0       -.8        9.7        5.0    STAB  6.20
```

Symbol	Description	Unit
MEMB	Member number	

Symbol	Description	Unit
FX	Axial force in the member (T = tension, C = compression)	kN
MYs	Start moment about the y-axis	kN·m
MYm	Mid moment about the y-axis	kN·m
MYe	End moment about the y-axis	kN·m
MYb	Buckling moment about the y-axis	kN·m
RATIO	Interaction ratio	
LOAD	The criticl load case number	
TABLE	Section type (HE, IPE, TUBE, etc.)	
MZs	Start moment about the z-axis	kN·m
MZm	Mid moment about the z-axis	kN·m
MZe	End moment about the z-axis	kN·m
MZb	Buckling moment about the z-axis	kN·m
COND	Critical condition	
DIST	Distance from the start of the member to the critical section	m

Output example for TRACK 9.0

Member in tension:

```
DETAILS FOR CODECHECK ACCORDING TO NS3472
                                (VERSION 06001)
          MEMBER NO            :    111
          MEMBER TYPE          : PIPE     SECTION PIP 600X15
          GOVERING LOADCASE    :   11

               MEMBER PROPERTY            UNITS CM
               ---------------
               Ax    :     275.7     iy    :      20.7
               Ay    :     137.8     iz    :      20.7
               Az    :     137.8     Sy    :    3933.5
               Ix    :  236010.1     Sz    :    3933.5
               Iy    :  118005.0     Iw    :        .0
               Iz    :  118005.0     Lw    :     345.9

               MATERIAL DATA              UNITS NEWTON MMS
               ---------------
               E     :      204960.    Gamma :     1.150
               Fy    :   344.966       Fd    :   299.970
               FORCES                     UNITS KNEWTON METERS
               ---------------
               Fx     :    299.78 T
               Msz    :   -426.86         Msy   :   -13.176
               Mmz    :    262.48         Mmy   :   -17.508
               Mez    :   -951.82         Mey   :    21.840
               LATERAL BUCKLING
               --------
               Mlatbuck:   670.43
               Mvd      :  1179.90
               IRtot    :      .786
               YIELD CHECK
               -----------
               STRESS : NEW MMS           FORCES: KNEW METERS
               STRESS AT POINT :   3      FORCES AT SECTION   3.459
               sigax  :    10.873         Fx    :   299.778 T
               sigb   :   242.016         Fy    :   398.539
               tau    :    28.914         Fz    :     2.505
               tors   :      .305         Mx    :     2.398
               sige   :   257.904         My    :    21.838
               IR     :      .860         Mz    :  -951.695

               Governing interaction ratio              .860
```

Member in compression:

```
DETAILS FOR CHECKING ACCORDING TO NS3472
                       (VERSION 06001)
MEMBER NO           :     50
MEMBER TYPE         : TUBE
SECTION : TUB16016010              (EUROPEAN SECTIONS)
GOVERNING LOADCASE  :    5

     MEMBER PROPERTY           UNITS CM
     ---------------
     Ax    :        57.4     iy    :         6.0
     Ay    :        32.0     iz    :         6.0
     Az    :        32.0     Sy    :       262.5
     Ix    :      3470.0     Sz    :       262.5
     Iy    :      2100.0     Iw    :         0.0
     Iz    :      2100.0     Lw    :       600.0

     MATERIAL DATA             UNITS NEWTON MMS
     ---------------
     E     :      205000.     Gamma    :     1.100
     Fy    :   355.000        Fd       :   322.727
     lamfy :    75.494        Gamma mk:     1.000
     BUCKLING PARAMETERS       UNITS KNEWTON METERS
     -------------------
     STRONG AXIS              WEAK AXIS              LATERAL BUCKLING
     L     :         6.000    L     :         6.000  L     :     6.000
     beta  :         1.000    beta  :         1.000  ny    :     1.000
     lambda:        99.197    lambda:        99.197  alfaLT:     0.490
     lambb :         1.314    lambb :         1.314  Mvd   :    84.716
     curve :      A           curve :      A
     ksi   :         0.463    ksi   :         0.463
     n/ksi :         0.032    n/ksi :         0.032
     betaM :         2.292    betaM :         2.305
     k     :         0.978    k     :         0.977
     m     :         0.014    m     :         0.066

     FORCES                    UNITS KNEWTON METERS
     ---------------
     STRONG AXIS               WEAK AXIS/BUCKLING
     Fx     :        27.013 C   Fx     :        27.013 C
     Ms     :         0.814     Ms     :         4.064
     Mm     :        -0.173     Mm     :        -0.784
     Me     :         1.159     Me     :         5.633
     psi    :        -0.702     psi    :        -0.721
     Mmax   :         1.159     Mmax   :         5.633
     IRx    :         0.032     IRx    :         0.032
     IRm    :         0.078     IRm    :         0.079
     IRtot  :         0.110     IRtot  :         0.110

     YIELD CHECK
     -----------
     STRESS : NEW MMS          FORCES: KNEW METERS
     STRESS AT POINT :   3     FORCES AT SECTION         6.000
     sigax  :         4.706        Fx    :        27.013 C
     sigb   :        24.257        Fy    :        -0.329
     tau    :         0.521        Fz    :         1.616
     tors   :         0.203        Mx    :         0.094
     sige   :        28.990        My    :         5.633
     IR     :         0.090        Mz    :         1.159

     Governing interaction ratio             0.110
```

Member in compression (pipe - NPD):

```
DETAILS FOR CODECHECK ACCORDING TO NPD94
                              (VERSION 96016.00)
       MEMBER NO            :     1
       MEMBER TYPE          : PIPE      762x  19 mm
       GOVERING LOADCASE    :    2

       UNITS [properties: cm][stesses:new mms][forces:kn me]
       -- PROPERTIES --
       D/t    :      40.0     Iy Iz :  306983.8     Ly    :    3074.8
       Ax     :     444.6     Sy Sz :    8057.3     Lz    :    3074.8
       Ay Az :      222.3     iy iz :      26.3     By    :        .7
       Ix     :  613967.7     Z     :  124263.4     Bz    :        .6

       -- MATERIAL --
       E      :   209979.     Fy    :      366.     Gamma m :     1.150
       lamfy :        91.     Fd    :      318.     Gamma mk:     1.000

       -- SHELL BUCKLING --
       -- Section npd 3.4.6.1 --
       fea :     2984.5 feb :    3076.4 fet :      528.2 fep :     131.2
       -- Section npd 3.4.4.1 --
       La   :       .350 Lb   :       .345 Lt   :       .632 Lp   :     1.670
       Ga   :     1.047 Gb   :     1.045 Gt   :     1.135 Gp   :     1.250
       Fk   :   365.952
       -- Section npd 3.4.7 --
       Sigj:      278.7 SECT:        1.0 Irshell       .876
       -- Section npd 3.4.9.2 --
       Bendingmoment stress in 3.4.4.1 increased by dsigb :    91.430
       d/t    40.0 >    12.0 interaction npd 3.2.3 a
       -- BEAM COLUMN BUCKLING --
       -- Section npd 3.4.4.1 --
       La   :       .350 Lb   :       .345 Lt   :       .632 Lp   :     1.670
       Ga   :     1.047 Gb   :     1.045 Gt   :     1.135 Gp   :     1.250
       Fa   :     249.6 Sect:        1.0
       -- Section ns3472 5.4.1 --
       lbz :    67.872 lbbz:       .745 crvz:  A    FkFy z:       .826
       lby :    78.404 lbby:       .860 crvy:  A    FkFy y:       .759
       -- Section npd 3.2.2 --
       SIGa   :     162.6     SIGby :       8.2     SIGbz :      32.2
       fE     :   337.131
       B      :     1.932     By    :     1.932     Bz    :     1.566
       SIGb* :       26.3     SIGbuc:     266.4     Irb   :     1.227
       -- Section npd 3.2.2.1 (ns3472 5.4.2) --
          z axis                          y axis
       Fx      :  7231.114 C       Fx    :  7231.114 C
       Ms      :   -87.503         Ms    :    28.677
       Mm      :  -154.416         Mm    :    36.766
       Me      :  -297.803         Me    :    87.503
       beta    :     -.294         beta  :     -.328
       m       :      .482         m     :      .469
       Mb      :   259.566         Mb    :    66.179
       -- Section npd 3.1.2 --
       Stress at point :    3      Forces at section  30.748
       sigax  :   162.614          Fx    :  7231.114 C
       sigb   :    38.519          Fy    :    46.293
       tau    :     2.156          Fz    :    12.386
       tors   :      .000          Mx    :      .000
       sigp   :    30.166          My    :    87.491
       sige   :   187.912          Mz    :  -297.764
       Irj fy :      .590          HSpres:     1.508
```



```
            Governing interaction ratio              1.227
```

D12.A.8.2 Output for Joint Capacity Code Checking

Output example for TRACK 99.0

$JOINT	BRACE	CHORD	D	T	d	t	GAP	FYc	FYb	THETA	TW	THETAT	JTYPE
1020	1016	1015	420.	15.	400.	15.	0.	340.	340.	90.	0.	0.	TY
1020	1016	1017	420.	15.	400.	15.	0.	340.	340.	90.	0.	0.	TY
2010	1016	1010	500.	20.	400.	15.	0.	340.	340.	45.	0.	0.	TY
2010	1016	2010	500.	20.	400.	15.	0.	340.	340.	45.	0.	0.	TY
2000	1017	1000	500.	20.	420.	15.	0.	340.	340.	45.	0.	0.	TY
2000	1017	2000	500.	20.	420.	15.	0.	340.	340.	45.	0.	0.	TY
1020	1018	1015	420.	15.	400.	15.	0.	340.	340.	90.	0.	0.	TY
1020	1018	1017	420.	15.	400.	15.	0.	340.	340.	90.	0.	0.	TY
2000	2015	1000	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
2000	2015	2000	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3010	2015	2010	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3010	2015	3010	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3010	3015	2010	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3010	3015	3010	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
2000	2005	1000	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
2000	2005	2000	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
2010	2005	1010	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
2010	2005	2010	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
3000	3005	2000	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
3000	3005	3000	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
3010	3005	2010	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
3010	3005	3010	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
2210	1215	1210	500.	20.	400.	15.	0.	340.	340.	45.	0.	0.	TY
2210	1215	2210	500.	20.	400.	15.	0.	340.	340.	45.	0.	0.	TY
2210	2215	1210	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
2210	2215	2210	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3200	2215	2200	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3200	2215	3200	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3200	3215	2200	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
3200	3215	3200	500.	20.	400.	15.	0.	340.	340.	40.	0.	0.	TY
2200	2205	1200	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY
2200	2205	2200	500.	20.	400.	10.	0.	340.	340.	90.	0.	0.	TY

Output example for TRACK 98.0

```
                 NPD 94 TUBULAR JOINT CHECK (VERSION 96016.00)
                             UNITS ARE  mm

  JOINT    CHORD     Dc        Tc      BRACE     db       tb     TYPE  LOAD   RATIO
=================================================================================
   1020     1015   420.0     15.0      1016    400.0     15.0     TY     3     .143
   1020     1017   420.0     15.0      1016    400.0     15.0     TY     3     .143
   2010     1010   500.0     20.0      1016    400.0     15.0     TY     3     .049
   2010     2010   500.0     20.0      1016    400.0     15.0     TY     3     .048
   2000     1000   500.0     20.0      1017    420.0     15.0     TY     3     .548
   2000     2000   500.0     20.0      1017    420.0     15.0     TY     3     .586
   1020     1015   420.0     15.0      1018    400.0     15.0     TY     3     .099
   1020     1017   420.0     15.0      1018    400.0     15.0     TY     3     .099
   2000     1000   500.0     20.0      2015    400.0     15.0     TY     3     .450
   2000     2000   500.0     20.0      2015    400.0     15.0     TY     3     .481
   3010     2010   500.0     20.0      2015    400.0     15.0     TY     3     .527
   3010     3010   500.0     20.0      2015    400.0     15.0     TY     3     .498
   3010     2010   500.0     20.0      3015    400.0     15.0     TY     3     .338
   3010     3010   500.0     20.0      3015    400.0     15.0     TY     3     .320
   2000     1000   500.0     20.0      2005    400.0     10.0     TY     3     .107
   2000     2000   500.0     20.0      2005    400.0     10.0     TY     3     .113
   2010     1010   500.0     20.0      2005    400.0     10.0     TY     3     .177
   2010     2010   500.0     20.0      2005    400.0     10.0     TY     3     .170
   3000     2000   500.0     20.0      3005    400.0     10.0     TY     3     .168
   3000     3000   500.0     20.0      3005    400.0     10.0     TY     3     .167
   3010     2010   500.0     20.0      3005    400.0     10.0     TY     3     .183
   3010     3010   500.0     20.0      3005    400.0     10.0     TY     3     .174
   2210     1210   500.0     20.0      1215    400.0     15.0     TY     3     .945
   2210     2210   500.0     20.0      1215    400.0     15.0     TY     3     .518
   2210     1210   500.0     20.0      2215    400.0     15.0     TY     3    1.146
   2210     2210   500.0     20.0      2215    400.0     15.0     TY     3     .617
   3200     2200   500.0     20.0      2215    400.0     15.0     TY     3     .575
   3200     3200   500.0     20.0      2215    400.0     15.0     TY     3     .579
   3200     2200   500.0     20.0      3215    400.0     15.0     TY     3     .232
   3200     3200   500.0     20.0      3215    400.0     15.0     TY     3     .234
   2200     1200   500.0     20.0      2205    400.0     10.0     TY     3     .183
   2200     2200   500.0     20.0      2205    400.0     10.0     TY     3     .177
   2210     1210   500.0     20.0      2205    400.0     10.0     TY     3    1.402
   2210     2210   500.0     20.0      2205    400.0     10.0     TY     3     .262
   3200     2200   500.0     20.0      3205    400.0     10.0     TY     3     .210
   3200     3200   500.0     20.0      3205    400.0     10.0     TY     3     .212
   3210     2210   500.0     20.0      3205    400.0     10.0     TY     3     .223
   3210     3210   500.0     20.0      3205    400.0     10.0     TY     3     .223
   2000     1000   500.0     20.0      1315    400.0     15.0     TY     3     .522
   2000     2000   500.0     20.0      1315    400.0     15.0     TY     3     .558
   2000     1000   500.0     20.0      2315    400.0     15.0     TY     3     .463
   2000     2000   500.0     20.0      2315    400.0     15.0     TY     3     .495
   3200     2200   500.0     20.0      2315    400.0     15.0     TY     3     .519
   3200     3200   500.0     20.0      2315    400.0     15.0     TY     3     .523
   3200     2200   500.0     20.0      3315    400.0     15.0     TY     3     .309
   3200     3200   500.0     20.0      3315    400.0     15.0     TY     3     .311
   2200     1200   500.0     20.0      2305    400.0     10.0     TY     3     .259
   2200     2200   500.0     20.0      2305    400.0     10.0     TY     3     .238
   2000     1000   500.0     20.0      2305    400.0     10.0     TY     3     .154
   2000     2000   500.0     20.0      2305    400.0     10.0     TY     3     .171
   3200     2200   500.0     20.0      3305    400.0     10.0     TY     3     .163
   3200     3200   500.0     20.0      3305    400.0     10.0     TY     3     .164
   3000     2000   500.0     20.0      3305    400.0     10.0     TY     3     .159
   3000     3000   500.0     20.0      3305    400.0     10.0     TY     3     .159
```

D12.A.8.3 Special prints (not code check)

Output example for TRACK 49

```
                              NS3472 JOINT OUTPUT (VERSION 95015.01.)
                              UNITS ARE  KNS  AND   METE

 JOINT  LOAD  MEMBER     FX         FY         FZ         MZ         MY         MZ
=============================================================================================
     1   12      1      46.8C      142.0      156.0       72.3       38.8      -31.7
                 8       9.5C       51.6      -13.0        4.4      -28.2     -109.4
               107     193.5T       33.8     -146.5       10.6     -181.7       27.3
     2   12      1      46.8C     -140.7     -156.0      -72.3       88.8      -83.9
                 2     110.8C       22.7        -.1        -.2      -11.8       82.0
                 9     156.1T      118.0      -64.0        1.9      -76.9      -72.5
     3   12      2     110.8C       22.2         .1         .2       11.4      -82.8
                 3      47.3C     -140.1     -158.7      -68.3      -86.7       84.2
                10     158.6T      117.8       63.5       -1.4       75.3      -68.0
     4   12      3      47.3C      141.3      158.7       68.3      -43.1       30.9
                 4      19.1C       51.6       13.0       -4.4       26.9     -107.4
               108     192.9T      -34.3     -139.6      -16.2     -175.6      -26.5
     5   12      4      19.1C      -44.8      -13.0        4.4       30.3     -104.8
                 5       1.2T       47.4      119.8      -69.3       43.3      -16.8
               109      92.2C      -11.8     -138.9      -13.0     -174.1       12.4
     6   12      5       1.2T      -48.6     -119.8       69.3       54.7      -22.4
                 6      64.6T      -22.6      278.3      278.5      149.3       21.1
                10     158.6T      -71.2      -63.5        1.4      204.0     -347.8
     7   12      6      64.6T      -22.3      279.4      279.2     -151.5      -20.5
                 7        .7T      -49.1     -123.3       65.1      -53.0       22.4
                 9     156.1T      -71.4       64.0       -1.9     -204.5     -344.3
     8   12      7        .7T       47.8      123.3      -65.1      -47.8       17.2
                 8       9.5C      -44.8       13.0       -4.4      -29.2     -102.8
               110      92.6C       12.4     -132.8       18.7     -167.8      -12.8
     9   12     11     218.4T       81.6       27.7       75.1        2.3      -94.9
                18     101.5T       48.6       -3.1       -7.8       -8.7     -101.8
               107     200.7T      -33.8      146.5      -10.6     -201.5     -115.7
               111     549.0T     -169.9        3.6        3.4       20.5      221.7
               115     333.3C         .3        2.2        -.2        3.9       -1.0
               117      40.2C        1.1         .0         .1         .1       -2.5
```

Output example for TRACK 31

MAX MIN OUTPUT FOR END NO: 1
UNITS ARE KNS AND METE

MEMBER	LOAD	FX	FY	FZ	MZ	MY	MZ
1	11	67.6C	153.8	155.2	71.9	38.2	-61.3
	16	-116.3T	60.0	-3.7	.1	1.2	-181.9
	11	67.6C	153.8	155.2	71.9	38.2	-61.3
	10	.9C	.1	.0	.0	.0	-.1
	13	48.0C	142.0	156.0	72.3	38.8	-31.6
	16	-116.3T	60.0	-3.7	.1	1.2	-181.9
	13	48.0C	142.0	156.0	72.3	38.8	-31.6
	1	3.0C	36.3	.3	-.2	-.1	9.0
	14	1.1C	96.6	77.1	41.7	39.2	-58.8
	3	-12.4T	29.1	45.2	24.7	-23.1	30.2
	4	-81.9T	7.5	-2.6	.1	-.6	103.1
	16	-116.3T	60.0	-3.7	.1	1.2	-181.9

Output example for TRACK 32

MAX MIN OUTPUT FOR END NO: 2
UNITS ARE KNS AND METE

MEMBER	LOAD	FX	FY	FZ	MZ	MY	MZ
1	11	67.6C	-152.5	-155.2	-71.9	88.7	-64.0
	16	-116.3T	-58.7	3.7	-.1	-4.2	133.4
	10	.9C	-.1	.0	.0	.0	.1
	11	67.6C	-152.5	-155.2	-71.9	88.7	-64.0
	16	-116.3T	-58.7	3.7	-.1	-4.2	133.4
	13	48.0C	-140.8	-156.0	-72.3	88.8	-84.0
	1	3.0C	-35.4	-.3	.2	-.2	20.3
	13	48.0C	-140.8	-156.0	-72.3	88.8	-84.0
	13	48.0C	-140.8	-156.0	-72.3	88.8	-84.0
	2	14.7C	-56.0	-91.8	-42.8	-52.3	31.6
	16	-116.3T	-58.7	3.7	-.1	-4.2	133.4
	4	-81.9T	-7.5	2.6	-.1	2.7	-97.0

D12.B. Norwegian Codes - Steel Design per NORSOK N-004

STAAD.Pro is capable of performing steel design based on the Norwegian code NORSOK N-004 Rev 2, October 2004. Code checks for tubular (pipe) members is performed per the code.

Please note the following:

- The code check is available for the pipe cross sections only.
- The design of conical transitions and joints with joint cans is not performed.

Design of members per NTC 1987 requires the *STAAD ECC Super Code* SELECT Code Pack.

D12.B.1 Member Resistances

The implementation of the NORSOK N-004 code in STAAD.Pro considers sections 4, 5, 6 & 7 in of that document. The details of the various clauses implemented from these sections is presented here for member checking and design.

D12.B.1.1 General Provisions

The general safety check is per Section 4. Checks are made to ensure that the design action effect (S_d) is less than or equal to the design resistance (R_d):

$$S_d \leq R_d$$

The design resistance is evaluated for each condition and this check is applied as described in the following sections.

D12.B.1.2 Steel selection and non destructive testing

Section 5 deals with the choice of “design class” for structural joints and components. The choice of design class will determine the choice of steel grade & quality and also the determination of inspection category for fatigue. The choice of design class (as per Table 5-1 of the code) is left to you and does not have any direct impact on how STAAD.Pro performs design checks.

D12.B.1.3 Ultimate Limit States

Clause 6.1 primarily deals with the section of material factors to be used in the various conditions or checks. The material factors chosen are dependent on the “section class” of a cross section. N-004 does not explicitly specify how to classify various cross sections. Therefore, the section classification is made as given in Section 5.5 of EN 1993-1-1:2005, except when specified explicitly along with member checks (See Member Subject to Axial Compression).

Also, N-004 does not specify steel grades to be used. Therefore, this STAAD.Pro uses the steel grades per EN 1993-1-1:2005 for designs per N-004.

Note: Ring stiffener design to CL. 6.3.6.2 is not included for this implementation.

D12.B.1.4 Tubular Members

Clause 6.3.1 deals with the general considerations while using tubular members.

Caution: Only tubular sections can be used with the N-004 code in STAAD.Pro. A warning is presented for any other section type.

The dimensions of the tubular sections are limited as follows:

- The thickness t ≥ 6 mm.
- The thickness t <150 mm.
- The slenderness ratio of the cross section D/t < 120.

  Where D is the diameter and t is the wall thickness of the section.
- The yield strength for tubular member ≤ 500 N/mm^2.

If any of these conditions are not met for a member selected for design, a warning will be issued by the engine and the design of that member is aborted.

> **Note:** N-004 uses “Y” to define the action effects that is in plane and ‘Z’ to define out of plane effects. This is the opposite to what STAAD uses, where ‘Z’ defines the in plane effects and ‘Y’ the out of plane effects. This document will follow the STAAD.Pro convention for the Z and Y axes.

The N-004 code also segregates members into those that are subject to hydrostatic pressure and those that are not subject to hydrostatic pressure. The program allows you to specify whether a member is subject to hydrostatic pressure or not and, if so, to specify the hydrostatic pressure for the element. By default the program will assume that all members are *not* subject to any hydrostatic pressure. The design parameter HYD is used to specify the maximum water level with respect to the origin.

If the HYD parameter is specified, the program will take that to be the water level and will evaluate the pressure distribution on each element assuming a linear increase in pressure with depth (The density of water is assumed to be 9.8 KN/m^3). Also, if the HYD parameter is specified, the program will assume that the hydrostatic loads have not been included in the analysis. For members that are subject to a combination of loads (i.e., bending plus compression) along with a hydrostatic pressure, the design will be done according to Clause 6.3.9 of the code. In the absence of any hydrostatic pressure on the member the design will be performed in accordance with Clause 6.3.8 of the code.

D12.B.1.5 Ultimate Limit State

Axial Tension

Clause 6.3.2 states that tubular members subject to axial tension shall satisfy the following condition:

$$N_{Sd} \le N_{t,Rd} = A \cdot f_y / \gamma_m$$

where

N_{Sd}	=	Design axial force (tension positive)
f_y	=	Characteristic yield strength
A	=	Cross section area
γ_m	=	Default material factor = 1.15

Axial Compression

Clause 6.3.3 states that tubular members subject to axial compression shall satisfy the following condition:

$$N_{Sd} \le N_{c,Rd} = A \cdot f_c / \gamma_m$$

where

N_{Sd}	=	Design axial force (compression positive)
f_c	=	Characteristic axial compressive strength
γ_m	=	Refer to clause 6.3.7

The design axial compressive strength for a member that is not subject to any hydrostatic pressure will be taken as the smaller of in plane or out of plane buckling strengths determined by the equations given below:

$$f_c = [1.0 - 028 \cdot {}^2]f_y \text{ when } \le 1.34$$

$$f_c = 0.9/ {}^2 \cdot f_y \text{ when } > 1.34$$

$$= \sqrt{(f_{cl}/f_E)} = k \cdot l/(\pi \cdot i)\sqrt{(f_{cl}/E)}$$

where

f_{cl}	=	Characteristic local buckling strength
$\bar{\lambda}$	=	Column slenderness parameter
f_E	=	Smaller Euler buckling strength in y or z direction
E	=	Young's modulus of elasticity = 2.1x10^5 MPa
k	=	Effective length factor, refer to Clause 6.3.8.2

l	=	Longer unbraced length in y or z direction
i	=	Radius of gyration

The characteristic local buckling strength is determined from:

$f_{cl} = f_y$ when $f_y/f_{cle} \leq 0.170$ (Plastic yielding)

$f_{cl} = [1.047 - 0.274 \cdot f_y/f_{cle}] \cdot f_y$ when $0.170 < f_y/f_{cle} \leq 1.911$ (Elastic/Plastic)

$f_{cl} = f_{cle}$ when $f_y/f_{cle} > 1.911$ (Elastic buckling)

where

f_{cle}	=	$2C_eE \cdot t / D$ (Characteristic elastic local buckling strength)
C_e	=	0.3 (Critical elastic buckling coefficient)
D	=	Outside diameter
t	=	wall thickness

For a member that is subject to pure compression, if $f_y/f_{cle} > 0.170$, the section will be classed as a CLASS 4 (slender section). In such cases, the value of the material factor (γ_m) used in the above checks is increased according to equation 6.22 (Cl. 6.3.7) of the code.

Bending

Clause 6.3.4 states that tubular members subject to pure bending alone shall satisfy:

$M_{Sd} \leq M_{Rd} = f_m \cdot W/\gamma_m$

where

M_{Sd}	=	Design bending moment
f_m	=	Characteristic bending strength
W	=	Elastic section modulus
γ_m	=	Refer to clause 6.3.7

The bending strength f_m is calculated as:

$f_m = Z/W \cdot f_y$ when $f_yD/(E \cdot t) \leq 0.0517$

$f_m = [1.13 - 2.58 \cdot f_yD/(E \cdot t)] \cdot Z/W \cdot f_y$ when $0.0517 < f_yD/(E \cdot t) \leq 0.1034$

$f_m = [0.94 - 0.76 \cdot f_yD/(E \cdot t)] \cdot Z/W \cdot f_y$ when $0.1034 < f_yD/(E \cdot t) \leq 120 \cdot f_y/E$

Shear

Clause 6.3.5 states that tubular members subject to shear shall satisfy:

$V_{Sd} \leq V_{Rd} = A \cdot f_y/(2\sqrt{3} \cdot \gamma_m)$

where

V_{Sd}	=	Design shear force
f_y	=	Yield strength
A	=	Cross section area
γ_m	=	Default material factor = 1.15

When torsional shear stresses are present, the following condition shall also be satisfied:

$M_{T,Sd} \leq M_{T,Rd} = 2 \cdot I_p f_y/(D\sqrt{3} \cdot \gamma_m)$

where

$M_{T,Sd}$	=	Design bending moment
I_p	=	Polar moment of inertia

Hydrostatic Pressure

Clause 6.3.6 states that tubular members subject to an external pressure shall primarily be checked for hoop buckling. The condition to be satisfied is:

$$\sigma_{p,Sd} \leq f_{h,Rd} = f_h/\gamma_{m)}$$

where

$\sigma_{p,Sd}$	=	$p_{Sd} \cdot D/(2 \cdot t)$
p_{Sd}	=	Design hydrostatic pressure
f_h	=	Characteristic hoop buckling strength
$\gamma_{m)}$	=	Refer to clause 6.3.7

The characteristic hoop buckling strength f_h, will be calculated as follows:

$$f_h = f_y \text{ when } f_{he} > 2.44 \cdot f_y$$

$$f_h = 0.7 \cdot f_y(f_{he}/f_y)_{0.4} \text{ when } 2.44 \cdot f_y \geq f_{he} > 0.55 \cdot f_y$$

$$f_h = f_{he} \text{ when } f_{he} \leq 0.55 \cdot f_y$$

where

=

The elastic hoop buckling strength, f_{he} , is evaluated as follows:

$$f_{he} = 2C_h E \cdot t/D$$

where

C_h	=	$0.44 \cdot t/D$ when $\mu \geq 1.6 \cdot D/t$
C_h	=	$0.44 \cdot t/D + 0.21 \cdot (D/t)^3/\mu^4$ when $0.825 \cdot D/t \leq \mu < 1.6 \cdot D/t$
C_h	=	$0.737/(\mu - 0.579)$ when $1.5 \leq \mu < 0.825 \cdot D/t$
C_h	=	0.8 when $\mu < 1.5$
μ	=	Geometric Parameter = $L/D\sqrt{(2 \cdot D/t)}$
L	=	Length of tubular member between stiffening rings, diaphragms, or end connections.

Combined Axial Tension and Bending (without Hydrostatic Pressure)

Clause 6.3.8.1 states that tubular members subject to axial tension and bending shall be designed to satisfy the following condition:

$$\left(\frac{N_{Sd}}{N_{t,Rd}}\right)^{1.75} + \frac{\sqrt{M_{y,Sd}^2 + M_{z,Sd}^2}}{M_{Rd}} \leq 1.0$$

where

$M_{y,Sd}$	=	the design bending moment about the y axis (out-of plane axis)
$M_{z,Sd}$	=	the design bending moment about the z axis (in plane axis)
N_{Sd}	=	the design axial force
M_{Rd}	=	the moment resistance (as determined by Clause 6.3.4)
$N_{t,Rd}$	=	the tension capacity of the section (as determined by Clause 6.3.2)

Combined Axial Compression and Bending (without Hydrostatic Pressure)

Clause 6.3.8.2 states that tubular members subject to axial tension and bending shall be designed to satisfy the following conditions:

$$\frac{N_{Sd}}{N_{c,Rd}} + \frac{1}{M_{Rd}}\sqrt{\left(\frac{C_{my}M_{y,Sd}}{1 - \frac{N_{Sd}}{N_{Ey}}}\right)^2 + \left(\frac{C_{mz}M_{z,Sd}}{1 - \frac{N_{Sd}}{N_{Ez}}}\right)^2} \leq 1.0$$

and

$$\frac{N_{Sd}}{N_{cl,Rd}} + \frac{\sqrt{M_{y,Sd}^2 + M_{z,Sd}^2}}{M_{Rd}} \le 1.0$$

where

N_{Sd}	=	the design axial compression
C_{my} and C_{mz}	=	the reduction factors corresponding to the Y and Z axes, respectively. You may specify a value for these using the `CMY` and `CMZ` design parameters, respectively (default0.85 for both).
N_{ey} and N_{ez}	=	the Euler buckling loads about y & z axes and are given by: $N_{Ey} = \frac{\pi^2 EA}{\left(\frac{k\ \ell}{i}\right)_y^2}$ $N_{Ez} = \frac{\pi^2 EA}{\left(\frac{k\ \ell}{i}\right)_z^2}$
k	=	the effective length factor and is given in table 6-2 of the code.
$N_{cl,\ Rd}$	=	the design axial local buckling resistance given by: $N_{cl,Rd} = \frac{f_{cl}A}{\gamma_M}$
f_{cl}	=	the characteristic local buckling strength (as determined by Clause 6.3.3)

The reduction factors used in this clause depend on the “structural element type” and will be as given in Table 6-2 of N-004. This requires the member to be classified under any one of the section types given in the table.

Combined Bending and Shear (without Hydrostatic Pressure)

Clauses 6.3.8.3 & 6.3.8.4 state that tubular members subject to beam shear force (excluding shear due to torsion) and bending moments shall satisfy:

$M_{Sd}/M_{Rd} \le \sqrt{(1.4 - V_{Sd}/V_{Rd})}$ when $V_{Sd}/V_{Rd} \ge 0.4$

$M_{Sd}/M_{Rd} \le 1.0$ when $V_{Sd}/V_{Rd} < 0.4$

If the member is subject to shear forces due to torsion along with bending moments, the condition to be satisfied is:

$M_{Sd}/M_{Red,Rd} \le \sqrt{(1.4 - V_{Sd}/V_{Rd})}$ when $V_{Sd}/V_{Rd} \ge 0.4$

$M_{Sd}/M_{Red,Rd} \le 1.0$ when $V_{Sd}/V_{Rd} < 0.4$

where

$M_{Red,Rd}$	=	$W \cdot f_{m,Red}/\gamma_m$
$f_{m,Red}$	=	$f_m\sqrt{[1 - 3(\tau_{T,Sd}/f_d)^2]}$
$\tau_{T,Sd}$	=	$M_{T,Sd}/(2\pi \cdot R^2 \cdot t)$
f_d	=	f_y/γ_m
R	=	Radius of the tubular member
γ_m	=	Refer to clause 6.3.7

Combined Loads with Hydrostatic Pressure

Clause 6.3.9 of NS-004 describes two methods to check for members subject to combined forces in the presence of hydrostatic pressure: depending on whether the hydrostatic forces were included as nodal forces in the analysis or not. If the hydrostatic forces have *not* been included in the analysis as nodal forces, Method A given in the code is used. If, however, the hydrostatic forces have been included in the analysis, then Method B in the code is used. Prior to proceeding with the checks described in the sections below, the section is verified for hoop stress limit per clause 6.3.6 (see Hydrostatic Pressure above).

The choice of method for checking members subject to combined forces and hydrostatic pressure used by STAAD.Pro will depend on the HYD parameter specified as a design parameter. If the HYD parameter has been specified, then the program will assume that the hydrostatic forces have not been included in the analysis and will perform the necessary checks as per Method A in code. If, on the other hand, the HYD parameter has not been specified, the program will use the section forces and use Method B in the code.

Combined Axial Tension, Bending, and Hydrostatic Pressure

Checks per Clause 6.3.9.1:

A. When HYD is specified:

The following condition is to be satisfied:

a. For the net axial tension condition ($\sigma_{a,Sd} \geq \sigma_{q,Sd}$)

$$\frac{\sigma_{a,Sd} - \sigma_{q,Sd}}{f_{th,Rd}} + \frac{\sqrt{\sigma^2_{my,Sd} + \sigma^2_{mz,Sd}}}{f_{mh,Rd}} \leq 1.0$$

where

$\sigma_{a,Sd}$	=	the design axial stress, excluding any axial compression from hydrostatic pressure.
$\sigma_{q,Sd}$	=	the design axial compressive stress due to hydrostatic pressure. (i.e., the axial load arising from the hydrostatic pressure being applied as nodal loads).
$\sigma_{my,Sd}$	=	the out of plane bending stress
$\sigma_{mz,Sd}$	=	the in plane bending stress
$f_{th,RD}$	=	$f_y/\gamma_m[\sqrt{(1 + 0.09 \cdot B^2 - B^{2\eta})} - 0.3B]$
$f_{mh,RD}$	=	$f_m/\gamma_m[\sqrt{(1 + 0.09 \cdot B^2 - B^{2\eta})} - 0.3B]$
B	=	$\sigma_{psd}/ f_{h,Rd}$
η	=	$5 - 4 \cdot f_h/f_y$

b. For the net axial compression condition ($\sigma_{a,Sd} < \sigma_{q,Sd}$)

$$\frac{|\ \sigma_{a,Sd} - \sigma_{q,Sd}\ |}{f_{cl,Rd}} + \frac{\sqrt{\sigma^2_{my,Sd} + \sigma^2_{mz,Sd}}}{f_{mh,Rd}} \leq 1.0$$

where

$f_{cl,Rd}$	=	f_{cl}/γ_m
f_{cl}	=	the characteristic local buckling strength (as determined by Clause 6.3.3)

Additionally, when:

$$\sigma_{c,Sd} > 0.5 \cdot f_{he}/\gamma_m$$

and

$$f_{cle} > 0.5 \cdot f_{he}$$

the following condition shall be satisfied in addition to the above check(s):

$$\frac{\sigma_{c,Sd} - 0.5\frac{f_{he}}{\gamma_M}}{\frac{f_{cle}}{\gamma_M} - 0.5\frac{f_{he}}{\gamma_M}} + \left(\frac{\sigma_{p,Sd}}{\frac{f_{he}}{\gamma_M}}\right)^2 \le 1.0$$

where

$\sigma_{c,Sd}$ = the maximum compressive stress at that section.

B. When HYD has not been specified:

$$\frac{\sigma_{ac,Sd}}{f_{th,Rd}} + \frac{\sqrt{\sigma^2_{my,Sd} + \sigma^2_{mz,Sd}}}{f_{mh,Rd}} \le 1.0$$

where

$\sigma_{ac,Sd}$ = the axial stress in the member

Combined Axial Compression, Bending, and Hydrostatic Pressure

Checks per Clause 6.3.9.2:

A. Method used when HYD has been specified:

The following condition is to be satisfied:

$$\frac{\sigma_{a,Sd}}{f_{ch,Rd}} + \frac{1}{f_{mh,Rd}}\sqrt{\left(\frac{C_{my}\sigma_{my,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{Ey}}}\right)^2 + \left(\frac{C_{mz}\sigma_{mz,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{Ez}}}\right)^2} \le 1.0$$

and

$$\frac{\sigma_{a,Sd} + \sigma_{q,Sd}}{f_{cl,Rd}} + \frac{\sqrt{\sigma^2_{my,Sd} + \sigma^2_{mz,Sd}}}{f_{mh,Rd}} \le 1.0$$

Where:

where

$\sigma_{a,Sd}$ = the design axial stress that excludes the stress from hydrostatic pressure

fch,Rd = $\frac{1}{2}\frac{f_{cl}}{\gamma_M}\left[\xi - \frac{2\sigma_{q,Sd}}{f_{cl}} + \sqrt{\xi^2 + 1.12\bar{\lambda}^2\frac{\sigma_{q,Sd}}{f_{cl}}}\right] when \bar{\lambda} < 1.34\sqrt{\left(1 - \frac{2\sigma_{q,Sd}}{f_{fl}}\right)^{-1}}$

$\frac{0.9 f_{cl}}{\bar{\lambda}^2\gamma_M} when \bar{\lambda} \ge 1.34\sqrt{\left(1 - \frac{2\sigma_{q,Sd}}{f_{fl}}\right)^{-1}}$

ξ = $1-0.28\bar{\lambda}^2$

Additionally, when:

$\sigma_{c,Sd} > 0.5 \cdot f_{he}/\gamma_m$

and

$f_{cle} > 0.5 \cdot f_{he}$

the following condition shall be satisfied in addition to the above check(s):

$$\frac{\sigma_{c,Sd} - 0.5\frac{f_{he}}{\gamma_M}}{\frac{f_{cle}}{\gamma_M} - 0.5\frac{f_{he}}{\gamma_M}} + \left(\frac{\sigma_{p,Sd}}{\frac{f_{he}}{\gamma_M}}\right)^2 \leq 1.0$$

B. Method used when HYD has *not* been specified:

The following condition is to be satisfied:

a. For the net axial tension condition ($\sigma_{ac,Sd} \geq \sigma_{q,Sd}$)

$$\frac{\sigma_{ac,Sd} - \sigma_{q,Sd}}{f_{ch,Rd}} + \frac{1}{f_{mh,Rd}}\sqrt{\left(\frac{C_{my}\sigma_{my,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{Ey}}}\right)^2 + \left(\frac{C_{mz}\sigma_{mz,Sd}}{1 - \frac{\sigma_{a,Sd}}{f_{Ez}}}\right)^2} \leq 1.0$$

and

$$\frac{\sigma_{ac,Sd}}{f_{cl,Rd}} + \frac{\sqrt{\sigma_{my,Sd}^2 + \sigma_{mz,Sd}^2}}{f_{mh,Rd}} \leq 1.0$$

(Refer to the previous section for an explanation of these terms).

b. For the net axial compression condition ($\sigma_{ac,Sd} < \sigma_{q,Sd}$)

$$\frac{\sigma_{ac,Sd}}{f_{cl,Rd}} + \frac{\sqrt{\sigma_{my,Sd}^2 + \sigma_{mz,Sd}^2}}{f_{mh,Rd}} \leq 1.0$$

(Refer to the previous section for an explanation of these terms).

Additionally, when:

$\sigma_{c,Sd} > 0.5 \cdot f_{he}/\gamma_m$

and

$f_{cle}/\gamma_m > 0.5 \cdot f_{he}/\gamma_m$

the following condition shall be satisfied in addition to the above check(s):

$$\frac{\sigma_{c,Sd} - 0.5\frac{f_{he}}{\gamma_M}}{\frac{f_{cle}}{\gamma_M} - 0.5\frac{f_{he}}{\gamma_M}} + \left(\frac{\sigma_{p,Sd}}{\frac{f_{he}}{\gamma_M}}\right)^2 \leq 1.0$$

where

$\sigma_{c,Sd}$ = the maximum compressive stress at that section.

D12.B.3 Design Parameters

Design parameters communicate specific design decisions to the program. They are set to default values to begin with and may be altered to suite the particular structure.

Table 189: Design Parameters for NORSOK N-004 design code

Parameter Name	Default Value	Description
CODE	none	Must be specified as NORSOK. **Note:** Do not use the shortened NOR, as this initiates an D12.A. Norwegian Codes - Steel Design per NS 3472 / NPD (on page 1790).
FYLD	235 [MPa]	Yield strength of steel, f_y (St37) **Note:** Note, if the SGR value is specified, then the associated value of f_y for that steel grade will be used for a member in lieu of the FYLD value.
KY	1.0	Effective length factor, k, in local Y-axis, usually minor axis.
KZ	1.0	Effective length factor, k, in local Z-axis, usually major axis.
LX		Effective length for lateral torsional buckling.
LY	Member Length	Length in local Y axis for slenderness value KL/r
LZ	Member Length	Length in local Z axis for slenderness value KL/r
CMY	0.85	Reduction factor C_m corresponding to the Y axis.
CMZ	0.85	Reduction factor C_m corresponding to the Z axis.
LSR		Length of Tubular between Stiffening Rings. This value is required to calculate Design Hoop Stress due to Hydrostatic Pressure to check Hoop Buckling as per clause 6.3.6.1.

Parameter Name	Default Value	Description
HYD	0.0	The Y-coordinate, current units, of the maximum water level with respect to the origin. **Note:** If `SET Z UP` command has been specified, then yi will be the Z co-ordinate of the max water level. For HYD > 0, the value of max. hydrostatic pressure calculated is reported for each member in a `TRACK 2.0` output.
PSD	0.0	Water pressure at each section in absence of HYD.
SGR	0	Steel Grade per EC3 (D5.C.6 Design Parameters (on page 1502)): 0 = S 235 grade steel 1 = S 275 grade steel 2 = S 355 grade steel 3 = S 420 grade steel 4 = S 460 grade steel
DMAX	100.0 [cm]	Maximum allowable depth of steel section.
DMIN	0.0 [cm]	Minimum allowable depth of steel section.
DFF	None (Mandatory for deflection check)	"Deflection length"/maximum allowable local deflection.
MAIN	0	Option to design for slenderness. 0 = Check for slenderness 1 = Do not check for slenderness Any value greater than 1.0 is used as the limit for slenderness in compression.
TMAIN	180.0	Slenderness limit in tension. Slenderness limit is checked based the `MAIN` parameter.
FU	420 [MPa]	Ultimate Tensile Strength of Steel

Parameter Name	Default Value	Description
TRACK	0	Output detail: 0 = Only a summary of the design checks performed is printed. 2 = All the details of the member checks and the various clause checks performed are printed. 3 = Prints detailed member check and joint check results.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
BEAM	0	Beam segment locations for design: 0 = design only for end moments and those at locations specified by `SECTION` command. 1 = Perform design for moments at twelfth points along the beam.
DJ1	Start Joint of member	Joint No. denoting start point for calculation of “deflection length”
DJ2	End Joint of member	Joint No. denoting end point for calculation of “deflection length”

Notes

a. C1 and C2 Parameters

The default values of these coefficients are taken from Table 6-4 of N-004 and depend on the joint and load type:

Table 190: Default values for C1 and C2 parameters

Joint Type	C1	C2
T or Y joints under brace axial load	25	11
X joints under brace axial load	20	22
K joints under balanced axial load	20	22
All joints under brace moment loading	25	30

Note: These values can be changed by setting the K, X, and Y values in the D12.B.8 External Geometry File (on page 1852).

Example

Note: This is a partial example containing only the information pertaining to the NORSOK N-004 steel design code; used at the end of the input file.

```
* Check tubular members according NORSOK N-004
CODE NORSOK
HYD 3.0 MEMB 1 TO 3
PSD 10 MEMB 7 10
SGR 2 MEMB 1 TO 3 7 10
TRACK 2 MEMB 1 TO 3 7 10
CHECK CODE MEMB 1 TO 3 7 10
```

D12.B.4 Code Checking

The purpose of code checking is to ascertain whether the provided section properties of the members are adequate as per N-004. Code checking is done using the forces and moments at specific sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed or failed the checks, the critical condition of NORSOK code, the value of the ratio of the critical condition (overstressed for value more than 1.0 or any other specified `RATIO` value), the governing load case, and the location (distance from the start of the number of forces in the member) where the critical condition occurs.

D12.B.5 Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (i.e., the lightest section which fulfills the code requirements for the specified member). The section selected will be of the same type section as originally designated for the member being designed. Member selection can also be constrained by the parameters `DMAX` and `DMIN` which limit the maximum and minimum depth of the members.

Selection of members whose properties are originally input from a user created table will be limited to sections in the user table.

D12.B.6 Tubular Joint Checking

The design of tubular joints for this implementation shall be based on section 6.4 of N-004 and will be applicable to joints formed from a connection of two or more members.

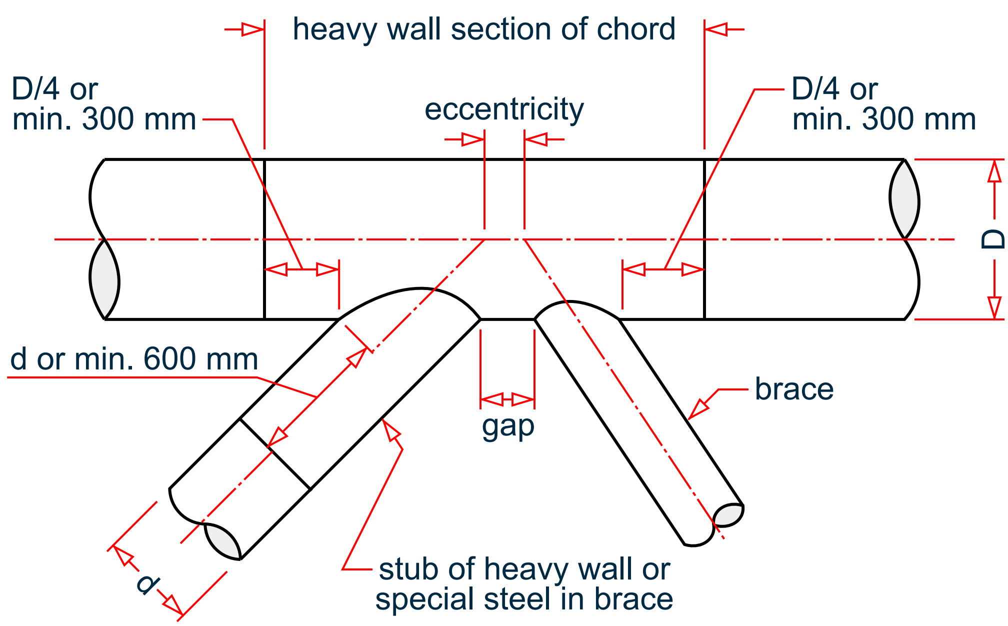


Figure 188: Typical Tubular Joint (Fig 6-1 in N004)

Prior to completing a joint design, the joint should be classified into one of the three categories given by the code. Joint classification is the process whereby a “brace” member connecting into a “chord” member is classified into one of these categories based on the axial force components in the brace. The classification normally considers all the members at a joint that lie in a plane. N-004 defines three joint classification categories: K, X, or Y (or a combination of these).

Joint Classification	Description
K	The axial force in the brace should be balanced by forces in the other braces in the same plane and on the same side of the joint. The code allows a 10% tolerance in the balancing force.
X	The axial force in the brace is reacted as a beam shear in the chord.
Y	The axial force in the brace is carried through the chord to braces in the opposite side.

Note: Typical examples of these joint types are given in Figure 6-3 of the N-004 code. It is worth noting that the joint class for each brace will be different for each load case.

Note: STAAD.Pro does not perform an automatic classification of the joints. This is left up to the engineer. All joints will initially be classified as Y in the generation of the D12.B.8 External Geometry File (on page 1852). Joints should be re-classified as necessary before performing the final joint capacity checks.

The checks for joint capacity are given in Cl. 6.4.3.2 to 6.4.3.6 and STAAD.Pro performs the checks as per these clauses. However, the program does not deal with conical joint transitions and joints with joint cans. The code also specifies checks and limits for the gaps and eccentricity of joints. This implementation will not perform such geometry checks.

The details of the checks done and the methodology will be discussed in the following sections.

Joint checks are performed as part of the `CODE CHECK` command and are displayed when either a `TRACK 0` or `TRACK 3` output design parameter is specified (with the former giving a summary and the latter giving detailed member & joint check results). Refer to D12.B.9 Examples (on page 1855) for example input and output including joint checks.

D12.B.6.1 Identification and Classification of CHORD and BRACE Members

This is a two step process where the program automatically identifies the CHORD and BRACE members at a joint and perform a default joint check. The input variables used for the initial joint checks will be generated in an external text file. You can then use this text file to edit or modify the input variables and perform a final check as necessary.

When a member is included in the `CHECK CODE` list, the program designs the member and also then proceeds to perform the joint checks at either ends of the member. It finds all members that connect to the end nodes of the member being designed and then proceeds to calculate the joint capacities based on each such connected member. The design process follows the section 6.4.3 of the code. This sections specifies certain pre-requisites for its application. If these pre-requisites are not satisfied, then the joint checks will not be performed. The following geometry checks must be satisfied in order for the joint check to be performed:

- $0.2 \leq \beta \leq 1.0$
- $10 \leq \gamma \leq 50$
- $30° \leq \theta \leq 90°$
- $g / D \geq -0.6$ (for K joints)

where

γ	=	$D / 2T$, where D = chord diameter and T = chord thickness.

The program will also produce an output file called `filename_Jnt.txt`, where "`filename`" will be the name of the `.std` input file. This format of this text file is explained in D12.B.8 External Geometry File (on page 1852).

You can then edit this text file to set up the necessary design parameters. Once the program finds of the `_JOINTS.txt` file, it will read in the necessary parameters from this file and perform the subsequent design checks. This is done by simply performing the design and analysis again for the input file.

Note: This file will be produced only once (i.e., when this file does not exist). If this file exists, it is assumed that you have already done a joint design check and hence the program reads the values from this file and uses these for joint checks.

D12.B.7 Tubular Joint Resistance

D12.B.7.1 Basic Joint Resistances

The characteristic joint resistance between a chord and a brace is given by:

$$N_{Rd} = \frac{f_y T^2}{\gamma_M \sin\theta} Q_u Q_f$$

$$M_{Rd} = \frac{f_y T^2 d}{\gamma_M \sin\theta} Q_u Q_f$$

Where:

N_{Rd} is the joint design axial resistance

M_{Rd} is the joint design bending moment resistance.
f_y is the yield strength
γ_m = Default material resistance =1.15
θ is the angle between the chord and the brace (max θ = 90 degrees)
Q_u = Strength factor which varies with the joint type and the action type in the brace. Refer to Table 6-3 and Clause 6.4.3.3 of N-004 for these equations.
$Q_f = 1.0 – \lambda A^2$

$$A^2 = C_1\left(\frac{\sigma_{a,Sd}}{f_y}\right)^2 + C_2\left(\frac{\sigma^2_{my,Sd} + \sigma^2_{mz,Sd}}{1.62 f^2_y}\right)$$

$\sigma_{p,Sd}$ is the design axial stress in the chord
$\sigma_{my,Sd}$ is the design in-plane bending stress in the chord
$\sigma_{mz,Sd}$ is the design out-of-plane bending stress in the chord
C1 is the coefficient used for the axial stress term in calculating the joint resistance. C2 is the coefficient used for the bending stress term in calculating the joint resistance. The default values of C1 and C2 are as given in Table 6-4 of N-004. The actual values used are dependent on the values of K, X, and Y specified for the joint in the D12.B.8 External Geometry File (on page 1852).
See also Figures 6-3 to 6-6 of N-004 for definition of the various terms for various joint classes.

D12.B.7.2 Strength Check for Joints

Each brace to chord joint to be checked will have to satisfy the following condition:

$$\frac{N_{Sd}}{N_{Rd}} + \left(\frac{M_{z,Sd}}{M_{z,Rd}}\right)^2 + \frac{M_{y,Sd}}{M_{y,Rd}} \le 1$$

Where:

N_{Sd} is the design axial force in the brace,
N_{Rd} is the joint design axial resistance
$M_{z,Sd}$ is the in plane bending moment in the brace
$M_{y,Sd}$ is the out of plane bending moment in the brace
$M_{z,Rd}$ is the in plane bending moment resistance
$M_{y,Rd}$ is the out of plane bending moment resistance

D12.B.8 External Geometry File

The data contained in the `filename_JOINTS.NGo` file should meet the following format. The overall process of performing punching shear checks consists of two steps which are explained in D12.B.6 Tubular Joint Checking (on page 1849).

General Format

```
LOAD LIST load_list
```

```
JOINT    NODE    K     X    Y     CHORD   CLEN       D      T      BRACE     BLEN
d           t GAP
```

```
j# n# K% X% Y% C# CLEN D T B# BLEN d t gap
```

Where:

j#	the joint number
n#	the node number
K%, X%, and Y%	The fractional contributions of K-type, X type and Y-type, respectively. Initially the joints will be classed as Y (i.e., K=0, X=0 and Y=1).
C#	the member numbers of the CHORD
CLEN	the length of chord member
D, T	Diameter and thickness of CHORD
B#	the member number of the brace
BLEN	the length of chord member
d, t	Diameter and thickness of BRACE
gap	Distance required to calculate gap factor for K bracing. Initially, the value of GAP is assumed as 0.

Note: All distances are provided in units of inches.

Example

```
LOAD LIST 1 2 4
JOINT    NODE    K     X    Y     CHORD     CLEN     D       T     BRACE
BLEN   d         t      GAP
  1      3        0    0    1       2       5.0   0.168    0.10      1
4.0    0.140   0.010   0
  2      3       0    0    1       2       5.0   0.168    0.10      16
6.043  0.075   0.005   0
```

D12.B.9 Tabulated Results

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

Member	the member number for which the design is performed.
TABLE	the steel section name which has been checked against the N-004 code or has been selected.
RESULTS	prints whether the member has PASSed or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark on front of the member.
CRITICAL COND	the section of the N-004 code which governs the design.
RATIO	prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
LOADING	the load case number which governed the design.
FX, MY, and MZ	provide the axial force, moment in local Y-axis, and the moment in local Z-axis respectively. Although STAAD does consider all the member forces and moments (except torsion) to perform design, only FX, MY and MZ are printed since they are the ones which are of interest, in most cases.

LOCATION specifies the actual distance from the start of the member to the section where design forces govern.

Note: If the parameter TRACK is set to 2.0, the program will block out part of the table and will print the allowable bending stressed in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV).

D12.B.9.1 Sample TRACK 2.0 Output

```
                       STAAD.PRO CODE CHECKING - NORSOK-N004 (V1.0)
                     ************************************************

ALL UNITS ARE - KN   METE (UNLESS OTHERWISE NOTED)

MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  ST   PIP13910.0               (BRITISH SECTIONS)
                          PASS     Eq. 6.44           0.170         1
                       0.01 C         1.01           6.39       0.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  S 355
     Modulus of elasticity    = 204999.98 N/mm2
     Design Strength  (py)    =355.00  N/mm2

  SECTION PROPERTIES (units - cm)
     Member Length =    400.00
     Gross Area of cross section =   40.70

                                    z-axis          y-axis
     Moment of inertia        :      862.000         862.000
     Plastic modulus          :      168.554         168.554
     Elastic modulus          :      123.407         123.407
     Radius of gyration       :        4.602           4.602
     Effective Length         :      400.000         400.000

  DESIGN PARAMETER (units - m)   N004/2004
  Height of water lavel   :        3.000
  CMZ  :   0.85      CMY  :   0.85
  KZ   :   1.00      KY   :   1.00

  SECTION CLASSIFICATION   :       Class 1

  CAPACITIES (units - kN,m)
     Tension Capacity                     :       1256.4
     Compression Capacity                 :        790.1
     Bending Capacity                     :         52.0
     Shear Capacity                       :        362.7
     Shear Capacity due to torsional moment:        44.0

  HYDROSTATIC PRESSURE CALCULATION (units - N,mm) - Cl.6.3.6
     Max design hydrostatic pressure, (psd) :        0.000
     Max design hoop stress, (sigma_psd))   :        0.000

  CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):
   CLAUSE            RATIO LOAD     FX       VY      VZ      MZ      MY
```

```
   Cl:6.3.2           0.000    1     0.0       -       -       -       -
   Cl:6.3.3           0.000    1     0.0       -       -       -       -
   Cl:6.3.4           0.102    1      -        -       -     -5.3     0.0
   Cl:6.3.5           0.031    1      -     -11.2     0.5      -       -
   Cl:6.3.8.(1 & 2)   0.124    1     0.0       -       -      6.4     1.0
   Cl:6.3.8.(3 & 4)   0.102    1     -      -0.5      0.5    -5.3     0.0
   Cl:6.3.9           0.170    1     0.0       -       -      6.4     1.0
 ======================================================================
```

D12.B.9 Examples

The following examples of using the NORSOK N-004

Norsok Joint 1

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Sep-10
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 4; 5 1 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
2 TABLE ST PIPD60
1 3 TO 5 TABLE ST PIPS50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1 LIST ALL
MEMBER LOAD
2 UNI GY -4
1 UNI PX 4
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE NORSOK
HYD 2 ALL
SGR 1 ALL
TRACK 3 MEMB 1
CHECK CODE MEMB 1
PRINT MEMBER FORCES LIST 1 2 4
```

```
PRINT MEMBER PROPERTIES LIST 1 2 4
FINISH
```

When the preceding input file is run, the following external geometry file (`*_Jnt.TXT` file) is created.

```
*MEMBER    NODE    K       X       Y       CHORD       CLEN
D           T     BRACE       BLEN          D              T      GAP
     1       2   0.000   0.000   1.000       1      393.701
5.563       0.241      4      556.777       5.563        0.241    0.00
```

And the output file section for design results produced are:

```
                       STAAD.PRO CODE CHECKING - NORSOK-N004:Re-2 (V1.0)
                     *****************************************************

 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE      Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *    1  ST   PIPS50                   (AISC SECTIONS)
                           FAIL     Eq. 6.43           1.434         1
                      11.00 C          0.00          37.69       10.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    = 204999.98 N/mm2
      Design Strength  (py)    =275.00  N/mm2

   SECTION PROPERTIES (units - cm)
      Member Length =   1000.00
      Gross Area of cross section =   25.87

                                   z-axis          y-axis
      Moment of inertia        :      595.211         595.211
      Plastic modulus          :      111.935         111.935
      Elastic modulus          :       84.248          84.248
      Radius of gyration       :        4.797           4.797
      Effective Length         :     1000.000        1000.000

   DESIGN PARAMETER (units - m)   N004/2004
   Height of water lavel    :        2.000
   CMZ  :   0.85      CMY  :   0.85
   KZ   :   1.00      KY   :   1.00

   SECTION CLASSIFICATION   :       Class 1

   CAPACITIES (units - kN,m)
      Slenderness ratio (kzlz/rz)          :        208.5
      Slenderness ratio (kyly/ry)          :        208.5
      Tension Capacity                     :        618.7
      Compression Capacity                 :         94.2
      Bending Capacity                     :         26.8
      Shear Capacity                       :        178.6
      Torsion Capacity                     :         23.3

   HYDROSTATIC PRESSURE CALCULATION (units - N,mm) - Cl.6.3.6
      Max design hydrostatic pressure, (psd) :        0.000
      Max design hoop stress, (sigma_psd))   :        0.000
     STAAD SPACE                                              -- PAGE NO.    5
```

```
   CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):
    CLAUSE       RATIO     LOAD    FX        VY        VZ        MZ
MY        MX
   Cl:6.3.2      0.000       0    0.000     -         -         -
-        -
   Cl:6.3.3      0.138       1   12.990     -         -         -
-        -
   Cl:6.3.4      1.408       1     -        -         -      37.691
0.000     -
   Cl:6.3.5      0.116       1     -      20.651    0.000     -
-       0.000
   Cl:6.3.8.1    0.000       0    0.000     -         -       0.000
0.000     -
   Cl:6.3.8.3    1.408       1     -      20.651    0.000   37.691
0.000    0.000
   Cl:6.3.9.2    1.434       1   11.002     -         -      37.691
0.000     -

==============================================================================
=======
 ** PUNCHING ** No Check for Joint      1 on Member      1

 NODE NO:      2  (All units are in KN-M)
 ----------------------------------------
 CHORD MEMBER:     1-PIPS50
 BRACE MEMBER:     4-PIPS50
 Angle Theta :45.00 deg.

 Critical Loadcase :     1
 Design Loads for Chord Memb: FX=   11.00 MZ=     37.69 MY=      0.00
 Design Loads for Brace Memb: FX=   22.28 MZ=      0.76 MY=      0.00

 Joint Class: 0.000k + 1.000y + 0.000x

 Geometry Factor (Qb)     = 1.796  Gap Factor (Qg) =  2.119
 Strength factor(Qu) for Axial                   :     28.01
 Strength factor(Qu) for in-plane bending        :     15.29
 Strength factor(Qu) for out-of-plane bending    :     10.87
 Chord Action Factor(Qf) for axial force         :      0.46
 Chord Action Factor(Qf) for in-plane bending    :      0.19
 Chord Action Factor(Qf) for out-of-plane bending:      0.62
 C1 = 25.00  C2 = 11.00

 Axial Capacity: N,Rd  =     163.41(Eq: 6.52)
 Moment Capacity: Mz,Rd =       5.21 ; My,Rd =      12.11(Eq: 6.53)

 Critical Ratio =    0.158(Eq: 6.57)   PASS
 ========================================================================
```

Norsok Joint 2

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Sep-10
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 4; 5 1 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
*STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 4 5 TABLE ST PIP886.3
MEMBER PROPERTY INDIAN
2 3 TABLE ST PIP1143H
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1 LIST 1 TO 5
MEMBER LOAD
2 UNI GY -4
1 UNI PX 4
1 UMOM Y 5 0 10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE NORSOK
HYD 2 ALL
SGR 1 ALL
TRACK 3 MEMB 1
CHECK CODE MEMB 1
PRINT MEMBER FORCES LIST 1 2 4
PRINT MEMBER PROPERTIES LIST 1 2 4
FINISH
```

When the preceding input file is run, the following external geometry file (`*_Jnt.TXT` file) is created.

```
*MEMBER    NODE    K        X        Y        CHORD       CLEN
D            T     BRACE        BLEN          D             T      GAP
     1        2   0.000   0.000   1.000        2      393.701
4.500        0.213      1      393.701        3.500        0.248    0.00
```

And the output file section for design results produced are:

```
                      STAAD.PRO CODE CHECKING - NORSOK-N004:Re-2 (V1.0)
                     *****************************************************

 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE      Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  ST   PIP886.3                 (BRITISH SECTIONS)
```

```
                           FAIL     Eq. 6.43           5.386        1
                        9.41 C       -24.30          31.86       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    = 204999.98 N/mm2
      Design Strength  (py)    =275.00  N/mm2

   SECTION PROPERTIES (units - cm)
      Member Length =   1000.00
      Gross Area of cross section =   16.30

                                    z-axis          y-axis
      Moment of inertia        :      140.000         140.000
      Plastic modulus          :       43.067          43.067
      Elastic modulus          :       31.496          31.496
      Radius of gyration       :        2.931           2.931
      Effective Length         :     1000.000        1000.000

   DESIGN PARAMETER (units - m)   N004/2004
   Height of water lavel   :        2.000
   CMZ  :   0.85      CMY  :   0.85
   KZ   :   1.00      KY   :   1.00

   SECTION CLASSIFICATION   :       Class 1

   CAPACITIES (units - kN,m)
      Slenderness ratio (kzlz/rz)           :        341.2
      Slenderness ratio (kyly/ry)           :        341.2
      Tension Capacity                      :        389.8
      Compression Capacity                  :         22.2
      Bending Capacity                      :         10.3
      Shear Capacity                        :        112.5
      Torsion Capacity                      :          8.7

   HYDROSTATIC PRESSURE CALCULATION (units - N,mm) - Cl.6.3.6
      Max design hydrostatic pressure, (psd) :        0.020
      Max design hoop stress, (sigma_psd))   :        0.138
     STAAD SPACE                                              -- PAGE NO.    5


   CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):
    CLAUSE       RATIO     LOAD    FX       VY       VZ       MZ
MY       MX
   Cl:6.3.2      0.000       0    0.000     -        -        -
-        -
   Cl:6.3.3      0.424       1    9.409     -        -        -
-        -
   Cl:6.3.4      3.530       1     -        -        -      36.295
8.543     -
   Cl:6.3.5      0.357       1     -      19.557    1.715     -
-       3.104
   Cl:6.3.8.1    0.000       0    0.000     -        -       0.000
0.000     -
   Cl:6.3.8.4    3.686       1     -      20.443    1.715   36.295
8.543    3.104
   Cl:6.3.9.2    5.386       1    9.409     -        -      31.861
24.304     -
```

```
===============================================================================
=======
 ** PUNCHING ** No Check for Joint      1 on Member      1

 NODE NO:      2  (All units are in KN-M)
 ----------------------------------------
 CHORD MEMBER:     2-PIP1143H
 BRACE MEMBER:     1-PIP886.3
 Angle Theta :90.00 deg.

 Critical Loadcase :     1
 Design Loads for Chord Memb: FX=    5.72 MZ=     36.92 MY=      4.74
 Design Loads for Brace Memb: FX=    8.16 MZ=     36.30 MY=      8.54

 Joint Class: 0.000k + 1.000y + 0.000x

 Geometry Factor (Qb)     = 1.095  Gap Factor (Qg) =  2.249
 Strength factor(Qu) for Axial                     :     17.46
 Strength factor(Qu) for in-plane bending          :     11.39
 Strength factor(Qu) for out-of-plane bending      :      6.53
 Chord Action Factor(Qf) for axial force           :      0.00
 Chord Action Factor(Qf) for in-plane bending      :      0.00
 Chord Action Factor(Qf) for out-of-plane bending:        0.00
 C1 = 25.00  C2 = 11.00

 Axial Capacity: N,Rd  =       0.00(Eq: 6.52)
 Moment Capacity: Mz,Rd =       0.00 ; My,Rd =       0.00(Eq: 6.53)

 Critical Ratio = ********(Eq: 6.57)   FAIL
 ========================================================================
```

Norsok Joint 3

```
STAAD SPACE NORSOK_JOINT3
START JOB INFORMATION
ENGINEER DATE 01-Oct-10
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 240 0; 3 240 240 0; 4 240 0 0; 5 240 96 0; 6 120 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 5; 4 5 3; 5 2 5; 6 2 6; 7 2 4;
DEFINE MATERIAL START
ISOTROPIC BS_STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283599
ALPHA 2.16e-005
DAMP 0.03
G 11435.7
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
```

```
DAMP 0.03
TYPE STEEL
STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TO 4 TABLE ST PIP35510.0
5 TO 7 TABLE ST PIP27310.0
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 6 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1 LIST 1 TO 4 6 7
JOINT LOAD
2 FX 5 FY -5 MX 1 MY 1 MZ 1
MEMBER LOAD
2 UNI GY -2
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT MEMBER FORCES LIST 1 2 4
PARAMETER 1
CODE NORSOK
SGR 1 ALL
HYD 39.37 ALL
STIFF 39.37 ALL
TRACK 3 ALL
CHECK CODE MEMB 1 4
PERFORM ANALYSIS
FINISH
```

When the preceding input file is run, the following external geometry file (`*_Jnt.TXT` file) is created.

```
*MEMBER    NODE    K       X       Y       CHORD       CLEN
D            T     BRACE       BLEN          D             T      GAP
     1        2   0.000   0.000   1.000        1      240.000
14.000        0.394      2      240.000       14.000        0.394    0.00
     1        2   0.000   0.000   1.000        1      240.000
14.000        0.394      5      279.886       10.748        0.394    0.00
     1        2   0.000   0.000   1.000        1      240.000
14.000        0.394      7      339.411       10.748        0.394    0.00
     4        5   0.000   0.000   1.000        4      144.000
14.000        0.394      5      279.886       10.748        0.394    0.00
```

And the output file section for design results produced are:

```
                       STAAD.PRO CODE CHECKING - NORSOK-N004:Re-2 (V1.0)
                     *****************************************************

 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE      Noted)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  ST   PIP35510.0               (BRITISH SECTIONS)
                             FAIL     Eq. 6.44           1.842         1
                     188.44 C          0.14        3844.85      240.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    = 204999.75 N/mm2
```

```
      Design Strength  (py)    =275.00  N/mm2

   SECTION PROPERTIES (units - cm)
      Member Length =    609.60
      Gross Area of cross section =  109.00

                                  z-axis          y-axis
      Moment of inertia        :    16220.002       16220.002
      Plastic modulus          :     1194.726        1194.726
      Elastic modulus          :      912.261         912.261
      Radius of gyration       :       12.199          12.199
      Effective Length         :      609.600         609.600

   DESIGN PARAMETER (units - m)   N004/2004
   Height of water lavel   :       1.000
   CMZ  :   0.85      CMY  :   0.85
   KZ   :   1.00      KY   :   1.00

   SECTION CLASSIFICATION   :       Class 1

   CAPACITIES (units - kN,m)
      Slenderness ratio (kzlz/rz)           :         50.0
      Slenderness ratio (kyly/ry)           :         50.0
      Tension Capacity                      :       2606.5
      Compression Capacity                  :       2358.8
      Bending Capacity                      :        285.6
      Shear Capacity                        :        752.4
      Torsion Capacity                      :        251.9

   HYDROSTATIC PRESSURE CALCULATION (units - N,mm) - Cl.6.3.6
      Max design hydrostatic pressure, (psd) :        0.000
      Max design hoop stress, (sigma_psd))   :        0.000
     NORSOK_JOINT3                                           -- PAGE NO.    6


   CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):
    CLAUSE       RATIO     LOAD    FX       VY       VZ       MZ
MY       MX
   Cl:6.3.2      0.000       0    0.000     -        -        -
-       -
   Cl:6.3.3      0.358       1  843.312     -        -        -
-       -
   Cl:6.3.4      1.521       1     -        -        -     434.409
0.016     -
   Cl:6.3.5      0.142       1     -     106.846    0.011     -
-      0.026
   Cl:6.3.8.1    0.000       0    0.000     -        -       0.000
0.000     -
   Cl:6.3.8.4    1.521       1     -     106.846    0.011  434.409
0.016    0.026
   Cl:6.3.9.2    1.842       1  838.208     -        -     434.409
0.016     -

==============================================================================
=======
 ** PUNCHING ** No Check for Joint      1 on Member      1

 NODE NO:      2  (All units are in KN-M)
```

```
----------------------------------------
CHORD MEMBER:     1-PIP35510.0
BRACE MEMBER:     2-PIP35510.0
Angle Theta :90.00 deg.

Critical Loadcase :     1
Design Loads for Chord Memb: FX=  838.21 MZ=    434.41 MY=      0.02
Design Loads for Brace Memb: FX=  274.77 MZ=    924.49 MY=      0.03

Joint Class: 0.000k + 1.000y + 0.000x

Geometry Factor (Qb)     = 1.796  Gap Factor (Qg) =  2.385
Strength factor(Qu) for Axial                     :     28.01
Strength factor(Qu) for in-plane bending          :     18.97
Strength factor(Qu) for out-of-plane bending      :     13.49
Chord Action Factor(Qf) for axial force           :      0.33
Chord Action Factor(Qf) for in-plane bending      :      0.00
Chord Action Factor(Qf) for out-of-plane bending:        0.53
C1 = 25.00  C2 = 11.00

Axial Capacity: N,Rd  =     221.30(Eq: 6.52)
Moment Capacity: Mz,Rd =       0.00 ; My,Rd =      60.96(Eq: 6.53)

Critical Ratio = ********(Eq: 6.57)   FAIL
========================================================================
    NORSOK_JOINT3                                           -- PAGE NO.    7


NODE NO:      2  (All units are in KN-M)
----------------------------------------
CHORD MEMBER:     1-PIP35510.0
BRACE MEMBER:     5-PIP27310.0
Angle Theta :59.04 deg.

Critical Loadcase :     1
Design Loads for Chord Memb: FX=  838.21 MZ=    434.41 MY=      0.02
Design Loads for Brace Memb: FX= -376.52 MZ=    181.45 MY=      0.03

Joint Class: 0.000k + 1.000y + 0.000x

Geometry Factor (Qb)     = 1.084  Gap Factor (Qg) =  2.385
Strength factor(Qu) for Axial                     :     23.03
Strength factor(Qu) for in-plane bending          :     14.57
Strength factor(Qu) for out-of-plane bending      :      7.47
Chord Action Factor(Qf) for axial force           :      0.33
Chord Action Factor(Qf) for in-plane bending      :      0.00
Chord Action Factor(Qf) for out-of-plane bending:        0.53
C1 = 25.00  C2 = 11.00

Axial Capacity: N,Rd  =     212.19(Eq: 6.52)
Moment Capacity: Mz,Rd =       0.00 ; My,Rd =      30.23(Eq: 6.53)

Critical Ratio = ********(Eq: 6.57)   FAIL
========================================================================
    NORSOK_JOINT3                                           -- PAGE NO.    8


NODE NO:      2  (All units are in KN-M)
```

```
----------------------------------------
CHORD MEMBER:     1-PIP35510.0
BRACE MEMBER:     7-PIP27310.0
Angle Theta :45.00 deg.

Critical Loadcase :     1
Design Loads for Chord Memb: FX=  838.21 MZ=    434.41 MY=      0.02
Design Loads for Brace Memb: FX=  125.90 MZ=    135.22 MY=      0.03

Joint Class: 0.000k + 1.000y + 0.000x

Geometry Factor (Qb)     = 1.084  Gap Factor (Qg) =  2.385
Strength factor(Qu) for Axial                  :     17.16
Strength factor(Qu) for in-plane bending       :     14.57
Strength factor(Qu) for out-of-plane bending   :      7.47
Chord Action Factor(Qf) for axial force        :      0.33
Chord Action Factor(Qf) for in-plane bending   :      0.00
Chord Action Factor(Qf) for out-of-plane bending:      0.53
C1 = 25.00  C2 = 11.00

Axial Capacity: N,Rd  =     191.77(Eq: 6.52)
Moment Capacity: Mz,Rd =       0.00 ; My,Rd =      36.65(Eq: 6.53)

Critical Ratio = ********(Eq: 6.57)   FAIL
========================================================================
    NORSOK_JOINT3                                            -- PAGE NO.    9


                      STAAD.PRO CODE CHECKING - (NORSOK-N004:Re-2)   v1.0
                      ********************************************

ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE      Noted)

MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
*    4  ST   PIP35510.0               (BRITISH SECTIONS)
                           FAIL     Eq. 6.44           3.062         1
                       234.52 C          0.22       -6729.95      144.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  S 275
     Modulus of elasticity    = 204999.75 N/mm2
     Design Strength  (py)    =275.00  N/mm2

  SECTION PROPERTIES (units - cm)
     Member Length =    365.76
     Gross Area of cross section =  109.00

                                  z-axis          y-axis
     Moment of inertia        :    16220.002       16220.002
     Plastic modulus          :     1194.726        1194.726
     Elastic modulus          :      912.261         912.261
     Radius of gyration       :       12.199          12.199
     Effective Length         :      365.760         365.760

  DESIGN PARAMETER (units - m)   N004/2004
  Height of water lavel   :        1.000
```

```
   CMZ  :   0.85      CMY  :   0.85
   KZ   :   1.00      KY   :   1.00

   SECTION CLASSIFICATION   :       Class 1

   CAPACITIES (units - kN,m)
      Slenderness ratio (kzlz/rz)           :         30.0
      Slenderness ratio (kyly/ry)           :         30.0
      Tension Capacity                      :       2606.5
      Compression Capacity                  :       2517.3
      Bending Capacity                      :        285.6
      Shear Capacity                        :        752.4
      Torsion Capacity                      :        251.9

   HYDROSTATIC PRESSURE CALCULATION (units - N,mm) - Cl.6.3.6
      Max design hydrostatic pressure, (psd) :        0.000
      Max design hoop stress, (sigma_psd))   :        0.000
     NORSOK_JOINT3                                            -- PAGE NO.   10


   CRITICAL LOAD FOR EACH CLAUSE CHECK (units - kN,m):
    CLAUSE       RATIO     LOAD    FX       VY       VZ       MZ
MY       MX
   Cl:6.3.2      0.000       0    0.000     -        -        -
-       -
   Cl:6.3.3      0.416       1  *******     -        -        -
-       -
   Cl:6.3.4      2.662       1     -        -        -     760.382
0.025     -
   Cl:6.3.5      0.365       1     -     274.768    0.003     -
-      0.011
   Cl:6.3.8.1    0.000       0    0.000     -        -       0.000
0.000     -
   Cl:6.3.8.4    2.662       1     -     274.768    0.003  760.382
0.025    0.011
   Cl:6.3.9.2    3.062       1  *******     -        -     760.382
0.025     -

==============================================================================
=======

 NODE NO:      5  (All units are in KN-M)
 ----------------------------------------
 CHORD MEMBER:     4-PIP35510.0
 BRACE MEMBER:     5-PIP27310.0
 Angle Theta :59.04 deg.

 Critical Loadcase :     1
 Design Loads for Chord Memb: FX= 1046.27 MZ=    244.61 MY=      0.02
 Design Loads for Brace Memb: FX= -376.52 MZ=    113.31 MY=      0.00

 Joint Class: 0.000k + 1.000y + 0.000x

 Geometry Factor (Qb)     = 1.084  Gap Factor (Qg) =  2.385
 Strength factor(Qu) for Axial                    :     23.03
 Strength factor(Qu) for in-plane bending         :     14.57
 Strength factor(Qu) for out-of-plane bending     :      7.47
 Chord Action Factor(Qf) for axial force          :      0.71
```

```
Chord Action Factor(Qf) for in-plane bending    :      0.57
Chord Action Factor(Qf) for out-of-plane bending:      0.80
C1 = 25.00  C2 = 11.00

Axial Capacity: N,Rd  =     458.80(Eq: 6.52)
Moment Capacity: Mz,Rd =      63.38 ; My,Rd =      45.52(Eq: 6.53)

Critical Ratio =    4.017(Eq: 6.57)   FAIL
========================================================================
** PUNCHING ** No Check for Joint      3 on Member      5
```

D13. Russian Codes

D13.A. Russian Codes - Concrete Design Per SNiP 2.03.01-84*

STAAD.Pro is capable of performing concrete design based on the Russian code СНиП 2.03.01-84*: *СТРОИТЕЛЬНЫЕ НОРМЫ И ПРАВИЛА БЕТОННЫЕ И ЖЕЛЕЗОБЕТОННЫЕ КОНСТРУКЦИИ* (SNiP 2.03.01-84* *Building Regulations: Concrete and Reinforced Concrete Construction*).

Design of members per SNiP 2.03.01-84* requires the *STAAD ECC Super Code* SELECT Code Pack.

D13.A.1 General

Russian Code SNiP 2.03.01–84* plain concrete and concrete structures is based on the method of limit states. Code SNiP 2.03.01–84* defines two groups of limit states.

Analysis according to the first group of limit states is performed to avoid the following phenomena:

- brittle, plastic or other type of failure,
- loss by structure of stable form or position,
- fatigue failure,
- failure due to the action of load actions and unfavorable environmental effects.

Analysis according to the second group of limit states is performed to avoid the following phenomena:

- excessive and long-term opening of cracks if they are allowed according to service conditions,
- excessive displacements.

Analysis of structures for the first group of limit states is performed with the use of the maximum (design) loads and actions. Analysis of structures for the second group of limit states is made in accordance with the operational (normative) loads and actions. Ratio between design and normative loads is called reliability coefficient for loads which is determined according to SNiP 2.01.07.-85 “Loads and actions”.

Reliability coefficient γ_n for destination according to SNiP 2.01.07.-85 shall be considered in determination of loads and their combinations.

STAAD.Pro makes it possible to calculate reinforcement for concrete members according to Russian Code SNiP 2.03.01-84*. Algorithms for calculation of reinforcement of concrete linear (beams, columns) and 2D (two dimensional; slabs, walls, shells) members are incorporated into the program. Not only Code SNiP 2.03.01-84* but also the “Guide for design of plain concrete and reinforced concrete structures from normal weight and lightweight concrete (to SNiP 2.03.01-84)” have been used in creation of these algorithms.

The program can automatically calculate reinforcement for beams of rectangular or T section and for columns of rectangular or circular section.

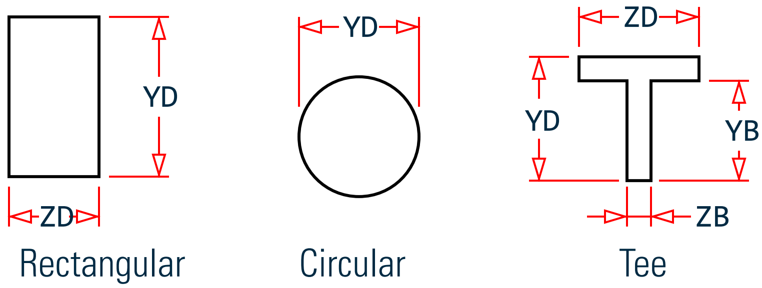


Figure 189: Section types for concrete design

Flange of T-shape beams may be situated at the top zone of the section if the angle β = 0° (`BETA 0`), or at the bottom zone of the section, if β = 180° (`BETA 180`).

D13.A.2 Design Parameters

Entry of data of cross-sections of beams and columns is made by the use of **MEMBER PROPERTIES** command, and thicknesses of 2D members are entered by **ELEMENT PROPERTY** command.

Example:

```
UNIT  MM
MEMBER  PROPERTIES
*  Columns of rectangular cross-section
1  TO  16  PRI  YD
 350.  ZD  350.
*  Columns of circular cross-section
17  TO  22  PRI  YD  350.
*  Beams of T cross-section
23  TO  40  PRI  YD  450.  ZD  550.  YB  230.  ZB  200.
UNIT  METER
ELEMENT  PROPERTY
41  TO  100 THICKNESS  0.14
101  TO  252 THICKNESS  0.16
* Flange of T beams is located at the bottom  zone of cross-section
BETA  180.  MEMB  23  TO
40
```

Commands for calculation of reinforcement are located in the input data file after the command of analysis and as a rule, after output commands to print results of calculation.

Example:

```
*  Command of analysis
PERFORM  ANALYSIS
.
.* Output command to print results of calculation  (according to user's
judgment)
.
*  Command of loading and their combinations  considered in design
```

```
LOAD  LIST  1  5  TO  9
*  Command to start reinforcement calculation  procedure
START  CONCRETE  DESIGN
CODE  RUSSIAN
.* List of parameters being used in reinforcement  calculation
.
.
BCL  20.  MEMB  17  TO   22
CL1  0.04  MEMB  1  TO   40
DD2  10.  MEMB  23  TO   40
CRA 0.036  MEMB  41  TO   252
.
.
.
* Command of beam reinforcement calculation
DESIGN  BEAM  23  TO  40
* Command of column reinforcement calculation
DESIGN  COLUMN  1  TO  22
* Command of calculation 2D elements (slabs,
 walls, shells)
DESIGN  ELEMENT  41  TO  252
* Command of interruption reinforcement calculation
END  CONCRETE  DESIGN
```

In tables 1, 2 and 3 information about parameters used for calculation of reinforcement for beams, columns and 2D (two dimensional) members is presented. Values of parameters do not depend on **UNIT** command. In the file of input data only such parameters have to be taken, the values of which differ from determined in the program.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 191: Names of parameters for Concrete design according to Russian Code -СНиП 2.03.01-84* for beams.

No.	Parameter name	Default Value	Description
1	NLT	1	Number of long-term loading case

No.	Parameter name	Default Value	Description
2	RCL	3	Class of longitudinal reinforcement: • RCL = 1, if class of reinforcement is A-I; • RCL = 2, if class of reinforcement is A-II; • RCL = 3, if class of reinforcement is A-III; • RCL = 33, if class of reinforcement is A-IIIb; • RCL = 4, if class of reinforcement is A-IV; • RCL = 5, if class of reinforcement is A-V; • RCL = 6, if class of reinforcement is A-VI; • RCL = 7, if class of reinforcement is A-VII; • RCL = 77, if class of reinforcement is K-7; • RCL = 8, if class of reinforcement is B-II; • RCL = 9, if class of reinforcement is Bp-II; • RCL = 10, if class of reinforcement is Bp-I; • RCL = 19, if class of reinforcement is K-19
2	RCL	3	Class of longitudinal reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European Grade: • 11 = S240; • 12 = S400; • 13 = S500;

No.	Parameter name	Default Value	Description
3	USM	1.	Total product of service conditions coefficients for longitudinal reinforcement **(g_s)**
4	UB2	0.9	Specific service conditions coefficient for concrete **(g_{b2})**
5	DD1	16.	Diameter of longitudinal reinforcement bars in beam tension zone
6	DD2	16.	Diameter of shear reinforcement bars for beam;
7	BCL	15.	Compression class of concrete

No.	Parameter name	Default Value	Description
7	BCL	15.	Compression Class of concrete. • 10 = B10; • 15 = B15 • 20 = B20; • 25 = B25; • 30 = B30; • 35 = B35; • 40 = B40; • 45 = B45; • 50 = B50; • 55 = B55; • 60 = B60; • 8.10 = C8/10 • 12.15 = C12/15; • 16.20 = C16/20 • 25.30 = C25/30 • 30.37 = C30/37 • 35.45 = C35/45 • 40.50 = C50/50 • 45.55 = C45/55 • 50.60 = C50/60 • 60.75 = C60/75 • 70.85 = C70/85 • 80.95 = C80/95 • 90.105 = C90/105
8	UBM	1.	Product of service conditions coefficients for concrete, except UB2 (g_b)
9	TEM	0.	Parameter of concrete hardening conditions: • TEM=0, for natural hardening conditions; • TEM=1, for steam hardening conditions
10	CL1	0.05	Distance from top/bottom fiber of beam cross section to the center of longitudinal reinforcement bar;

No.	Parameter name	Default Value	Description
11	CL2	0.05	Distance from left/right side of beam cross section to the center of longitudinal reinforcement bar
12	WST	0.4	Ultimate width of short-term crack
13	WLT	0.3	Ultimate width of long-term crack
14	SSE	0	Limit state parameter for beam design • SSE=0, if calculation of reinforcement amount must be carried out according to the requirements of load carrying capacity (the first limit state); • SSE=1, if calculation of reinforcement amount must be carried out according to the cracking requirements (the second limit state)

No.	Parameter name	Default Value	Description
15	RSH	1	Class of shear reinforcement: • RSH = 1, if class of reinforcement is A-I; • RSH = 2, if class of reinforcement is A-II; • RSH = 3, if class of reinforcement is A-III; • RSH = 33, if class of reinforcement is A-IIIb; • RSH = 4, if class of reinforcement is A-IV; • RSH = 5, if class of reinforcement is A-V; • RSH = 6, if class of reinforcement is A-VI; • RSH = 7, if class of reinforcement is A-VII; • RSH = 77, if class of reinforcement is K-7; • RSH = 8, if class of reinforcement is B-II; • RSH = 9, if class of reinforcement is Bp-II; • RSH = 10, if class of reinforcement is Bp-I; • RSH = 19, if class of reinforcement is K-19
15	RSH	1	Class of shear reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European grade: • 11 = S240; • 12 = S400; • 13 = S500;

No.	Parameter name	Default Value	Description
16	FWT	ZD	Design width of beam top flange. Use for beam design only with default value provided as ZD in member properties.
17	FWB	ZB	Design width of beam bottom flange. Use for beam design only with default value provided as ZB in member properties.
18	DEP	YD	Design depth of beam section. Use for beam design only with default value provided as YD in member properties.
19	SFA	0.	Face of support location at the start of the beam. Use for beam design only.
20	EFA	0.	Face of support location at the end of the beam. Use for beam design only.
21	NSE	13	Number of equally-spaced sections for beam design. Use for beam design only. Upper limit is equal to 20.

Table 192: Names of parameters for Concrete design according to Russian Code СНиП 2.03.01-84* for columns

No.	Parameter Name	Default Value	Description
1	NLT	1	Number of long-term loading case

No.	Parameter Name	Default Value	Description
2	RCL	3	Class of longitudinal reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European Grade: • 11 = S240; • 12 = S400; • 13 = S500;
3	USM	1.	Total product of service conditions coefficients for longitudinal reinforcement **(g_s)**
4	UB2	0.9	Specific service conditions coefficient for concrete **(g_{b2})**
5	DD1	16.	Minimum diameter of longitudinal reinforcement bars for column
6	DD2	16.	Maximum diameter of longitudinal reinforcement bars for column

No.	Parameter Name	Default Value	Description
7	BCL	15.	Compression class of concrete: • 10 = B10; • 15 = B15 • 20 = B20; • 25 = B25; • 30 = B30; • 35 = B35; • 40 = B40; • 45 = B45; • 50 = B50; • 55 = B55; • 60 = B60; • 8.10 = C8/10 • 12.15 = C12/15; • 16.20 = C16/20 • 25.30 = C25/30 • 30.37 = C30/37 • 35.45 = C35/45 • 40.50 = C50/50 • 45.55 = C45/55 • 50.60 = C50/60 • 60.75 = C60/75 • 70.85 = C70/85 • 80.95 = C80/95 • 90.105 = C90/105
8	UBM	1.	Product of service conditions coefficients for concrete, except UB2 $(\mathbf{g_b})$
9	TEM	0.	Parameter of concrete hardening conditions: • TEM=0, for natural hardening conditions; • TEM=1, for steam hardening conditions
10	CL1	0.05	Distance from edge of column cross section to the center of longitudinal reinforcement bar

No.	Parameter Name	Default Value	Description
11	ELY	1.	Column's length coefficient to evaluate slenderness effect in local Y axis
12	ELZ	1.	Column's length coefficient to evaluate slenderness effect in local Z axis
13	RSH	1.	Class of shear reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European grade: • 11 = S240; • 12 = S400; • 13 = S500;

Table 193: Names of parameters for Concrete design according to Russian Code (SNiP 2.03.01-84*) for slabs and/or walls

No.	Parameter Name	Default Value	Description
1	NLT	1	Number of long-term loading case

No.	Parameter Name	Default Value	Description
2	RCL	3	Class of longitudinal reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European Grade: • 11 = S240; • 12 = S400; • 13 = S500;
3	USM	1.	Total product of service conditions coefficients for longitudinal reinforcement **(g_s)**
4	UB2	0.9	Specific service conditions coefficient for concrete **(g_{b2})**
5	SDX	16.	Diameter of reinforcing bars located in the first local (X) direction of slab/ wall
6	SDY	16.	Diameter of reinforcing bars located in the second local (Y) direction of slab/ wall

No.	Parameter Name	Default Value	Description
7	BCL	15.	Compression class of concrete: • 10 = B10; • 15 = B15 • 20 = B20; • 25 = B25; • 30 = B30; • 35 = B35; • 40 = B40; • 45 = B45; • 50 = B50; • 55 = B55; • 60 = B60; • 8.10 = C8/10 • 12.15 = C12/15; • 16.20 = C16/20 • 25.30 = C25/30 • 30.37 = C30/37 • 35.45 = C35/45 • 40.50 = C50/50 • 45.55 = C45/55 • 50.60 = C50/60 • 60.75 = C60/75 • 70.85 = C70/85 • 80.95 = C80/95 • 90.105 = C90/105
8	UBM	1.	Product of service conditions coefficients for concrete, except UB2 **(g_b)**
9	TEM	0.	Parameter of concrete hardening conditions: • TEM=0, for natural hardening conditions; • TEM=1, for steam hardening conditions

No.	Parameter Name	Default Value	Description
10	CL	0.05	Distance from top/bottom face of slab/wall element to the center of longitudinal reinforcing bars located in first local (X) direction. (Main thickness of top/bottom concrete cover for slab/ wall element)
11	CRA	0.05	Distance from top/bottom face of slab/wall element to the center of transverse reinforcing bars located in second local (Y) direction (Secondary thickness of top/bottom concrete cover for slab/wall)
12	WST	0.4	Ultimate width of short-term crack
13	WLT	0.3	Ultimate width of long-term crack

No.	Parameter Name	Default Value	Description
14	STA	0	Parameter of limit state for slab/wall design: • STA=0, if calculation of nonsymmetrical reinforcement must be carried out according to the requirements of load carrying capacity (the first limit state); • STA=1, if calculation of symmetrical reinforcement must be carried out according to the requirements of load carrying capacity (the first limit state); • STA=2, if calculation of nonsymmetrical reinforcement must be carried according to the cracking requirements (the second limit state); • STA=3, if calculation of symmetrical reinforcement must be carried according to the cracking requirements (the second limit state)
15	SELX	0.	Design length of wall member to evaluate slenderness effect in local X axis
16	SELY	0.	Design length of wall member to evaluate slenderness effect in local Y axis

No.	Parameter Name	Default Value	Description
17	MMA	0	Design parameter of slab/ wall reinforcement: • MMA=0, if reinforcement calculation must be applied by stresses in local axis; • MMA=1, if reinforcement calculation must be applied by principal stresses
18	MMB	1	Design parameter of slab/ wall reinforcement: • MMB=0, if the effect of additional eccentricity is not taken into account; • MMB=1, if the effect of additional eccentricity is taken into account
19	RSH	1.	Class of shear reinforcement: Russian Grade: • 1 = A240; • 2 = A300; • 3 = A400; • 4 = A500; • 5 = B500; • 6 = A500SP; European grade: • 11 = S240; • 12 = S400; • 13 = S500;

D13.A.3 Beams

Reinforcement for beams of rectangular and T cross-section can be calculated. In calculation of longitudinal reinforcement bending moment about local axis Z_{loc} and torsional moments are considered, but influence of longitudinal forces and bending moments in relation to local axis Y_{loc} is ignored. In calculation of transverse reinforcement shear forces parallel to local axis Y_{loc} a nd torsional moments are taken into account.

Reinforcement for beams can be calculated either from conditions of strength or from conditions of open crack width limitation (see parameter SSE).

Parameters SFA and EFA are considered only in calculation of transverse reinforcement.

In general case calculation of reinforcement for beams is carried out two times – according to strength conditions and according to conditions of open crack width limitation. In reinforcement calculations from conditions of strength design values of load have to be taken and in calculations from conditions of crack width limitation – characteristic (normative) load values are used. Both calculations can be carried out in one session with the use multiple analysis possibility of the program STAAD.Pro.

In most cases calculation of reinforcement is carried out with account only of a part of loadings. In such cases command LOAD LIST is used, in which numbers of loads considered in calculation are indicated. Number of permanent and long-term loads equal to parameter NLT must be included into the list of considered loads.

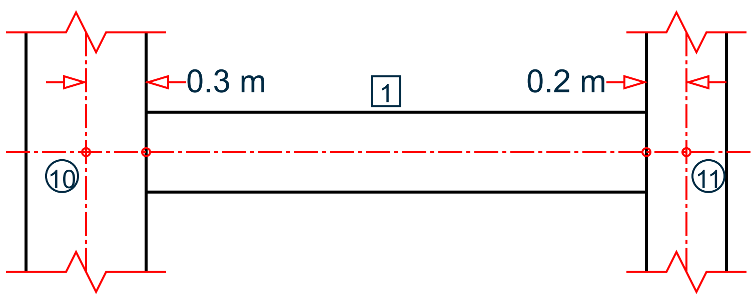
It has to be noted, that values of parameters DD1 and DD2 have influence not only on the width of opened crack but also in some cases, on design and normative reinforcement resistances.

Parameter BCL can be equal to any value of concrete compression strength class given in SNiP 2.03.01-84* and to any intermediate value as well.

It should be remembered, that accuracy of results of calculation of transverse reinforcement increases with the value of parameter NSE.

Parameters SFA and EFA are considered only in calculations of transverse reinforcement. Beam 1 is shown in Figure 2 with rigid intervals the lengths of which are: at the start of the beam 0.3m and at the end – 0.2m. In modeling of the beam the following command can be used.

```
MEMBER  OFFSET
1  START 0.3 0 0
1  END -0.2 0 0
```



When command MEMBER OFFSET is used forces corresponding to the beam the length of which is equal to the distance between points a and b are calculated and then used in calculation of reinforcement. In such case it is necessary to take into account default values of parameters SFA and EFA equal to zero.

When command MEMBER OFFSET is not used forces corresponding to the beam the length of which is equal to the distance between points *10* and *11* are calculated and then used in calculation of reinforcement. In this case it is necessary to consider values of parameters SFA=0.3 and >EFA=0,2 in reinforcement calculation.

In both cases calculated quantity of transverse reinforcement will be the same. Calculated quantity of longitudinal reinforcement in the second case will be greater.

For beam the following output is generated:

- beam number;
- method of calculation (according to conditions of strength or limitations of opened crack width);

- length and cross-sectional dimensions;
- distance from resultant of forces acting in bottom/top reinforcement to bottom/top edge of the section;
- distance from the side edge of cross-section of the beam web to the centroid of longitudinal bars located at this edge;
- concrete class;
- class of longitudinal and transverse reinforcement;
- assumed in calculations bar diameters of longitudinal and transverse reinforcement;
- calculation results of longitudinal and transverse reinforcement (in two tables).

In nine columns of the first table the following results are presented:

Table 194: Beam design output 1

Result	Description
Section	distance of the section from the “start” of the beam, *мм*
As-	cross-sectional area of longitudinal reinforcement in the bottom zone of cross-section of the beam, if angle *BETA*=0°, or in the top zone, if *BETA*=180° , *sq.cm*
As+	cross-sectional area of longitudinal reinforcement in the top zone of cross-section of the beam , if angle *BETA*=0°, or in the top zone, if *BETA*=180° , *sq.cm*
Moments (-/+)	values of bending moments, determining cross-sectional areas of longitudinal reinforcement *As*- and *As*+ , *kNm*
Load. N. (-/+)	numbers of loading versions, determining cross-sectional areas of longitudinal reinforcement
Acrc1	short-term opened crack width*, *mm*
Acrc2	long-term opened crack width*, *mm*

* Opened crack width is presented only in the case when calculation is performed according to conditions limiting opened crack width.

In ten columns of second table the following results are presented:

Table 195: Beam design output 2

Result	Description
Section	distance of the section from the “start” of the beam, *mm*
Qsw	intensity of transverse reinforcement, *kN/m*

Result	Description
Asw	cross-sectional area of transverse bars, *sq.cm*, if their step is 10, 15, 20, 25 or 30 *cm*
Q	value of shear force parallel to the local axis, *kN*
T	value of torsional moment, *kNm*
Load N.	number of loading version, determining intensity of transverse reinforcement

D13.A.4 Columns

Reinforcement for columns of rectangular or circular cross-section can be calculated. Flexibility of columns can be evaluated in two ways. In the case of usual analysis (command `PERFORM ANALYSIS`) flexibility is assessed by parameters `ELY` and `ELZ`, values of which should conform with recommendation of the Code SNiP 2.03.01-84*. If `P-DELTA` (analysis according to deformed diagram) or `NONLINEAR` (nonlinear geometry) analysis is performed, values of parameters `ELY` and `ELZ` should be close to zero, for example `ELY = ELZ`=0.01.

Longitudinal reinforcement for columns is calculated only from condition of strength. Longitudinal forces and bending moments in relation to local axes Y_{loc} and Z_{loc} are taken into account in longitudinal reinforcement calculations.

For rectangular columns the following output is generated:

- column number;
- column length and cross-sectional dimensions;
- distance of centroid of each longitudinal bar from the nearest edge of the cross-section;
- concrete class;
- longitudinal reinforcement class;
- range of longitudinal reinforcement bar diameters assumed in calculation;
- diameter of longitudinal reinforcement bars obtained in calculation;
- total quantity of longitudinal bars;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Y_{loc} ;
- quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Z_{loc} .

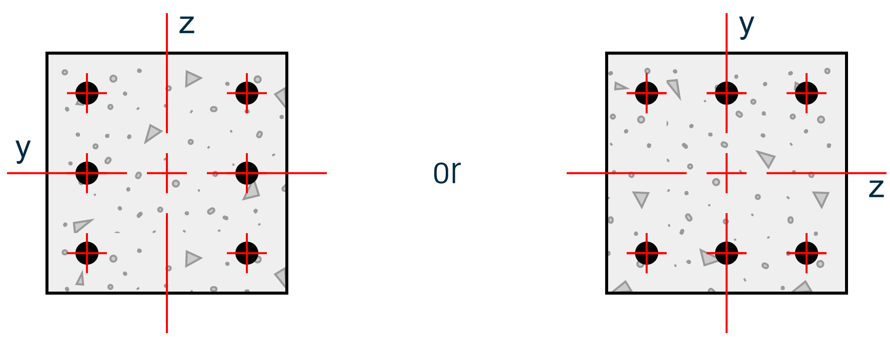
In nine columns of the table under the heading LONGITUDINAL REINFORCEMENT the following output is presented:

Table 196: Column design output 1

Result	
Section	distance of the section from the “start” of the column, *mm*
Astot	total cross-sectional area of longitudinal reinforcement, sq.cm

Result	
Asy	cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Y_{loc} , sq.cm
Asz	cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Z_{loc} , sq.cm
Percent	reinforcement percentage in the section
Nx, Mz, My	respective values of longitudinal force and bending moments in relation to the local axes Z_{loc} and Y_{loc} , determining cross-sectional area of longitudinal reinforcement
Load.N.	number of loading version, determining cross-sectional area of longitudinal reinforcement

Diameter of longitudinal reinforcement bars, total quantity of longitudinal bars as well as quantity of longitudinal bars at each edge of the section obtained from calculation should be considered as recommendation. In this case arrangement of reinforcement in the section depends on the orientation of the local axes and is as follows:



Calculated values of reinforcement cross-sectional areas are presented in the table and they may differ from recommended on the lower side.

When it is not possible according to detailing provisions to arrange in the column longitudinal reinforcement determined from calculation additional message is derived.

For columns of circular section the following output is generated:

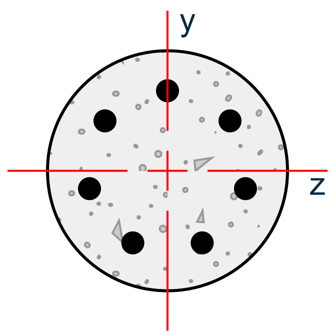
- column number;
- column length and diameter of cross-section;
- distance of centroid of each longitudinal bar to the edge of cross-section;
- longitudinal reinforcement class;
- assumed in calculation range of diameters of longitudinal reinforcement bars;
- diameter of longitudinal reinforcement bars obtained from calculation;
- quantity of longitudinal bars.

In seven columns of the table under the heading LONGITUDINAL REINFORCEMENT the following results are presented:

Section	distance of the section from the "start" of the column, *mm*
Astot	total cross-sectional area of longitudinal reinforcement, sq.cm
Per cent	percentage of longitudinal reinforcement
Nx, Mz, My	respective values of longitudinal force and bending moments in relation to local axis Z_{loc} and Y_{loc}, determining cross-sectional area of longitudinal reinforcement
Load. N.	number of loading version, determining cross-sectional area of longitudinal reinforcement

Diameter of longitudinal reinforcement bars, total quantity of longitudinal bars as well as quantity of longitudinal bars at each edge of the section should be considered as recommendation.

Arrangement of reinforcement in section in this case is shown below:



Calculated cross-sectional areas of reinforcement presented in the table may differ from recommended on the lower side.

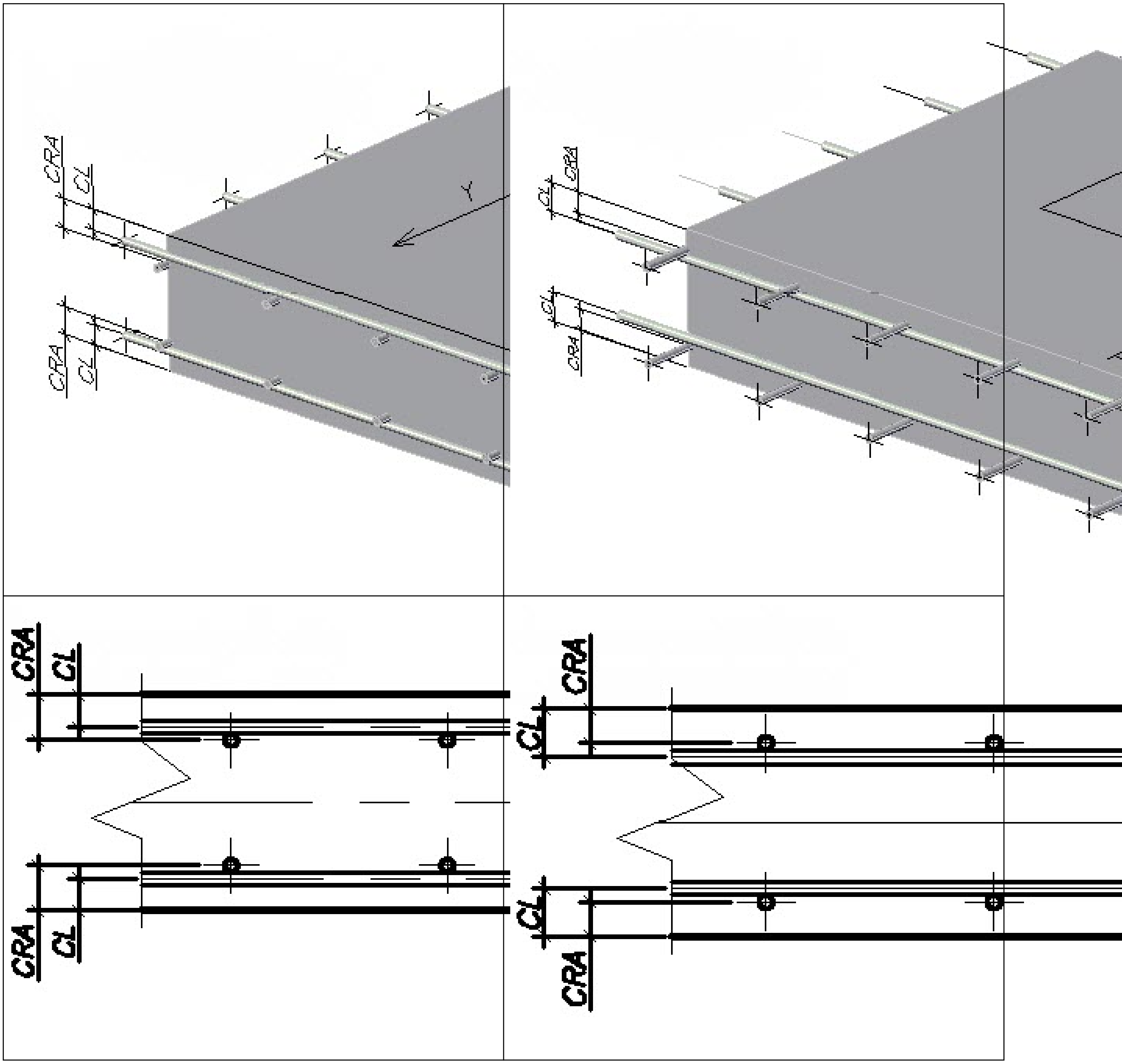
When according to detailing provisions it is not possible to arrange in the column longitudinal reinforcement obtained from calculation additional message is derived.

D13.A.5 Two Dimensional Elements (slabs, walls, shells)

In general case calculation of reinforcement for 2D members is carried out two times – according to conditions of strength and conditions of limiting opened width of cracks. If reinforcement is calculated according to conditions of strength, design values of loads have to be used, and for conditions of limiting crack width – characteristic (normative) loads are employed. Both calculations can be made in one session by using multiple analyses.

Symmetric or nonsymmetrical reinforcement of 2D members is calculated according to conditions of strength or according to conditions of limiting opened crack width (see for example **STA**).

In reinforcement calculation for 2D members it is necessary to pay attention to arrangement of local axes of member and direction of reinforcement (see for example **CL** and **CRA**).



An example of output of calculation results is presented bellow.

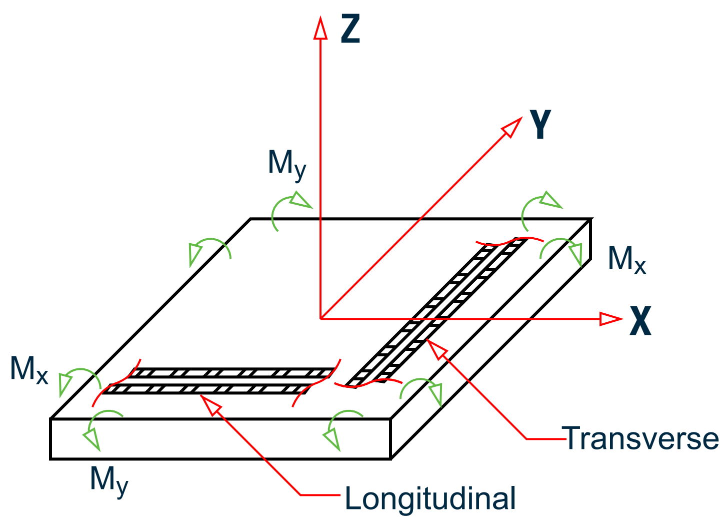
```
                    SLAB/WALL DESIGN RESULTS
    (by stresses in local axes for limitation of crack width)
--------------------------------------------------------------------
Element  Asx     Mx      Nx   Load. N.  Asy       My     Ny   Load N.
       sq.cm/m  kNm/m   kN/m   (X)    sq.cm/m  kNm/m   kN/m   (Y)
--------------------------------------------------------------------
60  TOP  0.00   - 4.9   0.0     1      0.00    - 4.5   0.0     1
    BOT  3.53   - 9.9   0.0     3      3.46    - 8.9   0.0     3
```

```
61  TOP  0.00   - 5.3   0.0     1       0.00    - 4.7   0.0       1
    BOT  3.87  - 10.7   0.0     3       3.65    - 9.4   0.0       3
62  TOP  0.00   - 5.6   0.0     1       0.00    - 4.8   0.0       1
    BOT  4.10  - 11.2   0.0     3       3.77    - 9.6   0.0       3
```

Where:

Table 197: Slab design output

Result	Description
Element	number of finite element, “TOP” - top zone of member, “BOT” - bottom zone of member (“top” zone of member is determined by positive direction of local axis, Z)
Asx	intensity of reinforcing in the longitudinal direction (parallel to the local axis, X), *sq.cm/m*
Mx	distributed bending moment in respect to the local axis, Y, *kNm/m*
Nx	distributed longitudinal force directed parallel to the axis *X*, *kNm/m*
Load N.(X)	number of loading version, determining intensity of reinforcing in the longitudinal direction
Asy	intensity of reinforcing in the transverse direction (parallel to the local axis Y), *sq.cm/m*
My	distributed bending moment in respect to the local axis X *kNm/m*
Ny	distributed longitudinal force directed parallel to the local axis *Y* *kN/m*
Load N.(Y)	number of loading version, determining intensity of reinforcing in the transverse direction



D13.B. Russian Codes - Steel Design Per SNiP 2.23-81* (Edition 1990)

STAAD.Pro is capable of performing steel design based on the Russian code СНиП II-23-81* Часть II *Нормы проектирования Стальные конструкции* (SNiP 2.23-81* Part II *Design Standards for Steel Construction*).

In STAAD.Pro V8i (SELECTseries 5) or later, design of members per SNiP 2.23-81* requires the *STAAD ECC. Super Code* SELECT Code Pack.

D13.B.1 General

Design Code SNiP Steel Structures –as is the case in the majority of modern codes– is based on the method of limit states. The following groups of limit states are defined in the Code.

- The first group is concerned with losses of general shape and stability, failure, qualitative changes in configuration of structure. Appearance of non-allowable residual deformations, displacements, yielding of materials or opening of cracks.
- The second group is concerned with states of structures making worse normal their service or reducing durability due to not allowable deflections, deviations, settlements, vibrations, etc.

Analysis of structures for the first limit state is performed using the maximum (design) loads and actions, which can cause failure of structures.

Analysis of structures for the second limit state is performed using service (normative) loads and actions. Relation between design and normative loads is referred to as coefficient of load reliability, which is defined in SNiP 2.01.07.- 85 “Loads and Actions”.

Coefficient of reliability for destination GAMA n according to SNiP 2.01.07.- 85 shall be taken in to account determining loads or their combinations.

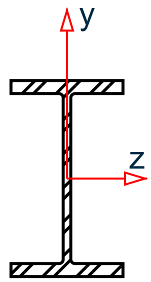
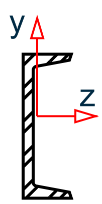
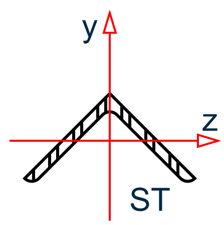
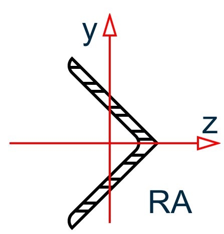
In this version of the program only members from rolled, tube and roll-formed assortment sections and also from compound such as double angles of T-type sections, double channels are presented. Design of other members of compound section will be presented in other versions of the program.

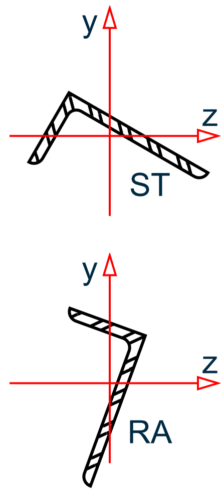
Economy of selected section is indicated by ratio (`RATIO`) $\sigma/R_y y_c$ presented in calculation results. A section is economical when said ratio equals to 0,9 – 0,95.

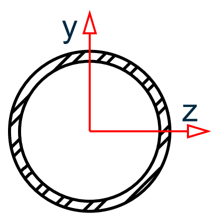
D13.B.2 Built-in Russian Steel Section Library

Typical sections of members being checked and selected according to SNiP 2.01.07.- 81* are presented in the following tables.

Table 198: Typical Sections for Russian Steel Design

Section	Section Type	Designation form
I-beam (GOST 8239-89)		`ST I12`
Regular I-beam (GOST 26020-83)		`ST B1-10`
Broad-flanged I-beam (GOST 26020-83)		`ST SH1-23`
Column I-beam (GOST 26020-83)		`ST K1-20`
Channel (GOST 8240-89)		`ST C14`
Equal legs angle (GOST 8509-89)		`ST L100x100x7` `RA L100x100x7`

Section	Section Type	Designation form
Unequal legs angle (GOST 8510-89)		`ST L125x80x10` `RA L125x80x10`

Pipes (GOST 10705 and GOST 10706; welded and for gas piping)		`ST PIP102x5.5`
		or
		`ST PIPE OD 0.102 ID 0.055`

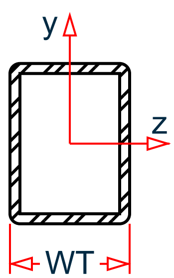
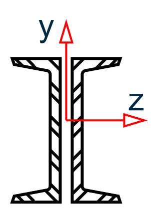
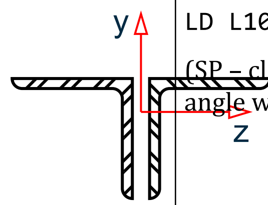
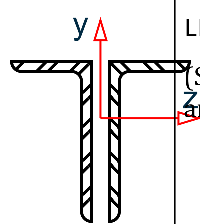
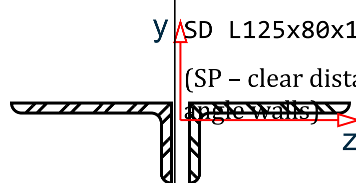
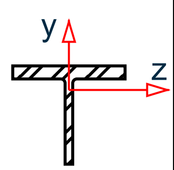
Roll-formed square and rectangular tubes (GOST 8639 and GOST 8645)		`ST TUB160x120x3`
		or
		`ST TUBE TH 0.003 WT 0.12 DT 0.16`

Table 199: Compound Sections for Russian Steel Design

Section	Section Type	Designation form
Double channels		`D C14 SP 0.01` (SP – clear distance between channel walls)

Section	Section Type	Designation form
Double equal legs angles		`LD L100x100x7 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with long legs back to back		`LD L125x80x10 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with short legs back to back		`SD L125x80x10 SP 0.01` (SP – clear distance between angle walls)

Tee with flange at the top
Note: Flange of Tee beam is at the top part of cross-section if beta angle = 0°, or at the bottom part if beta angle = 180°.
		`T I12`
		`T B1-10`
		`T SH1-23`
		`T K1-20`

For entry of cross-sectional dimensions command `MEMBER PROPERTIES RUSSIAN` is used.

Example

```
UNITS  METER
MEMBER  PROPERTY RUSSIAN
* I-beam
1 TO 6 TABLE ST  B1-10
* Channel
7  TO 11 TABLE ST C14
* Unequal  legs angle
12  TO 30 TABLE RA L125x80x10
* Round assortment pipe
31 TO 46 TABLE  ST PIP102x5.5
* round pipe of cross-sectional  dimensions defined by client
47  TO 60 TABLE ST PIPE OD 0.102 ID 0.055
* Square tube from assortment
61 TO 68 TABLE  ST TUB120x120x3
* Rectangular tube of cross-sectional dimension defined by client
69  TO 95 TABLE ST TUBE TH 0.003 WT 0.12 DT 0.16
```

```
* Double channel  (distance between walls 10 мм)
96  TO 103 TABLE D C14 SP 0.01
* Double unequal legs angles with short legs back-to-back (distance  between
walls 10 мм)
104  TO 105 TABLE SD L125x80x10 SP 0.01
* member of Tee section
106  TO 126 TABLE T SH1-23
* Flange of T-beams at the bottom of cross-section
BETA  180. MEMB 116 TO 126
* Orientation of the local angle axes in relation to the global axes of  the
structure
BETA  RANGLE MEMB 12 TO 30
```

D13.B.3 Member Capacities

Algorithms for selection and review of sections for steel members according to assortments and databases of the main rolled steel producers from given countries and according to international standards as well are included in STAAD.Pro program. In this program version only assortment sections can be utilized.

Example

```
* Command of analysis
PERFORM  ANALYSIS
* Command of loadings and their combinations considered in design
LOAD  LIST 1 5 TO 9
* Command to start design according to Russian Code
PARAMETER
CODE  RUSSIAN
* List of parameters used in checking and selecting
BEAM  1. ALL
```

Obligatory parameter

```
LY  4. MEMB 1 TO 4
LZ  4. MEM 1 TO 4
MAIN  1. ALL
SGR  3. ALL
SBLT  0 ALL
* Parameter of output amount of information on calculation results
TRACK  2. ALL
.
* Command to start section check procedure
CHECK  CODE ALL
* Command to start section selection procedure
SELECT  ALL
.
* Command of output to print content of assortment tables
PRINT  ENTIRE TABLE
* Command of output to print summary of steel according to sections
STEEL  TAKE OFF
* Command of output to print summary of steel according to members and
sections
STEEL  MEMBER TAKE OFF
```

D13.B.3.1 Axial Tension Members

Stress in a section of axial tension member shall not exceed design strength R_y of selected steel multiplied by coefficient of service conditions γ_c (KY and KZ), table 6 of SNiP 2.01.07.- 81*. Slenderness of tension member (CMM) shall not exceed slenderness limit indicated in table 20 of SNiP 2.01.07.- 81* (default value $\lambda_u = 200$, but another value can be defined). Net section factor (ratio A_{net}/A_{gross} (NSF)) is used for tension member to allow for reduction of design cross-section area.

D13.B.3.2 Axial Compression Members

All axial compression members are calculated as long bars, i.e., with allowance for slenderness ($\lambda = l_0/i_{min}$). The calculation is performed in accordance with the clause 5.3 of SNiP 2.01.07.- 81*, buckling coefficient φ is determined by formula 8-10. Effective bar lengths (within and out of plane) taking in to account role and location of the bar in the structure, as well as fixation of ends ($l_0 = \mu l$), are determined according to requirements of chapter 6 or addition 6 to SNiP 2.01.07.- 81* and are set by specification of members. Slenderness of compression members (CMN) shall not exceed limit values given in table 19 of SNiP 2.01.07.- 81*. Value of coefficient α being used in table 19 is taken within limits from 0,5 to 1,0. Limit slenderness value depends on stress acting in the member, section area, buckling coefficient and design resistance of steel.

Since slenderness can be different in various planes the greatest slenderness is assumed in calculations.

D13.B.3.3 Flexural members

Members subjected to the action of bending moments and shear forces are called flexural members.

Calculation of flexural members consists of verification of strength, stability and deflection.

Normal and tangential stresses are verified by strength calculation of members. Normal stresses are calculated in the outermost section fibres. Tangential stresses are verified in the neutral axis zone of the same section. If normal stresses do not exceed design steel strength and tangential stresses do not exceed design value of steel shear strength $R_s\gamma_s$ then according to clause 5.14 of SNiP 2.01.07.- 81* principal stresses are checked.

General stability of member subjected to bending in one plane are calculated in accordance with clause 5.15 of SNiP 2.01.07.- 81*, and subjected to bending in two planes – in accordance with “Guide to design of steel structures” (to SNiP 2.01.07.- 81*). Coefficient φ_b value is determined according to appendix 7 of SNiP 2.01.07.- 81*. Additional data about load (concentrated or distributed), numbers of bracing restrains of compression flanges, location of applied load are required. For closed sections it is assumed that coefficient $\varphi_b = 1.0$.

Simply supported (non-continuous) beams can be calculated in elastic as well as in elastic-plastic state according to requirements of clause 5.18 of SNiP 2.01.07.- 81*. Calculation can be selected by specification of structure in input data.

Stiffness of flexural members is verified comparing input value of deflection limit (through parameter DFF) with maximum displacement of a section of flexural member allowing for load reliability coefficient, which is specified, in input data. Limit values of deflection are determined in accordance with SNiP 2.01.07.- 85 “Loads and Actions. Addition chapter 10. Deflections and displacements”. Verification of deflection is performed only in the case of review (CHECK) problem.

D13.B.3.4 Eccentric Compression/Tension Members

Eccentric compression or tension members are subjected to simultaneous action of axial force and bending moment. Bending moment appears due to eccentric application of longitudinal force or due to transverse force.

Stress in eccentric compression/tension members is obtained as a sum of stresses due to axial force and bending.

Following the requirements of clause 5.25 of SNiP 2.01.07.- 81* resistance of eccentric compression/tension member taking into consideration condition $R_y < 530\ MPa$, $\tau < 0.5R_s$ and $N/(A_n R_y) > 0.1$ is calculated by formula 49, and in other cases-by formula 50. Calculations of stability verification are performed according to requirements of clauses 5.27, 5.30, 5.32 or 5.34.

Calculation for strength of eccentric tension members is made according to formula 50 of SNiP 2.01.07.- 81*.

When reduced relative eccentricity $m_{ef} > 20$ eccentric compression members are calculated as flexural members (N = 0), when $m_{ef} < 20$ strength by formula 49 is not verified (clause 5.24).

D13.B.4 Design Parameters

Information on parameters, data used for check and selection of sections in design of steel structures according to Russian Code is presented in the following table.

In this version of calculation according to requirements of SNiP 2.01.07.- 81* there is common database of equal legs angles and unequal legs angles, therefore solution of section selection problem may give equal legs angle as well as unequal legs angle irrespective of set at the beginning. The same is and with rectangular and square tubes.

Values of parameters do not depend on command `UNIT`. Only these values of parameters, which differ from, defined in the program need to be included in the input data file.

Review of sections (command `CHECK`) can be performed according to the first and the second group of limit states. Selection of section (command `SELECT`) can be performed only according to the first group of limit states with subsequent recalculation and verification of selected section with allowance for deflection.

Calculation for the first group of limit states involves selection of members according to strength and stability. Parameters `CMN` and `CMM` give opportunity to set slenderness limit for compression and tension members respectively for their stability calculation, or refuse consideration of slenderness by setting default parameters. In this case selection of sections will be performed with consideration only of strength check.

Check for deflection performed by setting parameter `DFF` (maximum allowable relative deflection value) different from set in the program.

In the case of application of steel not defined by SNiP and/or GOST it is necessary to set their design strength by parameters `UNL` and `PY`.

In determination of steel parameters `SBLT` and `MAIN` shall be approved (see Table 15B.4).

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 200: Parameters for Steel design according to Russian Code (SNiP II – 23 – 81*, edition 1990)

Parameter Name	Default Value	Description
CODE	-	Must be specified as `RUSSIAN 1990` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	1	Member design parameter: • BEAM = 0, Design members for forces at their ends or at the sections defined by SECTION command; • BEAM = 1, Calculate the major axis moment Mz at 13 points along the beam and design beam at the location of maximum Mz; • BEAM = 2, Same as BEAM=1, but additional checks are carried out at beam ends and at critical inter mediate section; • BEAM = 3, Calculate forces at 13 points and perform design checks at all locations including the ends
CB	1	Place of loading on beam: • CB = 1, for loading on top flange; • CB = 2, for loading on bottom flange
CMM	0	Slenderness limit value for tension members: • CMM = 0, if slenderness is suppressed; • CMM = 2, if ultimate slenderness value is "150"; • CMM = 2, if ultimate slenderness value is "200"; • CMM = 3, if ultimate slenderness value is "250"; • CMM = 4, if ultimate slenderness value is "300"; • CMM = 5, if ultimate slenderness value is "350"; • CMM = 6, if ultimate slenderness value is "400 Set slenderness limit value not equal to "0" for design with evaluation of buckling effect

Parameter Name	Default Value	Description
CMN	0	Slenderness limit value for compression members: • CMN = 0, if slenderness is suppressed • CMN = 1, if slenderness limit value is 180-60a • CMN = 2, if slenderness limit value is 120 • CMN = 3, if slenderness limit value is 210-60a • CMN = 4, if slenderness limit value is 220-40a • CMN = 5, if slenderness limit value is 220 • CMN = 6, if slenderness limit value is 180-60a • CMN = 7, if slenderness limit value is 210-60a • CMN = 8, if slenderness limit value is 200 • CMN = 9, if slenderness limit value is 150 Set slenderness limit value not equal to "0" for design with evaluation of buckling effect
DFF	0.	Allowable limit of relative local deflection (Member length/ Deflection Ratio): Default value 0 is valid if design is applied without deflection limitation. Set for deflection check only
DMAX [m]	1.	Maximum allowable section depth
DMIN [m]	0.	Minimum allowable section depth
GAMC1	1.0	Specific service condition coefficient for buckling design

Parameter Name	Default Value	Description
GAMC2	1.0	Specific service condition coefficient for strength design
GMF	1.0	Ratio between design and characteristic load value
KY	1.0	Coefficient of effective length in respect to local axis Y (in plane XZ)
KZ	1.0	Coefficient of effective length in respect to local axis Z (in plane XY)
LEG	4	Type and position of loading on beam: • LEG = 1, for loading concentrated in the middle span; • LEG = 2, for loading concentrated in the quarter of the span; • LEG = 3, for loading concentrated at the end of bracket; • LEG = 4, for loading uniformly distributed on beam; • LEG = 5, for loading uniformly distributed on bracket
LY [m]	Member length	Effective length in respect to local axis Y (in plane XZ) Default is selected member's length
LZ [m]	Member length	Effective length in respect to local axis Z (in plane XY) Default is selected member's length

Parameter Name	Default Value	Description
MAIN	1	Standard of steel grade (GOST): • MAIN = 1, if Standard of steel grade is GOST27772-88; • MAIN = 2, if Standard of steel grade is GOST10705-80; • MAIN = 3, if Standard of steel grade is GOST10706-76; • MAIN = 4, if Standard of steel grade is GOST8731-87; • MAIN = 5, if Standard of steel grade is TY14-3-567-76
NSF	1.0	Net section factor for tension members or web section area weakening factor for bending members
PY [MPa]	0	Design steel strength (yield strength): If parameters MAIN according to Standard of steel grade (GOST) and by SGR according to Steel grade (STAL) are not defined
RATIO	1.0	Permissible ratio of the actual capacities.
SBLT	0	Number of lateral bracing restraints along the span: • SBLT = 0, if beam not fixed; • SBLT = 1, one restraint in the middle of the span; • SBLT = 2, 3, etc. number of uniformly spaced lateral supports along the span
SGR	1	Steel grade (STAL). Refer to Table 12B.4 below.

Parameter Name	Default Value	Description
TB	0	Indication of elastic or elastic-plastic calculation: • TB = 0, for elastic calculation • TB = 1, for elastic-plastic calculation Set for members under bending or non-axial compression/tension only.
TRACK	0	Output parameter: • TRACK = 0, for suppressed output information; • TRACK = 1, for extended output information; • TRACK = 2, for advanced output information
UNL [MPa]	0	Design steel strength (ultimate strength): If parameters MAIN according to Standard of steel grade (GOST) and by SGR according to Steel grade (STAL) are not defined

Table 201: Steel types for design of steel structures according to SNiP 2.01.07.-81* (table 51 and 51a)

SGR Value	Steel	Parameter MAIN	GOST	For members*
1	C235	1	GOST 27772-88	GT, F
2	C245	1	“	GT, F
3	C255	1	“	GT, F
4	C275	1	“	GT, F
5	C285	1	“	GT, F
6	C345	1	“	GT, F
7	C345K	1	“	GT, F
8	C375	1	“	GT, F
9	C390	1	“	F

SGR Value	Steel	Parameter MAIN	GOST	For members*
10	C390K	1	“	F
11	C440	1	“	F
12	C590	1	“	F
13	C590K	1	“	F
14	BSt3kp	2	GOST 10705-80*	Tube
15	BSt3ps	2 3	GOST 10705-80* GOST 10706-76*	Tube
16	BSt3sp	2 3	GOST 10705-80* GOST 10706-76*	Tube
17	20	4	GOST 8731-87	Tube
18	16G2AF	5	TY 14-3-567-76	Tube

*GT – members from sheet and roll-formed tubes

F – rolled section steel

D13.B.5 Member Selection and Code Check

Both code checking and member selection options are available in SNiP 2.23-81*.

Output of selection and check results are given in suppressed, extended and advanced forms. Form of output results depends on value of parameter TRACK.

Results are presented in tables. Three versions of output results are possible: suppressed – results according the critical strength condition (TRACK=0), extended - results according to all check conditions (TRACK=1) and advanced – complete information on results of member design (TRACK=2).

In tables of results common data for all TRACKs are indicated:

(TRACK=2)

In tables of results common data for all TRACKs are indicated:

number of member;
type and number of cross-section;
result obtained (ACCEPTED – requirements are met, FAILURE – are not met);
abbreviated name of normative document (code, standard) (SNiP);
number of check clause;
safety of strength (ratio between design and normative values);
number of the most unfavorable loading;

value of longitudinal force acting in the member with subscript indicating its direction ("C" – compression, "P" – tension);

bending moments in relation to local member axes Z and Y;

distance to section, in which the most unfavorable combination of forces acts.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

Note: The output for the Russian code uses the axes system from STAAD.Pro, except where details of the equations from the code are printed, in which case the axes system is as defined in the code.

D13.B.5.1 Example of TRACK 0 output

In suppressed form (`TRACK 0`) results are presented according to the critical check for given member with indication of SNiP clause number, according to which strength safety of the member is minimum.

```
=========================================================================
MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
           SECTION NO.      FX            MZ            MY      LOCATION
=========================================================================
   1         I60            PASS      SNiP- 5.18       0.68       1
                        0.000E+00     -4.650E+02    0.000E+00   3.000E+00
```

D13.B.5.2 Example of TRACK 1 output

In extended form (`TRACK 1`) results are presented on the basis of all required by SNiP checks for given stress state.

```
=========================================================================
MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
           SECTION NO.      FX            MZ            MY      LOCATION
=========================================================================
   1         I60            PASS      SNiP- 5.18       0.68       1
                        0.000E+00     -4.650E+02    0.000E+00   3.000E+00
   1         I60            PASS      SNiP- DISPL      0.36       1
                        0.000E+00     -4.650E+02    0.000E+00   3.000E+00
```

D13.B.5.3 Example of a TRACK 2 output

In advanced form (`TRACK=2`) in addition to tabled results supplementary information is presented.

Material characteristics:

Steel;

Design resistance;

Elasticity modulus;

Section characteristics:

Length of member;

Section area;

Net area;

Inertia moment (second moment of area) (I);

Section modulus (W);
First moment of area (S);
Radius of gyration;
Effective length;
Slenderness;
Results are presented in two columns, Z and Y respectively.
Design forces:
Longitudinal force;
Moments;
Shear force.

Signs “+” and “-“ indicate direction of acting longitudinal force, bending moments and shear forces in accordance with sign rules assumed in program STAAD.

Check results in advanced form are presented with values of intermediate parameters by formulas in analytical and numerical expression with indication of SNiP clause.

```
========================================================================
MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
           SECTION NO.      FX           MZ             MY      LOCATION
========================================================================
   1         I60            PASS      SNiP- 5.18       0.68       1
                       0.000E+00     -4.650E+02    0.000E+00   3.000E+00
   1         I60            PASS      SNiP- DISPL      0.36       1
                       0.000E+00     -4.650E+02    0.000E+00   3.000E+00

MATERIAL DATA
   Steel                         =C245
   Modulus of elasticity         = 206.E+06 KPA
   Design Strength (Ry)          = 240.E+03 KPA

SECTION PROPERTIES (units - m)
   Member Length                 = 6.00E+00
   Gross Area                    = 1.38E-02
   Net Area                      = 1.38E-02
                                      z-axis      y-axis
   Moment of inertia (I)         :   768.E-06    173.E-07
   Section modulus (W)           :   256.E-05    182.E-06
   First moment of area (S)      :   149.E-05    156.E-06
   Radius of gyration (i)        :   236.E-03    354.E-04
   Effective Length              :   600.E-02    600.E-02
   Slenderness                   :   0.00E+00    0.00E+00

DESIGN DATA (units -kN,m)SNiP II-23-81*/1998
   Axial force                   :   0.00E+00
                                      z-axis      y-axis
   Moments                       :  -465.E+00    0.00E+00
   Shear force                   :   0.00E+00    500.E-02

CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
       F.(39)  M/(C1*Wmin)=-465.0E+00/ 1.12E+00* 2.56E-03= 162.1E+03
       F.(41)  Q/(H*T)= 500.0E-02/ 6.00E-01* 1.20E-02= 694.E+00
       RY*GAMAC= 240.0E+03
       ACTUAL SECTION DISPLACEMENT = 1.094E-02 M
       MAXIMUM MEMBER DEFLECTION = 1.094E-02 M  Loading No.    1
       ULTIMATE ALLOWABLE DEFLECTION VALUE = 3.000E-02 M
```

Conventional notations assumed in presentation of results: “+”, “-“, “/”, “*”,”**”, “SQRT”, their respective meanings (i.e., addition, subtraction, division, multiplication, raising to the second power (squared), and square root). Conventional notations of stresses, coefficients and characteristics of steel resistance comply with accepted in the SNiP standard. Only Greek letters are changed by their names (e.g., , γ_c-GAMAC; α-ALPHA; β-BETA, η-ETA, φ-PHI, etc.).

D13.C. Russian Codes - Steel Design Per SP 16.13330.2011

STAAD.Pro is capable of performing steel design based on the Russian code Сп 16.13330.2011 *стальные конструкции* (SP 16.13330.2011 *Steel Structures*).

Note: This code supersedes SNiP II-23-81*.

D13.C.1 General

Design Code SP *Steel Structures* –as is the case in the majority of modern codes– is based on the method of limit states. The following groups of limit states are defined in the Code.

- The first group concerns losses of general shape and stability, failure, and qualitative changes in configuration of the structure (i.e., ultimate limit states). Appearance of non-allowable residual deformations, displacements, yielding of materials or opening of cracks.

  Analysis of structures for the first limit state is performed using the maximum (design) loads and actions, which can cause failure of structures.
- The second group concerns states of the structure which worsen their service or reduce durability due to exceeding allowable deflections, deviations, settlements, vibrations, etc. (i.e., service conditions)

  Analysis of structures for the second limit state is performed using service (normative) loads and actions. Relation between design and normative loads is referred to as coefficient of load reliability, which is defined in SNiP 2.01.07.- 85 “Loads and Actions”.

The coefficient of reliability for destination GAMA n according to SP 20.13330.2011 shall be taken in to account determining loads or their combinations.

According to the European standards, the strength of steel is represented by the characteristic value. To obtain the design value, the steel reliability coefficient `GAMM` is used. The default value of `GAMM` is 1.0.

Note: If there are doubts about the conformance of the European steel type and the standard, it is necessary that the steel type designation used by the `ENSGR` parameter be set the same as the standard selected in the `ENMAIN` parameter. If the chosen steel type is *not* present in the chosen standard, then the program exits with the error code `MEMBER NO.145 STEEL S275 IS NOT PRESENT IN EN 10025-6 * ERROR *` The calculation will be terminated if the thickness of the designed member web or flange thickness is outside the limits of the steel standard. The error code will be issued: `MEMBER NO. 1000 CURRENT THICKNESS IS OUT OF EN 100219-1 * ERROR *`

Only members from rolled, tube, and roll-formed assortment sections, and compound sections (such as double angles of T-type sections, double channels) are may be designed in STAAD.Pro.

Economy of selected section is indicated by ratio the ration (which can be set using the `RATIO` parameter) $\sigma / R_y y_c$ presented in calculation results. A section is economical when said ratio equals to 0.9 – 0.95.

D13.C.2 Member Capacities

D13.C.2.1 Axial Tension Members

Stress in a section of an axial tension member shall not exceed design strength R_y of the selected steel multiplied by the coefficient of service conditions, γ_c (input by the `GAMC1` and `CAMC2` parameters), take from table 1 of SP 16.13330.2011.

The slenderness of tension members shall not exceed the slenderness limit, λ_u indicated in table 33 of SP 16.13330.2011 as equal to 150. This limit may be specified using the `CMM` parameter, which defaults to 150.

The net section factor (the ratio of A_{net}/A_{gross}) is specified by the `NSF` parameter and is used for tension members to allow for the reduction of design cross-section area.

D13.C.2.2 Axial Compression Members

All axial compression members are calculated as long bars (i.e., with allowance for slenderness - $\lambda = l_0/i_{min}$, where l_0 is the effective length of the element). Calculation is performed in accordance with clause 7.1.1 of SP 16.13330.2011, with the buckling coefficient ϕ determined by equation 8. Effective lengths of elements (within and out-of plane) take into account role and location of the bar in the structure, as well as fixity of the ends ($l_0 = \mu_1$), are determined according the requirements of section 10.3 of SP 16.13330.2011 and are set by the user specification of the members. Slenderness parameters, $\mu_{x(z)}$ and μ_y are set using the parameters `KZ` and `KY`, respectively. If the slenderness parameters of an element is not precisely known, then the effective length can be specified using the `LY` and `LZ` parameters, instead. The ultimate slenderness of compression members shall now exceed the limit values given in table 32 of SP 16.13330.2011, or a user-specified value provided through the `CMN` parameter. The value of the coefficient α used in Table 32 is taken within the limits of 0.5 and 1.0. The limiting slenderness value in compression elements depends on stress acting the member, buckling coefficient, and design resistance of the steel.

Since the slenderness can be different in various planes, the greatest slenderness ratio is assumed in calculations. A warning is given if the slenderness ratio of a compression element exceeds the limit, but the calculations are continued. If the slenderness ratio exceeds the limit value, the output line containing the slenderness check is preceded by a # (pound or hash symbol).

The calculations of single members are performed in this manner. If the member is subjected to axial forces and bending moment (e.g., due to self weight), then the calculation of load bearing capacity will be done taking into account the axial force and bending moments and the buckling resistance only under the axial compression according to clause 7.1 of SP 133330.2011. Local buckling of the web and flanges of centrally loaded members is checked. Stiffener ribs are recommended if necessary.

D13.C.2.3 Flexural Members

Member subjected to bending moments and shear forces are called flexural members.

There are three classes of flexural elements:

1. **Elastic** - in cross-section, the stress in the extreme compression fiber of the steel member –assuming an elastic distribution of stresses– can reach the yield strength. $\sigma \le R_y$, where σ is the absolute value of the stress.
2. **Elasto-plastic** - in on part of the cross-section, the stresses are $\sigma \le R_y$ and in another $\sigma = R_y$.
3. **Plastic state** (i.e., conditional plastic hinge) - across the entire cross-section, the stresses are $\sigma = R_y$.

The parameter `TB` is used to specify either class 1 (elastic) or class 2 (elasto-plastic).

The calculation of flexural members consists of verification of strength, stability, and deflection.

Normal and tangential stresses are verified by strength calculations of members. Normal stresses are calculated in the outermost section fibers. Tangential stresses are verified in the neutral axis zone of the same section. If the normal stresses do not exceed design steel strength and tangential stresses do not exceed the design value of steel shear strength, $R_s\gamma_s$, then according to clause 8.2.1 of SP 16.13330.2011 the principal stresses are checked.

For elements subjected to biaxial bending moments according to clauses 8.2.1 and 8.4.1 of SP 16.13330.2011

D13.C.2.4 Eccentrically Compressed/Tensioned Members

Eccentrically compressed or tensioned members are subjected to simultaneous action of axial force and bending moment. Bending moment appears to eccentrically applied longitudinal force or due to transverse force.

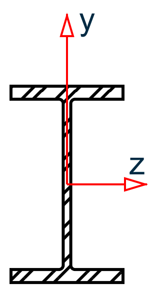
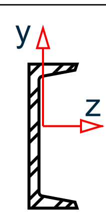
Related Links

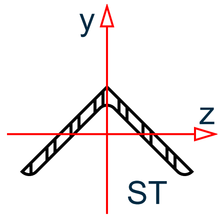
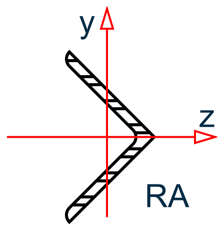
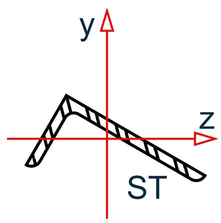
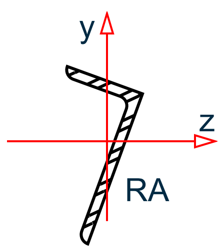
- *V.SNiP SP16 2011 - I Section with Axial Load* (on page 4469)
- *V.SNiP SP16 2011 - I Section with UDL* (on page 4472)

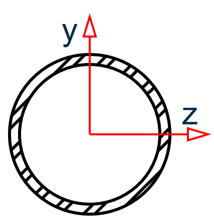
D13.C.3 Built-in Russian Steel Section Library

STAAD.Pro can check many standard sections in SP 16.13330.2011.

Table 202: Typical Sections for Russian Steel Design

Section	Section Type	Designation form
I-beam (GOST 8239-89)		`ST I12`
Regular I-beam (GOST 26020-83)		`ST B1-10`
Broad-flanged I-beam (GOST 26020-83)		`ST SH1-23`
Column I-beam (GOST 26020-83)		`ST K1-20`
Channel (GOST 8240-89)		`ST C14`

Section	Section Type	Designation form
Equal legs angle (GOST 8509-89)	 	`ST L100x100x7` `RA L100x100x7`
Unequal legs angle (GOST 8510-89)	 	`ST L125x80x10` `RA L125x80x10`

Pipes (GOST 10705 and GOST 10706; welded and for gas piping)		`ST PIP102x5.5`
		or
		`ST PIPE OD 0.102 ID 0.055`

Section	Section Type	Designation form

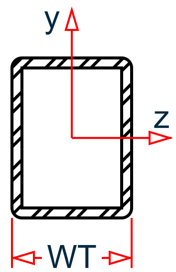
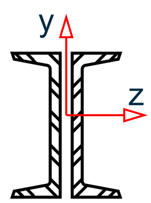
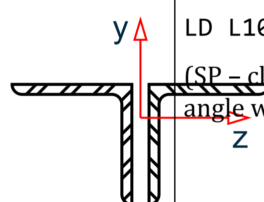
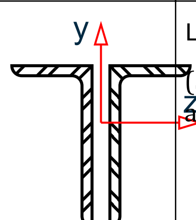
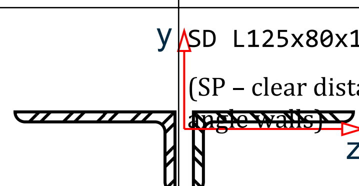
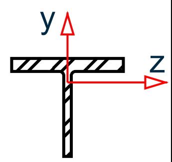
Roll-formed square and rectangular tubes (GOST 8639 and GOST 8645)		`ST TUB160x120x3`
		or
		`ST TUBE TH 0.003 WT 0.12 DT 0.16`

Table 203: Compound Sections for Russian Steel Design

Section	Section Type	Designation form
Double channels		`D C14 SP 0.01` (SP – clear distance between channel walls)
Double equal legs angles		`LD L100x100x7 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with long legs back to back		`LD L125x80x10 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with short legs back to back		`SD L125x80x10 SP 0.01` (SP – clear distance between angle walls)

Section	**Section Type**	**Designation form**

Tee with flange at the top
Note: Flange of Tee beam is at the top part of cross-section if beta angle = 0°, or at the bottom part if beta angle = 180°.
		`T I12`
		`T B1-10`
		`T SH1-23`
		`T K1-20`

For entry of cross-sectional dimensions command `MEMBER PROPERTIES RUSSIAN` is used.

Refer to D13.B.2 Built-in Russian Steel Section Library (on page 1890) for an example.

D13.C.4 Design Parameters

Table 204: Design parameters for design of steel members per SP 16.13330.2011

Parameter Name	**Default Value**	**Description**
CODE	-	Must be specified as `RUSSIAN 2011` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	1	Member design parameter: 0. Design member for forces at their ends or at the sections defined by the SECTION command (TR.41 Section Specification (on page 2665)). 1. Calculate the major axis moment, Mz, and 13 points along the beam and design the beam the location of the maximum Mz value. 2. Same as BEAM = 1, but additional checks are carried out at beam ends and at critical intermediate sections. 3. Calculate forces at 13 points and perform design checks at all locations, including at ends

Parameter Name	Default Value	Description
BMT		Bimoment for design of eccentrically loaded members per clause 8.4 or members subjected to two bending moments per clause 8.2.
CB	1	Location of loading on the beam: 1. top flange 2. bottom flange
CMM	0	Slenderness limit for tension members: 0. suppress slenderness check 1. 150 2. 200 3. 250 4. 300 5. 350 6. 400
CMN	0	Ultimate slenderness for compression members. Limits as per SP 16.13330.2011 clause 10.4, Table 32: 0. suppress slenderness check 1. 180-60a 2. 120 3. 210-60a 4.220-40a 5. 220 6. 180-60a 7. 210-60a 8. 200 9. 150 where a = alpha calculated as defined in SP 16.13330.2011 Table 32.
DFF	None	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Jointof member	

Parameter Name	Default Value	Description
DJ2	End Joint of member	
DMAX	1,000 in	Maximum allowable section depth
DMIN	0	Minimum allowable section depth
ENMAIN	1	The number of the steel standard taken from Table 3.1 in EN 1993-1-1: 2005
ENSGR	1	The number of the steel grade from Table 3.1 in EN 1993-1-1: 2005. D5.C.6 Design Parameters (on page 1502) for values. Also, see Note 1 below.
GAMC1	1.0	Specific service condition coefficient for buckling design
GAMC2	1.0	Specific service condition coefficient for strength design
GAMM	1.0	Specific partial coefficient for steel for European . Refer to D13.C.1 General (on page 1905) for details.
GMF	1.0	Ratio between design and characteristic load value
KY	1.0	Coefficient of effective length in respect to local axis Y (in plane XZ)
KZ	1.0	Coefficient of effective length in respect to local axis Z (in plane XY)
LEG	4	Describes the type of and position of loading on the beam: 1. concentrated in middle of the span 2. concentrated in the quarter of the span 3. concentrated at the end of bracket 4. uniformly distributed along beam 5. uniformly distributed on bracket

Parameter Name	Default Value	Description
LY	Member Length	Effective length in respect to local axis Y (in plane XZ)
LZ	Member Length	Effective length in respect to local axis Z (in plane XY)
MAIN	1	Standard of steel grade (GOST): 1. GOST27772-88 2. GOST10705-80 3. GOST10706-76 4. GOST8731-87 5. TY14-3-567-76
NSF	1.0	Net section factor for tension members or web section area weakening factor for bending members
PY	0	Design steel strength (yield strength) **Note:** Used when parameters `MAIN` and `SGR` are not defined for Russian steel materials or `ENMAIN` and `ENSGR` for European steel materials. See Note 1 below.
RATIO	1.0	Permissible ratio of the actual capacities.
SBLT	0	Number of lateral bracing restraints along the span: 0. beam is not fixed laterally 1. singe restraint at mid-span 2. or higher is the number of uniformly spaced lateral restraints along the span

Parameter Name	Default Value	Description
SGR	1	Steel grade (STAL): See Note 1 below. 1. S235 2. S245 3. S255 4. S285 5. S345 6. S345K 7. S375 8. S390 9. S440 10. S590 11. S590K 12. VSt3kp 13. VSt3ps 14. VSt3sp 15. VSt3ps4 16. St3ps4 17. 20
TB	1	Indication of stress strain state per Cl 4.2.7: 1. 1st Class (Elastic) 2. 2nd Class (Elasto-plastic) 3. 3rd Class (Plastic) The engineer shall consider the appropriate class of the member, as it is used in multiple checks under Ch.8. for elastic or elastic-plastic calculation for bending members (non-axially compressed or tensioned): Cl 8.2.1, Cl. 8.2.3, Cl. 8.4.1, Cl. 8.4.4, and Cl 8.4.6.

Parameter Name	Default Value	Description
TRACK	0	Output details: 0. suppressed output information 1. extended output information for the critical section under the critical load case 2. full output information for the critical section under the critical load case 3. extended output information for all sections under all load cases.
UNL	0	Design steel strength (ultimate strength) **Note:** Used when parameters MAIN and SGR are not defined.

Notes

1. It is common practice to design only Russian or European shapes to the SNiP codes. For Russian steel sections, the steel design strength value, Ry, of a particular steel grade can be obtained from Table C.5 of SP 16.13330.2011 and is used for the SGR parameter. For European sections, the ENSGR and ENMAIN parameters are used accordingly. The ENSGR parameter is valid only with a corresponding value of the ENMAIN parameter when specifying European steel grades. If steel sections from other countries must be used, the PY parameter is used to specify the steel strength.

D13.C.5 Member Selection and Code Check

Both code checking and member selection options are available in SP 16.13330.2011.

Output of selection and check results are given in suppressed, extended and advanced forms. Form of output results depends on value of parameter TRACK.

Results are presented in tables. Three versions of output results are possible: suppressed – results according the critical strength condition (TRACK=0), extended - results according to all check conditions (TRACK=1) and advanced – complete information on results of member design (TRACK=2).

In tables of results common data for all TRACKs are indicated:

(TRACK=2)

In tables of results common data for all TRACKs are indicated:

number of member;
type and number of cross-section;
result obtained (ACCEPTED – requirements are met, FAILURE – are not met);

abbreviated name of normative document (code, standard) (SNiP);
number of check clause;
safety of strength (ratio between design and normative values);
number of the most unfavorable loading;
value of longitudinal force acting in the member with subscript indicating its direction (“C” – compression, “P” – tension);
bending moments in relation to local member axes Z and Y;
distance to section, in which the most unfavorable combination of forces acts.

Note: Slenderness criteria is not reported in the Design Results table of Post-processing mode because Slenderness check is performed to check overall limit; not a guiding criteria for member check.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

Note: The output for the Russian code uses the axes system from STAAD.Pro, except where details of the equations from the code are printed, in which case the axes system is as defined in the code.

D13.C.5.1 Example of TRACK 0 output

In suppressed form (TRACK 0) results are presented according to the critical check for given member with indication of SP 16 clause number, according to which strength safety of the member is minimum.

```
ALL UNITS ARE - KN METE
   ============================================================================
   MEMBER      CROSS           RESULT/   CRITICAL COND/     RATIO/     LOADING/
               SECTION NO.       FX            MZ             MY       LOCATION
   ============================================================================
       2   I      IPE300          PASS      SNiP- 8.4.1        0.76        1
                              0.000E+00    -4.446E+01    0.000E+00   4.000E+00
```

D13.C.5.2 Example of TRACK 1 output

In extended form (TRACK 1) results are presented on the basis of all required by SP 16 checks for given stress state.

```
ALL UNITS ARE - KN METE
   ============================================================================
   MEMBER      CROSS           RESULT/   CRITICAL COND/     RATIO/     LOADING/
               SECTION NO.       FX            MZ             MY       LOCATION
   ============================================================================
       2   I      IPE300          PASS      SNiP- 8.2.1        0.22        1
                              0.000E+00    -4.446E+01    0.000E+00   4.000E+00
       2   I      IPE300          PASS      SNiP- 8.2.1        0.00        1
                              0.000E+00    -4.446E+01    0.000E+00   4.000E+00
       2   I      IPE300          PASS      SNiP- 8.2.1        0.20        1
                              0.000E+00    -4.446E+01    0.000E+00   4.000E+00
       2   I      IPE300          PASS      SNiP- 8.4.1        0.76        1
                              0.000E+00    -4.446E+01    0.000E+00   4.000E+00
```

D13.C.5.3 Example of a TRACK 2 output

In advanced form (TRACK=2) in addition to tabled results supplementary information is presented.

Material characteristics:
Steel;
Design resistance;
Elasticity modulus;
Section characteristics:
Length of member;
Section area;
Net area;
Inertia moment (second moment of area) (I);
Section modulus (W);
First moment of area (S);
Radius of gyration;
Effective length;
Slenderness;
Results are presented in two columns, Z and Y respectively.
Design forces:
Longitudinal force;
Moments;
Shear force.

Signs "+" and "-" indicate direction of acting longitudinal force, bending moments and shear forces in accordance with sign rules assumed in program STAAD.

Check results in advanced form are presented with values of intermediate parameters by formulas in analytical and numerical expression with indication of SNiP clause.

```
ALL UNITS ARE - KN METE
========================================================================
MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
           SECTION NO.      FX            MZ            MY      LOCATION
========================================================================
   2  I      IPE300        PASS      SNiP- 8.2.1        0.22        1
                        0.000E+00     -4.446E+01    0.000E+00   4.000E+00
   2  I      IPE300        PASS      SNiP- 8.2.1        0.00        1
                        0.000E+00     -4.446E+01    0.000E+00   4.000E+00
   2  I      IPE300        PASS      SNiP- 8.2.1        0.20        1
                        0.000E+00     -4.446E+01    0.000E+00   4.000E+00
   2  I      IPE300        PASS      SNiP- 8.4.1        0.76        1
                        0.000E+00     -4.446E+01    0.000E+00   4.000E+00

MATERIAL DATA
   Steel                         =S355        EN10025-2
   Modulus of elasticity         = 206.E+06 kPa
   Design Strength (Ry)          = 355.E+03 kPa

SECTION PROPERTIES (units - m)
   Member Length                 = 8.00E+00
   Gross Area                    = 5.38E-03
   Net Area                      = 5.38E-03
                                      z-axis      y-axis
```

```
      Moment of inertia (I)         :   836.E-07    604.E-08
      Section modulus (W)           :   557.E-06    805.E-07
      First moment of area (S)      :   314.E-06    625.E-07
      Radius of gyration (i)        :   125.E-03    335.E-04
      Effective Length              :   160.E-01    800.E-02
      Slenderness                   :   0.00E+00    0.00E+00

   DESIGN DATA (units -kN,m)SNiP II-23-81*/2011
      Axial force                   :   0.00E+00
                                         z-axis      y-axis
      Moments                       :  -445.E-01    0.00E+00
      Shear force                   :   0.00E+00    0.00E+00
      Bi-moment                     :   0.00E+00 Value of Bi-moment not being
entered!!!

   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(41)  M/(Wn,min*Ry*GAMAc)=-444.6E-01/( 5.57E-04* 355.0E+03* 1.00E+00=
2.25E-01=<1
      F.(44)  0.87/(Ry*GAMAc)*SQRT(SIGMz^2+3*TAUzy^2)=
              0.87/( 355.0E+03* 1.00E+00)*SQRT(-798.2E+02^2+3* 0.000E+00)=
              1.96E-01=<1
              TAUzy/(Rs*GAMAc)= 0.000E+00/( 205.9E+03* 1.00E+00)= 0.00E+00=<1
      F.(69)  M/(FIb*Wcx*Ry*GAMAc)=
             -444.6E-01/( 2.97E-01* 5.57E-04* 355.0E+03* 1.00E+00)= 2.25E-01=<1
```

Conventional notations assumed in presentation of results: “+”, “-“, “/”, “*”,”**”, “SQRT”, their respective meanings (i.e., addition, subtraction, division, multiplication, raising to the second power (squared), and square root). Conventional notations of stresses, coefficients and characteristics of steel resistance comply with accepted in the SP standard. Only Greek letters are changed by their names (e.g., , γ_c-GAMAC; α-ALPHA; β-BETA, η-ETA, φ-PHI, etc.).

D13.D. Russian Codes - Concrete Design Per SP 63.1330.2012

STAAD.Pro is capable of performing concrete design for beams, columns, slabs, and walls based on the Russian code Сп63.1330.2012 *Бетонные и железобетонные конструкции. Основные положения* (SP 63.1330.2012 *Concrete and Reinforced Concrete Structures. Basic Provisions*).

Note: This code supersedes SNiP 2.03.01-84*.

Design of members per SP 63.1330.2012 requires the *STAAD ECC Super Code* SELECT Code Pack.

Note: This feature requires STAAD.Pro 2007.11 or greater.

D13.D.1 General

Russian Code SP 63.1330.20121 *Concrete and Reinforced Concrete Structures - Basic Provisions* is based on the method of limit states. This code defines two groups of limit states.

Analysis according to the first group of limit states is performed to avoid the following phenomena:

- brittle, plastic or other type of failure,
- loss by structure of stable form or position,

Analysis according to the second group of limit states is performed to avoid the following phenomena:

- excessive opening of cracks (if they are allowed according to service conditions)

Analysis of structures for the first group of limit states is performed for the maximum (design) loads and actions. Analysis of structures for the second group of limit states is made in accordance with the operational (normative) loads and actions. Ratio between design and normative loads is called reliability coefficient for loads which is determined according to SNiP 2.01.07.-85 *Loads and actions*.

STAAD.Pro calculates reinforcement for concrete members. Algorithms for calculation of reinforcement of concrete linear (beams, columns) and 2D (two dimensional) (slabs, walls, shells) members are incorporated into STAAD.Pro. In addition to SP5 2.13330.2011, the *Guide for design of plain concrete and reinforced concrete structures from normal weight and lightweight concrete (to SNiP 2.03.01-84)* has been incorporated in the calculation algorithms.

The program can automatically calculate reinforcement for beams of rectangular or T section and for columns of rectangular or circular section.

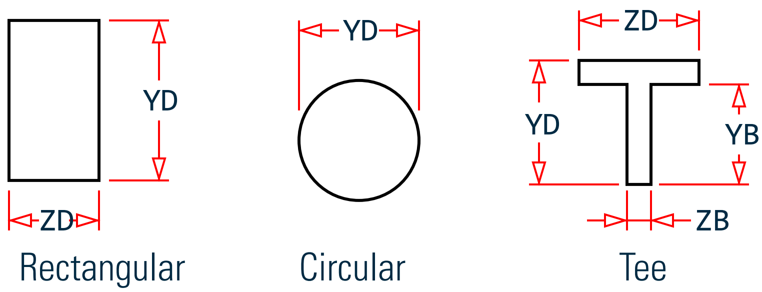


Figure 190: Section types for concrete design

Flange of T-shape beams may be situated at the top zone of the section if the angle β = 0° (`BETA 0`), or at the bottom zone of the section, if β = 180° (`BETA 180`).

D13.D.2 Design Parameters

Values of parameters do not depend on the `UNIT` command.

Commands for calculation of reinforcement are located in the input data file after the command of analysis and as a rule, after output commands to print results of calculation.

Table 205: Parameters for concrete design according to Russian Code SP 63.1330.2012

Parameter Name	Default Value	Description
CODE	-	Must be assigned as `RUSSIAN` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BCL	15	Compression class of concrete

Parameter Name	Default Value	Description
CL	0.05	(Slabs/Walls) Distance from top/ bottom face of slab/wall element to the center of the longitudinal reinforcing bars located in the first local (x) direction (i.e., main thickness of top/bottom concrete cover for slab/wall elements)
CL1	0.05	(Beams) Distance from the top or bottom edge of the beam cross section to the center of longitudinal reinforcement (Columns) Distance from the edge of the column cross section to the center of the longitudinal reinforcing bars
CL2	0.05	(Beams) Distance from left/right side of the beam cross section to the center of longitudinal reinforcement
CRA	0.05	(Slabs/Walls) Distance from top/ bottom face of slab/wall element to the center of the transverse reinforcing bars located in the second local (y) direction (i.e., secondary thickness of top/bottom concrete cover for slab/wall elements)
DD1	16	(Beams) Diameter of the longitudinal reinforcement bars in the beam tension zone (Columns) Minimum diameter of longitudinal reinforcement bars for the column
DD2	16	(Beams) Diameter of the shear reinforcement bars for the beam (Columns) Maximum diameter of longitudinal reinforcement bars for the column

Parameter Name	Default Value	Description
DEPTH	YD	(Beams) Design depth of beam the section. Used for design only, with the default value taken as the YD dimension in the member properties.
EFA	0	(Beams) Face of support location at the end of the beam
ELY	1	(Columns) Length coefficient to evaluate slenderness effects in the local Y axis
ELZ	1	(Columns) Length coefficient to evaluate slenderness effects in the local Z axis
FWB	ZB	(Beams) Design width of the beam's bottom flange. Used for design only, with the default value taken as the ZB dimension in the member properties.
FWT	ZD	(Beams) Design width of the beam's top flange. Used for design only, with the default value taken as the ZD dimension in the member properties.
MMA	0	(Slabs/Walls) Design parameter of slab/wall reinforcement: 0. reinforcement calculation applied by stresses in local axis 1. reinforcement calculation applied by principal stresses
MMB	1	(Slabs/Walls) Design parameter of slab/wall reinforcement: 0. the effect of additional eccentricity is *not* considered 1. the effect of additional eccentricity is considered
NLT	1	Load case ID number of the long-term load case

Parameter Name	Default Value	Description
NSE	13	(Beams) Number of equally spaced sections used for beam design. The upper limit is 20.
RCL	3	Class of longitudinal reinforcement: 1. A240 2. A400 3. A500 4. A600
RSH		(Beams) Class of shear reinforcement: 1. A240 2. A400 3. A500 4. A600
SDX	16	(Slabs/Walls) Diameter of reinforcing bars located in the first local (X) direction
SDY	16	(Slabs/Walls) Diameter of reinforcing bars located in the second local (Y) direction
SELX	0	(Slabs/Walls) Design length of wall member to evaluate slenderness effects in local X axis
SELY	0	(Slabs/Walls) Design length of wall member to evaluate slenderness effects in local Y axis
SFA	0	(Beams) Face of support location at the start of the beam

Parameter Name	Default Value	Description
SSE		(Beam) Limit state parameter for beam design: 0. reinforcement amount calculated for required load capacity (i.e., the first limit state) 1. reinforcement amount calculated for cracking requirements (i.e., the second limit state)
STA	0	(Slabs/Walls) Parameter of limit state for slab/wall design: 0. non symmetric reinforcement amount calculated for required load capacity (i.e., the first limit state) 1. symmetric reinforcement amount calculated for required load capacity (i.e., the first limit state) 2. non symmetric reinforcement amount calculated for cracking requirements (i.e., the second limit state) 3. symmetric reinforcement amount calculated for cracking requirements (i.e., the second limit state)
TEM	0	Parameter of concrete hardening conditions: 0. normal concrete 1. fine-grain concrete 2. (Beams and Columns) fine-grain, steam hardened concrete
UB2	0.9	(Slabs/Walls) Specific service conditions coefficient for concrete, γ_b
UBM	1	Product of service conditions coefficients for concrete, except for γ_b in the case of slabs or walls (see UB2)

Parameter Name	Default Value	Description
USM	1	(Beams and Columns) Total product of service conditions coefficients for longitudinal reinforcement, γ_s
WLT	0.3	(Beams and Slabs/Walls) Ultimate width of long-term crack
WST	0.4	(Beams and Slabs/Walls) Ultimate width of short-term crack.

Example

```
STAAD

…

UNIT  MM
MEMBER  PROPERTIES
*  Columns of rectangular cross-section
1  TO  16  PRI  YD  350.  ZD  350.
*  Columns of circular cross-section
17  TO  22  PRI  YD  350.
*  Beams of T cross-section
23  TO  40  PRI  YD  450.  ZD  550.  YB  230.  ZB  200.
UNIT  METER
ELEMENT  PROPERTY
41  TO  100 THICKNESS  0.14
101  TO  252 THICKNESS  0.16
* Flange of T beams is located at the bottom zone of cross-section
BETA  180.  MEMB  23  TO  40

…

*  Command of analysis
PERFORM  ANALYSIS
…
* Output command to print results of calculation (according to user’s
judgment)
…
*  Command of loading and their combinations considered in design
LOAD  LIST  1  5  TO  9
*  Command to start reinforcement calculation procedure
START  CONCRETE  DESIGN
CODE  RUSSIAN
* List of parameters being used in reinforcement calculation
…
…
BCL  20.  MEMB  17  TO  22
CL1  0.04  MEMB  1  TO  40
DD2  10.  MEMB  23  TO  40
CRA 0.036  MEMB  41  TO  252

…
```

```
* Command of beam reinforcement calculation
DESIGN  BEAM  23  TO  40
* Command of column reinforcement calculation
DESIGN  COLUMN  1  TO  22
* Command of calculation 2D elements (slabs, walls, shells)
DESIGN  ELEMENT  41  TO  252

* Command of interruption of the reinforcement calculation
```

D13.D.3 Beams

Reinforcement areas are calculated for beams of rectangular and T cross section. When calculating the longitudinal reinforcement bending moment about the local axis Z and torsional moments are considered; the influence of longitudinal forces and bending moments in relation to local axis Y is ignored. When calculating transverse reinforcement shear forces parallel to local axis Y and torsional moments are considered.

Reinforcement for beams can be calculated either for strength conditions or from crack width limits using the `SSE` parameter.

In general, the calculation of reinforcement for beams is carried out two times: once according to strength conditions and again according to crack width limitation. In reinforcement calculations for strength conditions (i.e., the first limit state), design load values must be used. In reinforcement calculations for crack width limitation (i.e., the second limit state), characteristic (i.e., normative or service) load values are used. Using the multiple analysis capability of STAAD.Pro allows you to carry out both calculations in a single analysis and design run.

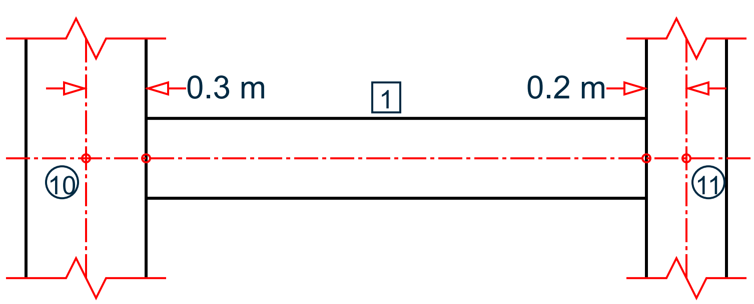
In most cases, calculation of reinforcement is carried out with only a partial number of load cases. In such cases, the `LOAD LIST` command is used to indicate the load case numbers. The load case number indicated by the `NLT` parameter –used to indicate the load cases for permanent and long-term loads– must be included in the list of considered loads.

The parameter `BCL` can be equal to any value of concrete compression strength class given in SNiP 2.03.01-84*, as well as any intermediate value.

It should be noted that the accuracy of results in calculating transverse reinforcement increases with the value of the `NSE` parameter.

Parameters `SFA` and `EFA` are considered only in the calculation of transverse reinforcement. Beam 1 shown in the following figure has rigid end-links of 0.3m at the start of the beam and 0.2m at the end of the beam. In the STAAD input file, this is modeled as:

```
MEMBER OFFSET
1 START 0.3 0 0
1 END -0.2 0 0
```



When the `MEMBER OFFSET` command is used, the forces used in calculation of reinforcement are for a beam whose length is taken from point a to point b (i.e., between the faces of the supporting columns) in the figure. In this case, it is necessary to use the default values of the parameters `SFA` and `EFA` (zero).

When the `MEMBER OFFSET` command *not* is used, the forces used in calculation of reinforcement are for a beam whose length is taken from node 10 to node 11. In this case, it is necessary to assign the distances to the design parameters `SFA` and `EFA`.

```
CODE RUSSIAN
SFA 0.3 1
EFA 0.2 1
```

Note: The sign is positive for the `EFA` parameter, as it is assumed to be taken away from the gross beam length.

In both cases, the calculated quantity of transverse reinforcement is the same. The calculated quantity of longitudinal reinforcement in the second case will be greater.

For beams, the following output is generated:

beam number;
method of calculation (according to conditions of strength or limitations of opened crack width);
length and cross-sectional dimensions;
distance from resultant of forces acting in bottom/top reinforcement to bottom/top edge of the section;
distance from the side edge of cross-section of the beam web to the centroid of longitudinal bars located at this edge;
concrete class;
class of longitudinal and transverse reinforcement;
assumed in calculations bar diameters of longitudinal and transverse reinforcement;
calculation results of longitudinal and transverse reinforcement (in two tables).

In nine columns of the first table the following results are presented:

Section – distance of the section from the “start” of the beam, мм;
As- – cross-sectional area of longitudinal reinforcement in the bottom zone of cross-section of the beam, if angle BETA=0°, or in the top zone, if BETA=180°, sq.cm;
As+ – cross-sectional area of longitudinal reinforcement in the top zone of cross-section of the beam , if angle BETA=0°, or in the top zone, if BETA=180°, sq.cm;
Moments (-/+) – values of bending moments, determining cross-sectional areas of longitudinal reinforcement As* and As* , kNm;
Load. N. (-/+) – numbers of loading versions, determining cross-sectional areas of longitudinal reinforcement;
Acrc1 – short-term opened crack width, mm;
Acrc2 – long-term opened crack width, mm.

Opened crack width is presented only in the case when calculation is performed according to conditions limiting opened crack width.

In ten columns of second table the following results are presented:

Section – distance of the section from the “start” of the beam, mm;
Qsw – intensity of transverse reinforcement, kN/m;
Asw – cross-sectional area of transverse bars, sq.cm, if their step is 10, 15, 20, 25 or 30 cm;
Q – value of shear force parallel to the local axis, kN;

T – value of torsional moment, kNm;
Load N. – number of loading version, determining intensity of transverse reinforcement.

D13.D.4 Columns

Reinforcement for columns of rectangular or circular cross section can be calculated. The flexibility of columns can be evaluated in two ways.

- In the case of linear analysis (i.e., the `PERFORM ANALYSIS` command), the effective length is evaluated using the `ELY` and `ELZ` parameters, conforming to provisions of SP5 2.13330.2011.
- If a `P-DELTA` or `NONLINEAR` (i.e. nonlinear geometry) analysis is performed, the values of the `ELY` and `ELZ` parameters should be close to zero (e.g., ELY = ELZ = 0.01).

Longitudinal reinforcement for columns is calculated only for the strength condition. Longitudinal forces and bending moments in relation to local axes Y and Z are taken into account in longitudinal reinforcement calculations.

For rectangular columns, the following output is generated:

column number;
column length and cross-sectional dimensions;
distance of centroid of each longitudinal bar from the nearest edge of the cross-section;
concrete class;
longitudinal reinforcement class;
range of longitudinal reinforcement bar diameters assumed in calculation;
diameter of longitudinal reinforcement bars obtained in calculation;
total quantity of longitudinal bars;
quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Y ;
quantity of longitudinal bars at each cross-section edge, directed parallel to the local axis Z.

In nine columns of the table under the heading LONGITUDINAL REINFORCEMENT, the following output is presented:

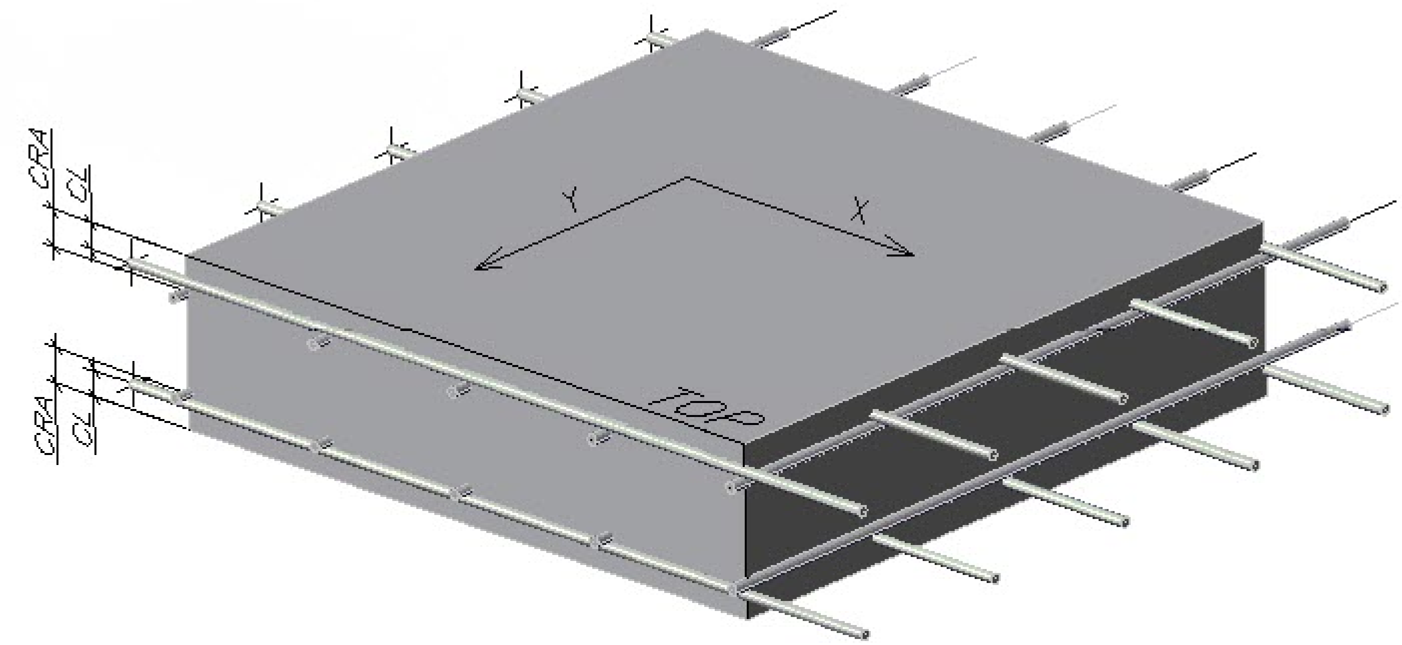
Section – distance of the section from the “start” of the column, mm;
Astot – total cross-sectional area of longitudinal reinforcement, sq.cm;
Asy – cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Y , sq.cm;
Asz – cross-sectional area of longitudinal reinforcement bars at each edge of section, directed parallel to the local axis Z , sq.cm;
Percent – reinforcement percentage in the section;
Nx, Mz, My – respective values of longitudinal force and bending moments in relation to the local axes Z and Y , determining cross-sectional area of longitudinal reinforcement;
Load.N. – number of loading version, determining cross-sectional area of longitudinal reinforcement.

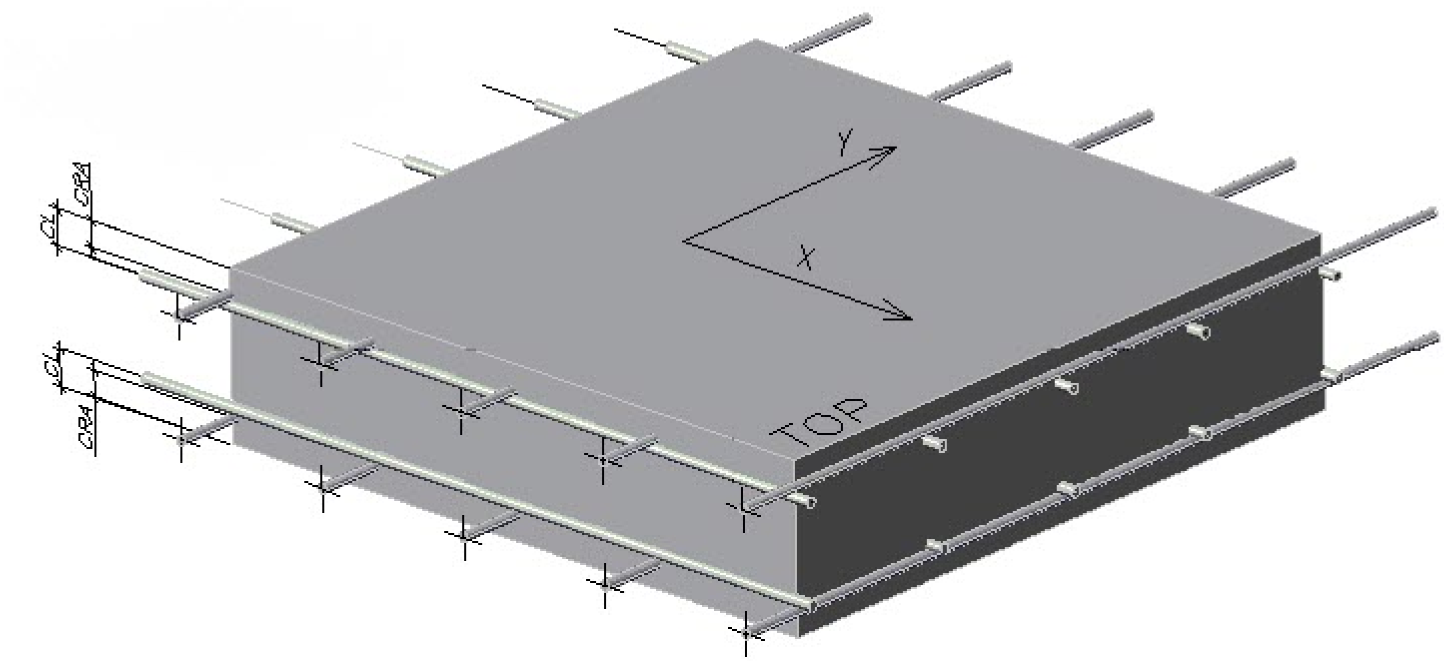
D13.D.5 2DElements: Slabs, Walls, and Shells

In general, the calculation of reinforcement for two dimensional elements (i.e., slabs, walls, and shells) is performed two times: once according to conditions of strength and again according to limiting of width of cracks. In reinforcement calculations for strength conditions (i.e., the first limit state), design load values must be used. In reinforcement calculations for crack width limitation (i.e., the second limit state), characteristic (i.e., normative or service) load values are used. Using the multiple analysis capability of STAAD.Pro allows you to carry out both calculations in a single analysis and design run.

Symmetric or non symmetric reinforcement of elements is calculated according to either first or second limit states by using the STA parameter.

It is necessary to pay close attention to the arrangement of local axis of the element with respect to the direction of reinforcement for the calculation of reinforcement. This is controlled using the CL and CRA parameters, as shown in the following figure.


CRA
CL
CRA
CL
Y
X
TOP


CL
CRA
CL
CRA
Y
X
TOP

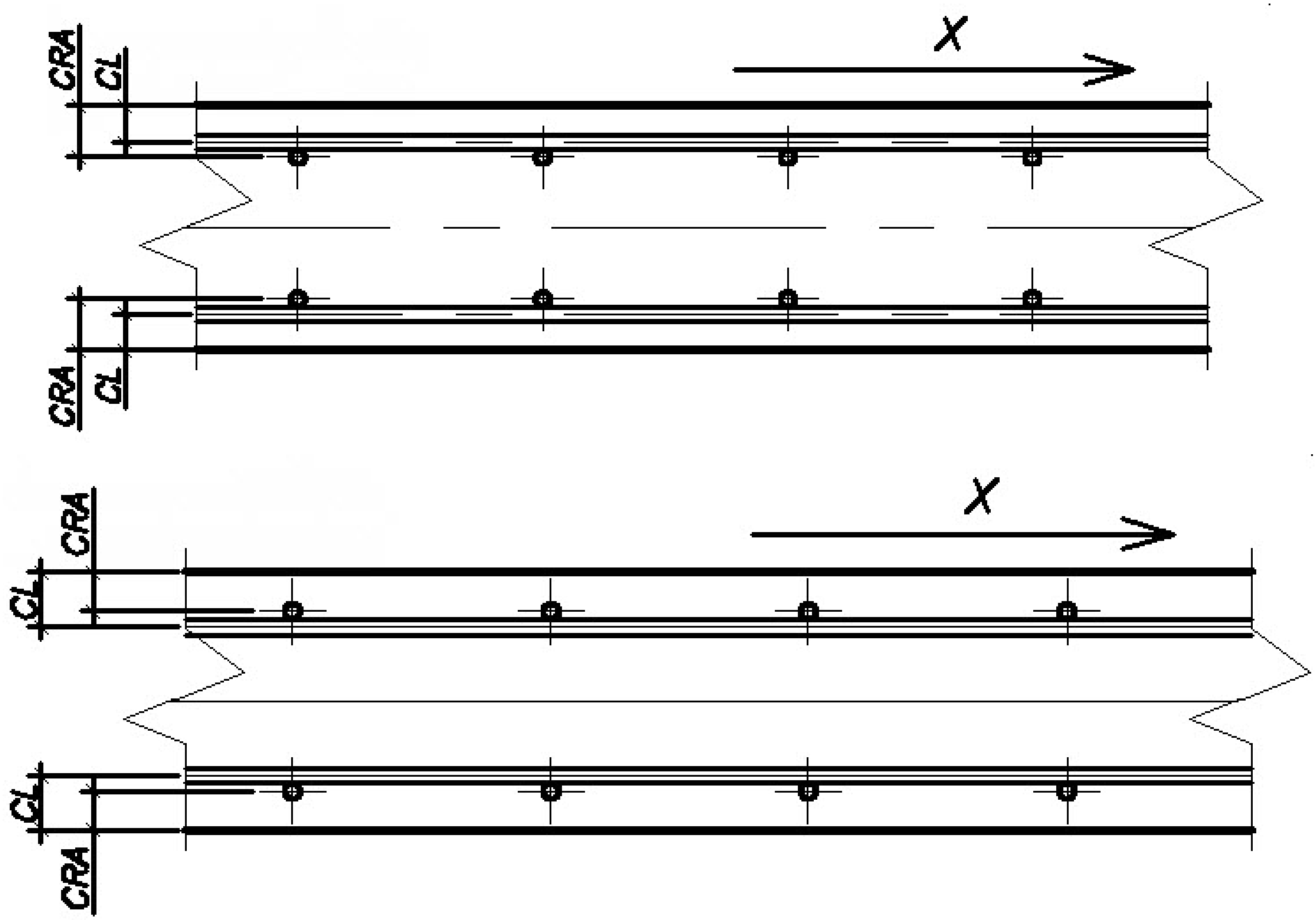


Figure 191: Russian concrete design parameters CL and CRA

An example output of reinforcement calculations:

```
                                 SLAB/WALL DESIGN RESULTS
                  (by stresses in local axes for limitation of crack width)
-------------------------------------------------------------------------------
---------------
  Element    Asx        Mx       Nx      Load. N.     Asy            My
Ny        Load N.
           sq.cm/m   kNm/m    kN/m       (X)       sq.cm/m      kNm/m    kN/
m         (Y)
-------------------------------------------------------------------------------
----------------
60  TOP      0.00     *4.9     0.0        1       0.00        *4.5
0.0         1
    BOT      3.53     *9.9     0.0        3       3.46        *8.9
0.0         3
61  TOP      0.00     *5.3     0.0        1       0.00        *4.7
```

```
0.0        1
    BOT      3.87    *10.7     0.0        3       3.65        *9.4
0.0        3
62  TOP      0.00     *5.6     0.0        1       0.00        *4.8
0.0        1
    BOT      4.10    *11.2     0.0        3       3.77        *9.6
0.0        3
```

Where:

Table 206: Slab design output

Result	Description
Element	number of finite element, “TOP” - top zone of member, “BOT” - bottom zone of member (“top” zone of member is determined by positive direction of local axis, Z)
Asx	intensity of reinforcing in the longitudinal direction (parallel to the local axis, X), *sq.cm/m*
Mx	distributed bending moment in respect to the local axis, Y, *kNm/m*
Nx	distributed longitudinal force directed parallel to the axis *X*, *kNm/m*
Load N.(X)	number of loading version, determining intensity of reinforcing in the longitudinal direction
Asy	intensity of reinforcing in the transverse direction (parallel to the local axis Y), *sq.cm/m*
My	distributed bending moment in respect to the local axis X *kNm/m*
Ny	distributed longitudinal force directed parallel to the local axis *Y* *kN/m*
Load N.(Y)	number of loading version, determining intensity of reinforcing in the transverse direction

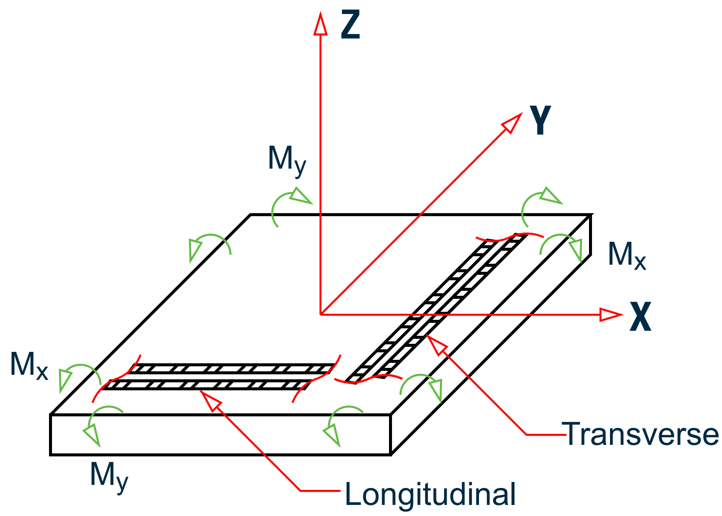


Figure 192: Local coordinate system of 2D member and notation of forces

D13.E. Russian Codes - Steel Design per SP 16.13330.2017

STAAD.Pro is capable of performing steel design based on the Russian code Сп 16.13330.2017 *стальные конструкции* (SP 16.13330.2017 *Steel Structures*).

D13.E.1 General

Design Code SP *Steel Structures* –as is the case in the majority of modern codes– is based on the method of limit states. The following groups of limit states are defined in the Code.

- The first group concerns losses of general shape and stability, failure, and qualitative changes in configuration of the structure (i.e., ultimate limit states). Appearance of non-allowable residual deformations, displacements, yielding of materials or opening of cracks.

  Analysis of structures for the first limit state is performed using the maximum (design) loads and actions, which can cause failure of structures.
- The second group concerns states of the structure which worsen their service or reduce durability due to exceeding allowable deflections, deviations, settlements, vibrations, etc. (i.e., service conditions).

  Analysis of structures for the second limit state is performed using service (normative) loads and actions. Relation between design and normative loads is referred to as coefficient of load reliability, which is defined in SP 16.13330.2017.

The coefficient of reliability for destination `GAMA n` according to SP 20.13330.2017 shall be taken into account determining loads or their combinations.

According to SP 16.13330.2017, the strength of steel is represented by the characteristic value. To obtain the design value, the steel reliability coefficient `GAMM` is used. The default value of `GAMM` is 1.0.

Only members from rolled, tube, and roll-formed assortment sections, and compound sections (such as double angles of T-type sections, double channels) are may be designed in STAAD.Pro.

Various checks are made according to SP 16.13330 Design Code, particularly implementing Clauses:

- 7.1. Calculation of elements in central tension and compression with solid sections

- Cl. 7.1.1 Eq. (5)
- Cl. 7.1.3 Eq. (7)
- 7.3. Stability check of walls and flanges of center-compressed elements with solid sections
- 8. Calculation of elements in bend
    - Strength calculation of bending elements with solid sections
        - Cl. 8.2.1 Eq. (41), (42), (43), (44) – for 1st Class Elements
        - Cl. 8.2.3 Eq. (50), (51), (53), (54), (55) – for 2nd and 3rd Class Elements
    - Calculation of general stability of bending elements with solid sections
        - Cl. 8.4.1 Eq (69), (70)
- 9. Calculation of elements in sinuous central force
    - Strength calculation of elements with solid sections
        - Cl. 9.1.1 Eq. (105), (106)
        - Cl. 9.1.2
    - Calculation of stability of elements with solid sections
        - Cl. 9.2.1
        - Cl. 9.2.2 Eq. (109)
        - Cl. 9.2.4 Eq. (111)
        - Cl. 9.2.5 Eq. (112), (113), (114)
        - Cl. 9.2.7
        - Cl. 9.2.8
        - Cl. 9.2.9 Eq. (116), (117)
        - Cl. 9.2.10 Eq. (120), (121), (122)
- 10.4. Ultimate Slenderness of Elements
    - Cl. 10.4.1
- Steel Grades are determined from Appendix C, Tables C.3, C.4, C5.
- Coefficients for the calculation of the stability of centrally- and eccentrically- compressed elements are determined from Appendix E (normative), Tables E.1, E.2, E.3, E.4, E.6.
- Coefficients for calculation of constructional elements with regard to the development of plastic deformations are determined from Appendix F (normative), Table F.1
- Stability coefficient ϕb at a bend is calculated according to Appendix G (obligatory).

Also, the Selection of steel of hot rolled sections, plates, tubes and pipes is implemented.

D13.E.2 Member Capacities

D13.E.2.1 Axial Members in Tension

Stress in a section of an axial tension member shall not exceed design strength Ry of the selected steel multiplied by the coefficient of service conditions, γ_c (input by the `GAMC1` and `CAMC2` parameters), take from table 1 of SP 16.13330.2017.

The slenderness of tension members shall not exceed the slenderness limit, λ_u indicated in table 33 of SP 16.13330.2017 as equal to 150. This limit may be specified using the `CMM` parameter, which defaults to 150.

The net section factor (the ratio of A_{net}/A_{gross}) is specified by the `NSF` parameter and is used for tension members to allow for the reduction of design cross-section area.

D13.E.2.2 Axial Compression Members

All axial compression members are calculated as long bars (i.e., with allowance for slenderness - $\lambda = l_0 / i_{min}$, where l_0 is the effective length of the element). Calculation is performed in accordance with clause 7.1.1 of SP 16.13330.2017, with the buckling coefficient ϕ determined by equation 8. Effective lengths of elements (within and out-of plane) take into account role and location of the bar in the structure, as well as fixity of the ends (l_0 = μ_1), are determined according the requirements of section 10.3 of SP 16.13330.2017 and are set by the user specification of the members. Slenderness parameters, $\mu x(z)$ and μy are set using the parameters `KZ` and `KY`, respectively. If the slenderness parameters of an element is not precisely known, then the effective length can be specified using the `LY` and `LZ` parameters, instead. The ultimate slenderness of compression members shall now exceed the limit values given in table 32 of SP 16.13330.2017, or a user-specified value provided through the `CMN` parameter. The value of the coefficient α used in Table 32 is taken within the limits of 0.5 and 1.0. The limiting slenderness value in compression elements depends on stress acting the member, buckling coefficient, and design resistance of the steel.

Since the slenderness can be different in various planes, the greatest slenderness ratio is assumed in calculations. A warning is given if the slenderness ratio of a compression element exceeds the limit, but the calculations are continued. If the slenderness ratio exceeds the limit value, the output line containing the slenderness check is preceded by a "#" (pound or hash symbol).

The calculations of single members are performed in this manner. If the member is subjected to axial forces and bending moment (e.g., due to self weight), then the calculation of load bearing capacity will be done taking into account the axial force and bending moments and the buckling resistance only under the axial compression according to clause 7.1 of SP 133330.2017. Local buckling of the web and flanges of centrally loaded members is checked. Stiffener ribs are recommended if necessary.

Note: Wall cross section area Aw used in SP 16.13330.2017 code is the area of web excluding thickness of flange(s).

D13.E.2.3 Flexural Members

Member subjected to bending moments and shear forces are called flexural members.

There are three classes of flexural elements:

- Elastic - in cross-section, the stress in the extreme compression fiber of the steel member –assuming an elastic distribution of stresses– can reach the yield strength. $\sigma \leq R_y$, where σ is the absolute value of the stress.
- Elasto-plastic - in on part of the cross-section, the stresses are $\sigma \leq R_y$ and in another $\sigma = R_y$.
- Plastic state (i.e., conditional plastic hinge) - across the entire cross-section, the stresses are $\sigma = R_y$.

The parameter `TB` is used to specify either class 1 (elastic), class 2 (elasto-plastic), or class 3 (plastic), with elastic being the default.

The calculation of flexural members consists of verification of strength, stability, and deflection.

Normal and tangential stresses are verified by strength calculations of members. Normal stresses are calculated in the outermost section fibers. Tangential stresses are verified in the neutral axis zone of the same section. If the normal stresses do not exceed design steel strength and tangential stresses do not exceed the design value of steel shear strength, $R_s \gamma_s$, then according to clause 8.2.1 of SP 16.13330.2017 the principal stresses are checked.

For elements subjected to biaxial bending moments according to clauses 8.2.1 and 8.4.1 of SP 16.13330.2017.

Stability check as per clause 8.4.4 should be performed for I shaped section only. If stability is not ensured as per 8.4.4 for I shaped sections, then a check as per clause 8.4.1 is performed. For other types of sections, the program skips clause 8.4.4 and checks per clause 8.4.1.

D13.E.2.4 Eccentrically Compressed/Tensioned Members

Eccentrically compressed or tensioned members are subjected to simultaneous action of axial force and bending moment. Bending moment appears to eccentrically applied longitudinal force or due to transverse force.

Related Links

- *V. SNiP SP16 2017 - Channel section with UDL* (on page 4476)
- *V. SNiP SP16 2017 - CLASS 2 UPT I section* (on page 4484)
- *V. SNiP SP16 2017 - Column in compression* (on page 4488)
- *V. SNiP SP16 2017 - Interaction check of a column* (on page 4497)
- *V. SNiP SP16 2017 - I section with biaxial moment* (on page 4502)
- *V. SNiP SP16 2017 - I section with UDL* (on page 4506)

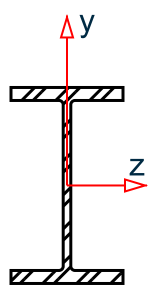
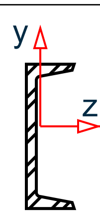
D13.E.3 Supported Section Profiles

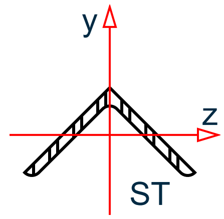
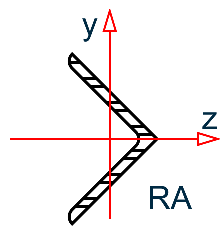
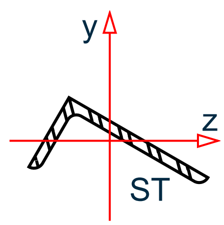
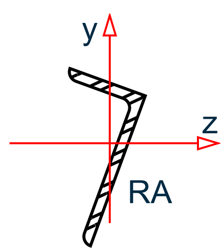
STAAD.Pro can check many standard sections in SP 16.13330.2017.

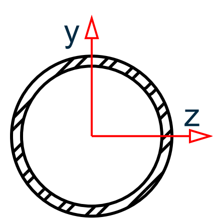
D13.E.3.1 Built-in Russian Steel Section Library

These profiles use the SGR to specify the grade of steel.

Table 207: Typical Sections for Russian Steel Design

Section	Section Type	Designation form
I-beam (GOST 8239-89)		`ST I12`
Regular I-beam (GOST 26020-83)		`ST B1-10`
Broad-flanged I-beam (GOST 26020-83)		`ST SH1-23`
Column I-beam (GOST 26020-83)		`ST K1-20`
Channel (GOST 8240-89)		`ST C14`

Section	Section Type	Designation form
Equal legs angle (GOST 8509-89)	 	`ST L100x100x7` `RA L100x100x7`
Unequal legs angle (GOST 8510-89)	 	`ST L125x80x10` `RA L125x80x10`

Pipes (GOST 10705 and GOST 10706; welded and for gas piping)		`ST PIP102x5.5`
		or
		`ST PIPE OD 0.102 ID 0.055`

Section	Section Type	Designation form

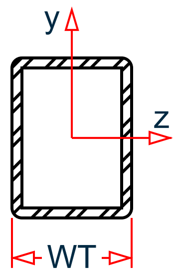
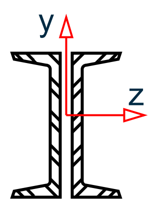
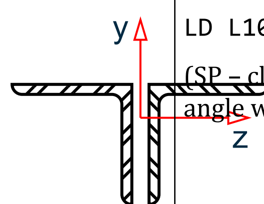
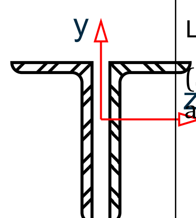
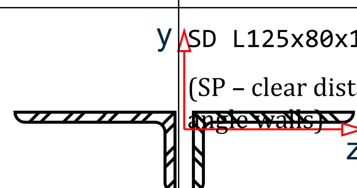
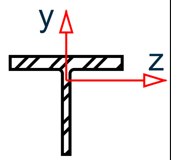
Roll-formed square and rectangular tubes (GOST 8639 and GOST 8645)		`ST TUB160x120x3`
		or
		`ST TUBE TH 0.003 WT 0.12 DT 0.16`

Table 208: Compound Sections for Russian Steel Design

Section	Section Type	Designation form
Double channels		`D C14 SP 0.01` (SP – clear distance between channel walls)
Double equal legs angles		`LD L100x100x7 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with long legs back to back		`LD L125x80x10 SP 0.01` (SP – clear distance between angle walls)
Double unequal legs angles with short legs back to back		`SD L125x80x10 SP 0.01` (SP – clear distance between angle walls)

Section	Section Type	Designation form

Tee with flange at the top
Note: Flange of Tee beam is at the top part of cross-section if beta angle = 0°, or at the bottom part if beta angle = 180°.
		`T I12`
		`T B1-10`
		`T SH1-23`
		`T K1-20`

For entry of cross-sectional dimensions command `MEMBER PROPERTIES RUSSIAN` is used.

Refer to D13.B.2 Built-in Russian Steel Section Library (on page 1890) for an example.

D13.E.3.2 Standard European Profiles

The following profile types are supported using the `ENSGR` parameter to specify steel grade.

- Wide Flange: IPE, HD, HE, HL, HP, DIL, IPE
- Channel: U, UPE, UPN
- Angle: L
- Circular Hollow: CHC, Pipe
- Rectangular Hollow: RHS, SHS, Tube

D13.E.3.3 Other Supported Standard Profiles

The following profiles are supported using the `PY` parameter to specify steel yield strength in place of steel grade.

- Wide Flange
- Channel
- Angle
- Circular Hollow
- Rectangular Hollow

D13.E.3.4 User Table Profiles

The following UPT shapes are supported either using the `PY` parameter to specify steel yield strength or the `SGR` or `ENSGR` in place of steel grade.

- Wide Flange
- Channel
- Angle
- Double Angle
- Tee
- Pipe
- Tube

Not supported: ISection, General, or Prismatic

D13.E.4 Design Parameters

Table 209: Design parameters for design of steel members per SP 16.13330.2017

Parameter Name	Default Value	Description
CODE	-	Must be specified as: `RUSSIAN` or `RUSSIAN 2017` Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BMT		Bimoment for design of eccentrically loaded members per clause 8.4 or members subjected to two bending moments per clause 8.2.
CB	1	Location of loading on the beam: 1) top flange 2) bottom flange
CMM	0	Slenderness limit for tension members: 0) suppress slenderness check 1) 150 2) 200 3) 250 4) 300 5) 350 6) 400 **Note:** Slenderness check for member is performed against `CMM` or `CMN` parameters depending on the tension/compression nature of axial force. If the member is not subjected to axial force, slenderness check is not performed as there is no relevant clause in the SP 16.13330.2017.

Parameter Name	Default Value	Description
CMN	0	Ultimate slenderness for compression members. Limits as per SP 16.13330.2017 clause 10.4, Table 32: 0) suppress slenderness check 1) 180-60a 2) 120 3) 210-60a 4) 220-40a 5) 220 6) 180-60a 7) 210-60a 8) 200 9) 150 where a = alpha calculated as defined in SP 16.13330.2017 Table 32. **Note:** Slenderness check for member is performed against `CMM` or `CMN` parameters depending on the tension/compression nature of axial force. If the member is not subjected to axial force, slenderness check is not performed as there is no relevant clause in the SP 16.13330.2017.
DFF	None	"Deflection Length" / Maximum allowable local deflection
DJ1	Start Joint of member	
DJ2	End Joint of member	
DMAX	1,000 in	Maximum allowable section depth
DMIN	0	Minimum allowable section depth
ENSGR	1	The number of the steel grade from Table 3.1 in EN 1993-1-1: 2005. D5.C.6 Design Parameters (on page 1502) for values. Also, see Note 1 below.
FU	0	Design steel strength (ultimate strength) **Note:** Used when the `SRG` parameter is not defined.
GAMC1	1.0	Specific service condition coefficient for buckling design
GAMC2	1.0	Specific service condition coefficient for strength design

Parameter Name	Default Value	Description
GAMM	1.0	Specific partial coefficient for steel, γ_m, for European or Russian steel. Refer to Table 3 in the SP 16.13330.2017 code. 1) Gamma(n) = 1.025 2) Gamma(n) = 1.05
GMF	1.0	Ratio between design and characteristic load value
KY	1.0	Coefficient of effective length in respect to local axis Y (in plane XZ)
KZ	1.0	Coefficient of effective length in respect to local axis Z (in plane XY)
LEG	4	Describes the type of and position of loading on the beam: 1) concentrated in middle of the span 2) concentrated in the quarter of the span 3) concentrated at the end of bracket 4) uniformly distributed along beam 5) uniformly distributed on bracket
LY	Member Length	Effective length in respect to local axis Y (in plane XZ)
LZ	Member Length	Effective length in respect to local axis Z (in plane XY)
NSF	1.0	Net section factor for tension members or web section area weakening factor for bending members
PY	0	Design steel strength (yield strength) **Note:** Used when parameters `MAIN` and `SGR` are not defined for Russian steel materials or `ENMAIN` and `ENSGR` for European steel materials. See Note 1 below.
SBLT	0	Number of lateral bracing restraints along the span: 0) beam is not fixed laterally 1) single restraint at mid-span 2) or higher is the number of uniformly spaced lateral restraints along the span

Parameter Name	Default Value	Description
SGR	1	Steel grade: See Note 1 below. Grades from Table C5: hot-rolled profiles except of type K, SH, and B of I section beams (See Note 2 below) 1) S245* (4-40 mm) 2) S255* (4-40 mm) 3) S345* (4-40 mm) 4) S345K* (4-10 mm) 5) S355*; S355-1* (8-40 mm) 6) S390* (8-40 mm) Grades from Table C3: sheets, wide strips, plates, pipes, and tubes 7) S235 (2-4 mm) 8) S245 (2-20 mm) 9) S255 (2-40 mm) 10) S345 (2-160 mm) 11) S345K (4-10 mm) 12) S355 (8-160 mm) 13) S355-1; S355-K (8-60 mm) 14) S355P (8-40 mm) 15) S390; S390-1 (8-50 mm) 16) S440 (8-50 mm) 17) S550 (8-50 mm) 18) S590 (8-50 mm) 19) S690 (8-50 mm) Grades from Table C4: I-beams with parallel flange faces, type K, SH, and B 20) S255B; S255B-1 (0-100 mm) 21) S345B (0-60 mm) 22) S345B-1 (0-60 mm) 23) S355B (0-100 mm) 24) S355B-1 (0-60 mm) 25) S390B (31-100 mm) 26) S440B (0->100 mm)
STP	1	Section type: 1) Rolled section 2) Welded section

Parameter Name	Default Value	Description
TB	1	Indication of stress strain state per Cl 4.2.7: 1) 1st Class (Elastic) 2) 2nd Class (Elasto-plastic) 3) 3rd Class (Plastic) The engineer shall consider the appropriate class of the member, as it is used in multiple checks under Ch.8. for elastic or elastic-plastic calculation for bending members (non-axially compressed or tensioned): Cl 8.2.1, Cl. 8.2.3, Cl. 8.4.1, Cl. 8.4.4, and Cl 8.4.6.
TRACK	0	Output details: 0) design summary only 1) extended output information for the critical section under the critical load case 2) full output information for the critical section under the critical load case 3) design summary only (Russian language) 4) extended output information for the critical section under the critical load case (Russian language) 5) full output information for the critical section under the critical load case (Russian language)

Notes

1. It is common practice to design only Russian or European shapes to the Russian steel codes. For Russian steel sections, the steel design strength value, Ry, is obtained from Tables C.3, C.4, and C.5 of SP 16.13330.2017 determined by the assigned the SGR parameter. For European sections, the ENSGR should be used instead. If steel sections from other countries are used, the PY parameter should be used to specify the steel strength.
2. SGR parameter values marked with an “*” are from Table C.5. These grades are also repeated in Table C.3 with different labels.

D13.E.5 Member Selection and Code Check

Both code checking and member selection options are available in SP 16.13330.2017.

Output of selection and check results are given in suppressed, extended and advanced forms. Form of output results depends on value of parameter TRACK.

Results are presented in tables. Three versions of output results are possible: suppressed – results according the critical strength condition (TRACK=0), extended - results according to all check conditions (TRACK=1) and advanced – complete information on results of member design (TRACK=2).

In tables of results common data for all TRACKs are indicated:

(TRACK=2)

In tables of results common data for all TRACKs are indicated:

- number of member;
- type and number of cross-section;
- result obtained (`ACCEPTED` – requirements are met, `FAILURE` – are not met);
- abbreviated name of normative document (code, standard) (SNiP);
- number of check clause;
- safety of strength (ratio between design and normative values);
- number of the most unfavorable loading;
- value of longitudinal force acting in the member with subscript indicating its direction ("C" – compression, "T" – tension);
- bending moments in relation to local member axes Z and Y;
- distance to section, in which the most unfavorable combination of forces acts.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

Note: The output for the Russian code uses the axes system from STAAD.Pro, except where details of the equations from the code are printed, in which case the axes system is as defined in the code.

D13.E.5.1 Example of TRACK 0 output

In suppressed form (`TRACK 0`) results are presented according to the critical check for given member with indication of SP 16 clause number, according to which strength safety of the member is minimum.

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************

   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4        PASS      SP cl.9.1.1      0.23      3981
                          1.186E+00 C    9.442E+02    5.044E+00   1.000E+00
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4        PASS         DISPL         0.01      3981
                          1.186E+00 C    9.442E+02    5.044E+00   5.833E-01
   1820  I      SH3-50         PASS      SP cl.9.1.1      0.41      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50         PASS         DISPL         0.31      3655
                          1.627E+02 C    1.435E+00    7.986E+01   1.667E+00
   1820  I      SH3-50         PASS      SP cl.10.4.1C    0.33      3655
                          1.627E+02 C    1.435E+00   -7.986E+01   0.000E+00
```

D13.E.5.2 Example of TRACK 1 output

In extended form (TRACK 1) results are presented on the basis of all required by SP 16 checks for given stress state.

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************

   ALL UNITS ARE - KN METRE
   ===========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ===========================================================================
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4       PASS      SP cl.9.1.1      0.23      3981
                          1.186E+00 C    9.442E+02    5.044E+00   1.000E+00
    384  I      SH100-4       PASS      SP cl.8.2.3      0.21      3981
                          1.186E+00 C    9.442E+02    5.044E+00   1.000E+00
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4       PASS        DISPL          0.01      3981
                          1.186E+00 C    9.442E+02    5.044E+00   5.833E-01
   1820  I      SH3-50        PASS      SP cl.9.1.1      0.41      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.2      0.03      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.4      0.03      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.9      0.03      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.8.2.3      0.34      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.8.4.1      0.39      3655
                          1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS        DISPL          0.31      3655
                          1.627E+02 C    1.435E+00    7.986E+01   1.667E+00
   1820  I      SH3-50        PASS      SP cl.10.4.1C    0.33      3655
                          1.627E+02 C    1.435E+00   -7.986E+01   0.000E+00
```

D13.E.5.3 Example of a TRACK 2 output

In advanced form (TRACK=2) in addition to tabled results supplementary information is presented.

Material characteristics:

Steel;

Design resistance;

Elasticity modulus;

Section characteristics:

Length of member;

Section area;

Net area;

Inertia moment (second moment of area) (I);
Section modulus (W);
First moment of area (S);
Radius of gyration;
Effective length;
Slenderness;
Results are presented in two columns, Z and Y respectively.
Design forces:
Longitudinal force;
Moments;
Shear force.

Signs “+” and “-” indicate direction of acting longitudinal force, bending moments and shear forces in accordance with sign rules assumed in program STAAD.Pro.

Check results in advanced form are presented with values of intermediate parameters by formulas in analytical and numerical expression with indication of SP code clause.

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************

   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4       PASS      SP cl.9.1.1      0.23      3981
                          1.186E+00 C    9.442E+02    5.044E+00   1.000E+00
    384  I      SH100-4       PASS      SP cl.8.2.3      0.21      3981
                          1.186E+00 C    9.442E+02    5.044E+00   1.000E+00
*    384 GENERAL STABILITY CAN NOT BE CHECKED
     FACTOR "ALPHA" < 0,1
            (look at appendix G, table G.1) * Element is too short for checking the
stability  *
    384  I      SH100-4       PASS         DISPL         0.01      3981
                          1.186E+00 C    9.442E+02    5.044E+00   5.833E-01

   MATERIAL DATA
      Steel                          = C345B      SP16.13330
      Modulus of elasticity          = 206.E+06 kPa
      Design Strength (Ry)           = 335.E+03 kPa

   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                  = 1.00E+00
      Gross Area                     = 4.01E-02
      Net Area                       = 4.01E-02
                                          x-axis      y-axis
      Moment of inertia (I)          :   655.E-05    178.E-06
      Section modulus (W)            :   129.E-04    111.E-05
      First moment of area (S)       :   729.E-05    877.E-06
      Radius of gyration (i)         :   405.E-03    667.E-04
      Effective Length               :   1.00E+00    1.00E+00
```

```
      Slenderness                   :   247.E-02    150.E-01

   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   118.6E-02
                                         x-axis      y-axis
      Moments                       :   944.2E+00    504.4E-02
      Shear force                   :   410.6E-02    946.7E+00
      Bi-moment                     :   0.000E+00 Value of Bi-moment not being
entered!!!

   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(106) (N/A+Mx*y/Ix+My*x/Iy+B*w/Iw)/(Ry*GammaC)=
              ( 118.6E-02/ 4.0E-02+ 944.2E+00* 5.07E-01/ 6.55E-03+ 504.4E-02* 1.60E-01/
               1.78E-04+ 0.000E+00* 2.50E-01/ 4.27E-05)/( 335.0E+03* 1.00E+00)
               = 2.31E-01=<1
       m_x =24.6E+02>20.    m_y =15.3E+01>20.
      F.(51)  Mx/(Cx*beta*Wxn,min*Ry*GammaC)+My/(Cy*Wyn,min*Ry*GammaC)=
               944.2E+00/( 1.11E+00* 9.99E-01* 1.29E-02* 335.0E+03* 1.00E+00)+
               504.4E-02/( 1.15E+00* 1.11E-03* 335.0E+03* 1.00E+00)=
               2.07E-01=<1
              TAUX=Qx/Aw= 946.7E+00/ 197.5E-04= 479.E+02 <= 0,9*RS= 174.9E+03
              TAUy=Qy/Af= 410.6E-02/ 208.0E-04= 197.E+00 <= 0,5*RS= 971.5E+02
          LIMIT SPAN/DEFLECTION (DFF) =    250.00   (DEFLECTION LIMIT=      0.004 M)
          SPAN/DEFLECTION = 177.2E+02 (DEFLECTION=  5.643E-05M)
          LOAD= 3981     RATIO=    0.014     LOCATION=    0.583


                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************

   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
   1820  I      SH3-50        PASS      SP cl.9.1.1      0.41      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.2      0.03      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.4      0.03      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.9.2.9      0.03      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.8.2.3      0.34      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS      SP cl.8.4.1      0.39      3655
                           1.627E+02 C    1.435E+00    7.986E+01   0.000E+00
   1820  I      SH3-50        PASS         DISPL         0.31      3655
                           1.627E+02 C    1.435E+00    7.986E+01   1.667E+00
   1820  I      SH3-50        PASS      SP cl.10.4.1C    0.33      3655
                           1.627E+02 C    1.435E+00   -7.986E+01   0.000E+00

   MATERIAL DATA
      Steel                         = C345B      SP16.13330
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 335.E+03 kPa

   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
```

```
      Member Length                 = 4.00E+00
      Gross Area                    = 1.99E-02
      Net Area                      = 1.99E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   842.E-06    925.E-07
      Section modulus (W)           :   340.E-05    617.E-06
      First moment of area (S)      :   192.E-05    462.E-06
      Radius of gyration (i)        :   206.E-03    681.E-04
      Effective Length              :   6.00E+00    4.00E+00
      Slenderness                   :   292.E-01    587.E-01

   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   162.7E+00
                                         x-axis      y-axis
      Moments                       :   143.5E-02    798.6E-01
      Shear force                   :   126.7E-01    400.1E-02
      Bi-moment                     :   0.000E+00 Value of Bi-moment not being
entered!!!

   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(106) (N/A+Mx*y/Ix+My*x/Iy+B*w/Iw)/(Ry*GammaC)=
              ( 162.7E+00/ 2.0E-02+ 143.5E-02* 2.48E-01/ 8.42E-04+ 798.6E-01* 1.50E-01/
               9.25E-05+ 0.000E+00* 2.50E-01/ 5.19E-06)/( 335.0E+03* 1.00E+00)
               = 4.12E-01=<1
      P.9.2.2  m_ef=eta*mx= 0.00E+00* 5.17E-02= 0.00E+00
      F.(109)  N/(FIe*A*Ry*GammaC)= 162.7E+00/( 9.07E-01* 1.99E-02* 335.0E+03* 1.00E
+00)
               = 2.69E-02=<1
      F.(112)  C=beta/(1+alfa*mx)= 1.00E+00/(1+7.00E-01* 5.17E-02)= 9.65E-01
                    Cmax= 9.99E-01
      F.(111)  N/(c*FIy*A*Ry*GammaC)
               = 0.16E+03/( 0.97E+00* 0.77E+00* 0.20E-01* 335.0E+03* 1.00E+00)
               = 3.30E-02=<1
      F.(117)  FIexy=FIey*(0.6*C**(1/3)+0.4*C**(1/4))=
               7.61E-01*(0.6*9.65E-01**(1/3)+0.4*9.65E-01**(1/4))= 7.53E-01
               N/(FIexy*A*Ry*GammaC)= 162.7E+00/( 7.53E-01* 1.99E-02* 335.0E+03*1.00E
+00)
               =3.24E-02=<1

       m_y =24.8E+00>20.
      F.(51)  Mx/(Cx*beta*Wxn,min*Ry*GammaC)+My/(Cy*Wyn,min*Ry*GammaC)=
               143.5E-02/( 1.08E+00* 1.00E+00* 3.40E-03* 335.0E+03* 1.00E+00)+
               798.6E-01/( 1.15E+00* 6.17E-04* 335.0E+03* 1.00E+00)=
               3.37E-01=<1
              TAUX=Qx/Aw= 400.1E-02/ 767.2E-05= 521.E+00 <= 0,9*RS= 174.9E+03
              TAUy=Qy/Af= 126.7E-01/ 123.0E-04= 103.E+01 <= 0,5*RS= 971.5E+02
      F.(70)  Mx/(FIb*Wcx*Ry*GammaC)+My/(Wy*Ry*GammaC)+
              B/(Ww*Ry*GammaC)=
               1.44E+00/( 1.00E+00* 3.40E-03* 335.0E+03* 1.00E+00)+
               7.99E+01/( 6.17E-04* 335.0E+03* 1.00E+00)+
               0.00E+00/( 2.08E-05* 335.0E+03* 1.00E+00)= 3.88E-01=<1
          LIMIT SPAN/DEFLECTION (DFF) =    250.00   (DEFLECTION LIMIT=      0.016 M)
          SPAN/DEFLECTION = 809.8E+00 (DEFLECTION=  4.939E-03M)
          LOAD= 3655     RATIO=    0.309     LOCATION=    1.667
      ULTIM. SLENDERNESS >= Lambda_y 180.0E+00 >= 587.0E-01
```

Conventional notations assumed in presentation of results: “+”, “-”, “/”, “*”,“**”, “SQRT”, their respective meanings (i.e., addition, subtraction, division, multiplication, raising to the second power (squared), and square root). Conventional notations of stresses, coefficients and characteristics of steel resistance comply with accepted in the SP standard. Only Greek letters are changed by their names (e.g., , γ_c-GAMAC; α-ALPHA; β-BETA, η-ETA, φ-PHI, etc.).

D14. South African Codes

D14.A. South African Codes - Concrete Design per SABS-0100-1

STAAD.Pro is capable of performing concrete design based on the South African code SABS-0100-1 2000 *Code of Practice for Structural Use of Concrete Part1: Design*. Design can be performed for beams (flexure, shear, and torsion) and columns (axial load + biaxial bending). Given the width and depth (or diameter for circular columns) of a section, the program calculates the required reinforcement.

D14.A.1 Design Parameters

The program contains a number of parameters which are needed to perform and control the design to SABS 0100-1. These parameters not only act as a method to input required data for code calculations but give the engineer control over the actual design process. Default values of commonly used parameters for conventional design practice have been chosen as the basis. Table 17A.1 contains a complete list of available parameters with their default values.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 210: South African Concrete Design SABS 0100-1 Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified as `SABS0100`. Design code to follow. See TR.53.2 Concrete Design-Parameter Specification (on page 2685).
BRACE	0.0	Column bracing: 0. Column braced in both directions. 1. Column braced about local Y direction only 2. Column unbraced about local Z direction only 3. Column unbraced in both Y and Z directions
CLB	20mm	Clear Cover for outermost bottom reinforcement

Parameter Name	Default Value	Description
CLS	20mm	Clear Cover for outermost side reinforcement
CLT	20mm	Clear Cover for outermost top reinforcement
DEPTH	YD	Depth of concrete member, in current units. This value default is as provided as YD in MEMBER PROPERTIES.
ELY	1.0	Member length factor about local Y direction for column design.
ELZ	1.0	Member length factor about local Z direction for column design.
FC	30N/mm^2	Concrete Yield Stress / cube strength, in current units.
FYMAIN	450 N/mm^2	Yield Stress for main reinforcement, in current units.
FYSEC	450N/mm^2	Yield Stress for secondary reinforcement a, in current units. Applicable to shear bars in beams
MAXMAIN	50mm	Maximum required reinforcement bar size Acceptable bars are per MINMAIN above.
MINMAIN	8mm	Minimum main reinforcement bar size Acceptable bar sizes: 6 8 10 12 16 20 25 28 32 36 40 50 60
MINSEC	8mm	Minimum secondary bar size a. Applicable to shear reinforcement in beams

Parameter Name	Default Value	Description
TRACK	0.0	Output detail 0. Critical Moment will not be printed with beam design report. Column design gives no detailed results. 1. For beam gives min/max steel % and spacing. For columns gives a detailed table of output with additional moments calculated. 2. Output of TRACK 1.0 List of design sag/hog moments and corresponding required steel area at each section of member
WIDTH	ZD	Width of concrete member, in current units. This value default is as provided as ZD in MEMBER PROPERTIES.

D14.A.2 Member Dimensions

Concrete members that are to be designed by STAAD.Pro must have certain section properties input under the `MEMBER PROPERTIES` command.

The following example demonstrates the required input:

```
UNIT  MM
MEMBER  PROPERTIES
*RECTANGULAR  COLUMN 300mm WIDE X 450mm DEEP
1 3  TO 7 9 PRISM YD 450. ZD 300.
*CIRCULAR  COLUMN 300mm diameter
11 13  PR YD 300.
* T-SECTION  - FLANGE 1000.X 200.(YD-YB)
* - STEM 250(THICK) X 350.(DEEP)
14 PRISM  YD 550. ZD 1000. YB 350. ZB 250.
```

In the above input, the first set of members are rectangular (450mm depth x 300mm width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 300mm diameter. Note that area (AX) is not provided for these members. If shear area areas (AY & AZ) are to be considered in analysis, the user may provide them along with YD and ZD. Also note that if moments of inertias are not provided, the program will calculate them from YD and ZD. Finally a T section can be considered by using the third definition above.

D14.A.3 Beam Design

Beam design includes flexure, shear and torsion. For all types of beam action, all active beam loadings are scanned to create moment and shear envelopes and locate the critical sections. The total number of sections considered is thirteen. From the critical moment values, the required positive and negative bar pattern is developed. Design for flexure is carried out as per clause no. 4.3.3.4.

Shear design as per SABS 0100 clause 4.3.4 has been followed and the procedure includes computation of critical shear values. From these values, stirrup sizes are calculated with proper spacing. If torsion is present, the program will also consider the provisions of SABS 0100 clause 4.3.5. Torsional reinforcement is separately reported.

A `TRACK 2` design output is presented below.

```
==============================================================================
         B E A M  N O.      4   D E S I G N  R E S U L T S


     M30                     Fe450 (Main)                Fe450 (Sec.)


    LENGTH:  6000.0 mm      SIZE:   715.0 mm X  380.0 mm   COVER: 40.0 mm




     DESIGN LOAD SUMMARY (KN MET)
     ----------------------------------------------------------------------------
     SECTION |FLEXURE  (Maxm. Sagging/Hogging moments)|          SHEAR
     (in mm) |    MZ    Load Case     MX   Load Case  |    VY       P   Load Case
     ----------------------------------------------------------------------------
     0.0     |     84.77       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     500.0   |     70.70       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     1000.0  |     56.64       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     1500.0  |     42.57       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     2000.0  |     28.50       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     2500.0  |     14.43       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     3000.0  |      0.37       1       -9.89      1   |  -28.13     4.39      1
             |      0.00       0                      |
     3500.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -13.70       1                      |
     4000.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -27.77       1                      |
     4500.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -41.84       1                      |
     5000.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -55.90       1                      |
     5500.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -69.97       1                      |
     6000.0  |      0.00       0       -9.89      1   |  -28.13     4.39      1
             |    -84.04       1                      |
     ----------------------------------------------------------------------------



     SUMMARY OF REINF. AREA FOR FLEXURE DESIGN (Sq.mm)
```

```
-----------------------------------------------------------------------------
SECTION |            TOP            |           BOTTOM          |   STIRRUPS
(in mm) | Reqd./Provided reinf.     | Reqd./Provided reinf.     |  (2 legged)
-----------------------------------------------------------------------------
0.0     |  543.40/ 549.78( 7-10í )|  680.71/ 706.86( 9-10í )|  8í  @ 115 mm
500.0   |  543.40/ 549.78( 7-10í )|  567.75/ 603.18( 3-16í )|  8í  @ 115 mm
1000.0  |  543.40/ 549.78( 7-10í )|  454.79/ 471.24( 6-10í )|  8í  @ 115 mm
1500.0  |  543.40/ 549.78( 7-10í )|  353.21/ 392.70( 5-10í )|  8í  @ 115 mm
2000.0  |  543.40/ 549.78( 7-10í )|  353.21/ 392.70( 5-10í )|  8í  @ 115 mm
2500.0  |  543.40/ 549.78( 7-10í )|  353.21/ 392.70( 5-10í )|  8í  @ 115 mm
3000.0  |  543.40/ 549.78( 7-10í )|  353.21/ 392.70( 5-10í )|  8í  @ 115 mm
3500.0  |  353.21/ 392.70( 5-10í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
4000.0  |  353.21/ 392.70( 5-10í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
4500.0  |  353.21/ 392.70( 5-10í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
5000.0  |  448.91/ 452.40( 4-12í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
5500.0  |  561.87/ 565.50( 5-12í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
6000.0  |  674.83/ 678.60( 6-12í )|  543.40/ 549.78( 7-10í )|  8í  @ 115 mm
-----------------------------------------------------------------------------


TORSION REINFORCEMENT : Not required
```

D14.A.4 Column Design

Columns are designed for axial force and biaxial bending at the ends. All active loadings are tested to calculate reinforcement. The loading which produces maximum reinforcement is called the critical load and is displayed. The requirements of SABS 0100-1 clause 4.7 are followed, with the user having control on the effective length in each direction by using the ELZ and ELY parameters as described in table 12A.1. Bracing conditions are controlled by using the BRACE parameter. The program will then decide whether or not the column is short or slender and whether it requires additional moment calculations. For biaxial bending, the recommendations of 4.7.4.4 of the code are considered.

Column design is done for square, rectangular and circular sections. For rectangular and square sections, the reinforcement is always assumed to be arranged symmetrically. This causes slightly conservative results in certain cases.

Using parameter `TRACK 1.0`, the detailed output below is obtained. `TRACK 0.0` would merely give the bar configuration, required steel area and percentage, column size and critical load case.

```
============================================================================

     C O L U M N   N O.      1  D E S I G N   R E S U L T S

     M30                   Fe450 (Main)              Fe450 (Sec.)

     LENGTH:  3000.0 mm   CROSS SECTION:  715.0 mm X  380.0 mm  COVER: 40.0 mm

     ** GUIDING LOAD CASE:    1 END JOINT:     2  SHORT COLUMN


     DESIGN FORCES (KNS-MET)
     -----------------------
     DESIGN AXIAL FORCE (Pu)             :   -14.6

     About Z         About Y
     INITIAL MOMENTS                     :    0.00            0.00
     MOMENTS DUE TO MINIMUM ECC.         :    0.28            0.29
```

```
        SLENDERNESS RATIOS                :    7.89            4.20
        ADDITION MOMENTS (Maddz and Maddy)  :    0.00            0.00

        TOTAL DESIGN MOMENTS                :   45.17            9.41

        REQD. STEEL AREA   :     26.99 Sq.mm.
        REQD. CONCRETE AREA:        1213.61 Sq.mm.
        MAIN REINFORCEMENT : Provide   4 - 12 dia. (0.17%,    452.40 Sq.mm.)
        (Equally Distributed)
        TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 140 mm c/c

        SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
        ----------------------------------------------------------
        Puz :     25.50   Muz1 :     45.22   Muy1 :     51.48


        ============================================================================
```

D14.B. South African Codes - Steel Design per SANS10162-1:1993

The South African Steel Design facility in STAAD.Pro is based on the SAB Standard SAB0162-1: 1993, *Limit States Design of Steel Structures*. A steel section library consisting of South African Standards shapes is available for member property specification.

D14.B.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The next few sections describe the salient features of the STAAD implementation of SAB0162-1: 1993. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D14.B.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. The user is allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Refer to TR.37 Analysis Specification (on page 2620) for additional information.

D14.B.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G.6 Member Properties (on page 2113) .

D14.B.4 Built-in Steel Section Library

A steel section library consisting of South African Standards shapes is available for member property specification.

The following information is provided for use when the built-in steel tables are to be referenced for member property specification. These properties are stored in a database file. If called for, the properties are also used for member design. Since the shear areas are built into these tables, shear deformation is always considered during the analysis of these members.

Refer to G.6.2 Built-In Steel Section Libraries (on page 2116) for additional information.

D14.B.4.1 I Shapes

The following example illustrates the specification of I- shapes.

```
1 TO 15 TABLE ST IPE-AA100
```

D14.B.4.2 H shapes

Designation of H shapes in STAAD is as follows.

For example,

```
18 TO 20 TABLE ST 152X37UC
```

D14.B.4.3 PG shapes

Designation of PG shapes in STAAD is as follows.

```
100 TO 150 TABLE ST 720X200PG
```

D14.B.4.4 Channel Sections (C & MC shapes)

C and MC shapes are designated as shown in the following example.

```
3 TABLE ST 127X64X15C
```

D14.B.4.5 Double Channels

Back to back double channels, with or without spacing between them, are specified by preceding the section designation by the letter D. For example, a back to back double channel section `PFC140X60` without spacing in between should be specified as:

```
100 TO 150 TABLE D PFC140X60
```

A back-to-back double channel section 140X60X16C with spacing 0.01 unit length in between should be specified as:

```
100 TO 150 TABLE D 140X60X16C  SP 0.01
```

Note: The specification SP after the section designation is used for providing the spacing. The spacing should always be provided in the current length unit.

D14.B.4.6 Angles

To specify angles, the letter L succeeds the angle name. Thus, a 70X70 angle with a 25mm thickness is designated as 70X70X8L. The following examples illustrate angle specifications.

```
100 TO 150 TABLE ST 70X70X8L
```

Note: This specification is for "standard" angles, designated by ST. In this specification, the local z-axis corresponds to the Y'-Y' axis shown in the section table. Another common practice of specifying angles assumes the local y-axis to correspond to the Y'-Y' axis. To specify angles in accordance with this convention, use the reverse angle designation, RA. Refer to G.4.2 Local Coordinate System (on page 2088) for details on the local axis systems for standard (ST) and reverse (RA) angles.

Refer to the following example for details.

```
100 TO 150 TABLE RA 45X45X3L
```

D14.B.4.7 Double Angles

To specify double angles, the specification ST should be substituted with LD (for long leg back-to-back) or SD (short leg back-to-back). For equal angles, either SD or LD will serve the purpose. Spacing between angles may be provided by using the word SP followed by the value of spacing (in current length unit) after section designation.

```
100 TO 150 TABLE LD 50X50X3L
3 TABLE LD 40X40X5L SP 0.01
```

The second example above describes a double angle section consisting of 40X40X5 angles with a spacing of 0.01 length units.

D14.B.4.8 Tees

Tee sections obtained by cutting W sections may be specified by using the T specification instead of ST before the name of the W shape. For example:

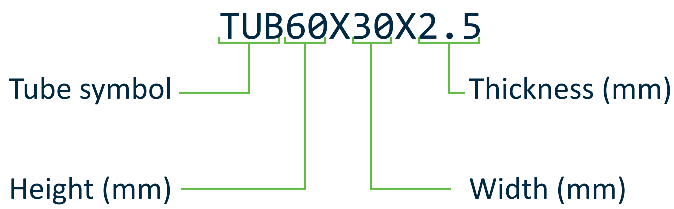
```
100 TO 150 TABLE T IPE-AA180
```

will describe a T section cut from a IPE-AA180 section.

D14.B.4.9 Rectangular Hollow Sections

These sections may be specified in two possible ways. Those sections listed in the SAB tables may be specified as follows.

```
100 TO 150 TABLE ST TUB60X30X2.5
```



In addition, any tube section may be specified by using the DT(for depth), WT(for width), and TH(for thickness) specifications. For example:

```
100 TO 150 TABLE ST TUBE  TH 3 WT 100 DT 50
```

will describe a tube with a depth of 50mm, width of 100mm. and a wall thickness of 3mm. Note that the values of depth, width and thickness must be provided in current length unit.

D14.B.4.10 Circular Hollow Sections

Sections listed in the SAB tables may be provided as follows:

```
100 TO 150 TABLE ST PIP34X3.0CHS
```

In addition to sections listed in the SAB tables, circular hollow sections may be specified by using the OD (outside diameter) and ID (inside diameter) specifications.

For example:

```
100 TO 150 TABLE ST PIPE  OD 50 ID 48
```

will describe a pipe with an outside diameter of 50 length units and inside diameter of 48 length units. Note that the values of outside and inside diameters must be provided in terms of current length unit.

D14.B.4.11 Example

A sample input file to demonstrate usage of South African shapes:

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 30-Mar-05
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 9 0 0; 3 0 6 0; 4 3 6 0; 5 6  6 0; 6 9 6 0; 7 0 10.5 0;
8 9 10.5 0; 9 2.25 10.5 0; 10 6.75 10.5  0; 11 4.5 10.5 0; 12 1.5 11.4 0;
13 7.5 11.4 0; 14 3 12.3 0; 15 6 12.3 0; 16 4.5 13.2 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4  5; 7 5 6; 8 7 12; 9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14; 16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21  9 11; 22 10 11; 23 8 10;
MEMBER PROPERTY SAFRICAN
```

```
1 TABLE ST IPE-AA100
2 TABLE T IPE120
3 TABLE ST 152X23UC
4 TABLE T 152X23UC
5 TABLE ST 812X200PG
6 TABLE T 812X200PG
7 TABLE ST 178X54X15C
8 TABLE D 178X54X15C
9 TABLE D 178X54X15C SP 0.1
10 TABLE ST 25X25X5L
11 TABLE RA 25X25X5L
12 TABLE LD 25X25X5L
13 TABLE SD 25X25X5L
14 TABLE LD 25X25X5L SP 0.1
15 TABLE SD 25X25X5L SP 0.1
16 TABLE ST TUB40X2.5SHS
17 TABLE ST TUBE TH 0 WT 0 DT 50
18 TABLE ST TUBE TH 0.02 WT 100 DT 50
20 TABLE ST PIP48X2.0CHS
21 TABLE ST PIPE OD 0.5 ID 0.48
PRINT MEMBER PROPERTIES
FINISH
```

D14.B.5 Section Classification

The SAB specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of SAB0162-1:1993). This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes. Design is performed for sections that fall into the category of Class 1,2, or 3 sections only. Class 4 sections are not designed by STAAD.Pro.

D14.B.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members' unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

All the members are checked against allowable slenderness ratio as per Cl.10.2 of SAB0162-1: 1993.

D14.B.6.1 Axial

Parameters FYLD, FU, and NSF are applicable for these calculations.

D14.B.6.2 Axial Compression

The compressive resistance of columns is determined based on Clause 13.3 of the code. The equations presented in this section of the code assume that the compressive resistance is a function of the compressive strength of the gross section (Gross section Area times the Yield Strength) as well as the slenderness factor (KL/r ratios). The effective length for the calculation of compression resistance may be provided through the use of the parameters KX, KY, KZ, LX, LY, and LZ (see D14.B.7 Design Parameters (on page 1960)). Some of the aspects of the axial compression capacity calculations are:

1. For frame members not subjected to any bending, and for truss members, the axial compression capacity in general column flexural buckling is calculated from Cl.13.3.1 using the slenderness ratios for the local Y-Y and Z-Z axis. The parameters KY, LY, KZ, and LZ are applicable for this.
2. For single angles, asymmetric or cruciform sections are checked as to whether torsional-flexural buckling is critical. But for KL/r ratio exceeding 50,as torsional flexural buckling is not critical, the axial compression capacities are calculated by using Cl.13.3. The reason for this is that the South African code doesn't provide any clear guidelines for calculating this value. The parameters KY, LY, KZ, and LZ are applicable for this.
3. The axial compression capacity is also calculated by taking flexural-torsional buckling into account. Parameters KX and LX may be used to provide the effective length factor and effective length value for flexural-torsional buckling. Flexural-torsional buckling capacity is computed for single channels, single angles, Tees and Double angles.
4. While computing the general column flexural buckling capacity of sections with axial compression + bending, the special provisions of 13.8.1(a), 13.8.1(b) and 13.8.1(c) are applied. For example, Lambda = 0 for 13.8.1(a), K=1 for 13.8.1(b), etc.)

D14.B.6.3 Bending

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified in STAAD with the help of the parameter UNL. If UNL is less than one tenth the member length (member length is the distance between the joints of the member), the member is treated as being continuously laterally supported. In this case, the moment resistance is computed from Clause 13.5 of the code. If UNL is greater than or equal to one-tenth the member length, its value is used as the laterally unsupported length. The equations of Clause 13.6 of the code are used to arrive at the moment of resistance of laterally unsupported members. Some of the aspects of the bending capacity calculations are:

1. The weak axis bending capacity of all sections except single angles is calculated as:

   For Class 1 & 2 sections

   $$Phi*Py*Fy$$

   For Class 3 sections

   $$Phi*Sy*Fy$$

   Where:

   Phi = Resistance factor = 0.9
   Py = Plastic section modulus about the local Y axis
   Sy = Elastic section modulus about the local Y axis
   Fy = Yield stress of steel
2. Single angles sections are not designed by STAAD, as the South African code doesn't provide any clear guidelines for calculating this value.
3. For calculating the bending capacity about the Z-Z axis of singly symmetric shapes such as Tees and Double angles, SAB0162-1: 1993 stipulates in Clause 13.6(b), page 31, that a rational method.

D14.B.6.4 Axial compression and bending

The member strength for sections subjected to axial compression and uniaxial or biaxial bending is obtained through the use of interaction equations. In these equations, the additional bending caused by the action of the axial load is accounted for by using amplification factors. Clause 13.8 of the code provides the equations for this purpose. If the summation of the left hand side of these equations exceeds 1.0 or the allowable value provided using the RATIO parameter (see D14.B.7 Design Parameters (on page 1960)), the member is considered to have FAILed under the loading condition.

D14.B.6.5 Axial tension and bending

Members subjected to axial tension and bending are also designed using interaction equations. Clause 13.9 of the code is used to perform these checks. The actual RATIO is determined as the value of the left hand side of the critical equation.

D14.B.6.6 Shear

The shear resistance of the cross section is determined using the equations of Clause 13.4 of the code. Once this is obtained, the ratio of the shear force acting on the cross section to the shear resistance of the section is calculated. If any of the ratios (for both local Y & Z axes) exceed 1.0 or the allowable value provided using the RATIO parameter (see D14.B.7 Design Parameters (on page 1960)), the section is considered to have failed under shear. The code also requires that the slenderness ratio of the web be within a certain limit (See Cl. 13.4.1.3, page 29 of SABS 0162-1:1993). Checks for safety in shear are performed only if this value is within the allowable limit. Users may by-pass this limitation by specifying a value of 2.0 for the MAIN parameter.

Related Links

- *V.I Section in Bending* (on page 4516)
- *V. I Section in Compression* (on page 4518)
- *V. I Section in Shear* (on page 4520)

D14.B.7 Design Parameters

The design parameters outlined in table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 211: South African Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified SANS10162-1:1993. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).

Parameter Name	Default Value	Description
BEAM	0	0 - Perform design at ends and those locations specified in the section command. 1 - Perform design at ends and 1/12th section locations along member length.
CB	1.0	Greater than 0.0 and less than 2.5,Value of Omega_2 (C1.13.6) to be used for calculation Equal to 0.0: Calculate Omega_2
CMY	1.0	1 - Do not calculate Omega-1 for local Y axis. 2 - Calculate Omega-1 for local Y axis
CMZ	1.0	1 - Do not calculate Omega-1 for local Z axis. 2 - Calculate Omega-1 for local Z axis
DFF	0	Default is 0 indicating that deflection check is not performed
DJ1	0	Start node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter
DJ2	0	End node of physical member for determining deflected pattern for deflection check and should be set along with DFF parameter
DMAX	1000	Maximum allowable depth
DMIN	0	Minimum required depth
FU	345Mpa	Ultimate strength of steel
FYLD	300Mpa	Yield strength of steel
KT	1.0	K value for flexural torsional buckling

Parameter Name	Default Value	Description
KY	1.0	K value in local Y-axis, usually minor axis
KZ	1.0	K value in local Z-axis, usually major axis
LT	Member length	Length for flexural torsional buckling
LY	Member length	Length in local Y axis for slenderness value KL/r
LZ	Member length	Length in local Z axis for slenderness value KL/r
MAIN	0	Flag for controlling slenderness check 0 - For Check for slenderness. 1 - For Do not check for slenderness
NSF	1.0	Net section factor for tension members
RATIO	1.0	Permissible ratio of applied load to section capacity Used in altering the RHS of critical interaction equations
SSY	0	Sidesway parameter 0 - Sideway about local Y-axis. 1 - No sideway about local Y-axis.
SSZ	0	Sidesway parameter 0 - Sideway about local Z-axis. 1 - No sideway about local Z-axis.

Parameter Name	Default Value	Description
TRACK	0	Track parameter 0. Print the design output at the minimum detail level. 1. Print the design output at the intermediate detail level. 2. Print the design output at maximum detail level
UNB	Member Length	Unsupported length in bending compression of bottom flange for calculating moment resistance
UNT	Member Length	Unsupported length in bending compression of top flange for calculating moment resistance

D14.B.8 Code Checking

The purpose of code checking is to determine whether the current section properties of the members are adequate to carry the forces obtained from the most recent analysis. The adequacy is checked as per the SANS10162-1:1993 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1 (which is also its default value), moments are calculated at every twelfth point along the beam. When no section locations are specified and the BEAM parameter is set to zero, design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. Using the TRACK parameter can control the extent of detail of the output.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

Example

Sample input data for South African Code Design

```
PARAMETER
CODE SANS10162-1: 1993
MAIN 1 all
LY 4 MEMB 1
LZ 4 MEMB 1
UNL 4 MEMB 1
CB 0 MEMB 1 TO 23
CMZ MEMB 2 1 TO  23
CMY MEMB 2  1 TO 23
SSY 0 MEMB 1 TO 23
SSZ 0 MEMB 1 TO 23
FU 450000 MEMB 1  TO 23
BEAM 1 ALL
NSF 0.85 ALL
```

```
KY 1.2 MEMB 3 4
RATIO 1.0 ALL
TRACK 2 ALL
FYLD 300000 1 TO  23
CHECK CODE ALL
FINISH
```

D14.B.9 Member Selection

The member selection process involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on members listed as `PRISMATIC`.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

D14.B.10 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. The term CRITICAL COND refers to the section of the SANS10162-1:1993 specification, which governed the design.

If the TRACK parameter is set to 1.0, the output will be displayed as follows:

```
                SSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSS
                       STAAD.PRO CODE CHECKING
               (SOUTHAFRICAN STEEL/SANS10162-1:1993 )
                **************************************


ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE NOTED)

MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================

      1 ST   406X67UB                (SOUTHAFRICAN SECTIONS)
                        PASS      SAB-13.8          0.543        1
                        0.00          0.00        -191.90       4.08
 |----------------------------------------------------------------------|
 |  FACTORED RESISTANCES FOR MEMBER-     1  UNIT - KN,M    PHI = 0.90  |
 |   MRZ=      353.27  MRY=       63.99                                 |
 |   CR=       453.21  TR=      2308.50  VR=       642.00              |
 |----------------------------------------------------------------------|
```

Factored member resistances will be printed out. Following is a description of some of the items printed out.

Output Term	Description
MRZ	Factored moment of resistance in z direction

Output Term	Description
MRY	Factored moment of resistance in y direction
CR	Factored compressive resistance for column
TR	Factored tensile capacity
VR	Factored shear resistance

Further details can be obtained by setting TRACK to 2.0. A typical output of track 2.0 parameter is as follows.

```
                 **************************************
                       STAAD.PRO CODE CHECKING
                (SOUTHAFRICAN STEEL/SANS10162-1:1993 )
                **************************************


 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE NOTED)

 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================

       1 ST   406X67UB                (SOUTHAFRICAN SECTIONS)
                         PASS       SAB-13.8           0.543         1
                         0.00           0.00       -191.90        4.08

   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------

   CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  7.00E+02
   IZ =  2.43E+04   SZ =  1.19E+03   PZ =  1.35E+03
   IY =  1.36E+03   SY =  1.52E+02   PY =  2.37E+02


   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------

   FYLD = 300.0   FU = 345.0


   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------

   CRY =  4.532E+02   CRZ =  2.016E+03
   CTORFLX =  4.532E+02
   TENSILE CAPACITY     =  2.308E+03   COMPRESSIVE CAPACITY =  4.532E+02
   FACTORED MOMENT RESISTANCE : MRY =  6.399E+01   MRZ =  3.533E+02
   FACTORED SHEAR RESISTANCE  : VRY =  6.420E+02   VRZ =  6.075E+02


   MISCELLANEOUS INFORMATION
   --------------------------

   NET SECTION FACTOR FOR TENSION = 85.000
   KL/RY =  175.514   KL/RZ =   41.522   ALLOWABLE KL/R =  300.000
```

```
UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  4.000
OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.75
SHEAR FORCE (KNS) : Y AXIS = -6.305E+01   Z AXIS =  0.000E+00
SLENDERNESS RATIO OF WEB (H/W) =  4.33E+01
```

Following is a description of some of the items printed out.

Output Term	Description
CRY	Factored compressive resistance for column buckling about the local y axis
CRZ	Factored compressive resistance for column buckling about the local z axis
CTORFLX	Factored compressive resistance against torsional flexural buckling
TENSILE CAPACITY	Factored tensile capacity
COMPRESSIVE CAPACITY	Factored compressive capacity
FACTORED MOMENT RESISTANCE	MRY = Factored moment of resistance in y direction MRZ = Factored moment of resistance in z direction
FACTORED SHEAR RESISTANCE	VRY = Factored shear resistance in y direction VRZ = Factored shear resistance in z direction

D14.C. South African Codes - Steel Design Per SANS 10162-1:2011

STAAD.Pro is capable of performing steel design based on the South African code SANS 10162-1:2011 *Design of steel structures*.

D14.C.1 General

The design philosophy embodied in this specification is based on the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized - ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation, members are proportioned to resist the design loads without exceeding the limit states of strength, stability, and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks whether code requirements for each selected section are met and identifies the governing criteria.

The next few sections describe the salient features of the STAAD implementation of SANS 10162-1:2011. A detailed description of the design process along with its underlying concepts and assumptions is available in the specification document.

D14.C.2 Analysis Methodology

Elastic analysis method is used to obtain the forces and moments for design. Analysis is done for the primary and combination loading conditions provided by the user. You are allowed complete flexibility in providing loading specifications and using appropriate load factors to create necessary loading situations. Depending upon the analysis requirements, regular stiffness analysis or P-Delta analysis may be specified. Dynamic analysis may also be performed and the results combined with static analysis results.

Refer to TR.37 Analysis Specification (on page 2620) for additional information.

D14.C.3 Member Property Specifications

For specification of member properties, the steel section library available in STAAD.Pro may be used. The next section describes the syntax of commands used to assign properties from the built-in steel table. Member properties may also be specified using the User Table facility. For more information on these facilities, refer to G. 6 Member Properties (on page 2113) .

D14.C.4 Built-in Steel Section Library

A steel section library consisting of South African Standards shapes is available for member property specification.

Refer to D14.B.4 Built-in Steel Section Library (on page 1955) for details on using the built-in library of South African steel shapes.

D14.C.5 Section Classification

The SANS 10162 specification allows inelastic deformation of section elements. Thus, local buckling becomes an important criterion. Steel sections are classified as plastic (Class 1), compact (Class 2), noncompact (Class 3), or slender element (Class 4) sections depending upon their local buckling characteristics (See Clause 11.2 and Table 1 of SANS 10162-1:2011).

The design procedures are different depending on the section class. STAAD.Pro determines the section classification for the standard shapes and user specified shapes.

Class 4 sections are those with ratios exceeding those listed for Class 3. This classification is a function of the geometric properties of the section. Design is performed accordingly for sections in Class 1, Class 2, and Class 3.

Class 4 sections per SANS 10162-1:2011 are *not* designed in STAAD.Pro. If a member design is requested for a section of this type, the analysis file will list reason for failure as “Class 4”.

D14.C.6 Member Resistances

The member resistances are calculated in STAAD.Pro according to the procedures outlined in section 13 of the specification. These depend on several factors such as members’ unsupported lengths, cross-sectional properties, slenderness factors, unsupported width to thickness ratios and so on. Note that the program automatically takes into consideration appropriate resistance factors to calculate member resistances. Explained here is the procedure adopted in STAAD.Pro for calculating the member resistances.

All the members are checked against allowable slenderness ratio as per Cl.10.2 of SANS 10162-1:2011.

D14.C.6.1 Resistance Factor

The strength resistance factor, ϕ used for structural steel is 0.90 per Section 13.1. This factor is applied to the equations directly in capacity calculations.

D14.C.6.2 Axial Tension

The factored tensile resistance, T_r , developed by a member subjected to an axial tensile force shall be taken as least of the following per Section 13.2:

$$T_r = \phi A_g f_y$$

$$T_r = 0.85\phi A_{ne} f_u$$

Parameters FYLD, FU, and NSF are applicable for these calculations.

D14.C.6.3 Flexural Buckling - Axial Compression

Per section 13.3.1, the factored axial compressive resistance, C_r , of doubly symmetric sections conforming to the requirements of clause 11 for class 1, 2, or 3 sections shall be taken as:

$$C_r = \phi A f_y (1+\lambda^{2n})^{-1/n}$$

where

n	=	1.34 for hot-rolled, fabricated structural sections, and hollow structural sections manufactured according to SANS 657-1
λ	=	$= \frac{kl}{r}\sqrt{\frac{f}{\pi^2 E}} = \sqrt{\frac{f}{f_e}}$

Doubly symmetric sections which may be governed by torsional-flexural buckling shall also meet the requirements of 13.3.2.

The parameters KY, LY, KZ, and LZ are applicable for this.

D14.C.6.4 Torsional or Torsional-Flexural Buckling

Per section 13.3.2, the factored compressive resistance, C_r , of asymmetric, singly symmetric, and cruciform or other bisymmetric sections not covered under 13.3.1 shall be computed using the expressions given in 13.3.1 with a value of $n = 1.34$ and the value of f_e taken as lesser of F_{ex} and F_{eyz} for single symmetric section, with the y axis taken as the axis of symmetry.

$$f_{eyz} = \frac{f_{ey} + f_{ez}}{2\Omega}\left[1 - \sqrt{1 - \frac{4 f_{ey} f_{ez} \Omega}{(f_{ey} + f_{ez})^2}}\right]$$

where

f_{ey}	=	$\frac{\pi^2 E}{\left(\frac{K_y L_y}{r_y}\right)^2}$
f_{ez}	=	$\left(\frac{\pi^2 E C_w}{K_Z^2 L_Z^2} + GJ\right)\frac{1}{A\bar{r}_0^2}$
Ω	=	$1 - \left(\frac{x_0^2 + y_0^2}{\bar{r}_0^2}\right)$

x_0, y_0	=	= the principal coordinates of the shear center with respect to the centroid of the cross-section.
$\bar{r}_0^2$	=	$= x_0^2 + y_0^2 + r_x^2 + r_y^2$

For asymmetric sections, f_e is the smallest root of:

$$\left(f_e - f_{ex}\right)\left(f_e - f_{ey}\right)\left(f_e - f_{ez}\right) - f_e^2\left(f_e - f_{ey}\right)\left(\frac{x_0}{\bar{r}_0}\right)^2 - f_e^2\left(f_e - f_{ex}\right)\left(\frac{y_0}{\bar{r}_0}\right)^2 = 0$$

where

f_{ex} = $\dfrac{\pi^2 E}{\left(\frac{K_x L_x}{r_x}\right)^2}$

The parameters KY, LY, KZ, and LZ are applicable for this.

D14.C.6.5 Shear

Per section 13.4.1, the factored shear resistance, V_r , developed by the web of a flexural member, shall be taken as

$$V_r = \phi A_v f_s$$

where

A_v = = the shear area $t_w h$

a. $f_s = 0.66 f_y$ when $\dfrac{h_w}{t_w} \le 440\sqrt{\dfrac{k_v}{f_y}}$

where

k_v = = the shear buckling coefficient defined as:

$$= 4 + \frac{5.34}{(s/h_w)^2} \text{ if } s/h_w < 1$$

$$= 5.34 + \frac{4}{(s/h_w)^2} \text{ if } s/h_w \ge 1$$

b. $f_s = f_{cri}$ when $440\sqrt{\dfrac{k_v}{f_y}} < \dfrac{h_w}{t_w} \le 500\sqrt{\dfrac{k_v}{f_y}}$

where

f_{cri} = $= 290\dfrac{\sqrt{f_y k_v}}{(h_w / t_w)}$

k_v = = is as defined in (a)

c. $f_s = f_{cri} + f_t$ when $500\sqrt{\dfrac{k_v}{f_y}} < \dfrac{h_w}{t_w} \le 620\sqrt{\dfrac{k_v}{f_y}}$

where

f_t = the tension field post-buckling stress defined as:

$= k_a(0.50 f_y - 0.866 f_{cri})$

k_a = the aspect coefficient defined as:

$$= \frac{1}{\sqrt{1 + (s/h_w)^2}}$$

d. $f_s = f_{cre} + f_t$ when $620\sqrt{\frac{k_v}{f_y}} < \frac{h_w}{t_w}$

where

f_t	=	the tension field post-buckling stress defined as:
		$= k_a(0.50f_y - 0.866f_{cre})$
f_{cre}	=	$= \frac{180,\ 000k_v}{(h_w / t_w)^2}$
k_v	=	as defined in (a)

D14.C.6.6 Bending

The laterally unsupported length of the compression flange for the purpose of computing the factored moment resistance is specified using the parameters UNB and UNT.

Laterally Supported Members

Per section 13.5, the factored moment resistance, M_r , developed by a member subjected to uniaxial bending moments about a principal axis and where continuous lateral support is provided to the compressive flange shall be taken as:

a. For class 1 and class 2 sections:

$$M_r = \phi Z_{pl} f_y = \phi M_p$$

b. For class 3 sections:

$$M_r = \phi Z_e f_y = \phi M_y$$

Laterally Unsupported Members

Per section 13.6, where continuous lateral support is not provided to the compression flange of a member subjected to uniaxial strong axis bending, the factored moment resistance, M_r , may be taken as follows:

a. For doubly symmetric class 1 and class 2 sections, except closed square and circular sections:

i. When $M_{cr} > 0.67M_p$,

$$M_r = 1.15\phi M_p\left(1 - \frac{0.28M_p}{M_{cr}}\right)$$

but not greater than ϕM_p .

ii. When $M_{cr} \le 0.67M_p$,

$$M_r = \phi M_{cr}$$

where

M_{cr}	=	= the critical elastic moment of the unbraced member,
		$= \frac{\omega_2 \pi}{KL}\sqrt{EI_y GJ + \left(\frac{\pi E}{KL}\right)^2 I_y C_w}$
KL	=	= the effective length of the unbraced portion of the beam, in mm.
ω_2	=	$= 1.75 + 1.05\kappa + 0.3\kappa^2 \le 2.5$ for unbraced lengths subjected to end moments, or
		$= 1.0$ when the bending moment at any point within the unbraced length is larger than the end moment or when the re is no effective lateral

support for the compression flange at one of the ends of the unsupported length.

Note: The value for ω_2 can be specified using the CB parameter. Otherwise, it is calculated as indicated here.

κ = = the ratio of the smaller factored moment to the larger factored moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.

C_w = = the ratio of the smaller factored moment to the larger factored moment at opposite ends of the unbraced length, positive for double curvature and negative for single curvature.

Note: Alternatively, E may be specified directly.

b. For doubly symmetric class 3 sections, except closed square and circular sections, and for channels:

i. When $M_{cr} > 0.67M_p$,

$$M_r = 1.15\phi M_y\left(1 - \frac{0.28M_y}{M_{cr}}\right)$$

but not greater than ϕM_y for class 3 sections and the value given in 13.5(c)(iii) for class 4 sections.

ii. When $M_{cr} \leq 0.67M_y$,

$$M_r = \phi M_{cr}$$

where M_{cr} and ω_2 are as defined in 13.6(a).

c. For closed sections and circular sections, M_{cr} shall be determined in accordance with section 13.5.

d. For biaxial bending, the member shall meet the following criterion:

$$\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \leq 1.0$$

e. For monosymmetric sections, a rational method of analysis should be used.

Note: STAAD.Pro uses AISC LRFD guidelines for the design of channels, double angles, tees, and single angle sections.

i. For tees and double angles:

$$= \frac{\pi\sqrt{EI_yGJ}}{L_b}\left(B + \sqrt{1 + B^2}\right)$$ (AISC LRFD equation F1-15)

where

B = $-2.3(d / UNL)\sqrt{I_y / J}$

minus sign considered for conservative side

ii. For channel sections:

When $M_{cr} \leq 0.67M_y$:

$$M_r = 1.15\phi M_y\left[1 - 0.28\frac{M_y}{M_{cr}}\right] \leq 0.9M_y$$

When $M_{cr} > 0.67M_y$

$$M_r = 0.9M_{cr}$$

iii. For angle sections:

When $M_{ob} \le M_y$:

$$M_n = M_{ob}\left[0.92 - 0.17\frac{M_{ob}}{M_y}\right]$$

When $M_{ob} > M_y$

$$M_n = M_y\left[1.92 - 1.17\sqrt{\frac{M_y}{M_{ob}}}\right] \le 1.5M_y$$

where

M_{ob} = $= 4.9E\frac{I_z}{l^2}CB\left[\sqrt{\beta_w^2 + 0.052(lt / r_z)^2} + \beta_w\right]$ for unequal leg angles.

Note: β_w is conservatively assumed as zero.

$= CB\frac{0.46Eb^2t^2}{l}$ for equal leg angles.

I_z = minor principal axis moment of inertia
r_z = radius of gyration for minor principal axis
β_w = 0 (conservative assumption)
CB = The design parameter corresponding to ω_2 .
b = width of the angle leg
t = thickness of the angle

D14.C.6.7 Member Strength and Stability

Class 1 and Class 2 I-Shaped Sections

Members required to resist both bending moments and an axial compressive force shall be proportioned so that:

$$\frac{C_u}{C_r} + \frac{0.85U_{1x}M_{ux}}{M_{rx}} + \frac{\beta U_{1y}M_{uy}}{M_{ry}} \le 1.0$$

where

C_u and M_u = = the maximum load effects in compression and bending, respectively, including stability effects as defined in Cl. 8.7.
β = $= 0.6 + 0.4\lambda_y \le 0.85$
λ_y = = the nondimensional slenderness parameter about the y-y axis

The capacity of the member shall be examined for:

a. cross-sectional strength (members in braced frames only), with $\beta = 0.6$, in which case:

C_r is as define din Cl. 13.3 with the value of $\lambda = 0$.
M_r is as defined in Cl. 13.5 (for the appropriate class of section), and
U_{1x} and U_{1y} are as defined in 13.8.4 but not less than 1.0, and

b. overall member strength, in which case:

C_r is as define din Cl. 13.3 with the value of $K = 1$, except that for strong-axis bending, $C_r = C_{rx}$ (see aslo 10.3.2),
M_r is as defined in Cl. 13.5 (for the appropriate class of section), and
U_{1x} and U_{1y} are as defined in 13.8.4 for members in braced frames, and

U_{1x} and U_{1y} are taken as 1.0 for members in unbraced frames, and

c. lateral torsional bucking strength, when applicable, in which case:

C_r is as define din Cl. 13.3, and is based on weak-axis or torsional-flexural buckling (see also 10.3.3),
M_{rx} is as defined in 13.6 (for the appropriate class of section),
M_{ry} is as defined in 13.5 (for the appropriate class of section),
U_{1x} and U_{1y} are as defined in 13.8.4 for members in braced frames, and
U_{1x} is as defined in 13.8.4 but not less than 1.0, for members in braced frames, and
U_{1y} is as defined in 13.8.4 for members in braced frames

There parameters SSY and SSZ are used to indicate the sidesway in the local Y and Z axes, respectively.

In addition, the member shall meet the following criteria:

$$\frac{M_{ux}}{M_{rx}} + \frac{M_{uy}}{M_{ry}} \leq 1.0$$

where

M_{rx} and M_{ry}	=	= as described in 13.5 or 13.6, as appropriate

All Other Sections

Members required to resist both bending moments and an axial compressive force shall be proportioned so that:

$$\frac{C_u}{C_r} + \frac{U_{1x} M_{ux}}{M_{rx}} + \frac{U_{1y} M_{uy}}{M_{ry}} \leq 1.0$$

where all terms are as defined in 13.8.2

The capacity of the member shall be examined for the following cases in a parallel manner to that in 13.8.2:

a. cross-sectional strength (members in braced frames and tapered members only),
b. overall member strength, and
c. lateral-torsional buckling strength.

Section Values of U_1

In lieu of a more detailed analysis, the value of U1 for the axis under consideration, accounting for the second-order effects due to the deformation of a member between its ends, shall be taken as:

$$U_1 = \frac{\omega_1}{1 - C_u / C_e}$$

where

ω_1	=	= for the axis under consideration as defined in 13.8.4
C_e	=	$= \frac{\pi^2 EI}{L^2}$ for the axis under consideration

D14.C.6.8 Combined Axial Tension and Bending

Members required to resist both bending moments and an axial tensile force shall be proportioned such that:

a. $\frac{T_u}{T_r} + \frac{M_u}{M_r} \leq 1.0$

where

M_r	=	ϕM_p for class 1 and class 2 sections

$= \phi M_y$ for class 3 and class 4 sections

b. $\frac{M_u}{M_r} - \frac{T_u Z_{pl}}{M_r A} \leq 1.0$ for class 1 and class 2 sections,

$\frac{M_u}{M_r} - \frac{T_u Z_e}{M_r A} \leq 1.0$ for class 3 and class 4 sections

where

M_r = as defined in 13.5 or 13.6

D14.C.7 Design Parameters

The design parameters outlined in table below may be used to control the design procedure. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note: Once a parameter is specified, its value stays at that specified number until it is specified again. This is the way STAAD.Pro works for all codes.

Table 212: South African Steel Design Parameters

Parameter Name	Default Value	Description
CODE	-	Must be specified `SANS10162-1:2011`. Design code to follow. See TR.48.1 Parameter Specifications (on page 2676).
BEAM	0	0 - Perform design at ends and those locations specified in the section command. 1 - Perform design at ends and 1/12th section locations along member length.
CAN	0	Deflection check for cantilever members. 0 - False 1 - True

Parameter Name	Default Value	Description
CB	1.0	Greater than 0.0 and less than 2.5,Value of ω_2 (C1.13.6) to be used for calculation Equal to 0.0: Calculate ω_2
CMY	1.0	Values between 0.4 – 1 used for ω_{1y} (C1.13.8.5). Input beyond the values consider as 1.
CMZ	1.0	Values between 0.4 – 1 used for ω_{1z} (C1.13.8.5). Input beyond the values consider as 1.
DFF	0	Default is 0 indicating that deflection check is not performed
DJ1	0	Start node of physical member for determining deflected pattern for deflection check and should be set along with `DFF` parameter
DJ2	0	End node of physical member for determining deflected pattern for deflection check and should be set along with `DFF` parameter
DMAX	1,000	Maximum allowable depth
DMIN	0	Minimum required depth
FU	345 Mpa	Ultimate strength of steel
FYLD	300 Mpa	Yield strength of steel
KT	1.0	K value for flexural torsional buckling
KY	1.0	K value in local Y-axis, usually minor axis
KZ	1.0	K value in local Z-axis, usually major axis
LT	Member length	Length for flexural torsional buckling
LY	Member length	Length in local Y axis for slenderness value KL/r

Parameter Name	Default Value	Description
LZ	Member length	Length in local Z axis for slenderness value KL/r
MAIN	0	Flag for controlling slenderness check 0 - For Check for slenderness. 1 - For Do not check for slenderness
NSF	1.0	Net section factor for tension members
RATIO	1.0	Permissible ratio of applied load to section capacity Used in altering the RHS of critical interaction equations
SSY	0	Sidesway parameter 0 - Sideway about local Y-axis. 1 - No sideway about local Y-axis.
SSZ	0	Sidesway parameter 0 - Sideway about local Z-axis. 1 - No sideway about local Z-axis.
STIFF	Member length	Center-to-center distance between transverse web stiffeners.
TRACK	0	Track parameter 0. Print the design output at the minimum detail level. 1. Print the design output at the intermediate detail level. 2. Print the design output at maximum detail level
UNB	Member Length	Unsupported length in bending compression of bottom flange for calculating moment resistance

Parameter Name	Default Value	Description
UNT	Member Length	Unsupported length in bending compression of top flange for calculating moment resistance

D14.C.8 Code Checking and Member Selection

D14.C.8.1 Code Checking

The purpose of code checking is to determine whether the current section properties of the members are adequate to carry the forces obtained from the most recent analysis. The adequacy is checked as per the SAB0162-1: 1993 requirements.

Code checking is done using forces and moments at specified sections of the members. If the BEAM parameter for a member is set to 1 (which is also its default value), moments are calculated at every twelfth point along the beam. When no section locations are specified and the BEAM parameter is set to zero, design will be based on member start and end forces only. The code checking output labels the members as PASSed or FAILed. In addition, the critical condition, governing load case, location (distance from the start joint) and magnitudes of the governing forces and moments are also printed. Using the TRACK parameter can control the extent of detail of the output.

Refer to D1.B.1.3 Code Checking (on page 1141) for general information on Code Checking. Refer to TR.49 Code Checking Specification (on page 2677) for details the specification of the Code Checking command.

D14.C.8.2 Member Selection

The member selection process involves determination of the least weight member that PASSes the code checking procedure based on the forces and moments of the most recent analysis. The section selected will be of the same type as that specified initially.

For example, a member specified initially as a channel will have a channel selected for it. Selection of members whose properties are originally provided from a user table will be limited to sections in the user table. Member selection cannot be performed on members listed as `PRISMATIC`.

Refer to D1.B.1.4 Member Selection (on page 1142) for general information on Member Selection. Refer to TR. 49.1 Member Selection Specification (on page 2678) for details the specification of the Member Selection command.

7

Postprocessing and Reports

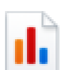
This section describes how to review output, perform post-processing tasks, generate reports, and plot from STAAD.Pro.

P. To view analysis results

There are several ways to open analysis results in the STAAD.Pro Editor.

1. Either:

   On the **Utilities** ribbon tab, select the **View Analysis Output** tool in the **Utilities** group.

   or

   Drag a STAAD output file (file extension `.anl`) icon from Windows Explorer and drop it on to the STAAD.Pro Editor icon.

   or

   In the Windows Explorer, double-click the icon for the file, `Bentley.Staad.Editor.exe`.

2. If you opened the STAAD.Pro Editor, then:
   - **a.** select **File** > **Open.**
   - **b.** Select **STAAD Analysis Output Files (*.anl)** in the file type drop-down list.
   - **c.** Navigate to and select a STAAD output file and the click **Open**.

The output results open in the STAAD.Pro Editor application.

Note: You may need to change the character encoding used depending on the design code selected.

Related Links

- *I. STAAD.Pro Editor* (on page 2043)

P. Postprocessing Workflow

This workflow offers graphical result verification and visualization facilities. A comprehensive custom report generation facility is also incorporated. The customized reports may contain tabular results as well as graphics.

P. To create a animated video file from analysis results

AVI files are a mechanism by which a dynamic result, such as, a deflection diagram in animation, may be captured and recorded. In an animated view, the movement from one extremity to the other is captured as several frames. Presently, this facility is available in STAAD for deflection, section displacement, mode shape and plate stress contour diagrams.

1. Perform a successful analysis on a STAAD input file.
2. On the **Utilities** ribbon tab, select the **Create AVI File** tool in the **Utilities** group.
   The **Create AVI File** dialog opens.
3. (Optional) Change the values of the **Total No. of Frames** and **Frame Rate /sec** to alter the speed and length of the animation.
4. Select a result to animate: **Deflection**, **Sectional Displacement**, **Mode Shape**, or **Stress Contour**.

   Note: Mode Shape and Stress Contour are not active if the required data of that type are not present in the STAAD file, such as a modal extraction, or finite elements.
5. Click **OK**.
   A **Save As** dialog opens.
6. Click **Save**.
   The **Video Compression** dialog opens.
7. (Optional) Select the means of **Video Compression** (codec) to use and change settings for the selection as needed.
8. Click **OK**.
   A message indicating the status of the AVI creation opens.
9. Click **OK**.

   The AVI Video window opens and begins playing the saved animation.

P. Nodal Results

This section describes how to display and review analysis results for nodes.

P. To display displacements

To view the displaced shape along with nodal and beam displacements for the current load case, use the following procedure.

1. Either:

   Select the **Displacements** page in the **Postprocessing** workflow

   or

Select the **Deflection** tool in the **View Results** group on the **Results** ribbon tab

The nodal displacements are shown for the current load case. The **Node Dispalcements** and **Beam Relative Displacement Detail** tables open.

2. (Optional) Select different **Load** in the **View Results** group on the **Results** ribbon tab.
   The displaced shape updates with the selected load.

Related Links

- *P. Node Displacements table* (on page 2002)

P. To display the deflect shape

To display the deflected shape of the structure including beam section displacements, use the following procedure.

Tip: This displays a more accurate estimation of the section displacements than using just the nodal displacements, but can also take longer to render on large models.

1. On the **Results** ribbon tab, select the **Displacement** tool in the **View Results** group.

   The deflected shape is drawn for the current load case.
2. (Optional) Select different **Load** in the **View Results** group on the **Results** ribbon tab.
   The displaced shape updates with the selected load.

Related Links

- *P. Node Displacements table* (on page 2002)

P. To display the support reaction values

To display the support reaction values for the current load case, use the following procedure.

1. Either:

   select the **Reactions** page in the **Postprocessing** workflow

   or

   select the **Layouts** > **Reaction** tool in the **View Results** group on the **Results** ribbon tab

   The six values —namely, the three forces along global X, Y and Z, and the three moments Mx, My and Mz, in the global axis system— are displayed in the annotation box for each support node.

   The **Support Reactions** table opens.
2. (Optional) Select different **Load** in the **View Results** group on the **Results** ribbon tab.
   The reaction values update with the selected load.
3. To reposition an annotation box to aid visibility:
   a. On the **Select** ribbon tab, select the **Text** tool in the **Cursors** group.

The mouse pointer changes to the text selection cursor. 

b. Click-and-drag on any annotation (while holding down the left mouse button) it to the desired location in the View window.
The leading arrow is redrawn from the support to the new annotation position.

c. Press **<Esc>** to return to the default selection mode.

You can use the **Annotation** tool to control the reactions displayed.

Related Links

- *P. Support Reactions table* (on page 2003)

P. To graphically display reactions at each support

1. Select the **Reactions** page in the Postprocessing workflow.
   The reactions are labeled for the current load.
2. On the **Results** ribbon tab, select the **Annotate** tool in the **Configuration** group.

   The **Annotation** dialog opens.
3. Select the Reactions tab.
4. Select the **Show Line** option.
5. (Optional) Change the scale for force and moment if needed.
6. Click **Annotate** to display the reactions as scaled load arrows.
7. Click **Close**.

Related Links

- *Annotation dialog* (on page 2997)
- *Annotation dialog* (on page 2997)

P. To display load versus displacement graph

To display the load level versus displacement curve for a selected node and load case, use the following procedure.

Load versus displacement is only available after a geometric nonlinear analysis is performed on the STAAD.Pro model.

1. Select the **Layouts** > **Node Displacement** tool in the **View Results** group on the **Results** ribbon tab.
   The **Node Displacement Curve** and **Node Displacements** table open.
2. Either:

   use the **Node Cursor** tool in the **Cursors** group on the **Select** ribbon tab to select a node in the graphical view

   or

   select a node in the **Node Displacements** table

The plot for the selected node is displayed. If you select multiple nodes, then the graph will display the plot for the lowest node number.

3. Select a **Load** in the **View Results** group on the **Results** ribbon tab.

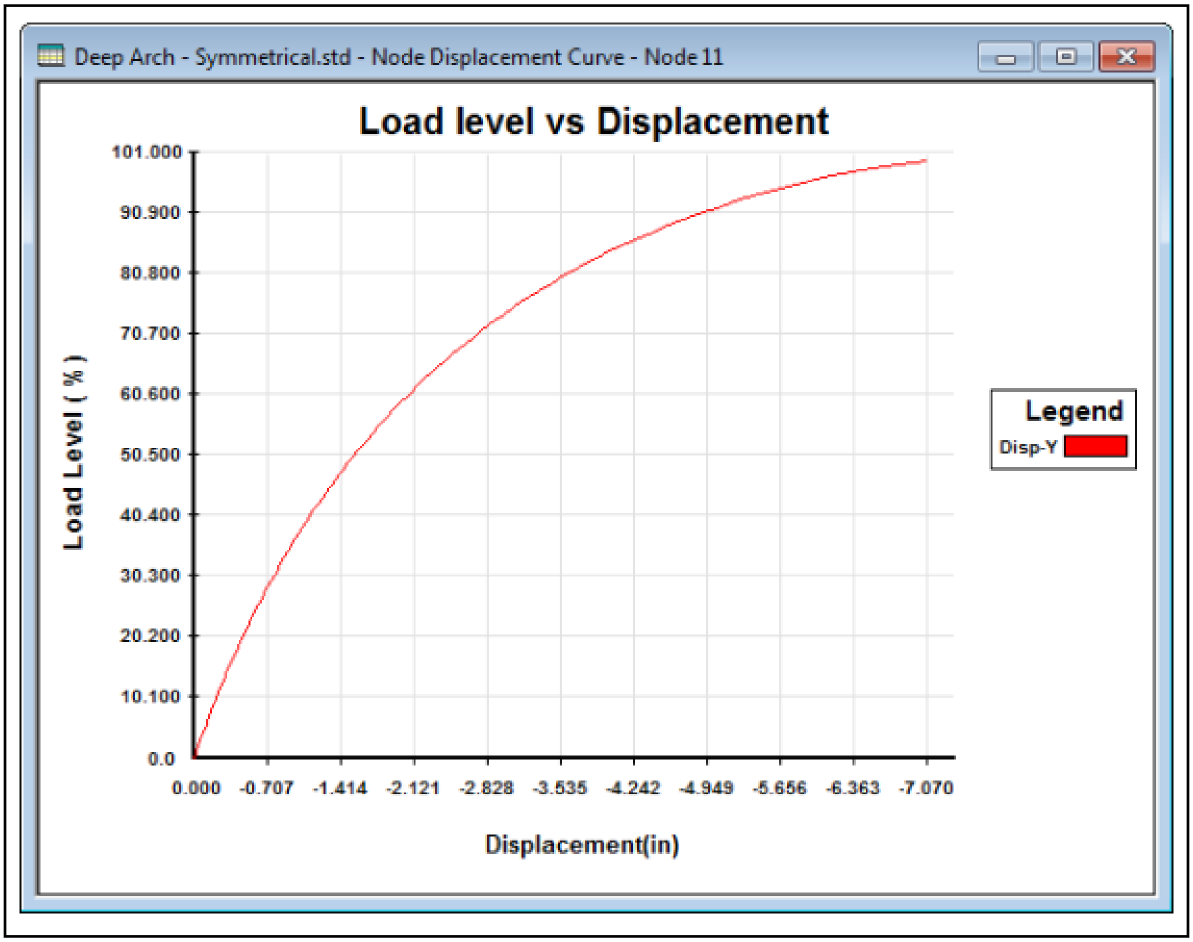
   A nonlinear analysis must be performed for the selected load case.
4. (Optional) Specify the **Select Load Step** in the **Node Displacements** table.

   > **Note:** The maximum number of load steps for the current load case is displayed on the right-hand side of the table controls.
5. (Optional) Check the **Limit Maximum Load Step for Graph** option in the **Node Displacements** table to limit the maximum load step displayed in the graph.
6. To change the degree of freedom plotting on the graph:
   a. Right-click anywhere on the **Node Displacements** curve window.
   b. Select **Geometry** from the pop-up menu.
   The **Diagram** dialog opens.
   c. Select the displacement degree of freedom you want displayed on the graph.

   You can select multiple degrees of freedom to plot simultaneously.
   d. Click **OK**.

   The graph and legend update to include the selected curves.
7. (Optional) To include the current graph in a report:
   a. Right-click anywhere on the **Node Displacements** curve window.
   b. Select **Take Picture** from the pop-up menu.
   A **Picture #** dialog opens to name the image.
   c. (Optional) Type an **ID** and **Caption**.
   d. Click **OK**.

   The picture is added to the picture album for inclusion in reports.



P. To display base pressure results

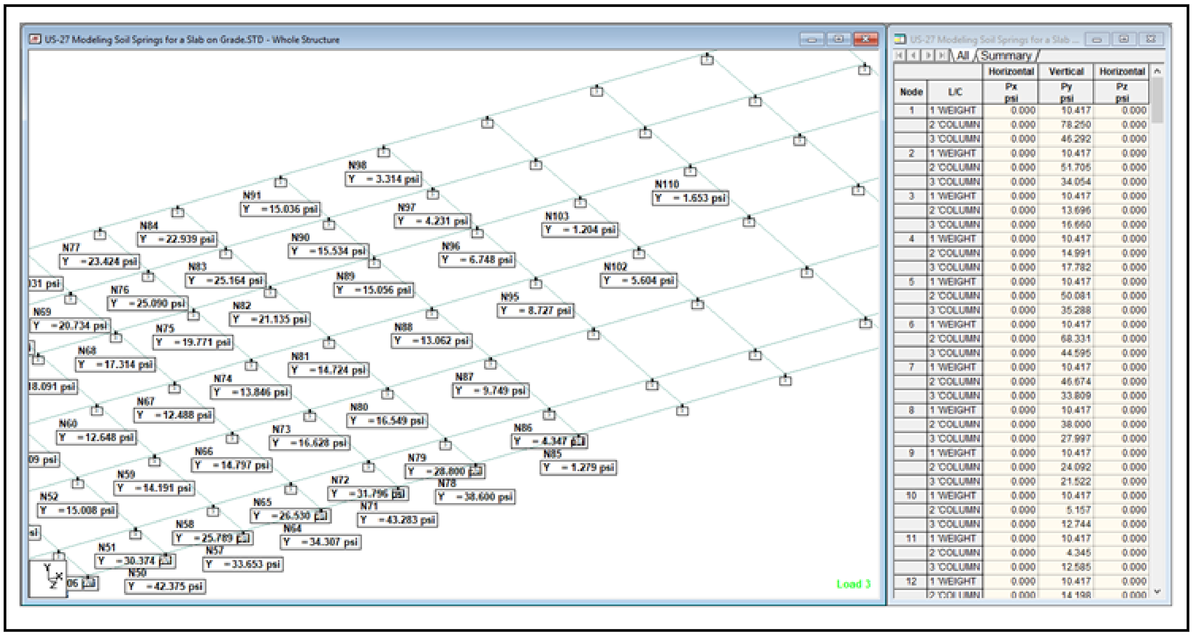
To display soil pressure at support node locations for slabs consisting of plate elements which have been generated using elastic mat or plate mat commands, use the following procedure.

This feature is only available for analyzed models that include these mat support conditions.

1. Select the **Layouts** > **Base Pressure** tool in the **View Results** group on the **Results** ribbon tab.
   The vertical pressure value at each support node is displayed. The **Base Pressure** has tabs to display the pressure values at each supported node for all load cases, along with a summary tab.

   Tip: You may need to zoom in to see individual labels for some models.

2. (Optional) Select a different **Load** in the **View Results** group on the **Results** ribbon tab.
   The labels update to reflect the base pressure for the selected load case.



P. To display nodal instabilities

To graphically examine nodes and degrees of freedom on the structure where instability conditions have been reported in the output, use the following procedure.

This feature is only available when instabilities have been detected during the analysis.

1. Select the **Layouts** > **Instability** tool in the **View Results** group on the **Results** ribbon tab.

   The locations of instabilities are highlighted on the structure in large blue circles. The Unstable Joint table provides information on the joint number and the degree of freedom associated with an instability at that joint. An unstable condition is marked with a "1".

You can apply restraints at these joints in unstable directions in an attempt to stabilize the joint.

P. Beam Results

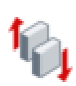
This section describes how to display and review analysis results for members.

P. To display results diagrams on the members

To display the results diagrams drawn on to the members, use the following procedure.

1. On the **Results** ribbon tab, select one of the graphical results tools in the **View Results** group:

Displacement



FY

MY

Utilization Ratio

FX

MZ

FX

MX

Beam Stress

> **Tip:** Alternatively, you can select the **Layout** > **Beam Forces** tool in the **View Results** group on the **Results** ribbon tab to display the major axis bending graphs on the members (**MZ**).

The selected diagram is drawn on the members.

> **Note:** The selected tool is shaded to indicate it is active.

2. (Optional) Either:

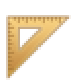
   repeat step 1 to select different diagrams as needed

   or

   select the active diagram tool to hide the diagram

   > **Tip:** Multiple diagrams may be displayed simultaneously.

3. To change the scale of any diagrams:
   a. On the **Results** ribbon tab, select the **Scale** tool in the **Configuration** group.

   The **Diagrams** dialog opens to the **Scales** tab.
   b. Change the scale factor for the relevant item in the Results Scales group.

   > **Tip:** Smaller values result in more exaggerated diagram sizes.

   c. Click **Apply** to update the diagram.
   d. Click **OK** to close the dialog.

P. To view stress contour of a beam

To display the full section stress contour on a beam, use the following procedure.

Stress contours can be displayed for sections that have predefined dimensions (e.g., catalog and prismatic sections) and general sections with stress distances defined.

1. On the **Results** ribbon tab, select the **Layouts** > **Beam Stress** tool in the **View Results** group.
   The **3D Beam Stress Contour** dialog opens.
2. In the **Whole Structure** view window, click on any member to select it.

The **Select Section Plane** dialog opens.

The **3D Beam Stress Contour** displays a rendered view of the member with the stress contours on the surface on the left and the current cross-section stress contour on the right. Compression is display in blue and tension in red.

The table displays the corner stress values at points along the member length.

3. Use the slider in the **Select Section Plane** dialogs to update the cross section stress contour display.
4. (Optional) In the **Select Section Plane** dialog, type a Y Point and Z Point value to display the cross-section stress at an arbitrary point within the section.

P. To view stress contour of a General UPT beam

To display the full section stress contour on a beam that uses a general UPT profile, use the following procedure.

Stress contours can be displayed for general UPT sections that do not yet have stress distances defined by defining those in the Postprocessing workflow.

1. On the **Results** ribbon tab, select the **Layouts** > **Beam Stress** tool in the **View Results** group.
   The **3D Beam Stress Contour** dialog opens.
2. In the **Whole Structure** view window, click on any member which uses a General UPT profile to select it.
3. Either:

   select the **Define Section Profile** tool in the **Model** group on the **Beam** ribbon tab

   or

   right-click and then select **Define Section Profile** from the pop-up menu

   The **Define Section Profile** dialog opens.
4. Type the local Y and Z coordinates of the extreme fibers for stress results.

   Tip: You can also “swap” the top and bottom sides ( ) or left and right sides ( ) in order to facilitate entering stress points by clicking the corresponding tools.
5. Click **OK**.
6. Use the slider in the **Select Section Plane** dialogs to update the cross section stress contour display.
7. (Optional) In the **Select Section Plane** dialog, type a Y Point and Z Point value to display the cross-section stress at an arbitrary point within the section.

Related Links

- *Define Section Profile dialog* (on page 3049)

P. To display steel design utilization ratios

To display member utilization values and color-coded design results on the structure, use the following procedure.

The utilization ratios for members are only available for steel design performed in the batch mode.

The utilization ratio is for the load envelope used for design and is not affected by the current load case in the Postprocessing workflow.

Note: For AISC 360-16, multiple parameter blocks (multiple steel designs) can be reviewed in the user interface.

1. (Optional) On the **Results** ribbon tab, select the parameter block or maximum design from the **Design Parameters** drop-down list.

   If an input file contains multiple parameter blocks (steel design parameter sets) or multiple design commands (steel code check or member selections) within a single parameter block, then the results are available in the Postprocessing workflow. Parameter sets are numbered and multiple design commands in the same parameter block are given as decimals.

   Note: This feature is compatible with AISC 360-16 only in this release.
2. O

   on the **Results** ribbon tab, select the **Layouts** > **Utilization** tool in the **View Results** group.

   or

   on the **Results** ribbon tab, select the **Utilization Ratio** tool in the **View Results** group

   The utilization ratio label is shown next to each member included in the steel design.

   Note: Members are color coded: green for passing design, blue for failing less than 50% over utilization, and red for failing more than 50% over utilization.

   The **Design Results** table includes a tab for **All** members as well as for **Failed** members.
3. (Optional) Change the color coded limits:
   a. On the **Results** ribbon tab, select the **Structure** tool in the **Configuration** group.
   The **Diagrams** dialog opens.
   b. Select the **Design Results** tab.
   c. In the color-coding table, change the **To** cell values for the green and blue rows.

   For example, you may want to change the green (passing) utilization ratio to 1.003 and the blue (nearly passing) utilization ratio to 1.05 to get a “feel” for where the members are nearly adequate.
   d. Click **Apply**.
   The diagram color coding updates accordingly.
   e. Click **OK**.

P. To display bending and shear diagrams

To display the bending (both major and minor axes) and shear diagrams, use the following procedure.

1. On the **Results** ribbon tab, select the **Layouts** > **Graphs** tool in the **View Results** group.
   The **Beam Graphs** windows open.
2. Select a member in the view window.
3. Select the current **Load** on the **Results** ribbon tab in the **View Results** group.
4. Change the diagrams displayed in any of the **Beam Graphs** windows:
   a. Right-click and select **Diagrams** from the pop-up menu.
   The **Diagram** dialog opens.
   b. Set the checkbox for the graphs you want to add to the window.
   c. Click **OK**.

   The diagrams are updated.

P. Plate Results

This section describes how to display and review analysis results for plate elements.

P. To display plate results contours

To display analysis results of plate elements as contour lines on the plates, use the following procedure.

Plate stresses can be displayed as contour lines along the plate elements graphically for a variety of stress types.

1. Either:

   on the **Results** ribbon tab, select the **Plate Stress** tool in the **View Results** group

   or

   select the **Plate Results** page in the Postprocessing workflow bar
   The **Diagrams** dialog opens to the Plate Stress Contour tab.

   The tables in the Plate Results page also display results data for many of the stress types.
2. (Optional) Select the **Load Case**.

   Tip: The current load case in the Results ribbon tab will be selected.
3. Select the **Stress Type** to display.
   STAAD.Pro can display a variety of component element stresses or stress combinations.
4. Select the display options for the contours.
5. Click **Apply**.
6. Make any different selections to change the contour display as needed and then click **Apply** again to update the view.
7. Click **OK**.
   The dialog closes.

Related Links

- *Diagrams dialog* (on page 2751)

P. To display plate results along a cut line

To view the plate stresses along an arbitrary line cut across plate elements, use the following procedure.

Note: Cut lines are not supported for models containing plate offsets.

1. On the **Results** ribbon tab, select the **Layouts** > **Results Along Line** tool in the **View Results** group.
   The **Results Along Line** dialog and layout open.
2. In the **Results Along Line** dialog, click **Cut by a Line**.
3. In the view window, click the start and end points of the line.

4. Click a third point to define the cut direction.

   A perpendicular line is rubber banded to the mid-point of the line drawn in step 3.

   The line is added to the list and the in Max Absolute results along the line are displayed in the graph and table.
5. Customize the stress results:
   a. (Optional) Type a **Name** for the line.
   b. Select the **Stress Type** from the drop-down list.

      Options Max Absolute through Tressca Bottom (i.e., options with "(Line)") represent values across indiviual elements.
      Options St-Line through Qt-Line represent values that have been mathematical processed to be continuous across element boundaries and have forces in equlibrium along the cut line.

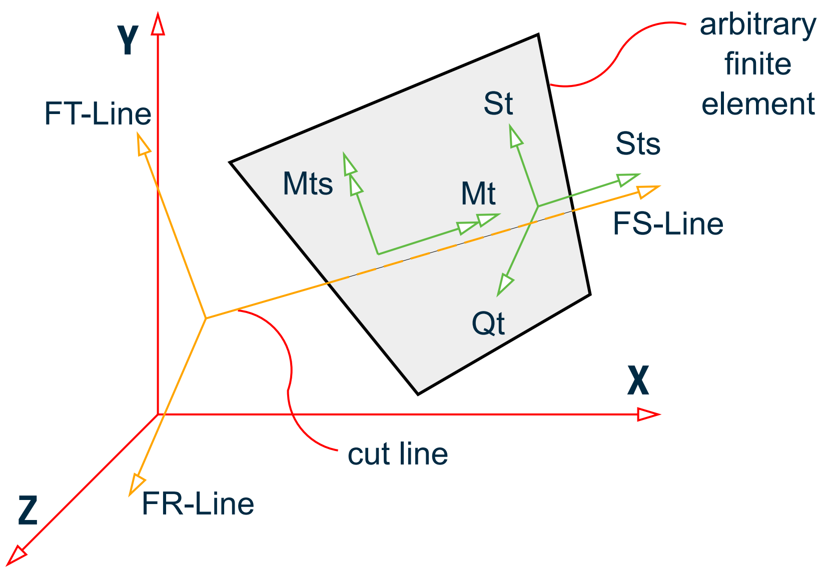


Figure 193: Stresses along a cut line for "-Line" stress types

St Normal stress perpendicular to the cut line, σ_t.

Sts Shear stress along the cut line, τ_t.

Mt Bending moment along the cut line, M_t.

Mts Bending Moment about the T axis. This is in the plane of the plate and perpendicular to the cut line (i.e. the S axis).

Qt Out-of-plane shear, Q_t

   c. Type a **Max No Div** to specify the number of equal divisions displayed in the graph.

      Tip: Higher values result in a clearer graph, with 10-20 being a normal range.

   d. Click **Update**.
6. (Optional) Add the cut line results to the report:
   a. Click **Create Report**.
      The **Report Title** dialog opens.
   b. Check the **Save Report** option.

c. Click **OK**.

The graph displays the selected stress values along the cut line using the number of divisions per element specified.

Forces listed in the table when these results are selected are given in the in a coordinate system relative to the cut line (i.e., “FS” is along the cut line, “FT” is in the plane of the element, and “FR” is normal to the element). This is indicated by the “-Line” suffix of the results.

The table has two tabs: **Stresses** and **Total Force**. In the case of “-Line” stress types, the forces are given in the local cut line coordinate system:

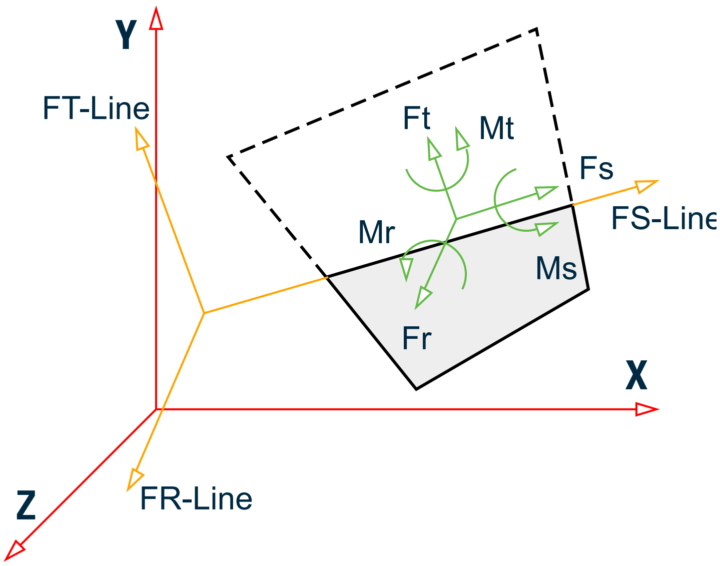


Figure 194: Forces along a cut line for “-Line” stress types

Fs Resultant force along the cutline

Ms Resultant moment about the cutline

Ft Resultant force perpindicular to the culine in the plane of the plate

Mt Resultant moment about the line perpindicular to the cutline in the plane of the plate

Fr Resultant force perpindicular to the cutline out of the plane of the plate

Mr Resultant moment about the line perpindicular to the cutline out of the plane of the plate

P. Solid Results

This section describes how to display and review analysis results for solid elements.

P. To display solid results contours

To display analysis results of solid elements as contour lines on the solid faces, use the following procedure.

Solid stresses can be displayed as contour lines along the solid element faces graphically for a variety of stress types.

1. Either:

   on the **Results** ribbon tab, select the **Solid Stress** tool in the **View Results** group

   or

   select the **Solid Results** page in the Postprocessing workflow bar

   The **Diagrams** dialog opens to the Solid Stress Contour tab.

   The tables in the Solid Results page also display results data for many of the stress types for both solid corners and solid centers.

   Note: The stress values displayed in the Postprocessing workflow tables are “smoothed” by averaging stresses from adjacent elements. Therefore, they may not be equal to those corner stresses reported in the STAAD.Pro `.anl` for the same model.
2. (Optional) Select the **Load Case**.

   Tip: The current load case in the Results ribbon tab will be selected.
3. Select the **Stress Type** to display.

   STAAD.Pro can display a variety of component element stresses or stress combinations.
4. Select the display options for the contours.
5. Click **Apply**.
6. Make any different selections to change the contour display as needed and then click **Apply** again to update the view.
7. Click **OK**.
   The dialog closes.

Related Links

- *Diagrams dialog* (on page 2751)
- *P. Solid Corner Stress table* (on page 2010)
- *P. Solid Center Stress table* (on page 2011)

P. Dynamic Results

P. To display mode shapes

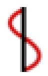
To display the mode shape diagram and a table of mode shape data, use the following procedure.

Mode shapes are available for models with dynamic load cases.

Tip: You can limit the number of modes included in the Postprocessing results in the **Results Setup** dialog **Modes** tab.

1. Either:

   on the **Results** ribbon tab, select the **Mode Shape** tool in the **Dynamics** group



or

on the **Results** ribbon tab, select the **Layouts** > **Mode Shape** tool in the **Dynamics** group

or

select the **Dynamics** page in the **Postprocessing** workflow bar

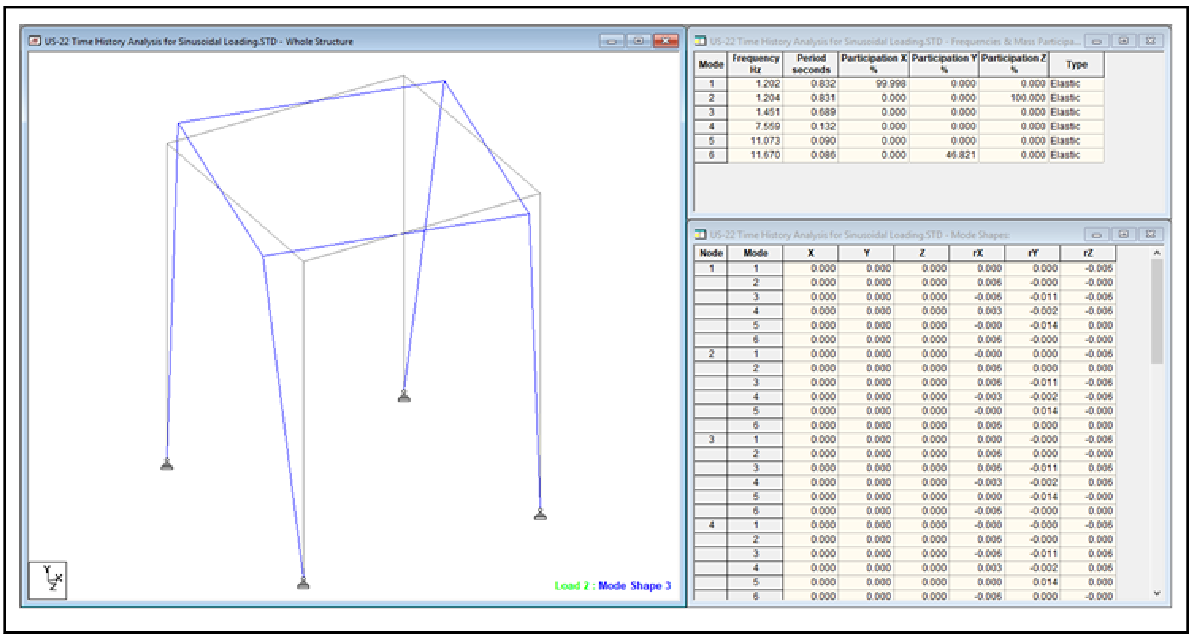
The **Mode Shape** tool is only active if the current load is a dynamic load case.

The mode shape for the currently selected mode and load case is displayed.

If you selected the Layout or the Page control, the **Frequencies & Mass Participation** and **Mode Shapes** tables are also displayed.

2. (Optional) Select a different **Load** in the **View Results** group on the **Results** ribbon tab.
3. (Optional) Select a different **Mode** in the **Dynamics** group on the **Results** ribbon tab.

   This option is only active if the current load is a dynamic load case.



You can control the color and display of the mode shapes diagram from the **Load and Results** tab of the **Diagrams** dialog.

Related Links

- *Diagrams dialog* (on page 2751)

P. To display time history graphs

To display data of time versus displacement, velocity, or acceleration, use the following procedure.

Time history plots are only available for STAAD.Pro models with time history loads.

1. Select the time history response plot you want to view:

To view...	Select...
Displacement data	the **Layouts** > **Displacement** tool in the **Dynamics** group
Velocity data	the **Layouts** > **Velocity** tool in the **Dynamics** group
Acceleration data	the **Layouts** > **Acceleration** tool in the **Dynamics** group

A set of three plots opens.

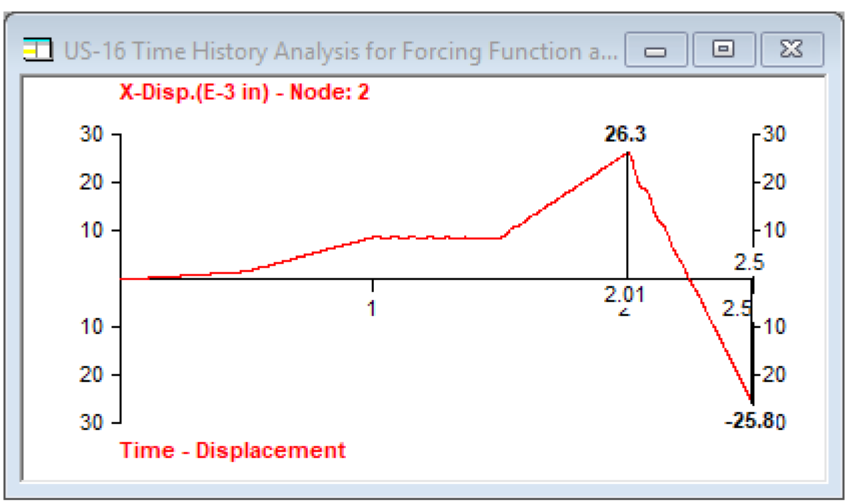
If you do not have any node groups defined in the model, you will asked if you want to define any now. If you opt to not create any now, you must select the Analytical workflow to define node groups.

2. Either:

To graph data for...	Select...
a single node	the node in the graphical view
a node group that defines a floor	the name of the node group in the **Select Groups** dialog **Tip:** You can use the **Group** tool in the **Attributes** group on the **Select** ribbon tab to display the **Select Groups** dialog.

Selecting a node group will display calculated average time history results for the desired response. This feature can be used to generate floor spectra at each floor level in a structure which can then be used as input for other dynamic analysis such as analysis of vibrating machinery existing on the floor.

3. To select the global direction of action displayed on the y-axis for a plot:
   a. Right-click on the graph and select **Diagrams** from the pop-up menu.
      The **Diagram** dialog opens.
   b. Select global direction for the action.
      More than one global direction can be displayed on a plot.
   c. (Optional) Click the color block beside a global direction to specify a different color for the plot.
   d. Click **OK**.

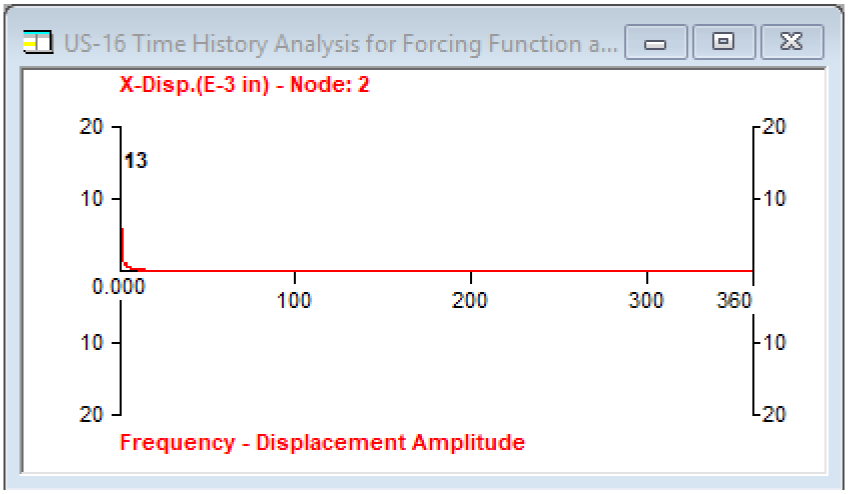


4. To change the x-axis of a graph to display frequency instead of time:

This option performs a time-domain to frequency-domain transformation of the nodal displacement response due to time-varying load. STAAD.Pro graphs the amplitude information from a Fourier transform.

a. Right-click on the graph and select **Diagrams** from the pop-up menu.
   The **Diagram** dialog opens.
b. Select **Frequency Domain** and then click **OK**.

The graph updates to display frequency.



5. (Optional) To save the current set data to an external file:
   a. Right-click on the graph and then select **Save data in a text file** from the pop-up menu.
      A Windows **File Open** dialog opens.
   b. Select a location to save the data.
   c. Type a **File Name** and then click **Save**.

   The plot data is saved in a plan text file (file extension `.txt`).
6. You can use pictures of the graph in other reports or external programs by right-clicking on the graph and selecting:

Select...	To...
Copy Picture	copy the graph to the Windows clipboard
Take Picture	save the image for use in reports for later use. A Picture dialog will open for a label and caption. Click **OK** to save the graph to the report Picture Album.

P. To display floor spectrum results

To display the generated floor spectra results for a time history load case, use the following procedure.

These results are only available for a STAAD.Pro model with a time history load and a generated floor spectrum command.

1. On the **Results** ribbon tab, select the **Layouts** > **Floor Spectrum** tool in the **Dynamics** group.
   The **Floor Spectrum Graph** window and **Floor Spectrums** table open.

   Tip: The peak acceleration value is labeled.
2. On the **Floor Spectrums** table, select the **Floor Spectrum Set** from the drop-down list.
   The tabulated values and the graph update to display the selected set.
3. (Optional) You can change the display of the graphed data by right-clicking on the graph and selecting:

Select...	To...
Linear Graph	display the response spectrum graph with linear axes
Log-Log Graph	display the response spectrum graph with logarithmic axes (default display)
Show Points	display the individual calculated points on the plot

4. (Optional) To save the current set data to an external file:
   a. Right-click on the graph and then select **Save data in a text file** from the pop-up menu.
      A Windows **File Open** dialog opens.
   b. Select a location to save the data.
   c. Type a **File Name** and then click **Save**.

   The plot data is saved in a plan text file (file extension `.txt`).
5. You can use pictures of the graph in other reports or external programs by right-clicking on the graph and selecting:

Select...	To...
Copy Picture	copy the graph to the Windows clipboard
Take Picture	save the image for use in reports for later use. A Picture dialog will open for a label and caption. Click **OK** to save the graph to the report Picture Album.

Related Links
- *AD.2007-04.1.2 Floor Response Spectrum* (on page 238)

P. To display buckling analysis results

To display the calculated buckling factors and buckling modes from a buckling analysis, use the following procedure.

You must perform a successful buckling analysis of a structure using the Eigen method to access this feature.

1. On the **Results** ribbon tab, select the **Layouts** > **Buckling** tool in the **Dynamics** group
   The **Buckling Factors** table and with the **Buckling Modes** table open. The first buckling mode is displayed on the structure.
2. Select a row corresponding to a Mode number in the **Buckling Factors** table
   The graphical display shows the selected buckling mode shape.

P. Reports

STAAD.Pro can generate the following specialized reports.

P. To generate a node displacement report

1. In the **Postprocessing** mode, use the **Node Cursor** tool to select nodes for inclusion in the report.
2. On the Results ribbon tab, select **Reports** > **Node Displacements** in the **Reports** group.

The **Node Displacement** dialog opens.

3. (Optional) Select the sorting criteria you want to use in the report.
4. (Optional) Select the **Load Cases** on the **Loading** tab you want to include in the report.
5. Specify the report metadata:
   a. Select the **Report** tab.
   b. Type the **Title** for the report.
   c. (Optional) Check the **Save Report** option and type an **Id** value if you want to change it.
6. Click **OK**.

Related Links

- *Node Displacement dialog* (on page 3001)

P. To generate transfer forces report

To generate transfer forces for a selected set of beams and columns to include in your report, use the following procedure.

1. In the **Postprocessing** workflow, use the **Beam Cursor** tool to select all the members framing into a connection.
2. On the Results ribbon tab, select **Reports** > **Column Transfer Force** in the **Reports** group.

   The **Transfer Force for Selected Members** dialog opens.
3. (Optional) Click any load cases you to *exclude* from the transfer forces report.

   By default, all load cases will be included.
4. Click **Insert to Table**.
   The **Transfer Force Report** table opens.
5. Click **Close**.

The **Transfer Force Report** can be found within the **Reports** group on the **Items** tab in the **Report Setup** dialog.
Related Links

- *Transfer Forces for Selected Members dialog* (on page 3002)

P. To generate a floor vibration report

To generate a floor vibration report per the procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11, use the following procedure.

Floor vibration reports are performed for composite decks only. You must define and run an analysis for a composite deck structure in order to generate a floor vibration report.

1. On the **Results** ribbon tab, select the **Reports** > **Floor Vibration Report** tool in the **Reports** group.

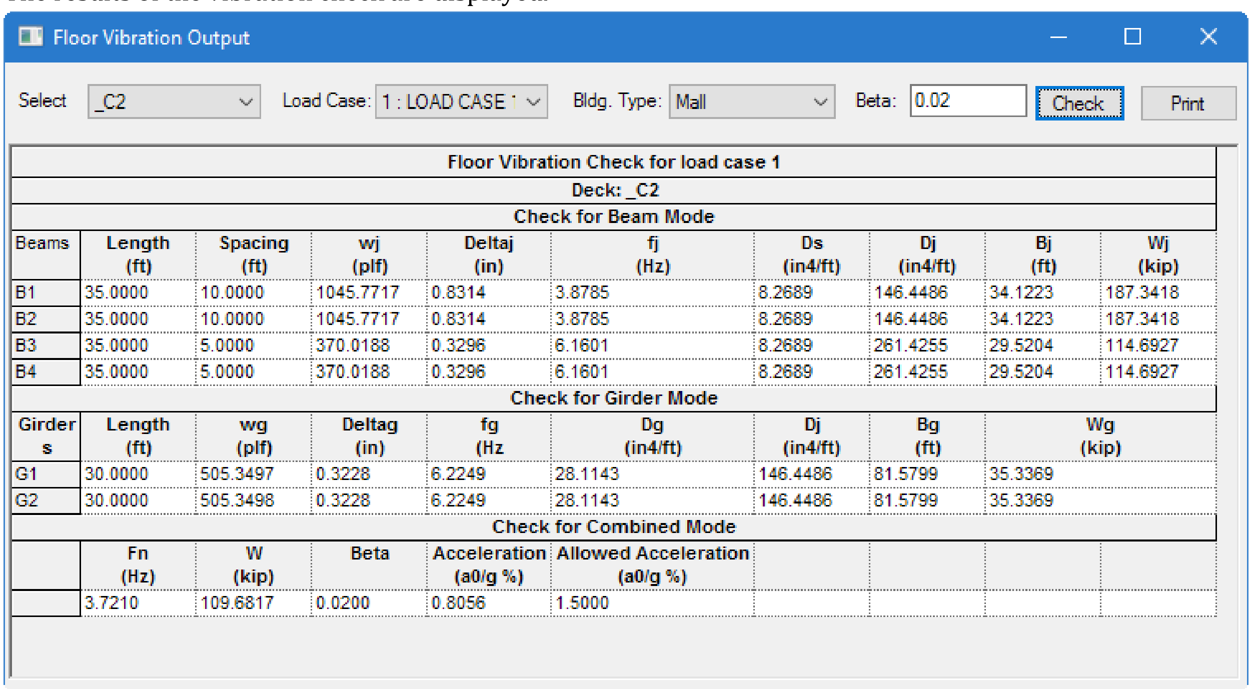
   

   The **Floor Vibration Report** dialog opens.
2. Select the composite deck name from the **Select** drop-down list and then select the **Load Case**.
3. Either:

   select the closes description for the type of building in the **Bldg Type** drop-down list in order to select a predefined Beta value

   or

   type the **Beta** value directly
4. Click **Check**.
   The results of the vibration check are displayed.

Floor Vibration Check for load case 1									
Deck: _C2									
Check for Beam Mode									
Beams	Length (ft)	Spacing (ft)	wj (plf)	Deltaj (in)	fj (Hz)	Ds (in4/ft)	Dj (in4/ft)	Bj (ft)	Wj (kip)
B1	35.0000	10.0000	1045.7717	0.8314	3.8785	8.2689	146.4486	34.1223	187.3418
B2	35.0000	10.0000	1045.7717	0.8314	3.8785	8.2689	146.4486	34.1223	187.3418
B3	35.0000	5.0000	370.0188	0.3296	6.1601	8.2689	261.4255	29.5204	114.6927
B4	35.0000	5.0000	370.0188	0.3296	6.1601	8.2689	261.4255	29.5204	114.6927
Check for Girder Mode									
Girders	Length (ft)	wg (plf)	Deltag (in)	fg (Hz	Dg (in4/ft)	Dj (in4/ft)	Bg (ft)	Wg (kip)	
G1	30.0000	505.3497	0.3228	6.2249	28.1143	146.4486	81.5799	35.3369	
G2	30.0000	505.3498	0.3228	6.2249	28.1143	146.4486	81.5799	35.3369	
Check for Combined Mode									
	Fn (Hz)	W (kip)	Beta	Acceleration (a0/g %)	Allowed Acceleration (a0/g %)				
	3.7210	109.6817	0.0200	0.8056	1.5000				

   Note: The comparison check is the **Acceleration** versus the **Allowed Acceleration** in the Check for Combined Mode section.
5. (Optional) Click **Print** and then select the printer to which you want to send the copy.

Note: Floor vibration reports must be output from this dialog. They cannot be included in the STAAD.Pro report.

Related Links

- *P. Floor Vibrations Engineering Theory* (on page 1998)
- *Floor Vibration Output dialog* (on page 3002)
- *M. Composite Decks* (on page 676)

P. Floor Vibrations Engineering Theory

The fundamental natural frequency of the joist mode and the girder mode can be determined from equation 3.3 on page 11 of the design guide:

$$f_{j/g} = 0.18\ (g/\Delta)^{1/2} \qquad 1$$

where

$f_{j/g}$	=	fundamental natural frequency of the joist or the girder mode
g	=	acceleration due to gravity
Δ	=	midspan deflection of the member due to the weight supported.

For the combined mode, the fundamental natural frequency can be determined from equation 3.4 on page 11 of the design guide:

$$f_n = 0.18\ [g/\ (\Delta_j + \Delta_g)\ ]_{1/2} \qquad 2$$

where

f_n	=	fundamental natural frequency for the combined mode
g	=	acceleration due to gravity
Δ_j	=	joist deflection due to the weight supported
Δ_g	=	girder deflection due to the weight supported

Δ_j and Δ_g are the local deflection of the joist and the girder determined from a secondary operation after the analysis. The stiffness analysis will yield the global deflection values for the girder beams. A line joining the start and the end nodes of the girder beam in its deflected position is created as a base line. Relative to this base line, the deflection values are zero for the start and end nodes. The local deflection values of the intermediate points of the girder beam are evaluated from the global deflection values relative to this base line.

It is this local deflection that is used in calculating the fundamental natural frequency as shown in the earlier equations. Further, the local deflection is also used in calculating the equivalent uniform loading on the joist and the girder, wj and wg, as shown in the equation on page 21 of the AISC Design Guide:

3

where

$I_{j/g}$	=	moment of inertia of the effective joist or girder section
$L_{j/g}$	=	joist or girder span
E_g	=	modulus of elasticity of steel

In addition to the terms f_j , Δ_j , w_j shown above , the following additional terms —D_s , D_j , B_j , and W_j — which are explained below, are also reported for the joist mode.

where

D_s	=	the transformed slab moment of inertia per unit width and is determined from the equation at the bottom right corner of page 17 of the AISC Design Guide:
d_e	=	effective depth of the concrete slab, usually taken as the depth of the concrete above the form deck plus one-half the depth of the form deck
N	=	dynamic modular ratio = $E_s\ /\ 1.35\ E_c$
E_c	=	modulus of elasticity of concrete
D_j	=	the joist or the or beam transformed moment of inertia per unit width, and is determined from the equation shown at the top left of page 18 of the AISC Design Guide:
		= I_j/S
S	=	joist or beam spacing

B_j	=	the effective width for the beam or joist panel mode and is determined from equation 4.3a on page 17 of the AISC Design Guide:
		$= C_j (D_s / D_j)^{1/4} L_j \leq 2/3\times$ floor width
C_j	=	1.0 for interior panels
		2.0 for edge panels
W_j	=	the weight of the beam panel and is calculated from equation 4.2 on page 17 of the AISC Design Guide and page 21 left side:
		$W_j = w_j B_j L_j$ (x 1.5 if continuous)

For the girder mode, the terms reported include f_g , Δ_j , w_g which were explained earlier, and, D_g , B_g , and W_g which are described below.

where

D_g	=	the girder transformed moment of inertia per unit width described on page 18 of the AISC Design Guide:
		$= I_g/L_j$ for all but edge girders
		$= I_g/2L_j$ for edge girders
B_g	=	the effective width for the girder panel mode and is determined by equation 4.3b on page 18:
		$= C_g (D_g / D_j)^{1/4} L_g \leq 2/3\times \mathit{floor\ width}$ for interior panels
		$= 2/3\ L_g$ for edge panels
C_g	=	defined on page 18 as:
		= 1.6 for girders supporting joists connected to girder flange (e.g., joist seats)
		= 1.8 for girders supporting beams connected to the girder web.
W_g	=	the weight of the girder panel and is calculated by equation 4.2 on page 17 and described on page 21:
		$= w_g B_g L_g$ (x 1.5 if continuous)

For the combined mode of vibration the parameters reported are f_n , W, β, Peak Acceleration and Acceleration Limit.

f_n is calculated from equation 2 shown above.

W is the equivalent panel weight in the combined mode and is calculated from the equation shown on page 21 of the AISC Design Guide:

W

β is the value of the damping ratio as per Table 4.1 on page 18 of AISC Design Guide.

The peak acceleration due to walking excitation is then determined from the equation 4.1 on page 17 and on page 21 of the AISC Design Guide:

where

a_p	=	peak acceleration value due to walking excitation
P_0	=	a constant force representing excitation and is determined as per Table 4.1 of the Design Guide.
f_n	=	fundamental natural frequency in combined mode

The acceleration limit is determined from Table 4.1 on page 18 of the Design Guide.

Related Links

- *Floor Vibration Output dialog* (on page 3002)
- *P. To generate a floor vibration report* (on page 1996)
- *M. Composite Decks* (on page 676)

P. Reviewing Pushover Analysis Results

This section describes how to view some of the specific results from a pushover analysis.

Once a successful pushover analysis is performed, pushover results are available in the **Postprocessing** worfklow.

P. To review pushover load steps

To review the pushover load applied at each load step, use the following procedure.

1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Loads** tool in the **Pushover** group.
   The **Load Value** table opens.
3. Either type a integer or use the arrows in the **Select Load Step** field.
   The pushover loads at each node and base shear are displayed for this load step.

P. To view capacity curve and determine target displacement

To determine the performance point of a pushover analysis for a given performance level, use the following procedure.

The capacity curve displays the plot of the base shear versus the displacement at control joint.

1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Graphs** tool in the **Pushover** group.
   The Capacity Curve graph (on page 2013) and table opens.
3. Right-click on the **Capacity Curve** graph and select **Determine Target Displacement** from the pop-up menu.
   The **Define Modification Factor C0** dialog opens.
4. Specify the Modification Factor C0:
   a. Either:

To...	Do the following...
use a table value	select the **Select from Table** option and then select a C0 factor value in the table
use an interpolated value	select the **Define** option and then select or type the specific building parameters using the options

   The **Modification Factor C0** value to be used is displayed in the dialog.
   b. Click **Next.**

   The **Define Cm to Calculate Modification Factor C1** dialog opens.
5. Specify the Modification Factor C1:
   a. Select the Damping ratio (in percent) from the drop-down list.

   b. Select the effective mass factor, Cm, to use by clicking the value in the table.
      The **Cm** value to be used is displayed in the dialog.
   c. Click **Next**.

   The **Define Modification Factor C2** dialog opens.
6. Specify the Modification Factor C2:
   a. Select the **Structure Performance Level** from the drop-down list.
   b. Select the **Framing Type** from the drop-down list.
      The **Modification Factor C2** value to be used is displayed in the dialog.
   c. Click **Finish**.

   The calculated target displacement and base shear at target displacement values are displayed on the **Capacity Curve** graph.
7. (Optional) Right-click on the **Capacity Curve** graph and select **Show Idealized Capacity Curve** from the pop-up menu.
   The idealized capacity curve is shown in blue on the graph, along with the curve parameters. These include the slopes of the linear portions of the idealized capacity curve along with the base shear and deflection at the inflection point.
8. (Optional) Repeat steps 3 through 6 and select a different **Structure Performance Level** as needed.

P. To display pushover node results

To display the nodal displacements and support reactions for a pushover analysis, use the following procedure.

1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Node Results** tool in the **Pushover** group.
   The **Node Displacements** and **Support Reactions** tables open.
3. Either type a integer or use the arrows in the **Select Load Step** field.
   The tables display the results for this load step. The view window displays the deflected shape of the structure for this load step.

P. To review pushover beam results

Once a successful pushover analysis is performed, pushover results are available in the **Postprocessing** worfklow.

1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Loads** tool in the **Pushover** group.
   The **Beam Hinge Results** and **Beam Force Detail** tables open.
3. Either type a integer or use the arrows in the **Select Load Step** field.
   The values in the tables are updated for this load step. The

The hinges are graphically displayed on the structure for each load step using the color coding for acceptance criteria:

- (Green) Initial Hinge format at the top of the elastic range, represents Immediate Occupancy.
- (Blue) Hinge in the Life Safety range
- (Purple) Hinge in the Collapse Prevention range.
- (Red) Hinge greater than Collapse Prevention range.

Related Links

- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *Capacity Curve graph* (on page 2013)

P. Pages in the Postprocessing Workflow

- Displacements
- Reactions
- Beam Results
- Plate Results
- Solid Results
- Dynamics
- Reports

P. Node Results - Layouts

Layout	Purpose
Displacement	Displays the nodal and beam sectional displacements, along with the P. Node Displacements table (on page 2002) and P. Beam Relative Displacement Detail table (on page 2003). This has the same effect as selecting the **Displacements** page in the Postprocessing workflow.
Reaction	Displays the support reactions, along with the P. Support Reactions table (on page 2003) and P. Statics Check Results table (on page 2004). This has the same effect as selecting the **Reactions** page in the Postprocessing workflow.
Instability	Displays unstable joints if any were detected in the analysis along with the P. Unstable Joints table (on page 2004). **Note:** This tool is only active if instabilities are detected during the analysis.
Base Pressure	Displays the soil pressure at support node locations for slabs consisting of plate elements generated using either the elastic mat or plate mat commands along with the P. Base Pressure table (on page 2004). **Note:** This tool is only active if `ELASTIC MAT` or `PLATE MAT` commands are present in the input file.

P. **Node Displacements** table

Displays the displacements and rotations for each node for the selected load case. For geometric nonlinear analysis, you can select the load step for which you want to display the displacements.

Opens when the **Displacements** page is selected in the **Postprocessing** workflow.

Select Load Step	Select the Load Step value to display in the view window, **Node Displacement Curve** dialog, and the **Node Displacements** table. The Maximum Number of Load Steps for the current load case is displayed on the right-hand side of the table controls.
Limit Maximum Load Step for Graph	Select this option to limit the Load Level scale on the Node Displacement Curve to the Selected Load Step.

All tab

This tab presents all nodal displacements in tabular form for all load cases and all degrees of freedom.

Summary tab

The Summary tab shown below, presents the maximum and minimum nodal displacements (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

Related Links

- *P. To display displacements* (on page 1979)
- *P. To display the deflect shape* (on page 1980)

P. **Beam Relative Displacement Detail** table

Used to view numerical values for sectional displacements along the length of members.

Opens when the **Displacements** page is selected in the **Postprocessing** workflow.

Note: You can set custom increments for beam results for using the **Increments** input on the Results Setup dialog (on page 2999).

All Relative Displacement tab

Displays the displacements of members at intermediate section points. All specified members and all specified load cases are included. The table shows displacements along the local axes of the members, as well as their resultants.

Max Relative Displacement tab

Displays the summary of maximum sectional displacements. This table includes the maximum displacement values and location of its occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the member to the resultant maximum section displacement of the member.

P. **Support Reactions** table

Used to display values of support reactions in tabular format. Three tabs help organize the results: All, Summary, and Envelope.

Opens when the **Reactions** page is selected in the **Postprocessing** workflow.

All tab

Displays the reactions at support nodes for all load cases.

Summary tab

Displays the maximum and minimum support reactions (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

Envelope tab

Displays maximum and minimum support reaction envelopes along with the Load Case number that caused the reaction. All degrees of freedom and all specified Load Cases are considered.

Related Links

- *P. To display the support reaction values* (on page 1980)

P. **Statics Check Results** table

Displays the global force and moment load and reaction totals.

Opens when the **Reactions** page is selected in the **Postprocessing** workflow.

Note: You can set custom increments for beam results for using the **Increments** input on the Results Setup dialog (on page 2999).

P. **Unstable Joints** table

Used to view the nodes and degrees of freedom associated with instabilities.

Opens when the **Layouts** > **Instability** tool in the **View Results** group on the **Results** ribbon tab is selected.

The locations of instabilities are highlighted on the structure in large blue circles. The Unstable Joint table provides information on the joint number and the degree of freedom associated with an instability at that joint. An unstable condition is marked with a “1”.

P. **Base Pressure** table

Displays the numeric values for base pressure at support nodes.

Opens when the **Layouts** > **Base Pressure** tool in the **View Results** group on the **Results** ribbon tab is selected.

All tab

Displays a table containing pressures, along global X, Y and Z directions for all load cases, at each support node where a spring is generated.

Summary tab

Displays a table containing a) the maximum and minimum pressure along each of the 3 global directions b) the support node where they occur c) the load case responsible for that value.

P. Beam Results - Layouts

Layout	Purpose
Beam Forces	Displays the force diagram on the structure for the selected load case along with the P. Beam End Forces table (on page 2005) and **Beam Force Detail** table. This has the same effect as selecting the **Beam Results** page in the Postprocessing workflow.
Beam Stress	Displays the member stress diagram on the structure (same as the **Beam Stress** tool), along with the **3D Beam Stress Contour** view and P. Beam Combined Axial and Bending Stress table (on page 2007).
Utilization	Displays the design code based utilization ratio, for code checks and member selection operations, in annotated form on the structure diagram (same as the **Utilization** tool) along with the P. Design Results table (on page 2008).
Graphs	Displays the current loading on the structure in the main view along with a set of three Beam Graph windows which display force or bending diagrams for the currently selected member.

P. **Beam End Forces** table

This table displays the numerical values of the forces that are applied on to a beam from the nodes that define its start and end. There are thus two rows of data per member per load case. The range of beams and load cases that are included in the table are selected from the Results Setup dialog,(see Results Setup dialog (on page 2999)).

The units for the data are defined in the **Display Options** dialog > Force Units sheet as the Force unit for the FX, FY and FZ results and Moment for the MX, MY and MZ results. Note that it is possible to select an alternative unit for forces or moments by right-clicking over a value and selecting the unit from the options in the pop-up menu 'Change Unit for this column'

This table, along with the **Beam Force Detail** table is opened when the **Beam Results** page is selected. The table has three tabs which allows the data to be displayed thus:

Opens when the **Beam Results** page is selected.

All tab

The results for all load cases and beams selected in the Results Setup dialog are displayed. Each result being the force on the node as applied to the beam.

Summary tab

This tab presents maximum and minimum End Force results for each degree of freedom. All specified members and all load cases are considered.

In order to account for the fact that the direction of the forces is inverted at the far end of the beam to induce an equivalent force on the beam, the signs of the force at the far end of the beam are first inverted and the maximum/minimum values are then determined.

For example consider a beams which have been loaded in two load cases such that the first is in compression by 10kN and in the second is in tension by 5kN. This would be reported in the All results tab thus:

Beam	L/C	Node	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
1	1 Compression	1	10.000	0.000	0.000	0.000	0.000	0.000
		2	-10.000	0.000	0.000	0.000	0.000	0.000
	2 Tension	1	-5.000	0.000	0.000	0.000	0.000	0.000
		2	5.000	0.000	0.000	0.000	0.000	0.000

As the values of the far end results are inverted, the value of Max FX reports the maximum compression and Min FX reports the maximum tension thus:

	Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx kip-in	My kip-in	Mz kip-in
Max Fx	1	1 Compression	1	10.000	0.000	0.000	0.000	0.000	0.000
Min Fx	1	2 Tension	1	-5.000	0.000	0.000	0.000	0.000	0.000

Note: The inversion of the far end results applies to all degrees of freedom to provide similar insights on the forces experienced on the beam.

Envelope tab

This tab reports the details for each beam which load case provides the most positive and most negative effect for each degree of freedom, (i.e. 8 results per beam).

This is displayed on 4 rows,

1. On the start node:
   a. The most positive effect in each degree of freedom.
   b. The load case that produce the most positive effect.
   c. The most negative effect in each degree of freedom.
   d. The load case that produce the most negative effect.
2. On the far node:
   a. The most positive effect in each degree of freedom.
   b. The load case that produce the most positive effect.
   c. The most negative effect in each degree of freedom.
   d. The load case that produce the most negative effect.

If there is no load case that produces a positive or negative effect, that is indicated with “-” as the load case.

P. **Beam Combined Axial and Bending Stress** table

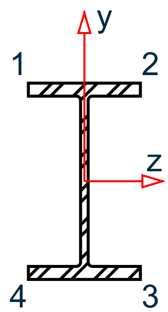
Presents the combined bending stress results at key cross-section points along the member length for each load case.

Opens when the **Layouts** > **Beam Stress** tool in the **View Results** group on the **Results** ribbon tab is selected.

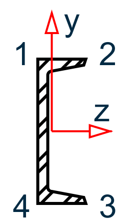
All tab

This tab presents combined axial and bending stress at four corners of the member cross section at the specified number of intermediate sections. The Maximum Compressive and Maximum Tensile stresses are also listed. The result is given for all selected Load Cases.

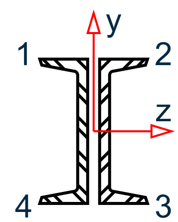
The points on the cross-section where the stresses are reported are identified by the numbers 1, 2, 3, and 4. The locations of these points are identified in the following figure by section type.



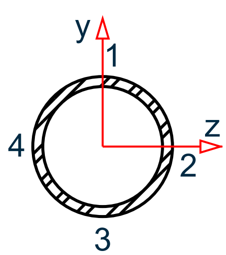
I shape



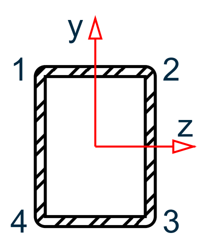
channel



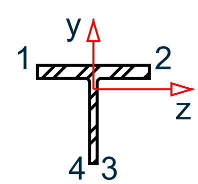
double channel



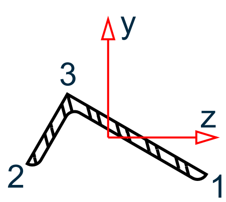
pipe



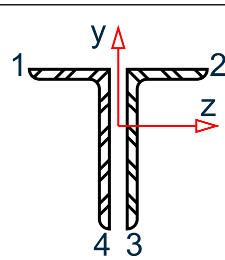
tube



T shape



single angle



double angle

Max Stresses tab

This tab presents the Maximum Compressive and Maximum Tensile Stresses for all Members for all selected Load Cases. The stresses at all intermediate sections and at both ends are considered. The point of occurrence (position along length) of the Maximum stress is listed in the Length column. The corner of the beam at which the stress occurs is listed under the Corner column.

Profile Stress Points tab

The forces and stresses (Combined Axial and Bending Stresses) can be obtained at certain selected points on a cross section and can be added to the table. Type in the **Dist** value, local Y distance on the cross section (**Stress Pt(Y)**), and the local Z distance on the cross section (**Stress Pt(Z)**).

P. **Design Results** table

This table presents the results from a steel or aluminum design in a tabular form.

Opens when the page is selected.

The results given are the **Design Property** designated by a member selection, **Utilization Ratio** for the member from a code check or member selection operation, cross sectional areas about the member's local axes, **Ay**, **Az**, **Ax**, web depth **Dw**, flange width **Bf**, and moments of inertia about the member's local axes, **Iz**, **Iy** and **Ix**.

All tab

This table displays the following:

a. The member number
b. The cross-section name used for the initial analysis and for the final design result is displayed. In the event of multiple operations, the cross-section used in the last operation is displayed.
c. Actual and allowable design ratios. When a `RATIO` parameter has been specified, this is listed as the allowable value.
d. The normalized ratio is given, which is the actual ratio divided by the allowable. This is the critical value that indicates the suitability of the member. Normally, a value higher than 1.0 indicates the extent to which the member is over-stressed and a value below 1.0 indicates the reserve capacity available.
e. The member property values of the section used in the design operation.

Failed Members tab

Shows the values for only those members which have failed from the standpoint of the code requirements. These usually have a utilization ratio higher than 1.0.

P. Plate Results - Layouts

Layout	**Purpose**
Plate Stress	Displays the graphical plate stress contour diagram superimposed on the elements along with the P. Plate Corner Stress table (on page 2009) and P. Plate Center Stress table (on page 2009). This item will also open the Diagrams dialog to the Plate Stress Contour tab.

Layout	Purpose
Results Along a Line	Opens the **Results Along Line** dialog along with the line graph and **Result along selected line** table.

P. **Plate Corner Stress** table

This table displays the stresses at corner nodes of plate elements.

Opens when the **Plate Results** page is selected.

Shear, Membrane and Bending tab

Displays all Shear, Membrane, and Bending Stresses for all specified plates and for all specified Load Cases. Stresses corresponding to all the nodal points of the plates are tabulated.

Stress Envelope

Displays the maximum positive plate stress and maximum negative plate stress values (shear, membrane, and bending) at each corner node for the envelope.

P. **Plate Center Stress** table

This table displays the stresses at the center of plate elements.

Opens when the **Plate Results** page is selected.

Shear, Membrane and Bending tab

This tab presents all Shear, Membrane and Bending Stresses for all specified Plates for all specified Load Cases.

Stress Summary tab

This table provides results of maximum Shear, Membrane and Bending stresses. The Plate element number and associated Load Case number are listed.

Stress Envelope

Displays the maximum positive plate stress and maximum negative plate stress values (shear, membrane, and bending) at each plate center for the envelope.

Principal and Von Mis tab

This table provides Principal and Von Mises Stresses for all specified Plates for all specified Load Cases similar to the Shear, Membrane and Bending tab.

Principal Summary tab

This table provides results of maximum Principal and Von Mises stresses in each row. The Plate element number and the associated Load Case number are also listed.

Principal Envelope

Displays the maximum positive and maximum negative Principal and Von Mises stresses values at each plate center for the envelope.

Global Moments tab

Depending on the various parameters specified by the user, for those plates whose plane is parallel to the selected global plane, the local axis moments MX, MY and MXY can be transformed into a moment about the selected global axis and plotted.

Combined Stress

P. Solid Results - Layouts

Layout	**Purpose**
Plate Stress	Displays the graphical Solid Stress Contour diagram superimposed on the elements along with the P. Solid Corner Stress table (on page 2010) and the P. Solid Center Stress table (on page 2011). This has the same effect as selecting the **Solid Results** page in the Postprocessing workflow. This item will also open the Diagrams dialog to the Solid Stress Contour tab.

P. **Solid Corner Stress** table

This table presents stresses at corner nodes of solid elements.

Opens when:

- the **Solid Results** page is selected in the Postprocessing workflow
- the **Solid Stress** tool is selected from the **View Results** group on the **Results** ribbon tab

- the **Layouts** > **Solid Stress** tool is selected from the **View Results** group on the **Results** ribbon tab

'

Note: The stress values displayed in the Postprocessing workflow tables are “smoothed” by averaging stresses from adjacent elements. Therefore, they may not be equal to those corner stresses reported in the STAAD.Pro `.anl` for the same model.

Shear, Member and Bending tab

This tab presents all Shear, Membrane and Bending Stresses for all specified solids for all specified Load Cases.

(Shear, Membrane and Bending) Summary tab

This table provides results of maximum Shear, Membrane and Bending stresses. The solid element number, node number and associated Load Case number are listed.

Principal and Von Mis tab

This table provides Principal and Von Mises Stresses for all solids for all Load Cases similar to the Shear, Membrane and Bending tab.

(Principal and Von Mis) Summary tab

This table provides results of maximum Principal and Von Mises stresses in each row. The solid element number and the associated Load Case number are also listed.

Related Links

- *P. To display solid results contours* (on page 1990)

P. **Solid Center Stress** table

This table presents stresses at center of solid elements.

Opens when:

- the **Solid Results** page is selected in the Postprocessing workflow
- the **Solid Stress** tool is selected from the **View Results** group on the **Results** ribbon tab

- the **Layouts** > **Solid Stress** tool is selected from the **View Results** group on the **Results** ribbon tab

'

Note: The stress values displayed in the Postprocessing workflow tables are “smoothed” by averaging stresses from adjacent elements. Therefore, they may not be equal to those corner stresses reported in the STAAD.Pro `.anl` for the same model.

Shear, Member and Bending tab

This tab presents all Shear, Membrane and Bending Stresses for all specified solids for all specified Load Cases.

(Shear, Membrane and Bending) Summary tab

This table provides results of maximum Shear, Membrane and Bending stresses. The solid element number, node number and associated Load Case number are listed.

Principal and Von Mis tab

This table provides Principal and Von Mises Stresses for all solids for all Load Cases similar to the Shear, Membrane and Bending tab.

(Principal and Von Mis) Summary tab

This table provides results of maximum Principal and Von Mises stresses in each row. The solid element number and the associated Load Case number are also listed.

Related Links

- *P. To display solid results contours* (on page 1990)

P. Geometric Nonlinearity - Layouts

Layout	Purpose
Loads	Displays the deflected shape for the selected load case and load step, along with the **Node Displacements Curve** graph and **Node Displacement** table.
Graph	Opens the Load Level vs Section Forces diagram and **Beam Force Detail** table.

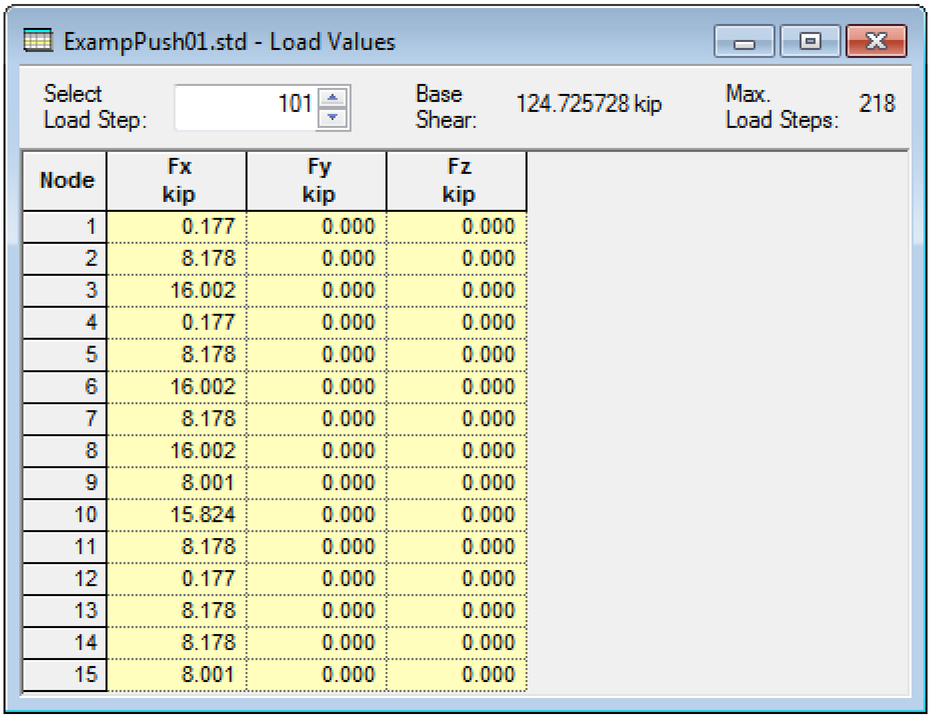
P. Pushover - Layouts

The pushover analysis results layouts in the post-processing mode are described here. These pages are available only upon a successful pushover analysis.

Layout	Purpose
Loads	Opens the (Pushover) Load Values table (on page 2012), which displays the nodal load values and total base shear for the selected load step, along with the **Capacity Curve** table. The magnitude and direction of the loads in the current load step are displayed in the view window.
Graph	Opens the Capacity Curve graph (on page 2013), which displays displacement vs. force for the pushover analysis. The **Capacity Curve** table contains the roof or control joint displacement and base shear values recorded for each load step.
Node Results	Displays nodal displacements and rotations in the (Pushover) Node Displacements table (on page 2014) and the support reactions in the (Pushover) Support Reactions table (on page 2015) for the selected load step. The deflected shape is overlaid on the structure in the view window.
Beam Results	Opens the (Pushover) Beam Hinge Results table (on page 2016) which displays the status of the beam and hinge conditions along each beam's length. The (Pushover) Beam Force Details table (on page 2017) displays the cross-sectional forces and moments at sections along each beam's length are displayed for the selected load step. The status of beams and hinges are displayed in the view window.

(Pushover) **Load Values** table

Displays the nodal load values and total base shear for the selected load step.



Node	Fx kip	Fy kip	Fz kip
1	0.177	0.000	0.000
2	8.178	0.000	0.000
3	16.002	0.000	0.000
4	0.177	0.000	0.000
5	8.178	0.000	0.000
6	16.002	0.000	0.000
7	8.178	0.000	0.000
8	16.002	0.000	0.000
9	8.001	0.000	0.000
10	15.824	0.000	0.000
11	8.178	0.000	0.000
12	0.177	0.000	0.000
13	8.178	0.000	0.000
14	8.178	0.000	0.000
15	8.001	0.000	0.000

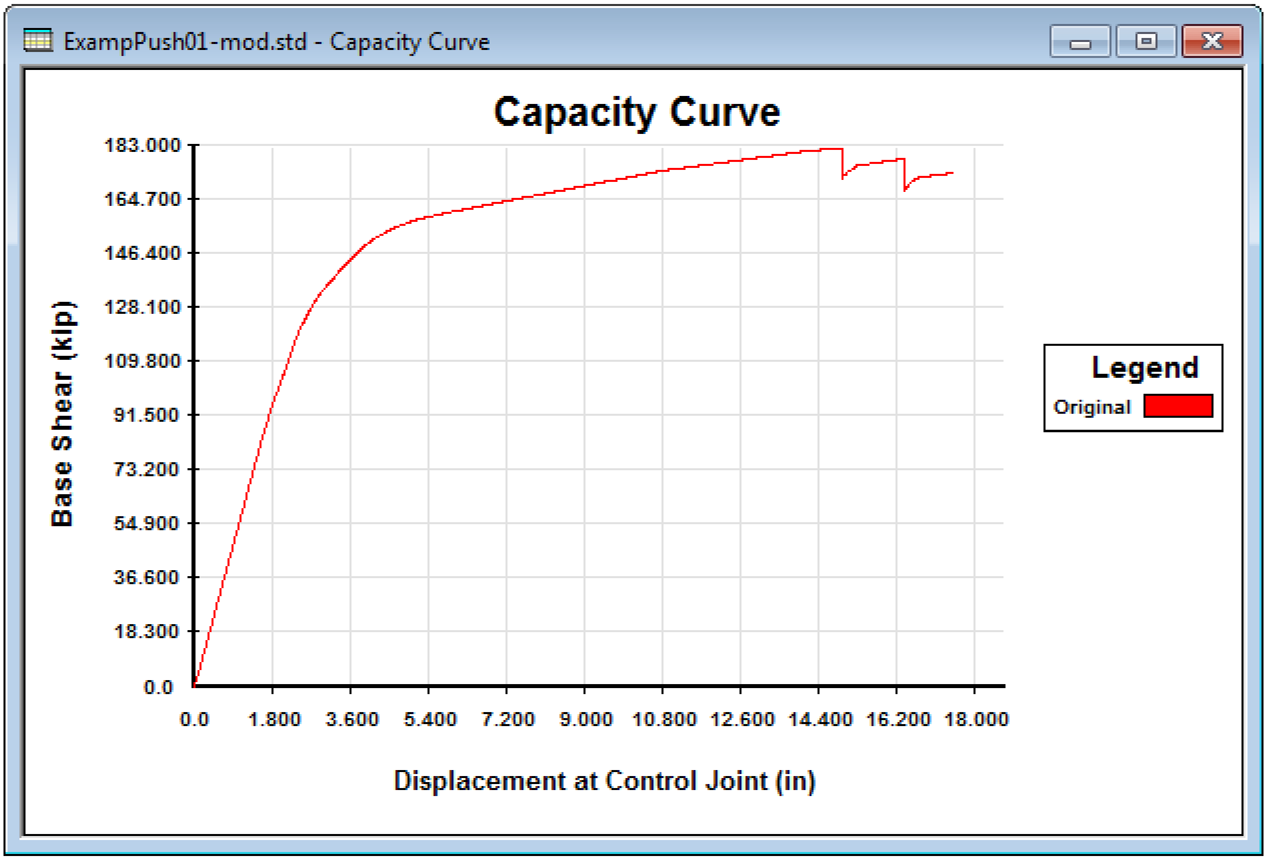
Figure 195: Load Values table

Select Load Step	Use the arrow controls or type the number of the load step. The tabulated load values, base shear, and view window load display update automatically.

Capacity Curve graph

Displays the displacement versus base shear graph for the pushover analysis. The displacement at control joint is given if a control joint was specified as a solution control method. Otherwise, the displacement is taken largest Y nodes (i.e., the "roof" level).

Opens when the **Pushover | Capacity Curve** page is selected.



The individual values of the curve are displayed in the **Capacity Curve** table.

Related Links

- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *P. To review pushover beam results* (on page 2001)

(Pushover) **Node Displacements** table

Displays the nodal displacements and rotations for the current load step.

ExampPush01.std - Node Displacements

Select Load Step: 101 | Base Shear: 124.725728 kip | Max. Load Steps: 218

Node	X in	Y in	Z in	rX rad	rY rad	rZ rad
1	0.000	0.000	0.000	0.000	0.000	0.000
2	0.719	0.011	0.000	0.000	0.000	-0.008
3	0.723	-0.023	0.000	0.000	0.000	-0.007
4	0.000	0.000	0.000	0.000	0.000	0.000
5	1.775	0.019	0.000	0.000	0.000	-0.008
6	1.777	-0.041	0.000	0.000	0.000	-0.007
7	2.633	0.023	0.000	0.000	0.000	-0.005
8	2.635	-0.053	0.000	0.000	0.000	-0.005
9	3.173	0.024	0.000	0.000	0.000	-0.004
10	3.170	-0.058	0.000	0.000	0.000	-0.003
11	0.722	-0.036	0.000	0.000	0.000	-0.008
12	0.000	0.000	0.000	0.000	0.000	0.000
13	1.776	-0.062	0.000	0.000	0.000	-0.008
14	2.635	-0.079	0.000	0.000	0.000	-0.005
15	3.170	-0.086	0.000	0.000	0.000	-0.003

Figure 196: Node Displacements table

Select Load Step Use the arrow controls or type the number of the load step. The tabulated node displacement & rotation values, base shear, and view window displacement display update automatically.

(Pushover) **Support Reactions** table

Displays the nodal displacements and rotations for the current load step.

ExampPush01.std - Support Reactions

Select Load Step: 101 | Base Shear: 124.725728 kip | Max. Load Steps: 218

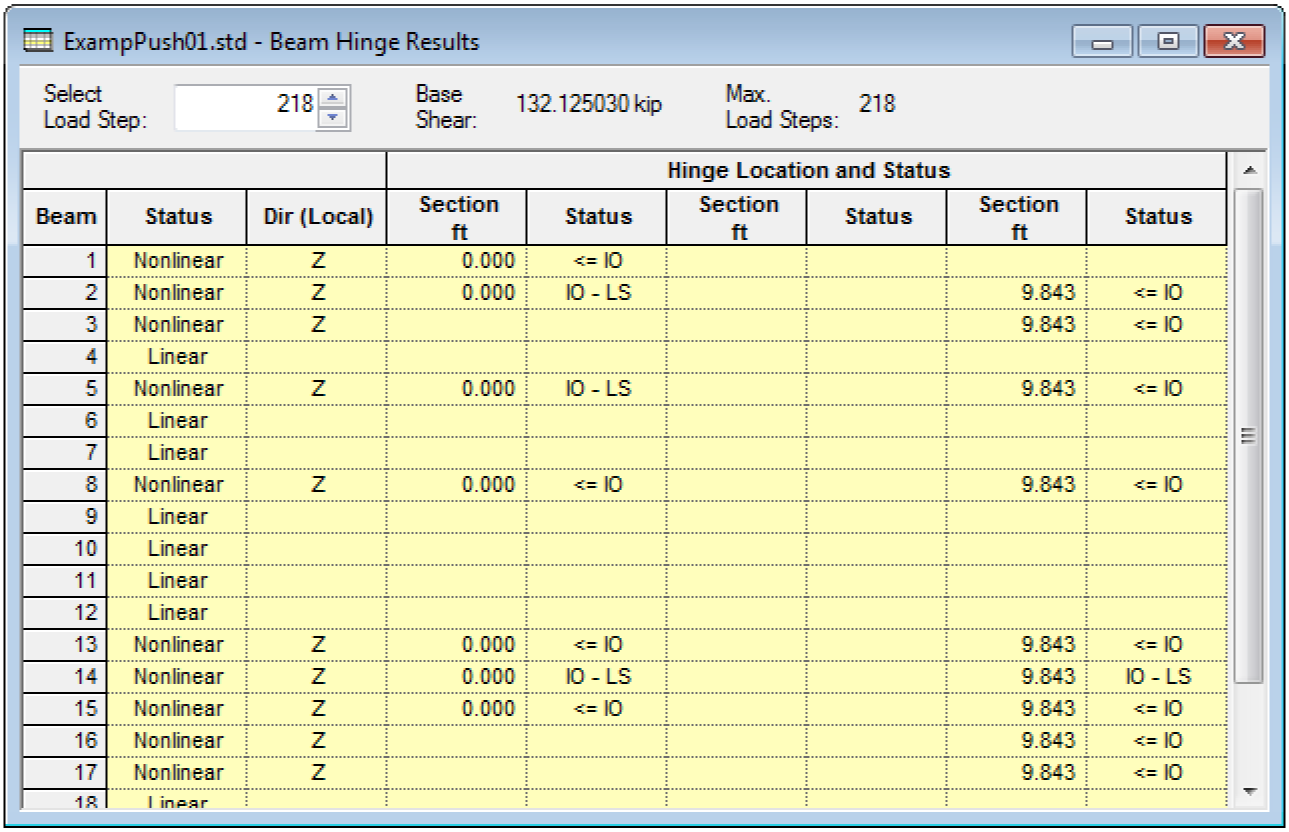
Node	Fx kip	Fy kip	Fz kip	Mx kip-in	My kip-in	Mz kip-in
1	-41.782	-45.601	0.000	0.000	0.000	4813.276
4	-46.467	96.106	0.000	0.000	0.000	4734.615
12	-36.477	147.023	0.000	0.000	0.000	4287.480

Figure 197: Support Reactions table

Select Load Step Use the arrow controls or type the number of the load step. The tabulated reaction values and base shear update automatically.

(Pushover) **Beam Hinge Results** table

Displays the cross section forces and moments at 1/12th points along the beam's length for the current load step.



ExampPush01.std - Beam Hinge Results

Select Load Step: 218 Base Shear: 132.125030 kip Max. Load Steps: 218

			Hinge Location and Status					
Beam	Status	Dir (Local)	Section ft	Status	Section ft	Status	Section ft	Status
1	Nonlinear	Z	0.000	<= IO				
2	Nonlinear	Z	0.000	IO - LS			9.843	<= IO
3	Nonlinear	Z					9.843	<= IO
4	Linear							
5	Nonlinear	Z	0.000	IO - LS			9.843	<= IO
6	Linear							
7	Linear							
8	Nonlinear	Z	0.000	<= IO			9.843	<= IO
9	Linear							
10	Linear							
11	Linear							
12	Linear							
13	Nonlinear	Z	0.000	<= IO			9.843	<= IO
14	Nonlinear	Z	0.000	IO - LS			9.843	IO - LS
15	Nonlinear	Z	0.000	<= IO			9.843	<= IO
16	Nonlinear	Z					9.843	<= IO
17	Nonlinear	Z					9.843	<= IO
18	Linear							

Figure 198: Beam Hinge Results table

Select Load Step Use the arrow controls or type the number of the load step. The tabulated section forces & moments, base shear, and view window hinge display update automatically.

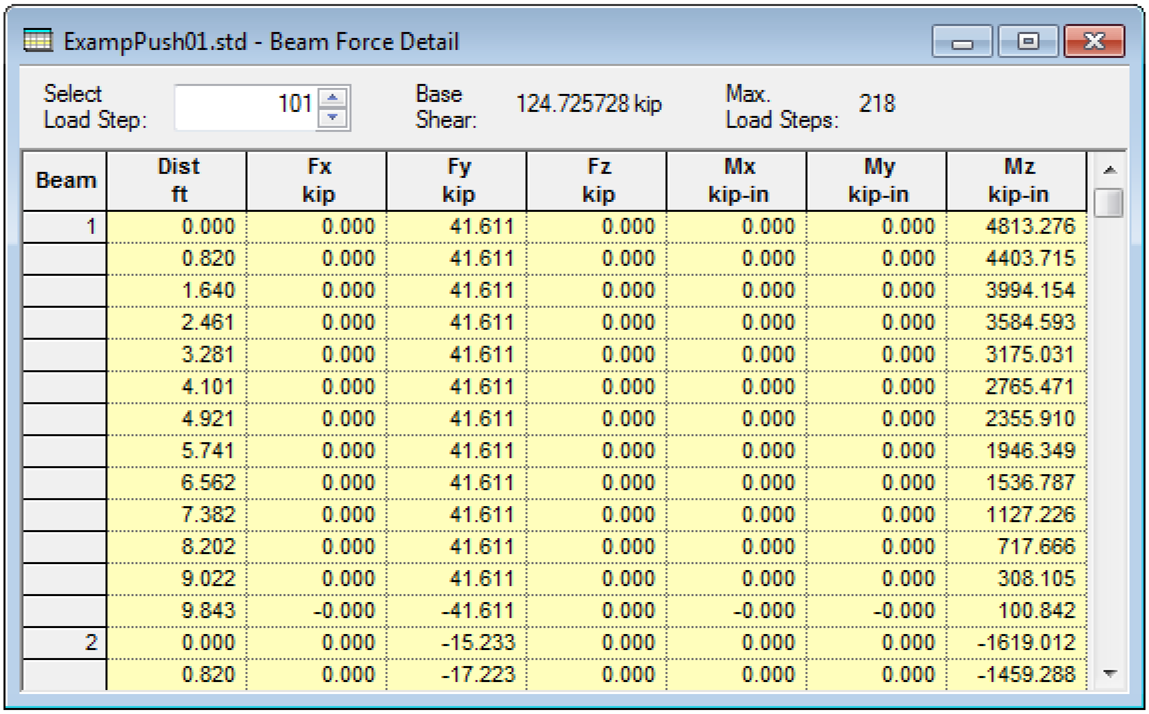
table The table displays the overall beam status as well as the location and status of any hinges which have formed. Up to three hinges may be listed. The hinge location (Section) and Status are displayed in physical order, from start to end node, of their location along the beam.

The table contains the following columns:

- Status - The material status of the beam for current load step. Linear indicates no hinges have formed in the beam. Nonlinear indicates that one or more hinges have formed. Inactive means that three hinges have formed or that one or hinges are beyond the collapse prevention (CP) limit.
- Dir (Local) - the direction of bending for resulting in the formation of hinges.
- Hinge Section & Status - The location along the length of the hinge and the acceptance criteria status. The location and status of hinges are displayed graphically on the structure in the view window.

(Pushover) **Beam Force Details** table

Displays the cross section forces and moments at 1/12th points along the beam's length for the current load step.



ExampPush01.std - Beam Force Detail

Select Load Step: 101 | Base Shear: 124.725728 kip | Max. Load Steps: 218

Beam	Dist ft	Fx kip	Fy kip	Fz kip	Mx kip-in	My kip-in	Mz kip-in
1	0.000	0.000	41.611	0.000	0.000	0.000	4813.276
	0.820	0.000	41.611	0.000	0.000	0.000	4403.715
	1.640	0.000	41.611	0.000	0.000	0.000	3994.154
	2.461	0.000	41.611	0.000	0.000	0.000	3584.593
	3.281	0.000	41.611	0.000	0.000	0.000	3175.031
	4.101	0.000	41.611	0.000	0.000	0.000	2765.471
	4.921	0.000	41.611	0.000	0.000	0.000	2355.910
	5.741	0.000	41.611	0.000	0.000	0.000	1946.349
	6.562	0.000	41.611	0.000	0.000	0.000	1536.787
	7.382	0.000	41.611	0.000	0.000	0.000	1127.226
	8.202	0.000	41.611	0.000	0.000	0.000	717.666
	9.022	0.000	41.611	0.000	0.000	0.000	308.105
	9.843	-0.000	-41.611	0.000	-0.000	-0.000	100.842
2	0.000	0.000	-15.233	0.000	0.000	0.000	-1619.012
	0.820	0.000	-17.223	0.000	0.000	0.000	-1459.288

Figure 199: Beam Force Details table

Select Load Step — Use the arrow controls or type the number of the load step. The tabulated section forces & moments, base shear, and view window hinge display update automatically.

P. Generating Reports

P. To setup report contents

To select the items to include in a STAAD.Pro report, use the following procedure.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Report** tab and then click **Setup**.
   The **Report Setup** dialog opens.
3. Select the **Items** tab and then specify the sections to include in the report.

   The list on the right contains the included report items in the order they will appear in the report.

   a. Select a report category in the drop-down list.
      The list of available items updates.

b. Either:

Select one or more items in the available list and then click **>** to add to the report.

Tip: Hold **<Ctrl>** to click on multiple items. Hold **<Shift>** to select a series of adjacent items.

or

Click **>>** to include all items within the report category in the report.

c. Repeat steps 3a and 3b to include items from additional report categories.

4. (Optional) Use the up and down buttons to reorder items in the report list.
5. (Optional) Click **Print** to output the report immediately.
   The Print dialog opens to select the system printer.
6. Click **OK**.

You can limit the report contents by load cases, modes (for dynamic results), and ranges of objects.

Related Links

- *Report Setup dialog* (on page 2697)

P. To add a custom header and logo to reports

To add custom header text and a logo for your organization to your reports, use the following procedure.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Report** tab and then click **Setup**.
   The **Report Setup** dialog opens.
3. Select the **Name and Logo** tab.
4. In the text field, type the custom header text.
5. (Optional) Customize the text placement and appearance:
   a. Click **Font** to select a font face, style, and size.
   b. Select the font **Alignment**.
      This is active when the logo is aligned to right or left.
6. Click **File** to locate and select a bitmap image (file extension `.bmp`) to use for your header logo.

Related Links

- *Report Setup dialog* (on page 2697)

P. Steel AutoDrafter Workflow

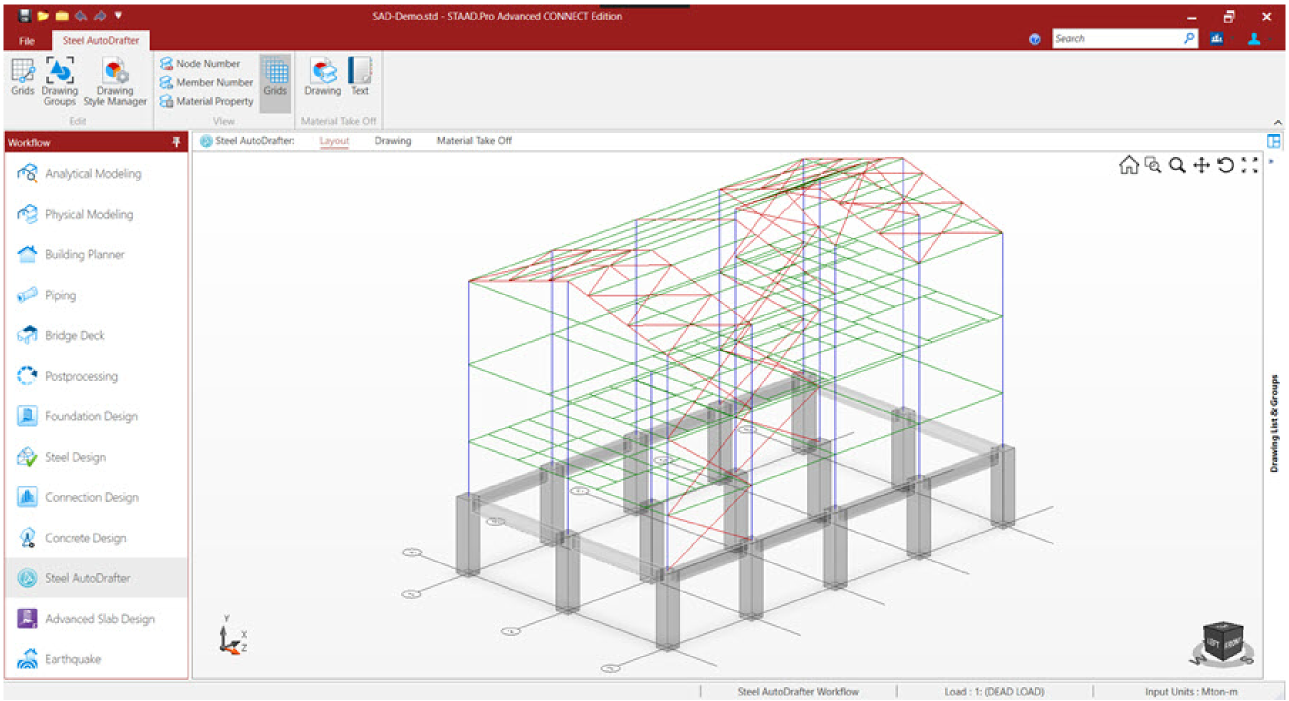
The Steel AutoDrafter workflow extracts planar drawings and material take-off from a structural steel model prepared in STAAD.Pro.

Steel AutoDrafter produces plans at any level and sections in any of the orthogonal directions.

Note: The model needs to be analyzed and saved before using Steel AutoDrafter workflow.

Note: Steel AutoDrafter requires models with Y-up orientation only. Z-up models are not supported.

Overview



Related Links

RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10

- *RR 21.03.00-3.1 Static Seismic Loads per IBC 2015 / ASCE 7-10* (on page 136)

P. To open the Steel AutoDrafter workflow

You must perform an analysis before using the Steel AutoDrafter workflow.

1. Perform a successful analysis.
2. Select **Steel AutoDrafter** in the **Workflows** panel.

The **Steel AutoDrafter** ribbon tab and the Steel AutoDrafter workflow **Layout** page open.

Related Links

- *Steel AutoDrafter tab* (on page 3003)

P. To configure units in Steel AutoDrafter

To change the system of units for the interface as well as in drawings and material take-off, use the following procedure.

1. On the **Steel AutoDrafter** ribbon tab, select the **Drawing Style Manager** tool in the **Edit** group.

   The **Drawing Style Manager** dialog opens.
2. Select the **Appearance** tab.
3. Select the **Drawing Unit** to use.

   Metric

   or

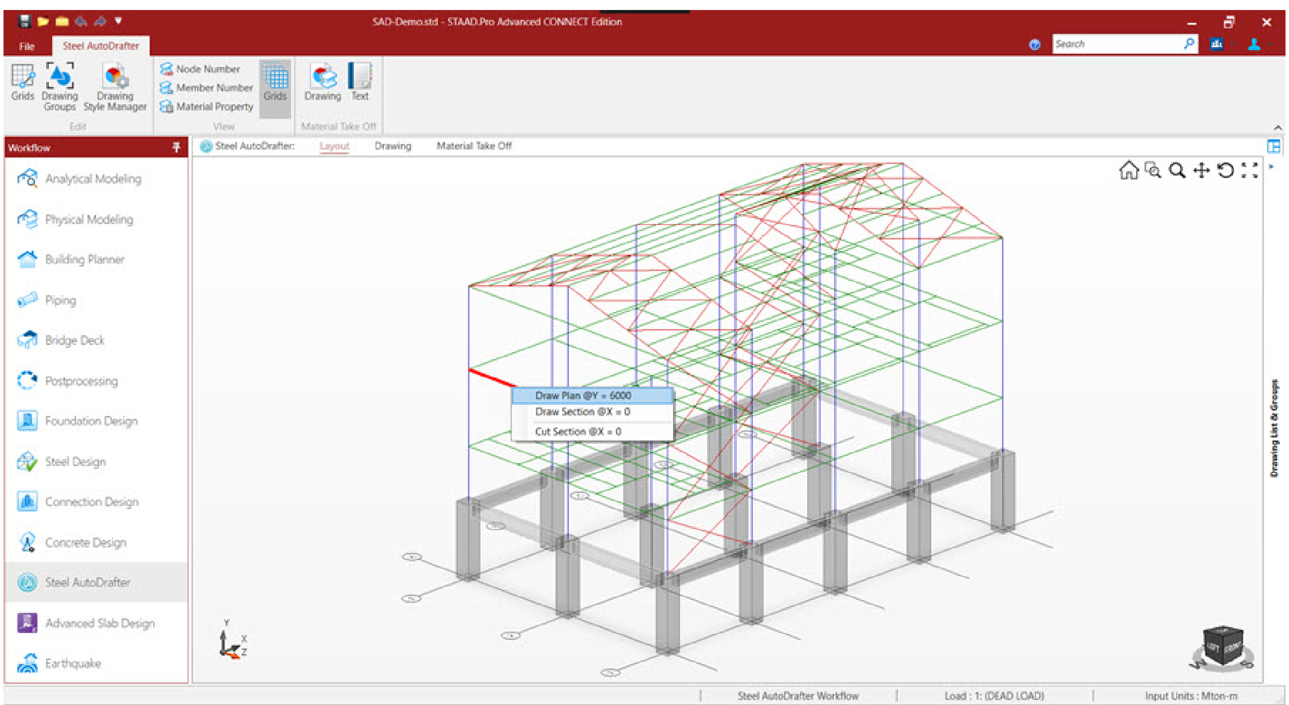
   English
4. Click  **OK**.

Grids, material take-offs, etc. are all now provided in the selected system of units.

Related Links

- *Drawing Style Manager dialog* (on page 3006)
- *Member Labels* (on page 3007)

P. Drawing



Drawing Logic

For built-up section made of two members, batten plates are drawn as indicative.

For connection between concrete and steel, indicative base plates are shown.

For elevation Bracing's the angle is adjusted as per the change of center line of the beams as per top of steel (TOS).

For bottom chord of the trusses the node between bottom chord and the top chord is adjusted as per the size of the bottom chord member. Accordingly the inclination of the top chord is adjusted without changing the top of truss. Similarly the lengths of the members connecting bottom chord and top chord are adjusted.

Similar to a truss, for inclined portals an indicative joint with connection plates is shown between the column and the portal member.

For moment connections there is no gap provided between the connecting members. For other connections an indicative gap is shown between the connecting members.

Shapes

These shapes are supported:

- Standard rolled steel sections, including T (Tee section) and DT (double tee) sections, are supported
- Section cover plates, TC (top cover plate), BC (bottom cover plate), and TB (top & bottom cover plates), are supported

The following prismatic shapes are also supported:

- Solid Circle (defined with YD),
- Solid Rectangle (defined with YD and ZD),
- Tee (defined with TD, ZD, YB and ZB),
- Trapezoidal (defined with YD, ZD and ZB),
- General (displayed as a rectangular solid using the values of YD and ZD),
- Tapered I (displayed using the average dimensions of web and flange),
- Tapered Tube (displayed using the average dimensions).

Note: The circle, rectangle, trapezoidal, tee, and general sections will be shown as concrete sections. The tapered I and tapered tube will be shown as steel sections.

User-provided table (UPT) shapes:

- All user table sections, such as wide-flange, channel, angle, double angle, etc.
- Shapes imported from Free Sketch as general shapes

These shapes are *not* supported:

- Other shapes with varying cross-sections (not listed under Prismatic sections above)
- Cold-formed steel sections, timber sections, and aluminum sections
- Composite sections (CM),
- American castellated sections, solid rods, and cables, steel joists,
- Dutch plates, strips, solid round sections, and solid square sections

Notes

Drawing generated are as per the members on a given plane, Any members that cut across different planes are not shown.

Curved members are currently not supported.

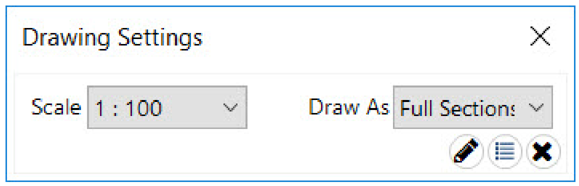
In case if the section defined in STAAD.Pro is not identified by Steel AutoDrafter, Then the member will not be displayed in drawing.

P. To add a plan drawing

To create a plan drawing of members at the same elevation, use the following procedure.

1. Select a horizontal member in the **Layout** page.

   The Y coordinate of the member is used to define the level of the plan. All members at this elevation are included.
2. Right-click and then select **Draw Plan @Y = *elevation*** from the pop-up menu.
   The **Drawing Settings** dialog opens.



3. Select the **Scale** for your drawing from the drop-down list.

   The Scale of drawing can be set as 1:25, 1:50, 1:80, or 1:100.
4. Select either **Full Sections** or **Part Sections** from the **Draw As** drop-down list.

   This will control the extents of how members are drawn.

   - **Full Section -** All edges of the section are drawn for the full length of the member
   - **Part Section -** All edges of the section are drawn for the part length of the member
5. Either:

   click **Draw** to generate the drawing

   or

   click **Add to Drawing List** to add this drawing to the Drawing List tab

The top of steel (TOS) for beams is adjusted as Y coordinate of the floor. Accordingly the center line of the beam is shifted down by half the beam depth.

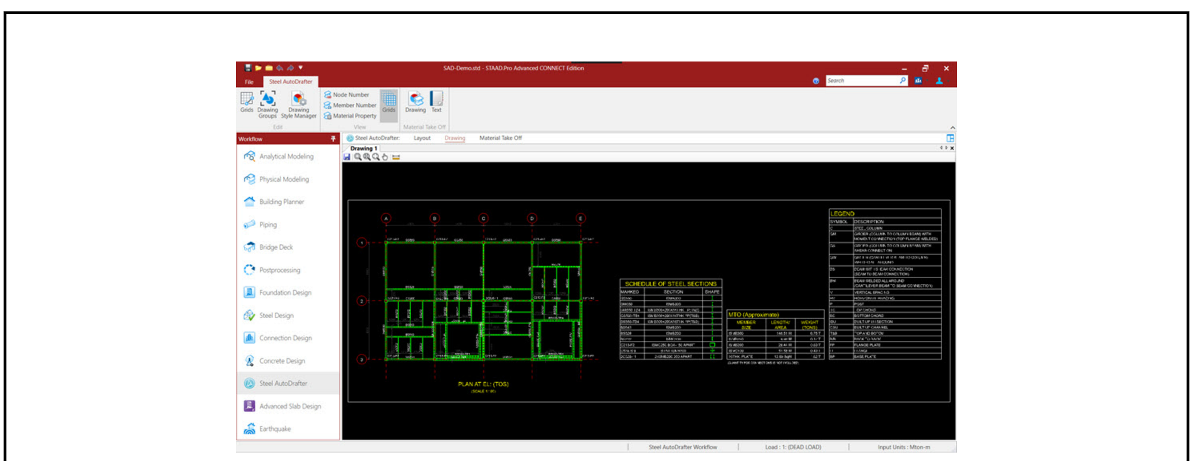


Figure 200: Full section example

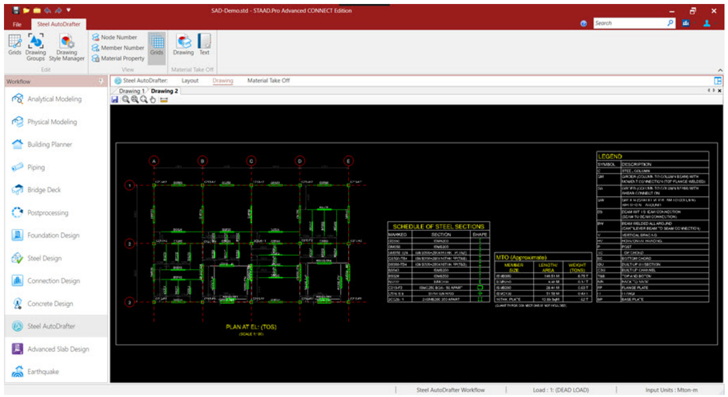


Figure 201: Part section example

Tip: Alternatively, you can use the options in the **Drawing List** tab to select **Plan** from the **Type** drop-down list to create a plan drawing at a selected **Location.**

If you selected to add this drawing to the drawing list, then click the  **Draw** tool below the list on the **Drawing List** tab to generate all of your drawings.

P. To add a section drawing

To add a new section drawing at a vertical or diagonal member, use the following procedure.

In Steel AutoDrafter, a standard "section" drawing is used draw either a elevation or a full section across the entire structure.

1. Select either a vertical or diagonal member in the **Layout** page.

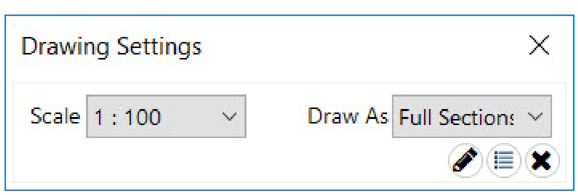
   The Z or X coordinate of this member is used to define the plane of the section. All members in this plane are included.
2. Right-click and then select one of the following options from the pop-up menu:

   Draw Section @X = ***Location***

   or

   Draw Section @Z = ***Location***

   The **Drawing Settings** dialog opens.



3. Select the **Scale** for your drawing from the drop-down list.

   The Scale of drawing can be set as 1:25, 1:50, 1:80, or 1:100.
4. Select either **Full Sections** or **Part Sections** from the **Draw As** drop-down list.

   This will control the extents of how members are drawn.

   - **Full Section -** All edges of the section are drawn for the full length of the member
   - **Part Section -** All edges of the section are drawn for the part length of the member
5. Either:

   click **Draw** to generate the drawing

   or

   click **Add to Drawing List** to add this drawing to the Drawing List tab

> **Tip:** Alternatively, you can use the options in the **Drawing List** tab to select **Section @X** or **Section @Z** from the **Type** drop-down list to create a plan drawing at a selected **Location.**

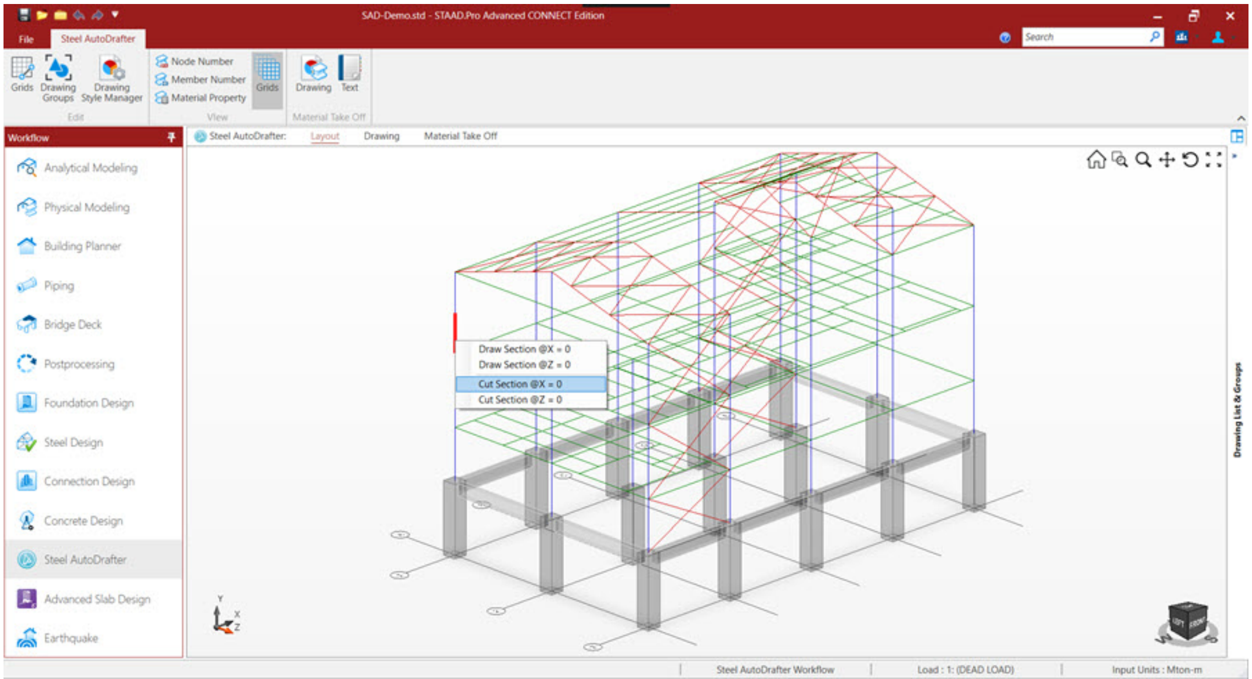
If you selected to add this drawing to the drawing list, then click the **Draw** tool below the list on the **Drawing List** tab to generate all of your drawings.

P. To create a drawing group

To create groups (physical members) for inclined members such as bracing, truss chords, portals, etc., use the following procedure.

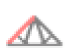
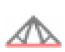
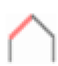
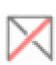
Steel AutoDrafter will identify physical members automatically for vertical and horizontal members.

1. Right-click on a member and select **Cut Section @*`<grid or plane>`***.



The **Section View** dialog opens.

2. Select the group type tool corresponding to the group of members you want to create:

Tool	Description
	Top chord - can include purlins
	Bottom chord
	Portal - can include purlins at a specified spacing
	Bracing
	Inclined Column

The mouse pointer changes to a selection box.

3. Select the members to be grouped graphically:

click on them individually

or

click-and-drag a window box around the members

Tip: Use the view control tools to assist in selecting members.

Members in the selection set are highlighted in red.

4. Right-click anywhere in the **Section View** dialog view area and then select **Create Group** from the pop-up menu.

   For Portal and Top Chord groups, the **Create Group for <group type>** dialog opens. For other groups, the group is created.
5. (Optional) For Portal and Top Chord groups, select the **Purlin Section** shape and section size to use and then click **OK**.

   Note: Purlins are not part of the analysis but this allows you to add these to the drawings for accuracy.
6. Repeat Steps 2 through 5 to create additional groups.
7. Click the **Draw** tool.

   The **Drawing Settings** dialog opens.
8. Select the **Scale** for your drawing from the drop-down list.

   The Scale of drawing can be set as 1:25, 1:50, 1:80, or 1:100.
9. Select either **Full Sections** or **Part Sections** from the **Draw As** drop-down list.

   This will control the extents of how members are drawn.

   - **Full Section -** All edges of the section are drawn for the full length of the member
   - **Part Section -** All edges of the section are drawn for the part length of the member
10. Either:

    click **Draw** to generate the drawing

    or

    click **Add to Drawing List** to add this drawing to the Drawing List tab

Tip: To delete a group, right-click on the group name in either the **Section View** dialog or in the **Drawing Groups** tab of the **Drawing List & Groups** panel and then select **Delete** from the pop-up menu.

Related Links

- *Section View dialog* (on page 2026)

Section View dialog

Used to create groups (physical members) for inclined members in a section.

Opens when **Cut Section** ***@grid or plane*** is selected from the right-click pop-up menu of a selected member in the view area.

A list of group at the current section selection is displayed on the left side tree.

Tip: A list of all groups in the model can be viewed on the **Drawing Groups** tab of the **Drawing Lists & Groups** panel.

The view area of the dialog displays the members in the current section. You can manipulate this section view using the view controls to the right.

View Area Right-click Pop-up Menu

Create Group (available when you have selected members to place in a group type) For bottom chord, bracing, and inclined column group types, the group is created.

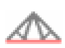
For top chord or portal group types, the **Create Group for <group type>** dialog opens to specify purlin parameters.

Show Node Number Toggles the display of node numbers in the dialog view area.

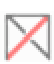
Show Member Number Toggles the display of member numbers in the dialog view area.

Show Member Property Toggles the display of member section sizes in the dialog view area.


Top chord - can include purlins


Bottom chord


Portal - can include purlins at a specified spacing


Bracing


Inclined Column


Opens the **Drawing Settings** dialog, which is used to select the drawing scale, member extents drawn, and if the drawing is to generated immediately or saved to the drawing list.

Related Links

- *P. To create a drawing group* (on page 2024)

P.To open a drawing in MicroStation

To open the current drawing in MicroStation, use the following procedure.

You must have either MicroStation or Bentley PowerDraft installed to use this feature.

Note: If you have MicroStation PowerDraft installed instead of MicroStation, you will have that product's icon instead of the MicroStation icon in the drawing toolbar.

1. Open a drawing in the Steel AutoDrafter workflow.
2. On the drawing toolbar, select the **Open in MicroStation** tool.

You are prompted to save the drawing file (`.dxf` file extension).

The drawing opens in MicroStation.

Tip: The resulting `.dxf` can be opened within any compatible program.

P. To edit grid labels

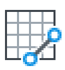
To toggle the display of grids and elevation marks or to change the label for each, use the following procedure.

Steel AutoDrafter identifies the grids from column (vertical elements) centers from the analysis file.

Note: The model in STAAD.Pro is required to be created using Y up axes system.

Tip: Grid can be imported from another `adfx` file using the import grid option provided. The `adfx` file will be available along with the relevant STAAD.Pro file in the same folder. Click  **Import** to select a `adfx` file.

1. On the **Steel AutoDrafter** ribbon tab, select the **Grids** tool in the **Edit** group.

   The **Grid Manager** dialog opens.
2. To toggle the display of a grid line or elevation mark, use the check box associated with that grid or elevation mark.
3. To edit the label of a grid or elevation mark, type the label in **Mark** cell associated with that grid or elevation mark.
4. Click  **OK**.

Related Links

- *Grid Manager dialog* (on page 3005)

P. Built-Up Sections in Steel AutoDrafter

Steel AutoDrafter recognises and draws built-up sections. These sections have to be defined in STAAD.Pro, using a particular format within the user table section.

Section	User Table Prop Name
Angle Star	AS-ISA75X75X8-10
I/H Star	IS-ISMB500
Built Up I	IBU-DPT-WD-WT-FT
Built Up Channel	CBU-DPT-WD-WT-FT
Two I/H Sxn	2I-ISMB250-100

where

DPT	=	total depth of the section
WD	=	total width of section

WT	=	web thickness
FT	=	flange thickness

“10” in the Angle Star represents the clear gap between the two angles.
“100” in the 2I represents the clear gap between the two sections.

Example of creating user table in STAAD as per above format:

```
START USER TABLE
TABLE 1
UNIT CM MTON
GENERAL
IEU-750-250-12-20
185.2 75 1.2 25 2 169049 5218.56 85.28 4507.98 417.48 0 0 0 0 0 0
IS-ISMB200
61.6 20 0.57 10 1 2257 14 225.7 451.4 0 0 0 0 0 0
AS-ISA75X75X6-8
17.32 15 0.58 15 0.58 192.58 148.6 1.97 33.62 28.02 12.99 12.99 51.47 -
44.04 19690.9 13.84
2I-ISMB200-200
64.66 20 1.62 30 1.07 6766 4470.8 21.85 451.067 447 48.49 48.49 656.48 -
517.95 218592 16.76
END
```

P. Drawing List & Groups panel

Drawings that are to be created can be added to the Drawing List using any of the following options. Groups of members in drawings can be managed here.

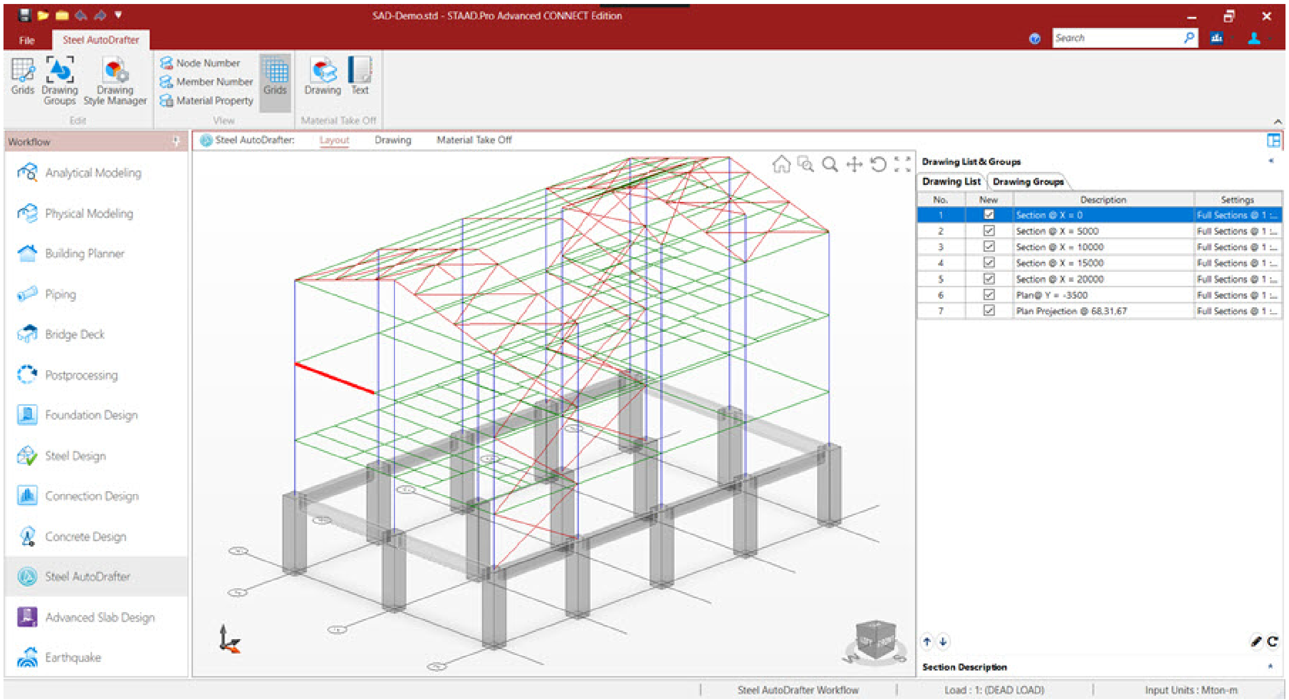
Drawing List tab

Section Description	This is used to add drawings to the list.

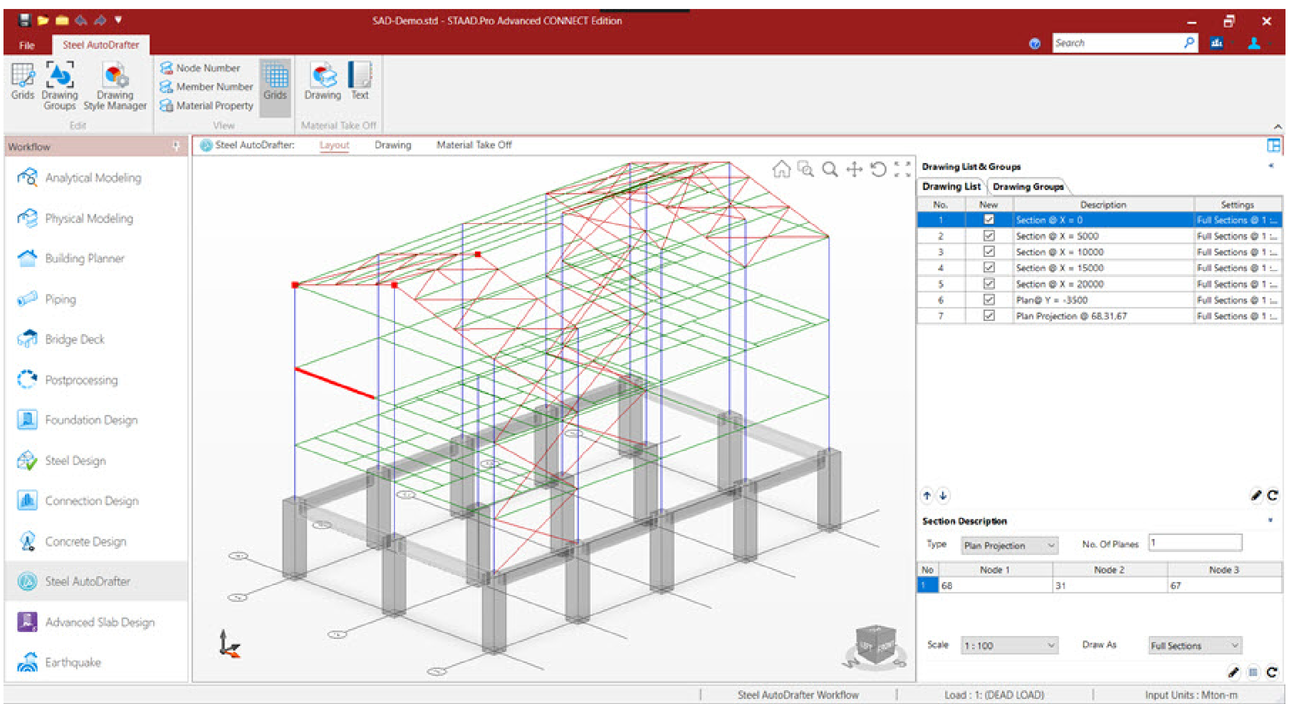
- By selecting a member in the view area and then selecting the required drawing from the options in the Drawings List tab.
- By using the options in the Section Description.
- A projected view can be created by selecting three nodes.

Drawings added to the list can all be generated at once using the draw options

New Option in the list can be used to generate drawings together in a single sheet. (Default each drawing is generated in a new sheet)



Selection of points for projected view



P. To generate a material take-off

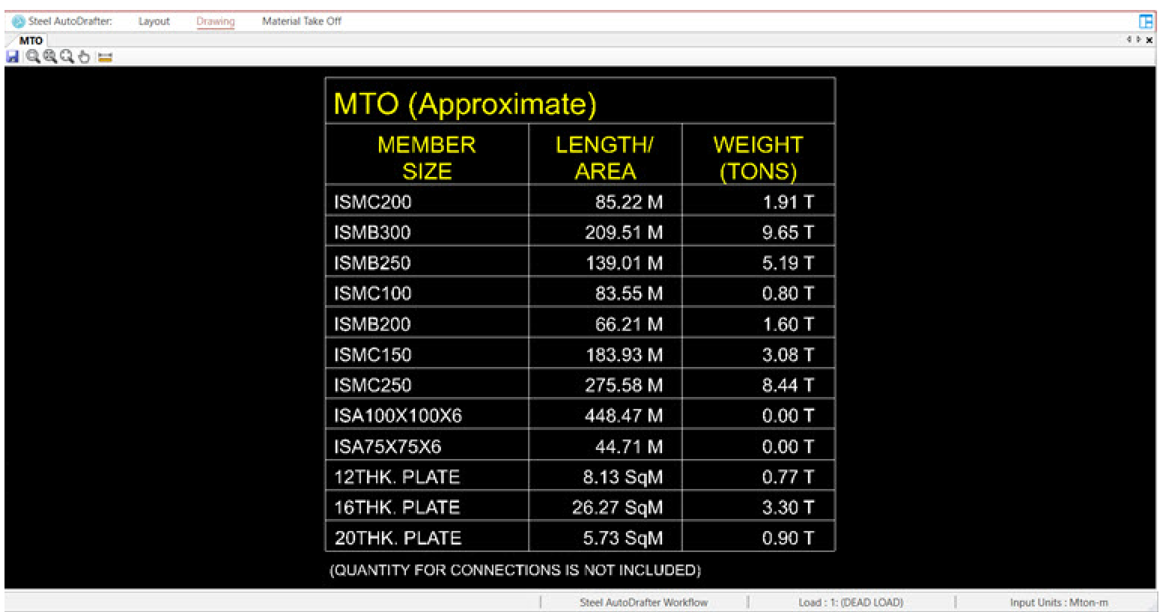
1. Steel AutoDrafter generates material take-offs for the entire structure in two formats:

Format	Select...
drawing	the **Drawing** tool in the **Material Take Off** group on the **Steel AutoDrafter** ribbon tab 
text	the **Text** tool in the **Material Take Off** group on the **Steel AutoDrafter** ribbon tab 

The material take-off opens in the selected format. The drawing format will open in the **Drawing** page. The text format will open in the **Material Take Off** page.

2. (Optional) Click the **Save** tool in the page toolbar to save the material take-off to an external file.

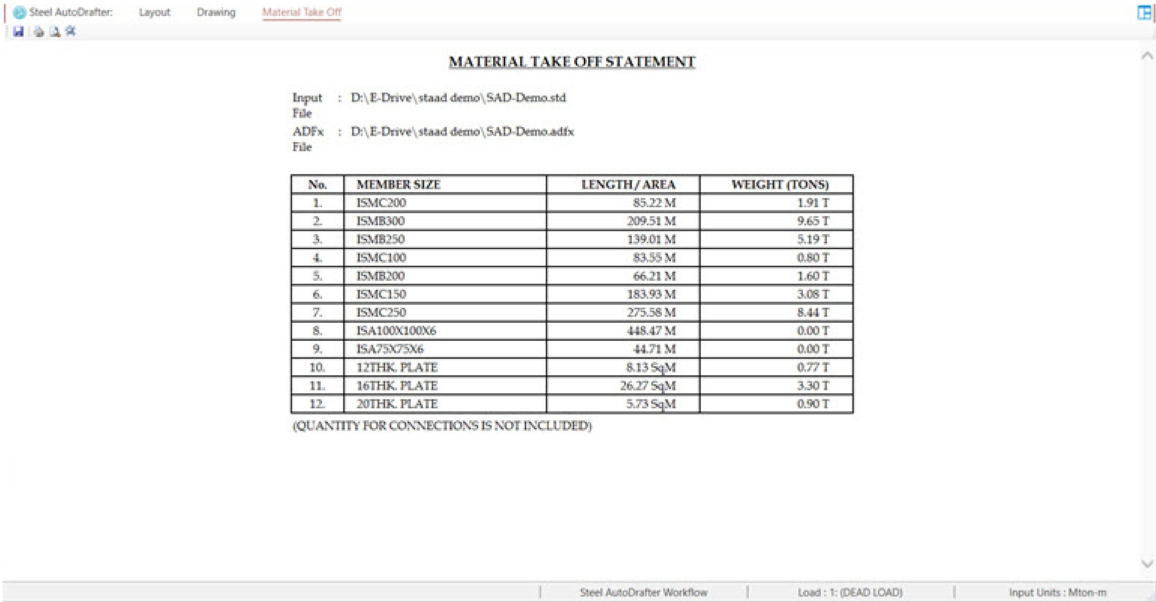
   The drawing is saved as a `.dxf` file and the text is saved as an `.html` file.



MTO (Approximate)		
MEMBER SIZE	LENGTH/ AREA	WEIGHT (TONS)
ISMC200	85.22 M	1.91 T
ISMB300	209.51 M	9.65 T
ISMB250	139.01 M	5.19 T
ISMC100	83.55 M	0.80 T
ISMB200	66.21 M	1.60 T
ISMC150	183.93 M	3.08 T
ISMC250	275.58 M	8.44 T
ISA100X100X6	448.47 M	0.00 T
ISA75X75X6	44.71 M	0.00 T
12THK. PLATE	8.13 SqM	0.77 T
16THK. PLATE	26.27 SqM	3.30 T
20THK. PLATE	5.73 SqM	0.90 T

(QUANTITY FOR CONNECTIONS IS NOT INCLUDED)

Figure 202: Drawing material take-off



MATERIAL TAKE OFF STATEMENT

Input File : D:\E-Drive\staad demo\SAD-Demo.std

ADFx File : D:\E-Drive\staad demo\SAD-Demo.adfx

No.	MEMBER SIZE	LENGTH / AREA	WEIGHT (TONS)
1.	ISMC200	85.22 M	1.91 T
2.	ISMB300	209.51 M	9.65 T
3.	ISMB250	139.01 M	5.19 T
4.	ISMC100	83.55 M	0.80 T
5.	ISMB200	66.21 M	1.60 T
6.	ISMC150	183.93 M	3.08 T
7.	ISMC250	275.58 M	8.44 T
8.	ISA100X100X6	448.47 M	0.00 T
9.	ISA75X75X6	44.71 M	0.00 T
10.	12THK. PLATE	8.13 SqM	0.77 T
11.	16THK. PLATE	26.27 SqM	3.30 T
12.	20THK. PLATE	5.73 SqM	0.90 T

(QUANTITY FOR CONNECTIONS IS NOT INCLUDED)

Figure 203: Text material take-off

P. Earthquake workflow

This STAAD.Pro workflow is used to check if the structure conforms to the basic geometric recommendations made in Eurocode 8 (EC8). This workflow is in addition to the normal post-processing workflow which gives the various analysis results. These checks are intended to give you a "feel" for the structure and are not mandatory to proceed to the design phase.

Eurocode 8: Part 1 [EN 1998-1-1:2004] contains specific requirements and recommendations for building structures that are to be constructed in seismic regions. Essentially, these fundamental requirements have been provided to ensure that the structures can sustain the seismic loads without collapse and also –where required– avoid suffering unacceptable damage and can continue to function after an exposure to a seismic event.

Tip: The program feature assumes Y as vertical and that seismic forces act in the global X and Z directions.

P. Using the Earthquake Workflow

To perform the geometry checks per Eurocode 8, STAAD.Pro must first calculate the center of mass for each defined floor. Specifying a response spectrum loading accomplishes this.

All the analysis methods for design and evaluation of the performance of a structure mentioned in Eurocode, as listed in the following, are all available in STAAD.Pro:

- Linear modal response spectrum analysis (i.e., a linear elastic analysis used in conjunction with an EC8 response spectrum load)
- Nonlinear static analysis (i.e., a pushover analysis)
- Nonlinear dynamic analysis (i.e., a time-history analysis)

Note: In order to make use of the EC8 response spectrum load, a linear modal response spectrum analysis should be used. This is required to locate the center of mass which is needed to perform the associated geometry checks.

Loads are distributed by element stiffness to all members during analysis. Thus, for the purposes of Eurocode 8, all members are assumed to be "primary seismic" elements.

Note: The EC8 response spectrum load must be specified in the model in order to perform any further checks, design, or detailing per Eurocode 8.

Related Links

- *M. To add an EC8 response spectrum* (on page 867)

P. To open the Earthquake workflow

You must perform an analysis on a model with a load cases including an EC8 response spectrum load.

1. Perform a successful analysis.
2. Select **Earthquake** in the **Workflows** panel.

The **EC8 Stiffness** page opens along with the **Floors** dialog, **Storey Stiffness** table, and **Soft Storeys** table.

Tip: You can nominally use the pages in the Earthquake workflow in any order. However, it is recommended that you work through the checks in a left-to-right order (i.e., EC8 Stiffness, EC8 Plans, and then EC8 Elevations).

P. To check stiffness of a structure per EC8

To review the stiffness of a structure per Eurocode 8 requirements, use the following procedure.

This set of checks is to enables you to judge whether the stiffness characteristics of the structure meet the recommendations set by EC 8. This implementation will:

i. Display the center of mass and center of stiffness graphically for each floor to inform you of some inherent eccentricities and their variation along the height of the building.
ii. Calculate the total stiffness of every story in each direction and provide you with a ratio of the stiffnesses in the two directions. This aides you to judge whether the structure conforms to the condition in EC8 that the stiffness should be similar in both directions.
iii. Check to see if there are any soft stories in the structure and will highlight them in the GUI. A soft story is defined as a story that has a stiffness in a particular direction that is less than 70% of the stiffness of the story above.

1. Select the **EC8 Stiffness** page in the **Earthquake** page control bar.
   The center of mass and center of stiffness are drawn at each floor level.
2. For a particular floor, you can expand the entry in the **Floors** dialog to review the location of these items.

   The selected floor and centers of mass or stiffness are highlighted in the view area.
3. The stiffness ratios are reported in the Storey Stiffness table for each floor as well as for the total structure.
4. Any soft stories detected by the program are listed in the Soft Storey table.

> **Note:** The stiffness of only the vertical elements are considered while working out the total stiffness of a story in a particular direction as the seismic loads are primarily resisted by the vertical elements in a structure.

P. To check for plan irregularities

To check for plan irregularities per Eurocode 8 such as re-entrant corners, slenderness, and torsional radius, use the following procedure.

This set of checks is to enable you to be able to classify the building as being 'regular in plan.' Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8 viz.

- **i.** Checks for re-entrant corners in floor slabs – The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied.
- **ii.** Check for slenderness of structure – The program calculates a “slenderness ratio” for the structure based on the maximum dimensions on plan and reports whether the EC-8 criteria are satisfied.
- **iii.** Checks for torsional radius: **-** The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8.

1. Select the **EC8 Plans** page in the **Earthquake** page control bar.
   The **Check Regularity in Plan** dialog opens along with the **Slab Re-Entrant Corners** and **Torsional Radius Check** tables.

   > **Tip:** You may have to re-arrange the dialog and tables to display all windows.
2. To check for re-entrant corners:
   a. In the **Check Regularity in Plan** dialog, expand **Clause 4.2.3.2(3) - [Re-Entrant Corners]**.
      A child entry is present for each floor in the structure.
   b. Select each floor to review any detected re-entrant corners.
      The details of any detected re-entrant corners are displayed in the **Slab Re-Entrant Corners** table.
3. To check for structure slenderness, expand **Clause 4.2.3.2(5) - [Slenderness]** in the **Check Regularity in Plan** dialog.
   The slenderness ratio and status for the structure is reported.
4. To check torsional radius, select the **Torsional Radius Check** table.
   The ratio of each torsional radius check for each floor along with the status of the check is reported here.

P. To check for elevation irregularities

To check for elevation irregularities as defined by EC8, use the following procedure.

This set of checks enables you to classify the building as a 'regular in elevation' structure. Checks will be performed and the results will be displayed graphically.

1. Select the **EC8 Elevations** page in the **Earthquake** page control bar.
   The **Earthquake Elevation Criteria** dialog opens along with the **Elevation Regularity** table.
2. In the **Earthquake Elevation Criteria** dialog, select to check elevation along **X- Axis** or **Z- Axis.**
3. Select the appropriate elevation criteria which applies to your structure.

   > **Note:** Refer to clause 4.2.3.3 and Figure 4.1 in Eurocode 8 (EN 1998-1:2004) for additional information.
4. Click **Perform Checks**.
   The calculated values and results of this check are displayed in the **Elevation Regularity** table.

5. Repeat steps 2 through 4 to check the other direction or for other criteria if needed.

P. Pages in the Earthquake Workflow

The Pages in the Earthquake workflow page control bar are described below in brief. Detailed description of these pages is available in the pages to follow.

Earthquake Page	Description
EC8 Stiffness	Used evaluate whether the stiffness characteristics of the structure meet the recommendations set by EC 8.
EC8 Plan	Used to classify the building as being “regular in plan.” Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8.
EC8 Elevations	Used to classify the building as a “regular in elevation” structure. Checks will be performed and the results will be displayed graphically.

P. EC8 Stiffness page

Used evaluate whether the stiffness characteristics of the structure meet the recommendations set by EC 8.

Note: The stiffness of only the vertical elements are considered while working out the total stiffness of a story in a particular direction as the seismic loads are primarily resisted by the vertical elements in a structure.

When the Eurocode 8 **|** Stiffness Checks page is selected, the **Floors** dialog, **Story Stiffness** table, and **Soft Story** table open. The active view window displays the Whole Structure with the Center of Mass and Center of Gravity plotted for each floor. This is used to inform you of some inherent eccentricities and their variation along the height of the building.

P. **Floors** dialog

Displays the Center of Mass (CM), Center of Stiffness (CS), and Eccentricity (difference between CM and CS) for each defined floor in the structure. This information is graphically plotted onto the structure diagram in the Whole Structure view window.

Opens when the **Eurocode 8 | Stiffness Checks** page is selected.

Tip: You can select on of the entries in the **Floors** dialog to have the corresponding item highlighted in the **Whole Structure** view window.

P. Story Stiffness table

The program calculates the total stiffness of every story in each direction and provides you with a ratio of the stiffnesses in the two directions. This aides you to judge whether the structure conforms to the condition in EC8 that the stiffness should be similar in both directions.

Opens when the **Eurocode 8 | Stiffness Checks** page is selected.

P. Soft Story table

The program will check if there are any soft stories in the structure and will display any detected soft story and the direction in this table. These stories will also be highlighted in the active View window. A soft story is defined as a story that has a stiffness in a particular direction that is less than 70% of the stiffness of the story above.

Opens when the **Eurocode 8 | Stiffness Checks** page is selected.

P. EC8 Plan page

Used to classify the building as being "regular in plan." Checks will be performed and the results will be displayed graphically in the GUI. The program primarily checks for the main three plan regularity conditions set out in EC-8.

When the EC8 Plan page is selected, the **Check Regularity in Plan** dialog opens and the active view window displays the Whole Structure.

Selecting various structure elements listed in the **Check Regularity in Plan** dialog will open either the **Torsional Radius Check** table or the **Slab Re-Entrant Corners** table.

P. Check Regularity in Plan dialog

Displays the results of the following checks for the main three plan regularity conditions set out in EC-8:

i. Checks for re-entrant corners in floor slabs

The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied.

ii. Check for slenderness of structure

The program calculates a "slenderness ratio" for the structure based on the maximum dimensions on plan and reports whether the EC-8 criteria are satisfied.

iii. Checks for torsional radius

The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8. These are summarized in the **Check Regularity in Plan** dialog and the **Torsional Radius Check** table will display details.

Opens when the **Eurocode 8 | Plan Regularity** page is selected.

P. Torsional Radius Check table

The program calculates the torsional radius for each floor and checks against the conditions set out in EC-8.

Opens when the **Eurocode 8 | Plan Regularity** page is selected.

P. Slab Re-Entrant Corners table

The program performs checks on each floor slab and reports whether the condition set out in EC-8 for re-entrant corners has been satisfied. The details of any corners detected which fail this check are displayed here. The geometry which fails is highlighted in the active view window.

Opens when the **Eurocode 8 | Plan Regularity** page is selected.

P. Eurocode 8 | Elevation Regularity page

Used to classify the building as a “regular in elevation” structure. Checks will be performed and the results will be displayed graphically.

When the Eurocode 8 **|** Elevation Regularity page is selected, the **Earthquake Elevation Criteria** dialog and **Elevation Regularity** table open.

The structure diagram is displayed in the Whole Structure view window. When a Elevation Regularity check is performed, the elevation profile is displayed on the structure diagram.

P. Earthquake Elevation Criteria dialog

Used to specify global seismic direction and the elevation criteria per Eurocode 8 against which a Elevation check will be performed.

Opens when the **Eurocode 8 | Elevation Regularity** page is selected.

Control	Description
Choose elevation along	Select the global axis along which the seismic forces are acting for this check.
Choose elevation criteria	Select the criteria which best describes the elevation variation per Clause 4.2.3.3 and Figure 4.1 of EN 1998-1:2004 (EC8).
Perform Checks	Click this button once the appropriate criteria is selected for a direction. The program will evaluate the elevation geometry against code requirements. The results of this evaluation will populate the Elevation Regularity table and the profile checked will be displayed onto the structure diagram in the active view window.

P. Elevation Regularity table

Reports the status of a Elevation Regularity check performed based on criteria set in the **Earthquake Elevation Criteria** dialog.

Opens when the **Eurocode 8 | Elevation Regularity** page is selected.

P. Plotting from STAAD.Pro

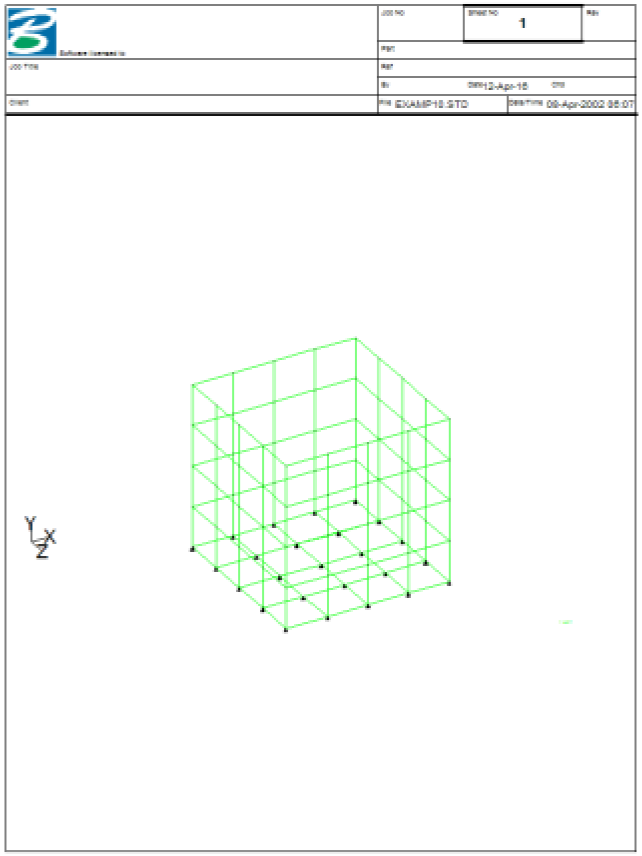
Explained below are five methods for plotting the drawing of the STAAD model and STAAD result diagrams.

P. Plot Using the Print Current View Tool

1. (Optional) Right-click in the view window and then select the **Print Preview Current View** from the pop-up menu.

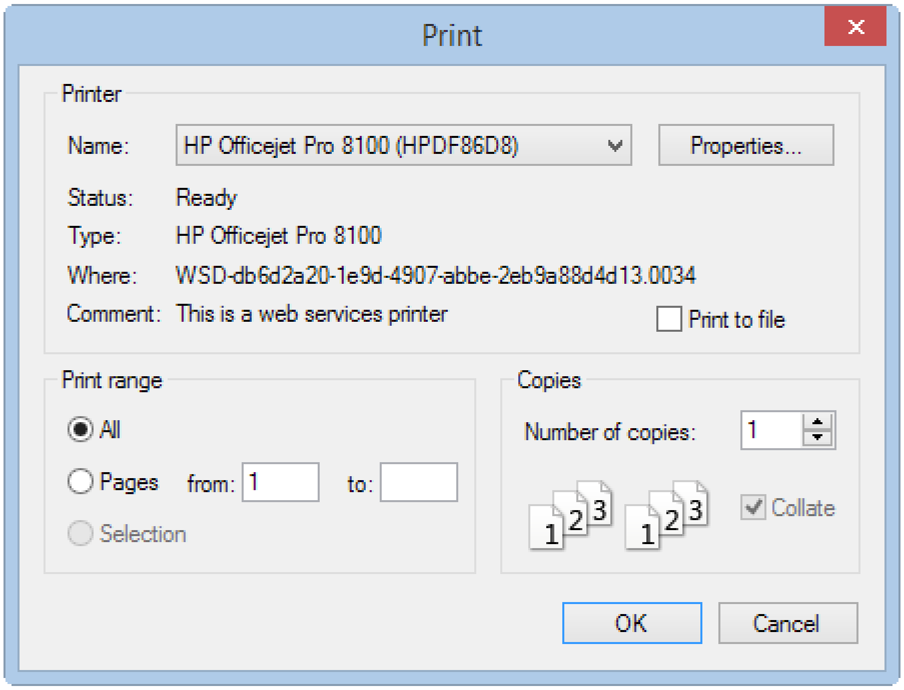
   This is used to preview the image as it will appear on paper.

   The **Print Preview** window opens to display the view in a report format.



2. Select **Print** in the Print Preview toolbar.

   The standard Windows **Print** dialog opens, which is used to select the printer or plotter where you wish to send the image.



3. Select the desired printer and click **OK.**

P. Plot Using the Take Picture Tool

1. On the **Utilities** ribbon tab, select the **Take Picture** tool in the **Utilities** group.

The **Picture #** dialog opens.

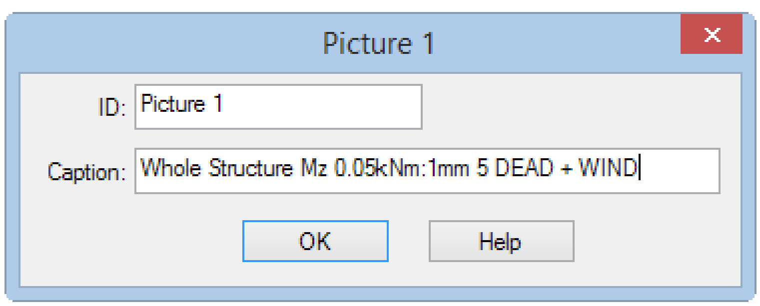


Figure 204: The Picture # dialog with the ID and Capture automatically generated

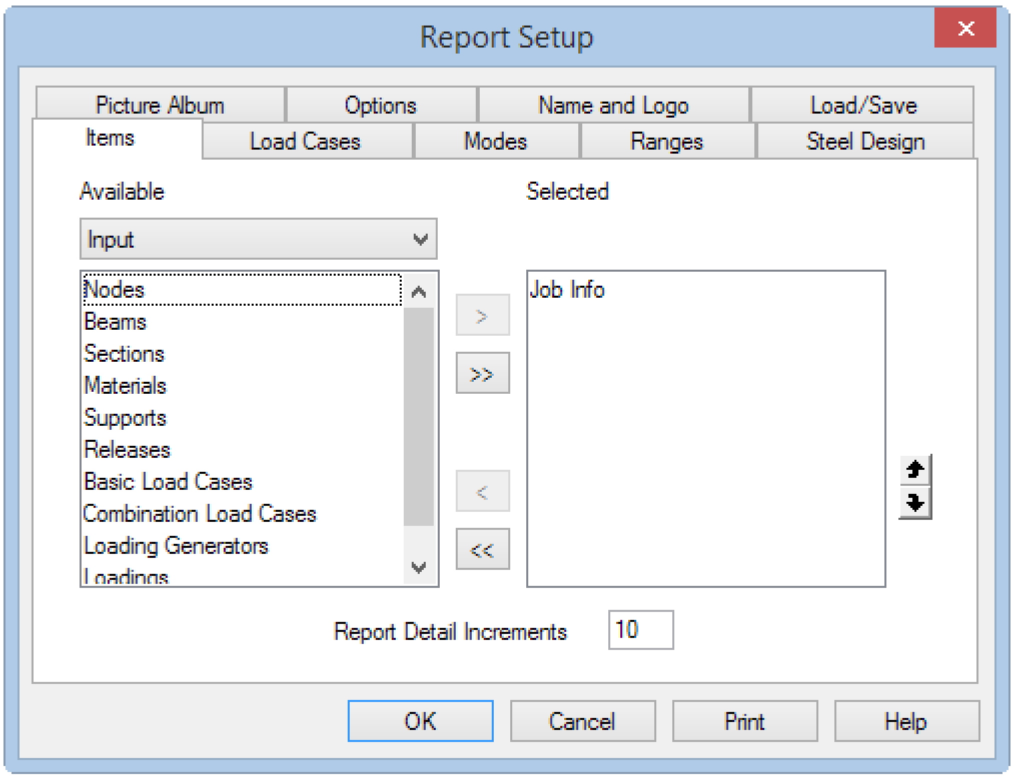
2. (Optional) Type a Picture **ID**

   The ID is incremented automatically.
3. (Optional) Type a **Caption**

Note: For most View window contents, the caption will be automatically completed with a description of the contents.

4. Click **OK**.
   This picture is saved for use in reports.
5. Open the Report Setup:
   a. Select the **File** ribbon tab.
      The Backstage view opens.
   b. Select the **Report** tab and then **Setup**.

   The **Report Setup** dialog opens.



Note: The Available items list is filtered by the drop-down selection list above it. This allows you to sort your reports.

6. On the **Items** tab, select **Pictures** from the available list of items.
7. Add one or more pictures to the Selected list by clicking [>].
8. (Optional) Select the **Picture Album**

Tip: You can manage pictures taken using the **Take Picture** tool here.

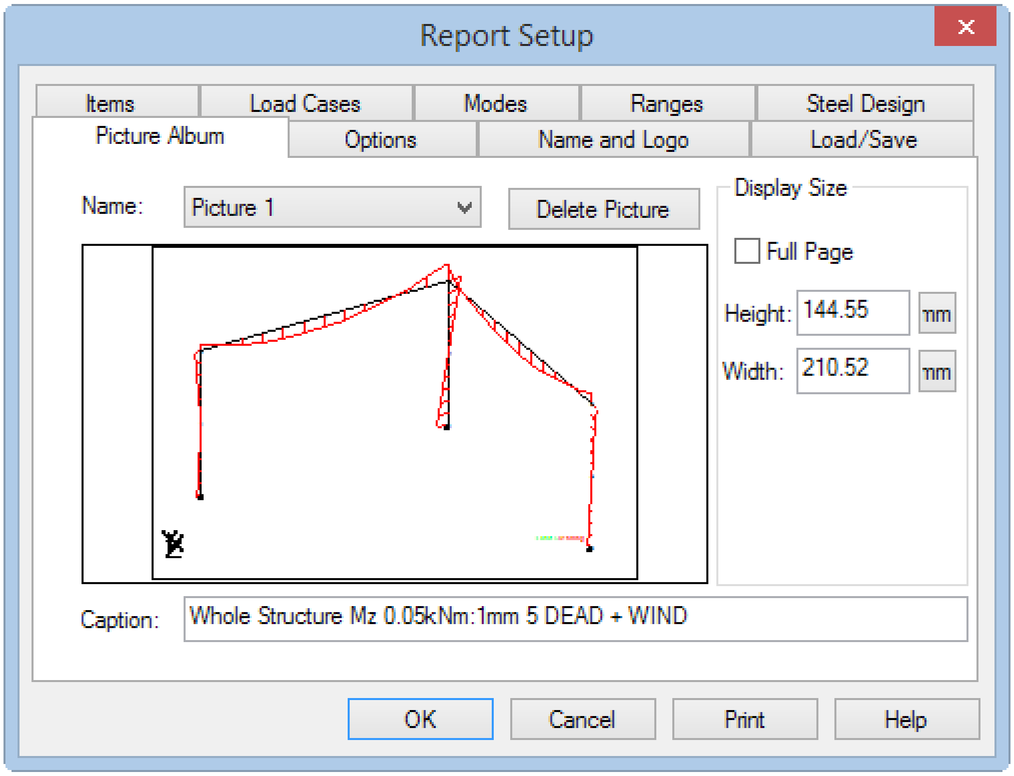


Figure 205:

9. (Optional) Select the **Full Page**
10. Click **OK**.
11. Select **File** > **Print Report**.
    The diagram will be plotted.

Alternatively, Select **File** > **Export Report** > **MS Word File** to create a report you can print from Microsoft Word.

In the **Save As** dialog, specify a name for the file and click **Save**. In the template dialog, select the **Normal** template (or use any custom Word templates you have created).

Figure 206:

Microsoft Office Word builds the file with the picture in it. Once this task is completed, the new file is opened in Microsoft Office Word.

P. Plot Using the Export View Option

STAAD.Pro has a facility to export the drawing to a graphic image file.

Use the program to generate the diagram or plot you want to export.

1. Select the **Export View** tool on the **Print** toolbar. 
   The **Save As** dialog opens.
2. Select the graphic format you want to use for your image file (i.e., Bitmap, JPEG, TIFF, GIF, etc.)
3. Type a file name and save the file.

P. Plot Using the Copy Picture Option

1. Highlight the window containing the diagram that you wish to plot.

   This can be done by making sure the title bar of that window has the color which indicates that it is in focus.
2. Select **Edit** > **Copy Picture**.
3. Open a graphics program capable of cropping the graphics image to your needs (e.g., Microsoft Paint, Corel PaintShop Pro, Adobe Photoshop, etc.).
4. Paste the clipboard contents into an empty file in this program.

   The drawing from the STAAD window is pasted in that program.

Alternatively, you can also copy graphic contents of the View window by using the **<PrtSc>** (Print Screen key).

8 Data Files and Interoperability

This section describes how to manually edit STAAD.Pro input files, import and export data, and interact with STAAD.Pro using external applications.

Note: You can also interact with STAAD.Pro via an API. Refer to OpenSTAAD (on page 7000) for more information.

I. STAAD.Pro Editor

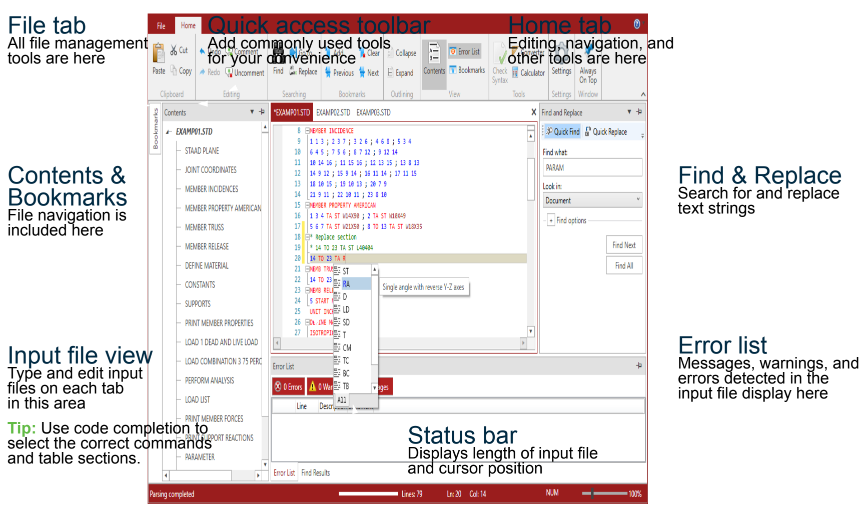
This section describes the various facilities available in the STAAD.Pro Editor. The tools and menus in the editor are described here.

Related Links

- *P. To view analysis results* (on page 1978)

I. Getting Started

I. Quick Overview



Related Links

- *I. File tab* (on page 2055)
- *I. Home tab* (on page 2056)

I. To start the editor from STAAD.Pro

When the STAAD.Pro Editor is opened from within STAAD.Pro, the model that is currently open is saved, the STAAD input file automatically parsed and a check on the data is performed. If any issues detected are then reported in an error message that identifies the number of Errors/Warning detected. The details of the issues are then reported in the Error List panel. For more details for the messages. Refer to I. To check for syntax errors (on page 2048) for details.

> **Note:** Note that when the STAAD.Pro Editor is opened from within STAAD.Pro, it is not possible to open an additional STAAD input file or start a new file. This is only possible when starting the STAAD.Pro Editor externally. Refer to I. To start the editor externally (on page 2044).

1. Open a STAAD file in STAAD.Pro.
2. On the **Utilities** ribbon tab, select the **Command File Editor** tool in the **Utilities** group.

   The STAAD.Pro Editor window opens with the current STAAD input file.

I. To start the editor externally

STAAD.Pro Editor can be run seperately from STAAD.Pro.

> **Note:** STAAD.Pro Editor is *not* capable of checking for errors when run externally from STAAD.Pro. Additionally, STAAD.Pro Editor cannot perform an analysis on an input file. This must always be done using STAAD.Pro.

1. In Windows Explorer, navigate to the Editor folder in the STAAD.Pro installation folder.
   This is typically
   `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Editor.`
2. Either:

   double-click the `Bentley.Staad.Editor.exe` `Bentley.TCL.Editor.exe` file

   or

   drag-and-drop a STAAD input file (file extension `.std` onto the STAAD.Pro Editor icon.

> **Tip:** You can create a Windows shortcut to the STAAD.Pro Editor program file and add it to your Start menu, desktop, or Windows task bar.

> **Tip:** You can right-click on an `.std` and select **Open with** to add the STAAD.Pro Editor to the program list associated with a STAAD input file.

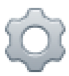
To select the character encoding

To manually select the character encoding used in the editor, use the following procedure.

You may typically only need to change the encoding to view some analysis output files. The auto-detect setting will work for STAAD input files.

For example, to correctly view the design results for steel design per the Russian SP 16.13330.2017 code, you must select "Cyrillic (Windows) (1251: windows-1251)" as the method of encoding.

1. On the **Home** ribbon tab in the STAAD.Pro Editor, select the **Settings** tool in the **Settings** group.

   The **Settings** dialog opens.
2. On the **General** tab, select the appropriate encoding system from the **Default Encoding** drop-down list.
3. Click **OK**.
   You are prompted to reload the file if one is open.
4. Click **Yes**.

I. To Pin a Panel

The **Contents**, **Bookmarks**, **Find and Replace**, and **Error List** panels call be "pinned" to stay open. Alternatively, to free up screen space, they can be "unpinned" to display only when selected.

1. Either:

   click the tab name of the panel you want to display

   or

   click the corresponding took name in the Searching or View groups on the **Home** ribbon tab
   The panel opens.
2. Click the pin icon in the top, right-hand corner of hte panel.
   The panel is now pinned in place.

Tip: Clickin the pin icon again will hide the panel once you click elsewhere.

Related Links
- *I. Bookmarks* (on page 2049)

I. STAAD Input Files

These are text files which list structure geometry, material properties, loading data, analysis commands, design commands, and other related structure input data which the STAAD.Pro analysis and design engine interprets upon running an analysis.

These files are the primary files created in the STAAD.Pro graphical interface and can be edited directly using the STAAD.Pro Editor.

Tip: STAAD input files use the `.std` file extension.

STAAD Command Language and Syntax

The advantage of using the STAAD.Pro Editor over other plain text editors is that the syntax of the STAAD command language is natively understood and will be appropriately marked (this functionality may be customized in the STAAD.Pro Editor **Settings** dialog). The following represents an overview of the language structure:

- English based - Input commands use full English words (though these may be shortened in many instances) or common engineering nomenclature.
- Case insensitive - Input data is not case sensitive.
- Line continuation - Lines containing lists may be continued to the next line by ending the line with a blank and a hyphen. Some lists have special rules or considerations.
- Comments - Lines may be marked as a comment (information which is ignored by the program) by including an asterisk (*) as the first non-blank character in any line. By default, comment lines will appear in green in the STAAD.Pro Editor window.

Refer to TR.1 Command Language Conventions (on page 2197) for additional information on the STAAD Command Language.

I. To create a new STAAD input file

1. Either:

   Select the **New** tool on the **File** ribbon tab.

   or

   Press **<Ctrl+N>**.

   A new STAAD input file opens with the empty template.

You can now start typing STAAD commands. To check syntax or to run the STAAD input file, you must use STAAD.Pro.

I. To open an existing STAAD input file

Tip: Multiple input files can be open simultaneously. Each file is displayed on a separate tab across the top of the text editor.

1. Either:

   Select the **Open** tool on the **File** ribbon tab.

   or

   Press **<Ctrl+O>**.

   The **Select STAAD File** dialog opens. The controls are similar to a typical Windows File open dialog.
2. Navigate to and select the STAAD input file you want to edit.
3. Click **Open**.

I. To save chagnes to a STAAD input file

1. Either:

   select the **Save** tool on the **File** ribbon tab

or

press **<Ctrl+S>**.

> **Tip:** Select the **Save As** tool on the **File** ribbon tab to save the file with a different file name or in a different location.

If this is a newly create STAAD input file, you will be prompted to type a file name and select a location to save the file.

2. (Optional) Click **Save** if you are saving for the first time or saving as a new file.

I. Getting Help

When you are typing, press **<F1>** to open the Technical Reference help to the topic pertaining to the current input command.

To open the general STAAD.Pro help, click the **Help** tool () on the right side of the ribbon bar.

I. Editing Input Files

I. Typing Commands

Code Completion

As you type commands, the STAAD.Pro Editor will offer suggestions to complete the command you are typing in the form of a drop-down list. This list is context-sensitive based on the preceding commands.

As you continue typing, the list will respond by narrowing to complete the word you are typing. You can use your mouse or the arrow keys to select a command from the list. Press **<Enter>** or **<Tab>** to accept the highlighted command keyword.

Table Sizes

Similarly, within a `MEMBER PROPERTY` *`country`* block, the corresponding country section tables are available for selecting sections. After typing `TABLE` and any additional specification options, the table sections will open as a drop-down list. You can continue typing to narrow the list or use your mouse to select one of the shape tabs for that table.

Context-Sensitive Command Help

Pressing **<F1>** opens the Technical Reference help to the topic corresponding to the current input command.

I. To add a comment

To change one or more lines in the input file to a comment, do the following.

Comments can be used to add notes or remarks in the input. They also can be used to turn 'off' commands so they are ignored by the analysis and design engine.

1. Either:

   place your cursor anywhere on the line you want to comment

or

select text across mulitple lines

2. Either:

   select the **Comment** tool in the **Editing** group on the **Home** ribbon tab

   or

   right-click and select **Comment Selection**

If...	Then...
No content was selected in Step 1	the current line is made a comment
Multiple lines were selected in Step 1	all the lines in the selection are made comments

Tip: You can remove comments using the **Uncomment** tool n the **Editing** group on the **Home** ribbon tab or by selecting **Uncomment Selection** in the right-click menu.

I. To check for syntax errors

Note: In order to check for errors, STAAD.Pro must be running. If you have launched the STAAD.Pro Editor externally, this feature is disabled.

Note: Input file data is also parsed when STAAD.Pro is first opened as well as when the STAAD.Pro Editor is first opened. You will be notified of any errors at that time.

1. Select the **Check Syntax** tool in the **Tools** group on the **Home** ribbon tab.
   A message opens to indicate if no errors are detected.

When the file is checked, if any issues are detected, the details are reported in the **Error list** panel. The data there is displayed in four colums:

i. An icon which indicates the type of issue detected: error, warning, or message.
ii. The line number on which the detected issue occurs. Double-click on the issue will point the input view to this line number.

   Note: If a line number cannot be identified by the program, then this will be blank.

iii. A description that provides details of why the issue has been identified and possible assistance in determining a resolution.
iv. A copy of the data in the line what has been identified as an issue.

You can review individual errors or warnings in the **Error List** panel.

I. Navigation

Multiple Files Open

You can open multiple STAAD input file simultaneously. Each file is displayed on a separate tab across the top of the text editor panel by the filename. The currently selected tab is active and all operations are performed on this file only.

Blocks

Blocks can be nested. For example, `DEFINE MATERIAL START` is a block, and each `ISOTROPIC` material within that block is a sub-block. The current block is indicated by a vertical bar to the right of the commands.

Clicking the minus sign will collapse the block down to the fist command. Once a block is collapsed, a plus sign is displayed and the first command is displayed in a box.

Tip: Select the **Collapse** tool in the **Outlining** group on the **Home** ribbon tab to collapse all blocks in the current input file.

Contents Panel

The top-level blocks are listed, in order, in the Contents pane.

Double-clicking an entry in the contents moves the cursor to that point in the text editor.

I. To go to a line

To move the cursor to a particular line in the input file, do the following:

1. Select the **Go to** tool in the **Searching** group on the **Home** ribbon tab.
   The **Go To Line** dialog opens.
2. Type the **Line Number**.
3. Click **OK**.
   The cursor is placed at the beginning of the indicated line.

   Note: A message indicates if the specified line number is higher than the number of lines in the file.

I. Bookmarks

You can add bookmarks at any point in an input file to quickly return to that point.

To add a bookmark

1. Either:
   - select the **Add** tool in the **Bookmarks** group on the **Home** ribbon tab, or
   - right-click and select **Toggle Bookmark** from the pop-up menu.

To navigate to the next or previous bookmark in the file, select the **Next** or **Previous** tool in the **Bookmarks** group on the **Home** ribbon tab.

To jump to a specify bookmark, double-click the name in the **Bookmark** panel.

To remove *all* the bookmarks in the file, select the **Clear** tool in the **Bookmarks** group on the **Home** ribbon tab.

Related Links

- *I. To Pin a Panel* (on page 2045)

I. Find and Replace

I. To find something

1. Either:

   select the **Find** tool in the **Searching** group on the **Home** ribbon tab

   or

   press **<Ctrl+F>**

   The **Quick Find** tab on the **Find and Repalce** pane opens.
2. Type a string to match in the **Find what** field.
3. (Optional) Select the **Look in** and **Find options** to specify the extents of the search and how the string should be matched, respectively.
4. Click **Find Next**.
   The next matching string is selected in the input file.

Related Links

- *I. Search Methods* (on page 2051)

I. To replace something

1. Either:

   select the **Replace** tool in the **Searching** group on the **Home** ribbon tab

   or

   press **<Ctrl+H>**

   The **Quick Find** tab on the **Find and Repalce** pane opens.
2. Type a string to match in the **Find what** field.
3. (Optional) Select the **Look in** and **Find options** to specify the extents of the search and how the string should be matched, respectively.
4. Type a replacement string in the **Replace with** field.
5. Either:

   click **Find Next** to highlight the next matching string

   or

   click **Replace** to replace the currently highlighted match

   or

   click **Replace All** to automatically replace all the matching strings in the file

Related Links

- *I. Search Methods* (on page 2051)

I. To find all occurances of a string

1. Press **<Ctrl+I>**.
2. Start typing a string.
   As you type, all occurances of the input string are highlighted in the file. The cursor jumps to the next occorance of the string.

I. Search Methods

STAAD.Pro Editor can use a variety of methods for matching strings when searching an input file.

In either the **Quick Find** or **Quick Replace** tabs on the **Find and Replace** panel, expand the **Find options** section to display the avaialable matching options.

Look in	Select either the **document** (entire file) or **selection** to limit the search extents.
Match case	Select use case-sensitive matching in the search string.
Match whole word	Select to match entire words (separated by white space) only. (e.g., searching for `JOI` will find "JOI" but not "JOINTS").
Search type	

Normal	Match the input string literally, using the options selected above.
Regular Expressions	Use regular expressions (also referred to as regex or regexp) in the search string. For additional
Wildcard	Use standard Windows wildcards to substitute a single character for one or more other characters in the search string. "?" represents a single character, and "*" represents any combination of characters
Acronym	
Shorthand	

Related Links

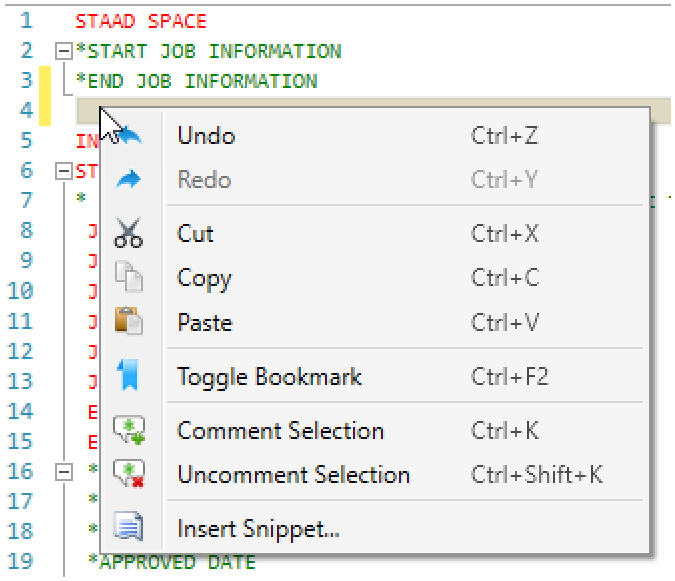
- *I. To find something* (on page 2050)
- *I. To replace something* (on page 2050)

I. Snippets

Code snippets are short sections which you commonly need to insert into different input files. Saving this code as a separate snippet allows you to easily access these portions of predefined code.

I. To insert a code snippet

1. Place your cursor at the point you want to insert a code snippet.
2. Right-click and select **Insert Snippet** from the pop-up menu.

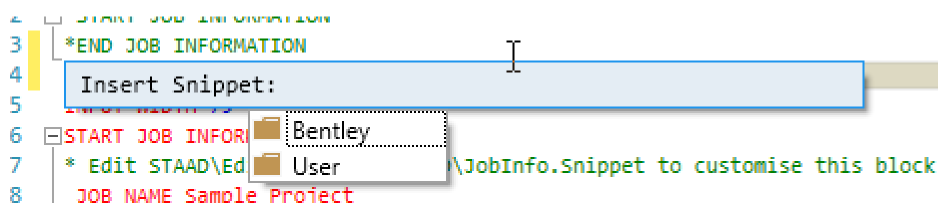


The inline snippet tool opens.

3. Either:

   Use the arrow keys or mouse to select the name of the snippet you want to insert.

   Tip: If you have organized snippet files into multiple folders, select the folder first.



   or

   Start typing to select the snippet by the Shortcut string in the snippet file.

4. Double-click with the mouse, press **<Enter>**, or press **<Tab>** to select and place the snippet.
   The text is inserted.

I. To create a code snippet

To create a custom code snippet using a sample as a template, do the following:

Some familiarity with editing XML files (e.g., HTML) is helpful to create snippet files, but not necessary. Further, no special tools or software are required, but a plain text editor which can validate XML is recommended.

1. In Windows Explorer, navigate to the .../`STAAD/Editor/Snippets/Std/` folder in your STAAD.Pro installation folder.
2. Make a copy of one of the existing snippet files (file extension `.snippet`).

   You can place snippets in the same folder or create a new folder within .../`STAAD/Editor/Snippets/` to orgnanize snippet files as needed.
3. Rename this copy to something meaningful.
4. Open the reanmed file in a plain text editor.

5. Change the header information:
   a. Type a new title within the <Title> element.
   b. (Optional) Type a your name or your organization's name within the <Author> element.
   c. Type a tool tip description within the <Description> element.
   d. Type a shortcut string within the <Shortcut> element.

   Note: Do *not* edit any of the XML tags or attibutes. Only change the plain text inside elements.
6. (Optional) If you want to add replacement strings in the snippet, create a <Literal> element inside the <Declarations> section for each.
7. Type or paste the actual snippet STAAD input code into the CDATA section, between the inner-most square brackets.

   Example:
   ```
   <![CDATA[snippet contents here]]>
   ```
   The contents here will be copied directly when the snippet is used (except for variables), including line breaks and commented lines.

   Tip: There is no syntax checking in the snippet file itself, so it is best practice to copy and paste the STAAD input snippet code from a STAAD input file you have checked for errors.
8. Save and close the snippet file.

I. Creating a New Code Snippet

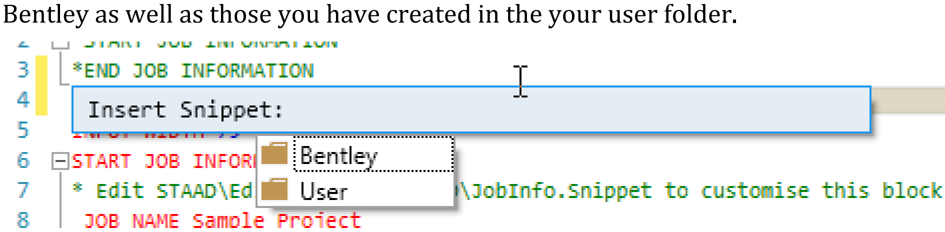
Code snippet files (file extension .snippet) are XML files which contain one or more code snippets to insert along with metadata about the snippet, such as tool tips, titles, etc. They can be created or edited in any plain-text editor.

Tip: It is easiest to copy an existing snippet file, rename it, and edit as needed.

You can save it in a separate folder to organize your snippets. This will add a folder section to your Insert Snippet menu. Otherwise, you can simply save them in the /STAAD/Editor/Snippets/Std/ folder.

The application comes with a few sample snippet files in the installation. However, to add your own Snippets, add the .snippet file to the folder \Users*(user name)*\AppData\Local\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default \Editor\Snippets.

When you select Insert Snippet from the right-click pop-up menu, you will then be provided folders for the samples from Bentley as well as those you have created in the your user folder.



Code Snippet Syntax

When editing a snippet file, you should note that there are two primary components to the XML: the `<Header>` and the `<Snippet>`. The Header contains the metadata about the code snippet, including the `<Title>` text and `<Description>`, which is used as a tool tip in the STAAD.Pro Editor.

The actual code inserted when the snippet is used by the STAAD.Pro Editor is contained in a CDATA section within the `<Code>` section.

```
<Code Language="STD">
  <![CDATA[UNIT FEET KIP]]>
</Code>
```

Tip: If you want line returns before or after the inserted code, you can place those at the beginning or end of the CDATA text. Whitespace (i.e., spaces, tabs, or line returns) *outside* of the CDATA section have no effect on the inserted snippet.

```
<Code Language="STD">
  <![CDATA[
  UNIT FEET KIP
  ]]>
</Code>
```

Snippet Replacements

You can create replacement strings in a snippet which will be highlighted for the snippet user. These are added in the `<Declarations>` section within the `<Snippet>` section.

Each replacement definition has the following structure:

```
<Literal>
   <ID>name</ID>
   <ToolTip>Tool tip text.</ToolTip>
   <Default>Default text</Default>
</Literal>
```

Where:

`name` is the replacement name to use within the snippet code.

`Tool tip text.` is the string to show as a tool tip in the STAAD.Pro Editor

`Default text` is the text that will be inserted (and highlighted) in the STAAD.Pro Editor when the snippet is used

To use the replacement within the snippet code, just place dollar signs ("$") before and after the *`name`* value.

Snippet Editing Tools

You can use an IDE such as Microsoft Visual Studio, but any plain text editor is capable of editing code snippet files. An editor that is capable of checking XML syntax is recommended, though.

Tip: Notepad++ is a free text editor which, when the XML Tools plugin is installed, is capable of checking XML syntax.

Related Links

- *http://www.herongyang.com/XML/NPP-XML-Tools-Plugin-Download-and-Install.html*
- *https://msdn.microsoft.com/en-us/library/ms165394.aspx*

I. Ribbons

I. File tab

Table 213: Controls on the File ribbon tab

Tool Name	What it Does	Shortcut
 **New**	Creates a new STAAD input file.	**<Ctrl+N>**
 **Open**	Opens an existing STAAD input file.	**<Ctrl+O>**
 **Close**	Closes the current STAAD input file (current tab only).	
 **Save**	Saves any changes made to the current STAAD input file since the last save. These changes are highlighted with yellow bars in the text view. After saving, these changes are marked green.	**<Ctrl+S>**
 **Save As**	Opens a **Save As** dialog, which is used to save the current STAAD input file with a different name or in a different location.	
 **Print Preview**	Opens a **Print Preview** window, which is used to review the print of the file.	

Tool Name	What it Does	Shortcut
 **Print**	Opens a **Print** dialog, which is used to select a printer device and print out a copy of the file.	**<Ctrl+P>**
 **Send in email**	Creates a draft e-mail in your default e-mail client with the current STAAD input file attached. The file path to the input file is inserted in the body of the message.	
 **About**	Opens the **About** dialog, which contains version and copyright information for the program.	
 **Exit Editor**	Closes the STAAD.Pro Editor program.	**<Alt+F4>**

Related Links

- *I. Quick Overview* (on page 2043)

I. Home tab

Tip: Double-click the **Home** ribbon tab or select the **^** button to collapse the ribbon. Clicking the tab again displays the toolbar.

Table 214: Clipboard group

Tool Name	Description	Shortcut
 **Paste**	Pastes the clipboard contents (text only) at the cursor position.	**<Ctrl+V>**

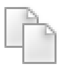
Tool Name	Description	Shortcut
 **Cut**	Copies the selected contents to the clipboard and deletes the original.	**<Ctrl+X>**
 **Copy**	Copies the selected contents to the clipboard.	**<Ctrl+C>**

Table 215: Editing group

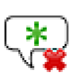
Tool Name	Description	Shortcut
 **Undo**	Undoes the last action in the editor.	**<Ctrl+Z>**
 **Redo**	Reverses the last undo action.	**<Ctrl+Y>**
 **Comment**	Changes the current line (or lines, if a selection spans multiple lines) to a comment (i.e., adds an asterisk (*) at the beginning of the line so it is ignored by STAAD.Pro).	
 **Uncomment**	Changes a commented line or lines back to commands to be interpreted by STAAD.Pro.	

Table 216: Searching group

Tool Name	Description	Shortcut
 **Find**	Opens the **Find and Replace** panel to the **Quick Find** tab, which is used to find strings that match the input.	**<Ctrl+F>**
 **Go to**	Opens the **Go To Line** dialog, which is used to move the cursor to a specified line number.	
 **Replace**	Opens the **Find and Replace** panel to the **Quick Replace** tab, which is used to replace any matched strings with a different string.	**<Ctrl+H>**

Table 217: Bookmarks group

Tool Name	Description	Shortcut
 **Add**	Adds a bookmark to the line where the cursor is located.	**<Ctrl+F2>**
 **Clear**	Clears *all* bookmarks from the current input file.	
 **Previous**	Jumps the cursor to the previous bookmark, if present, in the current input file.	**<Shift+F2>**

Tool Name	Description	Shortcut
 **Next**	Jumps the cursor to the next bookmark, if present, in the current input file.	**<F2>**

Table 218: Outlining group

Tool Name	Description
 **Collapse**	Collapses all top-level blocks in the text editor for the current input file.
 **Expand**	Expands all blocks in the text editor for the current input file.

Table 219: View group

Tool Name	Description
 **Contents**	Opens then **Contents** panel.
 **Error List**	Opens then **Error List** panel.
 **Bookmarks**	Opens then **Bookmarks** panel.

Table 220: Tools group

Tool Name	Description
 **Check Syntax**	Checks for errors in the current input file. Any errors, warnings, or messages are dipslayed in the **Error List** panel.
 **Converter**	Opens then **STAAD.Pro Unit Converter** program, which can be used to convert a vareity of common units.
 **Calculator**	Opens then **Calculator** program, which can be used for basic calculations.

Table 221: Misc controls on the Home ribbon tab

Tool Name	Description
 **Settings**	Used to change settings in the STAAD.Pro Editor program.
 **Always On Top**	Used to keep the STAAD.Pro Editor in the foreground, in front of all other Windows (regardless of focus). This feature is active when the icon is highlighted. Select again to deactivate.

Related Links

- *I. Quick Overview* (on page 2043)

I. Settings dialog

This dialog is used to control display settings for the STAAD.Pro Editor.

General tab

Display options	**Display Line Number**	Set this option to include line numbers to the left of the command lines in the editor. Line numbers are not printed.
	Highlight Current Line	Set this option to highlight the entire line where the cursor is located in the editor.
	Outline Group	Set this option to have sections of the input file “grouped” together in the editor.
	Color Coded Text	Set this option to highlight input using different colors by keyword, values, etc. in the input file. Printed input files will use the same color coding. **Note:** The output file (`.ANL`) does not use color coding for the input echo. Only warnings, errors, and messages will be displayed in color when the information is indicated for this.
Default Encoding	Select the default character encoding system used by the viewer.	
Editor options	These options are only available in the Editor mode.	
	Enable IntelliSense	Set this option to enable code-completion for input file editing.
	Enable Snippets	Set this option to enable the snippets functionality in the right-click pop-up menu in the editor. Snippets allow you quickly add predefined blocks of code to your input file.

Fonts and Colors tab

File Type	Select the file extension corresponding to the type of file you want to style.
Font	Select an installed Windows font from the drop-down list to use for the text editor.
Size	Select a font size from the drop-down list to use for the text editor.
Items	Select one of the text window display items from this list. The color can be selected from the **Item Color** drop-down pallet.
Item Color	Click the down arrow to select a color for the selected **Items**.

Print tab

Header	Set this option to include a header with the selected options:	
	Logo	(Input files only) Set this checkbox to include the file path in the **Company Logo File** field. Click **Browse** to locate an image file.
	Job Information	(Input files only) Set this option to include the `JOB NAME`, `JOB CLIENT`, and `ENGINEER NAME` job information values from the `START JOB INFORMATION` command block.

Date/Time (Output files only) Select the time stamp to use for the file.

Footer Set this option to include a footer with the selected options:

Path and Filename Set this option to include the file path, file name, and timestamp of the file in the footer of each page of the printout.

Page Number Set this option to include page numbers in the footer of each page of the printout.

Margins Set the page margins around the page here.

I. Keyboard Shortcuts

Exit the programs.	**<Alt+F4>**
Opens the help topic for the current input command.	**<F1>**
Select all the contents in the current input file.	**<Ctrl+A>**
Copy the current selection to the clipboard.	**<Ctrl+C>**
Open the **Quick Find** tool to search the document.	**<Ctrl+F>**
Open the **Quick Replace** tool to find and replace strings in the document.	**<Ctrl+H>**
Highlight all occurances of a typed string.	**<Ctrl+I>**
Delete the current line.	**<Ctrl+L>**
Start a new input file.	**<Ctrl+N>**
Open an existing STAAD input file.	**<Ctrl+O>**
Open the **Print** dialog.	**<Ctrl+P>**
Save changes made to the current input file. If the file has not been saved previously, you will be prompted to type a file name and select a location.	**<Ctrl+S>**
Reverse the order of the previous two characters before the cursor.	**<Ctrl+T>**
Paste the clipboard contents at the cursor location.	**<Ctrl+V>**
Cut the selection from the document to the clipboard.	**<Ctrl+X>**
Undo the last redo action.	**<Ctrl+Y>**
Undo the last action.	**<Ctrl+Z>**

I. Integrated Structural Modeling

STAAD.Pro can exchange structure information with Bentley Integrated Structural Model (ISM) repositories.

I. ISM Sync Tools Overview

STAAD.Pro can send structural data to and from an ISM repository through a set of ISM Syncing tools. These tools allow both creation and updating of STAAD.Pro models as well as ISM repositories. Further, these flexible tools allow you to begin models and move data as your workflow dictates.

If you need to ...	... use this tool	Description
Create a new ISM repository from an existing STAAD.Pro model	 **Create Repository**	transfers the current model opened in STAAD.Pro and generates a new ISM repository. This is the most common way in which an ISM repository is initially created.
Create a new STAAD.Pro model from an existing ISM repository	 **New From Repository**	creates a new STAAD.Pro model from an existing ISM repository. This is used to transfer model data from other tools used in the workflow.
Update an existing repository to reflect changes made in a STAAD.Pro model	 **Update Repository**	will coordinate changes made to the model in STAAD.Pro and coordinate some or all of those changes with an existing ISM repository.
Update an existing STAAD.Pro model to reflect changes in an ISM repository	 **Update From Repository**	used to update a STAAD.Pro model with some or all of the changes which have been made to the ISM repository.

Not all tools are available at all times. For example,

- If ISM is not installed these items will be grayed out and not available.
- If ISM is installed but no STAAD.Pro model is loaded, only **New From Repository** will be available.

iTwin Analytical Synchronizer

The program provided by ISM for accepting or rejecting model data changes is called " iTwin Analytical Synchronizer ". iTwin Analytical Synchronizer provides the user with a powerful set of tools for moving data between the applications used in a daily workflow. Even relatively small structural models have enormous amounts of data and ISM allows this data to be re-used with ease. Care must be taken that only desired data is transferred between applications.

When accepting changes made by client applications to an ISM Repository, some attention must be paid to what changes are actually being made. A small change in a client application can have unintended repercussions, if accepted. A repository is intended to represent the data that is *common* to all the client applications. Some client application models will use only a subset of the repository data but changes made to them can affect the entire repository if these changes are accepted when a repository update action is performed.

Note: This brief overview is not intended as a sufficient description on using the full capabilities of iTwin Analytical Synchronizer. Please refer to the iTwin Analytical Synchronizer product documentation for more detailed information.

Related Links

- *ISM tab* (on page 2702)

I. What is ISM?

Bentley's Integrated Structural Model (ISM) is a technology for sharing structural engineering project information among structural modeling, analysis, design, drafting, and detailing applications. ISM is similar to Building Information Modeling (BIM), but focuses on the information that is important in the design, construction, and modification of the load bearing components of buildings, bridges, and other structures.

I. Purpose of ISM

There are two related purposes for ISM:

1. The transfer of structural information between applications.
2. The coordination of structural information between applications.

To provide for the first purpose (transferring information), ISM provides a means of defining, storing, reading and querying ISM models.

To provide for the second purpose (coordination of information), ISM additionally provides capabilities to detect differences between ISM models and to selectively (based on user selection) update either an ISM repository or an application's data to provide a user-controlled level of consistency between the two data sets.

I. ISM and Application Data

ISM is not intended to store all of the information that all of its client applications contain. Rather, it is intended to store and communicate a *consensus* view of data that is *common* to two or more of its client applications, such as STAAD.Pro.

STAAD.Pro continues to hold and maintain its own private copy of project data. Some of the application data will duplicate that of the associated ISM repository. The application data may even conflict with that in the ISM repository. STAAD.Pro (or the user) may decide that a conflict gives the best data for the different uses in STAAD.Pro versus ISM.

I. Backups

STAAD.Pro has an auto-saving and backup managing feature to prevent accidental data loss.

I. To enable auto-recovery

To turn on the auto-recovery feature and set the frequency of auto-recovery backups, use the following procedure.

This feature will automatically create backups based on a specified time interval (default of 5 minutes) of your entire STAAD project, including physical models. In the event of a program crash or if the program is simply closed without saving changes, the program will detect the auto recovery files and prompt you to open from that recovery state when you re-open the associated STAAD model.

The auto-recovery process will only run if changes are detected in the files since your last save action.

1. From the Start page, select the **Configure** tool.
   The **Application Configuration** dialog opens.
2. Select the **Options** tab.
3. Check the option to **Save AutoRecover information every X minutes**.
4. (Optional) Type the how often (in minutes) you want the program to save recovery state for open files.

   The default of 5 minutes is recommended. Intervals shorter than this can degrade performance.
5. Click **OK**.

Tip: If you find that the auto-recovery process is causing performance issues when running (which may occur in large models), then you can toggle it off in the Quick Access toolbar. This will not stop a current auto-recovery process, but will prevent it from running subsequently until toggled back on again.

Related Links
- *GS. Quick Access Toolbar* (on page 62)

I. To create a restore point

To create a restore point used to backup your entire STAAD project, use the following procedure.

A restore point is a backup of all files associated with a STAAD input model, including a physical model if one is present. The option to include analysis output files is also available when creating a restore point.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Backup/Restore** tab and then click **Create Restore Point**.
3. (Optional) Provide the restore point details:
   a. Type a **Name** for the restore point.

      The name value will be populated and incremented automatically if you don't type a differenet value.
   b. Type a **Description** to provide additional information for this restore point.
   c. Check the option to **Include Analysis Data** if required.
4. Click **Create**.
   A message dialog informs you the restore point was created.
5. Click **Ok**.

You restore your model to this point at any time.
Related Links

- *Backup/Restore tab* (on page 2695)

I. To restore a model from a backup

To restore a model from a previously created restore point, use the following procedure.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Select the **Backup/Restore** tab and then click **Restore Model**.
3. Select the restore point and then click the **Restore** tool associated with that backup.
   A message dialog opens to confirm you want to overwrite the model with the restore point data.
4. Click **Yes**.
   The model opens with the data from the restore point and a message dialog opens to confirm the restore point was successfully loaded.

Related Links
- *Backup/Restore tab* (on page 2695)

I. To compare backups

To compare two restore points or a restore point with the current model state, use the following procedure.

Your model must have one or more restore points created.

1. Select the **File** ribbon tab.
   The STAAD.Pro Backstage view opens.
2. Either:

To...	Select...
compare the current model state with a previous restore point	select a restore point and then click **Compare with the Current Model**
compare two restore points	hold **<Ctrl>** and click on the two restore points of interest. Then click **Compare Models**.

   The **Backup and Restore - Compare Models** window opens to compare the input file data from the selected model states.
3. (Optional) To restore the contents of one of the selected files, click either: **Restore Left** or **Restore Right**.
4. Close the **Backup and Restore - Compare Models** window.

Related Links
- *Backup/Restore tab* (on page 2695)

I. Archives

Archive files can be used to collect all of the files within a STAAD.Pro into a single file, which is useful for records or sending project files to others.

STAAD.Pro projects consist of multiple files, such as the input file (file extension `.std`), analysis results (file extension `.anl`, etc.

An archive file uses the extension `.stz`.

Related Links

- *I. To share a STAAD.Pro project in ProjectWise* (on page 2076)

I. To create an archive

To create an archive file for a STAAD project, use the following procedure.

You cannot create an archive when a STAAD input file is open in the program.

1. Select the **Archive** tab on the Start page.
2. Select **Create**.

3. Select the **File Name** of the STAAD input file you want to archive.

   Click **Browse** to navigate to the file using a Windows open dialog.
4. Type the **File Name** of the archive file.
5. (Optional) Select a **Location** where you want the archive file created.

   The default location is the same folder containing the STAAD input file.
6. (Optional) Select the files associated with the input file you want included in the archive.

   All files are selected by default.
7. Click **Create**.

You can repeat this process and overwrite an existing archive file to update with any changes made to the STAAD input file, analysis results, etc.

Related Links

- *GS. Start Page* (on page 54)

I. To open an archive file

To open an archive file in STAAD.Pro, use the following procedure.

1. Select the **Open** tab on the **Start** page.
2. Select the Archive tab as the source type.

   > **Tip:** You can open recent Archive files with one click. These are listed on the **Archive** tab along the top of the Open tab

   The Windows **Open** dialog opens.

3. Select the **File Name** of the archive file (file extension `.stz`) you want to open.

   Tip: Click the **[...]** button to navigate to the folder where the archive is saved.
4. Select the **Location** where you want to store the extracted project files.
5. Click **Open**.

You can work with the STAAD input file and associated files within the archive as you normally would. Select **File** > **Close** to close the archive when you are finished.
Related Links
- *GS. Start Page* (on page 54)

I. To extract an archive

To extract the files within an archive, use the following procedure.

You cannot extract an archive when a STAAD input file is open in the program.

1. Select the **Archive** tab on the Start page.
2. Select **Create**.

3. Select the **File Name** of the archive file you want to extract.
   Click **Browse** to navigate to the file using a Windows open dialog.
4. (Optional) Select a **Location** to where you want the files within the archive extracted.
   The default location is the same folder containing the archive file.
5. (Optional) Select the files in the archive you want to extract.
   All files are selected by default.
6. Click **Extract**.

Related Links
- *GS. Start Page* (on page 54)

Bentley CONNECT Features

ProjectWise Project Association

STAAD.Pro CONNECT Edition allows you associate a file with a ProjectWise Project.

A ProjectWise Project is a single definition of a project for your entire organization and represents a one-to-one relationship with the contracted work being done by your organization.

Note: In order to utilize this feature in STAAD.Pro, you must:

1. Have the Bentley CONNECTION Client running. The CONNECTION Client is typically installed with STAAD.Pro.

2. Register with Bentley Cloud Services.
3. Sign in using your credentials with the CONNECTION Client.

For additional details on the benefits of using ProjectWise Projects, please visit https://www.bentley.com/connect/.

Related Links

- *GS. To create a new STAAD.Pro model* (on page 35)
- *GS. To open a STAAD.Pro model* (on page 36)

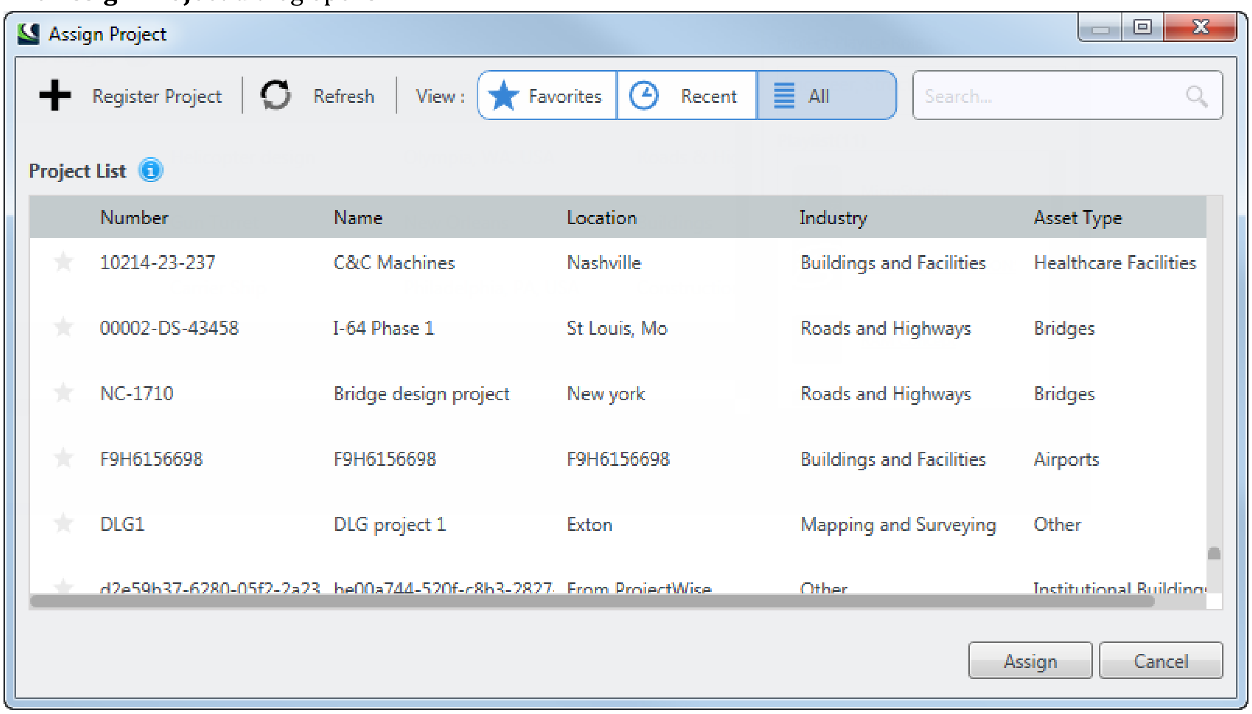
To Associate a ProjectWise Project with Your File

When you create a new file or open an existing file which is not associated with a project, use the following procedure to associate your file with a ProjectWise Project.

Note: You must be signed in using the CONNECTION Client to associate a ProjectWise Project with your file.

Tip: If you want to change the ProjectWise Project associated with your file, use the same following procedure.

1. On the **File** ribbon tab, select, **Cloud Services** > **Associate Project**
   The **Assign Project** dialog opens.



2. (Optional) If you want to register a new project, do the following:
   a. Click **Register Project**.

      The **Register a Project** page opens in your browser.

      Note: Only users with Admin/Co-admin roles can register a project.

   b. Type or select the required items (marked with an asterisk, "*")

   c. Click **Save**.
   A list of registered projects within your organization opens. The newly created project is highlighted in green.

   See Register a ProjectWise Project (on page 2072).

   > **Tip:** Alternately, you can visit connect.bentley.com and select **+New** on the **Recent Projects** tile on your personal dashboard.

3. Select the desired project from the list.

   > **Tip:** Use the View controls and Search tool to locate your project. If your project is not in the list, you can add a project following the steps given in Register a ProjectWise Project (on page 2072).

4. Click **Assign**.

Related Links

- *Assign Project dialog* (on page 2070)

To Disassociate a ProjectWise Project from a File

If you need to disassociate a file from a ProjectWise Project, select **Cloud Services** > **Disassociate Project** on the backstage view.

The project association is removed from the file.

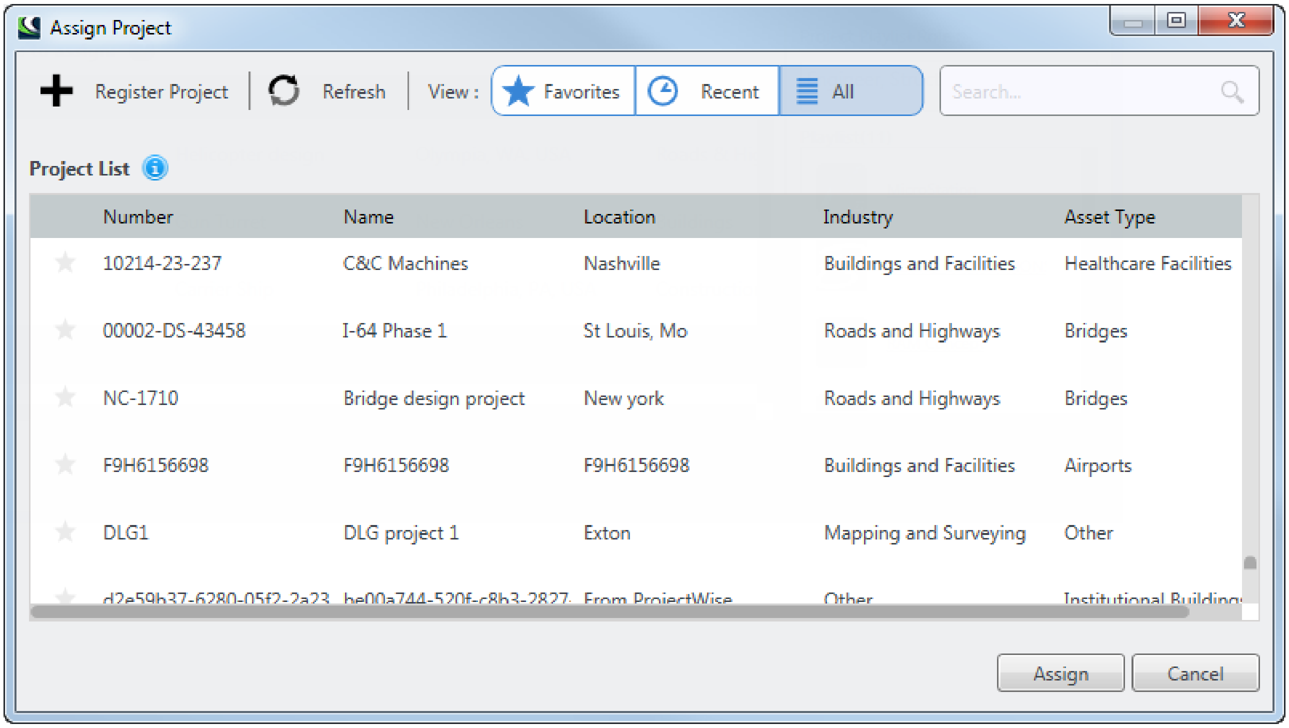
> **Tip:** If you want to change the ProjectWise Project association to another ProjectWise Project, this procedure is *not* necessary.

Related Links

- *Assign Project dialog* (on page 2070)

Assign Project dialog

Used to select a project to associate with your current file or model.



Register Project Opens the Register a Project page in your browser from where you can register a project.

Note: Only users with Admin/Co-admin roles can register a project.

Refresh Refreshes the list of available ProjectWise Projects.

View Allows you to choose the list of projects that you want to see in the list box. Following are the options:

- Favorites - Displays the projects that are marked as favorites.
- Recent - Displays the recently used projects.
- All - Displays all the projects.

Search Searches through the list of available projects.

List box Displays the following columns:

- Favorite - Allows you to favorite a project. Select the star icon in this column for the project that you want to mark as favorite.
- Number - Displays the number of the project.
- Name - Displays the name of the project.
- Location - Displays the geographic location of the project.
- Industry - Displays the industry of the project.
- Asset Type - Displays the asset type of the project.

Project Playlist Role For each user, the CONNECT Edition provides a personalized Bentley Playlist^{TM} of applications and apps specific to each participant's project role. Select your role from this drop-down list to see the applications and apps specific to your role.

Related Links

- *To Disassociate a ProjectWise Project from a File* (on page 2070)
- *To Associate a ProjectWise Project with Your File* (on page 2069)
- *To Register a ProjectWise Project* (on page 2072)

Register a ProjectWise Project

Organizations can enable CONNECTED Users to register and collaborate on ProjectWise Projects. These projects contain project information such as Project Name, Asset Industry, Asset Type, Location etc. While creating a file in a CONNECT Edition product, you can associate it to a ProjectWise Project where the project information is included in the data files as properties.

Note: Project files, such as DGN files and library files are not stored on the cloud. They can be stored locally, on a network, or in ProjectWise.

What is the ProjectWise Project Registration Utility?

The Project Registration utility is an administrative interface for registering an Organization's projects with Bentley. Registered projects are referred to as ProjectWise Projects. ProjectWise Projects provide information regarding the project themselves, as well as serving as a focal point for tying together other sources of project information.

For example, user and product usage for reporting and access to services available for each project.

Who can register a ProjectWise Project?

To register a ProjectWise Project, a user must have Administrator or Co-administrator privileges associated with their Bentley account. These privileges are required because registered ProjectWise Projects are Organization-wide resources that represent real-world projects and are used in many different locations for information organization and reporting. Therefore, access is limited to those members of an Organization with sufficient privileges to ensure that only recognized and permitted ProjectWise Projects be registered on behalf of an Organization.

Note: Users within the organization who were not designated as an Administrator or Co-Administrator who are requesting rights should contact their organizations Administrator. Bentley does *not* fulfill these requests.

To Register a ProjectWise Project

The Project Registration utility is used to provide information about a project as well as manage previously registered projects.

Note: Only users with Admin/Co-admin roles can register a project.

1. You can register a project from either the application or from the personal portal:

To use…	do this…
the personal portal	select **Bentley Cloud Services** > **CONNECTION Center** and then click the **+** (plus) at the top of the **ProjectWise Projects** panel.
STAAD.Pro	select **Bentley Could Services** > **Associate Project** and then click **+ Register Project** in the **Assign Project** dialog.

The **CONNECTION Center** page opens in your browser.

2. Fill out the form as needed. Required fields are marked with an asterisk ("*").

Project Number *	The unique project code or ID number that is officially used in your organization for internal tracking purposes. For example, `DMO-063 VP 778`.
Project Name *	The common name for the project within your organization. For example, `I-565 Interchange at County Line Road`.
Asset *	
Industry *	The asset industry this project belongs to. An asset industry is a group of like organizations with a common business function centered on a like set of infrastructure assets. For example, Electric Utility.
Type *	The type of asset this project will focus on. An asset type is a set of related assets. For example, the Asset Class Electric Network is comprised of the following assets: Distribution Network, Substation, and Transmission Network.
Use Location	Displays a **Location** field, where you can enter the name of the project location. For example, city/state/country.
Use Latitude/ Longitude	Displays the **Latitude** and **Longitude** fields, where you can enter the specific coordinates of where the project is located.
Engineering Location	
Time Zone	The time zone of the project location.
Billing Country *	
Status	The state of the project. **Active** means the project is open for participation. **Inactive** means the project is closed for participation.
Allow External Team Members	Allows the invitation process to include team members from external organizations.

3. Click **Save**.
A list of registered projects within your organization opens. The newly created project is highlighted in green.

Related Links

- *Assign Project dialog* (on page 2070)

Automated Updates via the CONNECTION Client

You will be notified of updates to STAAD.Pro automatically in the Bentley CONNECTION Client application.

This application is installed with STAAD.Pro CONNECT Edition and runs in the Windows system tray. You can manually check for updates by opening the CONNECTION Client and selecting the **Applications** tab.

Subscription Entitlement Service

Subscription Entitlement Service is Bentley's process for product activation and usage tracking, improving our licensing capabilities with features such as:

- License alert notifications when you are approaching a custom usage threshold
- Replacing site activation keys with user validation, enhancing security around your Bentley licenses and subscriptions

With traditional SELECT Licensing, product activation has been through an activation key that an Organization distributed to all users. With Subscription Entitlement Service, product activation is managed by user sign in through the CONNECTION Client, which is installed on each machine that uses Bentley applications. This offers a more secure and manageable system as it offers usage alerts, notifying your users when they are about to reach a certain usage limit set by the Administrator.

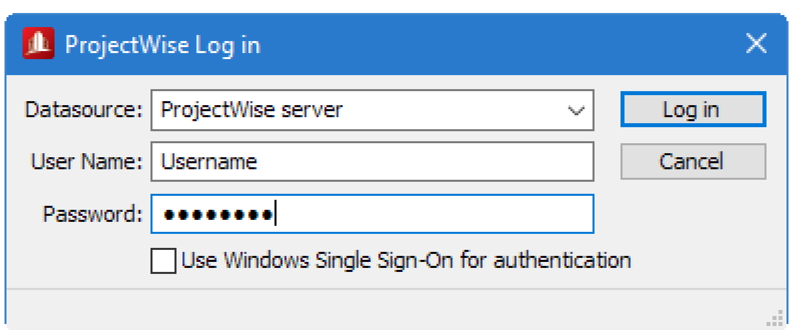
I. Using ProjectWise in STAAD.Pro

ProjectWise is an engineering project team collaboration system which is used to help teams improve quality, reduce rework, and meet project deadlines. One of the major pieces of functionality provided by ProjectWise is an Integration Server which allows data to be managed and shared across a distributed enterprise.

Note: This functionality requires that only *one* version ProjectWise be installed. It is *strongly* recommend that you use version 10.00.03.434 of the ProjectWise Design Integration client on the computer running STAAD.Pro. For more details on ProjectWise refer to the ProjectWise Design Integration client installation documentation. Refer to the ProjectWise Version Support Matrix for additional details.

Login

As authentication is required to access files stored on a ProjectWise repository, a login dialog allows the required details to be entered either with specific user credentials or by using the current windows login credentials.



Files that are accessed from a ProjectWise server are “Checked Out” and stored locally during the STAAD.Pro session until the file is closed and then it is returned to the server.

I. To open a STAAD input file from a ProjectWise repository

Note: Your most recent STAAD input files within PW repositories will be listed on the **ProjectWise** tab of the recent files list. You can click to select any of those files to open them.

1. On the STAAD.Pro Start page, select **Open** and then **ProjectWise**.
   You will be prompted to connect to a datasource if you have not already done so.

   The first time that a successful link to a ProjectWise server is established, a location in which checked out files are to be stored locally and additionally, where all the auxiliary data files are stored while STAAD is running, is required. Afterwards and on all future occasions, the ProjectWise open dialog presented is then presented where the repository can be navigated and filtered as defined in the ProjectWise documentation.
2. Navigate to and select the STAAD input file you want to check out and open.

   Note: The icons next to the STAAD filenames indicate the status of the file such as the current document, checked out to you, or locked as checked out to another user. Refer to the ProjectWise documentation for a full description of each icon.
3. Click **Open**.
   The STAAD input file is checked out and opened in STAAD.Pro.

Related Links

- *GS. Start Page* (on page 54)

I. To check in a STAAD input file to a ProjectWise repository

1. Either:

   Select the **File** ribbon tab and then select **Close** from the Backstage view menu.

   or

   Select the **Close** tool in the Quick Access toolbar.

   The **Check In** dialog opens.
2. (Optional) To increment the version of the file in the repository:
   a. Check the **Create new version during Check In** option.
   b. Type a **Version** label.

   If you choose to increment the option, you will not be able to only send updates or free the file. You must then Check In this version.
3. (Optional) To add a comment to the repository file, select the **Comment** tab and type your comment in the field.
4. Select the action you want to take with changes made to the file:

To...	Select...
Check in changes made to the file since it was checked out	Check In
Update the server copy with changes made since the file was last checked out, but keep the file checked out	Update Server Copy
Undo the check out and prevent any changes from being made to the server copy	Free

I. To share a STAAD.Pro project in ProjectWise

To share your STAAD.Pro input files and selected files associated with that project to a ProjectWise datasource, use the following procedure.

You cannot share a project when a STAAD input file is open in the program.

Shared files are saved as STAAD.Pro Archive files (file extension .stz).

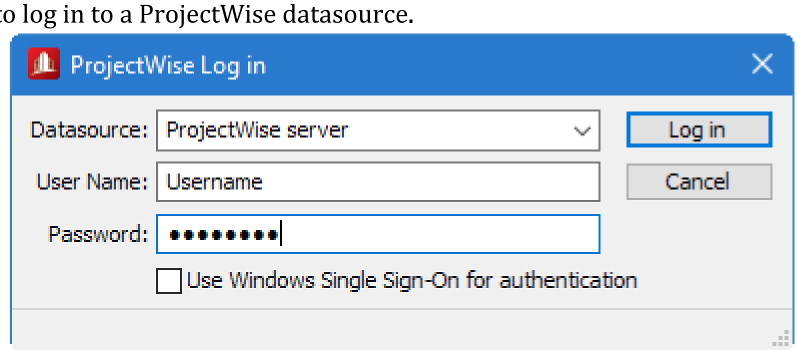
1. Select the **Share** page on the Start page.
2. Select **ProjectWise**.

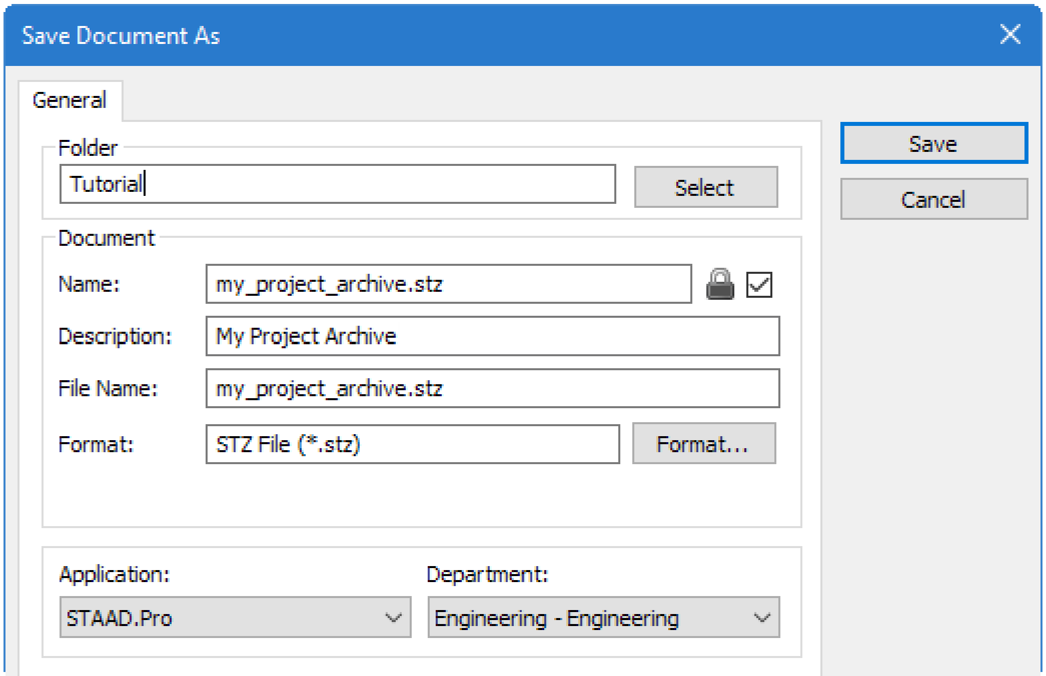
3. Select the **File Name** of the STAAD input file you want to share.

   Click **Browse** to navigate to the file using a Windows open dialog.
4. (Optional) Select the files associated with the input file you want included in the archive.

   All files are selected by default.
5. Click **Add**.
   You are prompted to log in to a ProjectWise datasource.

6. Select your **Datasource** and then click **Log in**.
   The **Save Document As** dialog opens.



Note: Depending on your ProjectWise Explorer configuration, you may further be prompted to select a Wizard for document creation. Please refer to the ProjectWise Explorer Help for further information on this feature.

7. Select a **Folder** in your ProjectWise datasource and provide any optional **Document** data.
8. Click **Save**.
   The **Add comment** dialog opens.
9. Type a descriptive comment for the and click **Add**.

Your project archive is now saved to your ProjectWise datasource.

You can open this archive file from the **Open** > **ProjectWise** tab in the **Start** page.

Related Links

- *GS. Start Page* (on page 54)
- *I. Archives* (on page 2066)

I. Importing Models

I. To import a DXF file

To import an AutoCAD® DXF™ drawing to use for model geometry, use the following procedure.

The following table maps the imported DXF objects to model entities.

DXF Object	STAAD Model Entity
line	member
3Dline	member
3Dface	plate
AcDbLine	member
AcDb3Dpolyline	one or more members
AcDbFace	finite element plates

You must have a model file already open to import data.

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select the **Import/Export** tab and then **DXF** in the **Import** group.
   A Windows **Open** dialog opens.
3. Select the DXF file you want to import and then click **Open**.
   The DXF Import dialog opens. This dialog is used to specify the vertical axis for data imported from a DXF file into a STAAD input file.
4. Select the **Structure Convention** to use.

   No Change - Use the vertical axis as defined in the 3D DXF file

   or

   Y Up - Adds a `SET Y UP` command to the STAAD input file.

   or

   Z Up - Adds a `SET Z UP` command to the STAAD input file.
5. Click **OK**.
   The DXF data is imported. The drawing objects are mapped to members and plates.

I. To import a CIS/2 file

To import a CIS/2 STEP file, use the following procedure.

The initial structural model will be created in your external 3D modeling software and exported as a CIS/2 STEP file, consisting of both analytical and physical model definitions. Limited load modeling can be done in some modeling software –such as SmartPlant® 3D– before the initial export.

The CIS/2 (CimSteel Integration Standard, Version 2) allows for the transfer of models using a prescribed data standard in the STEP (Part 21) format. There are various types of CIS/2 files. STAAD.Pro can import data from those types known as analysis models.

Note: If you are importing into an empty STAAD input file, then you will be creating a new file. Otherwise, the program assumes you are updating an existing file with changes in a CIS/2 file.

The following data can be imported through the CIS/2 format:

- Member properties
- Material constants (E, Density, Poisson, etc.) of steel profile sections
- Member orientation (Beta angles, Reference points)
- Member end conditions like Releases
- Support conditions
- Loading information

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select the **Import/Export** tab and then **CIS/2** in the **Import** group.
   A Windows **Open** dialog opens.
3. Select the CIS/2 (*.stp) file you want to import and then click **Open**.
   The Cis2Link dialog opens.
4. Either:

   for new models, click **Import Model**

   or

   for models with existing data, click **Update STAAD Model**

   The data is read and the import progress is reported in the log window.
5. (Optional) Click **Save Log File** to save this information to a text file.
6. Click **Close**.

The model data has been imported into your STAAD project.

If you need to update your STAAD.Pro model with changes made to the CIS/2 file, repeat this procedure.

Related Links

- *EX. CIS/2 Example Models* (on page 6252)

I. Exporting Models

I. To export to a DXF file

To export the model geometry to an AutoCAD DXF file, use the following procedure.

The following table maps the exported model entities to DXF objects.

STAAD Model Entity	DXF Object
member	line
element or surface	3Dface
solid	3Dface (multiple)

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select the **Import/Export** tab and then one of the following in the **Export** group:

Option	Description
3D DXF	Exports the entire model into a 3D DXF file. A Windows **Save As** dialog opens.
2D DXF	Exports a selected plane of the model into a 2D DXF file. This option opens the **Export 2D DXF** dialog.

3. If you select the 2D DXF option, you must specify the plane of the model:
   a. Select the global plane orientation to you want to use for the two dimensions.
   b. Select the node to define where this plane is cut in the model.
   c. Select the Options tab and then check the labels to include, if any, in the DXF drawing.
   d. Type a Text Size to use for the labels.
   e. Click **OK**.

   A Windows **Save As** dialog opens.
4. Type the name of the exported file and then click **Save**.

The model geometry is exported as a drawing file.

I. To export to a CIS/2 file

To export the model geometry to a CIS/2 STEP file, use the following procedure.

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select **CIS/2** in the **Import/Export** tab.
   A Windows **Save As** dialog opens.
3. Specify the file name and location and the click **Save**.
   The **Cis2Link** dialog opens.
4. Click **Export Model**.
   The data export progress is displayed in the log window.
5. (Optional) Click **Save Log File** to save this information to a text file.
6. Click **Close**.

I. To export structure date to AutoPipe

To export support frame data to a `.NTL` file for use in AutoPipe, use the following procedure.

A piping engineer who needs to consider the steelwork as their structural supports may need to import the STAAD.Pro model into AutoPipe. A macro called `ToAutoPipePub.vbs` is available in STAAD.Pro is used to facilitate this.

1. On the **Utilities** ribbon tab, select the **User Tools Export Model to AutoPipe** tool in the **Developer** group.

   The **Export STAAD Model to AutoPipe** dialog opens.
2. Type a file path and file name for the **AutoPipe Model file** (file extension `.ntl`) to which you want to export the STAAD.Pro support frame data.

> **Tip:** Click **[...]** to open a Windows **Save As** dialog.

3. Click **OK**.

I. To export to a SACS input file

To export the current STAAD.Pro model to a SACS input file, use the following procedure.

You must have an input file open in STAAD.Pro.

SACS is an offshore structural analysis and design software package offered by Bentley Systems, Inc.

1. On the **Utilities** ribbon tab, select the **User Tools Export Model to SACS** tool in the **Developer** group.

   The **Export STAAD Model to SACS** dialog opens.
2. Type a file path and file name for the **SACS Model file** (file extension `.inp`) to which you want to export the STAAD.Pro model data.

   > **Tip:** Click **[...]** to open a Windows **Save As** dialog.
3. Click **OK**.

I. Command Line Support

You can process STAAD.Pro input files using the STAAD analysis and design engine via the Windows command line. Using this method, you can run the analysis and design engine without opening the user interface at all.

> **Tip:** It is recommended to use a batch file with the necessary commands in it in order to simplify processing and reduce the need to repeatedly type commands. You can even use the Scheduled Tasks utility in Windows to automatically run these files for you on a schedule.

I. Command Line Syntax

The following format is used when running STAAD.Pro from the command line interface.

Syntax

```
<path>\SProStaad\SProStaad.exe STAAD <input_file> [Options]
```

where:

<path>	the installation path for STAAD.Pro, which is `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\ STAAD\` by default.

> **Tip:** You can add "`C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\`

STAAD\SProStaad\ ” to your Windows PATH environment variale to simply use the command SProStaad.

<input_file> the name of the STAAD input file you want to use as input.

Note: If the input file name includes spaces, you must use quote characters (") around the file name.

Expressions in square brackets (between '[' and ']') are optional, and can take one of the following values if used:

/s Run silently and auto close at the end of run

/h Run silently and hidden

Example

The following is a simple example which can be run from a directory containing `Example.std`.

```
C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\SProStaad
\SProStaad.exe STAAD Example.std /s
```

When the analysis and design engine is complete, it will close as a result of using the 'silent' option. Otherwise, this window must be manually closed.

Batch File Example

The following can be saved as a batch file (`.bat`) in any text editor. This batch file will analyze and design all the US examples which ship with the product by making use of a FOR command that loops through all STAAD input files beginning with the string “Examp”.

```
::Create a new variable with the location of the SProStaad.exe file
SET SProInst="C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition
\STAAD\SProStaad"
::Add this to the system path variable for this session
SET PATH="%PATH%";%SProInst%
::Create a variable to the default location of the STAAD US Example files
SET USExamps="%PUBLIC%\Documents\STAAD.Pro CONNECT Edition\Sample Models\US"

FOR %%f IN (%USExamps%\Examp*.std) DO (
    ECHO %%~nf
    SProStaad.exe STAAD "%%~nf.std" /h
)

EXIT /B
```

No additional windows will open while the analysis and design engine runs as a result of using the “hidden” option.

Note: The use of the variables *SProInst* and *USExamps* are *not* necessary. They are only used to highlight that if your installation folders differ from the default, this should be changed accordingly.

I. Copy/Paste from Spreadsheets

The Copy/Paste facility in input tables in STAAD.Pro allows different column sizes and configuration to be copied from an external spreadsheet file and then pasted into STAAD.Pro.

On any STAAD.Pro input grid table capable of handling manual input, rows and columns copied from a spreadsheet can be directly pasted into STAAD.Pro. If the columns in the spreadsheet do not match the column headers in a STAAD.Pro grid table, STAAD.Pro will ask to assign the columns in the spreadsheet to the appropriate columns in the STAAD.Pro grid table. In the figure shown below, a sample spreadsheet file containing the coordinates of some nodes is displayed. A selection of the spreadsheet's cells have been copied to the system clipboard.

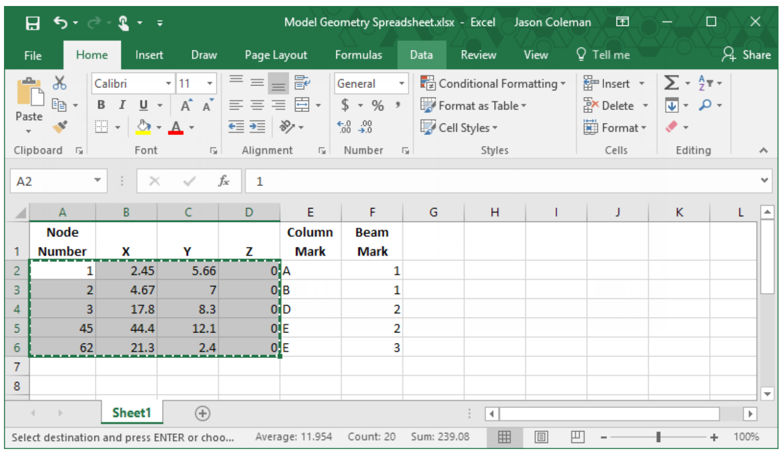


Figure 207: Example spreadsheet data

This data is then selected and attempted to be copied into the STAAD.Pro Nodes table shown in the next figure.

Tip: The entire row in the STAAD table should be selected before pasting the data. The paste function will *not* be active if only one cell is selected.

The number of columns in the first figure does not match the number of columns in the second figure. In this case, the **Select Column Mapping** dialog opens when attempting to paste the spreadsheet data.

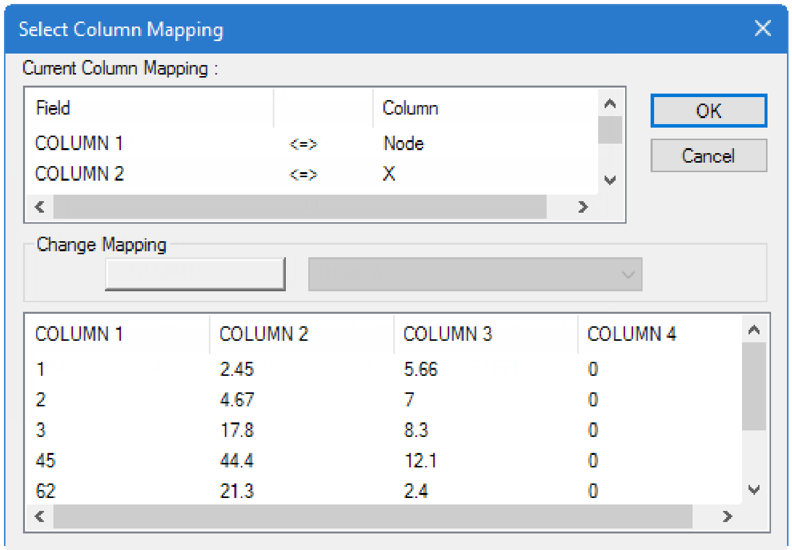


Figure 208: Select Column Mapping dialog box

The upper list box displays the current mapping between the spreadsheet (the **Field** column) and the STAAD.Pro table (the **Column** column). By clicking on a particular **Field**, the mapping between the spreadsheet and the STAAD.Pro table can be changed under the **Change Mapping** list box. The options listed under the **Change Mapping** list box are the available columns in the STAAD.Pro table. If a particular column coming from the spreadsheet is not relevant for the STAAD.Pro table, simply choose the option **Not Mapped** under the **Change Mapping** list box.

Tip: While the **Select Column Mapping** dialog is open, the system's ability to paste from the clipboard may be interrupted. Closing this dialog (either by clicking **OK** or **Cancel**) will restore normal clipboard functionality.

9

General Engineering Theory

This section serves to familiarize you with the basic principles involved in the implementation of the various analysis and design facilities offered by the STAAD.Pro engine. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the STAAD input file.

G.1 Input Generation

The STAAD input file can be created through the graphical user interface (GUI) modeling facility or using the STAAD.Pro Editor. The graphical modeling facility creates the input file through an interactive, graphics oriented procedure. In general, any plain-text editor may be utilized to edit or create the STAAD input file, but the STAAD.Pro Editor is recommended.

Note: Some of the automatic generation facilities of the STAAD command language will be reinterpreted by the GUI as lists of individual model elements upon editing the file using the GUI. A warning message is presented prior to this occurring. This does not result in any effective difference in the model or how it is analyzed or designed.

It is important to understand that STAAD.Pro is capable of analyzing a wide range of structures. While some parametric input features are available in the GUI, the formulation of input is the responsibility of you, the user. The program has no means of verifying that the structure input is that which was intended by the engineer.

G.2 Types of Structures

A `STRUCTURE` can be defined as an assemblage of elements. STAAD.Pro is capable of analyzing and designing structures consisting of frame, plate/shell, and solid elements. Almost any type of structure can be analyzed by STAAD.Pro.

SPACE A3D framed structure with loads applied in any plane. This structure type is the most general.

PLANE This structure type is bound by a global X-Y coordinate system with loads in the same plane.

TRUSS This structure type consists of truss members which can have only axial member forces and no bending in the members.

FLOOR A 2D or 3D structure having no horizontal (global X or Z) movement of the structure [FX, FZ, and MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a this type of structure. Columns can also be modeled with the floor in a `FLOOR` structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a `SPACE` structure.

Specification of the correct structure type reduces the number of equations to be solved during the analysis. This results in a faster and more economic solution for the user. The degrees of freedom associated with frame elements of different types of structures is illustrated in the following figure.

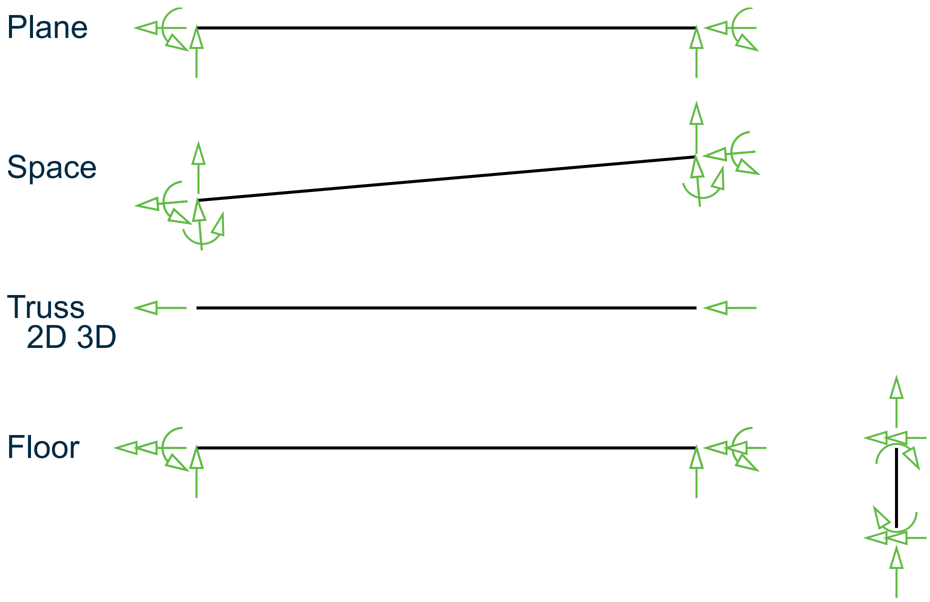


Figure 209: Degrees of freedom in each type of Structure

Related Links

- *TR.2 Problem Initiation and Model Title* (on page 2202)

G.3 Unit Systems

You are allowed to input data and request output in almost all commonly used engineering unit systems including “meter - kilonewton - second” (MKS), International System of Units (SI), and “feet - pound - second” (FPS). In the input file, you may change units as many times as required. Mixing and matching between length and force units from different unit systems is also allowed.

The input unit for angles (or rotations) is degrees. However, in `JOINT DISPLACEMENT` output, the rotations are provided in radians.

For all output, the units are clearly specified by the program.

Related Links

- *TR.3 Unit Specification* (on page 2203)

G.4 Coordinate Systems and Structure Geometry

A structure is an assembly of individual components such as beams, columns, slabs, plates etc.. In STAAD.Pro, frame elements and plate elements may be used to model the structural components. Typically, modeling of the structure geometry consists of two steps:

A. Identification and description of joints or nodes.

B. Modeling of members or elements through specification of connectivity (incidences) between joints.

In general, the term MEMBER will be used to refer to frame elements and the term ELEMENT will be used to refer to plate/shell and solid elements. Connectivity for MEMBERs may be provided through the MEMBER INCIDENCE command while connectivity for ELEMENTs may be provided through the ELEMENT INCIDENCE command.

STAAD.Pro uses two types of coordinate systems to define the structure geometry and loading patterns. The GLOBAL coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A LOCAL coordinate system is associated with each member (or element) and is utilized in MEMBER END FORCE output or local load specification.

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.12 Member Incidences Specification* (on page 2221)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2235)
- *TR.17 Rotation of Structure Geometry* (on page 2239)
- *TR.26.1 Define Material* (on page 2303)

G.4.1 Global Coordinate System

The following coordinate systems are available for specification of the structure geometry.

Note: The following figures depict the global coordinates with Y as vertical. The coordinate systems follow the similar right-hand rule when Z is vertical. Refer to SET { Y | Z } UP (on page 2210) for additional details.

Conventional Cartesian Coordinate System

This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the joint locations and loading directions.

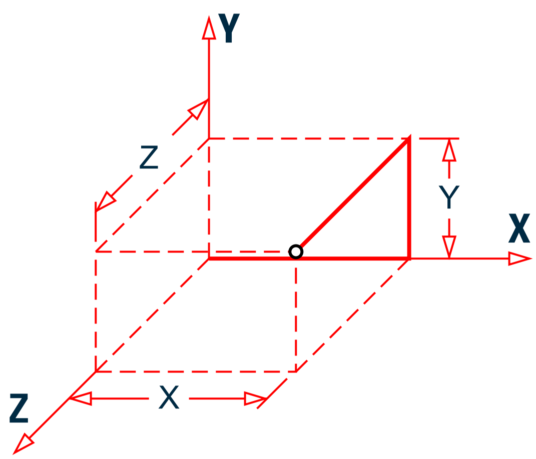


Figure 210: Cartesian (Rectangular) Coordinate System

Cylindrical Coordinate System

In this coordinate system, the X and Y coordinates of the conventional Cartesian system are replaced by R (radius) and Ø (angle in degrees). The Z coordinate is identical to the Z coordinate of the Cartesian system and its positive direction is determined by the right hand rule.

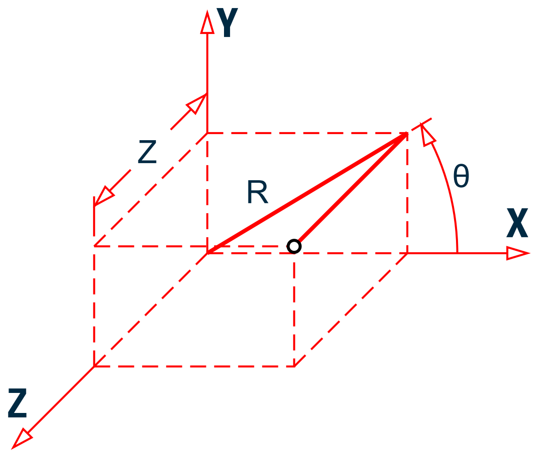


Figure 211: Cylindrical Coordinate System

Reverse Cylindrical Coordinate System

This is a cylindrical type coordinate system where the R- Ø plane corresponds to the X-Z plane of the Cartesian system. The right hand rule is followed to determine the positive direction of the Y axis.

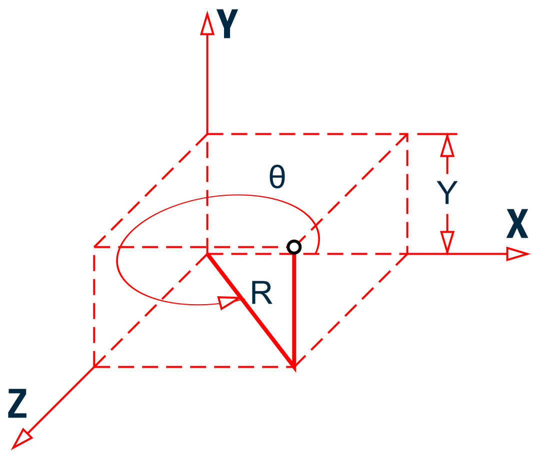


Figure 212: Reverse Cylindrical Coordinate System

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.12 Member Incidences Specification* (on page 2221)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2235)
- *TR.17 Rotation of Structure Geometry* (on page 2239)
- *TR.26.1 Define Material* (on page 2303)

G.4.2 Local Coordinate System

A local coordinate system is associated with each member. Each axis of the local orthogonal coordinate system is also based on the right hand rule.

The following figures show a beam member with start joint 'i' and end joint 'j'. The positive direction of the local x-axis is determined by joining 'i' to 'j' and projecting it in the same direction. The right hand rule may be applied to obtain the positive directions of the local y and z axes. The local y and z-axes coincide with the axes of the two principal moments of inertia. Note that the local coordinate system is always rectangular.

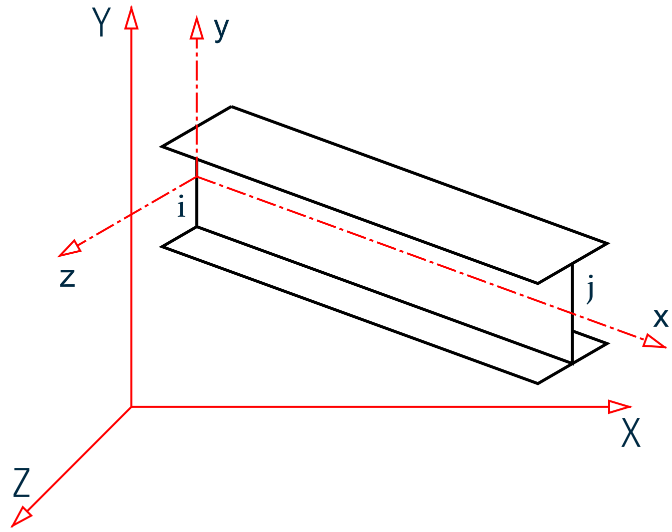


Figure 213: When Global-Y is Vertical

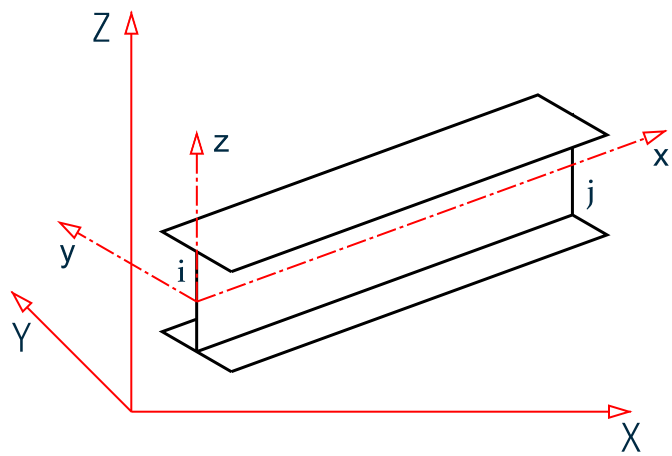
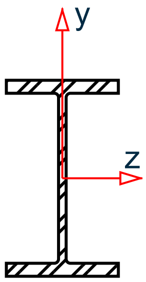
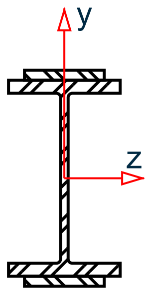
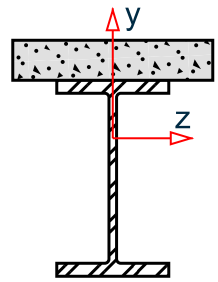
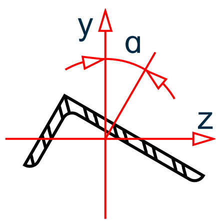
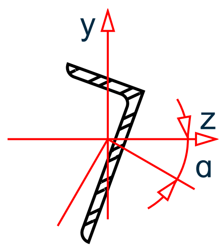
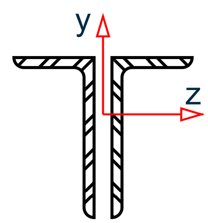
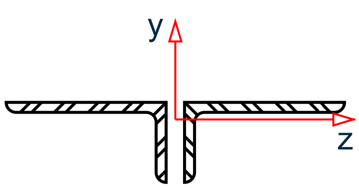
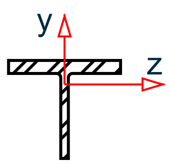
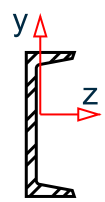
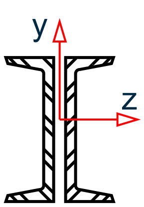
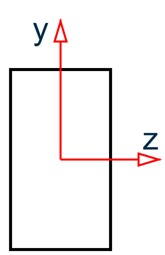
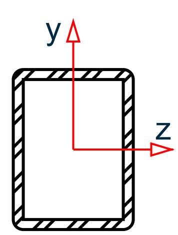


Figure 214: When Global-Z is Vertical (that is, SET Z UP is specified)

A wide range of cross-sectional shapes may be specified for analysis. These include rolled steel shapes, user-specified prismatic shapes etc.. The following table shows local axis system(s) for these shapes.

Table 222: Local axis system for various cross sections when global Y axis is vertical

 Wide Flange - ST	 Wide Flange - TB	 Wide Flange - CM
 Angle - ST	 Angle - RA	 Angle - LD (Long legs back-to-back)
 Angle - SD (Short legs back-to-back)	 Wide Flange - T	 Channel - ST
 Channel - D	 Prismatic	 Tube - ST

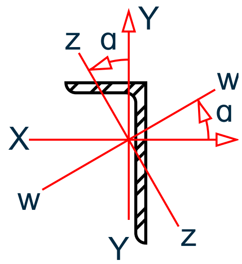
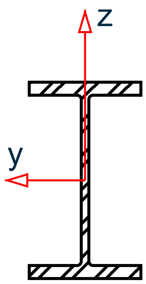
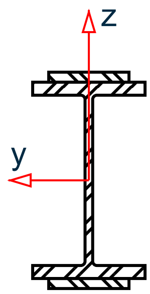
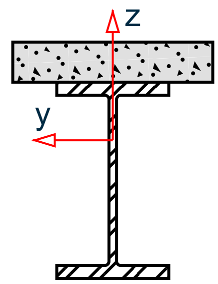


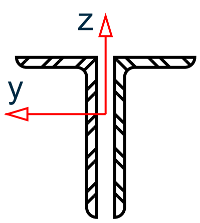
Figure 215: Labels for the local axes of a single angle as defined in AISC publications.

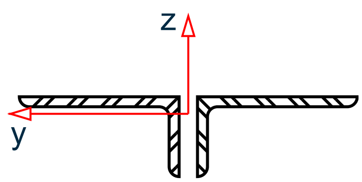
Table 223: Local axis system for various cross sections when global Z axis is vertical (SET Z UP is specified).

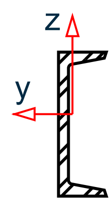
<table>
<tr>
<td>

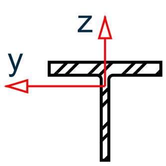
Wide Flange - ST</td>
<td>

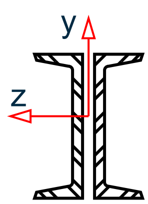
Wide Flange - TB</td>
<td>

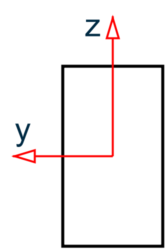
Wide Flange - CM</td>
</tr>
<tr>
<td>

Angle - LD</td>
<td>

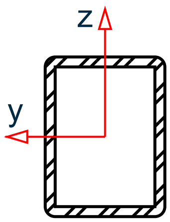
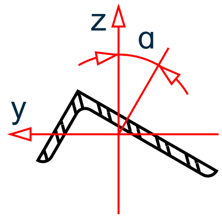
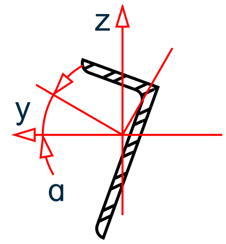
Angle - SD</td>
<td>

Channel - ST</td>
</tr>
<tr>
<td>

Wide Flange - T</td>
<td>

Channel - D</td>
<td>

Prismatic</td>
</tr>
</table>

 Tube - ST	 Angle - ST	 Angle - RA

Note: The local x-axis of the above sections is going into the screen.

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.12 Member Incidences Specification* (on page 2221)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2235)
- *TR.17 Rotation of Structure Geometry* (on page 2239)
- *TR.26.1 Define Material* (on page 2303)
- *TR.11 Joint Coordinates Specification* (on page 2217)

G.4.3 Relationship Between Global and Local Coordinates

Since the input for member loads can be provided in the local and global coordinate system and the output for member-end-forces is printed in the local coordinate system, it is important to know the relationship between the local and global coordinate systems. This relationship is defined by an angle measured in the following specified way. This angle will be defined as the beta (β) angle. For offset members the beta angle/reference point specifications are based on the offset position of the local axis, not the joint positions.

Beta Angle

When the local x-axis is parallel to the global Vertical axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis (or local Y for `SET Z UP`) has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis (global Y axis for `SET Z UP`).

When the local x-axis is not parallel to the global Vertical axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis (or local Y for `SET Z UP`) parallel to the global X-Z plane (or global X-Y plane for `SET Z UP`)and the local y-axis (or local z for `SET Z UP`) in the same positive direction as the global vertical axis. Figure 1.7 details the positions for beta equals 0 degrees or 90 degrees. When providing member loads in the local member axis, it is helpful to refer to this figure for a quick determination of the local axis system.

Reference Point

An alternative to providing the member orientation is to input the coordinates (or a joint number) which will be a reference point located in the member x-y plane (x-z plane for `SET Z UP`) but not on the axis of the member. From the location of the reference point, the program automatically calculates the orientation of the member x-y plane (x-z plane for `SET Z UP`).

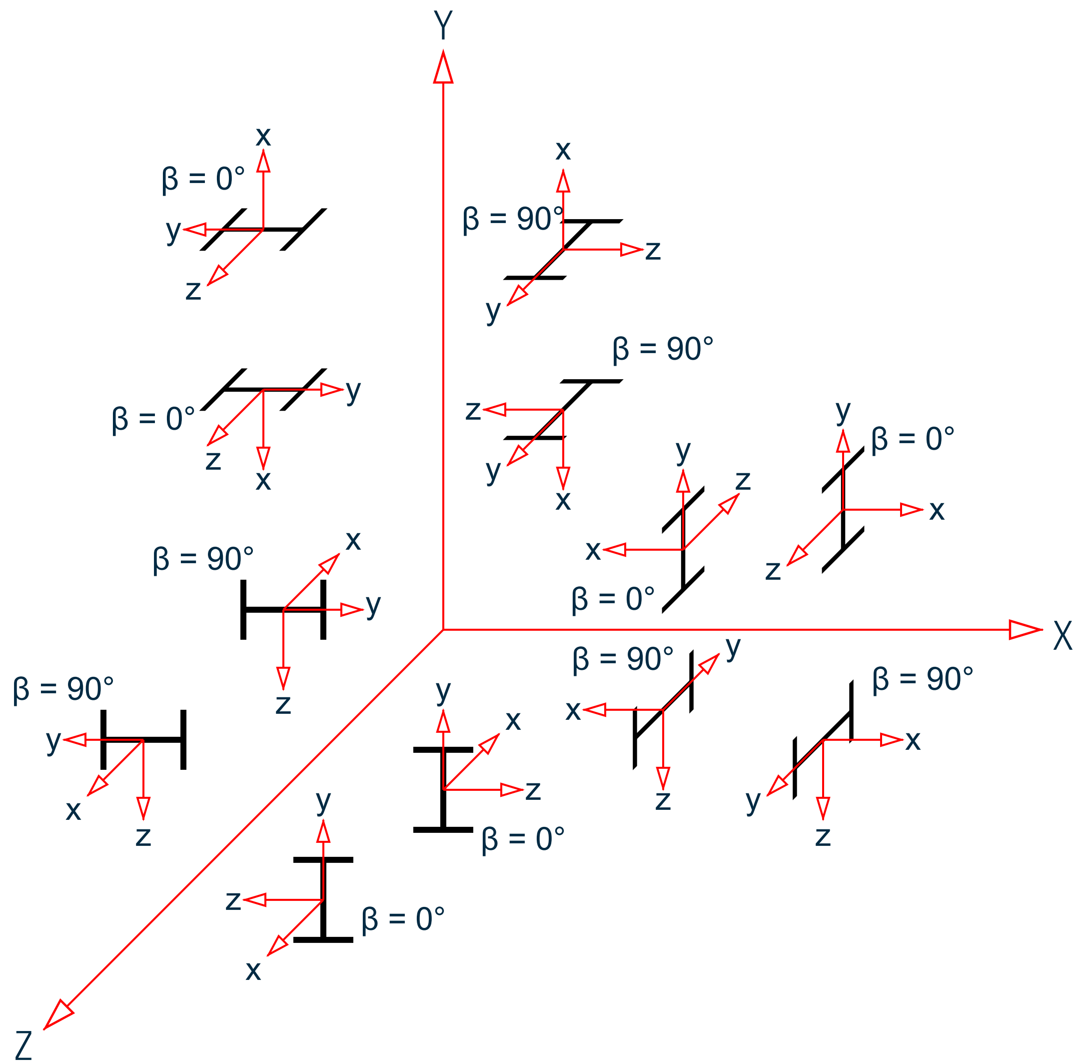


Figure 216: Relationship between Global and Local axes

Reference Vector

This is yet another way to specify the member orientation. In the reference point method described above, the X,Y,Z coordinates of the point are in the global axis system. In a reference vector, the X,Y,Z coordinates are specified with respect to the local axis system of the member corresponding to the BETA 0 condition.

A direction vector is created by the program. The program then calculates the Beta Angle using this vector.

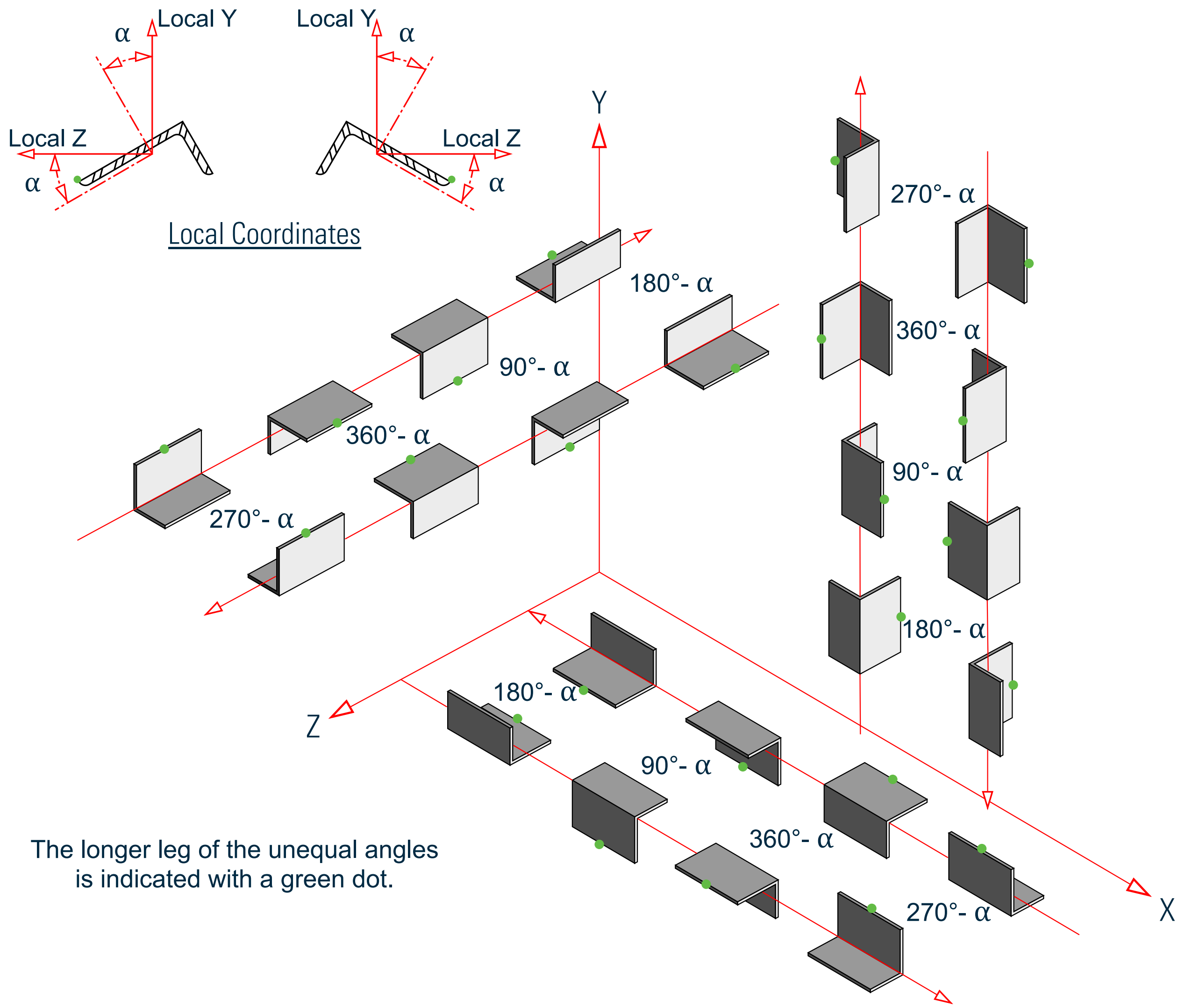


Figure 217: Beta rotation of equal & unequal legged 'ST' angles

Note: The order of the joint numbers in the `MEMBER INCIDENCES` command determines the direction of the member's local x-axis.

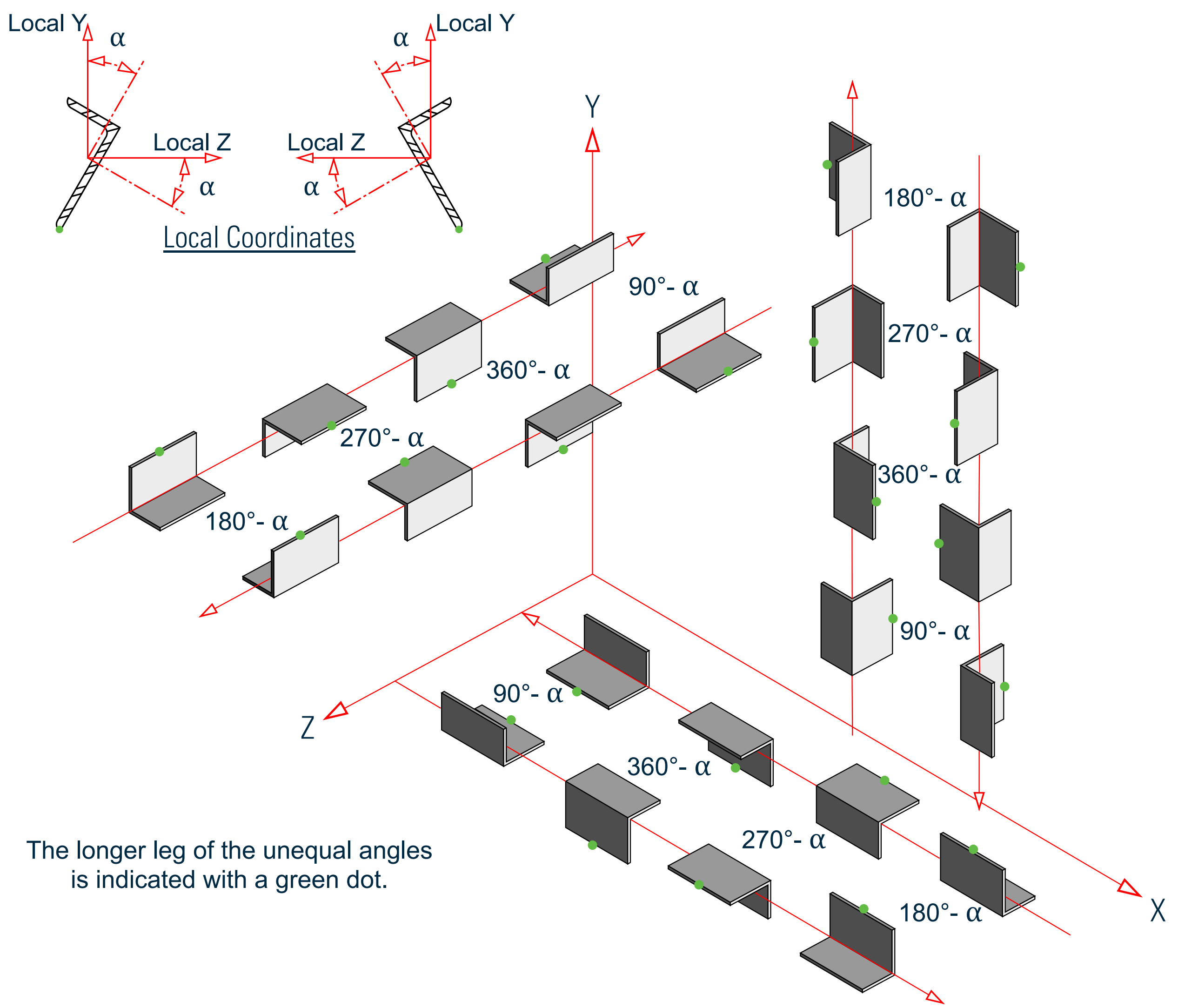


Figure 218: Beta rotation of equal & unequal legged 'RA' angles

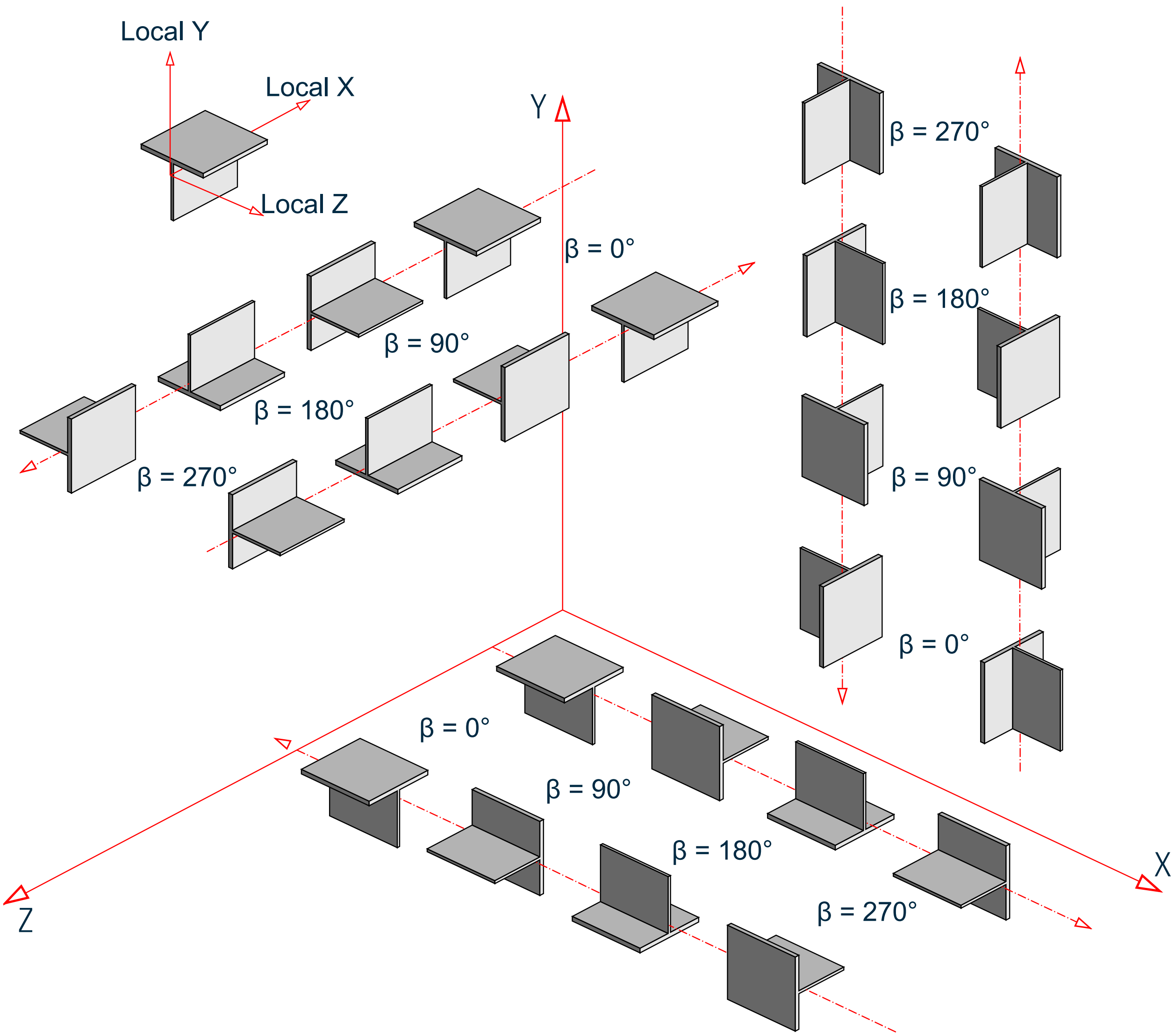


Figure 219: T-shape member orientation for various Beta angles when Global-Y axis is vertical

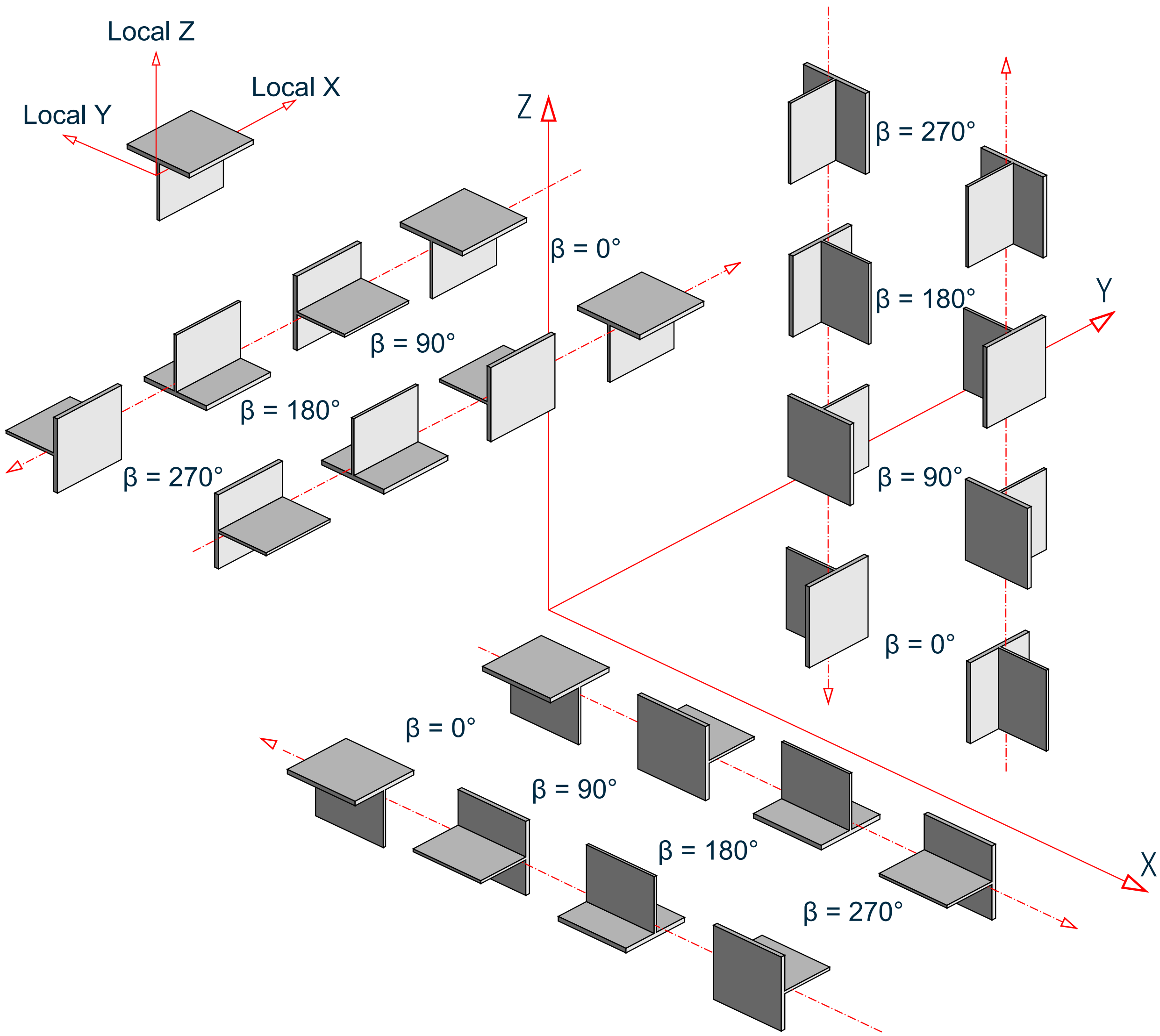


Figure 220: T-shape member orientation for various Beta angles when Global-Z axis is vertical (that is, `SET Z UP` *is specified)*

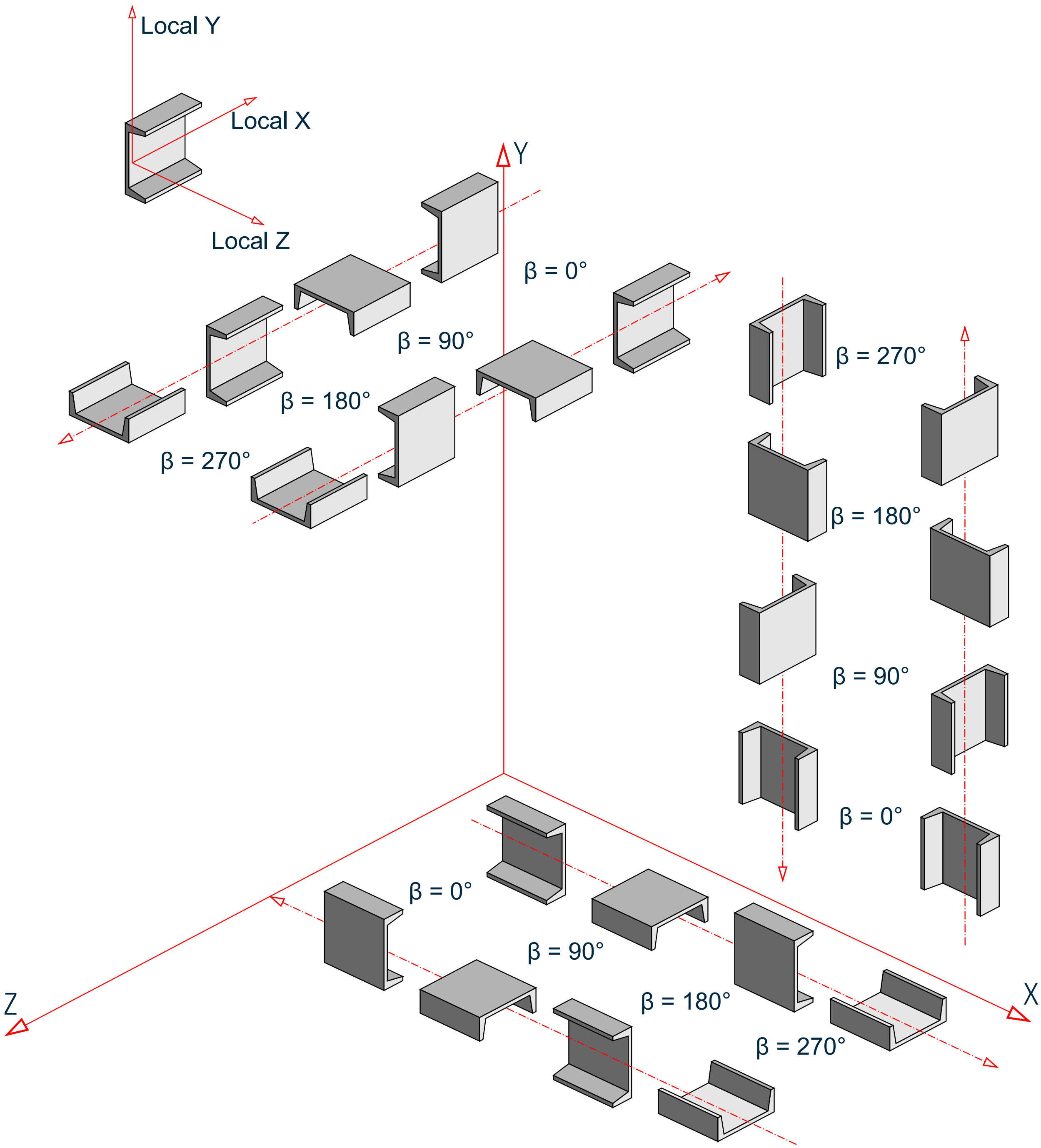


Figure 221: Channel member orientation for various Beta angles when Global-Y axis is vertical

Related Links

- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *Beta Angle dialog* (on page 2811)
- *M. To assign a member rotation angle* (on page 799)

- *M. To align a single angle to its flanges* (on page 800)
- *Reference Point dialog* (on page 2812)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *M. To align a member to a reference point* (on page 801)
- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.12 Member Incidences Specification* (on page 2221)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.16.1 Listing of Entities by Specifying Groups* (on page 2235)
- *TR.17 Rotation of Structure Geometry* (on page 2239)
- *TR.26.1 Define Material* (on page 2303)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *TR.20.8 Curved Member Specification* (on page 2273)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

G.5 Finite Element Information

STAAD.Pro is equipped with a plate/shell finite element, solid finite element and an entity called the surface element. The features of each is explained in the following sections.

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.13 Plate and Solid Elements* (on page 2224)
- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2224)
- *TR.13.2 Solid Element Incidences Specification* (on page 2225)
- *TR.14 Plate Element Mesh Generation* (on page 2227)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.21 Element/Surface Property Specification* (on page 2284)
- *TR.21.1 Element Property Specification* (on page 2284)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *TR.32.3 Element Load Specifications* (on page 2468)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)

G.5.1 Plate and Shell Elements

The plate (or shell) finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

"Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a `MESH GENERATION` facility is available.

You may also use the element for `PLANE STRESS` action only (i.e., membrane/in-plane stiffness only). The `ELEMENT PLANE STRESS` command should be used for this purpose.

Related Links

- *TR.21.1 Element Property Specification* (on page 2284)

- *Plate ElementProperty dialog* (on page 2813)
- *M. To specify plate thickness* (on page 811)
- *Plate Reference Point dialog* (on page 2813)
- *M. To align a plate to a reference point* (on page 811)
- *Plate Specs dialog* (on page 2791)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *M. To assign plates as plane stress* (on page 815)
- *Plate Specs dialog* (on page 2791)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *M. To assign inplane rotation behavior to plates* (on page 815)
- *Plate Specs dialog* (on page 2791)
- *TR.22.3 Element Ignore Stiffness* (on page 2288)
- *M. To ignore plate stiffness* (on page 816)
- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.13 Plate and Solid Elements* (on page 2224)
- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2224)
- *TR.13.2 Solid Element Incidences Specification* (on page 2225)
- *TR.14 Plate Element Mesh Generation* (on page 2227)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.21 Element/Surface Property Specification* (on page 2284)
- *TR.21.1 Element Property Specification* (on page 2284)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *TR.32.3 Element Load Specifications* (on page 2468)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)
- *TR.22.2 Element Release Specification* (on page 2287)

Geometry Modeling Considerations

1. The program automatically generates a fictitious, center node "0" (see the following figure) at the element center.

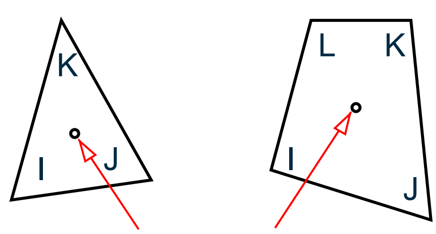


Figure 222: Fictitious center node (in the case of triangular elements, a fourth node; in the case of rectangular elements, a fifth node)

2. While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (see the following figure). For better efficiency, similar elements should be numbered sequentially.

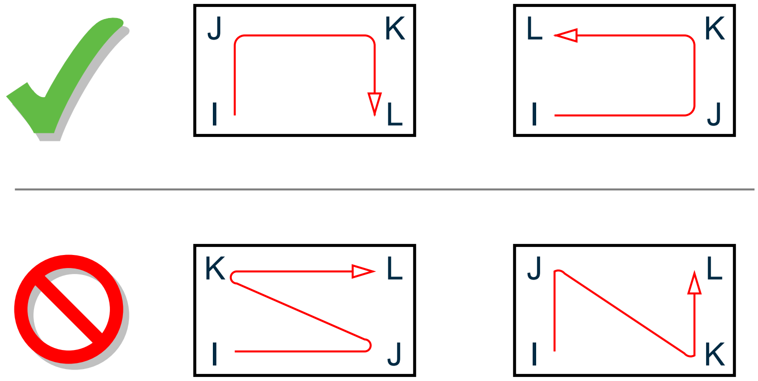


Figure 223: Examples of correct and incorrect numbering sequences

3. Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1.
4. Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 and never larger than 180.

   a. Opposite sides cross each other
   b. Ratio of the lengths of the longest side to the shortest side exceeds eight
   c. Ratio of the sides exceeds eight
   d. Angle between two adjacent sides exceeds 120 degrees

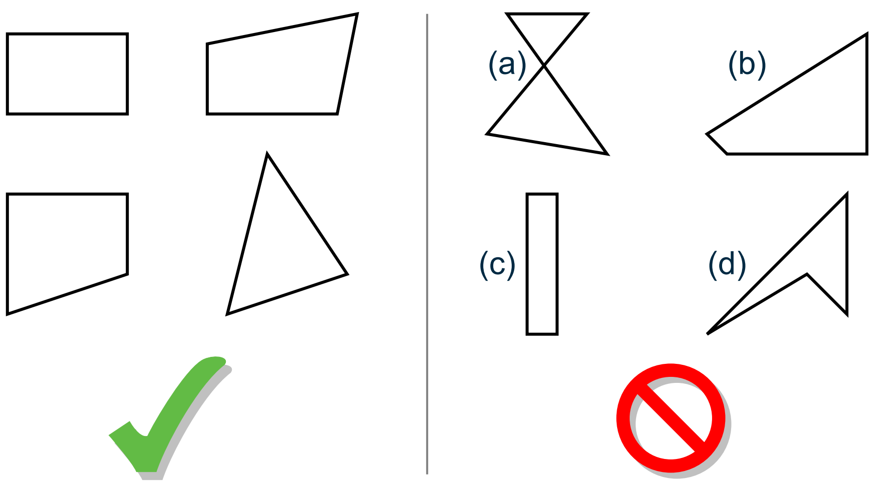


Figure 224: Some examples of good and bad elements in terms of the angles

Load Specification for Plate Elements

1. Joint loads at element nodes in global directions.
2. Concentrated loads at any user specified point within the element in global or local directions.
3. Uniform pressure on element surface in global or local directions.
4. Partial uniform pressure on user specified portion of element surface in global or local directions.
5. Linearly varying pressure on element surface in local directions.

6. Temperature load due to uniform increase or decrease of temperature.
7. Temperature load due to difference in temperature between top and bottom surfaces of the element.

Theoretical Basis

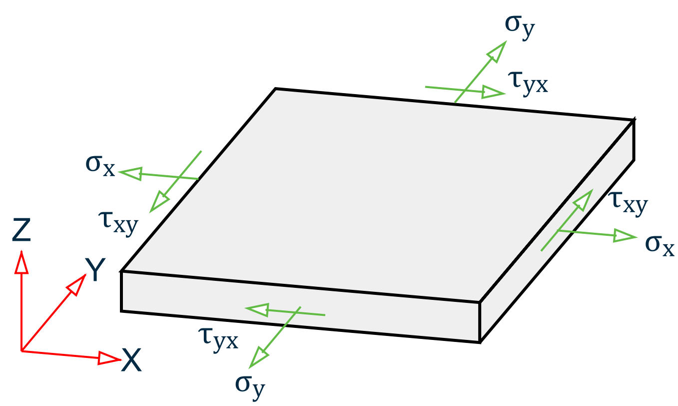


Figure 225: Assumed stress distribution

The incomplete quadratic assumed stress distribution:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & -x^2 \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ \ldots \\ a_{10} \end{pmatrix}$$

a_1 through a_{10} = constants of stress polynomials

The following quadratic stress distribution is assumed for plate bending action:

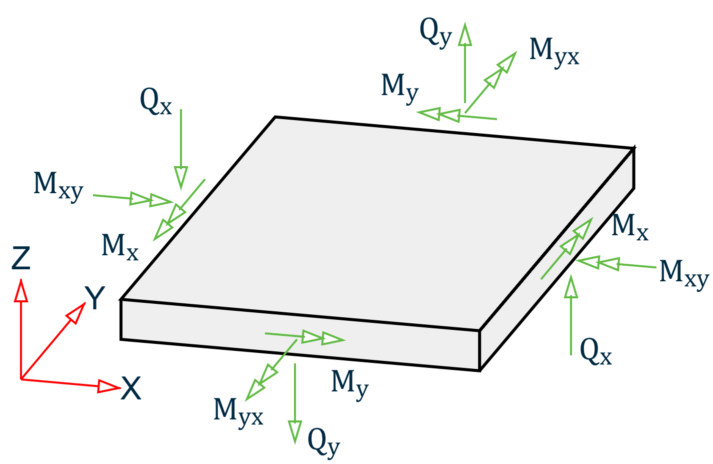


Figure 226: Quadratic stress distribution assumed for bending

The incomplete quadratic assumed stress distribution:

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & y^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & -xy \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & x & y \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ \dots \\ a_{12} \\ a_{13} \end{Bmatrix}$$

a_1 through a_{13} = constants of stress polynomials

The distinguishing features of this finite element are:

1. Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see the following figure) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.

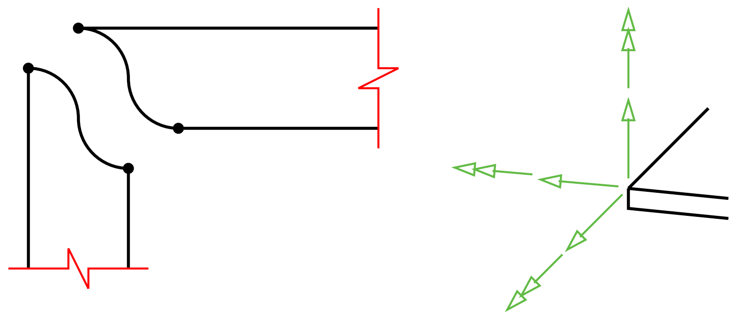


Figure 227: Adjacent elements at some angle

2. The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.
3. Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.
4. These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.
5. These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.
6. These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.
7. Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions.
8. The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.
9. The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of convergence. Thus the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.
10. Stress retrieval at nodes and at any point within the element.

Plate Element Local Coordinate System

1. The vector pointing from I to J is defined to be parallel to the local x- axis.
2. For triangles: the cross-product of vectors IJ and JK defines a vector parallel to the local z-axis, i.e., z = IJ x JK.

   For quads: the cross-product of vectors IJ and JL defines a vector parallel to the local z-axis, i.e., z = IJ x JL.
3. The cross-product of vectors z and x defines a vector parallel to the local y- axis, i.e., y = z x x.
4. The origin of the axes is at the center (average) of the four joint locations (three joint locations for a triangle).

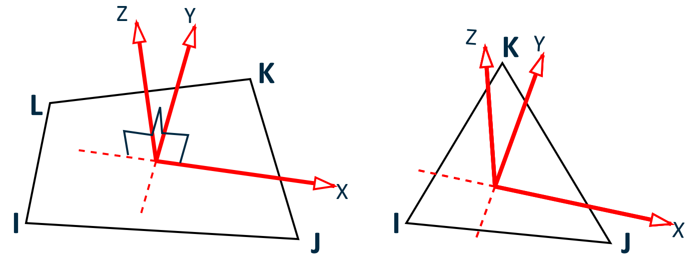


Figure 228: Element origin

Output of Plate Element Stresses and Moments

`ELEMENT` stress and moment output is available at the following locations:

A. Center point of the element.
B. All corner nodes of the element.
C. At any user specified point within the element.

Following are the items included in the `ELEMENT STRESS` output.

Table 224: Items included in the Stress Element output

Title	Description
SQX, SQY	Shear stresses (Force/ unit length/ thickness)
SX, SY	Membrane stresses (Force/unit length/ thickness)
SXY	Inplane Shear Stress (Force/unit length/ thickness)
MX, MY, MXY	Moments per unit width (Force x Length/length) (For Mx, the unit width is a unit distance parallel to the local Y axis. For My, the unit width is a unit distance parallel to the local X axis. Mx and My cause bending, while Mxy causes the element to twist out-of-plane.)
SMAX, SMIN	Principal stresses in the plane of the element (Force/ unit area). The 3rd principal stress is 0.0

Title	Description
TMAX	Maximum 2D shear stress in the plane of the element (Force/unit area)
VONT, VONB	3D Von Mises stress at the top and bottom surfaces, where: $$VM = 0.707[(SMAX - SMIN)^2 + SMAX^2 + SMIN^2]^{1/2}$$
TRESCAT, TRESCAB	Tresca stress, where TRESCA = MAX[|(Smax-Smin)| , |(Smax)| , |(Smin)|]

Notes

1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in the following section.
2. To obtain element stresses at a specified point within the element, you must provide the location (local X, local Y) in the coordinate system for the element. The origin of the local coordinate system coincides with the center of the element.
3. The 2 nonzero Principal stresses at the surface (SMAX & SMIN), the maximum 2D shear stress (TMAX), the 2D orientation of the principal plane (ANGLE), the 3D Von Mises stress (VONT & VONB), and the 3D Tresca stress (TRESCAT & TRESCAB) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.
4. The third principal stress is assumed to be zero at the surfaces for use in Von Mises and Tresca stress calculations. However, the TMAX and ANGLE are based only on the 2D inplane stresses (SMAX & SMIN) at the surface. The 3D maximum shear stress at the surface is not calculated but would be equal to the 3D Tresca stress divided by 2.0.

Sign Convention of Plate Element Stresses and Moments

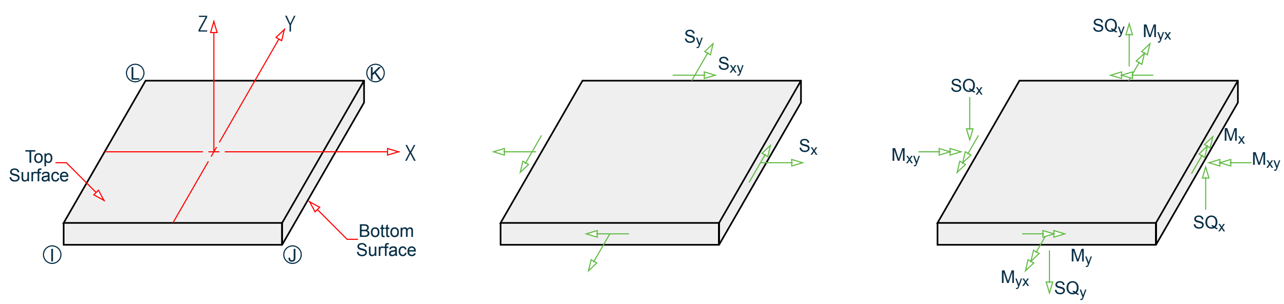


Figure 229: Sign conventions for plate stresses and moments

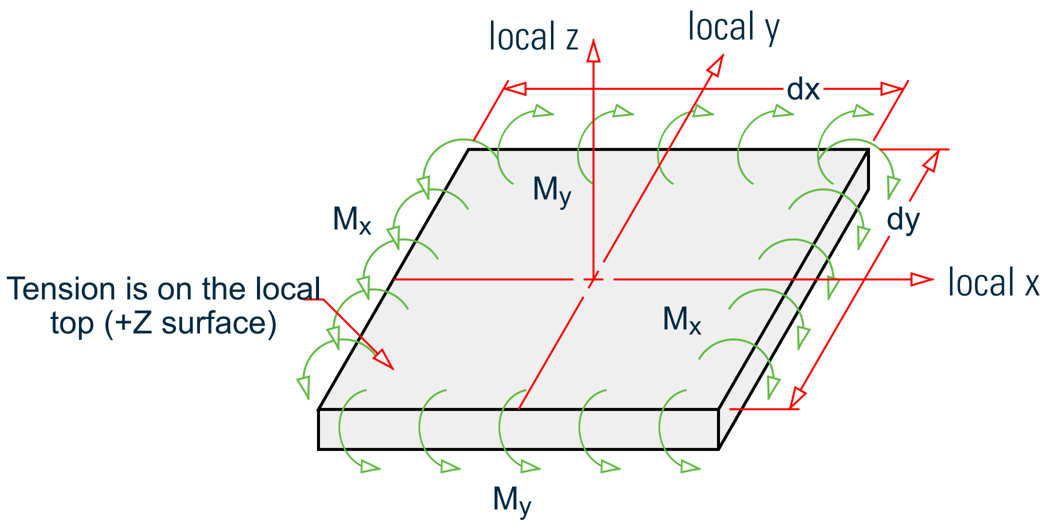


Figure 230: Sign convention for plate bending

Mx is the Bending Moment on the local x face and the local x-face is the face perpendicular to the local x-axis.

My is the Bending Moment on the local y face and the local y-face is the face perpendicular to the local y-axis.

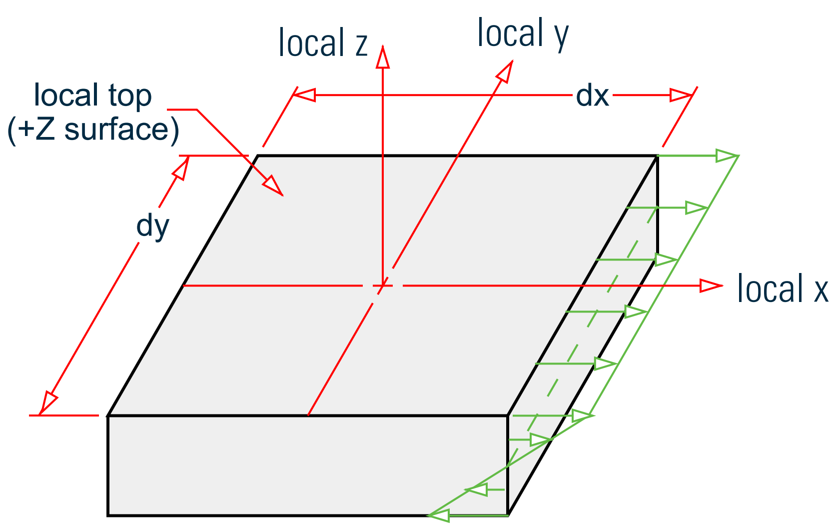


Figure 231: Stress caused by Mx

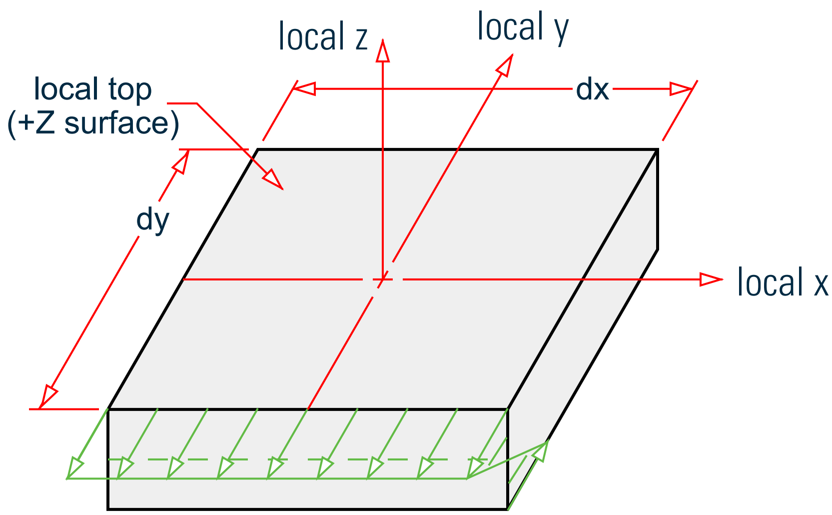


Figure 232: Stress caused by My

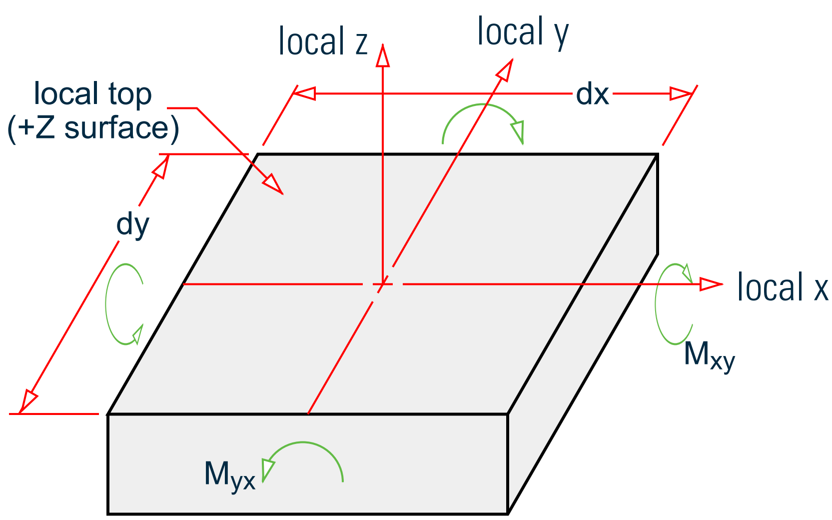


Figure 233: Torsion

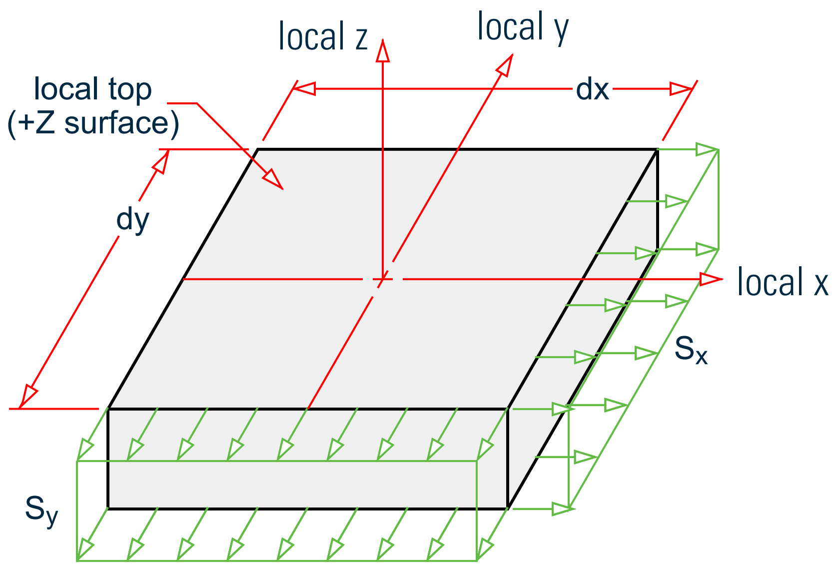


Figure 234: Membrane stress Sx and Sy

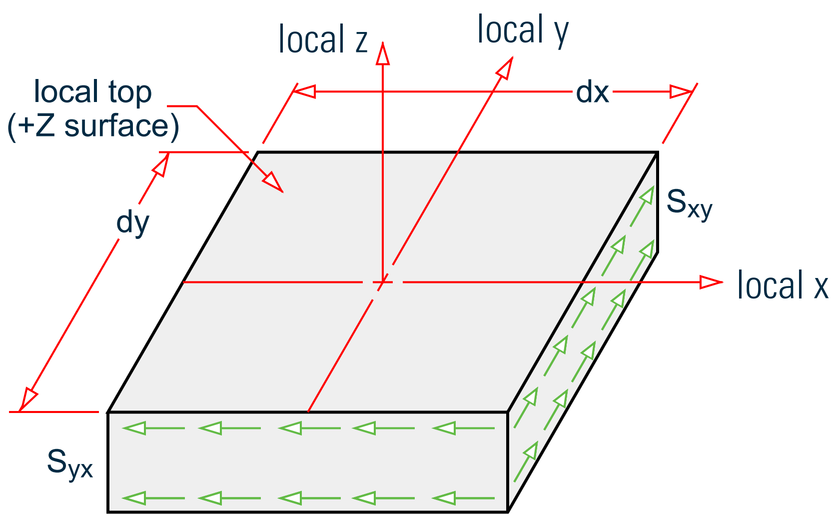


Figure 235: In-plane shear stresses Sxy and Syx

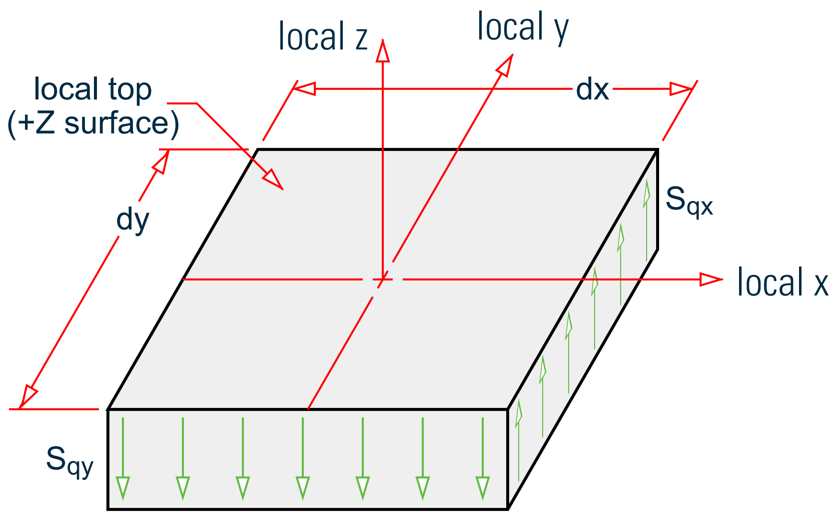


Figure 236: Out-of-plane shear stresses SQX and SQY

Members, plate elements, solid elements and surface elements can all be part of a single STAAD model. The `MEMBER INCIDENCES` input must precede the `INCIDENCE` input for plates, solids or surfaces. All `INCIDENCE`s must precede other input such as properties, constants, releases, loads, etc. The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

Plate Element Numbering

Therefore, to save some computing time, similar elements should be numbered sequentially. The following figure shows examples of efficient and non-efficient element numbering.

However, you have to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

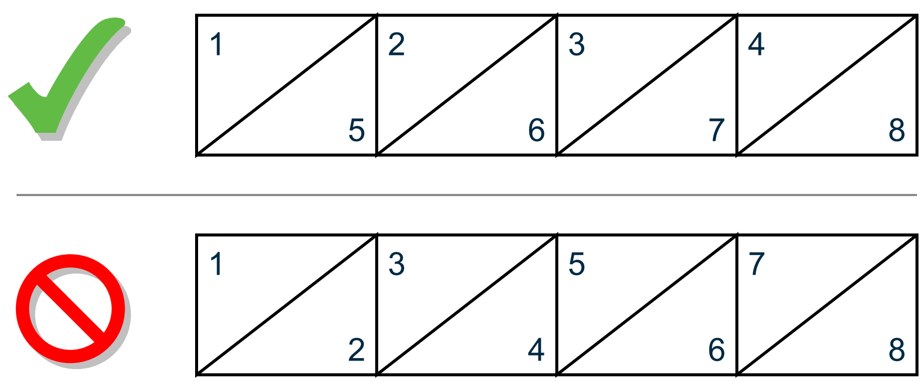


Figure 237: Examples of efficient and inefficient element numbering

G.5.2 Solid Elements

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams or soil and rock strata where finite element analysis using solid elements provides a powerful tool.

Theoretical Basis

The solid element used in STAAD.Pro is of eight-noded, isoparametric type. These elements have three translational degrees-of-freedom per node.

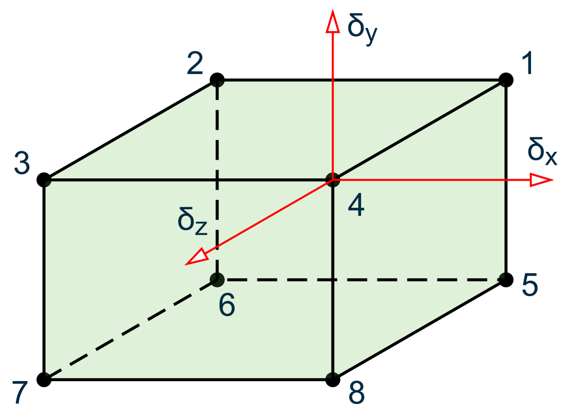


Figure 238: eight-noded, isoparametric solid element

By collapsing various nodes together, an eight noded solid element can be degenerated to the following forms with four to seven nodes. Joints 1, 2, and 3 must be retained as a triangle.

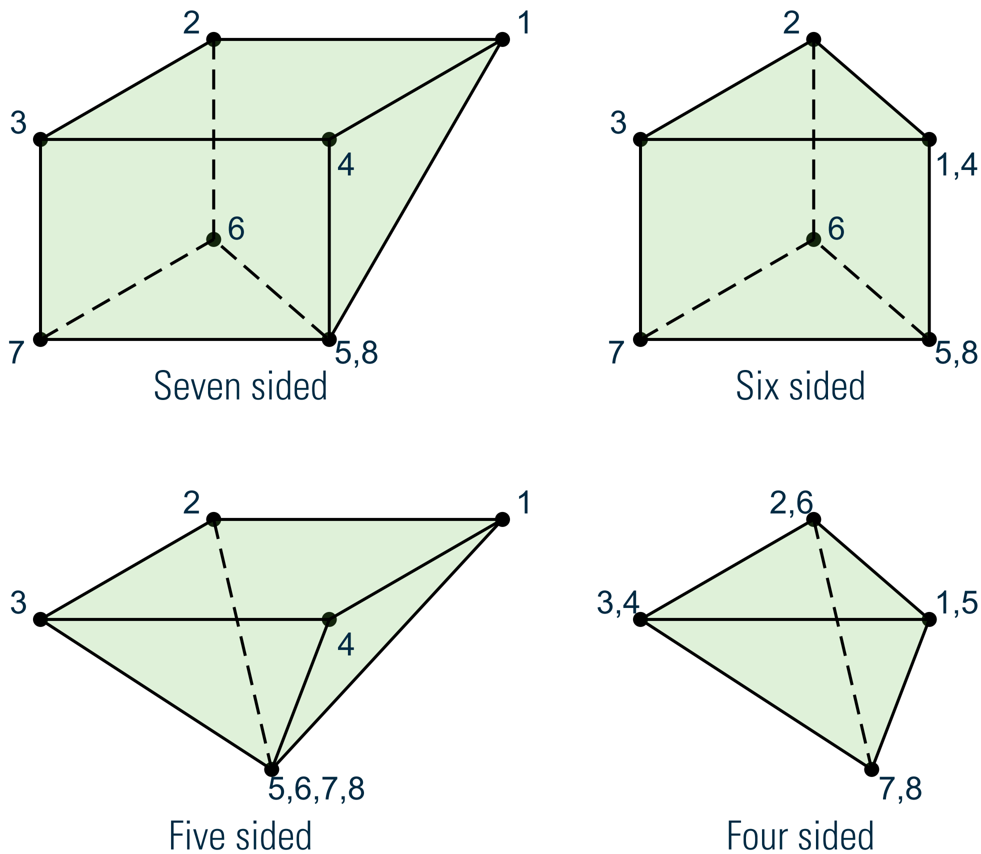


Figure 239: Forms of a collapsed eight-noded solid element

The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points (2x2x2). The integration is the reduced second order to prevent shear locking. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using the natural coordinate system, (r,s,t) of the element with its origin at the “center.” The interpolating functions are shown below:

$$X = \sum_{i=1}^{8} h_i x_i \text{ , } y = \sum_{i=1}^{8} h_i y_i \text{ , } z = \sum_{i=1}^{8} h_i z_i$$

where

x, y and z	=	the coordinates of any point in the element
xi, yi, zi	=	are the coordinates of nodes defined in the global coordinate system, i=1,…,8

The interpolation functions, hi are defined in the natural coordinate system, (r,s,t). Each of r,s and t varies between -1 and +1. The fundamental property of the unknown interpolation functions hi is that their values in natural coordinate system is unity at node, i, and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given below:

$$u = \sum_{i=1}^{8} h_i u_i \text{ , } v = \sum_{i=1}^{8} h_i v_i \text{ , } w = \sum_{i=1}^{8} h_i w_i$$

where u, v and w are displacements at any point in the element and u_i,v_i, w_i, i=1,8 are corresponding nodal displacements in the coordinate system used to describe the geometry.

Three additional displacement “bubble” functions which have zero displacements at the surfaces are added in each direction for improved shear performance to form a 33x33 matrix. A modified integration is used for the bubble functions to make the results invariant with respect to element orientation (a one-point integration at the center is used).

Static condensation is used to reduce this matrix to a 24x24 matrix at the corner joints.

Local Coordinate System

The local coordinate system used in solid element is the same as the global system.

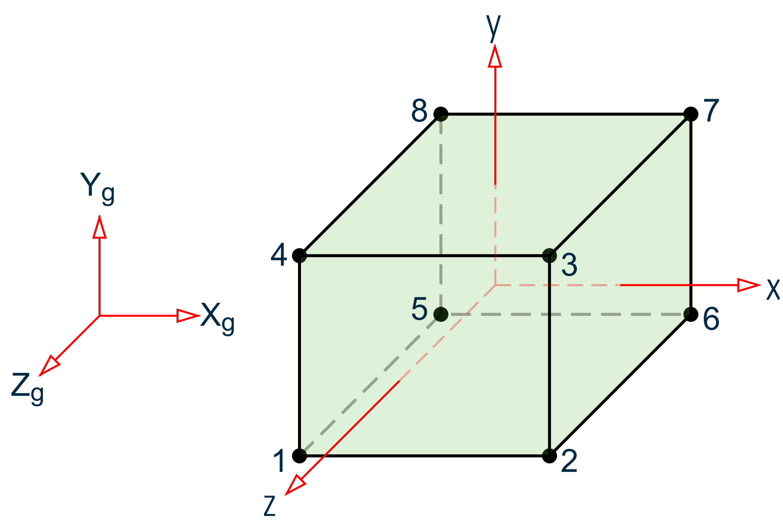


Figure 240: Local coordinate system for a solid element

Properties and Constants

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson’s ratio are to be specified. Also, density needs to be provided if selfweight is included in any load case.

Output of Element Stresses

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are :

- Normal Stresses : SXX, SYY and SZZ
- Shear Stresses : SXY, SYZ and SZX
- Principal stresses : S1, S2 and S3
- Von Mises stresses: $SIGE = 0.707\sqrt{(S1 - S2)^2 + (S2 - S3)^2 + (S3 - S1)^2}$

Direction cosines : six direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

Related Links

- *TR.13.2 Solid Element Incidences Specification* (on page 2225)

- *M. To draw a solid connecting existing nodes* (on page 681)
- *M. To check for negative volume solids* (on page 919)
- *TR.32.3.2 Element Load Specification - Solids* (on page 2472)
- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.13 Plate and Solid Elements* (on page 2224)
- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2224)
- *TR.13.2 Solid Element Incidences Specification* (on page 2225)
- *TR.14 Plate Element Mesh Generation* (on page 2227)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.21 Element/Surface Property Specification* (on page 2284)
- *TR.21.1 Element Property Specification* (on page 2284)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *TR.32.3 Element Load Specifications* (on page 2468)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)

G.5.3 Surface Elements (Deprecated)

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)
- *TR.13 Plate and Solid Elements* (on page 2224)
- *TR.13.1 Plate and Shell Element Incidence Specification* (on page 2224)
- *TR.13.2 Solid Element Incidences Specification* (on page 2225)
- *TR.14 Plate Element Mesh Generation* (on page 2227)
- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.21 Element/Surface Property Specification* (on page 2284)
- *TR.21.1 Element Property Specification* (on page 2284)
- *TR.24 Element Plane Stress and Ignore Inplane Rotation Specification* (on page 2295)
- *TR.32.3 Element Load Specifications* (on page 2468)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)

G.6 Member Properties

The following types of member property specifications are available in STAAD.Pro.

- G.6.1 Prismatic Properties (on page 2114) - four basic geometric shapes where the principal section properties are calculated from shapes dimensions or defined.
- G.6.2 Built-In Steel Section Libraries (on page 2116) - the principal dimensions and section properties from obtained from published documents and supplied in STAAD.Pro as standard databases
- G.6.3 User-Provided Steel Table (on page 2117) - the principal dimensions and some section properties for 10 different shape types which can be included in the model or located in an external file
- G.6.4 Tapered Sections (on page 2117) - either tapered I shaped or tapered tube sections defined by the dimensions at either end of the member for which the properties are calculated

- G.6.5 Assign Command (on page 2117) - a standard section profile from one of the standard database is used
- G.6.6 Steel Joist and Joist Girders (on page 2118) - standard joist or joist girders from publications
- G.6.7 Composite Beams and Composite Decks (on page 2119) - the section properties for standard steel profiles that have the additional effect of increased stiffness due to a positive connection to a concrete slab
- G.6.8 Curved Members (on page 2120) - the stiffness is calculated and modified based on the curvature of the member between the start and end nodes

Shear Area

Shear Area for members refers to the shear stiffness effective area. Shear stiffness effective area is used to calculate shear stiffness for the member stiffness matrix.

As an example: for a rectangular cross section, the shear stiffness effective area is usually taken as 0.83 (Roark) to 0.85 (Cowper) times the cross sectional area. A shear area of less than the cross sectional area will reduce the stiffness. A typical shearing stiffness term is

$$(12EI/L^3)/(1+\Phi)$$

where

Φ	=	$(12 EI) / (GA_s L^2)$
A_s	=	the shear stiffness effective area

Phi (Φ) is usually ignored in basic beam theory. STAAD will include the PHI term unless the `SET SHEAR` command is entered.

Shear stress effective area is a different quantity that is used to calculate shear stress and in code checking. For a rectangular cross section, the shear stress effective area is usually taken as two-thirds (0.67x) of the cross sectional area.

Shear stress in STAAD may be from one of three methods.

1. (Shear Force)/(Shear stress effective area)

   This is the case where STAAD computes the area based on the cross section parameters.
2. (Shear Force)/(Shear stiffness effective area)

   This is the case where STAAD uses the shear area entered.
3. *(V Q)/(I t)*

   In some codes and for some cross sections, STAAD uses this method.

The values that STAAD uses for shear area for shear deformation calculation can be obtained by specifying the command `PRINT MEMBER PROPERTIES`.

The output for this will provide this information in all circumstances: when AY and AZ are not provided, when AY and AZ are set to zero, when AY and AZ are set to very large numbers, when properties are specified using `PRISMATIC`, when properties are specified through a user table, when properties are specified through from the built-in-table, etc.

Related Links

- *TR.20 Member Property Specification* (on page 2255)

G.6.1 Prismatic Properties

The following prismatic properties are required for analysis:

AX = Cross sectional area
IX = Torsional constant
IY = Moment of inertia about y-axis.
IZ = Moment of inertia about z-axis.

In addition, you may choose to specify the following properties:

AY = Effective shear area for shear force parallel to local y-axis.
AZ = Effective shear area for shear force parallel to local z-axis.
YD = Depth of section parallel to local y-axis.
ZD = Depth of section parallel to local z-axis.

(when the global vertical axis is Y)

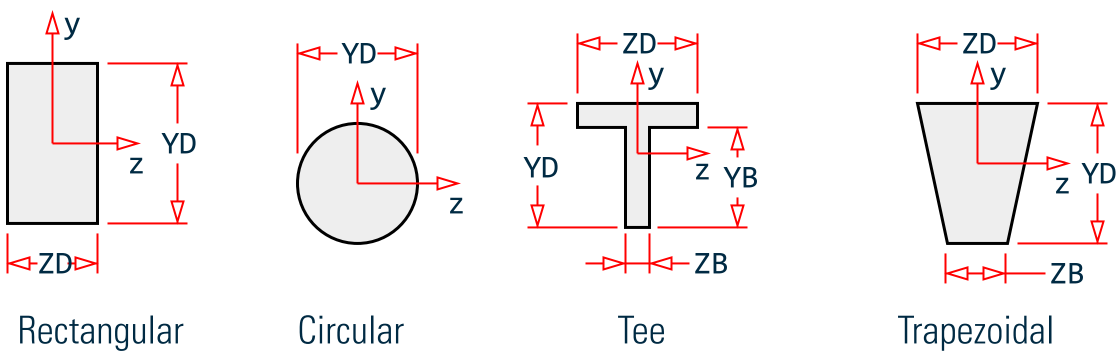


Figure 241: Prismatic property nomenclature

For T-beams, YD, ZD, YB & ZB must be specified. These terms are:

YD = Total depth of section (top fiber of flange to bottom fiber of web)
ZD = Width of flange
YB = Depth of stem
ZB = Width of stem

For Trapezoidal beams, YD, ZD & ZB must be specified. These terms, which too are shown in the next figure are:

YD = Total depth of section
ZD = Width of section at top fiber
ZB = Width of section at bottom fiber

The top & bottom are defined as positive side of the local Z axis, and negative side of the local Z axis respectively.

STAAD.Pro automatically considers the additional deflection of members due to pure shear (in addition to deflection due to ordinary bending theory). To ignore the shear deflection, enter a `SET SHEAR` command before the joint coordinates. This will bring results close to textbook results.

The depths in the two major directions (YD and ZD) are used in the program to calculate the section moduli. These are needed only to calculate member stresses or to perform concrete design. You can omit the YD & ZD values if stresses or design of these members are of no interest. The default value is 253.75 mm (9.99 inches) for YD and ZD. All the prismatic properties are input in the local member coordinates.

To define a concrete member,you must not provide AX, but instead, provide YD and ZD for a rectangular section and just YD for a circular section. If no moment of inertia or shear areas are provided, the program will automatically calculate these from YD and ZD.

The following table is provided to assist you in specifying the necessary section values. It lists, by structural type, the required section properties for any analysis.

Note: For the `PLANE` or `FLOOR` type analyses, the choice of the required moment of inertia depends upon the beta angle. If `BETA` equals zero, the required property is IZ.

Table 225: Required Section Properties

Structure Type	Required Properties
TRUSS	AX
PLANE	AX, IZ, or IY
FLOOR	IX, IZ or IY
SPACE	AX, IX, IY, IZ

Related Links

- *Property dialog* (on page 2794)
- *TR.20.2 Prismatic Property Specification* (on page 2263)
- *TR.20.2.1 Prismatic Tapered Tube Property Specification* (on page 2264)
- *M. To assign a prismatic section* (on page 728)
- *TR.20.2 Prismatic Property Specification* (on page 2263)

G.6.2 Built-In Steel Section Libraries

This feature of the program allows you to specify section names of standard steel shapes manufactured in different countries.

Since the shear areas of the sections are built into the tables, shear deformation is always considered for these sections.

Related Links

- *Properties - Whole Strucutre dialog* (on page 2793)
- *TR.20 Member Property Specification* (on page 2255)
- *Section Profile Tables dialog* (on page 2793)
- *TR.20.1 Assigning Properties from Steel Tables* (on page 2257)
- *M. To add a new table section property* (on page 715)
- *D1.A.4 Built-in Steel Section Library* (on page 1088)
- *TR.20.1 Assigning Properties from Steel Tables* (on page 2257)

G.6.3 User-Provided Steel Table

You can provide a customized steel table with designated names and proper corresponding properties. The program can then find member properties from those tables. Member selection may also be performed with the program selecting members from the provided tables only.

These tables can be provided as a part of a STAAD.Pro input or as separately created files from which the program can read the properties. If you do not use standard rolled shapes or only use a limited number of specific shapes, you can create permanent member property files. Analysis and design can be limited to the sections in these files.

Related Links

- *TR.19 User Steel Table Specification* (on page 2241)
- *TR.20.4 Property Specification from User Provided Table* (on page 2267)
- *User Table Manager dialog* (on page 2798)
- *User Provided Table dialog* (on page 2798)
- *M. To create a general section* (on page 743)
- *TR.19 User Steel Table Specification* (on page 2241)
- *TR.20.4 Property Specification from User Provided Table* (on page 2267)

G.6.4 Tapered Sections

Properties of tapered I-sections may be provided through `MEMBER PROPERTY` specifications. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis.

Tapered I-sections have constant flange dimensions and a linearly varying web depth along the length of the member.

Related Links

- *TR.20.3 Tapered Member Specification* (on page 2266)
- *Property: Tapered I dialog* (on page 2797)
- *M. To assign a tapered I section* (on page 729)
- *TR.20.3 Tapered Member Specification* (on page 2266)

G.6.5 Assign Command

If you want to avoid the trouble of defining a specific section name but rather leave it to the program to assign a section name, the `ASSIGN` command may be used. The section types that may be assigned include `BEAM`, `COLUMN`, `CHANNEL`, `ANGLE` and `DOUBLE ANGLE`.

When the keyword `BEAM` is specified, the program will assign an I-shaped beam section (Wide Flange for AISC, UB section for British).

For the keyword `COLUMN` also, the program will assign an I-shaped beam section (Wide Flange for AISC, UC section for British).

If steel design-member selection is requested, a similar type section will be selected.

Related Links

- *TR.20.5 Assign Profile Specification* (on page 2268)

G.6.6 Steel Joist and Joist Girders

STAAD.Pro includes facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called *Standard Specifications, Load Tables, and Weight Tables for Steel Joists and Joist Girders*, Fortieth edition. The following are the salient features of the implementation.

Steel joists are prefabricated, welded steel trusses used at closely spaced intervals to support floor or roof decking. Thus, from an analysis standpoint, a joist is not a single member in the same sense as beams and columns of portal frames that one is familiar with. Instead, it is a truss assembly of members. In general, individual manufacturers of the joists decide on the cross section details of the members used for the top and bottom chords, and webs of the joists. So, joist tables rarely contain any information on the cross-section properties of the individual components of a joist girder. The manufacturer's responsibility is to guarantee that, no matter what the cross section details of the members are, the joist simply has to ensure that it provides the capacity corresponding to its rating.

The absence of the section details makes it difficult to incorporate the true truss configuration of the joist in the analysis model of the overall structure. In STAAD, selfweight and any other member load applied on the joist is transferred to its end nodes through simply supported action. Also, in STAAD, the joist makes no contribution to the stiffness of the overall structure.

As a result of the above assumption, the following points must be noted with respect to modeling joists:

1. The entire joist is represented in the STAAD input file by a single member. Graphically it will be drawn using a single line.
2. After creating the member, the properties should be assigned from the joist database.
3. The 3D Rendering feature of the program will display those members using a representative Warren type truss.
4. The intermediate span-point displacements of the joist cannot be determined.

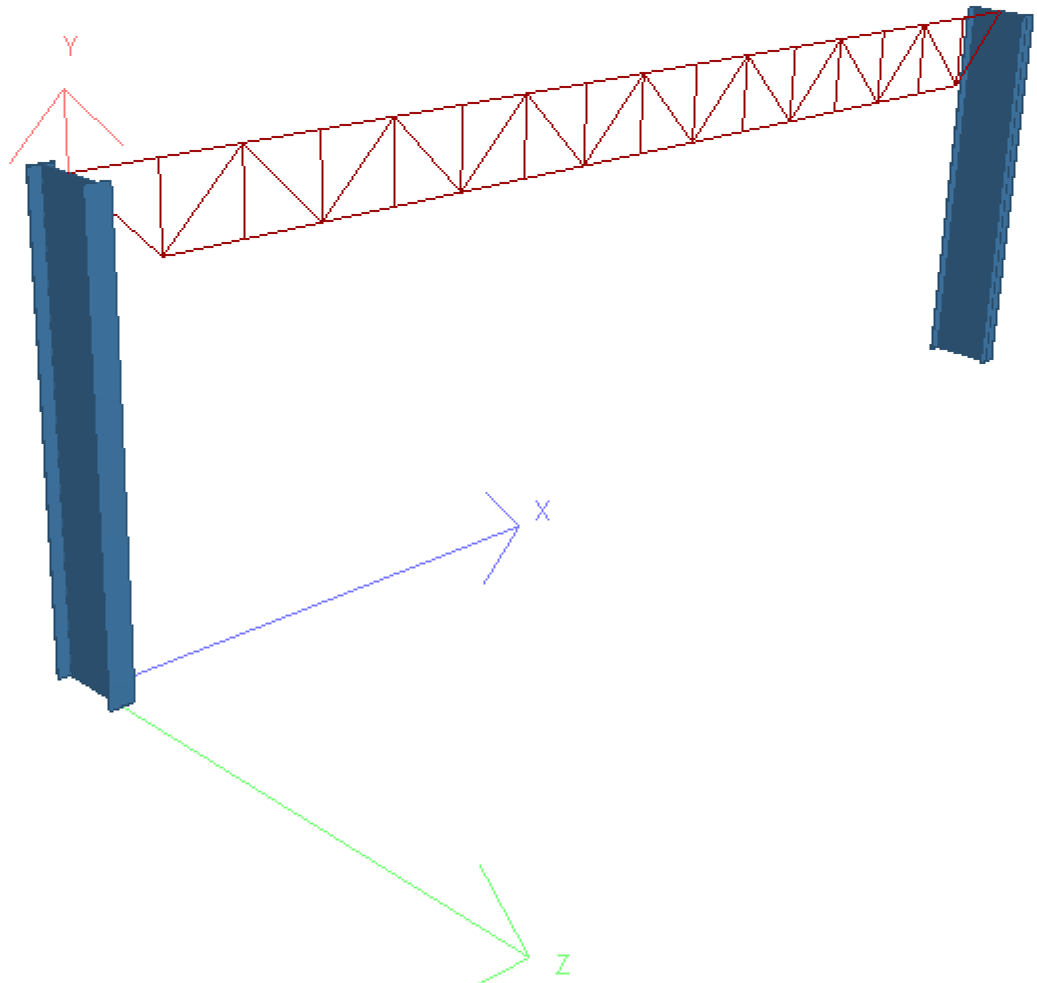


Figure 243: Example rendering of a joist member in STAAD.Pro

Related Links

- *M. To add an American steel joist section* (on page 716)

G.6.7 Composite Beams and Composite Decks

There are two methods in STAAD.Pro for specifying composite beams. Composite beams are members whose property is comprised of a rolled steel I-shape cross section (like an American W shape) with a concrete slab on top. The steel section and concrete slab act monolithically. The two methods are:

1. TR.20.1 Assigning Properties from Steel Tables (on page 2257) – In this method, the member geometry is first defined as a line. It is then assigned a property from the steel database, with the help of the "CM" attribute. Additional parameters like CT (thickness of the slab), FC (concrete strength), CW (effective width of slab), CD (concrete density), etc., some optional and some mandatory, are also provided.

   Hence, the responsibility of determining the attributes of the composite member, like concrete slab width, lies upon you, the user. If you wish to obtain a design, additional terms like rib height, rib width, etc. must also be separately assigned with the aid of design parameters. Hence, some amount of effort is involved in gathering all the data and assigning them.

2. TR.20.7 Composite Decks (on page 2270) – The laboriousness of the previous procedure can be alleviated to some extent by using the program's composite deck definition facilities. The program then internally converts the deck into individual composite members (calculating attributes like effective width in the process) during the analysis and design phase. The deck is defined best using the graphical tools of the program since a database of deck data from different manufacturers is accessible from easy-to-use dialogs. Since all the members which make up the deck are identified as part of a single object, load assignment and alterations to the deck can be done to just the deck object, and not the individual members of the deck.

Related Links

- *Composite Deck dialog* (on page 2741)
- *TR.20.7 Composite Decks* (on page 2270)
- *M. To add a floor load or one-way load* (on page 840)
- *TR.20.1 Assigning Properties from Steel Tables* (on page 2257)
- *TR.20.7 Composite Decks* (on page 2270)

G.6.8 Curved Members

Members can be defined as being curved.

Tapered sections are not permitted. The cross-section should be uniform throughout the length.

The design of curved members is not supported.

Related Links

- *TR.20.8 Curved Member Specification* (on page 2273)
- *M. To add a curved beam* (on page 660)
- *TR.20.8 Curved Member Specification* (on page 2273)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

G.7 Member and Element Release

STAAD.Pro allows releases for members and plate elements.

One or both ends of a member or element can be released in any combination of the six degrees of freedom. Members and elements are assumed to be rigidly framed into joints in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the member can be set to zero with member release statements. By specifying release components, individual degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each member. Partial moment releases are also allowed.

The translational degrees of freedom are denoted by u_1, u_2, u_3 and the rotational degrees of freedom are denoted by u_4, u_5 & u_6.

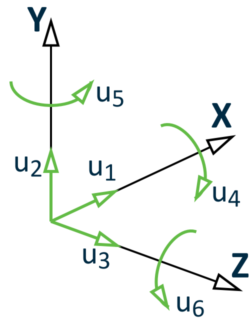


Figure 244: Degrees of Freedom

Note: A specific member can have *either* end releases (see G.7 Member and Element Release (on page 2120)) *or* axial-only specifications (see G.8 Axial-Only Specifications (on page 2121)) properties assigned. The last specification for a given member will be used. In other words, a `MEMBER RELEASE` should not be applied on a member which is declared `TRUSS`, `TENSION ONLY`, `COMPRESSION ONLY`, or `CABLE`.

Related Links

- *Plate Specs dialog* (on page 2791)
- *TR.22.2 Element Release Specification* (on page 2287)
- *M. To assign plate corner release* (on page 814)
- *TR.22.1 Member Release Specification* (on page 2286)
- *Member Specification dialog* (on page 2786)
- *M. To assign member end release* (on page 804)
- *TR.22 Member and Element Releases* (on page 2285)
- *TR.22.1 Member Release Specification* (on page 2286)
- *TR.22.2 Element Release Specification* (on page 2287)
- *TR.22.3 Element Ignore Stiffness* (on page 2288)

G.8 Axial-Only Specifications

STAAD.Pro has several member specifications which allow you to designate axial-only behavior for members.

Note: A specific member can have *either* end releases (see G.7 Member and Element Release (on page 2120)) *or* axial-only specifications (see G.8 Axial-Only Specifications (on page 2121)) properties assigned. The last specification for a given member will be used. In other words, a `MEMBER RELEASE` should not be applied on a member which is declared `TRUSS`, `TENSION ONLY`, `COMPRESSION ONLY`, or `CABLE`.

G.8.1 Truss and Tension- or Compression-Only Members

For analyses which involve members that carry axial loads only (i.e., truss members) there are two methods for specifying this condition. When all the members in the structure are truss members, the type of structure is declared as `TRUSS` whereas, when only some of the members are truss members (e.g., bracing members in a building), the `MEMBER TRUSS` command can be used where those members will be identified separately.

In STAAD.Pro, the `MEMBER TENSION` or `MEMBER COMPRESSION` command can be used to limit the axial load type the member may carry.

Related Links

- *TR.23.1 Member Truss Specification* (on page 2289)
- *TR.23.3 Member Tension/Compression Specification* (on page 2292)
- *Member Specification dialog* (on page 2786)
- *M. To assign axial action members* (on page 803)
- *TR.23.1 Member Truss Specification* (on page 2289)
- *TR.23.3 Member Tension/Compression Specification* (on page 2292)

G.8.2 Cable Members

STAAD.Pro supports the following types of analysis for cable members:

G.8.2.1 Linearized Cable Members

Cable members may be specified by using the `MEMBER CABLE` command. While specifying cable members, the initial tension in the cable must be provided.

Cable Stiffness

The increase in length of a loaded cable is a combination of two effects. The first component is the elastic stretch, which is governed by the familiar spring relationship:

$$F = Kx$$

where

$$K_{elastic} = EA/L$$

The second component of the lengthening is due to a change in geometry (as a cable is pulled taut, sag is reduced). This relationship can also be described by $F = Kx$, but here:

$$K_{sag} = \frac{12T^3}{w^2 L^3 \left(\frac{1}{\cos^2 \alpha}\right)}$$

where

w	=	weight per unit length of cable
T	=	tension in cable
α	=	angle that the axis of the cable makes with a horizontal plane (= 0, cable is horizontal; = 90, cable is vertical)

Therefore, the "stiffness" of a cable depends on the initial installed tension (or sag). These two effects may be combined as follows:

$$K_{comb} = \frac{1}{\left(1/K_{sag} + 1/K_{elastic}\right)} = \frac{(EA/L)}{\left(1 + \frac{w^2 L^2 EA \left(\cos^2 \alpha\right)}{12T^3}\right)}$$

Note: As $T \rightarrow \infty$ (infinity), $K_{comb} \rightarrow EA/L$ and that when $T = 0$, $K_{comb} = 0$. It should also be noted that as the tension increases (sag decreases) the combined stiffness approaches that of the pure elastic situation.

Notes

The following points need to be considered when using the cable member in STAAD.Pro:

1. The linear cable member is only a truss member whose properties accommodate the sag factor and initial tension. The behavior of the cable member is identical to that of the truss member. That is, it can carry axial loads only. As a result, the fundamental rules involved in modeling G.8.1 Truss and Tension- or Compression-Only Members (on page 2121) have to be followed when modeling cable members.

   For example, when two cable members meet at a common joint, if there is not a support or a 3rd member connected to that joint, it is a point of potential instability.
2. Due to the reasons specified in 1) above, applying a transverse load on a cable member is not advisable. The load will be converted to two concentrated loads at the two ends of the cable and the true deflection pattern of the cable will never be realized.
3. A tension only cable member offers no resistance to a compressive force applied at its ends. When the end joints of the member are subjected to a compressive force, they “give in” thereby causing the cable to sag. Under these circumstances, the cable member has zero stiffness and this situation has to be accounted for in the stiffness matrix and the displacements have to be recalculated. But in STAAD.Pro, merely declaring the member to be a cable member does not guarantee that this behavior will be accounted for.

   Important: It is also important that you declare the member to be a tension only member by using the `MEMBER TENSION` command, after the `CABLE` command.

   This will ensure that the program will test the nature of the force in the member after the analysis and if it is compressive, the member is switched off and the stiffness matrix re-calculated.
4. Due to potential instability problems explained in item 1) above, you should also avoid modeling a catenary by breaking it down into a number of straight line segments. The cable member in STAAD.Pro cannot be used to simulate the behavior of a catenary.

   By catenary, we are referring to those structural components which have a curved profile and develop axial forces due their self weight. This behavior is in reality a nonlinear behavior where the axial force is caused because of either a change in the profile of the member or induced by large displacements, neither of which are valid assumptions in an elastic analysis. A typical example of a catenary is the main U shaped cable used in suspension bridges.
5. The increase in stiffness of the cable as the tension in it increases under applied loading is updated after each iteration if the cable members are also declared to be `MEMBER TENSION`. However, iteration stops when all tension members are in tension or slack; not when the cable tension converges.

Related Links

- *TR.23.2 Member Cable Specification* (on page 2290)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)
- *TR.23.1 Member Truss Specification* (on page 2289)

G.8.2.2 Nonlinear Cable and Truss Members

Cable members for the Basic Nonlinear Cable Analysis may be specified by using the `MEMBER CABLE` command.

While specifying cable members, the initial tension in the cable or the unstressed length of the cable must be provided. You should ensure that all cables will be in sufficient tension for all load cases to converge. Use selfweight in every load case and temperature if appropriate; i.e., don’t enter component cases (e.g., wind only).

The nonlinear cable may have large motions and the sag is checked on every load step and every equilibrium iteration.

In addition there is a nonlinear truss which is specified in the Member Truss command. The nonlinear truss is simply any truss with pretension specified. It is essentially the same as a cable without sag. This member takes compression. If all cables are taut for all load cases, then the nonlinear truss may be used to simulate cables. The reason for using this substitution is that the truss solution is more reliable.

Notes

Nonlinear cable elements differ from linearized cable members in the following ways:

Notes 1, 2, and 4 G.8.2.1 Linearized Cable Members (on page 2122) will *not* apply to nonlinear cable analysis if sufficient pretension is applied, so joints may be entered along the shape of a cable (in some cases a stabilizing stiffness may be required and entered for the first loadstep). G.8.2.1 Linearized Cable Members (on page 2122): The Member Tension command is unnecessary and ignored for the nonlinear cable analysis. G.8.2.1 Linearized Cable Members (on page 2122): The cable tensions are iterated to convergence in the nonlinear cable analysis.

Related Links

- *TR.23.2 Member Cable Specification* (on page 2290)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)
- *TR.23.1 Member Truss Specification* (on page 2289)

G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis

Cable members used for a Advanced Cable Analysis are specified by using the `MEMBER CABLE` command.

Note: This analysis feature can be used only when Advanced Analysis license is active.

When specifying cable members, either the initial tension in the cable or the unstressed length of the cable must first be provided. The initial tension is sufficient for keeping the cable in tension because the catenary theory behind the element formulation will prevent the cable from behaving in compression. However, a larger initial tension (or smaller unstressed length) increases the numerical stability.

A nonlinear cable may have large deformation. The force equilibrium at cable ends is checked on every load step and on every equilibrium iteration.

Notes

Following considerations exist in Advanced Cable Analysis:

1. The nonlinear cable element provides stiffness and resistance forces to only three translational degrees (i.e., FX, FY, and FZ). It is not able to carry any moments. So, when two nonlinear cable elements meet at a common joint –and if there is not a support or a 3rd member connected to the joint– it is a point of potential instability. The program introduces a very weak spring to overcome this problem.
2. Due to the reason as described in item 1), applying any moment to a cable element is not advisable.
3. The cable is not able to carry any compression forces. So, when the load tends to cause compression in a cable –and if there are no other members connected to the cable and the cable can deform freely– there will be some numerical instability.
4. Due to the instability problem explained in item 1), subdividing a cable member into several smaller cable elements should be done *only* when it is necessary.

   An example of one such necessary case may be when there is force load applied in the middle of the cable member. For this case, the cable member has to be broken so that the forces can be applied as joint loads at cable nodes.

5. The increase in stiffness of the cable (by increasing the initial tension or decreasing unstressed length) is always beneficial for numerical stability. The iteration will stop only when the force equilibrium is reached.
6. In advanced cable analysis, the self weight of a cable member is initially considered to obtain the initial configuration of cable members under self weight. Any additional weight required along with self weight can be included using `FWY` parameter in the `MEMBER CABLE` command.

Related Links

- *TR.23.2 Member Cable Specification* (on page 2290)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)

G.9 Connection Tags

Connection tags provide a means for assigning both connection type data and connection capacities, which can be used for checking in STAAD.Pro.

Note: This feature is available in STAAD.Pro V8i (SELECTseries 4), release 20.07.09.21 and higher.

Connection tags consist of two pieces of data:

i. A Connection Tags XML file, which contains the connection categories, tag names, and member end releases for the connection tag. Connection capacities are also specified for each combination of member and connecting member which may utilize a connection tag. Refer to " D. Connection Tags XML File Schema (on page 1052)" in the STAAD User Interface help for additional information on the required structure of this XML file.
ii. Assignments of connection tags to members are stored in the STAAD input file. Though this is done within the `DEFINE MEMBER ATTRIBUTE` command, it is strongly recommended that the user interface features be used to make connection tag assignments as these must utilize only the connection categories and tag names in the associated XML file. See TR.29.2 Connection Tag Member Attribute (on page 2341) for additional information on this command.

Related Links

- *TR.29.2 Connection Tag Member Attribute* (on page 2341)
- *D. Connection Tags sub menu* (on page 1066)
- *TR.29.2 Connection Tag Member Attribute* (on page 2341)

G.11 Member and Plate Offsets

Some members or elements in a structure may not be concurrent with the incident joints thereby creating offsets. This offset distance is specified in terms of global or local coordinate system (i.e., X, Y, and Z distances from the incident joint).

Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual member or element forces.

In the case of members, the new offset centroid of the member can be at the start or end incidences and the new working point will also be the new start or end of the member. Therefore, any reference from the start or end of that member will always be from the new offset points.

Related Links

- *TR.25.2 Element Offset Specification* (on page 2298)
- *Plate Specs dialog* (on page 2791)

- *To assign plate offsets* (on page 813)
- *TR.25.1 Member Offset Specification* (on page 2296)
- *Member Specification dialog* (on page 2786)
- *M. To assign member end offsets* (on page 805)
- *TR.25.1 Member Offset Specification* (on page 2296)

G.12 Material Properties

STAAD.Pro allows you to define the properties of materials by two different methods. By assigning individual constants to members, elements, or solids or by creating a material definition and then assigning this.

Material Constants

The material constants are:

modulus of elasticity (`E`),
weight density (`DEN`),
Poisson's ratio (`POISS`),
coefficient of thermal expansion (`ALPHA`),
composite damping ratio,
and beta angle (`BETA`) or coordinates for any reference (`REF`) point.

The modulus of elasticity, E, value for members must be provided or the analysis will not be performed. Weight density (`DEN`) is used only when selfweight of the structure is to be taken into account. Poisson's ratio (`POISS`) is used to calculate the shear modulus (commonly known as G) by the formula,

$$G = 0.5 \cdot E/(1 + POISS)$$

If Poisson's ratio is not provided, STAAD.Pro will assume a value for this quantity based on the value of E.

Note: Poisson's Ratio must always be defined *after* the Modulus of Elasticity for a given member/element.

The coefficient of thermal expansion (`ALPHA`) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and `ALPHA` has to be the same.

The composite damping ratio is used to compute the damping ratio for each mode in a dynamic solution. This is only useful if there are several materials with different damping ratios.

`BETA` angle and `REF`erence point are discussed in G.4.3 Relationship Between Global and Local Coordinates (on page 2092) and are input as part of the member constants.

Material Definitions

Alternately, you may define the constants of a material in a material definition. This can include the analytical and design properties for the material. These material definitions are then assigned to the members, elements, and solids.

Note: A `BETA` is a geometric property and must be assigned separately from the material definition.

Related Links

- *TR.26 Specifying and Assigning Material Constants* (on page 2303)

G.13 Supports

STAAD.Pro allows specifications of supports that are parallel as well as inclined to the global axes.

Supports are specified as `PINNED`, `FIXED`, or `FIXED` with different releases. A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement.

The restraints of a fixed support can also be released in any desired direction as specified.

Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction.

For static analysis, Multi-linear spring supports can be used to model the varying, non-linear resistance of a support (e.g., soil).

The Support command is also used to specify joints and directions where support displacements will be enforced.

Related Links

- *M. To assign a fixed or pinned support* (on page 819)
- *Create Support dialog* (on page 2815)
- *Supports - Whole Structure dialog* (on page 2814)
- *TR.27.1 Global Support Specification* (on page 2316)
- *M. To assign a foundation support* (on page 824)
- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2319)
- *Create Support dialog* (on page 2815)
- *M. To assign an enforced support* (on page 820)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)
- *M. To assign an inclined support* (on page 823)
- *TR.27.2 Inclined Support Specification* (on page 2318)
- *Create Support dialog* (on page 2815)
- *M. To assign a spring support* (on page 821)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)
- *TR.27 Support Specifications* (on page 2315)
- *TR.27.1 Global Support Specification* (on page 2316)
- *TR.27.2 Inclined Support Specification* (on page 2318)

G.13.1 Tension- and Compression- Only Springs

In STAAD.Pro, the `SPRING TENSION` or `SPRING COMPRESSION` command can be used to limit the load direction the support spring may carry. The analysis will be performed accordingly.

Related Links

- *TR.27.5 Spring Tension/Compression Specification* (on page 2324)
- *TR.23.1 Member Truss Specification* (on page 2289)

G.14 Rigid Diaphragms

STAAD.Pro provides the following methods of modeling rigid diaphragms for structures.

Control/Dependent Joints

The control/dependent option is provided to enable you to model rigid links in the structural system.

This facility can be used to model special structural elements like a rigid floor diaphragm. Several dependent joints may be provided which will be assigned same displacements as the control joint. You are also allowed the flexibility to choose the specific degrees of freedom for which the displacement constraints will be imposed on the dependent joints. If all degrees of freedom (Fx, Fy, Fz, Mx, My, and Mz) are provided as constraints, the joints will be assumed to be rigidly connected.

Floor Diaphragms

Alternately, you can define a floor diaphragm without the need to specify a control joint. You can specify a floor by a height range or by a joint list using this method. The program will internally calculate the center of mass of the floor and generate an analytical node at that location.

Related Links

- *TR.28.1 Control/Dependent Specification* (on page 2327)
- *Node Specification dialog* (on page 2784)
- *M. To assign a rigid link between nodes* (on page 817)
- *TR.28.1 Control/Dependent Specification* (on page 2327)

G.15 Loads

Loads in a structure can be directly specified for joints, members, and elements. STAAD.Pro can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

G.15.1 Joint Loads

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

Related Links

- *TR.32.1 Joint Load Specification* (on page 2462)
- *Nodal Load tab* (on page 2841)
- *M. To add a nodal load* (on page 832)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *TR.32.1 Joint Load Specification* (on page 2462)
- *TR.32.10 Dynamic Loading Specification* (on page 2498)

G.15.2 Member Load

Three types of member loads may be applied directly to a member of a structure: uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal).

Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads.

Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length. See G.4.1 Global Coordinate System (on page 2087) to find the relation of the member to the global coordinate systems for specifying member loads. Positive forces act in the positive coordinate directions, local or global, as the case may be.

Uniform moment may not be applied to tapered members. Only uniform load over the entire length is available for curved members.

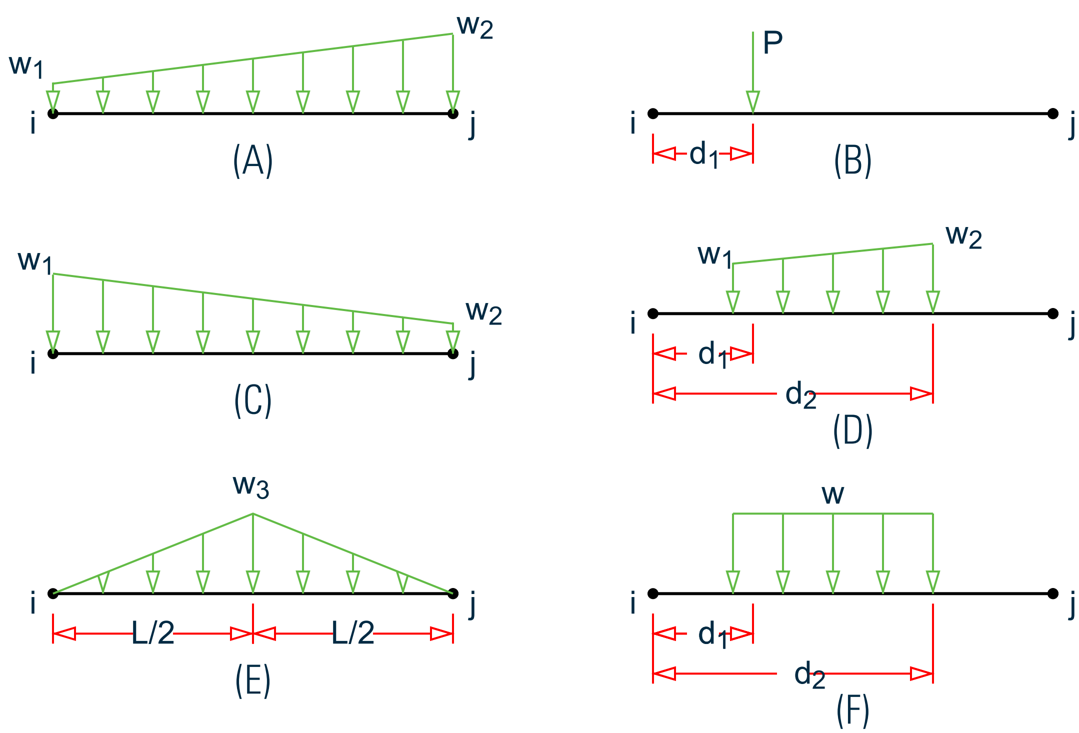


Figure 245: Member Load Configurations for A) linear loads, B) concentrated loads, C) linear loads, D) trapezoidal load, E) triangular (linear) loads, and E) uniform load

Related Links

- *TR.32.2 Member Load Specification* (on page 2464)
- *Member Load tab* (on page 2842)
- *M. To add a concentrated force or moment on members* (on page 834)
- *M. To add a uniform load to members* (on page 834)

- *M. To add a linear varying load to members* (on page 835)
- *M. To add a hydrostatic load to objects* (on page 842)
- *TR.32.2 Member Load Specification* (on page 2464)

G.15.3 Area, One-way, and Floor Loads

Often a floor is subjected to a uniform pressure. It could require a lot of work to calculate the equivalent member load for individual members in that floor. However, with the `AREA`, `ONEWAY` or `FLOOR LOAD` facilities, you can specify the pressure (load per unit square area). The program will calculate the tributary area for these members and calculate the appropriate member loads. The Area Load and Oneway load are used for one way distribution and the Floor Load is used for two way distribution.

The following assumptions are made while transferring the area/floor load to member load:

a. The member load is assumed to be a linearly varying load for which the start and the end values may be of different magnitude.

b. Tributary area of a member with an area load is calculated based on half the spacing to the nearest approximately parallel members on both sides. If the spacing is more than or equal to the length of the member, the area load will be ignored. Oneway load does not have this limitation.

c. These loading types should not be specified on members declared as `MEMBER CABLE`, `MEMBER TRUSS`, `MEMBER TENSION`, `MEMBER COMPRESSION`, or `CURVED`.

Note: Floor Loads and One-way Loads can be reduced when included in a load case defined as “Reducible” according to the UBC/IBC rules.

An example:

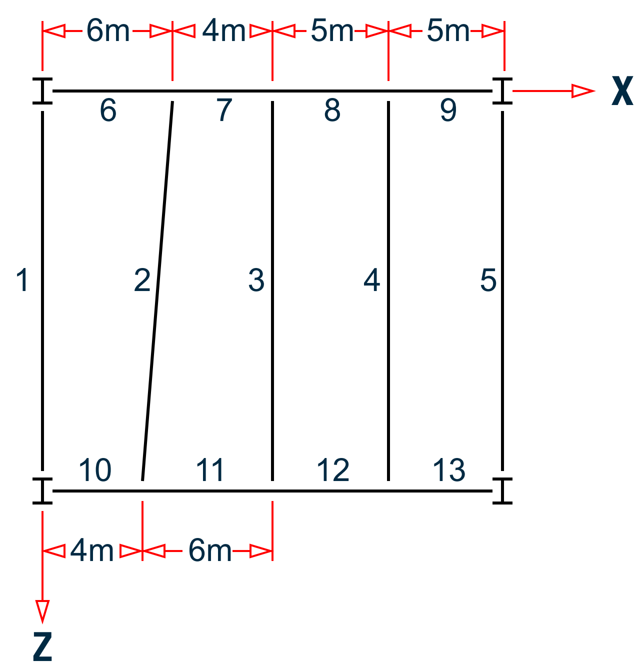


Figure 246: Example floor structure with area load specification of 0.1

Member 1 will have a linear load of 0.3 at one end and 0.2 at the other end. Members 2 and 4 will have a uniform load of 0.5 over the full length. Member 3 will have a linear load of 0.45 and 0.55 at respective ends. Member 5 will have a uniform load of 0.25. The rest of the members, 6 through 13, will have no contributory area load since the nearest parallel members are more than each of the member lengths apart. However, the reactions from the members to the girder will be considered.

Only member loads are generated from the Area, Oneway, and Floor load input. Thus, load types specific to plates, solids or surface are *not* generated. That is because, the basic assumption is that, a floor load or area load is used in situations where the basic entity (plate, solid or surface) which acts as the medium for application of that load, is not part of the structural model.

The Oneway load is intended to be used in areas with relatively large aspect rations between adjacent sides. It should not be used on members with square tributary areas unless the `TOWARDS` option is used, which then specifies to which of the two equal directions the load should be applied. Otherwise, the Floor Load type should be used.

Note: Failure to specify a `TOWARD` side on a Oneway load applied to a square tributary area will likely result in lost load or unintended load path changes.

Related Links

- *TR.32.4.1 Area Load Specification* (on page 2476)
- *Area Load tab* (on page 2845)
- *M. To add an area load* (on page 839)
- *Floor Load tab* (on page 2845)
- *TR.32.4.2 One-way Load Specification* (on page 2476)
- *TR.32.4.3 Floor Load Specification* (on page 2483)
- *M. To add a floor load or one-way load* (on page 840)
- *TR.32.4 Area, One-way, and Floor Load Specifications* (on page 2475)
- *TR.32.4.1 Area Load Specification* (on page 2476)
- *TR.32.4.2 One-way Load Specification* (on page 2476)
- *TR.32.4.3 Floor Load Specification* (on page 2483)

G.15.4 Fixed End Member Load

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

Related Links

- *TR.32.7 Fixed-End Load Specification* (on page 2494)
- *Member Load tab* (on page 2842)
- *M. To add fixed end member loads* (on page 837)
- *TR.32.7 Fixed-End Load Specification* (on page 2494)

G.15.5 Prestress and Poststress Member Load

Members in a structure may be subjected to prestress load for which the load distribution in the structure may be investigated. The prestressing load in a member may be applied axially or eccentrically. The eccentricities can be provided at the start joint, at the middle, and at the end joint. These eccentricities are only in the local y-axis.

A positive eccentricity will be in the positive local y-direction. Since eccentricities are only provided in the local y-axis, care should be taken when providing prismatic properties or in specifying the correct `BETA` angle when rotating the member coordinates, if necessary. Two types of prestress load specification are available; `PRESTRESS`, where the effects of the load are transmitted to the rest of the structure, and `POSTSTRESS`, where the effects of the load are experienced exclusively by the members on which it is applied.

1. The cable is assumed to have a generalized parabolic profile. The equation of the parabola is assumed to be

   $$y = ax^2 + bx + c$$

   where

a	=	$1/L^2 (2es - 4em + 2ee)$
b	=	$1/L (4em - ee - 3es)$
c	=	es
es	=	eccentricity of the cable at the start of the member (in local y-axis)
em	=	eccentricity of the cable at the middle of the member (in local y-axis)
ee	=	eccentricity of the cable at the end of the member (in local y-axis)
L	=	Length of the member

2. The angle of inclination of the cable with respect to the local x-axis (a straight line joining the start and end joints of the member) at the start and end points is small which gives rise to the assumption that

   $$\sin\theta = \theta = dy/dx$$

   Hence, if the axial force in the cable is P, the vertical component of the force at the ends is $P(dy/dx)$ and the horizontal component of the cable force is,

   $$P[1 - (dy/dx)^2]^{1/2}$$

   Users are advised to ensure that their cable profile meets this requirement. An angle under 5 degrees is recommended.
3. The member is analyzed for the prestressing / poststressing effects using the equivalent load method. This method is well documented in most reputed books on Analysis and Design of Prestressed concrete. The magnitude of the uniformly distributed load is calculated as

   $$udl = 8 \cdot Pe/L^2$$

   where

P	=	Axial force in the cable
e	=	$(es + ee)/2 - em$
L	=	Length of the member

4. The force in the cable is assumed to be same throughout the member length. No reduction is made in the cable forces to account for friction or other losses.
5. The term `MEMBER PRESTRESS` as used in STAAD.Pro signifies the following condition. The structure is constructed first. Then, the prestressing force is applied on the relevant members. As a result, the members deform and depending on their end conditions, forces are transmitted to other members in the structure. In other words, “`PRE`” refers to the time of placement of the member in the structure relative to the time of stressing.
6. The term `MEMBER POSTSTRESS` as used in STAAD.Pro signifies the following condition. The members on which such load is applied are first cast in the factory. Following this, the prestressing force is applied on them. Meanwhile, the rest of the structure is constructed at the construction site. Then, the prestressed members are brought and placed in position on the partially built structure. Due to this sequence, the effects of prestressing are “experienced” by only the prestressed members and not transmitted to the rest of the structure. In other words, “`POST`” refers to the time of placement of the member in the structure relative to the time of stressing.

7. As may be evident from Item (6) above, it is not possible to compute the displacements of the ends of the `POSTSTRESSED` members for the effects of poststressing, and hence are assumed to be zero. As a result, displacements of intermediate sections (See SECTION DISPLACEMENT command) are measured relative to the straight line joining the start and end joints of the members as defined by their initial `JOINT COORDINATES`.

Related Links

- *TR.32.5 Prestress Load Specification* (on page 2489)
- *Member Load tab* (on page 2842)
- *M. To add a prestress or post-tension load to members* (on page 836)
- *TR.32.5 Prestress Load Specification* (on page 2489)

G.15.6 Temperature and Strain Load

Uniform temperature difference throughout members and elements may be specified. Temperature differences across both faces of members and through the thickness of plates may also be specified (uniform temperature only for solids).. The program calculates the axial strain (elongation and shrinkage) due to the temperature difference for members. From this it calculates the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

Related Links

- *TR.32.6 Temperature Load Specification for Members, Plates, and Solids* (on page 2493)

G.15.7 Support Displacement Loads

Static Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rotational. Translational displacements are provided in the specified length while the rotational displacements are always in degrees. Note that displacements can be specified only in directions in which the support has an “enforced” specification in the Support command.

Related Links

- *TR.32.8 Support Joint Displacement Specification* (on page 2494)
- *Nodal Load tab* (on page 2841)
- *M. To add a support displacement* (on page 833)
- *TR.32.8 Support Joint Displacement Specification* (on page 2494)

G.15.8 Loading on Elements

On Plate/Shell elements, the types of loading that are permissible are:

1. Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over a small portion or over the entire surface of the element.
2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the plate element (causing only elongation / shortening) or may vary across the depth of a plate element causing bending on the plate element.. The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. The self-weight of the elements can be applied using the `SELFWEIGHT` loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

On Solid elements, the loading types available are:

1. The self-weight of the solid elements can be applied using the `SELFWEIGHT` loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.
2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
3. Temperature loads which may be constant throughout the solid elements (causing only elongation / shortening). The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
4. Pressure on the faces of solids.

Only translational stiffness is supported in solid elements. Thus, at joints where there are only solid elements, moments may not be applied. For efficiency, rotational supports should be used at these joints.

Related Links

- *Plate Loads tab* (on page 2847)
- *TR.32.3.1 Element Load Specification - Plates* (on page 2469)
- *M. To add pressure load on a plate* (on page 837)
- *M. To add a concentrated load on a plate* (on page 838)
- *TR.32.3 Element Load Specifications* (on page 2468)

G.16 Load Generator

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

For seismic loads, a static analysis method or a dynamic analysis method can be adopted. The static analysis method, which is the one referred to here, is based on codes such as UBC, IBC, AIJ, IS1893 etc. For dynamic analysis, see the sections in this chapter on response spectrum and time history analysis.

Input for the load generation facility consists of two parts:

1. Definition of the load system(s).
2. Generation of primary load cases using previously defined load system(s).

Related Links

- *TR.31 Definition of Load Systems* (on page 2345)

G.16.1 Moving Load Generator

This feature enables the user to generate static loads on members due to vehicles moving on a structure. Moving load systems consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis. American Association of State Highway and Transportation Officials (AASHTO) vehicles are available within the program and can be specified using standard AASHTO designations.

Related Links

- *Add New Vehicle Definitions dialog* (on page 2899)
- *TR.31.1 Definition of Moving Load System* (on page 2346)
- *M. To define a vehicle for loading* (on page 873)

- *Load Generation dialog* (on page 2866)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)
- *M. To generate moving load cases* (on page 874)
- *TR.31.1 Definition of Moving Load System* (on page 2346)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)

G.16.2 Seismic Load Generator

The STAAD.Pro seismic load generator follows the procedure of equivalent lateral load analysis explained in UBC, IBC and several other codes. It is assumed that the lateral loads will be exerted in X and Z (or X and Y if Z is up) directions (horizontal) and Y (or Z if Z is up) will be the direction of the gravity loads. Thus, for a building model, Y (or Z if Z is up) axis will be perpendicular to the floors and point upward (all Y (or Z if Z is up) joint coordinates positive). The user is required to set up his model accordingly. Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate equation from the code. IBC 2003, IBC 2000, UBC 1997, 1994, or 1985, IS:1893, Japanese, Colombian and other specifications may be used.

For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, soil characteristic parameters, etc.

Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Rayleigh quotient technique. This period is then utilized to calculate seismic coefficient C.

After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per UBC specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

Related Links

- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *M. To add a seismic load* (on page 860)
- *TR.31.2.5 Chinese Static Seismic per GB50011-2001* (on page 2369)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.14 IBC 2006/2009 Seismic Load Definition* (on page 2405)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.21 Turkish Seismic Code* (on page 2426)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1* (on page 2358)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.2 Canadian Seismic Code (NRC) - 1995* (on page 2355)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.20 NTC (Normas Técnicas Complementarias) Seismic Load* (on page 2423)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.1 RPA (Algerian) Seismic Load* (on page 2353)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.19 CFE (Comisión Federal De Electricidad) Seismic Load* (on page 2420)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.13 IBC 2000/2003 Load Definition* (on page 2401)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load* (on page 2387)

- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.18 Japanese Seismic Load* (on page 2418)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.7 Colombian NSR-98 Seismic Load* (on page 2382)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.23 UBC 1997 Load Definition* (on page 2432)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)
- *TR.31.2.22 UBC 1994 or 1985 Load Definition* (on page 2429)
- *TR.32.12.2 Generation of Seismic Loads* (on page 2598)

G.16.3 Wind Load Generator

The Wind Load Generator is a utility which takes as input wind pressure and height ranges over which these pressures act and generates nodal point and member loads.

This facility is available for two types of structures.

- Panel type or Closed structures
- Open structures

Closed Structures

Closed structures are ones like office buildings where non-structural entities like a glass facade, aluminum sheets, timber panels or non-load bearing walls act as an obstruction to the wind. If these entities are not included in the structural model, the load generated as a result of wind blowing against them needs to be computed. So, the steps involved in load generation for such structures are:

i. Identify the panels – regions circumscribed by members so that a polygonal closed area is formed. The area may also be formed between the ground level along one edge and members along the other.
ii. Calculate the center of gravity (CG) of each of the panels.
iii. For each panel, draw straight lines from the center of gravity to the midpoint of the members (or the ground) that form the panel boundary. The panel region will now contain several quadrilaterals whose two sides are made of portions of the respective members (or the ground) and the other two sides are lines going from the CG to the midpoint of the corresponding members.

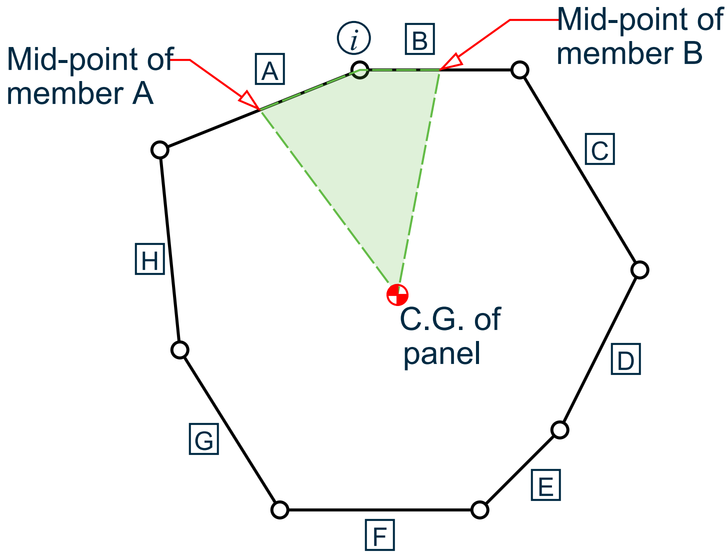


Figure 247: Influence area of Node i

iv. The area contained in any quadrilateral is allocated as the influence area for the node located at the meeting point of two members.

v. Multiply the influence area by the average wind pressure contained inside the influence area and by the exposure factor for the node. The resulting value is the concentrated wind force for the joint.

Plates and solids are not considered in the calculation of the panel area. Openings within the panels may be modeled with the help of exposure factors. An exposure factor is associated with each joint of the panel and is a fractional number by which the area affecting a joint of the panel can be reduced or increased.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for `Y UP`) should be loaded separately.

Open Structures

Open structures are those like transmission towers, in which the region between members is “open” allowing the wind to blow through. The procedure for load generation for open structures is

i. Calculate the exposed area of the individual members of the model.

ii. Multiply that exposed area by the wind pressure to arrive at the force and apply the force on individual members as a uniformly distributed load.

It is assumed that all members of the structure within the specified ranges are subjected to the pressure and hence, they will all receive the load. The concept of members on the windward side shielding the members in the inside regions of the structure does not exist for open structures. Members loaded as an open structure need not be vertical.

Related Links

- *Create Wind Type Definition dialog* (on page 2880)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *Add New Wind Definitions (data) dialog* (on page 2880)
- *M. To add a wind load definition* (on page 844)
- *TR.31.3 Definition of Wind Load* (on page 2435)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)

- *TR.32.12.3 Generation of Wind Loads* (on page 2603)
- *V.Wind On Closed Structure 1* (on page 3711)
- *GUID-6A46B9D1-50EA-417D-B730-A8877E54DA2C-low.svg*
- *V.Wind On Closed Structure 2* (on page 3718)
- *V. Wind On Open Structure* (on page 3726)

G.16.4 Snow Load

STAAD.Pro is capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

Related Links

- *Snow Load tab* (on page 2854)
- *TR.32.12.4 Generation of Snow Loads* (on page 2609)
- *M. To add an ASCE 7-02 snow load* (on page 870)
- *TR.31.5 Definition of Snow Load* (on page 2452)
- *TR.32.12.4 Generation of Snow Loads* (on page 2609)

G.17 Analysis Facilities

Salient features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.

Related Links

- *TR.37.1 Linear Elastic Analysis* (on page 2620)

G.17.1 Stiffness Analysis

The stiffness analysis implemented in STAAD.Pro is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components.

Structural systems such as slabs, plates, spread footings, etc., which transmit loads in two directions (frame members or finite elements) have to be discretized into a number of three- or four-noded finite elements connected to each other at their nodes. Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the joints. Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the joints. The plane stress effects as well as the plate bending effects are taken into consideration in the analysis.

Assumptions of the Analysis

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

1. The structure is idealized into an assembly of beam, plate and solid type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.

2. A beam member is a longitudinal structural member having a constant, doubly symmetric or near-doubly symmetric cross section along its length. Beam members always carry axial forces. They may also be subjected to shear and bending in two arbitrary perpendicular planes, and they may also be subjected to torsion. From this point these beam members are referred to as “members” in the manual.
3. A plate element is a three or four noded planar element having variable thickness. A solid element is a four-to-eight- noded, three dimensional element. These plate and solid elements are referred to as “elements” in the manual.
4. Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six degrees of freedom are considered at each node (i.e., three translational and three rotational) in the generation of relevant matrices. If the member is defined as truss member (i.e., carrying only axial forces) then only the three degrees (translational) of freedom are considered at each node.
5. Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems.

Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum established for all idealized elements so that element forces and displacements may be related to a common frame of reference.

Basic Equation

The complete stiffness matrix of the structure is obtained by systematically summing the contributions of the various member and element stiffness. The external loads on the structure are represented as discrete concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the displacements of the nodes by the equation:

$$A_j = a_j + S_j \cdot D_j$$

This formulation includes all the joints of the structure, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis.

Method to Solve for Displacements

There are many methods to solve the unknowns from a series of simultaneous equations.

In STAAD.Pro, the element stiffness matrices are assembled into a global stiffness matrix by standard matrix techniques used in FEA programs. The technique used by STAAD.Pro was developed based on routines made available in the public domain. The global stiffness matrix is then decomposed as

$$[\ K\ ] = [LT]\ [D]\ [L]$$

which is a modified Gauss method.

$$[K]\ \{d\} = \{F\}$$

becomes

$$[LT]\ [D]\ [L]\ \{d\} = \{F\}$$

which can be manipulated into a forward and a backward substitution step to obtain $\{d\}$. STAAD.Pro can detect singular matrices and solve then via a technique copied from Stardyne. To solve the matrices, the program uses an approach is used that is mathematically equivalent to the modified Choleski method. However the order of operations, memory use, and file use is highly optimized.

Consideration of Bandwidth

Internal storage order is automatically calculated to minimize time and memory.

Multiple Structures & Structural Integrity

The integrity of the structure is a very important requirement that must be satisfied by all models. You must make sure that the model developed represents one or more properly connected structures.

An "integral" structure may be defined as a system in which proper "stiffness connections" exist between the members/elements. The entire model functions as one or more integrated load resisting systems. STAAD.Pro checks structural integrity using a sophisticated algorithm and reports detection of multiple structures within the model. If you did not intend for there to be multiple structures, then you can fix it before any analysis. There are several additional model checking tools found on the **Utilities** ribbon tabs.

Modeling and Numerical Instability Problems

Instability problems can occur due to two primary reasons.

1. Modeling problem

   There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

   a. Local instability - A local instability is a condition where the fixity conditions at the end(s) of a member are such as to cause an instability in the member about one or more degrees of freedom. Examples of local instability are:

      i. Member Release: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) will be subjected to this problem.
      ii. A framed structure with columns and beams where the columns are defined as "TRUSS" members. Such a column has no capacity to transfer shears or moments from the superstructure to the supports.

   b. Global Instability - These are caused when the supports of the structure are such that they cannot offer any resistance to sliding or overturning of the structure in one or more directions. For example, a 2D structure (frame in the XY plane) which is defined as a SPACE FRAME with pinned supports and subjected to a force in the Z direction will topple over about the X-axis. Another example is that of a space frame with all the supports released for FX, FY or FZ.
2. Math precision

   A math precision error is caused when numerical instabilities occur in the matrix inversion process. One of the terms of the equilibrium equation takes the form $1/(1-A)$, where $A=k1/(k1+k2)$; k1 and k2 being the stiffness coefficients of two adjacent members. When a very "stiff" member is adjacent to a very "flexible" member, viz., when $k1>>k2$, or k1+k2 k1, A=1 and hence, $1/(1-A) =1/0$. Thus, huge variations in stiffnesses of adjacent members are not permitted. Artificially high E or I values should be reduced when this occurs.

   Math precision errors are also caused when the units of length and force are not defined correctly for member lengths, member properties, constants etc.

   You must also ensure that the model defined represents one single structure only, not two or more separate structures. For example, in an effort to model an expansion joint, you may end up defining separate structures within the same input file. Multiple structures defined in one input file can lead to grossly erroneous results.

Related Links

- *TR.37 Analysis Specification* (on page 2620)

G.17.2 Second Order Analysis

STAAD.Pro offers the capability to perform second order stability analyses.

G.17.2.1 P-Delta Analysis

Structures subjected to lateral loads often experience secondary forces due to the movement of the point of application of vertical loads. This secondary effect, commonly known as the P-Delta effect, plays an important role in the analysis of the structure.

In textbooks this secondary effect is typically referred to as stress stiffening for members in tension (or softening for compression). The stiffness changes due to P-Delta are known as geometric stiffness, [Kg]. There are two types of P-Delta effects for members. P-Δ which is due to the displacement of one end of a member relative to the other end (e.g., story drift of column members). A second effect is P-δ which is due to the bending of the member.

P-δ due to the bending of the member not only affects the local & global stiffness, nodal displacements, and member end forces; it also has an additional effect on the section displacements and section moments. The (axial compressive member force) times (the local relative to the ends section displacement) gives a section moment in addition to the flexural moment. This additional section moment will cause an additional sectional displacement; and so on. Normally this process will converge after 5-20 iterations if the member buckling load is not exceeded. STAAD.Pro uses up to 20 iterations unless convergence or divergence occurs.

P-δ due to the bending of the member can also occur with tension if the member has sufficient bending. STAAD only iterates once for tension.

STAAD.Pro does not include the effects of geometric stiffness for solids. If the part of the structure that deforms involves non-trivial motions of solids, then the results will be erroneous for P-Delta analysis (as well as for buckling analysis).

Related Links

- *P Delta Analysis tab* (on page 2917)
- *TR.37.2 P-Delta Analysis Options* (on page 2621)
- *A. To specify a P-Delta analysis* (on page 952)
- *V. 2D Frame 2 Step P-Delta Displacement* (on page 3731)
- *V. Single Column P-Delta Analysis* (on page 3746)

G.17.2.1.1 P-Delta Analysis – Large Delta and Small Delta

In STAAD.Pro, a procedure has been adopted to incorporate the P-Delta effect into the analysis without re-forming and factorizing the global stiffness matrix on each iteration. Actually, only the global stiffness matrix is formed and factorized; which must be done for any analysis. Only the relatively fast forward and backward substitution step for typically five to 25 iterations must be performed. This step is done simultaneously for however many cases are being solved. See G.17.2.1.2 P-Delta Kg Analysis (on page 2142) for an alternate formulation of P-Delta that may be used in dynamics.

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

If a structure is heavily loaded it may become unstable for some load cases. It may take 10 to 30 iterations for this instability to become obvious by the maximum displacements or bending moment envelope values becoming very large or infinite or reported as NaN (“not a number”).

The procedure consists of the following steps:

1. First, the primary deflections are calculated based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. By default the small delta effects are calculated. To include only the large delta effects, enter the `LARGEDELTA` option on the `PDELTA` command. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.

   The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The `REPEAT LOAD` facility has been created with this requirement in mind. This facility allows you to combine previously defined primary load cases to create a new primary load case.
3. The revised load vector is used with the static triangular factorized matrix to generate new deflections.
4. Element/Member forces and support reactions are calculated based on the new deflections.

Repeat steps 2 to 4 for several iterations. Three to 30 iterations are recommended. This procedure yields reasonably accurate results with small displacement problems. You are allowed to specify the number of iterations. If the Converged option is used, then set the displacement convergence tolerance by entering a `SET PDELTATOL` i_9 command before the Joint Coordinates. If all changes in displacement dof from one iteration to the next is less than the specified tolerance value, i_9, then that case is converged.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta has the most effect in structures where there are vertical and horizontal loads in the same load case.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits large buckling displacements if the loads make the structure unstable. You may need to repeat the analysis with only one to five iterations in order to get a pre-collapse solution in order to view the large displacement areas.

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

Related Links

- *TR.37.2 P-Delta Analysis Options* (on page 2621)

G.17.2.1.2 P-Delta Kg Analysis

In STAAD.Pro, an alternate procedure has been adopted to incorporate the P-Delta effect into the analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

   The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The `REPEAT LOAD` facility has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

This procedure yields reasonably accurate results with small displacement problems. STAAD.Pro allows the user to specify multiple iterations of this P-Delta-KG procedure; however one iteration is almost always sufficient.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case, a message is printed and the results of the case are set to zero. (In this case, repeat the analysis using `PDELTA 30 ANALYSIS SMALLDELTA` instead).

Related Links

- *TR.37.2 P-Delta Analysis Options* (on page 2621)

G.17.2.1.3 P-Delta K+Kg Dynamic Analysis

In STAAD.Pro, an alternate procedure has been adopted to incorporate the P-Delta effect into dynamic analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

This method uses the resulting [K+Kg] matrices from the last static case before the `PDELTA KG` command in the dynamic cases that precede the `PDELTA KG` command.

```
LOAD n
static case input
LOAD n+1
dynamic load case
PDELTA KG ANALYSIS
```

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading for case *n*.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The final triangular factorization for case *n* is then used in the dynamic case *n+1* along with the masses specified in case n+1 to solve the dynamic analysis.

Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The `REPEAT LOAD` facility has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only one to five iterations or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case a message is printed and the results of the case are set to zero. The dynamic results should be ignored if this type of failure should occur.

Related Links

- *TR.37.2 P-Delta Analysis Options* (on page 2621)

G.17.2.1.4 AISC 360 Direct Analysis

AISC 360-10/16 Chapter C and AISC 360-05 Appendix 7 describe a method of analysis, called “direct analysis”, which accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

The ANSI/AISC 360 direct analysis procedure has been adopted to incorporate the P-Delta effect into a static analysis by combining the global stiffness matrix and the global geometric stiffness matrix $[K+Kg]$; plus flexural stiffness reduction; plus axial stiffness reduction; plus an additional flexure reduction if member axial compression forces are above 50% of yield; plus the addition of notional loads.

This is a nonlinear, iterative analysis as the stiffness of the members is dependent upon the forces generated by the load.

1. The primary deflections are calculated by linear static analysis based on the provided external loading for case *n*. The stiffness reductions and notional loads are included here.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.
3. The final triangular factorization for case n is then used to calculate displacements and member forces.

   Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The `REPEAT LOAD` facility has been created with this requirement in mind. This facility allows you to combine previously defined primary load cases to create a new primary load case.
4. The axial force is compared to yield force to calculate τ_b (see Chapter C or AISC 360-10/16 or Appendix 7 of AISC 360-05). Flexure stiffness of selected members is set to ($0.80 \times \tau_b \cdot EI$)
5. Steps 2 to 4 are repeated until convergence or the iteration limit is reached.

   The analysis will iterate; in each step, changing the member characteristics until the maximum change in any τ_b is less than the specified τ tolerance. If the maximum change in any τ_b is less than 100 times the τ tolerance *and* the maximum change in any displacement degree of freedom is less than the specified displacement tolerance; then the solution has converged for this case.

General Procedure for Using the Direct Analysis Feature

There are three general steps required to set up a Direct Analysis:

1. Specify the definition with the `DEFINE DIRECT` command.
2. Specify the notional loads using the `NOTIONAL LOAD` load specification.
3. Specify a direct analysis method with the `PERFORM DIRECT ANALYSIS` command. Within this command, you will specify the tolerances or iteration limits.

Related Links

- *TR.31.7 Definition of Direct Analysis Members* (on page 2454)

- *Add New Direct Analysis Definition dialog* (on page 2899)
- *M. To define direct analysis parameters* (on page 871)
- *TR.32.13 Notional Loads* (on page 2610)
- *Repeat Load tab* (on page 2863)
- *M. To add a notional load case* (on page 872)
- *Perform Direct Analysis tab* (on page 2920)
- *TR.37.5 Direct Analysis* (on page 2630)
- *A. To specify a direct analysis* (on page 953)
- *D1.A. American Codes - Steel Design per AISC 360 Unified Specification* (on page 1086)
- *TR.31.7 Definition of Direct Analysis Members* (on page 2454)
- *TR.32.11 Repeat Load Specification* (on page 2595)
- *TR.32.13 Notional Loads* (on page 2610)
- *TR.33 Reference Load Cases - Application* (on page 2613)
- *TR.37.5 Direct Analysis* (on page 2630)
- *V. Direct Analysis of a Beam* (on page 3741)
- *V. Direct Analysis of a Column* (on page 3743)

G.17.2.2 Buckling Analysis

In STAAD.Pro, two procedures have been adopted to incorporate the calculation of the Buckling Factor for a load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

Note: This feature is available in STAAD.Pro 2007 Build 01 and greater.

STAAD.Pro does not include the effects of geometric stiffness for solids. If the part of the structure that deforms during buckling involves non-trivial motions of solids, then the results will be erroneous for buckling (as well as for P-Delta analysis).

Related Links

- *TR.37.4 Buckling Analysis* (on page 2628)
- *Perform Buckling Analysis tab* (on page 2925)
- *A. To specify buckling analysis* (on page 957)
- *TR.37.4 Buckling Analysis* (on page 2628)
- *V. Column Buckling Factor* (on page 3734)

G.17.2.2.1 Buckling Analysis - Iterative Method

In STAAD.Pro, a simple procedure has been adopted to incorporate the calculation of the Buckling Factor for any number of primary load cases. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are multiplied by the estimated BF (buckling factor) and then added to the global stiffness matrix K.

Buckling Kg matrix effects are calculated for frame members and plate elements only. They are not calculated for solid elements. So buckling analysis is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

3. For compressive cases, the Kg matrix is negative definite. If the buckling factor is large enough, then $[\ [K] + BF \times [Kg]\ ]$ will also be negative definite which indicates that BF times the applied loads is greater than the loading necessary to cause buckling.
4. STAAD.Pro starts an iterative procedure with a BF estimate of 1.0. If that BF causes buckling, then a new, lower BF estimate is used in the next trial. If the BF does not cause buckling, then a higher BF estimate is used. On the first iteration, if the determinant of the K matrix is positive and lower than the determinant of the K+Kg matrix, then the loads are in the wrong direction to cause buckling; and STAAD.Pro will stop the buckling calculation for that case.
5. After a few iterations, STAAD.Pro will have the largest BF that did not cause buckling (lower bound) and the lowest BF that did cause buckling (upper bound). Then each trial will use a BF estimate that is halfway between the current upper and lower bounds for BF.
6. After the default iteration limit is reached or the user specified iteration limit, `MAXSTEPS`, is reached or when two consecutive BF estimates are within 0.1% of each other; then the iteration is terminated.
7. Results for this load case are based on the last lower bound BF calculated.

- Only primary load cases may be solved
- Any number of buckling cases may be solved.
- Only the first buckling mode (lowest BF) is calculated.
- The buckling shape may not be as expected even though the buckling factor is OK. To enhance the mode shape result, apply small loads in the locations and directions where you expect the large displacements.

Note: During the buckling analysis using the iterative method, if the determinant of the matrix changes sign in one or more steps, then the resulting values of displacements and forces for the last successful step may not be accurate. If using STAAD.Pro Advanced, then it is recommended to check the buckling factor using the eigen method.

G.17.2.2.2 Buckling Analysis - Eigen Method

In STAAD.Pro, a second procedure has been adopted to incorporate the calculation of the Buckling Factor for one primary load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure. This procedure is an eigenvalue calculation to get buckling factors and buckling shapes.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects for members are calculated. These terms are the terms of the Kg matrix.
3. An eigenvalue problem is formed. $|\ [\ K\ ] - BF_i * [\ Kg\ ]\ | = 0$

There will be up to 4 buckling factors (BF) and associated buckling mode shapes calculated. The buckling factor is the amount by which the static load case needs to be multiplied by to just cause buckling (Euler buckling). BF less than 1.0 means that the load causes buckling; greater than 1.0 means buckling has not occurred. If BF is negative, then the static loads are in the opposite direction of the buckling load.

Notes

- Solid elements do not contribute to Kg in STAAD.Pro.

- Buckling shapes for the last buckling case only may be displayed in the postprocessor. If there are several buckling cases, then all will have their buckling factors printed.
- The displacement and member/element results are not calculated for the load case times the buckling factor.

G.17.2.3 Static Geometrically Nonlinear Analysis

In STAAD.Pro, a procedure has been adopted to incorporate the geometric nonlinearities into the analysis by updating the global stiffness matrix and the global geometric stiffness matrix $[K+Kg]$ on every step based on the deformed position. The deformations significantly alter the location or distribution of loads, such that equilibrium equations must be written with respect to the deformed geometry, which is not known in advance.

Note: This requires a STAAD.Pro Advanced license.

1. First, the primary deflections are calculated by linear static analysis based on the provided external loading.
2. Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.
3. Next the deflections are recalculated. Now equilibrium is computed in the deformed position to get out of balance forces. The tangential stiffness matrix is determined from each members new position; the Kg matrix is updated; and the out of balance forces are applied to get the next iteration result.
4. Repeat until converged. If displacements are much too large, then try using `ARC 5` to limit displacements on the first linear static step to 5 inches or some suitable value. The `STEP 10` parameter may help by loading the structure over many steps.
5. The options for Newton-Raphson, Kg, Steps = 1 are usually taken; but these options are available for some difficult cases.
6. Offset beams, curved beams, cables are not permitted. Tension/compression is not permitted.

Nonlinear effects are calculated for springs, frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for nonlinear analyses because this analysis type may result in buckling or large displacements.

The following limitations should be noted regarding static, geometrically nonlinear analyses:

- Large rotations in one step should be avoided by using more steps.
- Very large displacements, unstable structures, and/or post-buckling should be avoided.
- Geometrically nonlinear only. No tension/compression or contact is considered. No yield, plastic moment hinges or bilinear behavior is considered.
- Solids cannot be used for this analysis method.
- Temperature loads are *not* supported for geometric nonlinear analysis.

Related Links

- *Nonlinear Analysis tab* (on page 2922)
- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)
- *A. To specify a nonlinear analysis* (on page 954)
- *TR.37.8 Geometric Nonlinear Analysis* (on page 2654)

G.17.2.4 Imperfection Analysis

Structures subjected to vertical and lateral loads often experience secondary forces due to curvature imperfections in the columns and beams. This secondary effect is similar to the P-Delta effect. In STAAD.Pro the procedure consists of the following steps:

1. First, the deflections and the axial forces in the selected imperfect members are calculated based on the provided external loading.
2. The axial forces and the input imperfections are then used to compute an additional loading on the selected imperfect members that are in compression. These additional loads are combined with the originally applied loading.
3. The static analysis is performed with the combined loading to obtain the final result.

The section moment due to tension and the section displacements due to shear/bending are added to the moment diagram, if small delta is selected. This is no iteration performed for this step.

Related Links

- *TR.37.9 Imperfection Analysis* (on page 2657)
- *Perform Imperfection Analysis tab* (on page 2924)
- *A. To specify an imperfection analysis* (on page 956)
- *TR.26.6 Member Imperfection Information* (on page 2313)
- *TR.37.9 Imperfection Analysis* (on page 2657)

G.17.2.5 Multilinear Analysis

When soil is to be modeled as spring supports, the varying resistance it offers to external loads can be modeled using a multilinear analysis.

An example is when the soil's behavior in tension differs from its behavior in compression. Stiffness-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. The load cases in a multi-linear spring analysis must be separated by the `CHANGE` command and `PERFORM ANALYSIS` command. The `SET NL` command must be provided to specify the total number of primary load cases. There may not be any PDELTA, dynamic, or `TENSION/ COMPRESSION` member cases. The multi-linear spring command will initiate an iterative analysis which continues to convergence.

Related Links

- *TR.27.4 Multilinear Spring Support Specification* (on page 2322)

G.17.2.6 Tension- and Compression-Only Analysis

When some members or support springs are linear but carry only tension or only compression, then this analysis may be used.

This analysis is automatically selected if any member or spring has been given the tension or compression only characteristic. This analysis is an iterative analysis which continues to convergence. Any member/ spring that fails its criteria will be inactive (omitted) on the next iteration. Iteration continues until all such members have the proper load direction or are inactive (default iteration limit is 10).

This is a simple method that may not work in some cases because members are removed on interim iterations that are needed for stability. If instability messages appear on the second and subsequent iterations that did not appear on the first cycle, then do not use the solution. If this occurs on cases where only springs are the tension/compression entities, then use multilinear spring analysis.

Note:

If the list of members that are to be considered as tension changes between load cases (which would be characterized with the inclusion of an additional `MEMBER TENSION` command with the revised member list), then this should be followed by an additional analysis command (and a `CHANGE` command) before starting the next load case definition.

If the list of members does not change (i.e., there is no new `MEMBER TENSION` command defined between these load cases), then it is not necessary to include any additional analysis and change commands, the analysis will automatically reset the axial status and iterate to solve for all the specified tension members.

G.17.2.7 Nonlinear Cable or Truss Analysis

Note: This feature is available in limited form.

When all of the members, elements and support springs are linear except for cable and/or preloaded truss members, then this analysis type may be used. This analysis is based on applying the load in steps with equilibrium iterations to convergence at each step. The step sizes start small and gradually increase (15-20 steps is the default). Iteration continues at each step until the change in deformations is small before proceeding to the next step. If not converged, then the solution is stopped. You can then select more steps or modify the structure and rerun.

Structures can be artificially stabilized during the first few load steps in case the structure is initially unstable (in the linear, small displacement, static theory sense).

The user has control of the number of steps, the maximum number of iterations per step, the convergence tolerance, the artificial stabilizing stiffness, and the minimum amount of stiffness remaining after a cable sags.

This method assumes small displacement theory for all members/trusses/elements other than cables & preloaded trusses. The cables and preloaded trusses can have large displacement and moderate/large strain. Cables and preloaded trusses may carry tension and compression but cables have a reduced E modulus if not fully taut. Pretension is the force necessary to stretch the cable/truss from its unstressed length to enable it to fit between the two end joints. Alternatively, you may enter the unstressed length for cables.

The current nonlinear cable analysis procedure can result in compressive forces in the final cable results. The procedure was developed for structures, loadings, and pretensioning loads that will result in sufficient tension in every cable for all loading conditions. The possibility of compression was considered acceptable in the initial implementation because most design codes strongly recommend cables to be in tension to avoid the undesirable dynamic effects of a slack cable such as galloping, singing, or pounding. The engineer must specify initial preloading tensions which will ensure that all cable results are in tension. In addition this procedure is much more reliable and efficient than general nonlinear algorithms. To minimize the compression the `SAGMIN` input variable can be set to a small value such as 0.01, however that can lead to a failure to converge unless many more steps are specified and a higher equilibrium iteration limit is specified. SAGMIN values below 0.70 generally requires some adjustments of the other input parameters to get convergence.

Currently the cable and truss are not automatically loaded by selfweight, but the user should ensure that selfweight is applied in every load case. o not enter component load cases such as wind only; every case must be realistic. Member loads will be lumped at the ends for cables and trusses. Temperature load may also be applied to the cables and trusses. It is OK to break up the cable/truss into several members and apply forces to the intermediate joints. Y-up is assumed and required.

The member force printed for the cable is Fx and is along the chord line between the displaced positions of the end joints.

The analysis sequence is as follows:

1. Compute the unstressed length of the nonlinear members based on joint coordinates, pretension, and temperature.
2. Member/Element/Cable stiffness is formed. Cable stiffness is from EA/L and the sag formula plus a geometric stiffness based on current tension.
3. Assemble and solve the global matrix with the percentage of the total applied load used for this load step.
4. Perform equilibrium iterations to adjust the change in directions of the forces in the nonlinear cables, so that the structure is in static equilibrium in the deformed position. If force changes are too large or convergence criteria not met within 15 iterations then stop the analysis.
5. Go to step 2 and repeat with a greater percentage of the applied load. The nonlinear members will have an updated orientation with new tension and sag effects.
6. After 100% of the applied load has converged then proceed to compute member forces, reactions, and static check. Note that the static check is not exactly in balance due to the displacements of the applied static equivalent joint loads.

The load cases in a nonlinear cable analysis must be separated by the `CHANGE` command and `PERFORM CABLE ANALYSIS` command. The `SET NL` command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables and preloaded trusses:

1. Do not use Member Offsets.
2. Do not include the end joints in control/dependent command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

Related Links

- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)
- *TR.23.2 Member Cable Specification* (on page 2290)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)
- *TR.23.1 Member Truss Specification* (on page 2289)

G.17.2.8 Advanced Nonlinear Cable Analysis

When all of the members, elements, and support springs are linear except for cable members, then this analysis type may be used.

Note: This feature is available in limited form.

You have control of the number of steps, the maximum number of iterations per step, the convergence tolerance, include/exclude Kg matrix and use full/modified Newton-Raphson method.

The nonlinear static solver employs the Newton method (full Newton *or* modified Newton method) to analyze nonlinear problems. In STAAD.Pro, cable elements and P-Delta effect in beam/column and plates cause geometric nonlinearity.

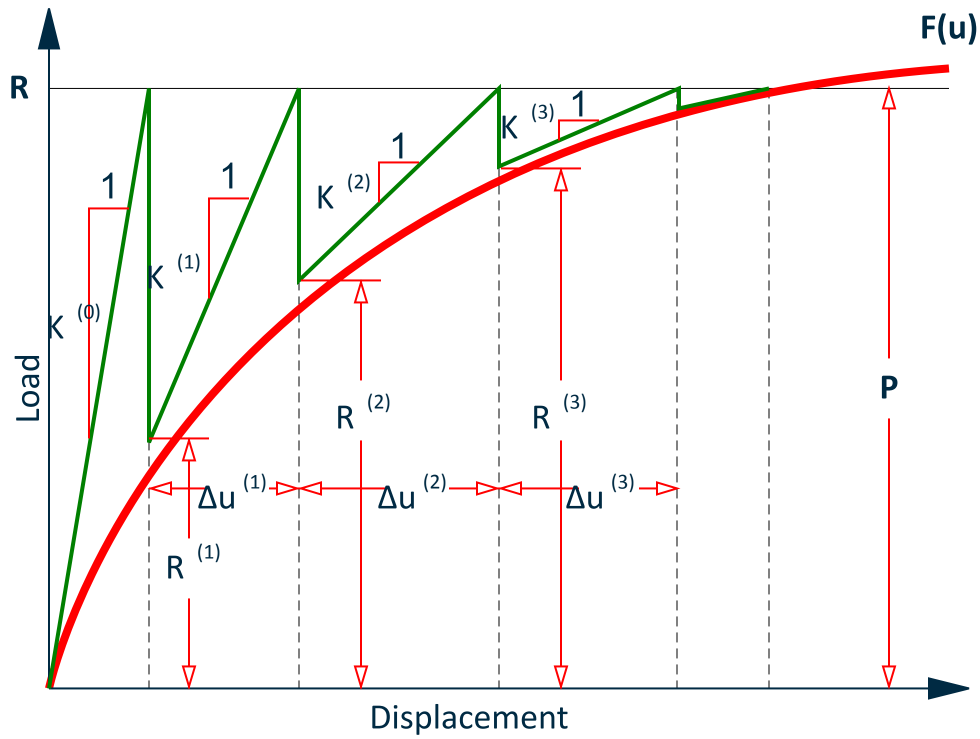


Figure 248: Numerical representation of the nonlinear static solver with full Newton method

Steps Included in the Analysis

The nonlinear static analysis has several steps.

1. Final applied loading vector *{Pext}* is assembled. Incremental load vector *{P}* = *{Pext}*/*nSteps* is calculated where nSteps is the number of load steps. *{uprev}*= *{0}* is also defined
2. For the first iteration, the unbalanced loading *{ΔP}*=*{P}* and displacement vector *{u}*=*{uprev}*, the stiffness matrix *[K]* are assembled. If any elements are performing nonlinearly, their element stiffness matrix will be determined based on their current nonlinear status.
3. The equation *[K]{Δu}* =*{ΔP}* is solved to find out the incremental displacement *{Δu}*.
4. The current displacement vector is calculated as *{u}* =*{u}*+*{Δu}*.
5. This newly calculated *{u}* is used to update all elements nodal coordinates.
6. Based on the updated elements, the element reaction *{R}* is calculated.
7. The unbalanced loading now becomes *{ΔP}*=*{P}*-*{R}*.
8. Convergence is checked by comparing |*{ΔP}*|/*{P}* with *ε*. If convergence is achieved the current displacement is saved as the displacement of previous iteration, i.e. *{uprev}* = *{u}*. The next load increment is applied and same all steps are repeated.
9. If convergence is not achieved, steps 2-7 are repeated until convergence is achieved or the maximum iteration number is reached.

Theory of Cable Elements

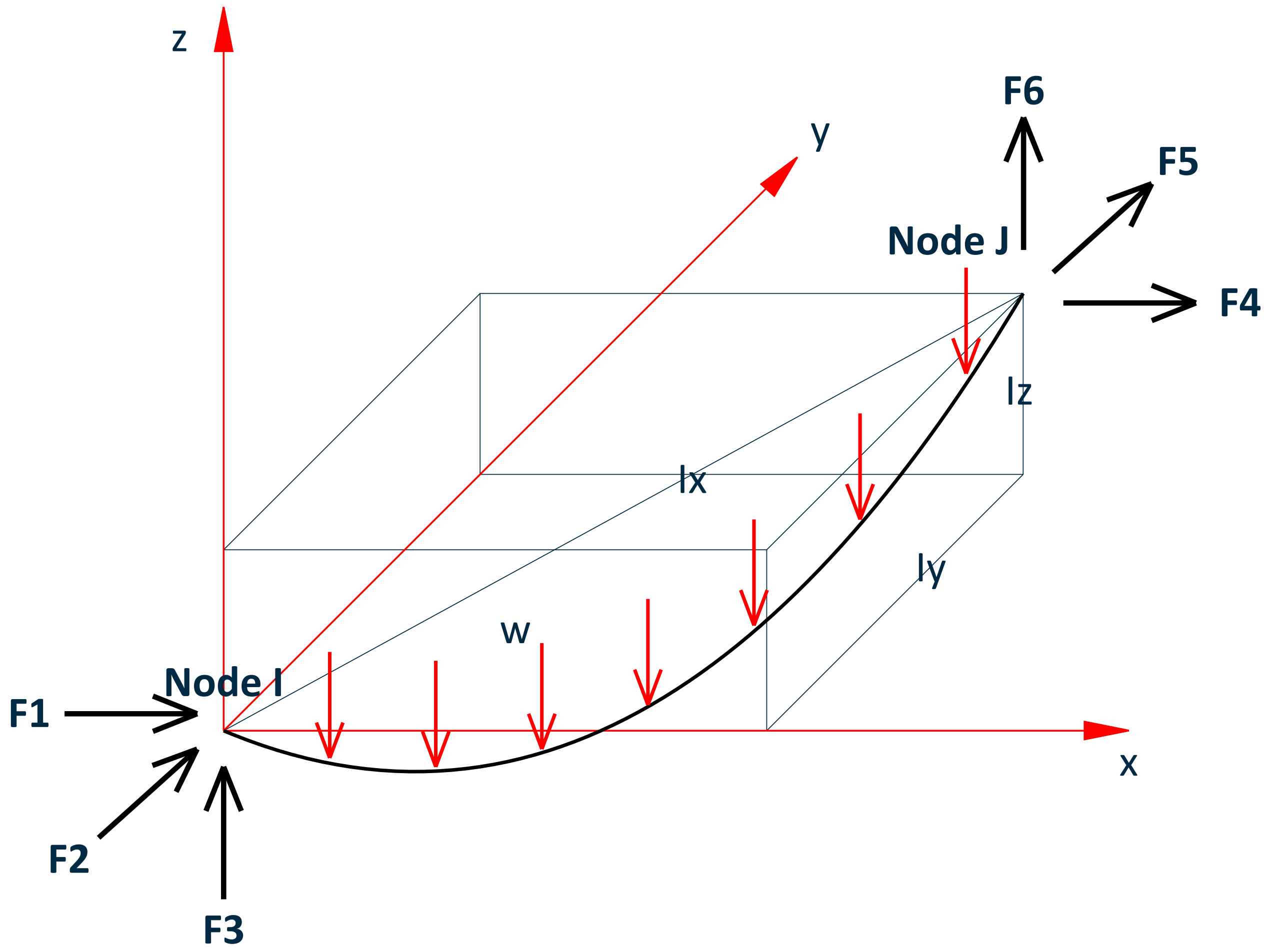


Figure 249: 3D cable sketch

The cable element formulation follows the catenary theory. It is a nonlinear element with geometric nonlinearity, but without material nonlinearity. For a 3D cable element shown in Figure 2, the free body equilibrium is:

$$T\frac{dx}{dp} = -F_1 \tag{1a}$$

$$T\frac{dy}{dp} = -F_2 \tag{1b}$$

$$T\frac{dz}{dp} = -F_3 + ws \tag{1c}$$

With geometrical constraint equations and constitutive equations, the relationship between chord length components l_x, l_y, lz and end support force components F_1, F_2, F_3 could be derived as:

$$l_x = -\frac{F_1 L_0}{EA} - \frac{F_1}{w}\left\{ln\left[\sqrt{F_1^2 + F_2^2 + (wL_0 - F_3)^2} + wL_0 - F_3\right] - ln\left(\sqrt{F_1^2 + F_2^2 + F_3^2} - F_3\right)\right\} \tag{2a}$$

$$l_y = -\frac{F_2 L_0}{EA} - \frac{F_2}{w}\left\{ln\left[\sqrt{F_1^2 + F_2^2 + (wL_0 - F_3)^2} + wL_0 - F_3\right] - ln\left(\sqrt{F_1^2 + F_2^2 + F_3^2} - F_3\right)\right\} \tag{2b}$$

$$l_z = -\frac{F_3 L_0}{EA} + \frac{wL_0^2}{2EA} + \frac{1}{w}\left[\sqrt{F_1^2 + F_2^2 + (wL_0 - F_3)^2} - \sqrt{F_1^2 + F_2^2 + F_3^2}\right] \tag{2c}$$

Then the derivative express of equation (2) is the flexibility matrix, as shown in equation (3). For brevity, the terms in the equation are not listed.

$$\begin{bmatrix} dl_x \\ dl_y \\ dl_z \end{bmatrix} = \begin{bmatrix} \frac{\partial l_x}{\partial F_1} & \frac{\partial l_x}{\partial F_2} & \frac{\partial l_x}{\partial F_3} \\ \frac{\partial l_y}{\partial F_1} & \frac{\partial l_y}{\partial F_2} & \frac{\partial l_y}{\partial F_3} \\ \frac{\partial l_z}{\partial F_1} & \frac{\partial l_z}{\partial F_2} & \frac{\partial l_z}{\partial F_3} \end{bmatrix} \begin{bmatrix} dF_1 \\ dF_2 \\ dF_3 \end{bmatrix} = \begin{bmatrix} f_{11} & f_{12} & f_{13} \\ f_{21} & f_{22} & f_{23} \\ f_{31} & f_{32} & f_{33} \end{bmatrix} \begin{bmatrix} dF_1 \\ dF_2 \\ dF_3 \end{bmatrix} = f \begin{bmatrix} dF_1 \\ dF_2 \\ dF_3 \end{bmatrix}$$

The inverse of the flexibility matrix is the stiffness matrix. So the deformation-force relationship and the stiffness matrix are found. Hence the nonlinear equations can be solved using the finite element analysis solvers.

In the step list of the nonlinear static solver, step 4 will provide the cable element the new updated displacement. And step 2 and step 5 will ask the cable element's stiffness matrix and reaction forces, which can be calculated with equation (3).

Limitation of cable elements

One limitation of using catenary theory is that the cable element cannot be loaded with non-uniformly distributed load, point load inside the element. In order to do such doing, the physical cable must be modeled with multiple analytical cable elements, so that the non-uniform load could be approximately simulated by uniform loads on each analytical cable elements, and there are nodes at locations where point loads are applied.

Notes

The load cases in a nonlinear cable analysis must be separated by the `CHANGE` command and `PERFORM CABLE ANALYSIS ADVANCED` command. The `SET NL` command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables:

1. Do not use Member Offsets.
2. Do not include the end joints in a Control/Dependent command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

Note: This analysis feature can be used only when Advanced Analysis License is active.

Related Links

- *TR.23.2 Member Cable Specification* (on page 2290)
- *TR.37.3 Nonlinear Cable Analysis* (on page 2624)

G.17.3 Dynamic Analysis

Available dynamic analysis facilities include solution of the free vibration problem (eigenproblem), response spectrum analysis, and forced vibration analysis.

Related Links

- *TR.32.10.2 Time Varying Load for Response History Analysis* (on page 2593)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)
- *TR.30 Miscellaneous Settings for Dynamic Analysis* (on page 2344)
- *TR.34 Frequency Calculation* (on page 2614)
- *TR.30 Miscellaneous Settings for Dynamic Analysis* (on page 2344)
- *TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004* (on page 2525)
- *TR.34 Frequency Calculation* (on page 2614)
- *TR.30 Miscellaneous Settings for Dynamic Analysis* (on page 2344)
- *TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994* (on page 2520)
- *TR.34 Frequency Calculation* (on page 2614)
- *TR.30.1 Cut-Off Frequency, Mode Shapes, or Time* (on page 2344)
- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *TR.34.2 Modal Calculation Command* (on page 2615)
- *TR.34.1 Rayleigh Frequency Calculation* (on page 2614)
- *G.15.1 Joint Loads* (on page 2128)
- *TR.32.10 Dynamic Loading Specification* (on page 2498)
- *TR.30.1 Cut-Off Frequency, Mode Shapes, or Time* (on page 2344)
- *TR.30.2 Mode Selection* (on page 2345)
- *TR.37.6 Steady State and Harmonic Analysis* (on page 2632)
- *TR.34 Frequency Calculation* (on page 2614)
- *TR.34.2 Modal Calculation Command* (on page 2615)

G.17.3.1 Solution of the Eigenproblem

The eigenproblem is solved for structure frequencies and mode shapes considering a diagonal, lumped mass matrix, with masses possible at all active degrees of freedom (d.o.f.) included. Two solution methods may be used: the subspace iteration method for all problem sizes (default for all problem sizes), and the Arnoldi/ Lanczos method for evaluating eigenvectors (Advanced Analysis only). Additionally, load dependent Ritz vectors (LDR) can be used for dynamically loaded structures.

For large scale eigen value problems, the Arnoldi method is very efficient.

Autoshifting of Eigenvectors

For large models having a large number of d.o.f and a large number of modes extracted (i.e., memory-bound), an incremental solver mode called autoshifting can be used by the Advanced Analysis. A mode shift value is specified to indicate the fixed number of modes the solution tries to find in each shift. The main benefit of using the incremental solver is that it is memory efficient (e.g., problems that were not solvable before on 32-bit systems might be solved with that technique). It is recommended to be used only when memory allocation failure takes place during eigen solution using subspace-iteration or Arnoldi/Lanczos methods. When the program fails to extract eigen vectors due to insufficient memory, you can use auto-shifting which significantly reduces memory demand.

Note: Generally, the Arnoldi/Lanczos is more robust than subspace iteration with autoshifting. The Arnoldi/ Lanczos method is very efficient at finding large eigen vectors with autoshift. Thus, if memory problems occur with subspace-iteration (regular mode), it is recommended to switch to the Arnoldi/Lanczos method (regular mode). If memory problems occur with this method in regular mode, then autoshift can be applied to reduce the memory demand.

Since autoshifting can provide significant reduction in memory demand in the solution, it can be ideal for the eigenvalue solution of large systems. In this case, the you are required to provide a “targeted” number of eigen modes to be searched in each shift. The solution starts with “0” shift and tries to find targeted number of eigen modes. Once completed, a new shift is applied and it tries to find next set of eigenvalues within the new shift. This continues until all required number of eigenvalues are found.

The subspace iteration method is very sensitive to calculated shift resulting in a partial extraction of mode. It is recommended to use this solution as a supplemental or alternative solution to existing (default mode). Whenever the program extracts partial set of eigen vectors, it issues a warning message.

If the solution return partial results, it means that not all the required number of eigen modes were found. But the solution guarantees that no eigen values are missed among returned results. Partial results can be still useable for dynamic analysis if they satisfy other analysis requirements. For example, “m” modes satisfy 90% mass participation.

Partial results can be also returned by the subspace method (Advanced Analysis) without using auto shifting method.

The program may miss some Eigen values because of applied shift while performing Eigen solution using Subspace-iteration method. In this case, a warning message is given. In this case, the results returned include missing modes. These results should be *used with caution* and it is strongly advised for further investigation (dynamic contribution from missing modes might be too important to ignore in analysis).

When the Arnoli/Lanczos method is used with autoshifting, an initial frequency shift may also be specified.

Load-Dependent Ritz Vectors

Research has indicated that considering the effect of natural free-vibration mode shapes may not be the most efficient basis for mode-superposition analysis of structures subjected to dynamic loads. This implies that dynamic analyses —like response spectrum and time history analysis— based on a special set of load-dependent Ritz vectors may yield more accurate results than the use of the same number of natural modes.

There are several reasons to consider Ritz vector analysis as a more efficient approach.

a. For large structural systems, the solution to find free-vibration modes and frequencies may require a significant amount of computational effort.
b. The Ritz vectors method takes into account the spatial distribution of the dynamic loading, whereas the direct use of natural modes neglects this information. Therefore, many of the natural mode shapes that are calculated may not have significant contribution to the dynamic response.
c. Ritz vector analysis by default does not include static correction due to higher mode truncation. The command `MIS` is required to be issued to include missing mass correction.

The Ritz vectors method is recommended where the solution with eigen vectors fails to capture 90% mass participation (a mandatory requirement of most country seismic codes) with a reasonable number of modes.

It is also recommended where eigen vectors capture irrelevant modes. Even though they are real modes, they are not relevant to the structural response due to the applied dynamic loading.

The spatial distribution of the dynamic load vector serves as a starting load vector to initiate the analysis process. The program will automatically generate this starting load or you can specify it using the `DEFINE STARTING LOAD` command. In the program generated method, the starting load vector is generated using the mass model of the structure. Assuming a mass matrix only has translational components, the resulting load vector will have force components in the directions of all the translational degrees of freedom. and will have zero values for all the rotational degrees of freedom. This will guarantee a static deflection with components in many directions with this initial mode assumption. As long as the initial mode assumption has a component in the direction of interest, it will yield a correct set of Ritz vectors.

If you define a set of starting load vectors, then this load vector will have a force component in one translational degree of freedom in the direction of the dynamic load. This results in a static deflection mainly in the direction of interest. In some models where mass participation is predominant in one translational direction for the initial few modes, this method can achieve 90% mass participation with only a few modes.

Note: A single set of Ritz vectors can be derived from a single starting load. If all starting loads that participate in dynamic loads are defined, a single set of Rtiz vectors can be extracted from all participating starting loads. Therefore, use of the `DEFINE STARTING LOAD` command is best suited if the structural response is predominant in one translational direction. If response in multiple translational d.o.f. are predominant, then it is recommended to use the program generated starting load vector.

Leger P, Wilson EL, Clough RW., *The use of load-dependent Ritz vectors for dynamic and earthquake analyses.* Technical Report UC13/EERC86/04, Earthquake Engineering Research Center, University of California Berkeley, Berkeley, CA, 1986.

Related Links

- *Add New Define Starting Mass Load dialog* (on page 2833)
- *TR.31.9 Defining Starting Load* (on page 2459)
- *M. To use starting vectors with load-dependent Ritz vectors* (on page 885)
- *TR.5 Set Command Specification* (on page 2206)
- *TR.30.1 Cut-Off Frequency, Mode Shapes, or Time* (on page 2344)
- *TR.30 Miscellaneous Settings for Dynamic Analysis* (on page 2344)

- *TR.32.10 Dynamic Loading Specification* (on page 2498)
- *TR.34 Frequency Calculation* (on page 2614)

G.17.3.2 Mass Modeling

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly depending upon the mass modeling. This variation, in turn, affects the response spectrum and forced vibration analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem.

In STAAD.Pro, all masses that are capable of moving should be modeled as loads applied in all possible directions of movement. Even if the loading is known to be only in one direction there is usually mass motion in other directions at some or all joints and these mass directions (applied as loads, in weight units) must be entered to be correct. Joint moments that are entered will be considered to be weight moment of inertias (force-length^2 units).

Note: Take care to enter selfweight, joint, and element loadings in global directions with the same sign as much as possible so that the representative masses do not cancel each other.

Member/Element loadings may also be used to generate joint translational masses. Note that loads (representing the masses) defined as member concentrated loads, or partially distributed member loads, on non-globally aligned members may result in additional mass being included in orthogonal directions at the nodes. This is because the resolution of these loads (masses) onto the nodes is only considered in the positive direction, and thus does not account for any directional sign of the effect at the node. Member end joint moments that are generated by the member loading (including concentrated moments) are discarded as irrelevant to dynamics. Enter mass moments of inertia, if needed, at the joints as joint moments.

STAAD.Pro uses a diagonal mass matrix of six lumped mass equations per joint. The selfweight or uniformly loaded member is lumped 50% to each end joint without rotational mass moments of inertia. The other element types are integrated but—roughly speaking—the weight is distributed equally amongst the joints of the element.

The members/elements of finite element theory are simple mathematical representations of deformation meant to apply over a small region. The finite element analysis (FEA) procedures will converge if you subdivide the elements and rerun; then subdivide the elements that have significantly changed results and rerun; an so on, until the key results are converged to the accuracy needed.

An example of a simple beam problem that needs to subdivide physical members to better represent the mass distribution (as well as the dynamic response and the force distribution response along members) is a simple floor beam between two columns will put all of the mass on the column joints. In this example, a vertical ground motion will not bend the beam even if there is a concentrated force (mass) at mid span.

Masses that are assigned to dependent degrees of freedom (dof) are moved to the control node with a rotatory mass moment of inertia applied at the control. This will be an approximation if the control node is not at the center of gravity (CG, i.e., center of mass) of the dependent masses.

In addition, the dynamic results will not reflect the location of a mass within a member (i.e., the masses are lumped at the joints). This means that the motion, of a large mass in the middle of a member relative to the ends of the member, is not considered. This may affect the frequencies and mode shapes. If this is important to the solution, split the member into two. Another effect of moving the masses to the joints is that the resulting shear/ moment distribution is based as if the masses were not within the member.

Note: If one end of a member is a support, then half of the member mass is lumped at the support and will not move during the dynamic response. Use ENFORCED supports to minimize this limitation.

Related Links

- *TR.31.8.2 Reference Load Mass Tables* (on page 2456)
- *TR.31.8.3 Mass Model Using Reference Load* (on page 2457)
- *Create Mass Model dialog* (on page 2870)
- *M. To generate a mass model* (on page 899)
- *TR.32 Loading Specifications* (on page 2461)
- *TR.31.6 Defining Reference Load Types* (on page 2453)
- *TR.28.2 Floor Diaphragm* (on page 2328)
- *TR.31.6 Defining Reference Load Types* (on page 2453)
- *TR.31.8.3 Mass Model Using Reference Load* (on page 2457)
- *TR.31.6 Defining Reference Load Types* (on page 2453)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *TR.32.10.2 Time Varying Load for Response History Analysis* (on page 2593)

G.17.3.3 Damping Modeling

Damping may be specified by entering values for each mode (either explicitly or calculated), by using a formula based on the first two frequencies, or by using composite modal damping. Composite modal damping permits computing the damping of a mode from the different damping ratios for different materials (steel, concrete, soil). Modes that deform mostly the steel would have steel damping ratio, whereas modes that mostly deform the soil, would have the soil damping ratio.

Modeling Methods

Damping Method	Related STAAD.Pro Command
A single specified value, used by all modes	DAMP
Composite damping based on values specified for each material which can include the effect of spring damping, if defined.	CDAMP
Modal damping which is explicitly defined for each mode.	MDAMP
Modal damping which is calculated for all modes.	MDAMP, using either the CALCULATE or EVALUATE method

Related Links

- *TR.37.6.4 Steady Ground Motion Loading* (on page 2634)
- *TR.26.1 Define Material* (on page 2303)
- *TR.37.6.4 Steady Ground Motion Loading* (on page 2634)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.37.6.5 Steady Force Loading* (on page 2635)
- *TR.26.1 Define Material* (on page 2303)

- *TR.37.6.5 Steady Force Loading* (on page 2635)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.37.6.6 Harmonic Ground Motion Loading* (on page 2636)
- *TR.26.1 Define Material* (on page 2303)
- *TR.37.6.6 Harmonic Ground Motion Loading* (on page 2636)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.37.6.8 Print Steady State/Harmonic Results* (on page 2639)
- *TR.26.1 Define Material* (on page 2303)
- *TR.37.6.8 Print Steady State/Harmonic Results* (on page 2639)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.11 Response Spectrum Specification per IBC 2012* (on page 2566)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.11 Response Spectrum Specification per IBC 2012* (on page 2566)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011* (on page 2589)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011* (on page 2589)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81* (on page 2584)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81* (on page 2584)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.10 Response Spectrum Specification per IBC 2006* (on page 2561)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004* (on page 2525)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004* (on page 2525)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994* (on page 2520)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994* (on page 2520)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *TR.26.1 Define Material* (on page 2303)
- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *TR.26.4 Modal Damping Information* (on page 2312)

G.17.3.3.1 Composite Damping

Composite modal damping is based on a weighted average of strain energies in each material, (or element), for each mode (eigenvector).

The critical composite damping term D_J for mode J is computed by:

$$D_i = \frac{\sum_{i=1}^{n} \{\phi_i\}^T b_i \cdot \left[k\right] \cdot \{\phi_i\}}{\{\phi_j\}^T \cdot \left[K\right] \cdot \{\phi_j\}}$$

where

n	=	total number of degrees of freedom
b_i	=	equivalent percent of critical damping associated with component i
$\{\varphi_J\}$	=	mode shape vector for mode J
$[k]_i$	=	stiffness associated with component i
$[K]$	=	stiffness matrix for the system

Related Links

- *Material Constant dialog* (on page 2809)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)
- *M. To assign a composite damping ratio* (on page 895)
- *TR.26.1 Define Material* (on page 2303)
- *TR.26.2 Specifying Constants for Members and Elements* (on page 2305)

G.17.3.3.2 Modal Damping

Explicit Damping

With the `EXPLICIT` option, you must provide unique modal damping values for some or all modes. Each value can be preceded by a repetition factor (rf*damp) without spaces.

Example

```
DEFINE DAMPING INFORMATION
EXPLICIT 0.03 7*0.05 0.04 -
0.012
END
```

In the above example, mode 1 damping is .03, modes 2 to 8 are .05, mode 9 is .04, mode 10 (and higher, if present) are 0.012.

If there are fewer entries than modes, then the last damping entered will apply to the remaining modes. This input may be continued to 10 more input lines with word `EXPLICIT` only on line 1; end all but last line with a space then a hyphen. There may be additional sets of `EXPLICIT` lines before the `END`.

Calculate Damping

The formula used to calculate the damping for modes i = 1 to N per modal frequency based on mass and/or stiffness proportional damping (for `CALCULATE`) is:

$$D(i) = (\alpha / 2\omega_i) + (\omega_i \beta / 2)$$

If the resulting damping is greater than MAX, then MAX will be used (MAX=1 by default). If the resulting damping is less than MIN, then MIN will be used (MIN=1.E-9 by default). This is the same damping as D = (αM + βK).

Example:

```
DEFINE DAMPING INFORMATION
CALC ALPHA 1.13097 BETA 0.0013926
END
```

To get 4% damping ratio at 4 Hz and 6% damping ratio at 12 Hz

Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	0.04
3	12.0	75.398	0.06

$D(i) = (\alpha / 2\omega_i) + (\omega_i \beta / 2)$

$0.04 = \alpha / 50.266 + 12.567\ \beta$

$0.06 = \alpha / 150.796 + 37.699\ \beta$

$\alpha = 1.13097$

$\beta = 0.0013926$

However they are determined, the α and β terms are entered in the CALC data above. For this example calculate the damping ratio at other frequencies to see the variation in damping versus frequency.

Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	0.040
3	12.0	75.398	0.060
	2	12.0664	0.05375
	8	50.2655	0.04650
	20	120.664	0.09200
	4.5	28.274	0.03969

The damping, due to β times stiffness, increases linearly with frequency; and the damping, due to alpha times mass, decreases parabolicly. The combination of the two is hyperbolic.

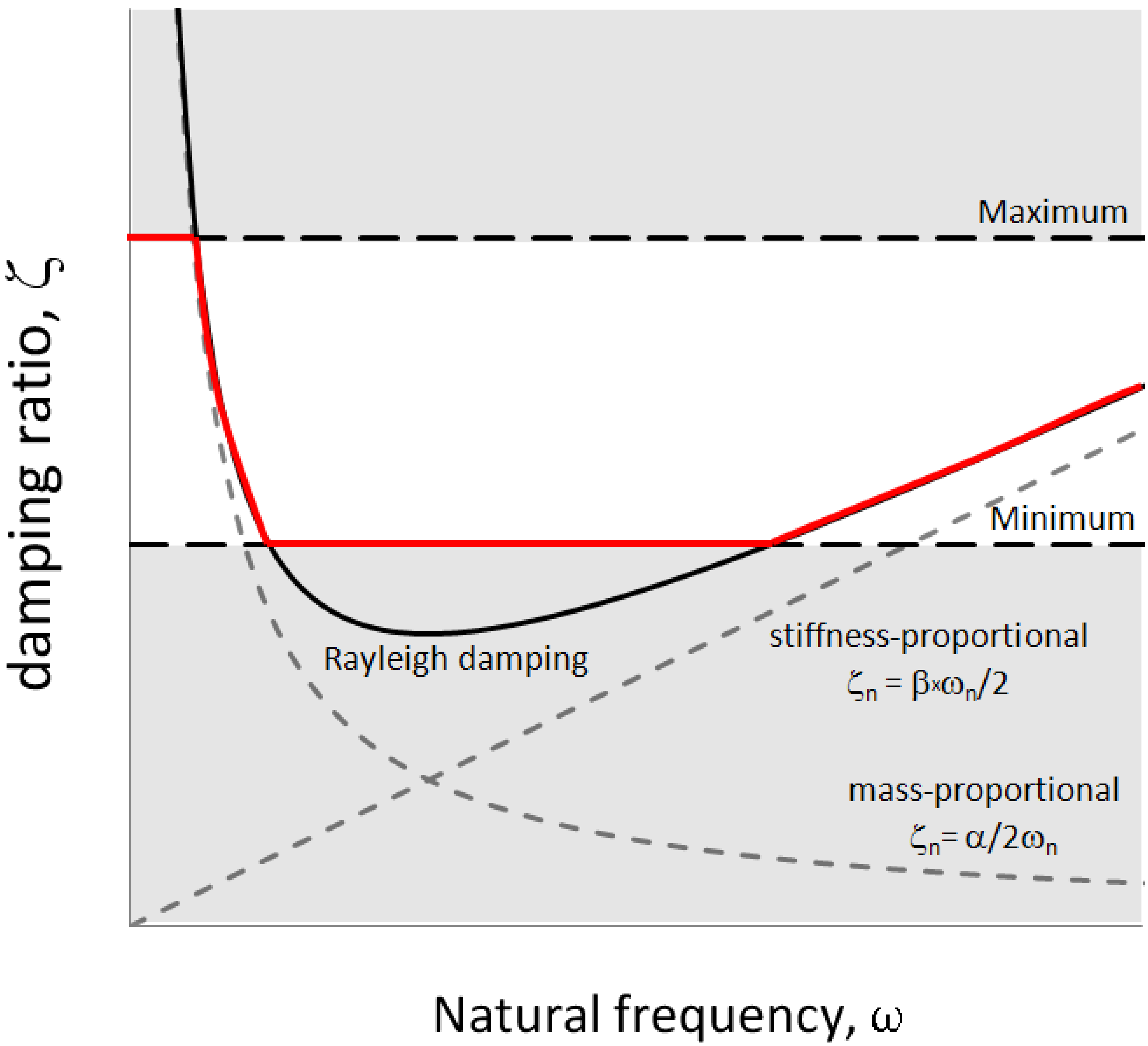


Figure 250: Graph of Damping Ration, D(i), versus Natural Frequency, ω with Max. and Min. values applied.

Evaluate Damping

The formula used for EVALUATE (to evaluate the damping per modal frequency) is:

Damping for the first 2 modes is set to dmin from input.

Damping for modes i = 3 to N given dmin and the first two frequencies ω_1 and ω_2 and the i^{th} modal frequency ω_i.

$$A_1 = \text{dmin} / (\omega_1 + \omega_2)$$

$$A_0 = A_1 * \omega_1 * \omega_2$$

$$D(i) = (A_0 / \omega_i) + (A_1 * \omega_i)$$

If the resulting damping is greater than the dmax value of maximum damping, then dmax will be used.

Example:

```
DEFINE DAMPING INFORMATION
EVALUATE 0.02 0.12
END
```

for dmin = .02 , dmax = .12 and the ω_i given below:

Mode	ω_i	Damping Ratio
1	3	0.0200
2	4	0.0200
3	6	0.0228568
N	100	0.1200 (calculated as .28605 then reset to maximum entered)

Related Links

- *Modal Damping dialog* (on page 2910)
- *TR.26.4 Modal Damping Information* (on page 2312)
- *M. To explicitly define damping values for modes* (on page 895)
- *M. To evaluate damping for modes* (on page 896)
- *TR.26.4 Modal Damping Information* (on page 2312)

G.17.3.4 Response Spectrum

This capability allows you to analyze the structure for seismic loading. For any supplied response spectrum (either acceleration vs. period or displacement vs. period), joint displacements, member forces, and support reactions are calculated for each mode used in the spectrum solution. These individual modal responses are combined using one of the square root of the sum of squares (SRSS), the complete quadratic combination (CQC), the ASCE4-98 (ASCE), the ten percent (TEN), or the absolute (ABS) methods to obtain the resultant responses. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results.

Calculation of Forces and Moments at Intermediate Sections

For static load cases, if there is no load applied within the span of a member, for any given degree of freedom (FX, FY, FZ, MX, MY, and MZ), the force or moment value at intermediate span locations can be calculated by linearly interpolating between the values for that degree of freedom at the start and end nodes of the member.

But for response spectrum load cases, this approach is applied at the individual mode basis following which the modal values are combined using the combination method specified in the input. The details of the procedure are as follows:

For any given member, we define the terms RAP and RBP as:

RAP = The force/moment value of the d.o.f under consideration for mode P at the start node of the member (End A)
RBP = The force/moment value of the d.o.f under consideration for mode P at the end node of the member (End B)

Note: RAP and RBP are quantities with signs because these are at the individual mode level.

Using linear interpolation, calculate the value of that d.o.f at each of 11 equally spaced intermediate sections along the member length.

So, we now define the term RIP as the value of the d.o.f under consideration at section location "I" for mode "P."

If the spectrum solution is based on "N" modes, the resultant value for that d.o.f at section location "I" is obtained as:

$$\mathrm{SRSS(RI1, RI2, RI3, RI4, \ldots, RIN)}$$

or

$$\mathrm{CQC(RI1, RI2, RI3, RI4, \ldots, RIN)}$$

or a similar calculation for the other modal combination methods.

The values calculated in the above fashion can then be obtained in the output file using the `PRINT SECTION FORCES` command and in tabular or graphical form in the post processing mode.

Complete Quadratic Combination Method

This method was first described in "A Replacement for the SRSS Method in Seismic Analysis" (*Earthquake Engineering and Structural Dynamics,* Vol. 9 p.187-192) by E.L. Wilson, et al in 1981. The method used in STAAD.Pro is adapted from the textbook *Three Dimensional Static and Dynamic Analysis Of Structures* by Edward L. Wilson.

Using the CQC method, the peak force is determined using the following:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

f_n	=	the modal force associated with mode n
ρ_{nm}	=	the cross-modal coefficient
		$= \dfrac{8\zeta^2(1+r)r^{3/2}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$
r	=	$\omega_n/\omega_m \leq 1.0$
ζ	=	the damping ratio. If you use a `DEFINE DAMPING INFORMATION` command to define modal damping, that will be used here for the corresponding modes. Otherwise, the constant damping or composite damping for the entire structure will be used.

Similar equations are used to determine displacements and other resultant responses.

Related Links

- *TR.32.10.1.1 Response Spectrum Specification - Custom* (on page 2499)
- *Spectrum Parameters dialog* (on page 2862)
- *Generated Spectrum dialog* (on page 2862)

- *M. To add an IBC 2000 response spectrum* (on page 862)
- *TR.32.10.1 Response Spectrum Analysis* (on page 2498)

G.17.3.5 Response Time History

STAAD.Pro is equipped with a facility to perform a response history analysis on a structure subjected to time varying forcing function loads at the joints and/or a ground motion at its base. This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Refer to G.17.3.2 Mass Modeling (on page 2157) for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes. The equilibrium equations are written as

$$[m]\{x''\} + [c]\{x'\} + [k]\{x\} = \{P(t)\}$$

Note: The double-prime notation ('') designates the second derivative (i.e., acceleration) and a prime notation (') designates the first derivative (i.e., velocity).

Using the transformation

$$\left\{x\right\} = \sum_{i=1}^{p} \{\phi\}_i q_i$$

The equation for *{P(t)}* reduces to "p" separate uncoupled equations of the form

$$q''_i + 2\,\xi_i\omega_i q'_i + \omega_i^2 q_i = R_i(t)$$

where

ξ	=	the modal damping ration
ω	=	the natural frequency for the i^{th} mode.

These are solved by the Wilson- θ method which is an unconditionally stable step by step scheme. The time step for the response is entered by you or set to a default value, if not entered. The q_is are substituted in equation 2 to obtain the displacements *{x}* at each time step.

Time History Analysis for a Structure Subjected to a Harmonic Loading

A Harmonic loading is one in which can be described using the following equation:

$$F(t) = F_0 \sin(\omega t + \phi)$$

where

$F(t)$	=	Value of the forcing function at any instant of time "t"
F_0	=	Peak value of the forcing function
ω	=	Frequency of the forcing function
ϕ	=	Phase angle

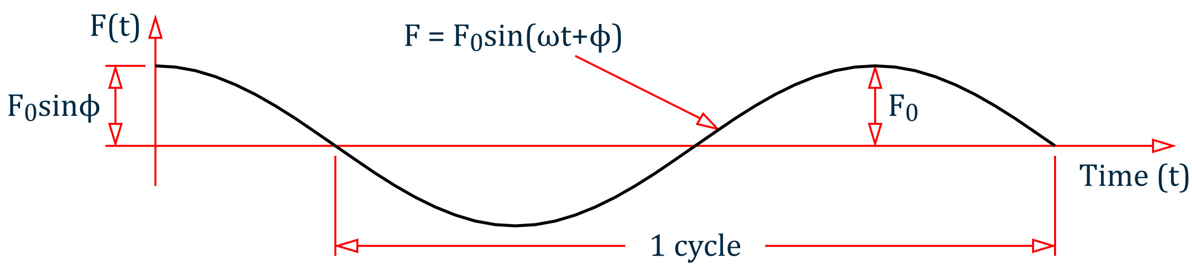


Figure 251: Plot of the harmonic loading function

The results are the maximums over the entire time period, including start-up transients. So, they do not match steady-state response.

Definition of Input in STAAD.Pro for the above Forcing Function

As can be seen from its definition, a forcing function is a continuous function. However, in STAAD.Pro, a set of discrete time-force pairs is generated from the forcing function and an analysis is performed using these discrete time-forcing pairs. What that means is that based on the number of cycles that you specify for the loading, STAAD.Pro will generate a table consisting of the magnitude of the force at various points of time. The time values are chosen from this time '0' to $n \times tc$ in steps of “STEP” where n is the number of cycles and tc is the duration of one cycle. STEP is a value that you may provide or may choose the default value that is built into the program. STAAD.Pro will adjust STEP so that a 1/4 cycle will be evenly divided into one or more steps. See TR. 31.4 Definition of Time History Load (on page 2441) for a list of input parameters that need to be specified for a Time History Analysis on a structure subjected to a Harmonic loading.

The relationship between variables that appear in the STAAD.Pro input and the corresponding terms in the equation shown above is explained below.

F_0 = Amplitude
ω = Frequency
ϕ = Phase

For control/dependent specifications, the forces applied at dependent dof will be ignored. Apply them at the control instead.

Related Links

- *TR.31.4 Definition of Time History Load* (on page 2441)
- *Add New Time History Definitions dialog* (on page 2907)
- *M. To define a time history type from tabular data* (on page 875)
- *M. To define a time history type from a function* (on page 876)
- *M. To define a time history type by spectrum* (on page 877)
- *M. To define a time history type by external file* (on page 879)
- *Define (Time History) Parameters dialog* (on page 2909)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *M. To define time history parameters* (on page 880)
- *Time History tab* (on page 2850)
- *TR.32.10.2 Time Varying Load for Response History Analysis* (on page 2593)
- *M. To add a time history load* (on page 880)
- *TR.31.4 Definition of Time History Load* (on page 2441)
- *TR.32.10.2 Time Varying Load for Response History Analysis* (on page 2593)

G.17.3.6 Steady State and Harmonic Response

A structure [subjected only to harmonic loading, all at a given forcing frequency and with non-zero damping] will reach a steady state of vibration that will repeat every forcing cycle. This steady state response can be computed without calculating the transient time history response prior to the steady state condition.

$$R(t) = R_0 \sin(\omega t + \phi)$$

The result, R, has a maximum value of R_0 and a phase angle ϕ . These two values for displacement, velocity, and acceleration at each joint may be printed or displayed.

This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. See G.17.3.2 Mass Modeling (on page 2157) for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes.

A Harmonic loading is one in which can be described using the following equation:

$$F(t) = F_0 \sin(\omega t + \phi)$$

where

$F(t)$	=	Value of the forcing function at any instant of time "t"
F_0	=	Peak value of the forcing function
ω	=	Frequency of the forcing function
ϕ	=	Phase angle

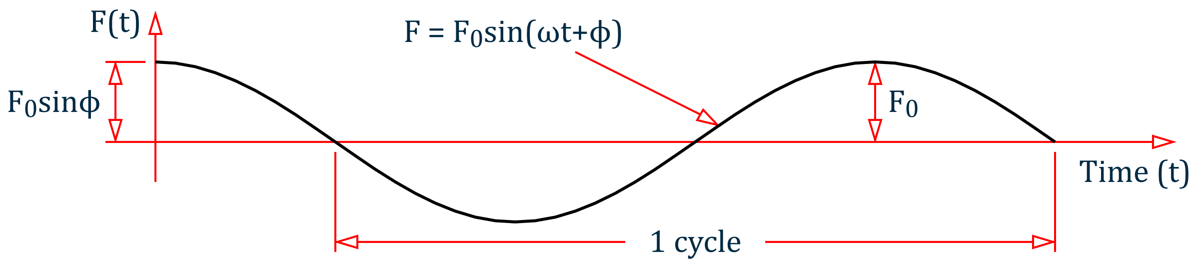


Figure 252: Plot of the harmonic loading function

The results are the steady-state response which is the absolute maximum of displacement (and other output quantities) and the corresponding phase angle after the steady state condition has been reached.

In addition, a Harmonic response can be calculated. This response consists of a series of Steady State responses for a list of frequencies. The joint displacement, velocity, or acceleration can be displayed as the response value versus frequency. Load case results are the maximums over all of the frequencies.

All results are positive as in the Response Spectrum and Time history analyses. This means section results should be ignored (BEAM 0.0 in Parameters for code checking). Because of this, you may want to add the steady state response to Dead & Live loads for one combination case and subtract the steady state response from those loads for another combination case.

Ground motion or a joint force distribution may be specified. Each global direction may be at a different phase angle.

Output frequency points are selected automatically for modal frequencies and for a set number of frequencies between modal frequencies. There is an option to change the number of points between frequencies and an option to add frequencies to the list of output frequencies.

The load case that defines the mass distribution must be the case just before the `PERFORM STEADY STATE ANALYSIS` command. Immediately after that command is a set of data starting with `BEGIN STEADY` and ending with `END STEADY`. The list of additional frequencies and the steady state load cases with joint loads or ground accelerations and phasing data are entered here. The optional print command for the maximum displacement and associated phase angle for selected joints must be at the end of this block of input.

Note: A license for the advanced analysis module is required to access this feature.

Related Links

- *TR.37.6 Steady State and Harmonic Analysis* (on page 2632)

G.17.4 Pushover Analysis

Pushover analysis is a static, nonlinear procedure using simplified nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis used to determine the force-displacement relationship, or the capacity curve, for a structure or structural element.

In STAAD.Pro, the basis for this analysis is the information published in the documents FEMA 356 : 2000 and ATC 40.

Note: A license for the advanced analysis module is required to access this feature.

Related Links

- *Perform Pushover Analysis tab* (on page 2926)
- *A. To specify a pushover analysis* (on page 957)
- *G.17.4.2 Pushover Analysis Engineering Reference* (on page 2178)
- *V. Column Pushover Displacement* (on page 3735)

G.17.4.1 Overview of Pushover Analysis

This section includes a an overview of the pushover analysis procedure as well as some of the concepts involved with how STAAD.Pro implements this form of analysis.

G.17.4.1.1 What is a Pushover Analysis?

A pushover analysis is a static, nonlinear procedure using a simplified, nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis which is used to determine the force-displacement relationship – or the capacity curve – for a structure or structural element.

The analysis involves applying horizontal loads, in a prescribed pattern, to the structure incrementally, i.e., pushing the structure and plotting the total applied shear force and associated lateral displacement at each increment, until the structure reaches a collapse condition or a prescribed limit.

This definition of a pushover analysis is in accordance with both the following references:

- FEMA 356 " Prestandard and Commentary for the Seismic Rehabilitation of Buildings " (2000, American Society of Civil Engineers for Federal Emergency Management Agency)
- ATC-40 " Seismic Evaluation and Retrofit of Concrete Buildings " (1996, Applied Technology Council)

G.17.4.1.2 Purpose of a Pushover Analysis

It is expected that most buildings rehabilitated in accordance with a standard, would perform within the desired levels when subjected to the design earthquakes. Structures designed according to the existing seismic codes provide minimum safety to preserve life and in a major earthquake, they assure at least gravity-load-bearing elements of non-essential facilities will still function and provide some margin of safety. However, compliance with the standard does not guarantee such performance. They typically do not address performance of non-structural components neither provide differences in performance between different structural systems. This is because it cannot accurately estimate the inelastic strength and deformation of each member due to linear elastic analysis. Although an elastic analysis gives a good indication of elastic capacity of structures and indicates where first yielding will occur, it cannot predict failure mechanisms and account for redistribution of forces during progressive yielding.

To overcome this disadvantages different nonlinear static analysis method is used to estimate the inelastic seismic performance of structures, and as the result, the structural safety can be secured against an earthquake. Inelastic analysis procedures help demonstrate how buildings really work by identifying modes of failure and the potential for progressive collapse. The use of inelastic procedures for design and evaluation is an attempt to help engineers better understand how structures will behave when subjected to major earthquakes, where it is assumed that the elastic capacity of the structure will be exceeded. This resolves some of the uncertainties associated with code and elastic procedures.

The practice of earthquake engineering is rapidly evolving to understand the behavior of buildings subjected to strong earthquakes. In order to be able to predict such behavior pushover analysis is performed. The overall capacity of a structure depends on the strength and deformation capacities of the individual components of the structure. In order to determine capacities beyond the elastic limit some form of nonlinear analysis, like Pushover Analysis, is required. It is a new performance based seismic design to achieve analytically a structural design that will reliably perform in a prescribed manner under one or more seismic environment.

G.17.4.1.3 Objective of a Pushover Analysis

Pushover analysis is a performance-based analysis that refers to a methodology in which structural criteria are expressed in terms of achieving a performance objective. This is contrasted to the conventional method in which structural criteria are defined by limits on member forces resulting from a prescribed level of applied shear force.

A performance level describes a limiting damage condition which may be considered satisfactory for a giving building and a given ground motion. The limiting condition is described by the physical damage within the building, the threat to life safety of the building's occupants created by the damage, and the post earthquake serviceability of the building. The basic approach is to improve the probable seismic performance of the building or to otherwise reduce the existing risk to an acceptable level.

Two key elements of a performance-based design procedure are demand and capacity. Demand is the representation of earthquake ground motion or shaking that the building is subjected to. In nonlinear static analysis procedures, demand is represented by an estimation of the displacements or deformations that the structure is expected to undergo. Capacity is a representation of the structure's ability to resist the seismic demand. The performance is dependent on the manner that the capacity is able to handle the demand. In other words, the structure must have the capacity to resist demands of the earthquake such that the performance of the structure is compatible with the objectives of the design. Performance objective is to obtain a desired level of seismic performance of the building, generally described by specifying maximum allowable (or acceptable) structural or nonstructural damage, for a specified level of seismic hazard.

There are two nonlinear procedures using pushover methods :

a. Capacity Spectrum Method
b. Displacement Coefficient Method

G.17.4.1.3.1 Capacity Spectrum Method

The objective of the Capacity Spectrum Method is to develop appropriate demand and capacity spectra for the structure and to determine their intersection point. During this process, performance of each structural component is also evaluated.

The demand curve is based on the earthquake response spectrum and the capacity curves are based on static, nonlinear pushover analysis. In pushover analysis, the structure is subjected to increasing levels of load, and the base shear versus roof displacement of the structure is charted along the way. The capacity spectrum is obtained by transforming the base shear versus roof displacement spectrum into a spectral acceleration versus spectral displacement spectrum. The intersection of an appropriate demand curve with the capacity curve is called the performance point. The performance point defines the estimated base shear and displacement of the structure when subjected to the earthquake represented by the demand curve. The behavior of the structure at the performance point is compared with predefined acceptance criteria to determine if the design objective is met.

Capacity (Pushover) Curve

The structure capacity is represented by a pushover curve, often termed as capacity curve. This represents the lateral displacement as a function of the force applied to the structure. The most convenient way to plot the force displacement curve is by tracking the base shear and roof displacement.

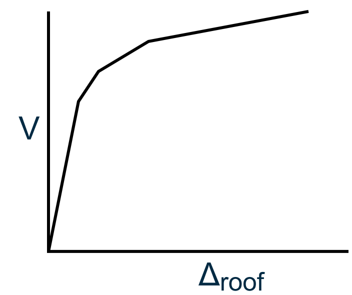


Figure 253: Roof deflection, Δ_{roof}, plotted versus base shear, V

Capacity Spectrum

It is the representation of spectral acceleration vs. spectral displacement derived from the capacity curve. The base shear and the roof displacement are converted to spectral acceleration and spectral displacement respectively by the procedure defined by FEMA to get the capacity spectrum from capacity curve.

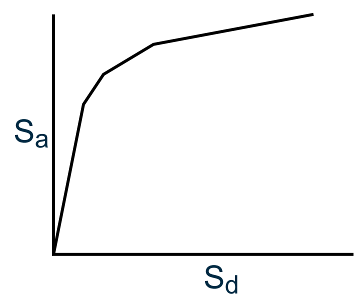


Figure 254: Spectral displacement, S_d plotted versus spectral acceleration, S_a

Conversion of Capacity Curve to Capacity Spectrum

Capacity curve, in terms of base shear and roof displacement, is converted to capacity spectrum, which is a representation of the capacity curve in Acceleration Displacement Response Spectra (ADRS) format (i.e., S_a versus S_d).

The required equations for conversion are the following:

$$PF_1 = \left[\frac{\sum_{i=1}^{N} (w_i \phi_{i1}) / g}{\sum_{i=1}^{N} (w_i \phi_{i1}^2) / g} \right] \quad (1\text{-}3\text{-}1)$$

$$a_1 = \frac{\left[\sum_{i=1}^{N}\left(w_i\phi_{i1}\right)/g\right]}{\left[\sum_{i=1}^{N} w_i/g\right]\left[\sum_{i=1}^{N} w_i\phi_{i1}^2/g\right]} \tag{1-3-2}$$

$$S_a = VW/\alpha_1 \tag{1-3-3}$$

$$S_d = \Delta_{roof}/(PF_1\varphi_{roof,l}) \tag{1-3-4}$$

where

PF_1	=	Modal participation factor for the first natural mode.
a_1	=	Modal mass coefficient for the first natural mode.
w_i/g	=	Mass assigned to level i.
φ_{i1}	=	Amplitude of mode 1 at level i.
N	=	Level N, the level which is the uppermost in the main portion of the structure.
V	=	Base shear
W	=	Building dead weight plus likely live loads.
Δ_{roof}	=	Roof displacement (V and the associated make up points on the capacity curve).
S_a	=	Spectral acceleration
S_d	=	Spectral displacement (and the associated make up points on the capacity spectrum).

Demand spectrum

This curve is obtained by redrawing the design earthquake response spectra as a curve of spectral acceleration vs spectral displacement.

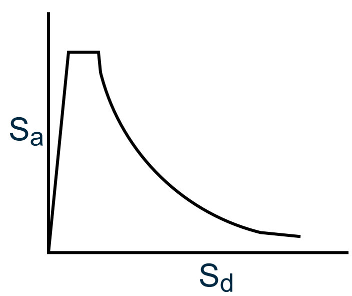


Figure 255: Response spectrum

Performance

Once capacity and demand spectra are defined, a performance check can be one. A performance check verifies that structural and nonstructural components are not damaged beyond the acceptable limits of the performance objective for the forces and displacements implied by the displacement demand.

G.17.4.1.3.2 Displacement Coefficient Method

The objective of the Displacement Coefficient Method is to find the target displacement which is the maximum displacement that the structure is likely to be experienced during the design earthquake. This is equivalent to the performance point in the Capacity Spectrum method. It provides a numerical process for estimating the displacement demand on the structure, by using a bilinear representation of capacity curve and a series of modification factors, or coefficients, to calculate a target displacement.

The structure, directly incorporating the nonlinear load-deformation characteristics of individual components and elements of the building, is subjected to monotonically increasing lateral loads representing inertia forces in an earthquake until a target displacement is exceeded. The damage state comprises deformations for all elements in the structure. Comparison with acceptability criteria for the desired performance goal leads to the identification of deficiencies for individual elements. Performance check at the expected maximum displacement is done to verify whether the lateral force resistance has not degraded by more than a desired percentage (generally 20%) of the peak resistance and the lateral drifts satisfy limits are per standard code.

Target Displacement, δ_t

The target displacement is calculated as per procedure described in Section 3.3.3.3.2 of FEMA 356 : 2000.

It is given by the following expression:

$$\delta_t = C_0 C_1 C_2 C_3 S_a [T_e^2/(4\pi^2)] g$$

where

C_0	=	Modification factor to relate spectral displacement to building roof displacement, as determined by Table 3-2 of FEMA 356.
C_1	=	Modification factor to relate expected maximum inelastic displacements to displacements calculated for linear elastic response $= 1.5$ for $T_e < 0.1$ sec $= 1.0$ for $T_e \geq T_s$ $= [1.0 + (R - 1) T_s / T_e] / R$ for $T_e < T_s$ Value of C1 should not be less than 1.0.
T_s	=	Characteristic period of the response spectrum , defined as period associated with transition from const accleration segment of the spectrum to the constant velocity segment of the spectrum (to be calculated from demand spectrum)
T_e	=	Effective fundamental time period $= T_i(K_i/K_e)^{1/2}$
T_i	=	Elastic fundamental period
K_i	=	Elastic lateral stiffness of the building
K_e	=	Effective lateral stiffness of the building. Taken as equal to the secant stiffness calculated at a base shear force equal to 60% of the effective yield strength of the Structure obtained from bilinear representation of Capacity Curve.
R	=	Ratio of elastic strength demand to calculated yield strength coefficient $= S_a/(V_y/W)C_m$
V_y	=	Effective yield strength calculated using the capacity curve . For larger elements or entire structural systems composed of many components, the effective yield point represents the point at which a sufficient number of individual components or elements have yielded and the global structure begins to experience inelastic deformation.
S_a	=	Response spectrum acceleration, at the effective fundamental period and damping ratio of the building (to be calculated from demand spectrum)
W	=	Effective seismic weight
C_m	=	Effective mass factor as determined by Table 3-1 of FEMA 356.
C_2	=	Modification factor to represent the effect of pinched hysteretic shape , stiffness degradation and strength deterioration on maximum displacement response. Taken from Table 3-3 of FEMA 356 for different

		framing systems and structural performance levels. Alternatively, C_2 may be taken as 1.0 for a nonlinear procedure.
C_3	=	Modification factor to represent increased displacement due to dynamic P-D effects
		= 1.0 for buildings with positive post - yield stiffness
		= $1.0 + \|\alpha\|(R - 1)^{3/2}/T_e$ for buildings with negative post - yield stiffness
α	=	Ratio of post - yield stiffness to effective elastic stiffness , where the non-linear force-displacement relation shall be characterized by a bilinear relation

Refer to Figure 3-1 of FEMA 356 for idealized force-displacement curves.

G.17.4.1.4 Types of Nonlinearity

Both geometric and material nonlinearities are considered in this static nonlinear pushover analysis.

Geometric Nonlinearity

This is a type of nonlinearity where the structure is still elastic, but the effects of large deflections cause the geometry of the structure to change, so that linear elastic theory breaks down. Typical problems that lie in this category are the elastic instability of structures, such as in the Euler bulking of struts and the large deflection analysis of a beam-column member. In general, it can be said that for geometrical non-linearity, an axially applied compressive force in a member decreases its bending stiffness, but an axially applied tensile force increases its bending stiffness. In addition, the P-Delta effect is also included in this concept.

Material Nonlinearity

In this type of nonlinearity, the material undergoes plastic deformation. Material nonlinearity can be modeled as discrete hinges at a number of locations along the length of a frame (beam or column) element and a discrete hinge for a brace element as discrete material fibers distributed over the cross-section of the element, or as a series of material points throughout the element.

Related Links

- *M. To manually define and assign hinges* (on page 882)
- *TR.37.7.5.1 User-Defined Hinge Property* (on page 2650)
- *TR.37.7.5.2 Assignment of Hinge Property to the Members* (on page 2651)
- *Define Hinge Property tab* (on page 2904)

G.17.4.1.5 Force controlled and deformation controlled actions

Force-controlled refers to components, elements, actions, or systems which are not permitted to exceed their elastic limits. This category of elements, generally referred to as brittle or nonductile, experiences significant degradation after only limited post-yield deformation.

Deformation-controlled refers to components, elements, actions, or systems which can, and are permitted to, exceed their elastic limit in a ductile manner. Force or stress levels for these components are of lesser importance than the amount of deformation beyond the yield point.

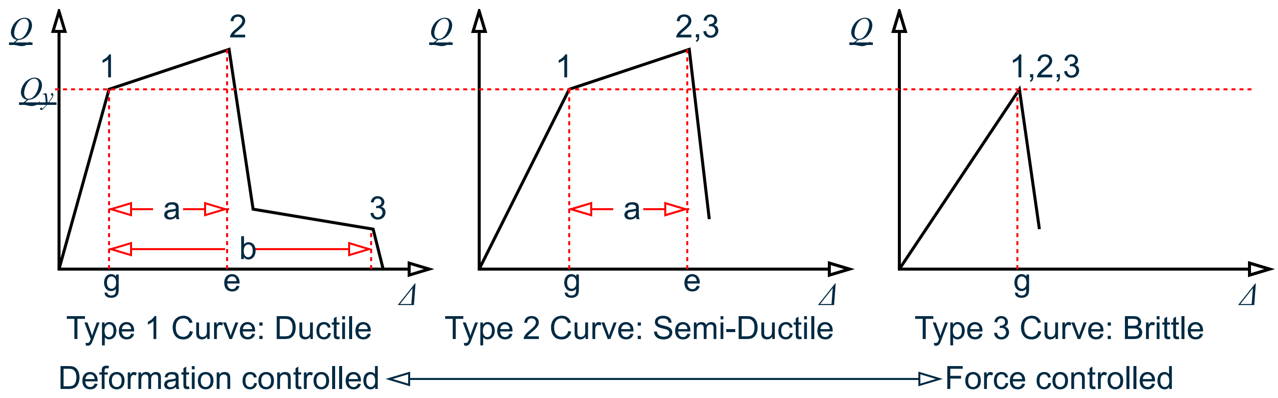


Figure 256: Component force versus deformation curves (Ref. Fig. 2-3 FEMA 356)

Note: Refer to Section 2.4.4.3.1 of FEMA 356 for detailed information on these curves.

G.17.4.1.6 Frame element hinge properties

Discrete hinge properties for frame elements are usually based on FEMA-356 criteria. As per section 5.5.2.2.2 of FEMA 356, in lieu of relationships derived from experiment or analysis, the generalized load deformation curve shown in the figure below, with parameters a, b, c, as defined in tables1.5.6 and 1.5.7, shall be used for components of steel moment frames. Modification of this curve shall be permitted to account for strain-hardening of components as follows:

a. a strain-hardening slope of 3% of the elastic slope shall be permitted for beams and columns unless a greater strain-hardening slope is justified by test data; and
b. where panel zone yielding occurs, a strain-hardening slope of 6% shall be used for the panel zone unless a greater strain-hardening slope is justified by test data.

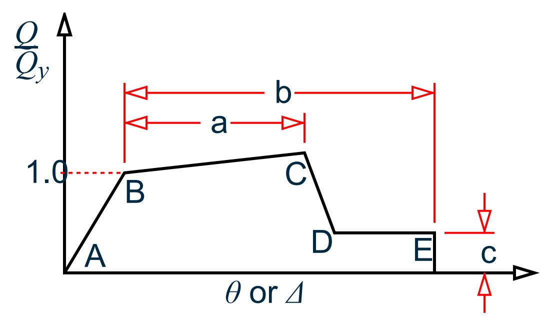


Figure 257: Generalized Force-Deformation Relationship for Components

- Point A is the origin
- Point B represents yielding. No deformation occurs in the hinge up to point B, regardless of the deformation value specified for point B. The displacement (rotation or axial elongation as the case may be) will be subtracted from the displacements at points C, D and E. Only plastic deformation beyond point B will be exhibited by hinge.
- Point C represents ultimate capacity of plastic hinge. At this point hinge strength degradation begins (hinge starts shedding load) until it reaches point D.
- Point D represents the residual strength of the plastic hinge. Beyond point D the component responds with substantially strength to point E.

- Point E represents total failure. At deformation greater than point E the plastic hinge will drop load to zero.

The parameters Q and Q_{CE} (Q_y) in Figure 1-6 are generalized component load and generalized component expected strength, respectively. For beams and columns, θ is the total elastic and plastic rotation of the beam or column, θ_y is the rotation at yield. For braces Δ is total elastic and plastic displacement, and Δ_y is yield displacement.

Use of equations (1-6-1) and (1-6-2) to calculate the yield rotation, θ_y, where the point of contraflexure is anticipated to occur at the mid-length of the beam or column, respectively, shall be permitted.

For beams:

$$\theta_y = Z \cdot F_{ye} L_b / (6 \cdot EI_b) \tag{1-6-1}$$

For columns:

$$\theta_y = Z \cdot F_{ye} L_c / (6 \cdot EI_c)\ (1 - P/P_{ye}) \tag{1-6-2}$$

Q and Q_{CE} are the generalized component load and generalized component expected strength, respectively. For flexural actions of beams and columns, Q_{CE} refers to the plastic moment capacity, which shall be calculated using equations (1-6-3) and (1-6-4):

For beams:

$$Q_{CE} = M_{CE} = Z \cdot F_{ye} \tag{1-6-3}$$

For columns:

$$Q_{CE} = M_{CE} = 1.18\, Z \cdot F_{ye}\ (1 - P/P_{ye}) \tag{1-6-4}$$

where

E	=	Modulus of elasticity
F_{ye}	=	Expected yield strength of the material
I	=	Moment of inertia
L_b	=	Beam length
L_c	=	Column length
M_{CE}	=	Expected flexural strength of a member or Joint, kip-in.
P	=	Axial force of the member
P_{ye}	=	Expected axial yield force of the member = $A_g F_{ye}$
Q	=	Generalized component load
Q_{CE}	=	Generalized component expected strength = Effective expected strength, which is defined as the statistical mean value of yield strengths, Q_y, for a population of similar components, and includes consideration of strain hardening and plastic section development.

Related Links

- *M. To manually define and assign hinges* (on page 882)
- *TR.37.7.5.1 User-Defined Hinge Property* (on page 2650)
- *TR.37.7.5.2 Assignment of Hinge Property to the Members* (on page 2651)
- *Define Hinge Property tab* (on page 2904)
- *P. To review pushover beam results* (on page 2001)
- *Capacity Curve graph* (on page 2013)

G.17.4.1.6.1 Calculation of Q_{CE}

The expected flexural strength is evaluated by the following means for beam or column elements.

Beams

The strength of elements of structural steel under flexural actions with negligible axial load present is calculated in accordance with this section.

The expected flexural strength, Q_{CE}, of beam components is determined using G.17.4.1.6 Frame element hinge properties (on page 2174). For nonlinear procedure flexural actions of beams is considered deformation controlled. Permissible plastic rotation deformation is as indicated in equations G.17.4.1.6 Frame element hinge properties (on page 2174) and G.17.4.1.6 Frame element hinge properties (on page 2174) (Ref. Sections 5.5.2.2.2 and 5.5.2.3.2 of FEMA 356).

Columns

This section is used to evaluate flexural and axial strengths of structural steel elements with non-negligible axial load present. These actions shall be considered force controlled.

The lower-bound strength, Q_{CL}, of steel columns under axial compression is taken to be the lowest value obtained for the limit states of column buckling, local flange buckling, or local web buckling. The effective design strength or the lower-bound axial compressive strength, P_{CL}, shall be calculated in accordance with the LRFD method, taking φ = 1.0 and using the lower-bound strength, F_{yLB}, for yield strength. The expected axial strength of a column in tension, Q_{CE}, is computed in accordance with equation (1-6-1-1) (Ref. Section 5.5.2.3.2 of FEMA 356):

$$Q_{CE} = T_{CE} = A_c F_{ye} \tag{1-6-1-1}$$

where

A_c	=	Area of column
F_{ye}	=	Expected yield strength of column
T_{CE}	=	Expected tensile strength of column
F_{yLB}	=	Lower-bound yield strength

Flexural loading of columns, with axial loads at a target displacement less than 50% of PCL shall be considered deformation controlled and maximum permissible plastic rotation demands on columns, in radians, shall be as indicated in tables 1.5.1 and 1.5.2, dependent on the axial load present and the compactness of the section.

Flexural loading of columns, with axial loads at the target displacement greater than or equal to 50% of PCL, shall be considered force-controlled and shall conform to equation (1-6-1-2) (Ref. Section 5.5.2.4.2 of FEMA 356).

$$P_{UF}/(2 \cdot P_{CL}) + M_x/(m_x M_{CEx}) + M_y/(m_y M_{CEy}) \leq 1.0 \tag{1-6-1-2}$$

where

P_{UF}	=	Axial force in the member
P_{CL}	=	Lower bound compression strength of the column
M_x	=	Bending moment in the member for the x-axis
M_y	=	Bending moment in the member for the y-axis
M_{CEx}	=	Expected bending strength of the column for the x-axis
M_{CEy}	=	Expected bending strength of the column for the y-axis
m_x	=	Value of m for the column bending about the x-axis in accordance with Table 5-5 of FEMA 365
m_y	=	Value of m for the column bending about the y-axis in accordance with Table 5-5 of FEMA 365

G.17.4.1.7 Elements

Major horizontal or vertical portions of the building’s structural systems that act to resist lateral forces or support vertical gravity loads such as frames, shear walls, frame-walls, diaphragms, and foundations.

Primary Elements

These are structural components or elements that provide a significant portion of the structure's lateral force resisting stiffness and strength at the performance point. These are the elements that are needed to resist lateral loads after several cycles of inelastic response to the earthquake ground motion.

Secondary Elements

These are structural components or elements that are not, or are not needed to be, primary elements of the lateral load resisting system. However, secondary elements may be needed to support vertical gravity loads and may resist some lateral loads.

G.17.4.1.8 Lateral Load Distribution

Lateral loads can be applied by any one of the following three methods per Section 3.3.3.2.3 of FEMA 356.

Method 1

The vertical distribution of the base shear shall be as specified in this section for all buildings. The lateral load applied at any floor level x shall be determined in accordance with equation (1-8-1) and equation (1-8-2):

$$F_x = C_{vx}V \tag{1-8-1}$$

where

$$C_{vx} = \frac{w_x h_x^k}{\sum_{i=1}^{n} w_i h_i^k} \tag{1-8-2}$$

Method 2

A vertical distribution proportional to the shape of the fundamental mode in the direction under consideration is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

$$F_x = \frac{w_x \phi_x}{\sum_{i=1}^{n} w_i \phi_i} \tag{1-8-3}$$

Method 3

A vertical distribution is performed consisting of lateral forces at each level proportional to the total mass at each level .

$$F_x = \frac{w_x}{\sum_{i=1}^{n} w_i} \tag{1-8-4}$$

where

C_{vx} = Vertical distribution factor

k = Linear interpolation shall be used to calculate values of for intermediate values of k for intermediate values of T.

= 2.0 for T ≥ 2.5 seconds

= 1.0 for T ≤ 0.5 seconds

V	=	User-defined base shear
w_i	=	Portion of the total building weight W located on or assigned to floor level i
w_x	=	Portion of the total building weight W located on or assigned to floor level x
h_i	=	Height (in ft) from the base to floor level i
h_x	=	Height (in ft) from the base to floor level x
F_x	=	Amplitude of mode a floor level x

Related Links

- *M. To add a pushover loading* (on page 884)
- *TR.37.7.3 Define Loading Pattern* (on page 2647)
- *TR.37.7.8 Pushover Loading Input* (on page 2653)
- *Define Loading Pattern tab* (on page 2902)

G.17.4.2 Pushover Analysis Engineering Reference

This section contains technical references on pushover analysis and how this has been implemented in STAAD.Pro.

Note: An Advanced Analysis license is required to access this feature.

Related Links

- *G.17.4 Pushover Analysis* (on page 2168)

G.17.4.2.1 Performing Pushover Analysis

Pushover analysis in STAAD.Pro is a static, non-linear procedure in accordance with FEMA 356 specification. Basically, in this method, the magnitude of the lateral push load is increased progressively according to a predefined loading pattern until either loading or the deflection reaches the described level.

Pushover analysis as it is currently implemented in STAAD.Pro is limited in application to buildings that are regular and do not have adverse torsional or multimode effects. The capacity curve is generally constructed to represent the first mode response of the structure based on the assumption that the fundamental mode of vibration is the predominant response of the structure.

G.17.4.2.1.1 Define Steel Moment and Braced Frames

Steel moment frames are those frames that develop their seismic resistance through bending of steel beams and columns and moment resisting beam-column connections. Steel braced frames are those frames that develop their seismic resistance primarily through axial forces in the components.

User must specify whether the structure is a moment frame or braced frame. By default the program considers it as moment frame.

Note: Currently only fully restraint (FR) moment frame and CBF (Concentric Braced frame) frame are considered.

Related Links

- *M. To define general pushover data* (on page 881)
- *TR.37.7.2.1 Type of Frame* (on page 2643)
- *Define Input tab* (on page 2900)

G.17.4.2.1.2 Define Gravity Loading

Gravity loads include dead loads and (typically) most live loads. Dead load can be taken as the calculated structure self-weight without load factors, plus realistic estimates of flooring, ceiling, partition and other structural and nonstructural components. Live loads should be evaluated for each structure; with consideration given to current and expected future occupancies.

G.17.4.2.1.3 Define Lateral (Push) Loading

The mathematical model directly incorporating the nonlinear load-deformation characteristics of individual components and elements of the building shall be subjected to monotonically increasing lateral loads representing inertia forces in an earthquake until a target displacement is exceeded.

A static, nonlinear pushover analysis usually requires multiple analyses cases. The first pushover load case is gravity load applied to the structure. The rest of the load cases may apply different lateral loads in terms of push load increments, whatever the case may be.

A pushover case may start from zero initial conditions or it may start from the results at the end of a previous pushover case. Thus, the gravity load case starts from zero initial conditions. The first lateral load case will start from the end of the gravity load case, the second lateral load case will start from the end of the first lateral load case, and so on until the target displacement is exceeded.

Note: When defining the incremental push load, you should take care to make its value small enough since larger values of incremental loading can prevent the analysis converging.

The lateral loads shall be applied in both positive and negative directions since it may lead to different results for asymmetric structures.

Related Links

- *M. To add a pushover loading* (on page 884)
- *TR.37.7.3 Define Loading Pattern* (on page 2647)
- *TR.37.7.8 Pushover Loading Input* (on page 2653)
- *Define Loading Pattern tab* (on page 2902)

Lateral Loading Pattern

Lateral loads should be applied in predetermined patterns that represent predominant distributions of lateral inertial loads during earthquake response.

Distribution of lateral load must be applied to the structure when performing a pushover analysis.

Typically push load is defined in any one of the following:

a. User defined static load pattern
b. User defined base shear to be distributed vertically

Incremental push load ΔP is calculated by using any of the following two methods:

a. You define Push load. In other words, you specify the incremental push load pattern on the structure by defining lateral load at nodes.
b. Or, you define the base shear which is distributed laterally as per methods described in Section 1.4.1. The lateral load at each floor is again divided by the number of load step increment to get actual push load incremental load. Thus:

$$\Delta P = V/N$$

where

V	=	Lateral load distributed from user defined base shear.
N	=	Total number of load step

The actual load acting on the structure at any load step $i = \Delta P_i = \Delta P \cdot S_{pi}$

where

S_{pi}	=	Stiffness Parameter at ith iteration
		= Slope of the capacity curve at (i-1)th iteration / Initial slope of the capacity curve

During linear stage (i.e., all members in the structure are linear), the stiffness parameter is 1.0. Whenever any member becomes nonlinear the stiffness parameter decreases since slope of the capacity curve becomes less than that during elastic stage. Thus, actual lateral load acting on the nonlinear structure at any load increment stage is less than that during linear stage.

Note: Currently, you must define gravity load case as primary loading. First lateral load case starts from the end of the gravity load case.

G.17.4.2.1.4 Define Primary/Secondary Elements and Components

Elements and components that provide the capacity of the structure to resist collapse under seismic forces induced by ground motion in any direction shall be classified as primary. Other elements and components shall be classified as secondary.

In a typical building, nearly all elements, including many nonstructural components, will contribute to the building;s overall stiffness, mass and damping, and consequently its response to the earthquake ground motion. However, not all these elements are critical to the ability of the structure to resist collapse when subjected to strong ground shaking. The secondary designation will be used when a component or element does not contribute significantly or reliably in resisting earthquake effects because of low lateral stiffness, strength or deformation capacity.

Note: Currently all elements are considered as primary elements.

G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria

At the beginning of the analysis user needs to define properties (Refer Section 1.3) and acceptance criteria for all pushover hinges. The program includes several built-in default hinge properties that are based on FEMA 356 tables 1.6.1 (Table 5-6 Pg 5-40 of FEMA 356) and 1.6.2 (Table 5-7 Pg 5-44 of FEMA 356) for steel structures.

While generating default in-built hinge properties following points have been considered.

- **Columns in axial tension in braced frame with negligible moment -** It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. Mz and My moment are negligible.
- **Columns in axial compression in braced frame with negligible moment -** It is considered as force-controlled action as per Table 1-5-1 of FEMA 356. Mz and My moment are negligible.
- **Columns in axial tension in braced and moment frame with moment -** As per FEMA 356 Pg 5-17, for steel columns under axial tension or combined axial tension and bending shall be considered deformation controlled. It will simply act as beam.
- **Columns in axial compression in braced and moment frame with moment -** As per FEMA 356 Pg 5-17, for steel columns under combined axial compression and bending stress, the column shall be considered deformation-controlled for flexural behavior, force-controlled for compressive behavior. i.e.

  a. if Fx / Pcl <= 0.5 consider deformation-controlled

   b. if Fx / Pcl > 0.2 and Fx / Pcl <= 0.5 consider deformation-controlled
   c. if Fx / Pcl > 0.5 consider force-controlled

   where Fx is axial compressive force and Pcl is lower bound compressive strength of column. (Refer Section 1.6.1 for detail).

- **Beams in axial tension in braced frame with negligible moment -** It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. Mz moment is negligible.
- **Beams in axial compression in braced frame with negligible moment -** It is considered as force-controlled action as per Table 1-5-1 of FEMA 356. Mz moment is negligible.
- **Beams in axial compression and tension in moment and braced frame with moment -** It is considered as deformation-controlled action as per Table 5-6 Pg 5-40 of FEMA 356.
- **Braces in axial tension and compression in braced frame -** It is considered as deformation-controlled action as per Table 5-7 Pg 5-44 of FEMA 356. For braces in compression out-of-plane is considered.

Please note that while generating built-in hinge properties in STAAD.Pro, the following components are *not* considered:

- Column panel zones
- Fully restrained moment connections
- Partially restrained moment connection
- EBF
- Steel plate shear walls

If any member has releases, the effect in member nonlinear stiffness is considered to be same as that for linear member stiffness.

Related Links

- *M. To manually define and assign hinges* (on page 882)
- *TR.37.7.5.1 User-Defined Hinge Property* (on page 2650)
- *TR.37.7.5.2 Assignment of Hinge Property to the Members* (on page 2651)
- *Define Hinge Property tab* (on page 2904)

Define Acceptance Criteria

The performance of a structure and its components is defined by the acceptance criteria to provide desirable information for evaluation or retrofit. It refers to the specific limiting values for the deformations and loadings, for deformation-controlled and force-controlled components respectively, which constitute for acceptable seismic performance. Following three criteria are there:

Immediate Occupancy (IO)	The post earthquake structural damage state in which only very limited structural damage has occurred. The basic vertical and lateral force resisting systems of the building retain nearly all of their pre-earthquake characteristics and capacities. The risk of life-threatening injury from the structural failure is negligible
Life Safety (LS)	The post earthquake damage state in which significant damage to the structure may have occurred but in which some margin against either total or partial structural collapse remains.
Collapse Prevention (CP)	The post earthquake structural damage state in which the building's structural system is on the verge of experiencing partial or total collapse.

The program considers only moment hinge for beam and column for steel structures. If you do not define hinge properties, the program will consider built-in default hinge properties based on FEMA 356.These automatic hinge properties include both moment and axial hinge.

G.17.4.2.1.6 Define Pushover Analysis Solution Control

Pushover analysis will continue until any of the following three conditions is satisfied:

i. Cumulative base shear is less than or equal to the base shear defined by the user

   You must define base shear up till which pushover analysis will be performed since design base shear (specific to particular seismic code) excludes non-linear effect. When the structure is subjected to strong earthquake the actual base shear may be very high compared to the design base shear. Under this condition there is no guarantee that the structure will maintain desired performance level. This option is chosen when the magnitude of base shear is known and the structure will be able to support that load.

ii. Displacement at the control joint in the specified direction exceeds specified displacement

   This option is chosen when the amount of displacement is known i.e. how far the structure will move but the amount of base shear that the structure will be subjected to is not known. While defining this option please make sure that the displacement component chosen at the control joint increases monotonically during loading. The control node shall be located at the center of mass at the roof of the building.

iii. The structure becomes unstable

   This happens whenever hinge formation is such that it renders the structure on the verge of collapse. If neither base shear nor displacement at control joint is known, define a higher value for both these options. During analysis instability will arise due to collapse of different members and make the structure unstable.

Related Links

- *M. To define solution control* (on page 884)
- *TR.37.7.4 Define Solution Control* (on page 2649)
- *Define Solution Control tab* (on page 2906)

G.17.4.2.1.7 Define Input for Demand Spectrum

Demand Spectrum is generated according to the method described in Section 1.6.1.5 of FEMA 356 : 2000.

The program generates Demand Spectrum for the purpose of finding T_s and S_a (corresponding to T_e for the purpose of calculating target displacement).

Related Links

- *M. To define pushover spectral data* (on page 883)
- *TR.37.7.6 Define Spectral Parameters* (on page 2652)
- *Define Spectrum Details tab* (on page 2904)

Generation of Demand Spectrum
The following criteria is used to develop the demand spectrum for a pushover analysis.

Site Classes

A. Hard rock with average shear wave velocity, v_s > 5,000 ft/sec
B. Rock with 2,500 ft/sec < v_s < 5,000 ft/sec
C. Very dense soil and soft rock with 1,200 ft/sec < v_s ≤ 2,500 ft/sec or with either standard blow count N > 50 or undrained shear strength s_u > 2,000 psf
D. Stiff soil with 600-ft/sec< v_s ≤ 1,200 ft/sec, with 15< N ≤ 50, or 1,000 psf ≤ s_u < 2,000 psf
E. Any profile with more than 10 feet of soft clay defined as soil with plasticity index PI > 20, or water content w > 40 percent, and s_u < 500 psf or a soil profile with v_s < 600 ft/sec.
F. Soils requiring site specific evaluations

Adjustment for site class

The design short period spectral response acceleration parameter, S_{XS}, and the design spectral response acceleration parameter at one second, S_{X1}, shall be obtained from equations (2-7-1) and (2-7-2), respectively, as follows:

$$S_{XS} = F_a S_S \tag{2-7-1}$$

$$S_{X1} = F_v S_1 \tag{2-7-2}$$

where

F_a and F_v	=	site coefficients determined respectively from Tables 1-4 and 10-5 in FEMA 365, based on the site class and the values of the response acceleration parameters S_S and S_1 for the selected return period.

General Horizontal Response Spectrum

A general response spectrum as shown in Figure 1-1 of FEMA 365 shall be developed using equations (2-6-3), (2-6-4) and (2-6-5) for spectral response acceleration, S_a, versus structural period, T, in the horizontal direction.

For $0 < T < T_s$,

$$S_a = S_{xs}\left[\left(\frac{5}{B_s} - 2\right)\frac{T}{T_s} + 0.4\right] \tag{2-7-3}$$

for $T < T_s$

$$S_a = S_{XS}/B_S \tag{2-7-4}$$

for $T > T_S$

$$S_a = S_{X1}/(B_1 T) \tag{2-7-5}$$

where

T_S	=	$S_{X1}B_S/(S_{XS}B_S)$	(2-7-6)
T_0	=	$0.2 \cdot T_S$	(2-7-7)

Now this response spectrum with S_a vs T is converted to demand spectrum with S_a vs S_d. (in case of Capacity Spectrum method)

Note: The program generates Demand Spectrum for the purpose of finding T_S and S_a (corresponding to T_e for the purpose of calculating target displacement).

G.17.4.2.1.8 Define Any Other Input

There are several other inputs which may be required for pushover analysis.

See TR.37.7 Pushover Analysis (on page 2643) for additional details.

G.17.4.2.1.9 Hinge Formation and Hinge Unloading

This section describes how the program handles the formation of hinges and the resulting redistribution of load.
Moment Diagrams With Hinges
For each load increment, member sectional forces are checked with section capacity in order to check formation of hinge. If sectional force exceeds section capacity, hinge formation starts. This implies member lies on or beyond point B on the load deformation curve. The point B in the load deformation curve denotes the yield point of the hinge. The hinge is assumed to be rigid between points A and B until it yields.

When the hinge reaches the deformation denoted by point C, it begins to lose load carrying capacity. When it reaches the deformation point E, the hinge loses all of its load carrying capacity.

Total numbers of 13 sections along the length of member are scanned for M_z and M_y moments. Maximum moment is located and checked with section capacity. If the sectional force exceeds section capacity the material starts yielding at that particular location and the hinge at this particular position lies on point B on load deformation curve.

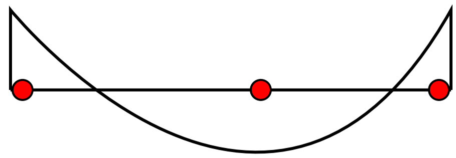


Figure 258: Typical beam moment diagram under gravity load, with super-imposed hinges

If bending moment diagram is like figure 1.8 (this happens at initial stage of load increment when push load is much low as compared to dead load) the chance of forming moment hinge is at two ends and at span of the member (as shown by red dots).

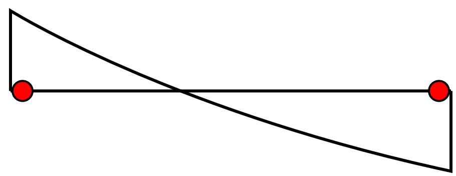


Figure 259: Typical beam moment diagram under significant lateral load, with super-imposed hinges

If bending moment diagram is like figure 1.9 (this happens when push load is much high as compared to dead load) the chance of forming moment hinge is only at sections at or nearer to two ends of the member (as shown by red dots).

Method of Hinge Unloading

When a hinge unloads, the program must find a way to remove the load that the hinge was carrying and redistribute it to the rest of the structure. Hinge unloading occurs whenever force-deformation or moment-rotation curve shows a drop in capacity, from point C to point D. The hinge unloads elastically without any plastic deformation (i.e., parallel to slope A-B).

When the hinge force reaches point C, entire structure is unloaded i.e. the program reverses the load on the whole structure until the hinge is unloaded up to the point D. When the hinge reaches point D, the load is again reversed. Other parts may now pick up the load that was removed from the unloading hinge.

G.17.4.2.1.10 Performance

A performance check can be done either by Capacity Spectrum method or Displacement Coefficient Method.

Note: The program considers Displacement Coefficient method.

G.17.4.2.2 Member Stiffness Matrix with Plastic Hinge

When numbers of springs are connected together in series, the force in each spring is the same. Here we will use the flexibility matrix rather than the stiffness matrix, which will enable us to avoid the intermediate displacements at the spring connections as variable.

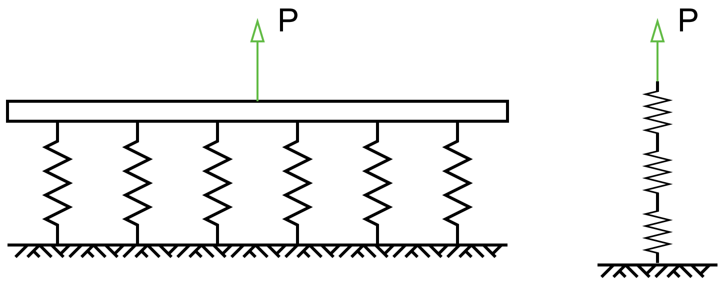


Figure 260: Springs in a) parallel and b) series.

Thus write for each spring,

Extension = flexibility x force

and add up all the individual extensions to give

Total extension = sum of flexibilities x force

To demonstrate this idea, we will take one composite member as shown in Fig. This member is assumed to consist of a number of sections whose F matrices are known.

At first if we consider a case where end A is fixed and a load px is applied to end X, then by definition the displacement at X is given $d_X = e_{AX} = F_{AX}\, p_X$. This displacement is the sum of the displacements due to the strains in the individual sections.

Let us consider, the part KX of the member, lying between end 2 of section k and X.The load acting on the end of KX attached to the section k is equal to -$H_{KX}p_X$, and the load acting on end 2 of section k is $H_{KX}p_X$. Since A is fixed and AJ is considered rigid, end 1 of member k is effectively fixed, and the displacement of its end 2 is given by $d_K = e_K = F_k\, p_{2A} = F_k\, H_{KX}\, p_X$ .Now KX is a rigid member, and the displacements of its ends therefore satisfy the equation for rigid-body movement. Thus the displacement of X due to the strain induced in section k by the load pX at X is $d_{Xk} = H^t{}_{KX}\, F_k\, H_{KX}\, p_X$. Thus the flexibility matrix F_k of section k appears as $H^t{}_{KX} F_k\, H_{KX}$ when viewed from the point X. Similar analysis may be applied to each section of AX, and adding up all the displacement contributions we obtain the complete flexibility matrix of AX in the form

$$F_{AX} = \Sigma\, H^t{}_{KX} \cdot F_k \cdot H_{KX}$$

Where the summation extends over all the sections making up the member Ax. Inversion of F_{AX} gives the matrix K_{AX}, and since the matrix H_{AX} may be obtained either from the overall equilibrium of the member or by multiplying together all the H matrices of the sections the other stiffness matrices.

Semi-Rigid Joint Connections

When a member is attached to the joints at its ends by flexible connections, each of which transmits a moment proportional to the difference between the rotation of the end of the member and the rotation of the joint to which it is attached. Such joints occur in bolted frames, and in welded frames after the onset of plasticity.

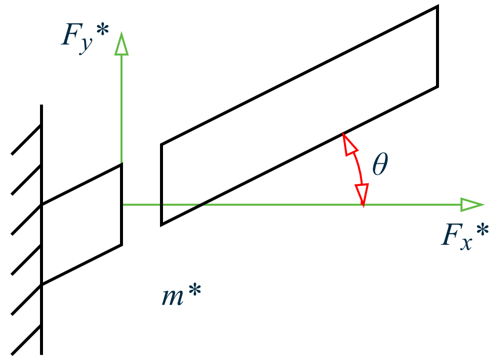


Figure 261: Free body diagram of semi-rigid joint

To have a clear understanding, first we will describe it for a two-dimensional plane frame and then we will extend it to three-dimensional space frame.

Let us consider a frame of length Land flexural rigidity EI, whose ends 1 and 2 are attached to the rest of the frame by connections, which exert moment's $4Eik_1/L$ and $4Eik_2/L$ respectively per unit difference in rotation. We ignore the length of each connection.

The displacement vector on the member side of the connection is given by the equation

G.17.4.2.3 Modeling Rules for Pushover Analysis in STAAD.Pro

Pushover analysis takes time. Each nonlinear problem is different. Since it is a step-wise linear analysis, analysis time and results is very much dependent on the incremental push load defined in the input file. Small changes in properties or loading can cause large change in nonlinear response. Hence it is important to consider different loading cases and to perform sensitivity studies on the effect of varying the properties of different structural elements. Analysis results depends on the selection of control node, the selection of lateral load patterns and the determination of fundamental period.

It is better to start with simple analytical model. The analytical model of the building should represent all new and existing components that influence the mass, strength, stiffness, and deformability of the structure at or near performance point. Elements and components shown not to significantly influence the building assessment need not be modeled.

The model should perform as expected under linear static loads and modal analysis. The control node shall be located at the center of mass at the roof of a building. It should be selected in such a way that the displacement component in the direction of lateral load monotonically increases as push load increases. Separate mathematical models representing the framing along two orthogonal axes of the building shall be developed for two-dimensional analysis. A mathematical model representing the framing along two orthogonal axes of the building shall be developed for three-dimensional analysis. Independent analysis must be done while using different methods of lateral load distribution.

Since Pushover analysis depends very much on the incremental push load defined in the input file by using `LDSTEP` parameter, it is better to give a lower value of `LDSTEP` parameter (preferably within 25) to start with. If the analysis does not run successfully and indicate warning message to increase `LDSTEP` parameter, parameter value will have to be increased. However, it may happen that the analysis has ran successfully and achieved the convergence criteria, the program is still issuing warning message to increase the `LDSTEP` parameter to get more accurate results. The value of `LDSTEP` parameter can be increased to get more accurate results but note that by how much `LDSTEP` parameter will be increased and what the exact value it will take depends upon the engineer's judgement. By looking at the Capacity Curve graph, hinge formation and the hinge status (IO, LS, CP or greater than CP) at different load steps, engineer has to judge whether to increase the `LDSTEP` parameter or not.

The type of warning message only gives an initial indication to the engineer whether the value of LDSTEP parameter will be increased or not.

```
*** WARNING : ANALYSIS RESULT IS NOT CONVERGING. UNABLE TO FIND A SOLUTION.
TRY TO REDUCE INCREMENTAL PUSH LOAD BY INCREASING LDSTEP PARAMETER.
```

If the warning message is issued as mentioned above, LDSTEP parameter has to be increased.

```
*** WARNING : STRUCTURE HAS REACHED AN UNSTABLE EQUILIBRIUM WHERE ALL OF A
SUDDEN MEMBERS HAVE STARTED TO FAIL. MORE ACCURATE RESULT MAY
BE OBTAINED BY REDUCING INCREMENTAL PUSH LOAD BY INCREASING LDSTEP
>PARAMETER.
```

If the warning message is issued as mentioned above, LDSTEP parameter may or may not be increased depending upon engineer's choice.

G.17.4.2.4 Scope of Pushover Analysis in STAAD.Pro

Pushover Analysis in STAAD.Pro will produce valid results only when following conditions are satisfied:

1. Buildings are regular and do not have adverse torsional or multimode effects. The capacity curve is generally constructed to represent the first mode response of the structure based on the assumption that the fundamental mode of vibration is the predominant response of the structure.
2. Fully rigid moment frame and concentric braced frame are considered. If any member release is defined, its effect will be same as that for linear analysis.
3. Only steel structures are considered (i.e., material cannot be concrete).
4. Only straight beam and column members are considered (i.e., no curved members, no plate or surface elements).
5. All members of a structure are considered as primary element. No secondary elements are considered.
6. For built-in FEMA hinge formation, out of six degrees of freedom, hinge formation is considered for bending moment about local z axis of beam. For column, hinge formation is considered for bending moment about local z axis of column or for bending moment about local y axis of column, whichever moment becomes the guiding factor. Depending upon the guiding criteria given in FEMA 356:2000, hinge formation in beam is deformation-controlled and that in column can be deformation-controlled or force-controlled. For braced (i.e. truss) member, axial hinge is considered. Beside moment and axial hinge, no other types of hinge formation (e.g. hinge formation due to shear or hinge formation due to coupled effect of axial force and bi-axial bending moment) are considered.
7. Built-in hinge properties and acceptance criteria for beam and columns given in Tables 5-6 and 5-7 of FEMA 356:2000 for beams, columns, and braces are considered.
8. User defined hinge properties are considered for beam and column for hinge formation for bending moment only. For column, only deformation-controlled action is considered as no axial force is considered for checking force-controlled action in column.
9. Out of five methods for lateral load distribution, three methods have been implemented as per Section 3.3.3.2.3 of FEMA 356:2000. The methods implemented are 1.1, 1.2, and 2.1 of Section 3.3.3.2.3 (See G. 17.4.1.8 Lateral Load Distribution (on page 2177)).
10. Performance point of a structure is calculated based on Displacement Coefficient method to find target displacement.

G.18 Member End Forces

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. The following figures show the member end actions with their directions.

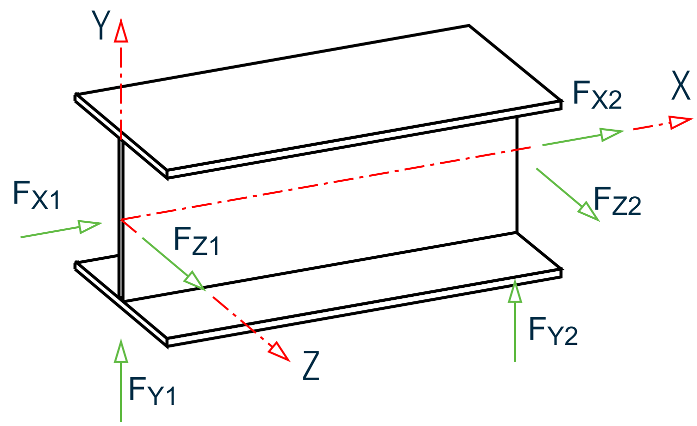


Figure 262: Member end forces when Global Y is vertical

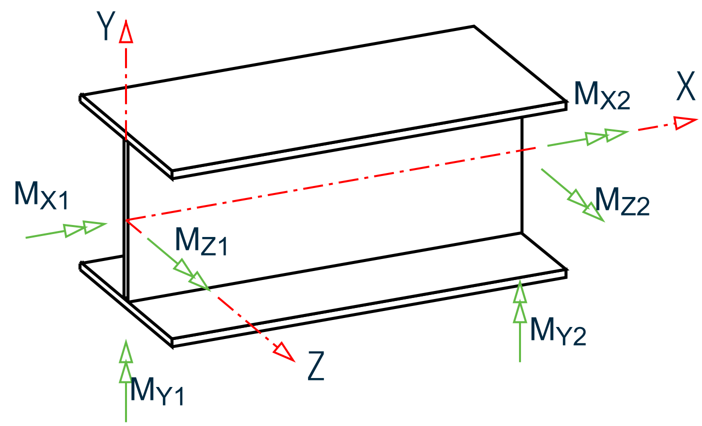


Figure 263: Member end moments when Global Y is vertical

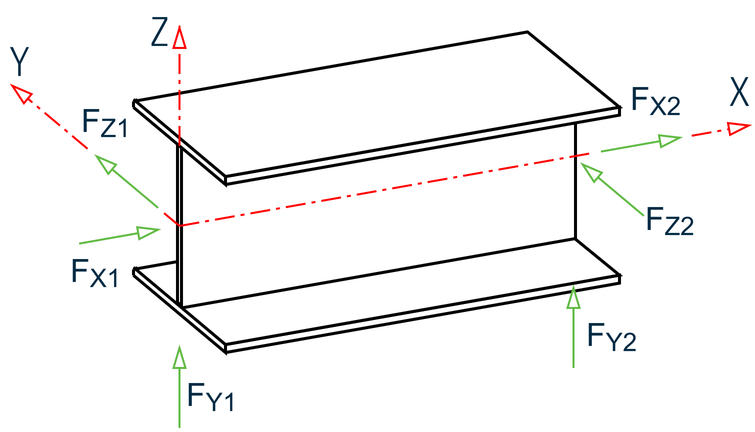


Figure 264: Member end forces when Global Z is vertical (that is, SET Z UP is specified)

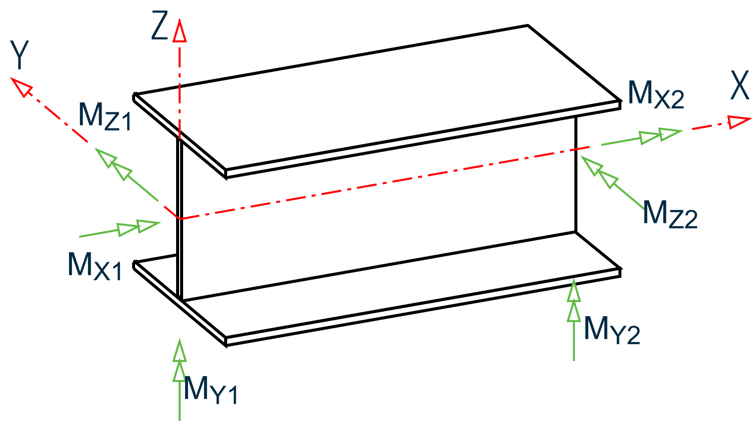
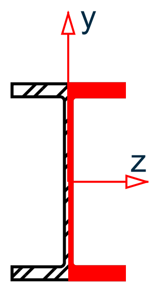
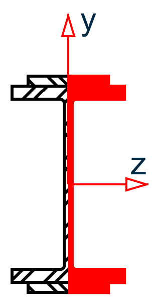


Figure 265: Member end moments when Global Z is vertical (that is, `SET Z UP` *is specified)*

Stress Zones Due to Bending





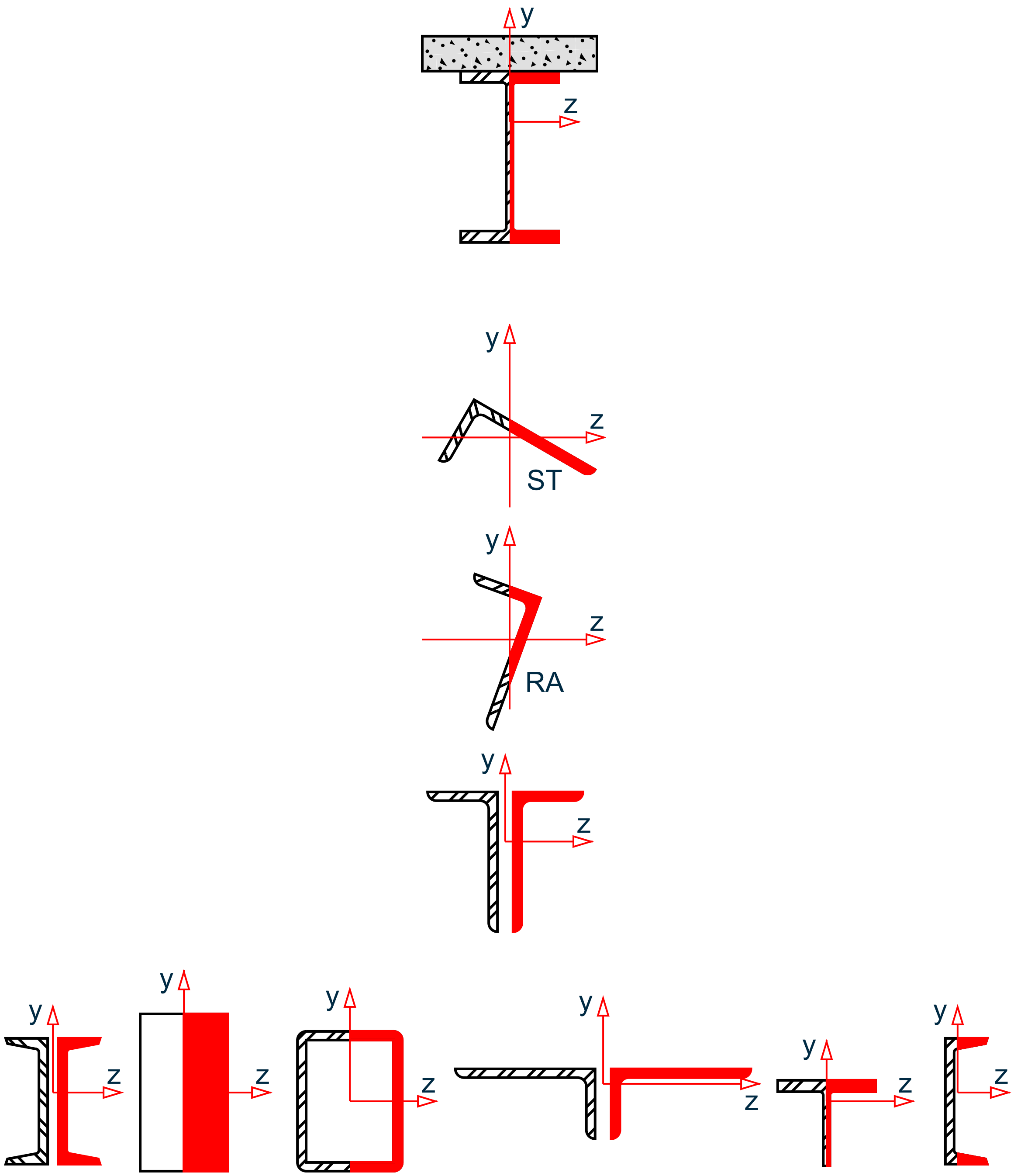
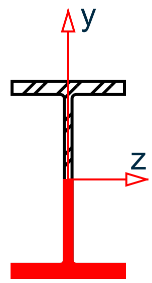
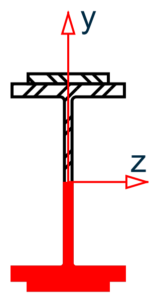


Figure 266: Stress zones due to bending about the Y axis (MY) for various section types

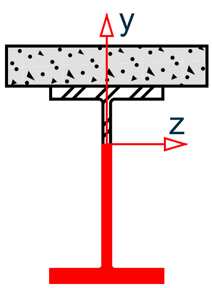
Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension



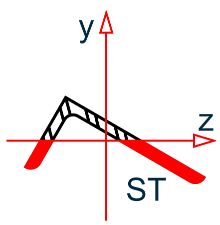
y
z



y
z



y
z



y
z
ST

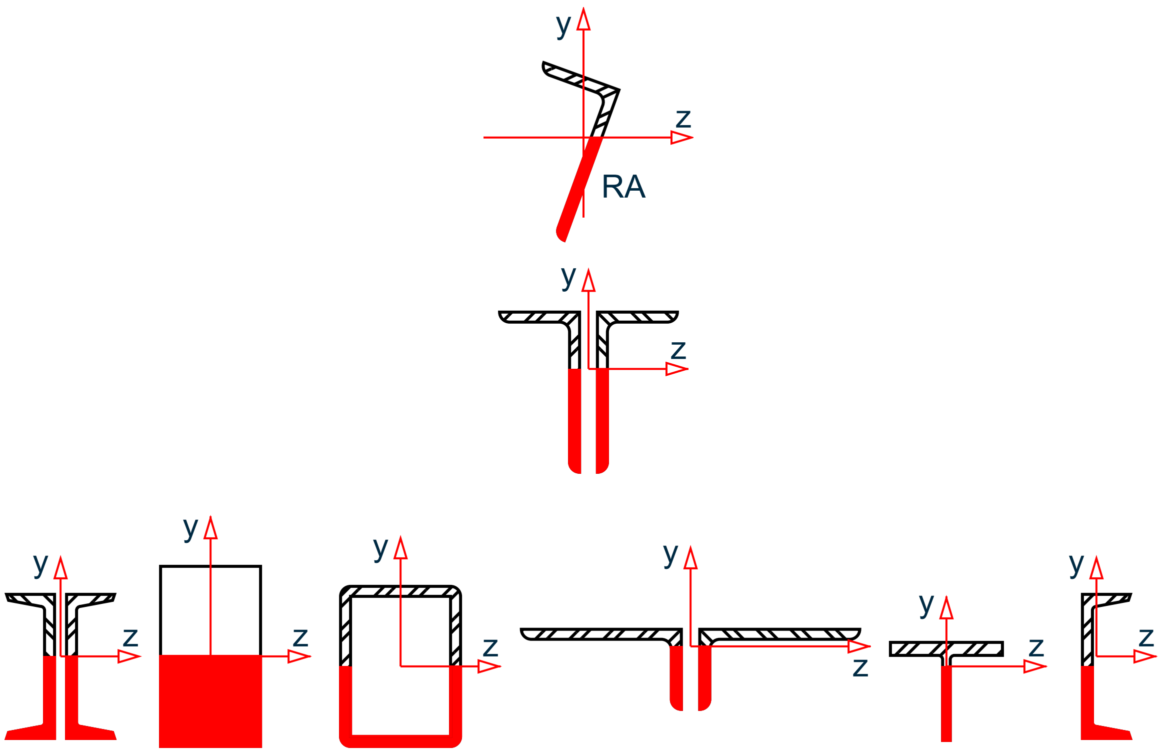


Figure 267: Stress zones due to bending about the Z axis (MZ) for various section types

Note: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension.

Related Links

- *TR.41 Section Specification* (on page 2665)
- *TR.42 Print Specifications* (on page 2666)
- *TR.40 Load Envelope* (on page 2663)

G.18.1 Secondary Analysis

Solution of the stiffness equations yield displacements and forces at the joints or end points of the member. STAAD.Pro is equipped with the following secondary analysis capabilities to obtain results at intermediate points within a member.

Related Links

- *TR.41 Section Specification* (on page 2665)
- *TR.42 Print Specifications* (on page 2666)
- *TR.44 Printing Section Displacements for Members* (on page 2672)
- *TR.45 Printing the Force Envelope* (on page 2674)

G.18.1.1 Member Forces at Intermediate Sections

With the `SECTION` command, you may choose any intermediate section of a member where forces and moments need to be calculated. These forces and moments may also be used in design of the members. The maximum number of sections specified may not exceed five, including one at the start and one at the end of a member. If no intermediate sections are requested, the program will consider the start and end member forces for design. However, of the sections provided, they are the only ones to be considered for design.

Related Links

- *TR.41 Section Specification* (on page 2665)
- *TR.42 Print Specifications* (on page 2666)
- *TR.40 Load Envelope* (on page 2663)

G.18.1.2 Member Displacements at Intermediate Sections

Like forces, displacements of intermediate sections of members can be printed or plotted. This command may not be used for truss or cable members.

Related Links

- *TR.41 Section Specification* (on page 2665)
- *TR.42 Print Specifications* (on page 2666)
- *TR.44 Printing Section Displacements for Members* (on page 2672)
- *TR.45 Printing the Force Envelope* (on page 2674)

G.18.1.3 Member Stresses at Specified Sections

Member stresses can be printed at specified intermediate sections as well as at the start and end joints. These stresses include:

a. Axial stress, which is calculated by dividing the axial force by the cross sectional area,
b. Bending-y stress, which is calculated by dividing the moment in local-y direction by the section modulus in the same direction,
c. Bending-z stress, which is the same as above except in local-z direction,
d. Shear stresses (in y and z directions), and
e. Combined stress, which is the sum of axial, bending-y and bending-z stresses.

All the stresses are calculated as the absolute value.

Related Links

- *TR.41 Section Specification* (on page 2665)
- *TR.42 Print Specifications* (on page 2666)
- *TR.40 Load Envelope* (on page 2663)

G.18.1.4 Force Envelopes

Force envelopes of the member forces FX (axial force), FY (Shear-y), and MZ (moment about local z-axis, i.e., strong axis) can be printed for any number of intermediate sections. The force values include maximum and minimum numbers representing maximum positive and maximum negative values.

The following is the sign convention for the maximum and minimum values:

FX A positive value is compression, and negative tension.

FY A positive value is shear in the positive y-direction, and negative in the negative y-direction.

FZ Same as above, except in local z-direction.

MZ A positive moment will mean a moment causing tension at the top of the member. Conversely, a negative moment will cause tension at the bottom of the member. The top of a member is defined as the side towards positive local y-axis.

MY Same as above, except about local y axis.

Related Links

- *TR.43 Stress/Force Output Printing for Surface Entities* (on page 2672)
- *TR.45 Printing the Force Envelope* (on page 2674)

G.19 Multiple Analyses

Structural analysis and design may require multiple analyses in the same run. STAAD.Pro allows you to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes.

For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an `INACTIVE` facility for this type of analysis.

Inactive Members

With the `INACTIVE` command, members can be made inactive. These inactive members will not be considered in the stiffness analysis or in any printout. The members made inactive by the `INACTIVE` command are made active again with the `CHANGE` command. This can be useful in an analysis where stage construction is modeled due to which, a set of members should be inactive for certain load cases. This can be accomplished by:

a. making the desired members inactive
b. providing the relevant load cases for which the members are inactive
c. performing the analysis
d. using the `CHANGE` command to make all the inactive members active
e. making another set of members inactive and providing the proper load cases for which the members are meant to be inactive, performing the analysis and repeating the procedure as necessary.

Related Links

- *TR.38 Change Specification* (on page 2660)
- *Change tab* (on page 2920)
- *A. To add a change command* (on page 958)
- *TR.18 Inactive/Delete Specification* (on page 2240)

G.20 Steel, Concrete, and Timber Design

Related Links

- *D1.F. American Codes - Concrete Design per ACI 318* (on page 1198)
- *D. Steel Design* (on page 969)
- *D1.G. American Codes - Timber Design per AITC Code* (on page 1248)

G.21 Printing Facilities

All input data and output may be printed using `PRINT` commands available in STAAD. The input is normally echoed back in the output. However, if required, the echo can be switched off.

Extensive listing facilities are provided in almost all `PRINT` commands to allow you to specify joints, members and elements for which values are required.

Related Links

- *TR.44 Printing Section Displacements for Members* (on page 2672)
- *TR.45 Printing the Force Envelope* (on page 2674)
- *TR.46 Post Analysis Printer Plot Specifications* (on page 2675)

G.22 Miscellaneous Facilities

STAAD.Pro offers the following miscellaneous facilities for problem solution.

Perform Rotation

This command can be used to rotate the structure shape through any desired angle about any global axis. The rotated configuration can be used for further analysis and design. This command may be entered after the Joint Coordinates or between two Joint Coordinate commands or after all Member/Element Incidences are specified.

Substitute

Joint and member numbers may be redefined in STAAD through the use of the `SUBSTITUTE` command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. This facility allows the user to specify numbering schemes that will result in simple input specification as well as easy interpretation of data.

Calculation of Center of Gravity

STAAD is capable of calculating the center of gravity of the structure. The `PRINT CG` command may be utilized for this purpose.

Related Links

- *TR.15 Redefinition of Joint and Member Numbers* (on page 2234)
- *TR.17 Rotation of Structure Geometry* (on page 2239)
- *TR.42 Print Specifications* (on page 2666)

10 Technical Reference of STAAD Commands

This section contains details of the STAAD.Pro commands used to create STAAD input files which are read by the STAAD engine.

Note: This section was previously included as Section 5 of the *Technical Reference Manual*. For convenience, the section numbers from that manual are maintained here with "TR." in place of "5." for the chapter number.

Introduction

The STAAD.Pro graphical user interface (GUI) is normally used to create all input specifications and all output reports and displays. These structural modeling and analysis input specifications are stored in STAAD input file – a text file with extension, `.STD`. When the GUI opens an existing model file, it reads all of the information necessary from the STAAD input file. You may edit or create this STAAD input file and then the GUI and the analysis engine will both reflect the changes.

The STAAD input file is processed by the STAAD analysis "engine" to produce results that are stored in several files (with file extensions such as `ANL`, `BMD`, `TMH`, etc.). The STAAD analysis text file (file extension `.ANL`) contains the printable output as created by the specifications in this manual. The other files contain the results (displacements, member/element forces, mode shapes, section forces/moments/displacements, etc.) that are used by the GUI in the post processing mode.

TR.0 STAAD Commands and Input Instructions

This section of the manual describes in detail various commands and related instructions for STAAD.Pro. The user utilizes a command language format to communicate instructions to the program. Each of these commands either supplies some data to the program or instructs it to perform some calculations using the data already specified. The command language format and conventions are described in TR.1 Command Language Conventions (on page 2197). This is followed by a description of the available commands.

Although the STAAD.Pro input can be created through the Analytical Modeling workflow, it is important to understand the command language. With the knowledge of this language, it is easy to understand the problem and add or comment data as necessary. The general sequence in which the commands should appear in an input file should ideally follow the same sequence in which they are presented in this section. The commands are executed in the sequence entered. Obviously then the data needed for proper execution of a command must precede the command (e.g., Print results after Perform Analysis). Otherwise, the commands can be provided in any order with the following exceptions.

- **i.** All design related data can be provided only after the analysis command.
- **ii.** All load cases and load combinations must be provided together, except in a case where the `CHANGE` command is used (see TR.38 Change Specification (on page 2660)). Additional load cases can be provided in the latter part of input.

All input data provided is stored by the program. Data can be added, deleted or modified within an existing data file.

In-Core Verus Out-of-Core

The engine can operate in two modes: *in-core* and *out-of-core*. The in-core solver will be used for models with under 20,000 joints and the out-of-core solver for models over 20,000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be overridden using the `SET STAR` command.

The full set of overrides for the advanced engine is:

`SET STAR 3` – default
`SET STAR 4` – use out-of-core solver regardless of size

TR.1 Command Language Conventions

This section describes the command language used in STAAD. First, the various elements of the language are discussed and then the command format is described in details.

TR.1.1 Elements of STAAD Commands

Integer Numbers

Integer numbers are whole numbers written without a decimal point. These numbers are designated as i_1, i_2, etc., and should not contain any decimal point. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number.

Floating Point Numbers

These are real numbers which may contain a decimal portion. These numbers are designated as f_1, f_2... etc.. Values may have a decimal point and/or exponent. When specifying numbers with magnitude less than 1/100, it is advisable to use the E format to avoid precision related errors. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number. Limit these to 24 characters.

Example

```
5055.32   0.73   -8.9   732
5E3   -3.4E-6
```

etc.

The decimal point may be omitted if the decimal portion of the number is zero.

Alphanumeric

These are characters, which are used to construct the names for data, titles or commands. Alphabetic characters may be input in upper or lower case letters. No quotation marks are needed to enclose them.

Example

```
MEMBER PROPERTIES
1 TO 8 TABLE ST W8X35
```

Repetitive Data

Repetitive numerical data may be provided in some (but not all) input tables such as joint coordinates by using the following format:

```
n*f
```

Where:

n = number of times data has to be repeated
f = numeric data, integer and floating point

Example

```
JOINT COORDINATES
1  3*0.
```

This joint coordinate specification is same as:

```
1  0.  0.  0.
```

TR.1.2 Command Formats

Free-Format Input

All input to STAAD.Pro is in free-format style. Input data items should be separated by blank spaces (not commas) from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles. Limit a data item to 24 characters.

Commenting Input

For documentation of a STAAD data file, the facility to provide comments is available. Comments can be included by providing an asterisk (*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

Example

```
JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT LOAD
2 3 7 FY 35.0
* etc.
```

Meaning of Underlining in the Manual

Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined.

For example, if the word `MEMBER` is used in a command, only the portion `MEMB` need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

Meaning of Braces and Parenthesis

In some command formats, braces enclose a number of choices, which are arranged vertically or separated by a | character. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (*) mark is located outside the braces.

Example

```
{XY | YZ | XZ}
```

In the above example, you must make a choice of XY or YZ or XZ.

Note: In some instances, the choices will be explicitly defined using "or" for clarification.

Example

```
*{FX | FY | FZ}
```

Here, you can choose one or all of the listing (FX, FY and FZ), in any order.

Parentheses, ( ), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.

Example

```
PRINT (MEMBER) FORCES
```

```
PERFORM ANALYSIS (PRINT LOAD DATA)
```

In the first line, the word `MEMBER` may be omitted with no change of the meaning of the command. In the second line, the `PRINT LOAD DATA` command may also be omitted, in which case the load data will not be printed.

Multiple Data Separator

Multiple data can be provided on a single line, if they are separated by a semicolon (;) character. One restriction is that a semicolon can not separate consecutive commands. They must appear on separate lines.

Example

```
MEMBER INCIDENCES
1  1  2;  2  2  3;  3  3  4
```

etc.

Possible Error:

```
PRINT FORCES; PRINT STRESSES
```

In the above case, only the `PRINT FORCES` command is processed and the `PRINT STRESSES` command is ignored.

Listing Data

In some STAAD command descriptions, the word "list" is used to identify a list of joints, members/elements, or loading cases. The format of a list can be defined as follows:

$$\text{list} = *\{\ i_1\ i_2\ i_3\ \ldots\ |\ i_1\ \underline{\text{TO}}\ i_2\ (\underline{\text{BY}}\ i_3)\ |\ \underline{X}\ \text{or}\ \underline{Y}\ \text{or}\ \underline{Z}\ \}$$

TO means all integers from the first (i_1) to the second (i_2) inclusive. BY means that the numbers are incremented by an amount equal to the third data item (i_3). If BY i_3 is omitted, the increment will be set to one. Sometimes the list may be too long to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also, only a list may be continued and not any other type of data.

Instead of a numerical list, a single group-name may be entered if that group was previously defined.

Instead of a numerical list, the specification X (or Y or Z) may be used. This specification will include all `MEMBER`s parallel to the global direction specified. Note that this is not applicable to `JOINT`s or `ELEMENT`s.

Note: ALL, BEAM, PLATE, SOLID. Do not use these unless the documentation for a command specifically mentions them as available for that command. ALL means all members and elements, `BEAM` means all members, etc.

Continuing a command to the next line

Only lists may be continued to the next line by ending the line with a blank and hyphen (see above) with few exceptions: Multilinear spring supports, Supports, Control/Dependent. Others have special types of continuations. Please follow the command descriptions.

Example

```
2 4 7 TO 13 BY 2 19 TO 22 -
28 31 TO 33 FX 10.0
```

This list of items is the same as:

```
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
```

Possible Error:

```
3 5 TO 9 11 15 -
FX 10.0
```

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

TR.1.3 Listing of Objects by Specification of Global Ranges

Used to specify lists of objects (e.g., joints, members, and/or elements) by providing global ranges. The general format of the specification is as follows.

General Format

```
{ XRANGE | YRANGE | ZRANGE } f1, f2
```

Where:

XRANGE, YRANGE, ZRANGE = direction of range (parallel to global X, Y, Z directions respectively)
f1, f2 = values (in current unit system) that defines the specified range.

Notes

1. Only one range direction (XRANGE, YRANGE etc.) is allowed per list. (Exceptions: Area/Floor load and Control/Dependent).
2. No other items may be in the list.
3. The values defining the range (f1, f2) must be in the current unit system.

Example

```
MEMBER TRUSS
XRANGE 20. 70.
CONSTANTS
E STEEL YRANGE 10. 55.
```

In the above example, a XRANGE is specified with values of 20. and 70. This range will include all members lying entirely within a range parallel to the global X-axis and limited by X=20 and X=70.

TR.1.4 Memory Allocation

To allow a larger allocation of memory for the CM array.

The `MEMORY` command is optional and if entered must be the first line of the `.STD` file.

General Format

```
MEMORY f1
```

where:

f1 The integer amount of blocks of CM memory to be allocated. *f1* is multiplied by 1,000,000 x 4-byte floating point words to get allocation in bytes (320 MBytes). The default value is 300 for STAAD.Pro 64-bit and 80 for STAAD.Pro 32-bit. The upper limit for 64-bit systems is 2,100.

Note: For older STAAD.Pro 32-bit (phased out in CE release May 2018), experience suggested that *f1* be between 80 and 400, if required to re-run an input file. For STAAD.Pro 64-bit , experience is limited so trial and error is recommended.

Usage

When the program finds that memory block was insufficient to complete the analysis, the following warning is issued:

```
    'ERROR-STAAD BUILT IN MEMORY BLOCK HAS OVERFLOWED.'
    'ALLOCATED BLOCKSIZE =',I11
    'NEEDED BLOCKSIZE =',I11
    'TRY REDUCING THE NUMBER OF LOAD CASES.'
```

In this instance, add the `MEMORY f1` command to the first line of the `.STD` file where *f1* is the “needed blocksize” value. This amount may be increased so long as this does not exceed the recommended amounts.

Example

```
MEMORY 200
STAAD SPACE
…
FINISH
```

TR.2 Problem Initiation and Model Title

This command initiates the STAAD run and is also used to specify the type of the structure and provide an optional title.

Any STAAD input file must start with the word `STAAD`. Following type specifications are available:

- PLANE= Plane frame structure
- SPACE= Space frame structure
- TRUSS= Plane or space truss structure
- FLOOR= Floor structure

General Format

```
STAAD { PLANE | SPACE | TRUSS | FLOOR } (any_title )
```

Where:

any_title = Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.

Notes

1. Care must be taken about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications. Detailed discussions are available in G.2 Types of Structures (on page 2085). PLANE indicates the XY plane for Y up and the XZ plane for Z up. FLOOR indicates the XZ floor for Y up and the XY floor for Z up.

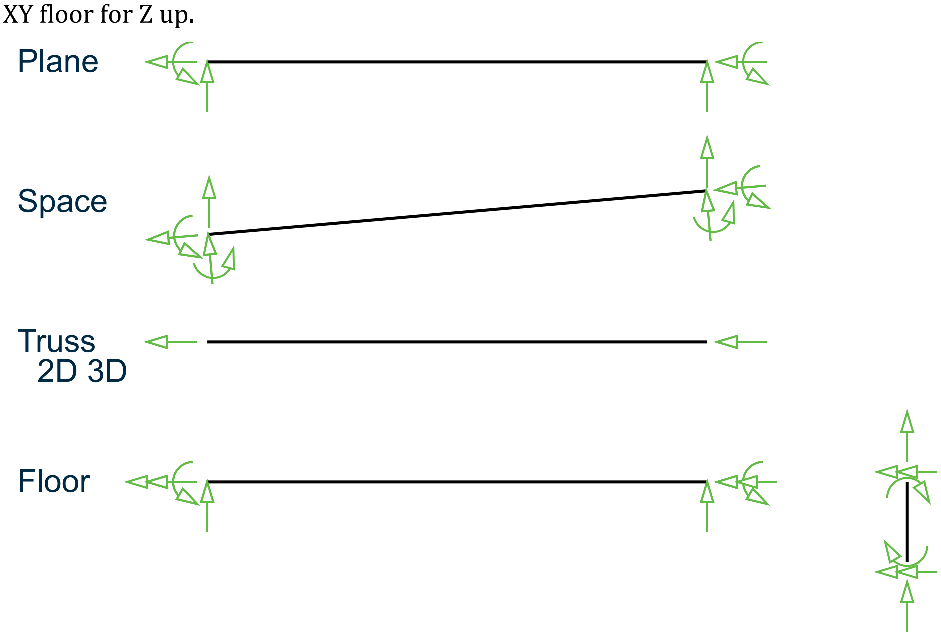


Figure 268: Structure type A) Plane, B) Space, C) Truss 2D or 3D, and D) Floor

2. The optional title provided by you are printed on top of every page of the output. You can use this facility to customize his output.

Limits

The following limits to model size are effective for STAAD.Pro CONNECT Edition V22 Update 2 (release 22.02.00) and later.

- Number of joints: 400,000*
- Joint number 1 to 999,999
- Number of Members, Physical Members, Plates, and Solids: 500,000*
- Member/Element numbers: 1 to 999,999
- Number of primary and combination cases: 10,101
- Load Case numbers: 1 to 99,999
- Number of modes and frequencies: 2,700
- Number of load cases that may be combined by a Repeat Load or Load Combination command: 550

* Some STAAD.Pro copies are available with much smaller limits, please check what limits you have purchased.

Notes:

The numerical limits should be considered as upper limits built into the software for those quantities on an individual basis. In practice, the actual maximums the program can handle are determined by the hardware resources as well as the limits imposed by the operating system. For example, it is highly improbable that a single model with 500,000 members and 10,101 load cases can be solved.

The memory demand of the program is determined by the combined effect of two or more of these terms. For example, when a steel design is performed, the memory required depends on the product of the members being designed (NMD) as well as the number of load cases being designed for (NL). That is, NMD ×NL. So the smaller the NMD, the larger the NL capacity and vice versa.

Related Links

- *G.2 Types of Structures* (on page 2085)

TR.3 Unit Specification

This command allows you to specify or change length and force units for input and output.

Description

The magnitude of numerical data that is entered in a STAAD model is determined by the unit of that data. STAAD.Pro supports numerical data based on both English and Metric unit systems.

The method used by STAAD.Pro to determine the unit of any value is by the previous `UNIT` command.

The `UNIT` command can set or change the unit of `LENGTH`, `FORCE`, or both. This can be set any number of times in a STAAD input file to specify data or generate output in the desired length or force units, subject to the conditions noted below. All data is assumed to be in the most recent unit specification preceding that data.

General Format

```
UNIT *{ length-unit  | force-unit }
```

where:

```
length-unit  = { INCHES | FT or FEET | METER | CM | DME | MMS | KM }
```

```
force-unit  = { KIP | POUND | KNS | | DNS | NEWTON | MNS | MTON | KG }
```

The following tables illustrate the unit and the relative factor against the primary base unit:

Table 227: Units of length in STAAD.Pro

Required Length Unit		Base Unit	Factor	UNIT command parameter
English	Inch	Inch	1	IN
	Foot	Inch	12	FT (or FE)
Metric	Meter	Meter	1	ME
	Decimeter	Meter	0.1	DM
	Centimeter	Meter	0.01	CM
	Millimeter	Meter	0.001	MM
	Kilometer	Meter	1,000	KM

Table 228: Units of force in STAAD.Pro

Required Force Unit		Base Unit	Factor	UNIT command parameter
English	Kip (kilopound)	Kip	1	KI
	Pound	Kip	0.001	PO
Metric	Kilonewton	Kilonewton	1	KN
	Decanewton	Kilonewton	0.1	DN
	Newton	Kilonewton	0.001	NE
	Meganewton	Kilonewton	1,000	MN
	Metric Ton	Kilonewton	9.80665	MT
	Kilogram	Kilonewton	$9.80665\ (10)^{-3}$	KG

Example

```
UNIT KIP FT
UNIT INCH
UNIT METER KNS
UNIT CM MTON
```

Notes

a. A `UNIT` command may be inserted before any primary level command (e.g., `JOINT COORD`, `MEMBER INCIDENCE`, etc.).
b. If no `UNIT` command is specified, then the default units are taken as:
   - Length: feet
   - Force: kips
c. Units can be mixed between systems of units, such as length of feet and force of kilonewtons.
d. Units that include both length and force components (e.g., stress = force/length^2) will be constructed from the current length and force units.

   Thus, if the current length unit is feet and force unit is kilonewtons, then when defining Young's modulus, the value for a stress unit is kN/ft^2. While this may be an odd definition, it is fully supported by STAAD.Pro.
e. Kilograms and metric tons are mass units which are converted to units of force assuming an acceleration due to gravity of 32.174 ft/s^2.
f. Temperature units are not explicitly defined in STAAD.Pro. The value of temperature loading and coefficient of thermal expansion are taken to be consistent units.
g. A `UNIT` command should be avoided immediately prior to the analysis command. Warnings will be presented in the output if one is included. Instead, to change the units of the output, the `UNIT` command should be included after the analysis command and prior to any post-analysis print commands.

Related Links

- *G.3 Unit Systems* (on page 2086)

TR.4 Input/Output Width Specification

These commands may be used to specify the width(s) of the lines of output file(s).

For INPUT width, 79 is always used. The program can create output using two different output widths - 72 (default) and 118. The narrower, 72-character width is used for display on most older monitors and for printing on “portrait” wide paper. The 118-character width may be used for printing on “landscape” wide paper.

Note: This is a customization facility that may be used to improve the presentation quality of the output documents.

General Format

```
{INPUT | OUTPUT} WIDTH i1
```

Where:

i1 = 72 or 118 depending on narrow or wide output. 72 is the default value.

TR.5 Set Command Specification

This command allows you to set various general specifications for the analysis/design run.

General Format

```
SET { NL i1 | {DISPLACEMENT i2 | PDELTA TOLERANCE i9} | SDAMP i3 | WARP i4 | ITERLIM
i5 | PRINT i7 | NOPRINT DIRECT | SHEAR | ECHO { ON | OFF } | GUI i6 | { Z | Y } UP |
DEFLECTION CUTOFF  f1 | FLOOR LOAD TOLERANCE f2 | EIGEN METHOD { LANCZOS | RITZ} }
```

Where:

Description

The following SET commands contain values with associated units and should appear *after* a UNIT command and before the first JOINT command.

Table 229: Commonly used SET commands which take units

Command	Parameter	Description
SET NL	*i1*	The SET NL command is used in a multiple analysis run if you want to add more primary load cases after one analysis has been performed. Specifically, for those examples which use the CHANGE command (see TR.38 Change Specification (on page 2660)), if you want to add more primary load cases, the NL value should be set to the maximum number (or slightly more) with the SET NL command. The program will then be able to set aside additional memory space for information to be added later. This command should be provided before any joint, member or load specifications. The value for *i1* is the maximum number of primary load cases (NL). This value should not be much greater than the maximum number of primary load cases actually used in the model. **Note:** If plates are present, the SET NL value includes the total number of primary and load combinations in the model.

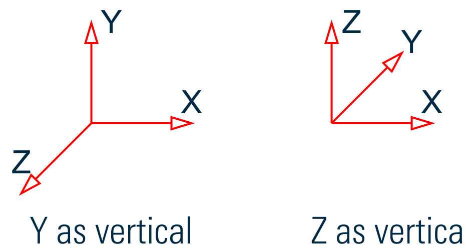
Command	Parameter	Description
SET DISPLACEMENT	*i2*	For `PDELTA ANALYSIS` (refer to TR.37.2 P-Delta Analysis Options (on page 2621) for additional information) The `SET DISPLACEMENT` *i2* command is used to specify the convergence tolerance. If the Euclidean norm RMS displacement of two consecutive iterations changes less than the value entered, then that load case is converged. This command should be placed before the `JOINT COORDINATE` specification. The default tolerance value, *i2* , is equal to the maximum span of structure divided by 120. The convergence tolerance for the Euclidean norm is difficult to know, so using this option is not recommended. *i2* = If the change in the Euclidean norm of the displacement vector from one `PDELTA` iteration to the next is less than this convergence tolerance value; then the iteration has converged for the case being analyzed.
SET DEFLECTION CUTOFF	*f1*	Used to arrest huge displacements in minor axis due to small delta effects. *f1* = If the absolute value of the maximum section displacement is less than *f1* after two iterations; then it is converged. Rapidly diverging minor axis displacement will not occur until after two iterations. *f1* is in current length units.
SET FLOOR LOAD TOLERANCE	*f2*	Used to specify the tolerance for out of plane nodes in a floor load. The program expects all nodes to lie in the same plain. If there is any minor variation in coordinates of the nodes, the program will excluded floor panels connected to those nodes. Hence, a tolerance value is used to so that all nodes with in the range (min. Y and min Y + tolerance). The default is take as 0.01% of the length of the longest beam of the beams in the floor load command. **Note:** Inclined floor panels are defined separately using the floor group option. Refer to TR.32.4.3 Floor Load Specification (on page 2483) for details on specifying a plane for floor loads. *f2* = the maximum distance any node can be from the plane, in the current length units

The following SET commands have dimensionless input

Table 230: Other commonly used SET commands

Command	Parameter	Description
SET SDAMP	*i3*	The SET SDAMP command will allow the damping of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. This command is not used unless CDAMP ratios are also entered for the members and elements in the CONSTANTS command. Composite damping is generally only used if there are many modes in the dynamic solution and there are a wide range of damping ratios in the springs, members, or elements. *i3* = the damping ratio to be used for all springs in computing the modal composite damping in dynamics.
SET WARP	*i4*	The SET WARP command will allow the I section member end warping restraint to be considered in calculating the torsional stiffness rigidity. Full or partial or no warping restraint are allowed.
SET ITERLIM	*i5*	This command sets the maximum number of iterations in an analysis with tension/compression only members or supports. The minimum iteration limit that may be entered is 3, then max is 150. Any value higher than this will be replaced with 150. The default value used is 10. The iterative procedure may not necessarily converge. Increasing the iterations may still not lead to convergence. Therefore it is recommended that after any tension/compression analysis, the output file should be reviewed for any warnings of non-convergence. The results from a load case that has not converged should not be used, but is provided for information only. *i5* = Maximum number of tension/compression iterations.

Command	Parameter	Description
SET PRINT	*i7*	The following values can be used to suppress the described warnings or include the described additional results in the output: 1 = Omit zero stiffness message, Rotational zero stiffness message due to solids, and “Node not connected. OK if control/ dependent” message. 2 = Omit Member in list does not exist message. 3 = Omit joint not connected message. 5 = Turn off floor load message. 8 = Omit self weight warnings. 10 = Turns on some iteration messages in direct analysis. 17 = Write rotational masses to mass text file; otherwise only translational masses written. Print scaled modal results for RSA and some force data by floor for RSA. *i7* = Used to suppress some warning messages or to include additional output.
SET NOPRINT DIRECT	(n/a)	Used to turn off the tau-b details in the output file when running a Direct Analysis (refer to TR.37.5 Direct Analysis (on page 2630)).
SET SHEAR	(n/a)	Including this command will omit the addition of pure shear distortion stiffness terms. This is often the method used in hand calculations and allows the analysis displacement results to be checked against those calculations.
SET ECHO {ON | OFF}	ON	This is a switch to turn on/off any commands that follow this command from being reported in the output file to help reduce the amount of data reported. This command can be specified at any point in the file.
SET GUI	0	A flag to indicate prevent the analysis engine from producing the results databases used by the GUI to display in the Post-Processing workflow. This can be used in very large models where the last action of the analysis to create the results databases can add a significant time to the overall analysis process. 0) Standard post-processing results are produced 1) No post-processing results are produced; only an output file (`.ANL`) is generated.

Command	Parameter	Description
SET { Z | Y } UP	Y UP	Determines the axis which defines the name of the global vertical direction in the model and the name of the minor axis of beam members. It is strongly recommended to use the Y UP axis system (default) as a number of commands and methods employed in STAAD.Pro do not support the Z UP axis convention. Both options follow the “right hand rule.”  **Note:** Both options follow the “right hand rule.” *Figure 269: Global axis orientation options in STAAD.Pro* This command will determine the alignment of angle profiles specified with the BETA ANGLE/RANGLE specification. See section CONSTANTs specification (see TR.26 Specifying and Assigning Material Constants (on page 2303) and G.4.2 Local Coordinate System (on page 2088)).
SET STAR	*i*	Instructs the program which solver to use. Refer to TR.0 STAAD Commands and Input Instructions (on page 2196) for additional information. 3 = STAAD Advanced Solver (in-core) - default 4 = STAAD Advanced Solver (out-of-core) **Note:** When SET STAR 3 is used and if the following limits are exceeded, then the solver will automatically switch to out-of-core: **i.** Number of nodes >30,000 **ii.** Number of nodes times the number of active primary load cases > 1,600,000

Command	Parameter	Description
SET EIGEN METHOD { LANCZOS | RITZ }	(n/a)	This command will override the default sub-space method for extracting eigen solutions as described in section G.17.3.1 Solution of the Eigenproblem (on page 2155). `LANCZOS` — Instructs the solver to use Arnoldi/Lanczos method for extraction of eigen vectors. `RITZ` — Use load dependent Ritz vectors method for extraction of eigen vectors. The use of either the Arnoldi/Lanczos method or the Ritz method requires a STAAD.Pro Advanced license. This command is not applicable with the basic solver. If neither method is specified, the default method of Subspace-iteration is used. **Note:** `SET EIGEN METHOD RITZ` is not used by pushover analysis or steady state analysis. The pushover analysis uses the lateral load distribution is based on the eigen vector. If the command is present, a warning message is given in the output.

Less frequently used SET commands

The following table contains a list of less frequently used SET commands

Command	Default	Description
SET BUCKLING MODES *i*	4	Number of buckling modes computed with advanced solver. This command is not applicable to the basic solver.
SET BYPASS { DIS | FOR | EJS | EJF }		Bypass generating certain outputs for the graphical user interface. DIS = section displacements FOR = section force EJS = plate nodal stress EJF = plate nodal force
SET CG TXT		This command will trigger the creation of a text file (`*.TXT`) that includes details of the section forces for the load cases generated with a load combination command with the `GENERATE` optional parameter. See TR.35 Load Combination Specification (on page 2616) for additional details. **Note:** The resulting file can be extremely large and thus this option should be used with care.
SET DIVISION *i*		Set the number of divisions used in meshing for surfaces (default is 10)
SET ENDFACTOR *f*		1.0 or -1.0 for combining spectrum cases

Command	Default	Description
SET GROUP DUPLICATES		Specifies the maximum number of groups any object (node, member, plate, or solid) can be included. A minimum of value 4 and a maximum of value 100. The default value is 10 groups.
SET INCLINED REACTION		Used to report the reactions of inclined supports in their inclined axis system. If not specified the reactions will be in the global axis.
SET INPLANE ROTATION		In-plane rotation (MX) in plates will be ignored.
SET LOAD PLATE		Include loads applied to inactive plates.
SET MASS *i*		1 = Use generated moments as masses
SET MULTI { 1 | 2 }		If multi-linear analysis fails to converge, try entering SET MULTI 2 and re-running.
SET NF TXT		Used to print section forces for each member corresponding to the critical load combination generation case to an external text file. See TR.35 Load Combination Specification (on page 2616) for additional details. This command should be used only for review of data and not in normal circumstances as the resulting text file can be very large.
SET NOSECT		This command is used to only produce results at the ends of members and no intermediate results. Note that this can result in a faster analysis for very large models, but parts of the Post Processing Workflow that rely on the intermediate section results will not be available.
SET NOTE { ON | OFF }		This commend toggles whether advice notes are displayed in the output file (`.anl`). When set to `OFF` the notes are not displayed. (`ON` is default).
SET NOWARNING		Switches off some warning messages.
SET PARTICIPATION FACTOR		Including this command will extend the details of a dynamic analysis to include participation factors normalised to 1.0.

Command	Default	Description
SET PLATE FLATNESS TOLERANCE *f*	30	This is used to set the tolerance for determining if any four-noded plate element is out of plane. The value entered in degrees is the maximum allowable deviation of one node from a plane defined by the other three nodes. **Note:** It is possible to test before the analysis using the tool **Utilities** > **Plate** menu. For more information see section M. To check for warped plates (on page 912). The value *f* is given in degrees.
SET PROFILE *s2*		A command to define the path of the folder that contains the collection of databases of section property data in SQLite (`.DB3`) files. Note also that the GUI will use this as the location of the section profile databases. The default file path used is `C:\ProgramData\Bentley\Engineering\STAAD.Pro CONNECT Edition\Sections\`.
SET RIGID DIAPHRAGM *i*	150	This command is used to reserve memory for the maximum number of diaphragms that may be defined in the model. See section TR.28.2 Floor Diaphragm (on page 2328) for more information on defining and using floor diaphragms in an analysis.
SET RS TXT		This command will trigger the creation of a text file (`*_RESP.TXT`) that includes the section forces for each mode used in a response spectrum load case. See TR.35 Load Combination Specification (on page 2616) for additional details. **Note:** This can generate a large datafile and thus should be used with care on larger models and models with large numbers of extracted mode shapes.
SET THCOPYS *i*		This command is used to set the maximum number of time history forcing functions (TYPES) to which a given node degree of freedom can be subjected. The default value is 4.
SET SOLUTION INCORE	(n/a)	For some smaller models, it may be possible to use an alternative “determinant search method” which is performed in-core. However this is limited to where the problem can be solved in a single matrix block. If not, then analysis will revert to using the subspace iteration method. This command is only applicable to the basic solver.

Command	Default	Description
SET PRINT STIFFNESS		This command will output the assembled global stiffness matrix to the output file (`*.ANL`). This command is only applicable to the basic solver. The output includes the stiffness matrix from the diagonal and only includes non-zero terms (with values of 10^{-20} or less are assumed to be zero). See the section below on how to read this output. See Stiffness Matrix Output (on page 2214) for details on how to interpret the stiffness matrix output.

Stiffness Matrix Output

When the `SET PRINT STIFFNESS` command is used with the Basic solver, the non-zero terms for the diagonal and upper half of the stiffness matrix are reported. The following example shows the output of adding this command to the file
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models \03 Static Beams\BEAM01.std`

```
 NON-ZERO STIFFNESS MATRIX VALUES, PRINTED BY ROWS FROM THE DIAGONAL.
                ROW        1  JOINT      1  DIRECTION  6
                1  1.180099515E+05      3 -1.498726372E+03       4  5.900497573E+04
                ROW        2  JOINT      2  DIRECTION  1
                2  3.172303322E+03
                ROW        3  JOINT      2  DIRECTION  2
                3  3.289796633E+01      4 -8.326259057E+02
                ROW        4  JOINT      2  DIRECTION  6
                4  1.966832440E+05
```

This describes position of each non-zero term in a row starting from the diagonal and then moving right. So, this matrix would be written as:

$$\begin{bmatrix} 118,009.9515 & 0 & -1,498.726372 & 59,004.97573 \\ & 3,172.303322 & 0 & 0 \\ \text{sym.} & & 32.89766633 & -832.6259057 \\ & & & 196,683.244 \end{bmatrix}$$

Related Links

- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)

TR.6 Data Separator

This command may be used to specify the desired separator character that can be used to separate multiple lines of data on a single line of input.

The semicolon (;) is the default character which functions as the separator for multiple line data on one line. However, this separator character can be changed by the `SEPARATOR` command to any character.

Note: Comma (,) or asterisk (*) may *not* be used as a separator character.

General Format

```
SEPARATOR a1
```

TR.7 Page Control Commands

TR.7.1 Page New

This command may be used to instruct the program to start a new page of output.

With this command, a new page of output can be started. This command provides the flexibility, the user needs, to design the output format.

Note: The presentation quality of the output document may be improved by using this command properly.

General Format

```
PAGE NEW
```

TR.7.2 Page Length

This command may be used to specify the page length of the output.

General Format

```
PAGE LENGTH i
```

Where:

i The page length in STAAD output is based on a default value of 58 lines . However, you may change the page length to any number i (number of lines per page) wanted.

TR.8 Ignore Specifications

This command allows you to provide member lists in a convenient way without triggering error messages pertaining to non-existent member numbers.

The `IGNORE LIST` command may be used if you want the program to ignore any nonexistent member(s) that may be included in a member list specification. For example, for the sake of simplicity, a list of members may be specified as MEMB 3 TO 40 where members 10 and 11 do not exist. An error message can be avoided in this situation by providing the `IGNORE LIST` command anywhere in the beginning of input. A warning message, however, will appear for each nonexistent member.

General format

```
IGNORE LIST
```

TR.9 No Design Specification

This command allows you to declare that no design operations will be performed during the run. The memory reserved for design will be released to accommodate larger analysis jobs.

STAAD always assumes that at some point in the input, you may want to perform design for steel, concrete, etc. members. These design processes require more computer memory. If memory availability is a problem, the above command may be used to eliminate extra memory requirements.

General Format

```
INPUT NODESIGN
```

TR.10 Job Information Data

This optional block of commands is used to project metadata for the STAAD input file.

General Format

The job information data block must begin with the START and END lines, with all lines in between being optional.

```
START JOB INFORMATION
ENGINEER DATA date
JOB NAME job-name
JOB CLIENT client-name
JOB NO job-number
JOB REV revision-number
JOB PART part-number
JOB REF reference-number
JOB COMMENT comments-string
ENGINEER NAME engineer-name
CHECKER NAME checked-by-name
APPROVED NAME approved-by-name
CHECKER-DATE checked-on-date
APPROVED DATE approved-on-date
CONNECTED PROJECT ID project-id
CONNECTED PROJECT NAME project-name
END JOB INFORMATION
```

Note: The job information block must be place *before* the JOINT COORDINATES command.

Caution: The CONNECTED Project *project-id* and *project-name* should not be directly edited. These are added through the User Interface.

TR.11 Joint Coordinates Specification

These commands allow you to specify and generate the coordinates of the joints of the structure. The JOINT COORDINATES command initiates the specification of the coordinates. The REPEAT and REPEAT ALL commands allow easy generation of coordinates using repetitive patterns.

General Format

```
JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) (NOREDUCE  BAND)
i1 xi1 yi1 zi1 (i2 xi2 yi2 zi2) (i3)
```

…repeat as needed to define each node or generated set of nodes

```
(in xin yin zin )
```

```
(REPEAT  n xr1 yr1 zr1 (xr2 yr2 zr2 … xrn yrn zrn) )
```

```
(REPEAT  ALL  n xt1 yt1 zt1 (xt2 yt2 zt2 … xtn ytn ztn) )
```

```
(JTORIGIN xOrigin yOrigin zOrigin)
```

Where:

i1, ***in***	the reference number of the node coordinate that follows
xi yi zi	the coordinates of node *i* in the given coordinate system.

Tip: To manually specify each node number and coordinate, separate nodes on a new line.

i2	optional last node number of a sequence of nodes generated from node *i1*
x2 y2 z2	the coordinates of node *i2* in the given coordinate system
i3	the increment in node number from *i1* to *i2*. The default increment is 1. Each node generated will be equally spaced along a linear path between *i1* and *i2*.
REPEAT *n*	A line starting with `REPEAT` will generate *n* copies of the previous line with an offset given by *xr1 yr1 zr1*. If additional offset values are used (e.g., *xr2 yr2 zr2* … *xrn yrn zrn*), then those offset values will be used by subsequent repeat generations. *n* is limited to 150
REPEAT ALL *n*	A line starting with `REPEAT ALL` will generate *n* copies of *all* the previous lines following any `REPEAT ALL` specification with an offset given by *xt1 yt1 zt1*. *n* is limited to 150
JTORIGIN	The optional parameter `JTORIGIN` causes the program to use a different origin than (0,0,0) for all joints entered with this `JOINT COORDINATES` command. It is useful in instances such as when the center of a cylinder is not at (0,0,0) but at a different point in space. The `JTORIGIN` command should be entered on a separate line. After the joint coordinates are entered or generated, then the *xOrigin*, *yOrigin*, *zOrigin* values are added to the respective X, Y, and Z coordinates of each joint.

The values of the coordinates (x y z) for cylindrical and reverse cylindrical systems are determined as follows:

The cylindrical coordinate system defines a point in space by virtue of the radius, an angle, and an offset distance:

1. *x1* = the radial distance (in current length units) from the origin in the XY plane
2. *y1* = an angle (in degrees) measured from the global X axis counter-clockwise about the global Z axis
3. *z1* = the offset distance (in current length units) from the XY plane along the global Z axis

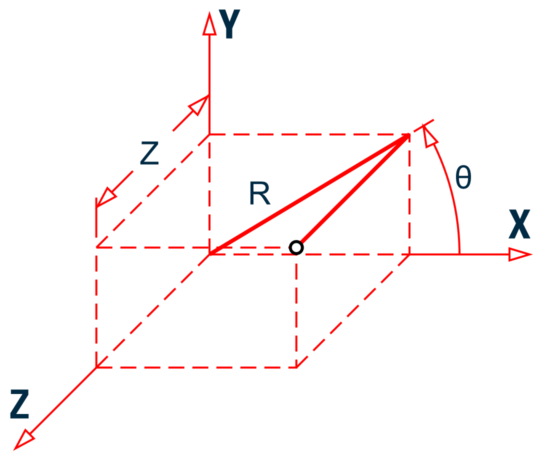


Figure 271: Cylindrical Coordinate System

The reference cylindrical coordinate system is an alternative cylindrical system that defines a point in space by virtue of the radius, an offset distance, and an angle:

1. *x1* = the radial distance (in current length units) from the origin in the XZ plane
2. *y1* = the offset distance (in current length units) from the XZ plane along the global Y axis
3. *z1* = an angle (in degrees) measured from the global X axis counter-clockwise about the global Y axis

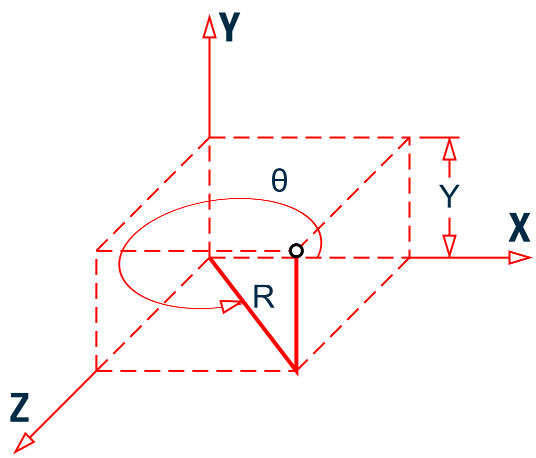


Figure 272: Reverse Cylindrical Coordinate System

Description

The command `JOINT COORDINATES` specifies a Cartesian coordinate system. Joints are defined using the global X, Y, and Z coordinates. The command `JOINT COORDINATES CYLINDRICAL` specifies a cylindrical coordinate system. Joints are defined using the r, q, and z coordinates. `JOINT COORDINATES CYLINDRICAL REVERSE` specifies a reverse cylindrical coordinate system. Joints are defined using the r, y, and q coordinates. Refer to G. 4.1 Global Coordinate System (on page 2087) for details and figures.

The multiple `JOINT COORDINATES` command concept allows `UNIT` changes and `PERFORM ROTATION` commands in between; such that these commands would apply to a selected portion of the joints. However, the `PERFORM`

`ROTATION` command applies to all prior defined joints, not just those in the previous `JOINT COORDINATE` command.

Example 1

```
JOINT COORDINATES
1 0 0 0  3 4 4 0  1
```

Generates the following three nodes:

1	0.0	0.0	0.0
2	2.0	2.0	0.0
3	4.0	4.0	0.0

Note: *i3* is given as 1 (the default value). Notably, *i3* could be omitted and the same three nodes would be generated.

Example 2

```
JOINT COORDINATES
1 0 0 0  5 5 0 0  2
```

Generates the following three nodes:

1	0.0	0.0	0.0
3	2.5	0.0	0.0
5	5.0	0.0	0.0

Example 3

```
JOINT COORDINATES CYLINDRICAL
1 10.0 0.0 0.0   5 10.0 180 0.0  1
```

Generated the following five nodes:

1	10.0	0.0	0.0
2	7.1	7.1	0.0
3	0.0	10.0	0.0
4	-7.1	7.1	0.0
5	-10.0	0.0	0.0

Example 4

The following examples illustrate various uses of the REPEAT command.

```
REPEAT 10  5. 10. 5.
```

The above REPEAT command will repeat the last input line 10 times using the same set of increments (i.e., x = 5., y = 10., z = 5.)

```
REPEAT 3  2. 10. 5. 3. 15. 3. 5. 20. 3.
```

The above REPEAT command will repeat the last input line three times. Each repeat operation will use a different increment set.

```
REPEAT 10  0. 12. 0. 15*0 0. 10. 0. 9*0
```

The above REPEAT command will repeat the last input line 10 times;

six times using x, y and z increments of 0., 12. and 0.,
and then four times using increments of 0., 10. and 0.

Each x, y, and z value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0., 0. and 0. (15*0) are supplied. The seventh repeat is done with increments of 0., 10. and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as 9*0.

Note: The PRINT JOINT COORDINATE command may be used to verify the joint coordinates provided or generated by REPEAT and REPEAT ALL commands. Also, you can use the Postprocessing workflow to verify geometry graphically.

Related Links

- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)
- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *G.4.2 Local Coordinate System* (on page 2088)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)
- *EX. US-9 Modeling Slabs and Shear Walls Using Finite Elements* (on page 6371)
- *EX. UK-9 Modeling Slabs and Shear Walls Using Finite Elements* (on page 6655)
- *EX. US-10 Finite Element Model for a Rectangular Tank* (on page 6380)
- *EX. UK-10 Finite Element Model for a Rectangular Tank* (on page 6665)

TR.12 Member Incidences Specification

This set of commands is used to specify members by defining connectivity between joints. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The member/element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD.Pro is capable of M. To check for multiple structures (on page 912).

General Format

```
MEMBER  INCIDENCES
i1 i2 i3 ( i4 i5 i6 )
REPEAT n mi ji
REPEAT ALL n mi ji
```

Description

The REPEAT command causes the previous line of input to be repeated *n* number of times with specified member and joint increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command or to the beginning of the specification if no previous REPEAT ALL command has been issued.

Note: When using REPEAT and REPEAT ALL commands, member numbering must be consecutive.

- ***i1*** Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.
- ***i2*** Start joint number.
- ***i3*** End joint number.

Note: Use REPEAT ALL 0 to start a set of members that will be repeated if you don't want to repeat back to the last REPEAT ALL.

The following parameters are used for member generation only:

- ***i4*** Second member number to which members will be generated.
- ***i5*** Member number increment for generation.
- ***i6*** Joint number increment which will be added to the incident joints. (*i5* and *i6* will default to 1 if left out.)
- ***n*** Number of times repeat is to be carried out.
- ***mi*** Member number increment
- ***ji*** Joint number increment

The PRINT MEMBER INFO command may be used to verify the member incidences provided or generated by REPEAT and REPEAT ALL commands.

Tip: Use the Post Processing facility to verify geometry graphically.

Example 1

```
MEMBER INCIDENCES
1  1  2
2  5  7  5
7  11  13  13  2  3
```

In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 7 will be from 11 to 13, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

Example 2

```
MEMBER INCIDENCES
1  1  21  20
21  21  22  23
REPEAT  4  3  4
36  21  25  39
REPEAT  3  4  4
REPEAT  ALL  9  51  20
```

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 5.12). The first input line creates the twenty columns of the first floor:

```
1  1  21  ;  2  2  22  ;  3  3  23  ; … ;  19  19  39  ;  20  20  40
```

The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor "floor" beams running, for example, in the east-west direction:

```
21  21  22;  22  22  23;  23  23  24
24  25  26;  25  26  27;  26  27  28
…    …    …
33  37  38;  34  38  39;  35  39  40
```

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor "floor" beams running in the north-south direction:

```
36  21  25;  37  22  26;  38  23  27;  39  24  28
40  25  29;  41  26  30;  42  27  31;  43  28  32
…    …    …    …
48  33  37;  49  34  38;  50  35  39;  51  36  40
```

The preceding commands have created a single floor unit of both beams and columns, a total of 51 members. The `REPEAT ALL` now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).

Related Links

- *M. To add beams with new nodes* (on page 658)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)

- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.13 Plate and Solid Elements

This section describes the commands used to specify plates (i.e., shells) and solids.

Related Links

- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.13.1 Plate and Shell Element Incidence Specification

This set of commands is used to specify elements by defining the connectivity between joints. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD.Pro is capable of M. To check for multiple structures (on page 912)

General Format

```
ELEMENT INCIDENCES (SHELL)
i1 i2 i3 i4 (i5) ( TO i6 i7 i8 )
REPEAT n ei ji
REPEAT ALL n ei ji
```

Description

ELEMENT INCIDENCES SHELL must be provided immediately after MEMBER INCIDENCES (if any) are specified. The REPEAT command causes the previous line of input to be repeated *n* number of times with specified element and joint increments. The REPEAT ALL command functions similar to the REPEAT command, except that it repeats all previously specified input back to the most recent REPEAT ALL command; or to the beginning of the specification if no previous REPEAT ALL command had been issued. Use REPEAT ALL 0 0 0 to start a set of elements that will be repeated if you don't want to repeat back to the last REPEAT ALL.

i1 Element number (any number up to six digits). If MEMBER INCIDENCE is provided, this number must not coincide with any MEMBER number.

i2 ... i5 Clockwise or counterclockwise joint numbers which represent the element connectivity. *i5* is not needed for triangular (3 noded) elements.

The following parameters are needed if elements are to be generated:

i6 Last element number to which elements are generated.

i7 Element number increment by which elements are generated. Defaults to 1 if omitted.

i8 Joint number increment which will be added to incident joints. Defaults to 1 if omitted.

The following data is needed if `REPEAT` or `REPEAT ALL` commands are used to generate elements:

n	Number of times repeat is to be carried out.
ei	Element number increment.
ji	Joint number increment.

The `PRINT ELEMENT INFO` command may be used to verify the element incidences provided or generated by `REPEAT` and `REPEAT ALL` commands.

Tip: Use the Post Processing facility to verify geometry graphically.

Example

```
ELEMENT INCIDENCE
1 1 2 7 6
2 3 4 8
3 8 9 11 10 TO 8
9 1 3 7 TO 14
```

Related Links

- *M. To draw plates connecting existing nodes* (on page 668)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.13.2 Solid Element Incidences Specification

Use the following commands to specify 4- through 8-noded elements, also known as solid elements.

Additional information on these elements is available in G.5.2 Solid Elements (on page 2110).

General Format

The element incidences for solid elements are to be identified using the expression `SOLID` to distinguish them from plate/shell elements.

```
ELEMENT INCIDENCES SOLID
```

```
i1 i2 i3 i4 i5 i6 i7 i8 i9 ( TO i10 i11 i12 )
```

```
REPEAT n ei ji
```

```
REPEAT ALL n ei ji
```

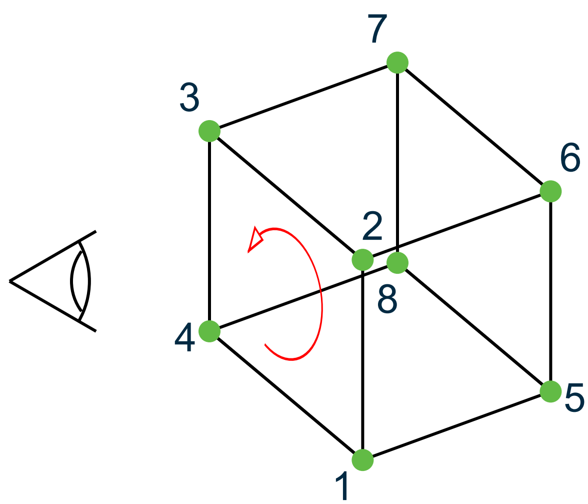
Description

`ELEMENT INCIDENCES SOLID` must be provided immediately after `MEMBER INCIDENCES` (if any) are specified as well as after the `ELEMENT INCIDENCES SHELL` (if any).

i1	Element number

i2 … i9	Joint number of the solid element
i10	Last element number to be generated
i11	Element number increment
i12	Joint number increment
n	Number of times REPEAT or REPEAT ALL is be to be carried out
ei	Element number increment
ji	Joint number increment

Specify the four nodes of any of the faces of the solid element in a counter-clockwise direction as viewed from the outside of the element and then go to the opposite face and specify the four nodes of that face in the same direction used in specifying the nodes of the first face. The opposite face must be behind the first face, as defined by the right hand rule, i.e., the opposite (back) face points to the first (front) face, which points to the viewer.



Use REPEAT ALL 0 to start a set of solids that will be repeated if you don't want to repeat back to the last REPEAT ALL.

Example

```
ELEMENT INCIDENCES SOLID
1 1 5 6 2 21 25 26 22 TO 3
4 21 25 26 22 41 45 46 42 TO 6
```

Related Links

- *G.5.2 Solid Elements* (on page 2110)
- *M. To draw a solid connecting existing nodes* (on page 681)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.13.3 Surface Entities Specification

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

TR.14 Plate Element Mesh Generation

There are several methods available in STAAD to model panel type entities like walls or slabs as an assembly of plate elements. This process is called meshing.

Two of those methods have a set of commands which can be provided in the STAAD input file.

1. Parametric mesh generator is the preferred method and is best to create this type of mesh using the STAAD.Pro Graphical User Interface. The aspect of this method, which enables commands to be written into the input file, is described in TR.14.1 Parametric Mesh Models (on page 2227).
2. The input mesh generation is described in TR.14.2 Element Mesh Generation (on page 2229) is based entirely on commands in the input file alone, and does not have any graphical interface for creation or modification.

Related Links

- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.14.1 Parametric Mesh Models

The STAAD.Pro Analytial Modeling workflow can be generate a plate element mesh using the Parametric Models dialog (on page 2734). Openings, density lines, and density points can be included in the mesh generation.

This is the recommended method for generating meshes in STAAD.Pro as it allows you to modify the mesh parameters after creation.

When a STAAD.Pro model has portions generated from a parametric mesh model, the parametric input (which is not otherwise part of the STAAD input data) is saved within a specially designated section of the STAAD input file. This gives you the flexibility to save mesh models at any time and make modifications at a later time, such as adding an opening or a density line.

It is important that this data not be modified or removed so as to preserve the parametric model. Any changes to the portions of the model marked between the `<!` and `!>` marks may have unintended consequences to the model.

General Format

```
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
parametric model data
!> END GENERATED DATA BLOCK
```

Where *parametric model data* is in following format:

```
PARAMETRIC MODEL name
TYPE s1
MESH PARAMTERS f1 f2 f3 f4
MESH ORIGIN f5 f6 f7
BOUNDARY f8
 f8 times coordinates of boundary nodes
GENERATED NODES node-list
GENERATED PLATES plate-list
```

Then the following optional inputs can be included one or more times:

```
(OPENING CIRCULAR f9 f10 f11 f12 f13)
(OPENING POLYGONAL f14
 f14 times coordinates that define the polygonal opening)
(REGION CIRCULAR f15 f16 f17 f18 f19)
(REGION POLYGONAL f20
 f20 times coordinates that define the polygonal region)
(DENSITY POINT f21
 f21 times coordinates of the density points)
(DENSITY LINE f22 f23 f24 f25 f26 f27 f28 f29)
```

Then the following closing command:

```
END
```

where:

s1 a category of surface used for post processing. Identifiers include `UNIDENTIFIED`, `SLAB`, or `WALL`. This is used to identify the object but does not otherwise affect the analysis or mesh produced.

> **Tip:** This value *is* used by design modules such as the Advanced Concrete Workflow (RCDC) to identify the type of surface generated.

f1 Meshing Method, 0 = Basic, 1 = Standard.

f5, f6, f7 Global coordinates for the origin point of the mesh.

Note: The control points for an `OPENING` or `REGION` cannot lie on the surface boundary.

Example

```
2072 1114 1113 1160; 2073 1045 1160 1113;
ELEMENT PROPERTY
810 TO 1779 1821 TO 2073 THICKNESS 1
                This entire section of the input file must not be edited.

<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL second_floor
MESH PARAM 0 3
MESH ORG 3 5 8
BOUNDARY 10
11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
OPENING CIRC 72 360 96 43.2666 12
OPENING POLY 5
216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360 100.8 2 216 360 100.8 2
DENSITY POINTS 2
```

```
180 360 168 1 360 360 168 1
DENSITY LINE 0 360 168 100 180 360 168 200
DENSITY LINE 180 360 168 1 360 360 168 1
DENSITY LINE 360 360 168 1 540 360 168 1
DENSITY LINE 180 360 0 1 180 360 168 1
DENSITY LINE 180 360 168 1 180 360 336 1
DENSITY LINE 360 360 0 1 360 360 168 1
DENSITY LINE 360 360 168 1 360 360 336 1
DENSITY LINE 54 360 302.4 1 162 360 201.6 1
DENSITY LINE 216 360 201.6 1 324 360 235.2 1
GENERATED PLATES ALL
END
!> END GENERATED DATA BLOCK

<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL roof
MESH PARAM 60 3
MESH ORG 2 3 5
BOUNDARY 6
36 1 65 1 66 1 53 1 52 1 51 1
GENERATED PLATES ALL
END
!> END GENERATED DATA BLOCK


DEFINE MATERIAL START
ISOTROPIC STEEL
```

Related Links

- *Parametric Models dialog* (on page 2734)

TR.14.2 Element Mesh Generation

This set of commands is used to generate finite element meshes. The procedure involves the definition of super-elements, which are subsequently divided into smaller elements.

Description

Tip: It is generally recommend that you use the parametric method of mesh generation (See TR.14.1 Parametric Mesh Models (on page 2227)) so you can modify the mesh parameters after creating the mesh.

Note: This method of generating element meshes is a one-type operation. That is, once you have saved the STAAD.Pro input file using these commands, the program will replace these commands with the resulting nodes and elements this command generates.

This is the second method for the generation of element incidences. If you needs to divide a big element into a number of small elements, you may use this facility which generates the joint numbers and joint coordinates, the element numbers and the element incidences automatically. Use of this feature consists of two parts:

1. Definition of the super-element boundary points: A super-element may be defined by either 4 boundary points or 8 boundary points (see figure below). A boundary point is denoted by a unique alphabet (A-Z in upper case or a-z in lower case) and its corresponding coordinates. Hence, any 4 or 8 of the 52 characters may be used to define the super-element boundary. If 4 points are used to define the super-element, each

side of the super-element will be assumed to have a straight edge connecting the 2 points defining that side. If 8 points are used, each side will be a smooth curve connecting the 3 points defining that side.

2. Generation of sub-elements: define the super-element using boundary points (4 or 8 as explained above) and specify the total number of sub-elements required.

General Format

```
DEFINE MESH
Ai xi yi zi ( { CYLINDRICAL, RCYLINDICRAL } ( x0, y0, z0 ) )
…
Aj xj yj zj ( { CYL, RCYL } ( x0, y0, z0 ) )
GENERATE ELEMENT { (QUADRILATERAL), TRIANGULAR }
MESH Ai Aj … n1 (n2)
MESH Am An … n3 (n4)
…
```

(up to 21 MESH input lines)

Where:

Ai*, *Aj	Letters A-Z or letters a-z. The maximum is 52.
xi*, *yi*, *zi	Coordinates for boundary point A_i.
x0*, *y0*, *z0	Optional Cartesian coordinates of the origin for cylindrical coordinates when CYL (cylindrical) or RCYL (reverse cylindrical) options are used. This defaults to the global origin of the model (0,0,0).

The 3 fields (*xi*, *yi*, *zi*) may be replaced by a joint number whose coordinates have been defined in the JOINT COORDINATE command by entering *Ai* JOINT *jn* instead.

***Ai*, *Aj*, *Ak*…**	A rectangular super-element defined by four or eight boundary points.
n1	Number of elements along the side $A_i A_j$ of the super-element. (Must not exceed 28).
n2	Number of elements along the side $A_j A_k$ of the super-element. (Must not exceed 28). If *n2* is omitted, that is, only *n1* is provided, then *n1* will indicate the total number of elements within the super-element. In this case, *n1* must be the square of an integer.

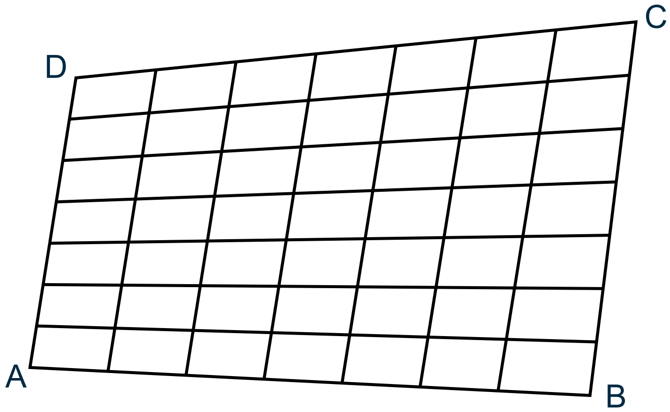
Limits

There is a limit of 21 Mesh commands. Up to 33,000 joints may be generated and up to 67,000 elements. Total number of joints in the model after this command is completed may not exceed 100,000.

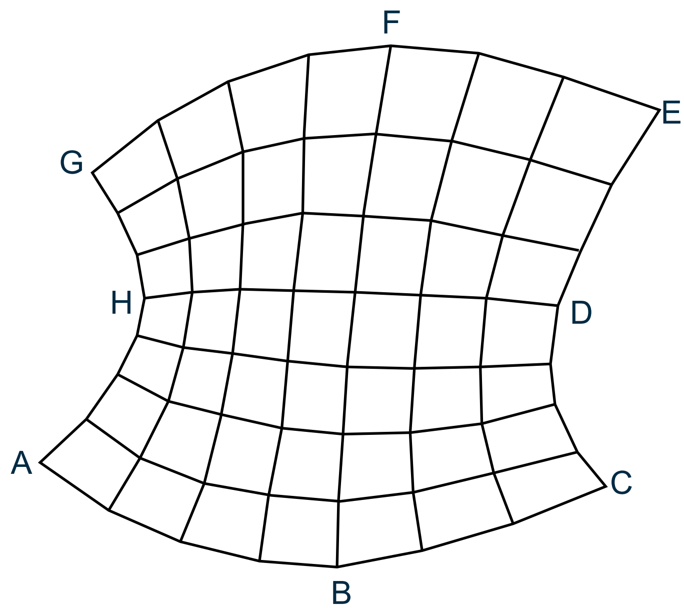
Notes

All coordinates are in current unit system. While using this facility you has to keep the following points in mind:

1. All super-elements must be 4-noded or 8-noded. Generated elements for 4-noded super-elements will retain the straight-line edges of the super-elements, while joints of elements generated from 8-noded super-elements will lie on a curved trajectory.



A) Mesh generated for a 4-noded super-element



B) Mesh generated for an 8-noded super-element

Figure 273: Mesh generation for super elements

2. Two super-elements, which have a common boundary, must have the same number of elements along their common boundary.
3. Sequence of super-elements - MESH commands define the super-elements. The sequence of this MESH command should be such that once one is defined, the next super-elements should be the ones connected to this. Therefore, for convenience, the first super-element should be the one, which is connected by the largest number of super-elements. In the example shown here for the tank, the bottom super-element is specified first.
4. This command must be used after the `MEMBER INCIDENCE` and `ELEMENT INCIDENCE` section and before the `MEMBER PROPERTIES` and `ELEMENT PROPERTIES` section. The elements that are created internally are numbered sequentially with an increment of one starting from the last member/element number plus one. Similarly the additional joints created internally are numbered sequentially with an increment of one starting from the last joint number plus one. It is advisable that users keep the joint numbers and member/element numbers in a sequence with an increment of one starting from one.
5. If there are members embracing a super-element which is being meshed, you must take care of the required additions/modifications in the `MEMBER INCIDENCE` section themselves since a few more new joints might appear on the existing common boundary as a result of meshing the super-element. See the following figure:

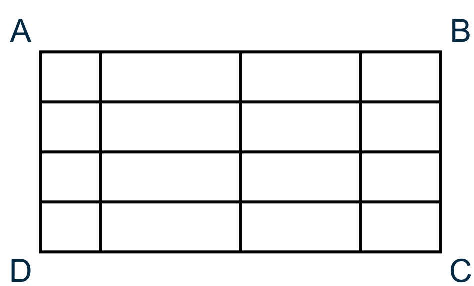


Figure 274: Additional joints on a super element

Note: If a member exists between points A and B, the user must breakup this member into 4 parts. Members will not be meshed automatically.

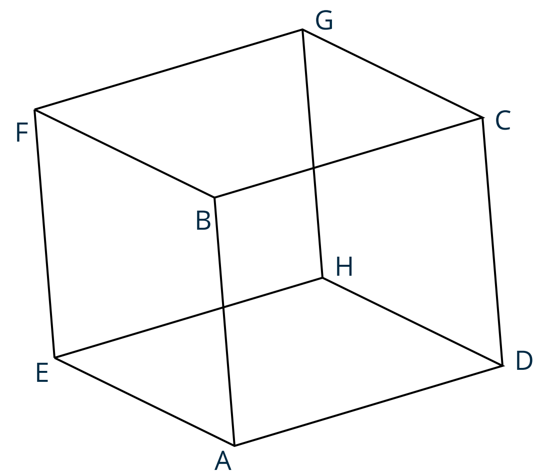
6. The sub-elements will have the same direction (Clockwise or Anti-clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. are in clockwise direction, CBAD or DCBA etc. are in anti-clock wise direction. If the particular super-element is denoted as ABCD, all the sub-elements in it will have a clockwise element incidence in this example.
7. Element incidences of the generated sub-elements may be obtained by providing the command 'PRINT ELEMENT INFORMATION' after the 'MESH...' command in the input file.
8. If the STAAD input file contains commands for `JOINT COORDINATES`, `MEMBER INCIDENCES`, `ELEMENT INCIDENCES`, and `MESH GENERATION`, they should be specified in the following order:

```
STAAD SPACE
UNIT  …
JOINT COORDINATES
…
MEMBER INCIDENCES
…
ELEMENT INCIDENCES
…
DEFINE MESH
…
GENERATE ELEMENT
…
```

9. Newly created joints will be merged with existing joints if they are within 0.001 inches of each other.

Example

The following section of input illustrates the use of `MESH GENERATION` facility, the user may compare this with the geometry inputs for Example Prob. No. 10:



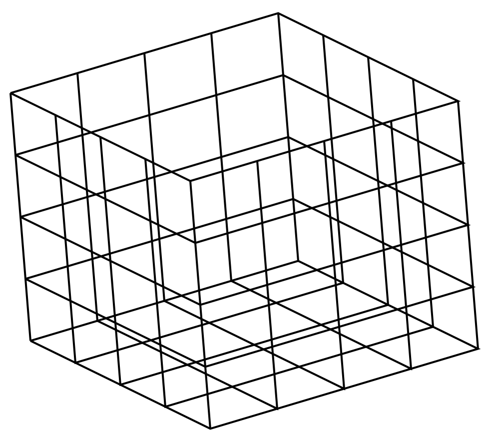
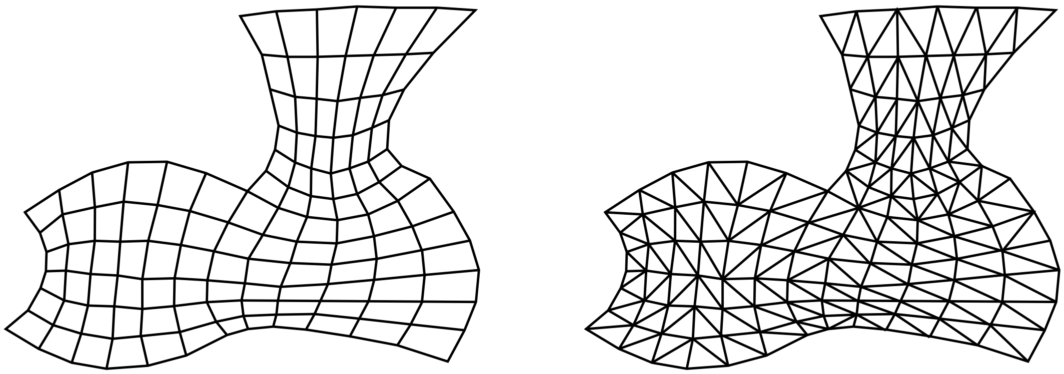


Figure 275: Mesh generation used in Example Problem 10

```
STAAD SPACE TANK STRUCTURE WITH
* MESH GENERATION
UNIT  . . .
DEFINE MESH
A 0 0 0 ; B 0 20 0 ; C 20 20 0
D 20 0 0 ; E 0 0 -20 ; F 0 20 -20
G 20 20 -20 ; H 20. 0. -20
GENERATE ELEMENT
MESH AEHD 16
MESH EABF 16
MESH ADCB 16
MESH HEFG 16
MESH DHGC 16
```



A) Typical generated Quad elements B) Typical generated Triangular elements

Figure 276: Typical generated elements

Related Links

- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)
- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)
- *EX. US-10 Finite Element Model for a Rectangular Tank* (on page 6380)
- *EX. UK-10 Finite Element Model for a Rectangular Tank* (on page 6665)

TR.15 Redefinition of Joint and Member Numbers

This command may be used to redefine `JOINT` and `MEMBER` numbers. Original `JOINT` and `MEMBER` numbers are substituted by new numbers.

General Format

```
SUBSTITUTE { { JOINT | MEMBER } { XRANGE | YRANGE | ZRANGE } | COLUMN } f1 f2 START i
```

Where:

f_1 and f_2 are two range values of x, y, or z
i is the new starting number.

Description

Joint and member numbers can be redefined in STAAD.Pro through the use of the `SUBSTITUTE` command. After a new set of numbers is assigned, input and output values will be in accordance with the new numbering scheme.

You can design numbering schemes that will result in simple input specification as well as easy interpretation of results. For example, all joints in first floor of a building may be renumbered as 101, 102, … etc., all second floor joints may be renumbered as 201, 202, …, etc.

Meaningful re-specification of `JOINT` and `MEMBER` numbers may significantly improve ease of interpretation of results.

This command may be in between incidence commands:

```
MEMBER INCIDENCE
…
SUBSTITUTE
ELEMENT INCIDENCE
…
```

Example

```
UNIT METER
SUBST JOINT YR  9.99  10.0 START  101
SUBST COLUMN START 901
```

Joints with Y coordinates ranging from 9.99 to 10 meters will have a new number starting from 101. Columns will be renumbered starting with the new number 901.

Related Links

- *G.22 Miscellaneous Facilities* (on page 2195)

TR.16 Entities as Single Objects

In the mathematical model, beams, columns, walls, slabs, block foundations, etc. are modeled using a collection of segments, which are known by the names members, plate elements, solid elements, etc. Hence, the bottom chord of a truss may be modeled using 5 members, with each member representing the segment between points where diagonals or vertical braces meet the bottom chord.

Often, it is convenient to cluster these segments under a single name so that assignment of properties, loads, design parameters. etc. is simplified. There are presently two options in STAAD for clustering entities - Group names and Physical members.

TR.16.1 Listing of Entities by Specifying Groups

This command allows you to specify a group of entities (e.g., members, joints, elements, etc.) and save the information using a 'group-name'. The 'group-name' may be subsequently used in the input file instead of a member/joint list to specify other attributes. This very useful feature allows avoiding of multiple specifications of the same member/joint list. Following is the general format required for the `GROUP` command.

General Format

```
START GROUP DEFINITION
```

followed by

```
(GEOMETRY)
_(group-name ) member/element/solid-list
…
```

or

```
JOINT
_(group-name ) joint-list
…
MEMBER
_(group-name ) member-list
…
ELEMENT
_(group-name ) element-list
SOLID
_(group-name ) solid-list
…
FLOOR
_(group-name ) member-list
…
```

followed by

```
END GROUP DEFINITION
```

Where:

group-name = an alphanumeric name specified to identify the group. The *group-name* must start with the '_' (underscore) character and is limited to 24 characters.
joint-list = the list of joints belonging to the group. TO, BY, and ALL are permitted.
member-list = the list of members/joints belonging to the group TO, BY, ALL, BEAM, PLATE, and SOLID are permitted.
ALL means all members, plates, *and* solids; BEAM means all beams; PLATE all plates; and SOLID all solids.

Notes

1. The GROUP definition must start with the START GROUP DEFINITION command and end with the END command.
2. More than one GROUP name may be specified within the same definition specification.
3. The words JOINT, MEMBER, ELEMENT, FLOOR, or SOLID may be provided if the you wish to identify the group name and lists with those specific items. However, if the group name and list is merely a means of grouping together more than one type of structural component under a single heading, the word GEOMETRY may be provided. In the absence of any of those words (GEOMETRY, JOINT, MEMBER, ELEMENT, FLOOR, or SOLID), the list is assumed to be that for GEOMETRY.

4. The same joint or member/element number may be included in up to four groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
5. If two or more consecutively entered groups have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
6. A member group may be used in lieu of a member-list with virtually any command which requires member lists, such as `MEMBER LOADS`, steel and concrete parameters, etc. There is one place however where a `MEMBER GROUP` will *not* suffice, and that is for defining panels during a `FLOOR LOAD` assignment.

   In Applying FLOOR LOAD onto a Floor Group (on page 2485), a panel has to be specified using a `FLOOR GROUP`, not a `MEMBER GROUP`. A `FLOOR GROUP` is not accepted in lieu of a member-list for any other command.
7. The maximum number of group allowed in an input file is equal to the total number of member, plates, solid, and nodes times the number of duplicate entities allowed to be in different group, which is by default 10 (may be changed using the `SET GROUP DUPLICATE i` command as described in TR.5 Set Command Specification (on page 2206)).

   For example, if a model has 10 members, 3 plates and 2 solids, and 100 nodes. The maximum number of groups could be $10 \cdot (10 + 3 + 2 + 100) = 1150$.
8. Group definitions must follow all geometry commands (e.g., joint coordinates, member/plate incidences) and any renumber (i.e., `SUBSTITUTE` commands). Likewise, group definitions must come before any load, reference loads, or load definition commands.

Example 1

```
START GROUP DEFINITION
_TRUSS 1 TO 20 25 35
_BEAM 40 TO 50
END
MEMBER PROPERTIES
_TRUSS TA LD L40304
_BEAM TA ST W12X26
```

Example 2

```
START GROUP DEFINITION
JOINT
_TAGA 1 TO 10
MEMBER
_TAGB 40 TO 50
GEOMETRY
_TAGC 101 TO 135
END

MEMBER PROPERTIES
_TAGB TA LD L40304
_TAGC TA ST W12X26
```

Related Links

- *M. To create a group from a selection* (on page 689)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)

- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.16.2 Physical Members

STAAD.Pro allows grouping analytical predefined members into physical members using a special member group `PMEMBER`. This command defines a group of analytical, collinear members with same cross section and material property.

To model using PMEMBER, you need to model regular analytical members and then, group those together.

While creating a PMEMBER, the following pre-requisites apply:

1. Existence of the analytical members in the member-list.
2. Selected members should be interconnected.
3. The selected individual members must be collinear (adjacent analytical members must lie within 5°).
4. Local axis of the individual members comprising the physical member should be identical (i.e., x, y and z are respectively parallel and in same sense).

   Tip: You may select the **Beam Tools** > **Beam Incidence** tool in the **Geometry Tools** group on the **Utilities** ribbon tab in the STAAD.Pro Analytical Modeling workflow for modifying analytical members which are pointing in the wrong direction

5. A member in one physical member group can not be part of any other physical member group.

Note: The `PMEMBER` option can only be used for codes which explicitly support that command. Refer to the appropriate design code specification to confirm if that code supports `PMEMBER` design.

Description

A `PMEMBER` can be created either in the Analytical Modeling workflow or in the Steel Designer workflow. Analytical Modeling workflow and Steel Designer workflow PMEMBERs will be labeled as M and D, respectively. Modeling workflow PMEMBERs allow variable cross-sections. Steel Designer workflow allows importing of PMEMBERs created in the modeling mode.

To define a Physical Member, the following command is used after the `MEMBER INCIDENCE` command:

```
DEFINE PMEMBER
```

```
Member-list  PMEMBER pmember-no
```

Example

```
JOINT COORDINATE
1 0 0 0 6 10.0 0 0
MEMBER INCIDENCE
1 1 2 5
DEFINE PMEMBER
1 TO 5 PMEMB 1
```

To define the member property of a Physical Member, the following command is used:

```
PMEMBER PROPERTY
```

```
pmember-list  PRIS …
```

The physical member supports all types of member properties available in STAAD.Pro.

If multiple definitions of member properties for a particular analytical member is encountered (e.g., analytical member properties is defined twice, once via PMEMBER PROP command and again via the MEMBER PROP command, then the MEMBER PROP command will override the PMEMBER PROP definition.

To define the material constants of a physical member, the following command is used:

```
PMEMBER CONSTANTS
```

```
E CONCRETE pmember-list
```

```
DEN CONCRETE pmember-list
```

```
...
```

Any member, which is a part of any PMEMBER is not allowed to be assigned constants explicitly.

Note: Loads are applied directly to physical members using the PMEMBER LOAD command. Refer to TR.32.2.1 PMember Load Specification (on page 2467) for details.

Design parameters are available for use with PMEMBERS by using the PMEMB list. The following syntax is used:

```
parameter value  PMEMB pmember-list
```

Example

```
RATIO 1.05 PMEMB 1 2
```

Note: There is not option to specify ALL for a PMEMB list, except for CODE CHECK or SELECT commands.

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

```
PRINT PMEMBER FORCE
```

This command will produce member forces for all the analytical members in the group.

Related Links

- *M. Physical Members* (on page 663)

TR.17 Rotation of Structure Geometry

This command may be used to rotate the currently defined joint coordinates (and the attached members/ elements) about the global axes. The rotated configuration is used for analysis and design. While specifying this command, the sense of the rotation should conform to the right hand rule.

General Format

```
PERFORM ROTATION *{ X d1 | Y d2 | Z d3 }
```

Where:

d1, *d2*, *d3* are the rotations (in degrees) about the X, Y, and Z global axes, respectively.

Example

```
PERFORM ROTATION X 20 Z -15
```

Related Links

- *G.22 Miscellaneous Facilities* (on page 2195)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)
- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.18 Inactive/Delete Specification

This set of commands may be used to temporarily inactivate or permanently delete specified `JOINT`s or `MEMBER`s.

General Format

```
INACTIVE { MEMBERS member-list | ELEMENTS element-list }
```

```
DELETE { MEMBERS member-list | JOINTS joint-list }
```

Description

These commands can be used to specify that certain joints or members be deactivated or completely deleted from a structure. The `INACTIVE` command makes the members and elements temporarily inactive; the user must re-activate them during the later part of the input for further processing. The `DELETE` command will completely delete the members/elements from the structure; you cannot re-activate them. The Delete Joint command must be immediately after the Joint Coordinates. The `DELETE` commands must be provided immediately after all member/element incidences are provided and before any `INACTIVE` commands.

Notes

a. The `DELETE MEMBER` command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements.

b. This command will also delete all the joints, which were not connected to the structure in the first place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a `DELETE MEMBER` command is used, a `DELETE JOINT` command should not be used.

c. The `DELETE MEMBER` command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to the next line by providing a blank space followed by a hyphen (-) at the end of the current line.

- **d.** If inactivating members causes joints to become unconnected in space (i.e., orphaned), a warning stating that those joints are not connected by any members or elements will be displayed in the output file. These warnings can normally be ignored.
- **e.** The inactivated members may be restored for further processes (such as an analysis or design for a 2nd set of load cases) by using the `CHANGE` command. See TR.37 Analysis Specification (on page 2620) and Example EX. US-4 Inactive Members in a Braced Frame (on page 6334) for more information.
- **f.** The `DELETE MEMBER` command should be used to delete elements too. Specify the command as `DELETE MEMBER j`, where *j* is the element number of the element you wish to delete. In the example shown below, 29 to 34 and 43 are element numbers.
- **g.** Loads that have been defined on members declared as `INACTIVE` members will not be considered in the analysis. This applies to `SELFWEIGHT`, `MEMBER LOADS`, `PRESTRESS`, and `POSTSTRESS LOADS`, `TEMPERATURE LOADs`, etc.

  For those joints that become orphaned (see note d above), loads applied to such joints, either explicitly through the `JOINT LOAD` command, or generated by some means as in moving, wind or seismic load generation, will be "lost", meaning, they will not get considered for the analysis for those load cases for which the associated member is inactivated.
- **h.** The `DELETE JOINT` command must be specified before all incidence commands such as `MEMBER INCIDENCE`, `ELEMENT INCIDENCE`, etc.

Example

```
INACTIVE MEMBERS 5 7 TO 10
DELETE MEMBERS 29 TO 34 43
```

Related Links

- *G.19 Multiple Analyses* (on page 2194)
- *EX. US-4 Inactive Members in a Braced Frame* (on page 6334)
- *EX. UK-4 Inactive Members in a Braced Frame* (on page 6618)

TR.19 User Steel Table Specification

STAAD.Pro allows you to create and use customized steel section tables for use in the property specification, code checking, and member selection. This set of commands may be used to create the tables and provide necessary data.

General Format

```
START  USER  TABLE
```

```
TABLE i1  (fn )
```

```
section-type
```

The following commands are repeated for each section within the user-table.

```
section-name
```

```
property-spec
```

For General sections, the following may be specified for each section:

```
(PROFILE_POINTS
zp1 yp1 zp2 yp2  … zpn ypn )
(STRESS_LOCATIONS
zs1 ys1 zs2 ys2 zs3 ys3 zs4 ys4 )
```

All tables end with the following command:

```
END
```

Where:

i1	table number (1 to 99). During the analysis process, the data in each user provided table is stored in a corresponding file with an extension `.U0`*n* . For example, the data of the 5th table is stored in `.U05`. The first part of the input file name is the same as that of the STAAD input file. These files are located in the same working directory as the input file. Hence, they may later be used as external user provided tables for other input files.
fn	external file path and file name containing the section name and corresponding properties (up to 72 characters). If an external file is used, the no other data should be provided for this table. **Note:** Hyphens (i.e., dashes) may not be used to break a line within the external file path and name. This line may be up to 250 characters long.
section-type	a steel section name including: `WIDE FLANGE`, `CHANNEL`, `ANGLE`, `DOUBLE ANGLE`, `TEE`, `PIPE`, `TUBE`, `GENERAL`, `ISECTION`, & `PRISMATIC`.
section-name	Any user designated section name, use 1 to 36 characters observing the following rules: • Only alphanumeric characters and digits are allowed for defining section names (i.e., spaces, asterisks, question marks, colons, underscores, semi-colons, etc. are *not* permitted). • The first three characters of the section names in pipe and tube tables must be `PIP` and `TUB`, respectively. • The first the characters of the section names in all other tables cannot start with `PIP` or `TUB`. • A name can be duplicated in different tables, but must be unique within any individual table.
property-spec	Properties for the section. The requirements are different for each section type as follows. Shear areas AY and AZ must be provided to ensure proper shear stress or shear strength calculations during design.

The default length units for properties are the current units. If a `UNIT` command is entered within the User Table in the input file then those units become the current units. However, a `UNIT` command on an external file only affects that file and has no effect on the units in subsequent input file commands. You may specify the desired length unit by using the `UNIT` command as the first command in the table (see example following this description).

If data is from input file, then use up to three lines of input per ***property-spec*** (end all but last with a hyphen, -).

General Section Definitions

The commands `PROFILE_POINTS` and `STRESS_LOCATIONS` can be used with a user-table general shape only. These define the vertices of a closed shape and the four locations to calculate stress, respectively.

zp1 yp1 zp2 yp2 ... zpn ypn are the X and Y pair of local coordinates which describe the points of the section. The points should be provided in either clockwise or counter-clockwise order.

zs1 ys1 zs2 ys2 zs3 ys3 zs4 ys4 are the X and Y pair of local coordinates which indicate where the stress is calculated. It is recommended that these start with the top,left-most point. The extreme points of the cross-section are then specified in a clockwise order around the section. Though any coordinates may be entered, points should lie on the section defined in order to represent true stress values.

Note: Hyphens (i.e., dashes) may not be used to break a line of PROFILE_POINTS or STRESS_LOCATIONS data.

Caution: Do not use comments (i.e., lines that start with an asterisk) within the PROFILE_POINTS and STRESS_LOCATIONS definitions, as these lines will *not* be ignored by the STAAD engine. This will result in unintended sections or errors.

Tip: Using the Graphical Interface is recommended for entering these points, as they may be entered with respect to any local coordinate system and will be translated to the section's center of mass and the section *property-spec* values will be calculated automatically. See M. To create a general section (on page 743) for details.

Example

```
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
P24X55-abcdefghijklmnopqrstuvwxyz111
16.2 23.57 0.375 7.005 0.505 1350 29.1 1.00688 8.83875 7.07505
P24X56
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
END
START USER TABLE
TABLE 2
GENERAL
L6x6x1
11 6 0 6 0 35.4621 35.4621 3.40525 8.57326 8.57326 5.09239 -
5.01615 21.4489 10.9903 8.98521 6
PROFILE_POINTS
 -1.86364 4.13636 -0.863636 4.13636 -0.863636 -0.863636 4.13636 -0.863636
4.13636 -1.86364 -1.86364 -1.86364
STRESS_LOCATIONS
 -1.8636 4.13636 -0.8638 -0.8636 4.13636 -1.8636 -1.8636 -1.8636
END
...
MEMBER PROPERTY
27 UPTABLE 1 P24X55-abcdefghijklmnopqrstuvwxyz111
39 UPTABLE 1 P24X56
```

The following example uses an external file, `C:\Structural Models\My_Profile.upt` (exported from Section Wizard), as the data source for a general section:

```
START USER TABLE
TABLE 1 C:\STRUCTURAL MODELS\MY_PROFILE.UPT
```

Related Links

- *G.6.3 User-Provided Steel Table* (on page 2117)

- *M. To create a general section* (on page 743)
- *G.6.3 User-Provided Steel Table* (on page 2117)

TR.19.1 Wide Flange

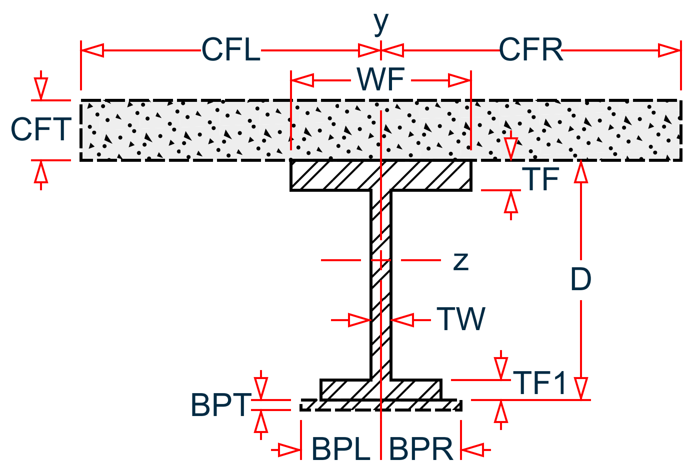


Figure 277: Wide flange section

General Format

For steel wide flange shapes:

```
property-spec  = AX D TW WF TF IZ IY IX AY AZ (WF1) (TF1)
```

For composite wide flange shapes:

```
property-spec  = -AX D TW WF TF IZ IY IX AY AZ (WF1) (TF1)
CFR CFL CFT MR
```

For composite wide flange shapes with bottom cover plate:

```
property-spec  = -AX D TW WF TF IZ IY IX AY AZ (WF1) (TF1)
-CFR CFL CFT MR
BPR BPL BPT
```

Note: There is a leading dash on the first line of section property specifications when composite slab properties are specified and again at the beginning of the composite slab properties when bottom cover plate dimensions are specified.

AX	Cross section area
D	Depth of the section
TW	Thickness of web
WF	Width of the top flange (or both flanges when `WF1` is not specified)
TF	Thickness of top flange (or both flanges when `WF1` is not specified)
IZ	Moment of inertia about local z-axis (usually strong axis)

IY Moment of inertia about local y-axis

IX Torsional constant

AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

WF1 (Optional) Width of the bottom flange. The width of the top flange will be used if this value is not specified.

TF1 (Optional) Thickness of bottom flange. The thickness of the top flange will be used if this value is not specified.

The following option parameters are used to include a composite concrete slab. If included, these must be on a separate line following a dash, -, on the line containing the required section parameters.

CFR Width of the composite slab to the right of the web center line

CFL Width of the composite slab to the left of the web center line

CFT Thickness of the composite slab

MR Modular ratio of the concrete in the composite slab

The following option parameters are used to include a bottom flange cover plate.

Tip: Bottom cover plates can only be added in association with a composite slab.

BPR Width of the additional bottom flange plate to the left of the web center line

BPL Width of the additional bottom flange plate to the right of the web center line

BPT Thickness of the additional bottom flange plate

Example

```
START USER TABLE
TABLE 1
UNIT MMS
WIDE FLANGE
UNEQUAL_FLANGE_I
16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 300 17
UNEQUAL_FLANGE_COMP_I
-16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 -
300 17
250 350 75 9.1
UNEQUAL_FLANGE_COMP_BOTPLT_I
-16855 600 10 405 15 1.10087e+009 1.21335e+008 1.13626e+006 6000 7450 -
300 17
-250 350 75 9.1
120 100 25
END
```

Related Links

- *M.To create a wide-flange user table section* (on page 731)

TR.19.2 Channel

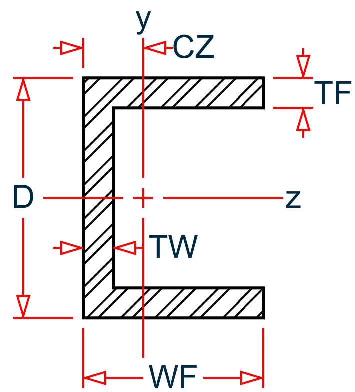


Figure 278: Channel section

`property-spec = AX D TW WF TF IZ IY IX CZ AY AZ`

Where:

AX	Cross section area
D	Depth of the section
TW	Thickness of web
WF	Width of flange
TF	Thickness of flange
IZ	Moment of inertia about local z-axis (usually strong axis)
IY	Moment of inertia about local y-axis
IX	Torsional constant
CZ	Distance from back of web to center of gravity (C.G.) of the shape along the local z-axis.
AY	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
AZ	Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

Related Links

- *M.To create a channel user table section* (on page 733)

TR.19.3 Angle

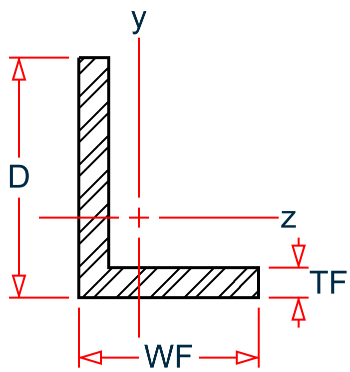


Figure 279: Angle section

```
property-spec = D WF TF R AY AZ
```

D Depth of angle (i.e., length of the leg along the local y-axis)

WF Width of angle (i.e., length of the leg along the local z-axis)

TF Thickness of angle leg

R Radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual (this must not be zero)

AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

Tip: If adding an user-provided table angle in the user interface, you can have the program calculate R, AY, and AZ values.

Related Links

- *M.To create an angle user table section* (on page 735)

TR.19.4 Double Angle

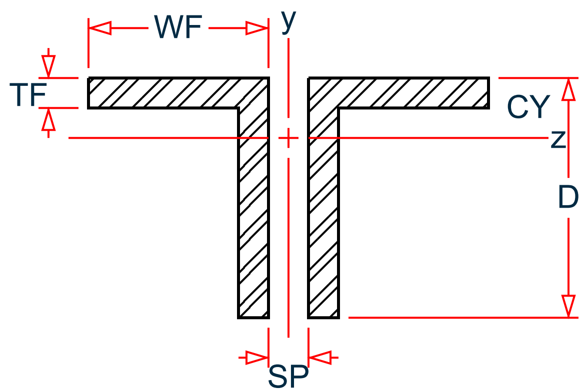


Figure 280: Double angle section

```
property-spec = D WF TF SP IZ IY IX CY AY AZ RVV
```

D Depth of angle (i.e., length of the leg along the local y-axis)

WF Width of angle (i.e., length of the leg along the local z-axis)

TF Thickness of angle leg

SP Space between angles.

IZ Moment of inertia about local z-axis (usually strong axis)

IY Moment of inertia about local y-axis

IX Torsional constant

CY Distance from the face of angles to the center of gravity (C.G.) along the local y-axis.

AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

RVV The radius of gyration about the minor principal axis for single angles - z-z axis for `TA ST` angles, and y-y axis for `TA RA` angles.

Related Links

- *M.To create a double angle user table section* (on page 737)

TR.19.5 Tee

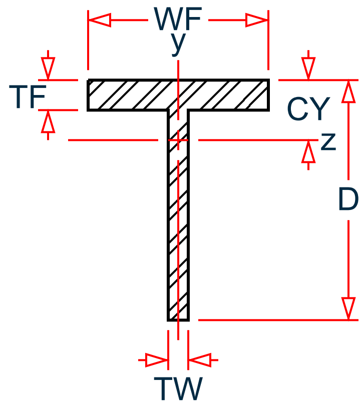


Figure 281: Tee section

```
property-spec = AX D WF TF TW IZ IY IX CY AY AZ
```

AX Cross section area

D Depth of the section

WF Width of flange

TF Thickness of flange

TW Thickness of web

IZ Moment of inertia about local z-axis (usually strong axis)

IY Moment of inertia about local y-axis

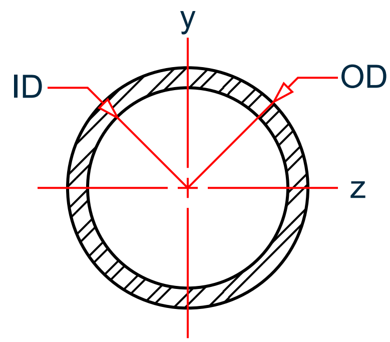
IX Torsional constant

CY Distance from back of web to center of gravity (C.G.) of the shape along the local y-axis.

AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

TR.19.6 Pipe




```
property-spec = OD ID AY AZ
```

OD Outer diameter

ID Inner diameter

AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

TR.19.7 Tube

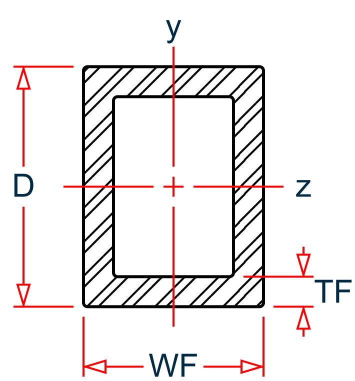


Figure 282: Tube section

```
property-spec = AX D WF TF IZ IY IX AY AZ
```

AX Cross section area

D Depth of the section

WF Section width

TF Tube wall thickness

IZ Moment of inertia about local z-axis (usually strong axis)

IY Moment of inertia about local y-axis

IX Torsional constant

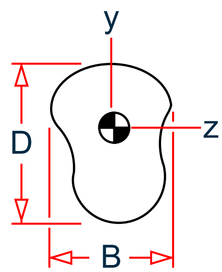
AY Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.

AZ Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

TR.19.8 General

The following cross-sectional properties should be used for this section-type. This facility allows you to specify a built-up or unconventional steel section. Provide both the Y and Z parameters for design or code checking.

Note: STAAD.Pro can perform some code checks on a General section where as a Prismatic section (TR.20.2 Prismatic Property Specification (on page 2263)) is for analysis only.




```
property-spec = AX D TD B TB IZ IY IX SZ SY AY AZ PZ PY HSS DEE
```

AX Cross section area

D Depth of the section

TD Thickness associated with section element parallel to depth (usually web). To be used to check depth/ thickness ratio

B Width of the section

TB Thickness associated with section element parallel to flange. To be used to check width/thickness ratio

IZ Moment of inertia about local z-axis (usually strong axis)

IY Moment of inertia about local y-axis

IX Torsional constant

SZ Section modulus about local z-axis

SY Section modulus about local y-axis

AY Shear area in local y-axis.

AZ Shear area in local z-axis.

PZ Plastic modulus about local z-axis

PY Plastic modulus about local y-axis

HSS Warping constant for lateral torsional buckling calculations

DEE Depth of web. For rolled sections, distance between fillets should be provided

Note: Properties PZ, PY, HSS, and DEE must be provided for code checking or member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

You can also specify points to define a general section. Refer to TR.19 User Steel Table Specification (on page 2241) for details. When this method is used, the section property-spec values can be calculated automatically by the graphical interface.

TR.19.9 I Section

This section type may be used to specify a generalized I-shaped section. The cross-sectional properties required are listed below. This section can be used to specify tapered I-shapes.

```
ISECTION
```

```
section-name
```

```
-DWW TWW DWW1 BFF TFF BFF1 TFF1 AYF AZF XIF
```

```
CFL -
```

```
CFR CFT MR
```

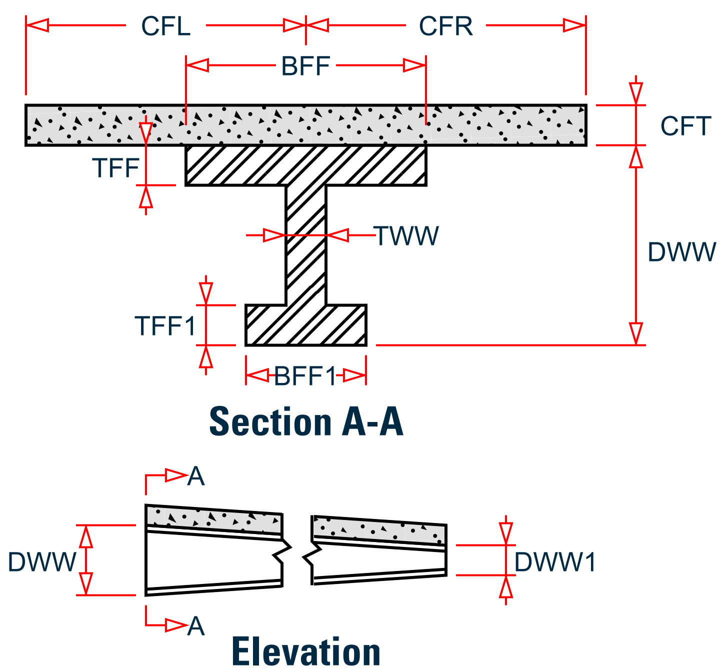


Figure 283: I section and tapered web

DWW	Depth of section at start node
TWW	Thickness of web
DWW1	Depth of section at end node (see note #1)
BFF	Width of top flange

TFF	Thickness of top flange
BFF1	Width of bottom flange
TFF1	Thickness of bottom flange
AYF	Shear area for shear parallel to Y-axis (see note #2)
AZF	Shear area for shear parallel to Z-axis (see note #2)
XIF	Torsional constant (IX or J) (see note #2)

The following option parameters are used to include a composite concrete slab. If included, these must be on a separate line following a dash, **-**, on the line containing the required section parameters.

CFR	Width of the composite slab to the right of the web center line
CFL	Width of the composite slab to the left of the web center line
CFT	Thickness of the composite slab
MR	Modular ratio of the concrete in the composite slab

Notes

1. `DWW` should never be less than `DWW1`. Therefore, you must provide the member incidences accordingly.
2. You are allowed the following options for the values `AYF`, `AZF`, and `XIF`:
   - If positive values are provided, they are used directly by the program.
   - If zero is provided, the program calculates the properties using the following formulae:

     $$AYF = D \times TWW$$

     $$AZF = 0.66\ ((BFF \times TFF) + (BFF1 \times TFF1))$$

     $$XIF = 1/3\ ((BFF \times TFF^3) + (DEE \times TWW^3) + (BFF1 \times TFF1^3))$$

     where

D	=	Depth at section under consideration
DEE	=	Depth of web of section

   - If negative values are provided, they are applied as factors on the corresponding value(s) calculated by the program using the above formula. The factor applied is always the absolute of the value provided. For example, if you specify a value of XIF as `-1.3`, then the program will multiply the value of XIF, calculated by the above formula, by a factor of 1.3.

Related Links

- *M.To create an I shape user table section* (on page 739)

TR.19.10 Prismatic

The property-spec for the `PRISMATIC` section-type is as follows:

AX	Cross-section area
IZ	Moment of inertia about the local z-axis
IY	Moment of inertia about the local y-axis
IX	Torsional constant

AY Shear area for shear parallel to local y-axis.

AZ Shear area for shear parallel to local z-axis.

YD Depth of the section in the direction of the local y-axis.

ZD Depth of the section in the direction of the local z-axis

Note: When listing multiple shapes, those section names must be provided in ascending order by weight since the member-selection process uses these tables and the iteration starts from the top.

Example

```
START USER TABLE
TABLE  1
UNIT . . .
WIDE FLANGE
W14X30
8.85  13.84  .27  6.73  .385  291.  19.6  .38  0  0
W21X50
14.7  20.83  .38  6.53  .535  984  24.9  1.14  7.92  0
W14X109
32.  14.32  .525  14.605  .86  1240  447  7.12  7.52  0
TABLE  2
UNIT . . .
ANGLES
L25255
2.5  2.5  0.3125 .489  0  0
L40404
4.  4.  .25  .795  0  0
END
```

Related Links

- *M.To create an prismatic user table section* (on page 741)

TR.19.11 Using Reference Table Files

The User-Provided Steel Tables may be created and maintained as separate files. The same files may be used for all models using sections from these tables. These files should reside in the same directory where the input file is located.

```
START USER TABLE
TABLE  1 TFILE1
TABLE  2 TFILE2
END
```

Where `TFILE1` and `TFILE2` are names of files which must be created prior to running STAAD, and where the file `TFILE1` contains the following:

On each file the first table should contain a `UNITS` command.

```
UNIT  . . .
WIDE FLANGE
W14X30
```

```
8.85  13.84  .27  6.73  .385  291.  19.6  .38  0  0
W21X50
14.7  20.83  .38  6.53  .535  984  24.9  1.14  7.92  0
W14X109
32.  14.32  0.525  14.605  .86  1240  447  7.12  7.52  0
```

and the file *TFILE2* contains:

```
UNIT  . . .
ANGLES
L25255
2.5  2.5  .3125  .489  0  0
L40404
4.  4.  .25  .795  0  0
```

TR.20 Member Property Specification

This set of commands may be used for specification of section properties for frame members.

The options for assigning properties come under two broad categories:

1. Those which are specified from built-in property tables supplied with the program, such as for steel, aluminum and timber.
2. Those which are *not* assigned from built-in tables, but instead are specified on a project-specific basis, such as for concrete beams and columns, or custom-made sections for industrial structures.

Properties which are specified from built-in property tables

1. General format for standard steel (hot rolled):

```
MEMBER PROPERTIES { AMERICAN | APLAPOLLOTUBE | AUSTRALIAN | BRAZILIAN | BRITISH |
CANADIAN | CHINESE | DUTCH | EUROPEAN | FRENCH | GERMAN | INDIAN | JAPANESE |
JINDAL | KOREAN | MEXICAN | RUSSIAN | SAFRICAN | SPANISH | STOASCHM | TATASTRUCTURA
| VENEZUELAN }
```

```
member-list  { TABLE type-spec section-name-in-table (additional-spec) | ASSIGN
profile-spec }
```

AMERICAN, BRITISH, EUROPEAN (etc.) option will instruct the program to read properties from the appropriate steel table. The default depends on the country of distribution.

- Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2257) for type-specs and additional-specs.
- Refer to TR.20.5 Assign Profile Specification (on page 2268) for ASSIGN profile-spec.
- contain information on the section types which can be assigned for the various countries named in the above list.
- Refer to TR.20.6 Examples of Member Property Specification (on page 2270) for examples.

The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

2. General format for cold formed steel:

```
MEMBER PROPERTIES { BUTLER | COLD AMERICAN | COLD AUSTRAILIAN | COLD BRITISH | COLD
EUROPEAN | COLD INDIAN | COLD JAPANESE | COLD RUSSIAN | KINGSPAN | LYSAGHT }
```

```
member-list TABLE ST section-name-in-table
```

Refer to D. Design Codes (on page 1086) information on the section types which can be assigned for the various countries/organizations named in the above list.

3. General format for steel joist:

```
MEMBER PROPERTIES SJIJOIST
```

```
member-list TABLE ST section-name-in-table
```

US Steel Joist Sections (on page 2261) contains information on the joist types which can be assigned from the Steel Joist Institute's tables.

4. General format for Aluminum:

```
MEMBER PROPERTIES ALUMINUM
```

```
member-list TABLE ST section-name-in-table
```

D1.H. American Codes - Aluminum Design per 1994 ADM (on page 1264) contains contains information on the section types which can be assigned for the aluminum table in the above list.

5. General format for Timber:

```
MEMBER PROPERTIES { AITC | TIMBER CANADIAN }
```

```
member-list TABLE ST section-name-in-table
```

D1.H. American Codes - Aluminum Design per 1994 ADM (on page 1264) contains information on the section types which can be assigned for the above list.

Properties that are *not* specified from built-in property tables

```
MEMBER PROPERTIES
```

```
member-list { PRISMATIC property-spec | TAPERED argument-list | UPTABLE i1section-
name }
```

- Refer to TR.20.2 Prismatic Property Specification (on page 2263) for specification of PRISMATIC properties.
- Refer to TR.20.3 Tapered Member Specification (on page 2266) for specification of TAPERED members.
- Refer to TR.20.4 Property Specification from User Provided Table (on page 2267) for specification from UPTABLES.
- Refer to TR.20.6 Examples of Member Property Specification (on page 2270) for examples.

The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

Related Links

TR.31.2.16 IBC 2015 Seismic Load Definition

- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *M. To add a new table section property* (on page 715)
- *G.6 Member Properties* (on page 2113)

TR.20.1 Assigning Properties from Steel Tables

The following commands are used for specifying section properties from built-in steel table(s). The section type specification is followed by additional specifications as needed.

General Format

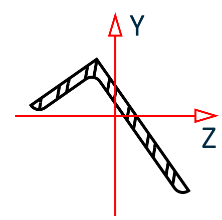
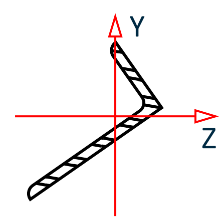
```
member-list TABLE type-spec  table-name additional-spec
```

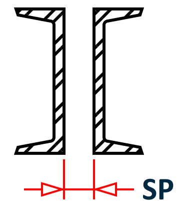
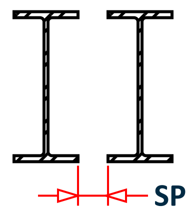
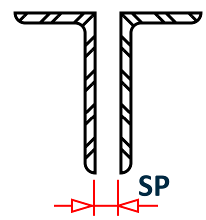
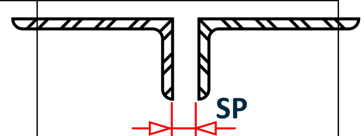
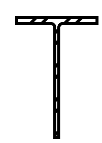
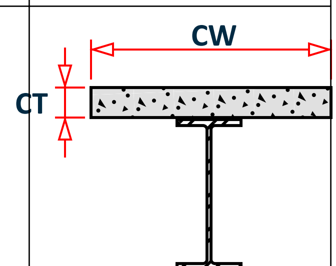
Where:

member-list A list of member numbers.

type-spec `= { ST | RA | D | LD | SD | T | CM | TC | BC | TB | FR }`

Table 234: Type-specs for various profile types

type-spec	**Description**	**Diagram**
ST	standard, single section from the built-in tables	
RA	single reverse angle (reverse Y-Z axes, refer to G. 4.2 Local Coordinate System (on page 2088))	

type-spec	**Description**	**Diagram**
D	double profile. In the case of channels, back-to-back The spacing between shapes is provided using the SP additional specification.	
LD	double angle with long legs back-to-back	
SD	double angle with short legs back-to-back	
T	tee section cut from I shaped section	
CM	composite section, available for I shaped sections	

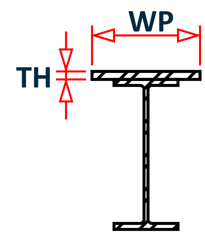
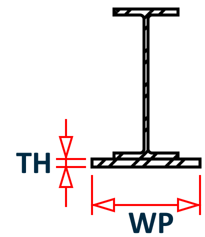
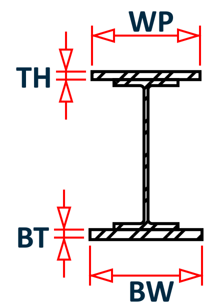
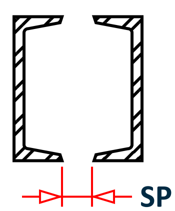
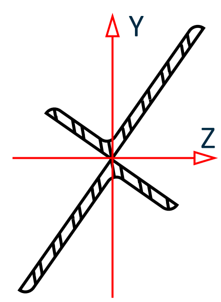
type-spec	**Description**	**Diagram**
TC	section with top cover plate	
BC	section with bottom cover plate	
TB	section with both top and bottom cover plate top plate dimensions are described using `WP` and `TH` parameter values and bottom plate dimensions are described using `BW` and `BT` parameter values	
FR	front-to-front (i.e., toe-to-toe) channels. **Note:** Spacing between the channels must be provided using the SP option mentioned in the additional spec specification described below.	
SA	double angle in a star arrangement (heel to heel)	

table-name Table section name like W8X18, C15X33 etc.

The documentation on steel design per individual country codes contains information regarding their steel section specification also. For details on specifying sections from the American steel tables, refer to D1.A.4 Built-in Steel Section Library (on page 1088).

additional-spec

```
= * {SP f1 | WP f2 | TH f3 | WT f4 | DT f5 | OD f6 | ID f7 | CT f8 |
FC f9 | CW f10 | CD f11 | BW f12 | BT f13 }
```

Table 235: Additional specifications for steel sections

Variable	Description
SP *f1*	Either: • Spacing between double angles, double channels, or double wide-flange sections (i.e., D, LD, SD, or FR). The default spacing is 0.0. • Rib height for composite sections
WP *f2*	Width of top cover plate for I shaped sections with cover plate.
TH *f3*	Thickness of plates (top plate for I shaped sections) or wall thickness for a parametric tube section.
WT *f4*	Width of a parametric tube section.
DT *f5*	Depth of a parametric tube section.
OD *f6*	Outer diameter of a parametric pipe section.
ID *f7*	Inside diameter of a parametric pipe section.
CT *f8*	Concrete thickness of the concrete for composite sections.
FC *f9*	Compressive strength of concrete for composite sections.
CW *f10*	Concrete width for composite sections.
CD *f11*	Concrete density for composite sections (default is 150 pounds per cubic ft.)
BW *f12*	With of bottom cover plate for I shaped sections with cover plates

Variable	Description
BT *f13*	Thickness of cover plates for I shaped sections with cover

Refer to G.6.2 Built-In Steel Section Libraries (on page 2116) for more information.

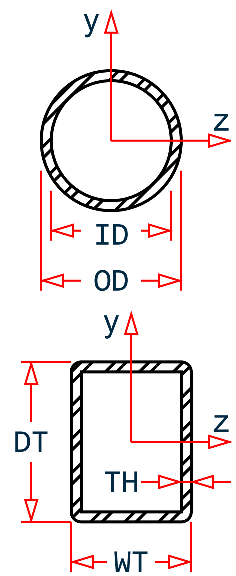
Parametric Pipe and Tube Sections

In lieu of specifying a table shape, you can specify pipe and tube sections parametrically. Only the standard (ST) single sections are used for pipes and tubes for table sections. So the input format for a pipe section is:

```
member-list TABLE ST PIPE OD f6 ID f7
```

The input format for a tube section is:

```
member-list TABLE ST TUBE DT f5 WT f4 TH f3
```



US Steel Joist Sections

Several tables of US steel joists and joist girders are available for use. Member properties can be assigned by specifying a joist designation contained in tables supplied with the program. The following joists and joist girder types have been implemented:

Open web steel joists – K series and KCS joists
Longspan steel joists – LH series
Deep Longspan steel joists – DLH series
Joist Girders – G series

The format for specifying steel joists is

```
MEMBER PROPERTIES SJIJOIST
```

```
member-list TABLE ST section-name-in-table
```

The STAAD.Pro joists database includes the weight per length of the joists. Thus, the weight of the joist is automatically considered for selfweight computations in the model.

The designation for most joists (K, KCS, LH, and DLH series) are similar to those given in the SJI publication (e.g., 16K2, 22KCS4, 24LH05, 60DLH13).

The designation for the G series Joist Girders is as shown in the Steel Joist Institute publication. STAAD.Pro incorporates the span length also in the name, as shown in the next figure.

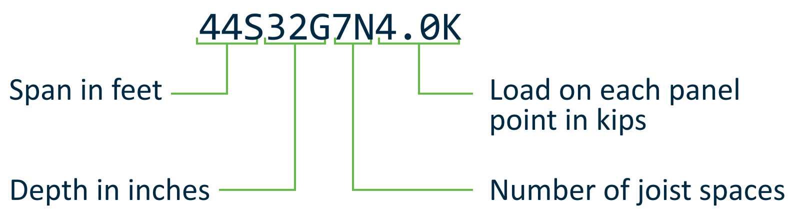


Figure 284: STAAD.Pro nomenclature for SJI joist girders

Examples

```
UNIT  …
MEMBER PROPERTIES
1  TO  5  TABLE  ST  W8X31
9  10  TABLE  LD  L40304  SP  0.25
12  TO  15  PRISMATIC  AX  10.0  IZ  1520.0
17  18  TA  ST  PIPE  OD  2.5  ID  1.75
20  TO  25  TA  ST  TUBE  DT  12.  WT  8.  TH  0.5
27  29  32  TO  40  42  PR  AX  5.  IZ  400.  IY  33. -
IX  0.2  YD  9.  ZD  3.
43  TO  47  UPT  1  W10X49
50  51  UPT  2  L40404
52  TO  55  ASSIGN  COLUMN
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0
```

See TR.20.6 Examples of Member Property Specification (on page 2270) for additional details on this example.

Notes

All values *f1* - *f9* must be supplied in current units.

Some important points to note in the case of the composite section are:

1. The CM parameter can be assigned to I-shaped sections only. A CM (composite) section is one obtained by considering a portion of a concrete slab to act in unison with the I shaped steel section. FC is the strength or grade of concrete used in the slab. In the USA, F_C is called the specified compressive strength of concrete. Typical values of F_C range between 2.0 and 5.0 ksi, and 20 to 50 Mpa.
2. The width of the concrete slab (CW) (if not entered) is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab.
3. In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:

   E_s = Modulus of elasticity of steel = 29,000 ksi.
   E_c = Modulus of elasticity of concrete = $1{,}802.5\sqrt{(F_c)}$ ksi

Where F_c (in ksi) is defined earlier.

Some other general notes on this subject of member property designations are:

1. The T parameter stands for a T-shaped section obtained by cutting an I-shaped section at exactly its mid height level along the web. Hence, the area of a T shape is exactly half the area of the corresponding I shape. The depth of a T shape is half the depth of the I shape it was cut from.

   What we refer to as I shaped sections are sections which look like the English alphabet I. The American Wide Flange, the British UB and UC sections, Japanese H sections, etc., all fall under this category. Consequently, the 'T' shape cut from a Japanese H shape is one obtained by cutting the H shape at exactly its mid-height level of the web.

   Not all I shaped sections have a corresponding T. This may be inferred by going through the section libraries of individual countries and organizations. In such cases, if a user were to specify such a T section, the program will terminate with the message that the section does not exist.
2. Steel Cover plates also can be added only to I shaped sections. Thus, the TC, BC, and TB are not applicable to any shape other than an I shape.

Related Links

- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *M. To add a new table section property* (on page 715)
- *M. Section Database Profiles* (on page 715)
- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *G.6.7 Composite Beams and Composite Decks* (on page 2119)

TR.20.2 Prismatic Property Specification

The following commands are used to specify section properties for prismatic cross-sections.

General Format

For the PRISMATIC specification, properties are provided directly (End each line but last with a hyphen "-") as follows:

```
property-spec  = * { AX f1 | IX f2 | IY f3 | IZ f4 | AY f5 | AZ f6 | YD f7 | ZD f8 |
YB f9 | ZB f10 }
```

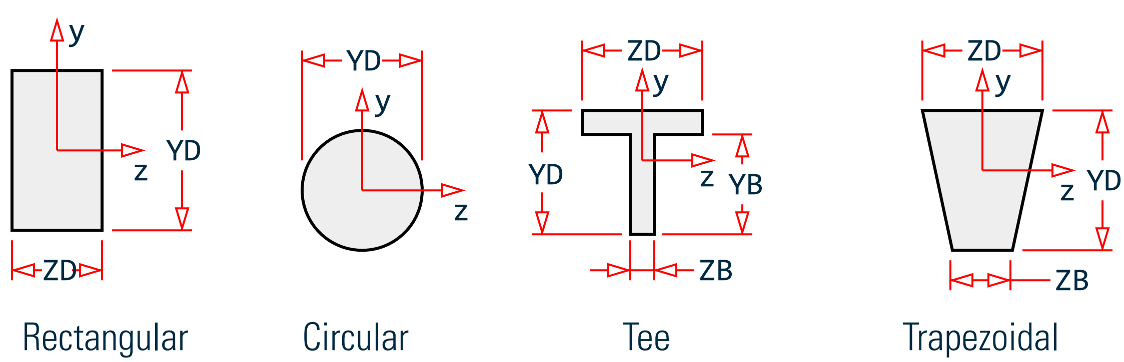


Figure 285: Prismatic property nomenclature for a Tee and Trapezoidal section

Where:

`AX` *f1* = Cross sectional area of the member. Set to zero for TEE, Rectangular, Trapezoid, or circular.
`IX` *f2* = Torsional constant.
`IY` *f3* = Moment of inertia about local y-axis.
`IZ` *f4* = Moment of inertia about local z-axis (usually major).
`AY` *f5* = Effective shear area in local y-axis.
`AZ` *f6* = Effective shear area in local z-axis.

If any of the previous six parameters are omitted, it will be calculated from the YD, ZD, YB, and/or ZB dimensions.

`YD` *f7* = Depth of the member in local y direction. Used as the diameter of section for circular members.
`ZD` *f8* = Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.
`YB` *f9* = Depth of stem for T-section.
`ZB` *f10* = Width of stem for T-section or bottom width for TRAPEZOIDAL section.

The values that STAAD calculates for the omitted terms can be obtained by specifying the command `PRINT MEMBER PROPERTIES`.

The values of many of the derived properties like shear areas (AY, AZ), section modulii (SY, SZ), etc. will be shown in the output file.

This command can be used regardless of the manner in which the properties are specified (e.g., PRISMATIC, user table, built-in table).

Related Links

- *G.6.1 Prismatic Properties* (on page 2114)
- *M. To assign a prismatic section* (on page 728)
- *D2.A.1 Section Types for Concrete Design* (on page 1350)
- *D4.A.1 Section Types for Concrete Design* (on page 1413)
- *D8.E.1 Section Types for Concrete Design* (on page 1669)
- *D8.G.2 Section Types for Concrete Design* (on page 1698)
- *D9.A.1 Section Types for Concrete Design* (on page 1713)
- *D10.A.2 Section Types for Concrete Design* (on page 1743)
- *D14.A.2 Member Dimensions* (on page 1951)
- *G.6.1 Prismatic Properties* (on page 2114)

TR.20.2.1 Prismatic Tapered Tube Property Specification

The following commands are used to specify section properties for prismatic tapered tube cross-sections. For the property types shown below, additional information can be obtained from Table 2.1 of the ASCE 72 document, 2nd edition.

General Format

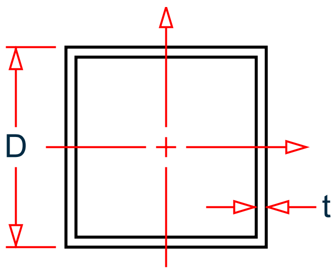
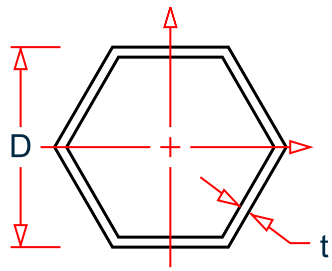
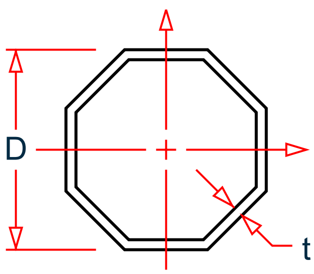
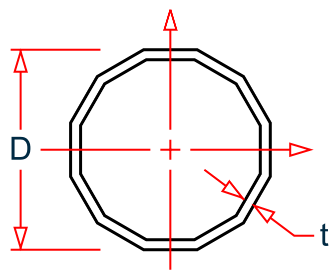
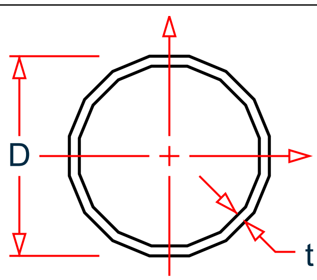
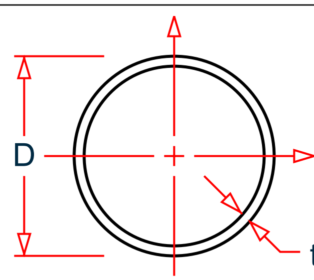
```
property-spec  = { ROUND | HEXDECAGONAL | DODECAGONAL | OCTAGONAL | HEXAGONAL |
SQUARE } START D1 END D2 THICK t
```

Where:

START *D1*	Depth of section at start of member.

END *D2*	Depth of section at end of member.
THICK *t*	Thickness of section (constant throughout the member length).

Figure 286: Prismatic tapered tube shapes

 Square	 Hexagonal (6 sides)
 Octagonal (8 sides)	 Dodecagonal (12 sides)
 Hexdecagonal (16 sides)	 Round

Notes

a. Section properties are calculated using the rules applicable for thin-walled sections.

b. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the `SET SHEAR` command has no effect on the deformation computed for members with these cross sections.

Example

```
UNIT …
MEMBER PROPERTY
1 PRIS ROUND STA 10 END  8  THI 0.375
```

```
2 PRIS HDC STA 15 END 10 THI 0.375
3 PRIS DOD STA 12 END 12 THI 0.375
```

See `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\US\Square_tapered.std` and `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\US\Round_tapered.std` for additional examples of designing members with prismatic tapered tube sections.

Related Links

- *G.6.1 Prismatic Properties* (on page 2114)
- *M. To assign a prismatic section* (on page 728)

TR.20.3 Tapered Member Specification

The following commands are used to specify section properties for tapered I-shapes.

General Format

```
argument-list  = f1 f2 f3 f4 f5 (f6 f7)
```

Where:

f1 Depth of section at start node. Must be greater than *f3*.

f2 Thickness of web.

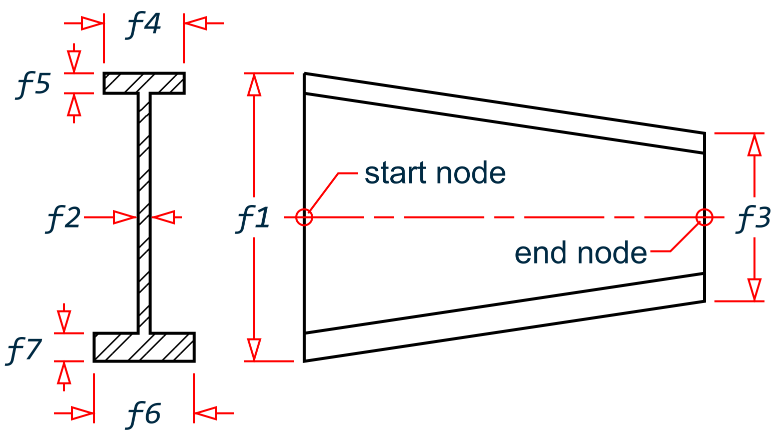
f3 Depth of section at end node.

f4 Width of top flange.

f5 Thickness of top flange.

f6 Width of bottom flange. Defaults to *f4* if left out.

f7 Thickness of bottom flange. Defaults to *f5* left out



Example

```
MEMBER PROPERTY
1 TO 5 TAPERED 15.98 0.285 11.98 6.745 .455 6.745 .455
```

See `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\US\Doubly_Symm-I_tapered.std` for an additional example of designing a member with a tapered I section.

Notes

a. All dimensions (*f1*, *f2*, ... *f7*) should be in current units.

b. *f1* (Depth of section at start node) should always be greater than *f3* (Depth of section at end node). You must provide the member incidences accordingly.

c. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the `SET SHEAR` command has no effect on the deformation computed for members with these cross sections.

Related Links

- *G.6.4 Tapered Sections* (on page 2117)
- *M. To assign a tapered I section* (on page 729)
- *G.6.4 Tapered Sections* (on page 2117)
- *D1.A.5.7 Design of Web-Tapered Members* (on page 1098)

TR.20.4 Property Specification from User Provided Table

The following commands are used to specify section properties from a previously created `USER-PROVIDED STEEL TABLE`.

General Format

```
member-list UPTABLE i1section-name
```

Where:

`UPTABLE` stands for User-Provided TABLE
i_1 = table number as specified previously (1 to 20)
section-name = section name as specified in the table.

See TR.20.6 Examples of Member Property Specification (on page 2270) for an example.

Related Links

- *G.6.3 User-Provided Steel Table* (on page 2117)
- *M. To create a general section* (on page 743)
- *G.6.3 User-Provided Steel Table* (on page 2117)

TR.20.5 Assign Profile Specification

The ASSIGN command may be used to instruct the program to assign a suitable steel section to a frame member based on the profile-spec shown below. The current country of the member properties is used to search for a suitable size based on the profile spec provided, if a such a shape is included in that country table.

General Format

```
profile-spec  = { BEAM | COLUMN | CHANNEL | ANGLE (DOUBLE) }
```

See TR.20.6 Examples of Member Property Specification (on page 2270) for an example.

Note: Sections are always chosen from the relevant built-in steel table. To find out the details of the sections that are chosen, the command PRINT MEMBER PROPERTIES should be provided after specification of all member properties. These commands may not work with certain tables such as cold-formed steel, Timber, Aluminum, and even some of the standard steel tables.

Sections Provided based on Country

The following table is a list of the actual section that is applied for each database and ASSIGN type.

Country	Beam	Column	Channel	Angle
0 US	W18X40	W14X43	C10X20	L40406
1 BRIT	UB457X191X89	UC254X254X107	CH229X76	UA100X100X12
10 SOUTH AFRICA	457x75UB	305x198UC	260x90x38C	100x100x10L
11 SPAIN	IPE450	HEA360	UPN260	L100x100x10
12 CHINA	I45A	HW300x300	CH25B	L100x100x10
13 DUTCH	IPE450	HE360A	UPN260	L100x10
14 Aluminum	I12.0x12.5	WF8x13.0	CS10x6.23	L4x4x0.500
15 KOREA	I450x175x11/20	H350x350x12x19	C250x90x9/13	L100x10
16 VENEZUELA	TUB26090	PIP106	(N/A)	(N/A)
17 ColdFormed	(N/A)	(N/A)	10CU1.25X045	4LU4x060
18 MEXICO	IR457x112.9	IE305x60.7	CE254x22.76	LI102x10
19 COLD FORMED INDIAN	(N/A)	(N/A)	200CU80X5	100LU100X4
2 INDIA	ISWB400	ISHB300	ISLC225	ISA100X100X12

Country	Beam	Column	Channel	Angle
20 Lysaght Sections (AUS Cold Formed)	(N/A)	(N/A)	C20019	(N/A)
21 JOIST	28K10	(N/A)	(N/A)	(N/A)
22 AITCTimber	aspn_ST_4x12	aspn_ST4x4	(N/A)	(N/A)
23 BSCOLD	TU150x5	PP1143x6	150CASH50x50	(N/A)
24 BUTLER	PO8960	PO8718	(N/A)	(N/A)
25 CAN TIMBER	DFL_SelStr_6X14_BM	DFL_SelStr_10x10_BM	(N/A)	(N/A)
26 KINGSPAN COLDFORMED	(N/A)	(N/A)	C25010040	(N/A)
28 JAP COLDFORMED	BCR30030012	BCR30030012	(N/A)	(N/A)
29 Australian Cold Formed	200X100X9.0RHS	100X100X9.0SHS	(N/A)	(N/A)
3 EURO	IPE450	HE360AA	UPN220	L100X100X12
30 Russian Cold Formed	PIP10X1	PIP10X1	(N/A)	(N/A)
31 STOASChM	B45-1	K35-1	(N/A)	(N/A)
32 Jindal	IPE450	HEA320	ISMC400x100	(N/A)
33 EuropeanColdFormed	RHS300150X150X12	SHS150150X150X10	(N/A)	(N/A)
34 TataStructura	300X150X10.0RHS	150X150X5.0SHS	(N/A)	(N/A)
35 Brazilian	W460x52.0	WH360x101.0	C250x22.8	EA100x100x9
4 RUSSIA	I45	K1-35	C27	L100x100x10
5 JAPAN	I300X150X10	H300X300X10x15	C200X90X8	L100x100x7
6 CANADA	W530X66	HP310x94	C230X30	L102x102x9.5
7 FRANCE	IPE450	HEA360	UAP80	L100X100X7
8 GERMANY	IPE450	HEA360	U260	L100x100x10

Country	Beam	Column	Channel	Angle
9 AUSTRALIA	UB200X22.3	UC200X46.2	PFC200	A50X50X3

Related Links

- *G.6.5 Assign Command* (on page 2117)

TR.20.6 Examples of Member Property Specification

This section illustrates the various options available for `MEMBER PROPERTY` specification.

```
UNIT  …
MEMBER PROPERTIES
1  TO  5  TABLE  ST  W8X31
9  10  TABLE  LD  L40304  SP  0.25
12  TO  15  PRISMATIC  AX  10.0  IZ  1520.0
17  18  TA  ST  PIPE  OD  2.5  ID  1.75
20  TO  25  TA  ST  TUBE  DT  12.  WT  8.  TH  0.5
27  29  32  TO  40  42  PR  AX  5.  IZ  400.  IY  33. -
IX  0.2  YD  9.  ZD  3.
43  TO  47  UPT  1  W10X49
50  51  UPT  2  L40404
52  TO  55  ASSIGN  COLUMN
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0
```

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; 43 to 47 are wide flanges selected from the user input table number 1; 50 and 51 are single angles from the user input table number 2; 52 through 55 are designated as COLUMN members using the ASSIGN specification. The program will assign a suitable I-section from the steel table for each member.

Member 56 is a wide flange W12X26 with a 4.0 unit wide cover plate of 0.3 units of thickness at the top. Member 57 is a composite section with a concrete slab of 5.0 units of thickness at the top of a wide flange W14X34. The compressive strength of the concrete in the slab is 3.0 force/length^2.

TR.20.7 Composite Decks

A composite deck generation facility is available in the program.

General Format

The command syntax for defining the deck within the STAAD input file is as shown below.

```
START DECK DEFINITION
_DECK deck-name
PERIPHERY member-list
DIRECTION d1 d2 d3
COMPOSITE member-list
OUTER member-list
VENDOR name
FC f1
CT f2
CD f3
RBH f4
RBW f5
PLT f6
PLW f7
DIA f8
HGT f9
DR1 f10
SHR f11
CMP f12
CW f13 MEMB cw-member-list
END DECK DEFINITION
```

Where:

_DECK *deck-name*	an alphanumeric name you specify to identify the deck. The *deck-name* line must start with '_DEC' or '_DECK'. The *deck-name* is the second word and is limited to 23 characters. This name must not be the same as any group name.
member-list	the list of members belonging to the deck. TO, BY, ALL, and BEAM are permitted. ALL means all members in structure; BEAM means all beams.
DIRECTION *d1 d2 d3*	the x, y, and z components of the direction of the deck, respectively

The following parameters may be in any order or omitted. They only apply to the composite members listed above. Do not enter a member list for these parameters.

FC *f1*	compressive strength of the concrete for all composite members listed above for this composite deck.

CT ***f2***	concrete thickness
CD ***f3***	concrete density
RBH ***f4***	height of rib of form steel deck. This is the distance from the top of the I beam to the bottom of the concrete deck.
RBW ***f5***	width of rib of form steel deck
PLT ***f6***	thickness of cover plate welded to bottom flange of composite beam
PLW ***f7***	width of cover plate welded to bottom flange of composite beam
DIA ***f8***	diameter of shear connectors
HGT ***f9***	height of shear connectors after welding
DR1 ***f10***	ratio of moment due to dead load applied before concrete hardens to the total moment
SHR [0 | 1]	temporary shoring during construction. 0 = no shoring 1 = with shoring
CMP [0 | 1 | 2]	composite action with connectors. 0 = no composite action in design 1 = composite action 2 = ignore positive moments during design
CW ***f11*** **MEMB** ***cw-member-list***	*f11* is the concrete width for each composite member listed. *cw-member-list* is the list of composite members in this deck that have this width. Enter as many CW lines as necessary to define the width of all composite members of this deck.

This Deck definition data should be entered after the member properties have been entered.

Notes

1. The DECK definition must start with the START DECK DEFINITION command and end with the END command.
2. More than one DECK may be specified between the START and END.
3. The same member number may be included in up to 4 deck/groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
4. If two or more consecutively entered decks have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
5. The _deck-name must be unique within the Deck definitions and the Group definitions.
6. PER, DIR, OUT are data created by the GUI. Do not edit this data.
7. This Deck definition data should be entered after the member properties have been entered.

Example

```
START DECK DEFINITION
_DECK DEC-1
PERIPHERY 4 1640 18 38 56 50 49
DIRECTION 0.0000000.000000 -1.000000
COMPOSITE 41 74 38
OUTER 7 8 3130
VENDOR USSTEEL
DIA 0.700
HGT 2.75
CT 11.0
FC 3.1
RBW 2.6
RBH 0.1
CMP 1.0
SHR 1
CD 0.0000870
CW 123.000000 MEMB41
CW 123.000000 MEMB7
CW 61.500000 MEMB4
CW 61.500000 MEMB38
END DECK DEFINITION
```

Related Links

- *M. Composite Decks* (on page 676)
- *G.6.7 Composite Beams and Composite Decks* (on page 2119)
- *M. To add a floor load or one-way load* (on page 840)
- *G.6.7 Composite Beams and Composite Decks* (on page 2119)

TR.20.8 Curved Member Specification

The following commands are used to specify that a member is curved. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees. Any non-tapered cross-section is permitted.

General Format

```
MEMBER CURVED
```

```
member-list RADIUS r GAMMA g PRESS p
```

Where:

RADIUS *r* = radius in length units

GAMMA *g* = The angle in degrees used to define the plane of the circle. The angle is defined using the same set of rules used to define the orientation (beta angle) of a straight member connected between the two nodes. See Gamma Angle (on page 2275).

PRESS *p* = Pressure/Flexibility parameter for pipe bends. See Pressure/Flexibility Parameter (on page 2274).

Notes

1. The radius should be in current units.
2. Certain attributes like releases, `TENSION/COMPRESSION` flags, and several member load types are currently not available. Section forces too are currently not available.
3. The design of curved members is not supported.

Pressure/Flexibility Parameter

This applies only to pipe bend (elbow) members (OD and ID entered). These members will flex more due to ovalization depending on internal pressure. The ASME Boiler and Pressure Vessel Code, Section III, NB-3687.2, 1971, for Class I components is used to calculate the flexibility reduction factor.

- Set p = 0 or omit for this flexibility increase calculation to occur with internal pressure equal to zero.
- Set p > 0 to specify internal pressure to use in this flexibility calculation. Pressure reduces the flexibility increase.
- Set p = -9999 to ignore this additional flexibility calculation and use only beam theory.
- Set p = flexibility reduction factor (-FLEXF below); which must be a negative number less than –1.0 .

ASME Pipe Elbow flexibility factors theory [ASME Section NB-3687.3]

This section only applies if *(Bend Radius/Mean Radius)* $\geq$ *1.70* or if *(Arclength)* > *(2x Mean Radius)*

Flexibility Factor, FLEXF = (1.65x (Mean Radius)^2)/[t·(Bend Radius)] x 1/[1 + (Press)(Mean Radius)(FACT.)]

where

FACT.	=	$6\cdot(MR/t)^{4/3}\cdot(BR/MR)^{1/2}/(Et)$
MR	=	Mean Radius of elbow wall
BR	=	Bend Radius
Press	=	Internal Pressure
t	=	elbow wall thickness
E	=	Modulus of Elasticity

If the Flexibility Factor computed is less than 1.0, then STAAD.Pro will use 1.0 . The Flexibility Factor directly multiplies or contributes to most non-shear terms in the elbow flexibility matrix.

Notes

1. The input for defining the curved member involves 2 steps. The first is the member incidence, which is the same as that for a straight line member. The second is the command described above, which indicates that the segment between the 2 nodes of the member is curved, and not a straight line.
2. Any non-tapered cross section property currently available in STAAD can be assigned to these members.
3. Currently, two load types are permitted on curved members. One is the `SELFWEIGHT` load type, described in TR.32.9 Selfweight (on page 2496). The other is the uniformly (`UNI`) distributed load type of the `MEMBER LOAD` options explained in TR.32.2 Member Load Specification (on page 2464). The uniformly distributed load has to be applied over the full span of the member. Other member loads such as `LINEAR`, `TRAP`, `CONCENTRATED` force or moment, `UNIFORM` moment, etc. are not supported.
4. Some of the other member load types such as `PRESTRESS`, `TEMPERATURE`, `STRAIN` loads, etc. are also not currently supported. These options too are expected to become available in future versions of the program.
5. The results of the analysis currently consist of the nodal displacements of the ends of the curved member, and the member end forces. The nodal displacements are in the global coordinate system. The member end forces are in the local coordinate system, with each end of the member having its own unique local axis

system. Results at intermediate sections, such as sectional displacements, and sectional forces will be available in future versions of the program.

Gamma Angle

The plane of the circle defines the plane formed by the straight line joining the two ends of the arc, and the local Y axis of an imaginary straight member between those two points. The positive value of the GAMMA angle is obtained using the same sense as the positive value of the beta angle of that imaginary straight line member whose local Y axis points towards the vertex of the arc.

Several diagrams intended to show the GAMMA angle for various segments lying in the three global planes are shown.

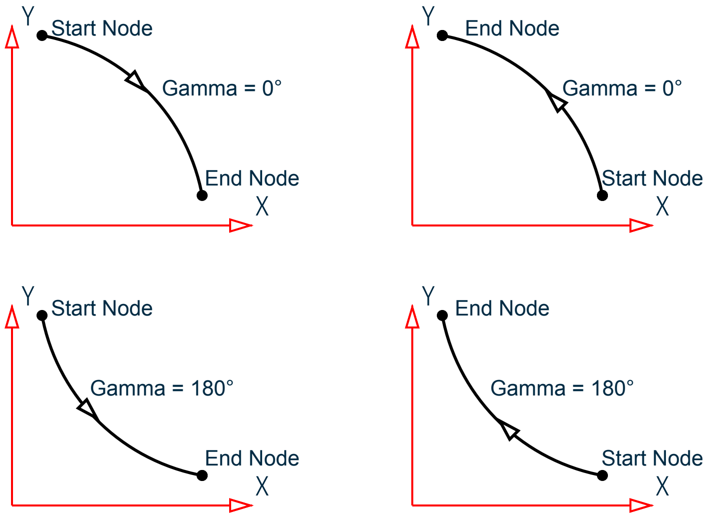


Figure 287: Gamma angle for various configurations of the circular arc lying in the global XY plane

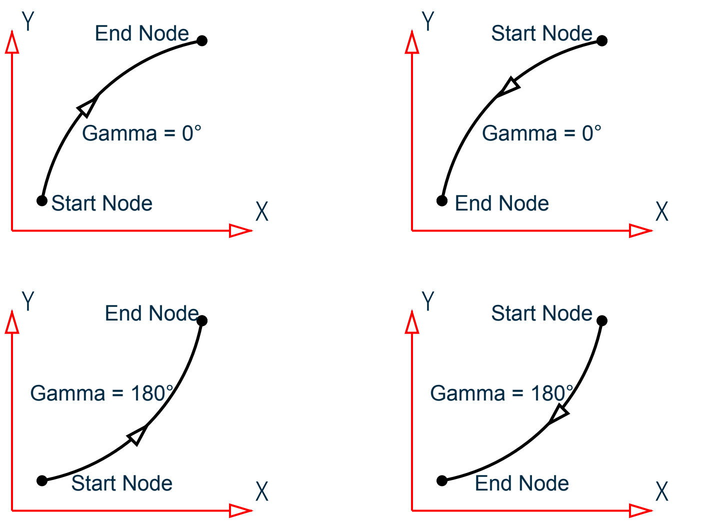


Figure 288: Gamma angle for various configurations of the circular arc lying in the global XY plane

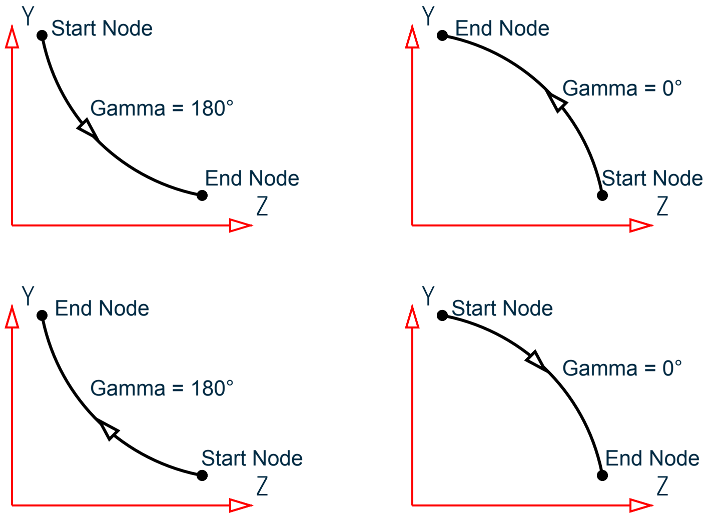


Figure 289: Gamma angle for various configurations of the circular arc lying in the global YZ plane

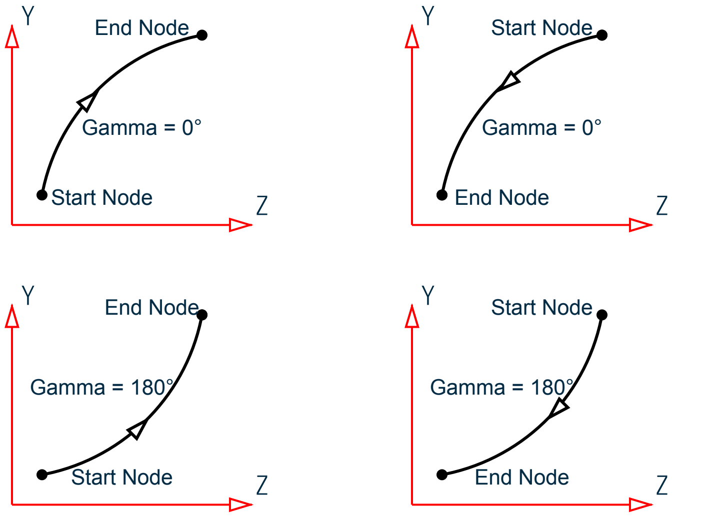


Figure 290: Gamma angle for various configurations of the circular arc lying in the global YZ plane

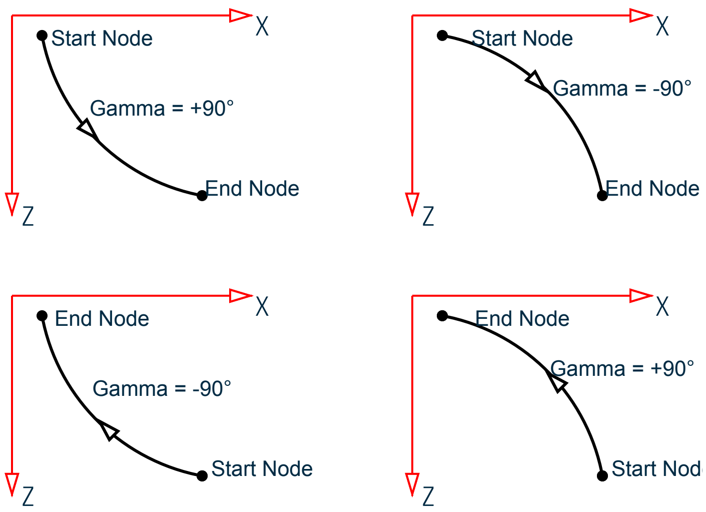


Figure 291: Gamma angle for various configurations of the circular arc lying in the global XZ plane

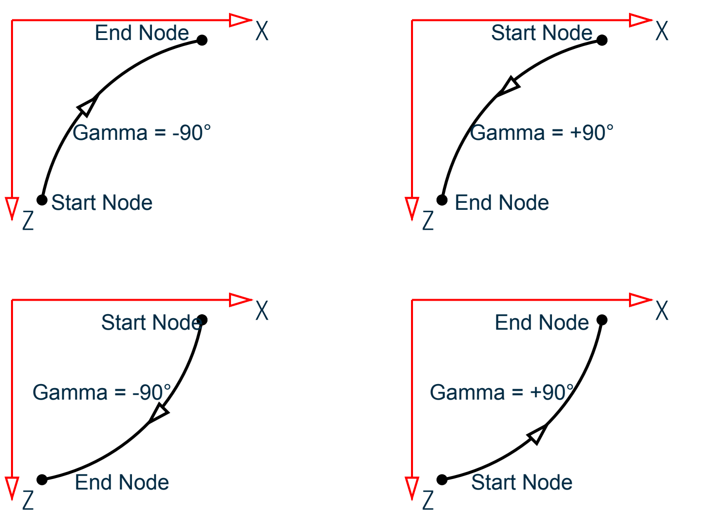


Figure 292: Gamma angle for various configurations of the circular arc lying in the global XZ plane

Member local axis system

The local axis directions for curved members are dependent on the point of interest along the curve. The general rules for local axis, as laid out in G.4.2 Local Coordinate System (on page 2088) are applicable. The figure shown later for member end forces indicates the directions of axes at the start and end nodes.

Rotation of local axis

There is a limited facility available to change the orientation of a curved member cross section. The cross-section may be at the default position where the strong axis (local y) is normal to the plane of the curve and the weak axis is in that plane.

The BETA ANGLE and REFERENCE POINT options, explained in G.4.3 Relationship Between Global and Local Coordinates (on page 2092) and TR.26.2 Specifying Constants for Members and Elements (on page 2305), are *not* available for curved members.

Sign conventions

The displacements of the nodes of the curved member are along the global axis system just as in the case of straight members.

The member end forces for curved members are quite similar to that for straight members. The only distinguishing item is that they are normal and tangential to the local axis at the corresponding ends. For example, FX at the start is tangential to the curve at the start node, and FX at the end is tangential to the curve at the end node. Similarly, FZ is along the radial direction at the two ends.

Member releases, offsets, tension/compression, truss and cable may not be specified for curved beams.

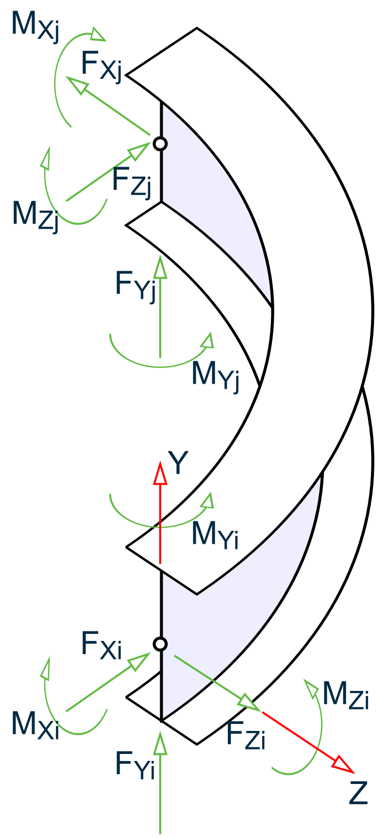


Figure 293: Sign conventions for member end actions; Global Y is vertical

Example

```
STAAD SPACE
UNIT KIP FEET
```

```
JOINT COORD CYL REVERSE
1 150 0 0 13 150 0 90
REPEAT 1 30 0 0
REPEAT ALL 1 0 15 0
MEMBER INCIDENCES
1 1 27 26
101 27 28 112
113 40 41 124
201 27 40 213
START GROUP DEFINITION
MEMBER
_COLUMN 1 TO 26
_CIRCUMFERENTIAL 101 TO 124
_RADIAL 201 TO 213
END GROUP DEFINITION
MEMBER PROPERTIES
_COLUMN PRIS YD 3.0
_CIRCUMFERENTIAL PRIS YD 3.0
_RADIAL PRIS YD 3.0
CONSTANT
E CONCRETE ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL
MEMBER CURVED
101 TO 112 RADIUS 150 GAMMA 90.0
113 TO 124 RADIUS 180 GAMMA 90.0
SUPPORTS
1 TO 26 PINNED
LOAD 1
SELF Y -1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT MEMBER FORCE LIST 101 113
FINISH
```

Related Links

- *G.6.8 Curved Members* (on page 2120)
- *M. To add a curved beam* (on page 660)
- *G.6.8 Curved Members* (on page 2120)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.20.9 Applying Fireproofing on members

STAAD.Pro includes a method to automatically consider the weight of fireproofing material applied to structural steel.

General Format

```
MEMBER FIREPROOFING
```

```
member-list FIRE { BFP | CFP } THICKNESS f1 DENSITY f2
```

Where:

THICKNESS *f1*	thickness (T in the figures below) in length units

DENSITY *f2*	density of fireproofing material in (force / length^3) units

In the actual load case itself, nothing besides the `SELFWEIGHT` command is necessary to instruct the program to include the weight of the fireproofing material in the selfweight calculation.

Two types of fireproofing configurations are currently supported. They are:

block fireproofing
contour fireproofing

Block Fireproofing (BFP)

The following figure shows this configuration. The fire-protection material forms a rectangular block around the steel section.

The area of fireproofing material (A_{fp}) at any section along the member length is calculated in the following manner.

For Wide Flanges (I-shaped sections), Channels and Tees,

$$A_{fp} = [(B_f + 2T)*(D+2T)] - A_{steel}$$

For single angles,

$$A_{fp}= [(B_f + 2T)*(D+2T)] - A_{steel}$$

where

B_f	=	the flange width
D	=	the overall depth of the steel section
T	=	the thickness of the fireproofing material beyond the outer edges of the cross section as dimensioned in the next figure
A_{steel}	=	area of the steel section

Where:

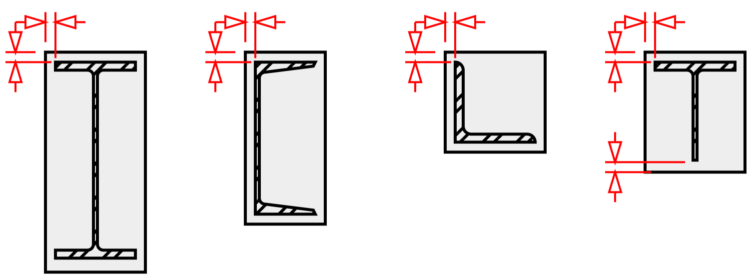


Figure 294: Block fireproofing (BFP) on various shapes

Contour Fireproofing (CFP)

In this configuration, the fire-protection material forms a coating around the steel section as shown in the next figure. The area of fireproofing material (A_{fp}) for this case is calculated in the following manner.

For Wide Flanges (I-shaped sections)

$$A_{fp} = [(B_f + 2T)*(T_f+2T)]*2 + [(D-2T-2T_f)*(T_w+2T)] - A_{steel}$$

For single angles,

$$A_{fp} = [(L_1+2T)*(2T+T_a) + (L_2-T_a)*(2T+T_a)] - A_{steel}$$

For Tees,

$$A_{fp} = [(B_f + 2T)*(T_f+2T)] + [(D-T_f)*(T_w+2T)] - A_{steel}$$

where

B_f	=	the flange width
D	=	the overall depth of the steel section
T	=	the thickness of the fireproofing material beyond the outer edges of the cross section as dimensioned in the next figure
T_f	=	the thickness of the flange for the I shape and Tee
T_a	=	the thickness of the leg of the angle
T_w	=	the thickness of the web for the I shape and Tee
A_{steel}	=	area of the steel section

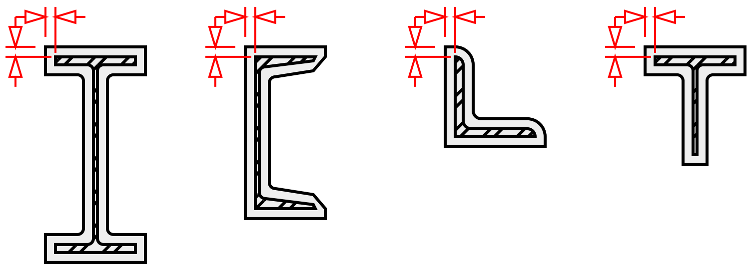


Figure 295: Contour fireproofing (CFP) on various shapes

The number of input items required to apply this attribute is:

a. the type of fireproofing
b. the thickness T shown in the above figures
c. the density of the fireproofing material
d. the members on which it is to be applied

For each such member, A_{fp} is calculated and multiplied by the density of the fireproofing material to obtain the weight per unit length of the member. This is added to the weight per unit length of the steel section itself and the total is used in calculating selfweight. Hence, `SELFWEIGHT` must be one of the load components of load cases if the weight of the fireproofing material should be considered as part of those load cases.

Notes

1. STAAD calculates the fire proofing weight only for the following sections:

   For block fireproofing - I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, single channels, angles and Tees.

   For CFP-contour fireproofing - the sections are I-beam straight or tapered, angle, Tee.

   I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, angles and Tees..
2. Fire proofing weight is not calculated for the following section types: Pipe, tube, composite I beams with slab on top, double channel, double angle, HSS, I-beam with cover plates, prismatic, solid circle or rectangle, castellated, cold formed sections, wood, aluminum, tapered poles, etc.

Example Problem

```
STAAD SPACE
UNIT KIP FEET
JOINT COORDINATES
1 0. 0. ;2 0. 15. ; 3 20. 15. ;4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X26
2 TABLE ST W14X34
CONSTANTS
E  STEEL ALL
POISSON STEEL ALL
DENSITY STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT POUND INCH
MEMBER FIREPROOFING
1 3 FIRE BFP THICK 2.0 DENSITY 40
2 FIRE CFP THICK 1.5 DENSITY 40
UNIT KIP FT
LOADING 1 DEADWEIGHT OF STEEL + FIREPROOFING
SELF Y -1.0
LOAD 2 LIVE
MEMBER LOAD
2 UNI GY -0.8
LOAD COMBINATION 3
1 0.75 2 0.75
PERFORM ANALYSIS
PRINT MEMBER FORCES
PRINT SUPPORT REACTIONS
FINISH
```

Related Links

- *M. To assign member fire proofing* (on page 808)

TR.20.10 Member Property Reduction Factors

Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. The methodology to handle cracked section properties is nonlinear in nature (i.e., the section capacities should be checked and modified depending upon the section forces the section is handling). The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

In STAAD.Pro, you can specify a set of reduction factors to be applied on the calculated section properties such as area, moments of inertia, and torsional constant. If you want to adopt this approach to account for cracking of concrete sections, refer to Section 10.11.1 of ACI 318 for a set of values to use for these reduction factors depending upon the nature of forces and moments the member is subjected to.

Note: The specifications in the AISC 13th edition manual suggest reducing the stiffness of the steel member during the analysis. In place of the `MEMBER CRACKED` command, the `REDUCEDEI` parameter may be used when the PERFORM DIRECT ANALYSIS command is used. See TR.37.5 Direct Analysis (on page 2630) for additional information.

General Format

The format of the command is:

```
MEMBER CRACKED (CODE IS1893 2016)
```

```
{ member-list | group-list } REDUCTION *{ RAX f1 | RIX f2 | RIY f3 | RIZ f4 }
```

Each reduction factor value, *f1* through *f4*, should be a fraction of unity.

RAX *f1* Reduction factor in the axial area.

RAX is *not* applicable for reduction factors for the IS1893 2016 code.

RIX *f2* Reduction factor of the torsion constant (about the local x-axis).

RIY *f3* Reduction factor in the moment of inertia about the local major axis (y-axis).

RIZ *f4* Reduction factor in the moment of inertia about the local minor axis (z-axis).

This is a multiplication factor on the property value. It does not signify the amount by which the property is reduced, but, it is simply a value by which the unreduced property is multiplied. Thus, the calculated (or the user-specified value) of the property will be multiplied by the reduction factor to arrive at the value used in the analysis.

For example, a factor of 0.45 defined for RAX will mean that if the cross sectional area of the gross section is 0.8 ft^2, the value used in the analysis will be $0.8 \times 0.45 = 0.36\ ft^2$.

Multiple factors can be assigned on the same line.

The reduction factor is considered only for analysis but not for design.

Code-Specific Reduction Factors

IS1893 2016 Clause 6.4.3 calls for a reduction of moment of inertia values by 0.35 for beams and 0.7 for columns of concrete members only while analyzing the structure for static seismic and response spectrum/linear dynamic analysis.

Note: Automated stiffness reduction requires a STAAD.Pro Advanced license.

Code-specific reduction factors will not affect non-concrete members. A separate stiffness matrix is generated for the use of IS1893 2016 static seismic and response spectra loads. For all other load cases, the analysis is performed using the unreduced stiffness matrix.

Tip: For a STAAD.Pro model containing multiple analysis commands, a different set of code-specific reduction factors may be used for each analysis command.

The CHANGE command has no affect or code-specific reduction factors.

Example

```
MEMBER CRACKED
1 REDUCTION RAX 0.35 RIX 0.40 RIY 0.45 RIZ 0.45
```

IS1893 2016 Example

```
START GROUP DEFINITION
MEMBER
_BEAM 1 4
_COLUMN 2 3 5 6
END GROUP DEFINITION
…
MEMBER CRACKED CODE IS1893 2016
_COLUMN REDUCTION RIY 0.700 RIZ 0.700
_BEAM REDUCTION RIY 0.350 RIZ 0.350
```

Related Links

- *M. To assign cracked section properties to a member* (on page 807)
- *M. To assign cracked section properties to a member* (on page 807)

TR.21 Element/Surface Property Specification

Individual plate elements, and the Surface element need to have their thickness specified before the analysis can be performed. The commands for specifying this information are explained in this section. No similar properties are required for solid elements. However, constants such as modulus of elasticity, Poisson's Ratio, etc. are required.

Related Links

- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.21.1 Element Property Specification

This set of commands may be used to specify properties of plate finite elements. Elements of uniform or linearly varying thickness may be modeled using this command. The value of the thickness must be provided in current units.

Unlike members and plate/shell elements, no properties are required for solid elements. However, constants such as modulus of elasticity and Poisson's ratio are to be specified.

General Format

```
ELEMENT PROPERTY
```

```
element-list   THICKNESS f1  (f2 f3 f4)
```

Where:

THICKNESS *f1*	thickness of the element
f2 … f4	thicknesses at other nodes of the element if different from *f1*

Example

```
UNIT …
ELEMENT PROPERTY
1 TO 8 14 16 THI 0.25
```

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To specify plate thickness* (on page 811)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.21.2 Surface Property Specification

This set of commands may be used to specify properties of surface entities.

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

General Format

```
SURFACE PROPERTY
```

```
surface-list THICKNESS t
```

Where:

t = Thickness of the surface element, in current units.

Example

```
SURFACE PROPERTY
1 TO 3 THI 18
```

TR.22 Member and Element Releases

STAAD.Pro allows specification of releases of degrees of freedom for frame members and plate elements.

Related Links

- *G.7 Member and Element Release* (on page 2120)

TR.22.1 Member Release Specification

This set of commands may be used to fully release specific degrees of freedom at the ends of frame members. They may also be used to describe a mode of attachment where the member end is connected to the joint for specific degrees of freedom through the means of springs.

General Format

```
MEMBER RELEASES
```

```
member-list  {START | END | BOTH } { *{ FX | FY | FZ | MX | MY | MZ } | *{KFX f1 | KFY
f2 | KFZ f3 | KMX f4 | KMY f5 | KMZ f6 } | {MP f7 | *{     MPX f8 | MPY f9 | MPZ
f10 }} }
```

Where:

FX ... MZ	represent force-x through moment-z degrees of freedom in the member local axes
KFX *f1* ... KMZ *f6*	spring constants for these degrees of freedom, in current units
MP *f7*	partial moment release factor for all three moments
MPX *f8*, MPY *f9*, MPZ *f10*	partial moment release factors for each moment separately. The moment related stiffness coefficient will be multiplied by a factor of (1 - *fn*) at the specified end. Release factors must be in the range of 0.001 through 0.999.

Note: If FX through MZ is used, it signifies a full release for that degree of freedom and if KFX through KMZ is used, it signifies a spring attachment.

Notes

a. Member releases are a means for describing a type of end condition for members when the default condition, namely, fully moment and force resistant, is not applicable. Examples are bolted or riveted connections. Partial moment releases are a way of specifying bending and torsional moment capacity of connections as being a fraction of the full bending and torsional strength.

b. It is important to note that the factor f_1 indicates a reduction in the stiffness corresponding to the rotational degrees of freedom MX, MY, and MZ. In other words, you should not expect the moment on the member to reduce by a factor of f_1 . It may be necessary to perform a few trials in order to arrive at the right value of f_1 that results in the desired reduction in moment.

c. The START and END are based on the MEMBER INCIDENCE specification. The BOTH specification will apply the releases at both ends.

d. At any end of the member—for any particular DOF—full, partial, and spring release cannot be applied simultaneously. Only one out of the three is permitted.

e. If MY (or MZ) is fully released at both ends, then VZ (or VY) cannot be transmitted through the member. The final shears in the member will be entirely due to loads applied directly to the member.

Example

```
MEMBER RELEASE
1 3 TO 9 11 12 START KFX 1000.0 MY MZ
1 10 11 13 TO 18 END MZ KMX 200.0
```

In this example, for members 1, 3 to 9, 11 and 12, the moments about the local Y and Z axes are released at their start joints (as specified in `MEMBER INCIDENCES`). Further, these members are attached to their START joint along their local x axis through a spring whose stiffness is 1000.0 units of force/length. For members 1, 10, 11 and 13 to 18, the moment about the local Z axis is released at their end joint. Also, the members are attached to their END joint about their local x axis through a moment-spring whose stiffness is 200.0 units of force-length/Degree. Members 1 and 11 are released at both start and end joints, though not necessarily in the same degrees of freedom.

Partial Moment Release

Moments at the end of a member may be released partially using the MP option (to provide the same partial release in all . This facility may be used to model partial fixity of connections. The following format may be used to provide a partial moment release. This facility is provided under the `MEMBER RELEASE` option and is in addition to the existing `RELEASE` capabilities.

Example

```
MEMBER RELEASE
15 TO 25 START MP 0.25
```

The above `RELEASE` command will apply a factor of 0.75 on the moment related stiffness coefficients at start node of members 15 to 25.

Related Links

- *G.7 Member and Element Release* (on page 2120)
- *M. To assign member end release* (on page 804)
- *M. To assign specifications to physical members* (on page 810)

TR.22.2 Element Release Specification

This set of commands may be used to release specified degrees of freedoms at the corners of plate finite elements.

General Format

```
ELEMENT RELEASE
```

```
element-list  { J1 | J2 | J3 | J4 } *{ FX | FY | FZ | MX | MY | MZ }
```

Where:

J1, J2, J3 and J4 = signify joints in the order of the specification of the element incidence. For example, if the incidences of the element were defined as 35 42 76 63, J1 represents 35, J2 represents 42, J3 represents 76, and J4 represents 63.
FX through MZ = represents forces/moments to be released per local axis system.

Note: Element releases at multiple joints cannot be specified in a single line. Those must be specified separately as shown below.

Examples

Example of Correct Usage

```
ELEMENT RELEASE
10 TO 50 J1 MX MY
10 TO 50 J2 MX MY
10 TO 50 J3 MY
10 TO 50 J4 MY
```

Example of Incorrect Usage

```
ELEMENT RELEASE
10 TO 50 J1 J2 MX MY
10 TO 50 J3 J4 MY
```

Notes

a. All releases are in the local axis system. See G.5.1 Plate and Shell Elements (on page 2099) for the various degrees of freedom. Fx and Fy have the same sense as Sx and Sy in G.5.1 Plate and Shell Elements (on page 2099). Fz has the same sense as SQx or SQy. Generally, do not over release. The element must still behave as a plate after the releases.

b. Selfweight is applied at each of the nodes as if there were no releases.

c. Thermal stresses will include the fixed-end thermal pre-stress as if there were no release.

d. May not be used with the Element Plane Stress or Element Ignore Inplane Rotation commands on the same element.

e. Note that the usual definitions of local Mx and My are reversed here. See G.5.1 Plate and Shell Elements (on page 2099) for the definitions of Mx and My. Releasing Fz, Mx, My will release all bending capability. Releasing Fx, Fy, Mz will release all in-plane stiffness.

Related Links

- *G.7 Member and Element Release* (on page 2120)
- *M. To assign plate corner release* (on page 814)
- *G.5.1 Plate and Shell Elements* (on page 2099)

TR.22.3 Element Ignore Stiffness

Structural units like glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these units are designed to carry those loads and transmit those loads to the rest of the structure, they are not designed to provide any additional stiffness to the structure. One way to handle the situation is to not input the unit as part of the structural model, and apply the load using load generation techniques like `AREA LOAD` or `FLOOR LOAD`.

STAAD provides another way of handling such units. This is through the help of the `ELEMENT IGNORE STIFFNESS` command. To use this feature, the glass panel or roof unit must be defined using plate elements. The `IGNORE STIFFNESS` command enables one to consider the unit just for the purpose of application of `ELEMENT LOAD` commands, while its stiffness will not be considered during the assembly of the stiffness matrix. In other words, it is like an `INACTIVE` member which is active for `ELEMENT LOAD` command application but `INACTIVE` for stiffness. Like the `INACTIVE` command, the plates listed here will become active at the next `CHANGE` command. To keep them inactive, re-enter this data after each `CHANGE`.

The `SELFWEIGHT`, `ELEMENT WEIGHT`, `TEMPERATURE`, and mass of the plates listed here will still be ignored.

General Format

```
IGNORE STIFFNESS {ELEMENT} element-list
```

Example

```
IGNORE STIFFNESS ELEMENT 78 TO 80
```

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To ignore plate stiffness* (on page 816)

TR.23 Axial Member Specifications

A member can have only *one* of the following specifications:

Truss
Tension-only
Compression-only
Cable

If multiple specifications are applied to the same member, only the last entered will be used (Warnings will be printed).

Note: `MEMBER TRUSS`, `MEMBER TENSION`, `MEMBER COMPRESSION`, and `MEMBER CABLE` are axial-only for stiffness. `MEMBER CABLE` are special truss members that may also be specified as tension-only.

TR.23.1 Member Truss Specification

This command may be used to model a specified set of members as TRUSS members.

This specification may be used to specify truss type members in a PLANE, SPACE, or FLOOR structure. The truss members are capable of carrying only axial forces. For example, bracing members in 2D (i.e., PLANE or FLOOR) or 3D (i.e., SPACE) frames will often be truss type members.

Tip: This command is unnecessary when a TR.2 Problem Initiation and Model Title (on page 2202) has already been specified using the command `STAAD TRUSS`.

General Format

```
MEMBER TRUSS
```

```
member-list (TENSION f1)
```

Where:

TENSION *f1* optional initial tension in the truss member, in current units

Note: The `TENSION` parameter is ignored except in Nonlinear Cable Analysis. For that analysis type, a truss with pretension is considered to be nonlinear (large displacement). Also for that analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

Example

```
MEMB TRUSS
1  TO  8  10  12  14  15
```

Notes

a. The `TRUSS` member has only one degree of freedom-the axial deformation.
b. Member Releases are not allowed for TRUSS members.
c. Self-weight and transverse loads may induce shear/moment distributions in the member.
d. Member loads are lumped at each end, whereas a frame member with moment releases only retains the axial component of the applied member load.

Related Links

- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)
- *M. To assign axial action members* (on page 803)
- *M. To assign specifications to physical members* (on page 810)
- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)
- *G.13.1 Tension- and Compression- Only Springs* (on page 2127)
- *G.8.2.1 Linearized Cable Members* (on page 2122)
- *G.8.2.2 Nonlinear Cable and Truss Members* (on page 2123)
- *G.17.2.7 Nonlinear Cable or Truss Analysis* (on page 2149)

TR.23.2 Member Cable Specification

This command may be used to model a specified set of members as cable members.

Cable members, in addition to elastic axial deformation, are also capable of accommodating the stiffness effect of initial tension and tension due to static loads. When used in a nonlinear cable analysis, cable members are capable of accommodating large displacements. See G.8.2 Cable Members (on page 2122) for a theoretical discussion of cable members.

General Format

```
MEMBER  CABLE
```

```
member-list cable-spec
```

```
cable-spec  = { TENSION f1 ( { START | END } )| LENGTH f2 } *{ (FWX f3) | (FWY f4) |
 (FWZ f5) }
```

Where:

TENSION *f1*	Initial tension in cable member (in current units), when the `TENSION` option is specified.
LENGTH *f2*	Unstressed cable length (in current units), when the initial `LENGTH` option is specified.
FWX *f3*, FWY *f4*, FWZ *f5*	Multiplying factors on self weight components applied in the global X, Y, and Z directions, respectively.
START, END	The initial tension is measured at the cable start node or cable end node, respectively. This is used for advanced cable analysis (refer to Note 5 in Nodes for use with Advanced Cable Analysis).

Notes for use with Standard Cable Analysis

1. The tension specified in the cable member is applied on the structure as an external load as well as is used to modify the stiffness of the member. The tension value must be positive to be treated as a cable otherwise it is a truss (See G.8.2 Cable Members (on page 2122)). If the `TENSION` parameter or the value is omitted, a minimum tension will be used.

   The end at which initial tension is measured is not used for standard cable analysis (i.e., START or END is ignored).
2. This is a truss member but not a tension-only member unless you also include this member in a `MEMBER TENSION` input (See TR.23.3 Member Tension/Compression Specification (on page 2292)). Note also that Member Releases are *not* allowed.
3. The tension is a preload and will not be the final tension in the cable after the deformation due to this preload.
4. The tension is used to determine the unstressed length. That length will be shorter than the distance between the joints by the distance that the tension will stretch the cable.
5. No weight (other than the assumed self weight) is used with standard cable analysis.
6. Cables should not be used in analysis involving frequency extraction or for dynamic loading conditions such as response spectrum, time history, or steady state.

Notes for use with Advanced Cable Analysis

1. A cable member is a truss member that has three translational degrees of freedom only. Note also that member releases are *not* allowed for a cable member.
2. `FWY` is used to add any additional weight that may be acting on the cable along with self weight before the application of external applied load. This additional weight will be added to cable self-weight in global Y direction. These are used to find the initial cable profile before the application of the external load.
3. If no self weight is included in the external applied load and also `FWY` parameter is not defined, the program will calculate self weight of the cable member and include it in the analysis.

   Warning message will be issued in the output file.
4. If self weight is included in the external applied load, during calculation cable self weight will not be considered in the external applied load vector. The reason is that it will be considered separately while finding the initial cable configuration under self weight and thus cannot be considered as external applied load.
5. If `START` or `END` is not specified with the initial `TENSION` parameter in the cable member, then the average tension is assumed. The average option is suitable *only for a taut cable*. This is because in a taut cable, the undeformed length is shorter than the chord length. due to this, the taut cable carries significant tension with little sag. The undeformed legth, Lu, is calculated as:

$$L_u = \frac{L_c}{(1 + T_{avg} / EA)}$$

where

L_c	=	chord length
T_{avg}	=	initial average tension in the taut cable
E	=	Young's modulus
A	=	area of the cross-section

However, for a slack cable, this does not hold true because the undeformed length is longer that the chord length and the cable member has significant sag. The catenary curve effect is required to be included. Thus, for a slack cable, initial TENSION should be defined either at the START or END node of the cable member. For a cable member which is considered to have significant sag, defining the average tension is *not* a suitable option.

6. Cables should not be used in analysis involving frequency extraction or for dynamic loading conditions such as response spectrum, time history, or steady state.

Example

A series of cable members with a tension of 15.5 specified for use with either standard or advanced cable analysis:

```
MEMB CABLE
20 TO 25 TENSION 15.5
```

A series of cable members with a tension of 15.5 at the cable start node for use with advanced cable analysis. An additional weight on the cable of 20% of the cable self weight which acts in the same direction of the cable's self weight is used.

```
MEMB CABLE
20 TO 25 TENSION 15.5 START FWY -0.2
```

Related Links

- *M. To assign nonlinear cable members* (on page 806)
- *G.8.2.1 Linearized Cable Members* (on page 2122)
- *G.8.2.2 Nonlinear Cable and Truss Members* (on page 2123)
- *G.17.2.7 Nonlinear Cable or Truss Analysis* (on page 2149)
- *G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis* (on page 2124)
- *G.17.2.8 Advanced Nonlinear Cable Analysis* (on page 2150)

TR.23.3 Member Tension/Compression Specification

This command may be used to designate certain members as Tension-only or Compression-only members.

Tension-only members are truss/cable members that are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them.

General Format

```
MEMBER TENSION
```

```
member-list
```

or

```
MEMBER COMPRESSION
```

```
member-list
```

or

```
MEMBER TENSION 0
```

Linear Tension/ Compression Analysis

Compression-only members are truss members that are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them. Member Releases are not allowed on members with this attribute.

The procedure for analysis of Tension-only or Compression-only members requires iterations for every load case and therefore may be quite involved. you may also consider using the `INACTIVE` specification (instead of Tension/Compression) if the solution time becomes unacceptably high.

If a `CHANGE` command is used (because of a change in the list of tension members, cable tension, supports, etc.), then the `SET NL` command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

Note: For Nonlinear cable analysis, this command is unnecessary and ignored. Cables are automatically assumed to be partially to fully tension only (except that there should always be selfweight) without this command. In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

a. Loads that have been defined on members declared as `MEMBER TENSION` or `MEMBER COMPRESSION` will be active even when the member becomes `INACTIVE` during the process of analysis. This applies to `SELFWEIGHT`, `MEMBER LOADS`, `PRESTRESS` & `POSTSTRESS LOADS`, `TEMPERATURE LOAD`, etc.

b. A member declared as a `TENSION` only member or a `COMPRESSION` only member will carry axial forces only. It will not carry moments or shear forces. In other words, it is a truss member.

c. Do not use Load Combination to combine these cases. Tension/Compression cases are Nonlinear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.

Example

```
MEMBER TENSION
12 17 19 TO 37 65
MEMBER COMPRESSION
5 13 46 TO 53 87
```

Member Tension 0

This command switches off all tension/compression only specifications for load cases which are specified subsequent to this command, usually entered after a `CHANGE` command. There is no list associated with this command. Hence, for any further primary load cases, the tension/compression only attributed is disabled for *all* members.

Example

The following is the general sequence of commands in the input file if the `MEMBER TENSION` or `MEMBER COMPRESSION` command is used. This example is for the `MEMBER TENSION` command. Similar rules are applicable for the `MEMBER COMPRESSION` command. The dots indicate other input data items.

```
STAAD …
SET NL …
UNITS …
JOINT COORDINATES
…
MEMBER INCIDENCES
…
ELEMENT INCIDENCES
…
CONSTANTS
…
MEMBER PROPERTY
…
SUPPORTS
…
MEMBER TENSION
…
LOAD 1
…
LOAD 2
…
LOAD 3
…
LOAD 4
…
LOAD 5
REPEAT LOAD
…
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT …
PRINT …
PARAMETER
…
CHECK CODE …
FINISH
```

Notes

a. See TR.5 Set Command Specification (on page 2206) for explanation of the `SET NL` command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
b. STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a `SET ITERLIM i` command (i > 10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high.
c. The principle used in the analysis is the following.

- The program reads the list of members declared as MEMBER TENSION and/or COMPRESSION.
- The analysis is performed for the entire structure and the member forces are computed.
- For the members declared as MEMBER TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. If the member cannot take that load, the member is "switched off" from the structure.
- The analysis is performed again without the switched off members.
- Up to 10 iterations of the above steps are made for each load case, unless a higher value is set using the command ITERLIM.
- This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable.

d. A revised MEMBER TENSION / COMPRESSION command and its accompanying list of members may be provided after a CHANGE command. If entered, the new MEMBER TENSION/COMPRESSION commands replace all prior such commands. If these commands are not entered after a CHANGE, then the previous commands will still be applicable.

e. The MEMBER TENSION command should not be used if the following load cases are present : Response Spectrum load case, Time History Load case, or Moving Load case. If used, the MEMBER TENSION / COMPRESSION will be ignored in all load cases.

f. If UBC Load cases are included, then follow each UBC load case with an Analysis command, then a Change command.

g. If the model requires a PDelta analysis, then only the first method as described in TR.37.2 P-Delta Analysis Options (on page 2621) (i.e., small or large delta) may be used.

Related Links

- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)
- *M. To assign axial action members* (on page 803)
- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)

TR.24 Element Plane Stress and Ignore Inplane Rotation Specification

These commands allow the user to model the following conditions on plate elements

a. PLANE STRESS condition

b. In-plane rotation stiffness reformulated to be rigid or to be zero.

General Format

```
ELEMENT { PLANE STRESS | RIG ID ( INPLANE ROTATION ) | IGNORE ( INPLANE ROTATION) }
element-list
```

The `PLANE STRESS` specification allows the user to model selected elements for PLANE STRESS action only [No bending or transverse shear stiffness].

The `RIGID INPLANE ROTATION` command causes the program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly. The STAAD plate element formulation normally produces a very soft Mz stiffness that improves the inplane shear deformation. However when the plate Mz is connected to a beam bending moment as the only load path for that moment, then the RIGID INPLANE option may be used to have that element carry the Mz moment to the other joints rigidly to avoid the instability at the beam end. Usually only the elements connected to beams in this manner would have this specification.

The `IGNORE INPLANE ROTATION` command causes the program to ignore "in-plane rotation" actions. The STAAD plate element formulation normally includes this important action automatically. However, it may be noted that some element formulations ignore this action by default. The user may utilize this option to compare STAAD results with solutions from these programs.

These options are exclusive of each other and also exclusive of element releases. No single element may have more than one of these options.

Example

```
ELEMENT PLANE STRESS
1 TO 10 15  20 25 35
ELEMENT IGNORE
30 50 TO 55
```

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To assign plates as plane stress* (on page 815)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To assign inplane rotation behavior to plates* (on page 815)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.25 Offset Specifications

The program can generate rigid links between the analytical joints and the ends of members or vertices of plate elements.

TR.25.1 Member Offset Specification

This command may be used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

General Format

```
MEMBER OFFSETS
```

```
member-list  { START | END } ( LOCAL ) dx dy dz
```

Where:

dx*, *dy*, and *dz	correspond to the distance, measured in localized or global coordinate system, from the joint (START or END as specified) to the centroid of the starting or ending point of the members listed.

LOCAL optional parameter, if not entered then *dx*, *dy*, *dz* are assumed to be in global. LOCAL means that the distances *dx*, *dy*, *dz* are in the same member coordinate system that would result if the member were not offset and BETA = 0.0.

Description

The `MEMBER OFFSET` command can be used for any member whose starting or ending point is not concurrent with the given incident joint. This command enables the user to account for the secondary forces, which are induced due to the eccentricity of the member. Member offsets can be specified in any direction, including the direction that may coincide with the member x-axis.

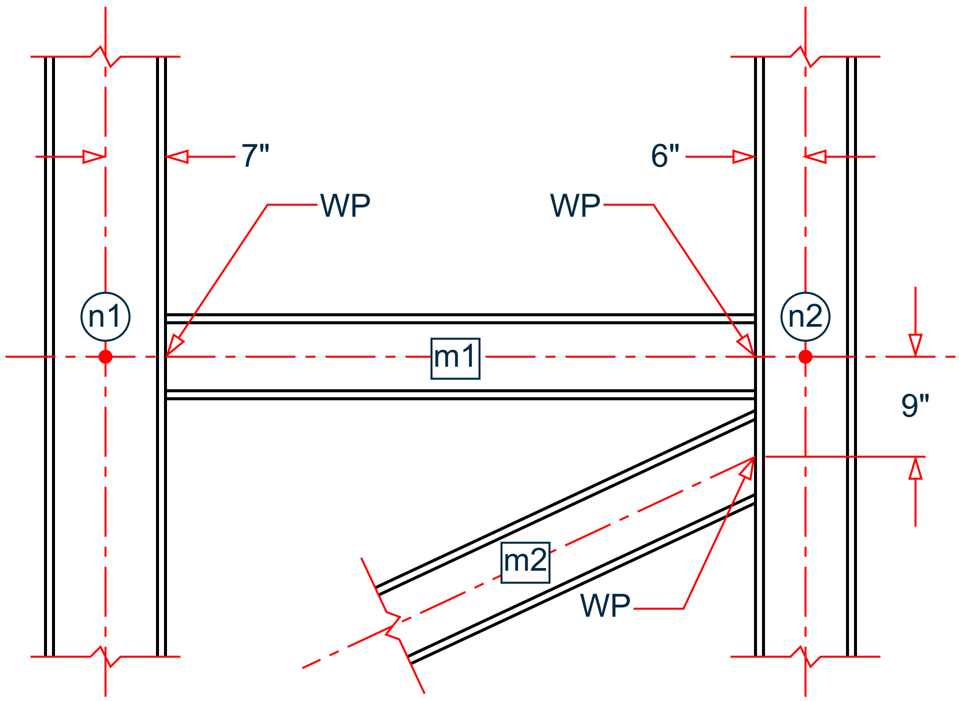


Figure 296: Example of working point (WP) represented by member end offsets

In the figure above, WP refers to the location of the centroid of the starting or ending point of the member.

Example

```
MEMBER OFFSET
1 START 7
1 END -6
2 END -6 -9
```

Notes

a. If a `MEMBER LOAD` (See TR.32.2 Member Load Specification (on page 2464)) is applied on a member for which `MEMBER OFFSETS` have been specified, the location of the load is measured not from the coordinates of the starting joint. Instead, it is measured from the offset location of the starting joint.

b. `START` and `END` is based on the user's specification of `MEMBER INCIDENCE` for the particular member.
c. Local offsets are defined in the local axes prior to rotation when a `BETA` angle is used.

Related Links

- *G.11 Member and Plate Offsets* (on page 2125)
- *M. To assign member end offsets* (on page 805)
- *M. To assign specifications to physical members* (on page 810)
- *G.11 Member and Plate Offsets* (on page 2125)

TR.25.2 Element Offset Specification

This command may be used to rigidly offset a plate element corner from a global joint to model the offset condition existing at the corners of plate elements.

General Format

The element offsets can be specified using one of two methods:

1. for individual joints in either local or global directions or
2. offsets along the local Z axis of the element.

```
element-list { JT1 | JT2 | JT3 | JT4 } ( LOCAL ) f1 f2 f3
```

or

```
element-list ZOFF { f4 | f5 f6 f7 f8 }
```

where

JT1 | JT2 | JT3 | JT4	The element joint number based on the joint incidence for the particular element. For example, JT1 is used for the first joint specified in that element's joint incidence list. Do not use JT4 for triangular plates.
f1, f2, f3	correspond to the distance, measured in local or global coordinate system, from a plate joint (JT1, JT2, JT3, or JT4 as specified) to the center plane of the associated corner point of the elements listed.

> **Note:** The joint offsets must result in offset corners which are co-planar to prevent a warped surface.

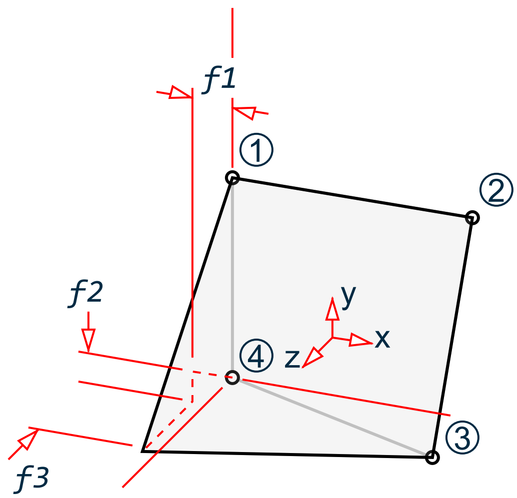


Figure 297: Local offsets given for JT4

LOCAL an option parameter that indicates the distances *ƒ1*, *ƒ2*, and *ƒ3* are in the member coordinate system with respect to the joint. If not entered, then *ƒ1*, *ƒ2*, and *ƒ3* are in the global coordinate system.

ƒ4 the Z offset distance to use for all four , measured in the local coordinate system from the joint perpendicular to the surface face in the local Z axis. This results in a constant offset of the entire element parallel to the joints.

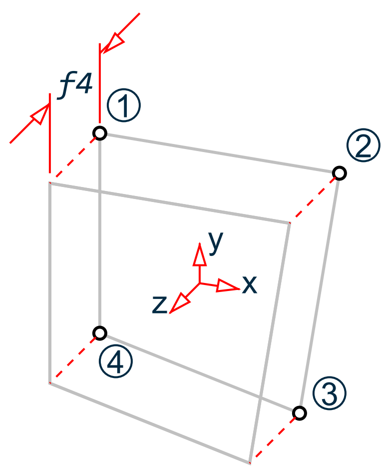


Figure 298: Constant Z offset

ƒ5*, *ƒ6*, *ƒ7*, *ƒ8 the Z offset distance to use for each of the four joints, in order. These are measured in the local coordinate system from the joint perpendicular to the surface in the local Z axis. Do not use *ƒ8* for triangular plates.

> **Note:** The joint offsets must result in offset corners which are co-planar to prevent a warped surface.

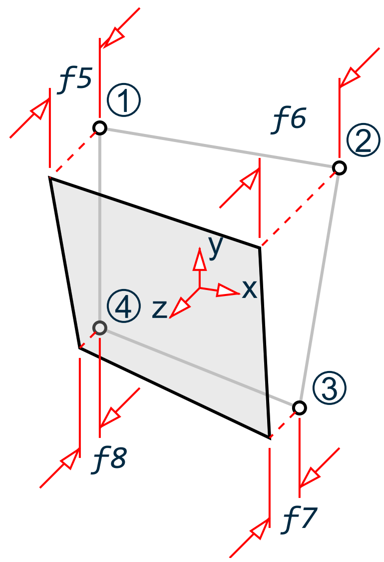


Figure 299: Z offset using different distances

Description

This command can be used for any element whose corner points are not concurrent with the given incident joint. This command enables you to account for the secondary forces which are induced due to the eccentricity of the element. Element offsets can be specified in any direction, including the direction that may coincide with the element in-plane axes.

Notes

a. If an element load (see TR.32.3 Element Load Specifications (on page 2468)) is applied on an element for which element offsets have been specified, then the location of the load is measured from the offset corner locations rather than the coordinates of the global joints.

b. Plate offsets are not applicable for steady state, geometric nonlinear, buckling, cable, or imperfection analysis types. An error message is generated if they are used with any of these analyses. Plate offsets are also not applicable for pushover analysis, as no elements (surface, plate, or solid) may be used with that analysis type.

c. The plate area, the warped element checks, and local element coordinates are based on the offset coordinates of the corner points.

d. Plate offsets are ignored static seismic loads.

e. Plate offsets are ignored for joint loads, since these are not applies to plates.

f. For floor diaphragms, use the `SET MASS 1` command to ensure that the plate offsets are considered during the center of mass calculations. Otherwise, the plate offsets will be ignored for this calculation. The mass modeling must be done using a `SET Y UP` command.

Examples

The following code example demonstrates from usage of this specification.

```
ELEMENT OFFSET
* All 4 joints of plate 1 are offset by 7 units in the local z direction.
1 ZOFF 7
```

```
* All 4 joins of plate 2 are offset by the units specified in the local z
* direction, in the order of the joint incidience.
2 ZOFF 7 7.5 7 7.5

* The first joint in the incidence of plate 3 is offset by 7 units in the
* local z direction.
3 JT1 LOCAL 0 0 7

* The second joint int he incidence of plate 4 is offset by 7.5 units in
* the global X direction.
4 JT2 7.5 0 0
```

Examples of Constant Offsets

The following offset methods all give the same results for element number 1:

Note: In practice, this model would only use *one* of these methods for element offsets.

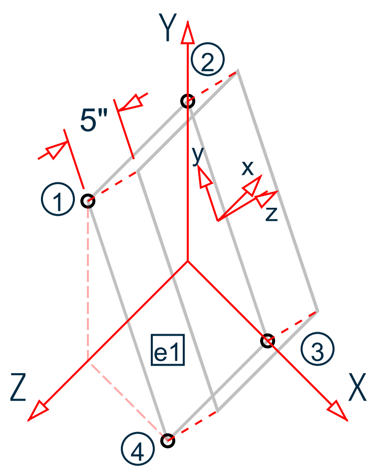


Figure 300: Example of uniform offsets for an element using different methods

```
UNIT INCH
JOINT COORDINATES
1 0 20 20
2 0 20 0
3 15 0 0
4 15 0 20
…
ELEMENT INCIDENCES
1 1 2 3 4
…
ELEMENT OFFSETS
* 1) Using global coordinates at each joint
1 JT1 3 4 0
1 JT2 3 4 0
1 JT3 3 4 0
1 JT4 3 4 0
```

```
* 2) Using local coordinates at each joint
1 JT1 LOCAL 0 0 5
1 JT2 LOCAL 0 0 5
1 JT3 LOCAL 0 0 5
1 JT4 LOCAL 0 0 5

* 3) Using a single Z offset
1 ZOFF 5

* 4) Using the same Z offset at each corner
1 ZOFF 5 5 5 5
```

Example of Physical Modeling Offset

The following example shows how element offsets can be used to model the "real world" physical properties of how concrete walls intersect with relation to the analytical model meshing centerlines. The adjoining wall on the right side has offsets from the analytical nodes of the elements (n1, n2) to the WP at the top and bottom of the wall intersection.

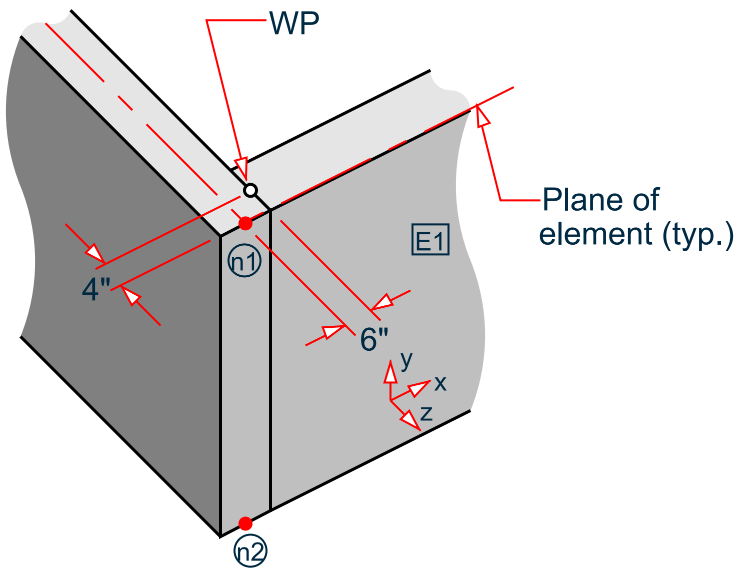


Figure 301: Example of working point (WP) represented by element offsets

```
UNIT INCH
ELEMENT OFFSETS
1 JT1 LOCAL 6 0 -4
1 JT2 LOCAL 6 0 -4
```

Related Links

- *G.11 Member and Plate Offsets* (on page 2125)
- *To assign plate offsets* (on page 813)
- *V. Element Offset Table Top Comparison* (on page 3242)
- *V. Element Offset Water Tank Comparison* (on page 3248)

TR.26 Specifying and Assigning Material Constants

Material constants are attributes like Modulus of Elasticity and Density which are required for operations like generating the stiffness matrix, computing selfweight, and for steel and concrete design.

In STAAD, there are two ways in which this data may be specified :

a. A two-step process that involves:

   a. Creating the material data by defining `MATERIAL` tags specified under the heading `DEFINE MATERIAL` (See TR.26.1 Define Material (on page 2303))

   b. Assigning them to individual members, plates and solids under the heading `CONSTANTS` (See TR.26.2 Specifying Constants for Members and Elements (on page 2305))

   This will create commands as shown below:

```
Part 1DEFINE MATERIAL
... name
...
...
END MATERIAL

Part 2CONSTANTS
MATERIAL name ...
```

b. Assign material attributes explicitly by specifying the individual constants (See TR.26.2 Specifying Constants for Members and Elements (on page 2305)).

```
CONSTANTS
E ...
POISSON .
```

See TR.26.3 Surface Constants Specification (on page 2311) for an explanation for the commands required to assign material data to Surface elements.

Related Links

- *G.12 Material Properties* (on page 2126)

TR.26.1 Define Material

This command may be used to specify the material properties by material name. You will then assign the members and elements to this material name in the `CONSTANTS` command (See TR.26.2 Specifying Constants for Members and Elements (on page 2305) for details and an example).

> **Note:** `ISOTROPIC` materials can be assigned to all element types, `2DORTHOTROPIC` materials should only be assigned to plate elements.

General Format

```
DEFINE MATERIAL
```

then

```
ISOTROPIC name
E f1
G f3
POISSON f6
DENSITY f7
ALPHA f8
DAMPING  f10
```

or

```
2DORTHOTROPIC name
E f1 (f2)
G f3 (f4) (f5)
POISSON f6
DENSITY f7
ALPHA f8 (f9)
DAMPING  f10
```

Repeat `ISOTROPIC` or `2DORTHOTROPIC` name and values for as many materials as desired then:

```
END MATERIAL (DEFINITION)
```

Where:

name	material name (name of up to 36 characters)
f1*, *f2	specifies Young's Modulus (E). (*f2* in local Y for 2DOthotropic materials)
f3*, *f4*, *f5	specifies Shear Modulus (G). For plates, the following are G values in local directions: *f3* is the G for in-plane shear; *f4* is the G for transverse shear in the local Y-Z direction; *f5* is the G for transverse shear in the local Z-X direction. (Enter only for beams when the Poisson ratio is *not* in the range of 0.01 to 0.499.)
f6	specifies Poisson's Ratio. If G is not entered, then this value is used for calculating the Shear Modulus (G = 0.5xE/(1+POISSON)). This value must be in the range of 0.01 to 0.499. Poisson's ratio must be entered for orthotropic plates or when Poisson cannot be computed from G.
f7	specifies weight density
f8*, *f9	Coefficient of thermal expansion. (*f9* in local Y for 2DOthotropic materials)
f10	the damping ratio to be used in computing the modal damping by the composite damping method in a dynamic analysis when `CDAMP` has been specified. Damping must be in the range of 0.001 to 0.990.

Note:

Any material property which you do not explicitly specify is assumed to be the default value.

- *f1* defaults to 0.0 but a positive value must be entered or an error will ensue
- *f2* defaults to *f1*

- *f3* defaults to 0.5xE/(1+POISSON)
- *f4* defaults to *f3*
- *f5* defaults to *f4*.
- *f6* defaults to a sliding scale value based on E (that is: 0.30 when E is near that of steel, 0.33 when E is near that of aluminum, 0.17 when E is near that of concrete) .
- *f7* defaults to 0.0
- *f8* defaults to 0.0
- *f9* defaults to *f8*
- *f10* defaults to 0.0

Tip: If one or more of the material properties is not explicitly specified, the results may be unpredictable or even incorrect with respect to the intended behavior. Therefore, it is best practice to always specify each material property for each defined material.

Related Links

- *M. To create a material definition* (on page 792)
- *M. To create an orthotropic material* (on page 795)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3.1 Composite Damping* (on page 2160)
- *G.4 Coordinate Systems and Structure Geometry* (on page 2086)
- *G.4.1 Global Coordinate System* (on page 2087)
- *G.4.2 Local Coordinate System* (on page 2088)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.26.2 Specifying Constants for Members and Elements

This command may be used to specify the individual material properties (Modulus of Elasticity, Poisson's ratio, Density, Coefficient of linear expansion, and material damping) of the members and elements. In addition, this command may also be used to specify the member orientation (BETA angle or reference point/vector).

General Format

```
CONSTANTS
```

To define the orientation:

```
BETA {  f5 | ANGLE | RANGLE } MEMBER memb/elem-list
```

```
{ REF f8 f9 f10 | REFJT f11 | REFVECTOR f12 f13 f14 } MEMBER memb/elem-list
```

To define the material properties:

```
MATERIAL name { MEMBER member/element-list | (ALL) }
```

or

```
{ E f1 | G f2 | POISSON f3 | DENSITY f4 | ALPHA f6 | CDAMP f7 } { MEMBER memb/elem-
list | BEAM | PLATE | SOLID | (ALL) }
```

Where:

name = material name as specified in the `DEFINE MATERIAL` command (Refer to TR.26.1 Define Material (on page 2303)).

```
{ REF f8 f9 f10 | REFJT f11 | REFVECTOR f12 f13 f14 } MEMBER memb/elem-list
```

Where:

memb/elem-list = MEM, BEA, PLA, SOL, ALL. Only MEM may be followed by a list. If none are specified, the default is to use `ALL`, which means all members and elements; BEA means all members; PLA, all plates; SOL, all solids.

Parameter	Value	Description
E	*f1*	Specifies the modulus of elasticity, E (i.e, Young's Modulus). This value must be provided before the `POISSON` for each member/ element in the Constants list.
G	*f2*	specifies Shear Modulus (G). Enter only for beams when Poisson would not be 0.01 to 0.499.
POISSON	*f3*	specifies Poisson's Ratio, υ. This value is used for calculating the Shear Modulus ($G = 0.5xE/(1+\upsilon)$).
DENSITY	*f4*	specifies weight per unit volume, γ
ALPHA	*f6*	Coefficient of thermal expansion, a
CDAMP	*f7*	Damping ratio to be used in computing the modal damping by the composite damping method in a dynamic analysis when `CDAMP` has been specified. Damping must be in the range of 0.001 to 0.990.
BETA	*f5*	Specifies member rotation angle in degrees (Refer to G.4.3 Relationship Between Global and Local Coordinates (on page 2092)).

The following values are used in various methods to define the BETA angle based on geometry:

f8, *f9*, *f10* = Global X, Y, and Z coordinates for the reference point f11= use location of joint *f11* for the reference point, from which the BETA angle will be calculated by STAAD.Pro.

f12, *f13*, *f14* = Establishes a Reference Vector along which the local y-axis is aligned. From the start node of the member, move by a distance of *f12* along the beam's local X axis, *f13* along the local Y axis, and *f14* along the local Z axis to define the end node of the reference vector. (The BETA angle is thus the angel between the local y-axis and the reference vector)

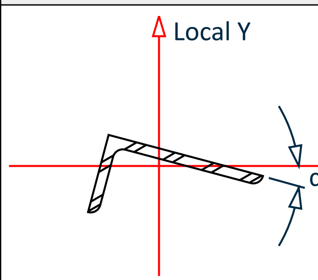
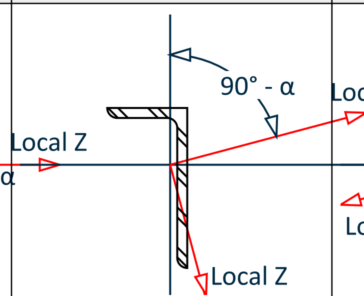
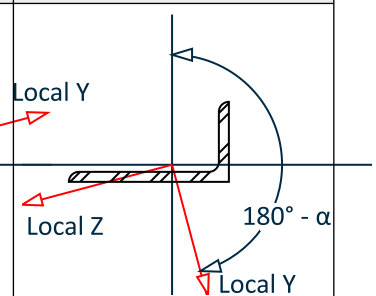
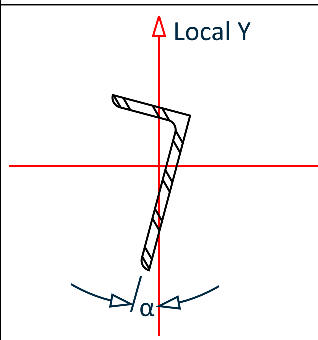
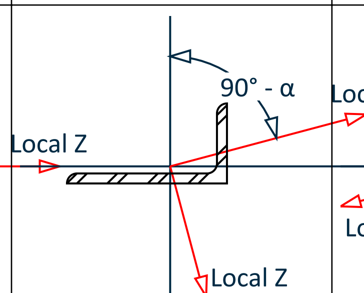
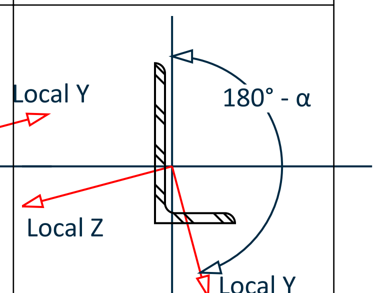
Using BETA ANGLE and RANGLE

Single angle sections are oriented according to their principal axes by default. If it is necessary to orient them such that their legs are parallel to the global axes, the BETA specification must be used. STAAD.Pro offers the following additional specifications for this purpose:

- `BETA ANGLE`
- `BETA RANGLE`

Both of the above options will result in an orientation with the legs parallel to the global axis. The ANGLE option rotates the section counterclockwise by the angle, α (where α = angle between the principal axis system and the geometric axis system of the angle). The RANGLE option rotates the section counterclockwise by an angle equal to (180° - α). For unequal angles, the right option must be used based on the required orientation.

Table 236: Effect of BETA ANGLE and BETA RANGLE commands

BETA value =	Zero (0)	ANGLE	RANGLE
ST Angle			
RA Angle			

Note: The figures in the preceding table are for a Y-UP system where the local x-axis goes into the screen. Global Y-axis is vertical. Refer to G.4.2 Local Coordinate System (on page 2088) for the orientation of angles when Z-UP is specified

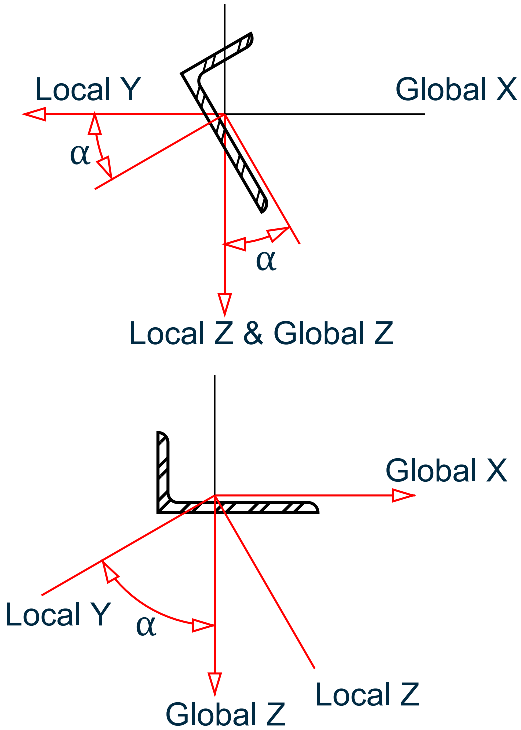


Figure 302: Orientation of an angle profile vertically aligned (i.e,. local x = Global Y) with (A) BETA = 0 (default) and (B) BETA = ANGLE

Note: The figures is for a Y-UP system where the local x-axis goes into the into the screen. Refer to G.4.2 Local Coordinate System (on page 2088) for the orientation of angles when Z-UP is specified.

Built-In Material Constants

For E, G, POISSON, DENSITY, ALPHA, and CDAMP, built-in material names can be entered instead of a value for *f1*. The built-in names are `STEEL`, `CONCRETE`, & `ALUMINUM`. Appropriate values will be automatically assigned for the built-in names.

Table 237: Constants (in Kip, inch, Fahrenheit units)

Constant	Material			Units
	Steel	Concrete	Aluminum	
E (US)	29,000	3,150	10,000	Kip/in^2
Poisson's	0.30	.17	.33	(ratio)
Density	.000283	.0000868	.000098	Kip/in^3
Alpha	6.5E-6	5.5E-6	12.8E-6	L/L/° F
CDAMP	.03	.05	.03	(ratio)

Constant	Material			Units
	Steel	Concrete	Aluminum	
E (nonUS)	29,732.736			Kip/in^2

Table 238: Constants (in MKS, Celsius units)

Constant	Material			Units
	Steel	Concrete	Aluminum	
E (US)	199, 947,960	21,718,455	68,947,573	kN/m^2
Poisson's	0.30	.17	.33	(ratio)
Density	76.819 541	23.561612	26.601820	kN/m^3
Alpha	12.0E-6	10.0E-6	23.0E-6	L/L/° C
CDAMP	.03	.05	.03	(ratio)
E (nonUS)	205, 000,000			kN/m^2

Note: E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

Example 1

```
DEFINE MATERIAL
ISOTROPIC CFSTEEL
E 28000.
POISSON 0.25
DENSITY 0.3E-3
ALPHA 11.7E-6
DAMP 0.075
END MATERIAL
CONSTANTS
MATERIAL  CFSTEEL MEMB 1 TO 5
CONSTANTS
E 2.1E5  ALL
BETA 45.0  MEMB  5  7  TO  18
DENSITY STEEL MEMB  14  TO  29
BETA  90  MEMB X
```

Example 2

The REFVECTOR command is used as in the following example:

```
REFVECTOR 0 2 1 MEMBER 27 TO 32
```

This command will set BETA as 90° for all members parallel to the X-axis and instructs the program to do the following procedure:

1. Establish the beam's local X,Y and Z axis corresponding to Beta = 0
2. Set the start node of the reference vector to be the same as the start node of the member.
3. From the start node of the reference vector, move by a distance of 0 along the beam's local X axis, 2 along the local Y axis, and 1 along the local Z axis. This establishes the end node of the reference vector.
4. At the end of step 3, the start node as well as the end node of the reference vector are known. That is the now the final direction of the member's local Y axis.

Since the local Y axis corresponding to Beta 0 is known, and the local-Y axis corresponding to the beam's final position has been established in step 4, Beta angle is calculated as the angle between these two vectors.

In this example, the angle is $Tan\text{-}1(1/2) = 26.5651$ degrees

Notes

a. The value for E must always be given first in the Constants list for each member/element.
b. All numerical values must be provided in the current units.
c. It is not necessary or possible to specify the units of temperature or ALPHA. You must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for temperature (see TR.32.6 Temperature Load Specification for Members, Plates, and Solids (on page 2493)).
d. If G is not specified, but Poisson (υ) *is* specified, G is calculated as $E/[2\ (1+v)]$.
e. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
f. If G and Poisson are both specified, the input value of G is used. That is, G is *not* calculated in this situation.
g. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated as $[(E/2G)-1]$.
h. To obtain a report of the values of these terms used in the analysis, specify the command `PRINT MATERIAL PROPERTIES`.
i. Local offsets are defined in the local axes prior to rotation when a BETA angle is used.

Related Links

- *GS.To change the system units* (on page 43)
- *M. To align a member to a reference point* (on page 801)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *M. To assign a member rotation angle* (on page 799)
- *M. To align a single angle to its flanges* (on page 800)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *M. To align a member to a reference point* (on page 801)
- *M. To assign material constants* (on page 798)
- *G.17.3.3.1 Composite Damping* (on page 2160)
- *M. To assign a composite damping ratio* (on page 895)
- *G.17.3.3.1 Composite Damping* (on page 2160)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *G.6.8 Curved Members* (on page 2120)
- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)

TR.26.3 Surface Constants Specification

Explained below is the command syntax for specifying constants for surface entities.

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

General Format

```
SURFACE CONSTANTS
```

```
{ E f1 | G f2 | POISSON f3 | DENSITY f4 | ALPHA f5 }  { LIST surface-list | ALL }
```

Where:

E *f1*	Young's Modulus (E)
G *f2*	Modulus of Rigidity (G)
POISSON *f3*	Poisson's ratio
DENSITY *f4*	weight density
ALPHA *f5*	coefficient of thermal expansion

In lieu of numerical values, built-in material names may be used for the above specification of constants. The built-in names are STEEL, CONCRETE, and ALUMINUM. See TR.26.2 Specifying Constants for Members and Elements (on page 2305) for the values used in the built-in materials.

Example

```
SURFACE CONSTANTS
E 3150 LIST 1 TO 4
POISSON 0.17 ALL
DENSITY 8.68e-005 LIST 1 TO 4
ALPHA 5.5e-006 LIST 1 TO 4
```

Alternatively, where the material constants have been defined in a material object such as CONCRETE:

```
SURFACE CONSTANTS
MATERIAL CONCRETE LIST 1 TO 4
```

Notes

a. If G is not specified, but Poisson is specified, G is calculated from $E/\ [2\ (1 + Poisson)]$.
b. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
c. If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
d. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from $[(E/2G) - 1]$.
e. To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.

TR.26.4 Modal Damping Information

To define unique modal damping ratios for every mode. STAAD.Pro allows you to specify modal damping either directly or by using Rayleigh damping (the algebraic combination of mass-proportional damping and stiffness-proportional damping). This will be used in a dynamic analysis when `MDAMP` has been specified.

Note: If all modes have the same damping, then you may simply enter damping within the define time history load or within the dynamic loading commands.

Damping may be entered:

- by specifying that the program `EVALUATE` each mode's damping based on the frequency of the mode and the minimum and maximum damping entered here. The formula used to evaluate the damping is given in Evaluate Damping (on page 2162).
- by specifying that the program `CALCULATE` each mode's damping based on the frequency of the mode and the mass factor, ALPHA, and the stiffness factor, BETA. The formula used to calculate the damping is given in Calculate Damping (on page 2160).
- explicitly for some or all modes (`EXPLICIT`).

The damping entered will be used in time history load cases and in response spectrum load cases that use the `CQC` or `ASCE4` methods and/or Spectra vs. Period curves versus damping.

General Format

```
DEFINE DAMPING INFORMATION
```

```
{ EVALUATE dmin (dmax) | CALCULATE ALPHA c1 BETA c2 (MAX dmax MIN dmin) | EXPLICIT d1
(d2 d3 d4 … dn ) }
```

```
END
```

Where:

dmin	the minimum damping ratio to be used in the evaluate damping formula
dmax	the optional maximum damping ratio to be used in the evaluate damping formula
ALPHA ***c1***	the mass-proportional damping coefficient, α, used in the cacluate damping formula
BETA ***c2***	the stiffness-proportional damping coefficient, β, used in the cacluate damping formula
MAX ***dmax***, **MIN** ***dmin***	optional minimum and maximum damping ratios, respectively, sed in the calculate damping formula
d1 d2 … dn	the damping ratios for each mode given in the explicit method

Note: Damping ratios must be in the range 0.0 through 1.0.

Related Links

- *G.17.3.3.2 Modal Damping* (on page 2160)
- *M. To explicitly define damping values for modes* (on page 895)
- *M. To evaluate damping for modes* (on page 896)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3.2 Modal Damping* (on page 2160)

TR.26.5 Composite Damping for Springs

This command may be used to designate certain support springs as contributing to the computation of modal damping by the composite damping method. The Response Spectrum or Time History dynamic response analyses must select composite damping, `CDAMP`, for this data to have any effect on results and the material used must have suitable damping values, `DAMP`, specified.

General Format

```
SPRING DAMPING
```

```
joint-list *{ KFX f1 | KFY f2 | KFZ f3 }
```

Where:

f1, f2, f3	damping ratios (0.001 to 0.990) in the local X, Y, and Z directions, respectively

Note: At least one of KFX, KFY, or KFZ must be entered and each one entered must have a spring damp value following it.

Description

If this Spring Damping command is entered, then all springs in the structure are included in the composite damping calculation, otherwise no spring is considered in that calculation.

This input command does not create a spring. Rather, if a support spring exists at the joint in the specified direction, then it will be assigned the damping ratio.

Note: This is not a discrete damper definition.

TR.26.6 Member Imperfection Information

To define camber and drift specifications for selected members. Drift is usually for columns and camber for beams.

General Format

```
DEFINE IMPERFECTION
```

```
CAMBER { Y | Z } (f1) RESPECT (f2) *{ XR f4 f5 | YR f4 f5 | ZR f4 f5 | MEM memb-list |
 LIST memb-list | ALL }
```

```
DRIFT { Y | Z } (f3) *{ XR f4 f5 | YR f4 f5 | ZR f4 f5 | MEM memb-list | LIST memb-list |
 ALL }
```

f_1 Camber value. Default = 300

f_2 Respect value. Default = 1.6

f_3 Drift value. Default = 200.

f_4, f_5 global coordinate values to specify X, Y, or Z range for member selection.

Imperfections will be simulated by loads. These loads will be generated for the specified members if there is an Imperfection Analysis specified and if the specified members are active, in compression, and are not truss or tension/compression only members.

Notes

Camber is the maximum offset of the neutral axes in the defined direction from a vector that passes through the ends of the beam (i.e., the local X axis) defined as the ratio of member length/offset.

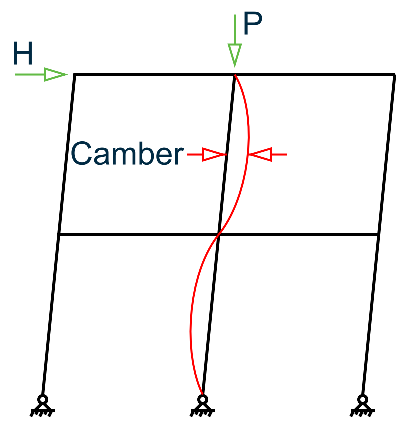


Figure 303: Camber definition

Drift is the offset of the end a member from its specified location defined as a ratio of member length/offset.

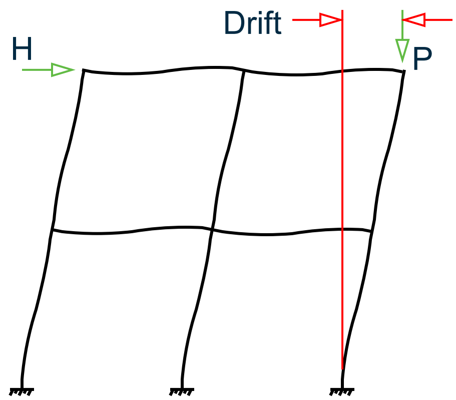


Figure 304: Drift definition

RESPECT is a non dimensional constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms is calculated and called EPSILON. If EPSILON is less than RESPECT, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

EPSILONy = Length * SQRT[(abs(axial load)) / EIz]

EPSILONz = Length * SQRT[(abs(axial load)) / EIy]

Example

```
SUPPORTS
1 FIXED
2 FIXED BUT FX
DEFINE IMPERFECTION
CAMBER Y 300 RESPECT 0.4 ALL
LOAD 1 GRAVITY + LATERAL
MEMBER LOAD
1 UNI GY -1
JOINT LOAD
2 FX -10
PERFORM IMPERFECTION ANALYSIS PRINT STATICS CHECK
```

Related Links

- *M. To assign member imperfection for members* (on page 806)
- *M. To assign member imperfection for members* (on page 806)
- *G.17.2.4 Imperfection Analysis* (on page 2148)

TR.27 Support Specifications

STAAD support specifications may be either parallel or inclined to the global axes.

See TR.27.1 Global Support Specification (on page 2316) for specification of supports parallel to the global axes. See TR.27.2 Inclined Support Specification (on page 2318) for specification of inclined supports.

Related Links

- *G.13 Supports* (on page 2127)

TR.27.1 Global Support Specification

This set of commands may be used to specify the SUPPORT conditions for supports parallel to the global axes.

For SURFACE elements, if nodes located along a straight line are all supported identically, as in the case of the base of a wall, support generation can be performed for assigning restraints to those nodes. See the GENERATE option in the command syntax below. You only need to provide the starting and ending nodes of the range, and the type of restraint.

General Format

```
SUPPORTS
```

```
{ joint-list | ni TO nj GENERATE } { PINNED | FIXED ( BUT release-spec (spring-spec) )
| ENFORCED ( BUT release-spec) }
```

```
release-sepc = { FX | FY | FZ | MX | MY | MZ }
```

```
spring-spec = *{KFX f1 | KFY f2 | KFZ f3 | KMX f4 | KMY f5 | KMZ f6 }
```

Where:

ni, nj = Start and end node numbers, respectively, for generating supports along a SURFACE element edge.

f_1 ... f_6 = Spring constants corresponding to support spring directions X, Y, and Z and rotations about X, Y, and Z, repsectively.

Description of Pinned and Fixed

A PINNED support is a support that has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A FIXED support has both translational and rotational restraints. A FIXED BUT support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a FIXED BUT support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are f1 through f6. The rotational spring constants are always per degree of rotation. All six releases may be provided as may be required when using the CHANGE command. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

1. See TR.38 Change Specification (on page 2660) for information on specification of SUPPORTS along with the CHANGE command specifications.
2. Spring constants must be provided in the current units.
3. All spring DOF must be entered after the last non-spring DOF is specified, if both are on the same line.
4. If there are two entries for the same joint, then:
   a. any direction that is pinned/fixed on either will be fixed in that direction.
   b. any direction released on one and is a spring on the other will use the spring.
   c. any direction that is pinned/fixed on one and a spring on the other will use the spring.
   d. any direction that is a spring on two or more entries will have the spring constants added.

Example 1

```
SUPPORTS
1 TO 4  7 PINNED
5  6 FIXED BUT FX MZ
8  9 FIXED BUT MZ KFX 50.0 KFY 75.
18  21 FIXED
27 FIXED BUT KFY 125.0
```

In this example, joints 1 to 4 and joint 7 are pinned. No moments are carried by those supports. Joints 5 and 6 are fixed for all DOF except in force-X and moment-Z. Joints 8 and 9 are fixed for all DOF except moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed for all translational and rotational degrees of freedom. At joint 27, all the DOF are fixed except the FY DOF where it has a spring with 125 units spring constant.

Description of Enforced

Enforced Support defines which translational and rotational directions, at a joint, may have a support displacement imposed. The support displacements are defined for each load case in TR.32.8 Support Joint Displacement Specification (on page 2494). If no support displacement is entered, then zero displacement will be imposed, as if that direction was FIXED. The enforced displacement directions will be fixed for dynamic load cases.

If there are two entries for the same joint, then any direction that is enforced on either will be enforced in that direction, overriding any other support specification for that joint-direction.

Currently the support generation command can only be used in conjunction with the Surface element support specifications.

Example 2

```
SUPPORTS
3 TO 7 GENERATE PIN
```

The above command will generate pinned supports for all joints located between nodes No. 3 and 7 along a straight line. This may include joints explicitly defined by the user or joints generated by the program for internal use only (e.g., as a result of SET DIVISION and SURFACE INCIDENCES commands).

Related Links

- *M. To assign a fixed or pinned support* (on page 819)
- *G.13 Supports* (on page 2127)
- *M. To assign an enforced support* (on page 820)
- *G.13 Supports* (on page 2127)
- *EX. US-24 Analysis of a Concrete Block Using Solid Elements* (on page 6495)
- *EX. UK-24 Analysis of a Concrete Block Using Solid Elements* (on page 6782)
- *M. To assign a spring support* (on page 821)
- *G.13 Supports* (on page 2127)
- *EX. US-3 Soil Springs for Portal Frame* (on page 6327)

- *EX. UK-3 Soil Springs for Portal Frame* (on page 6611)
- *G.13 Supports* (on page 2127)

TR.27.2 Inclined Support Specification

These commands may be used to specify supports that are inclined with respect to the global axes.

General Format

```
SUPPORT
joint-list INCLINED { f1 f2 f3 | REF  f4 f5 f6 | REFJT f7 } { PINNED | FIXED ( BUT
release-spec (spring-spec) ) | ENFORCED ( BUT release-spec) }
```

Where:

f1, f2, f3	x, y, z relative distances from the joint in the global directions to the reference point
f4, f5, f6	x, y, z global coordinates of an arbitrary reference point
f7	a joint number whose x, y, z global coordinates are the reference point

A vector from the joint location to the reference point location serves to define a local coordinate system (same as member with BETA = 0). The inclined support directions are in this local "Inclined Support Axis System (on page 2318)" (see more below).

Note: The release-spec and spring-spec are the same as in the Global Supports (See TR.27.1 Global Support Specification (on page 2316)). However, FX through MZ and KFX through KMZ refer to forces/moments and spring constants in the "Inclined Support Axis System" (see below).

Inclined Support Axis System

The INCLINED support specification is based on the "Inclined Support axis system." The local x-axis of this system is defined by assuming the inclined support joint as the origin and joining it with a "reference point" with coordinates of f1, f2, and f3 (see the following figure) measured from the inclined support joint in the global coordinate system.

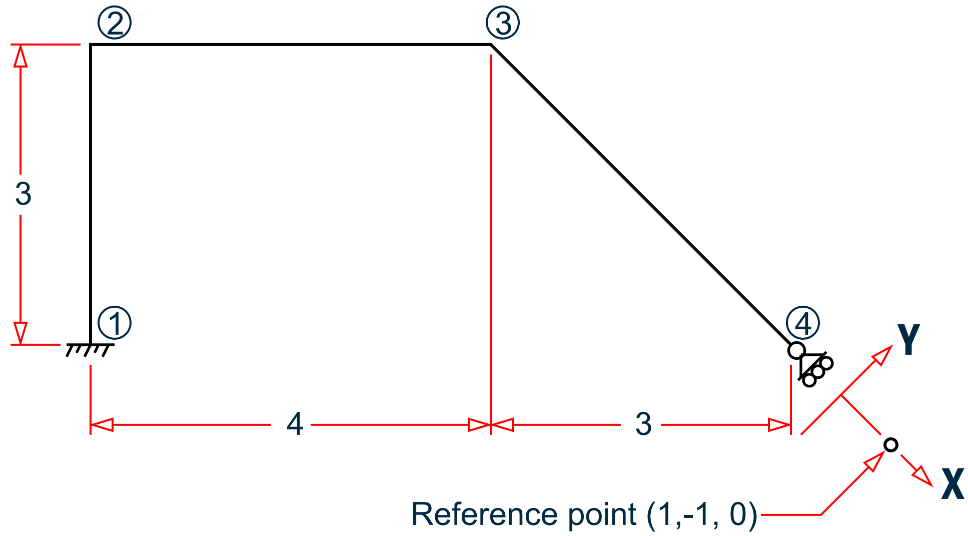


Figure 305: Reference point for defining an inclined support angle

The Y and Z axes of the inclined support axis system have the same orientation as the local Y and Z axes of an imaginary member whose BETA angle is zero and whose incidences are defined from the inclined support joint to the reference point. G.4.3 Relationship Between Global and Local Coordinates (on page 2092) for more information on these concepts.

Note: Inclined support directions are assumed to be same as global when computing some dynamic and UBC intermediate results (e.g., global participation factors). If masses and/or forces in the free directions at inclined supports are a relatively small portion of the overall forces, then the effect should be very small.

Example

```
SUPPORT
4 INCLINED 1.0 -1.0 0.0 FIXED BUT FY MX MY MZ
```

Related Links

- *M. To assign an inclined support* (on page 823)
- *G.13 Supports* (on page 2127)
- *EX. US-19 Inclined Supports* (on page 6458)
- *EX. UK-19 Inclined Supports* (on page 6744)
- *G.13 Supports* (on page 2127)

TR.27.3 Automatic Spring Support Generator for Foundations

STAAD.Pro has a facility for automatic generation of spring supports to model footings and foundation mats. This command is specified following the SUPPORT command.

General Format

```
SUPPORT
```

then either

```
{ joint-list ELASTIC FOOTING f1 (f2) | joint-list ELASTIC MAT | plate-list PLATE MAT }
DIR { X | XONLY | Y |  YONLY | Z | ZONLY } SUBGRADE f3
```

```
(PRINT) ( {COMP | MULTI } )
```

or

```
plate-list PLATE MAT DIR ALL SUBGRADE f4 (f5 f6)
```

```
(PRINT) ( {COMP | MULTI } )
```

Where:

FOOTING *f1 f2*	Length and width of the ELASTIC footing. If f_2 is not given, the footing is assumed to be a square with sides f_1.
X,Y,Z	Global direction in which soil springs are to be generated.
SUBGRADE *f3*	Soil subgrade modulus in current force/area/length units for elastic footings.
SUBGRADE *f4 f5 f6*	Soil subgrade modulus for mat foundations for use with ALL option in current force/area/length units in Y, X, Z directions respectively. f4, f5 default to f3 if omitted.

Note: For mat foundation subgrade values, the Y value is specified *first*, followed by X and then Z.

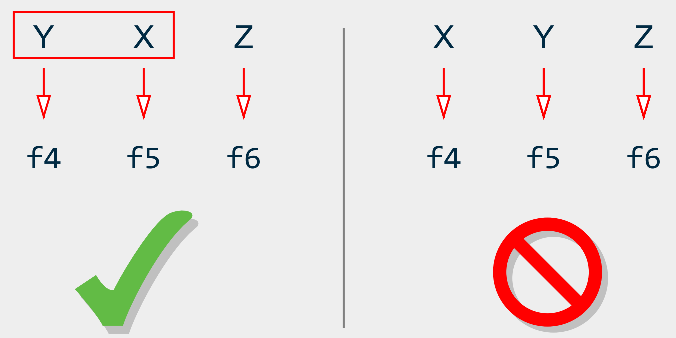


Figure 306: Correct and Incorrect order for specifying subgrade modulus values

Caution: Do not use this command with SET Z UP.

Description

If you want to specify the influence area of a joint yourself and have STAAD simply multiply the area you specified by the sub-grade modulus, use the ELASTIC FOOTING option. Situations where this may be appropriate are such as when a spread footing is located beneath a joint where you want to specify a spring support. A value for f_1 (and f_2 if its a non-square footing) is required for the FOOTING option.

If you want to have STAAD calculate the influence area for the joint (instead of you specifying an area yourself) and use that area along with the sub-grade modulus to determine the spring stiffness value, use the ELASTIC MAT option. Situations where this may be appropriate are such as when a slab is on soil and carries the weight of the structure above. You may have modeled the entire slab as finite elements and wish to generate spring supports at the nodes of the elements.

The `PLATE MAT` option is similar to the Elastic Mat except for the method used to compute the influence area for the joints. If your mat consists of plate elements and all of the influence areas are incorporated in the plate areas, then this option is preferable. Enter a list of plates or YRANGE f1 f2 at the beginning of the command, the joint influence areas are then calculated using the same principles as joint forces would be from uniform pressure on these plates. This method overcomes a major limitation of the Delaunay triangle method used in the ELASTIC MAT option, which is that the contour formed by the nodes of the mat must form a convex hull.

The `PLATE MAT DIR ALL` option is similar to the Plate Mat except that the spring supports are in 3 directions. If the compression only option is also specified, then the compression direction will be assumed to be in the Y direction. If the Y spring at a joint goes slack (lift off), then the X and Z spring stiffnesses for that joint will also be set to zero. Otherwise the X and Z springs act in both directions. The influence area for the X and Z springs is the same as used for the Y spring. Three values of subgrade reaction may be entered, the first is for the Y direction, the second for X and the third for Z.

The keyword `DIR` is followed by one of the alphabets X, Y or Z (or XONLY, YONLY, or ZONLY) which indicate the direction of resistance of the spring supports. If X or Y or Z is selected then a spring support is generated in that direction plus 3 other directions receive a fixed support, e.g., if Y is selected, then FY is supported by a spring; FX and FZ and MY are fixed supports; and MX and MZ are free. If XONLY, YONLY, or ZONLY are selected then only a spring support in that direction is generated.

The keyword `SUBGRADE` is followed by the value of the subgrade reaction. The value should be provided in the current unit system signified by the most recent `UNIT` statement prior to the `SUPPORT` command.

The `PRINT` option prints the influence area of each joint.

Use the `COMP` option generated will be compression only springs

Use the `MULTI` option to generate multilinear springs. Add the associated multilinear curve input after each `MAT` command (with the multi option) to describe the displacement-spring constant curve. See TR.27.4 Multilinear Spring Support Specification (on page 2322) for additional information on this input format. The actual spring constant used will be the subgrade modulus (f3 entered above) times the influence area (computed by STAAD) times the si values entered in the curve (so the curve stiffness values will likely be between 0.0 and 1.0).

```
SPRINGS d1 f1 d2 f2 … dn fn
```

Example

```
SUPPORTS
1 TO 126 ELASTIC MAT DIREC Y SUBG 200.
1 TO 100 PLATE MAT DIREC Y SUBG 200.
YR -.01 .01 PLA MAT DIR Y SUBG 200 MUL
SPRINGS -100 2.0 -0.51 2.0 -0.50 1.0 0.0 0.0 1000 0.0
```

The first command above instructs STAAD to internally generate supports for all nodes 1 through 126 with elastic springs. STAAD.Pro first calculates the influence area perpendicular to the global Y axis of each node and then multiplies the corresponding influence area by the soil subgrade modulus of 200.0 to calculate the spring constant to be applied to the node. In the 2nd example, the nodes of plates 1 to 100 are assigned spring supports, generated using a subgrade modulus of 200 units.

Notes

a. A closed surface is generated by the program based on the joint-list that accompanies the `ELASTIC MAT` command. The area within this closed surface is determined and the share of this area for each node in the list is then calculated.

Hence, while specifying the joint-list, one should make sure that these joints make up a closed surface. Without a proper closed surface, the area calculated for the region may be indeterminate and the spring constant values may be erroneous. Consequently, the list should have at a minimum, 3 nodes.

b. The internal angle formed by 2 adjacent segments connecting 3 consecutive nodes in the list should be less than 180 degrees. In other words, the region should have the shape of a convex polygon. The example below explains the method that may be used to get around a situation where a convex polygon is not available.

c. For the model comprised of plate elements 100 to 102 in the figure below, one wishes to generate the spring supports at nodes 1 to 8. However, a single `ELASTIC MAT` command will not suffice because the internal angle between the edges 1-8 and 8-7 at node 8 is 270 degrees, which violates the requirements of a convex polygon.

So, you should break it up into two commands:

```
1 2 3 8 ELASTIC MAT DIREC Y SUBG 200.
3 4 5 6 7 8 ELASTIC MAT DIREC Y SUBG 200.
```

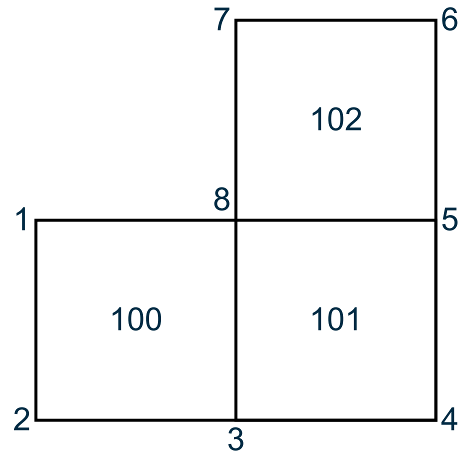


Figure 307: Example for elastic mat generation for a convex polygonal shape

Joints 3 and 8 will hence get the contribution from both of the above commands.

The command works only when the plane of the closed region is parallel to one of the global planes X-Y, Y-Z or X-Z. For regions that are inclined to one of the global planes, the spring constant will have to be evaluated manually and specified using the FIXED BUT type of spring support.

Related Links

- *M. To assign a foundation support* (on page 824)
- *G.13 Supports* (on page 2127)
- *EX. US-23 Spring Support Generation for a Slab on Grade* (on page 6484)
- *EX. UK-23 Spring Support Generation for a Slab on Grade* (on page 6771)

TR.27.4 Multilinear Spring Support Specification

When soil is modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression.

General Format

```
MULTILINEAR SPRINGS
```

joint-list SPRINGS d_1 s_1 d_2 s_2 … d_n s_n

Where:

d_i s_i pairs represent displacement and spring constant pairs (s_i is zero or positive), starting from the maximum negative displacement to the maximum positive displacement.

The first pair defines the spring constant from negative infinity displacement up to the displacement defined in the second pair. The second pair define the spring constant when the support displaces in the range from the displacement defined in the second pair, up to the displacement defined in the third pair. This continues for each displacement and spring constant pair until the last pair which defines the spring constant for displacements greater than the displacement in the last pair to positive infinity.

Each load case in a multilinear analysis must be separated by a `CHANGE` command and have its own `PERFORM ANALYSIS` command. There may not be any `PDELTA`, `NONLIN`, dynamics, `CABLE`, or `TENSION/COMPRESSION` analysis included. The multilinear spring command will initiate an iterative analysis and convergence check cycle. The cycles will continue until the root mean square (RMS) of the effective spring rates used remain virtually the same for two consecutive cycles.

Example

```
UNIT …
SUPPORT
1 PINNED; 2 4 FIXED BUT KFY 40.0
MULTILINEAR SPRINGS
2 4 SPRINGS -1 40.0 -0.50 50.0 0.5 65.0
```

Load-Displacement characteristics of soil can be represented by a multilinear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. A typical spring characteristic of soil may be represented as the step curve as shown in the figure below. In the above example, the multilinear spring command specifies soil spring at joints 2 and 4. (Note that the amplitude of the step curve does not change after the first point.)

Notes

a. `SUPPORT` springs must have previously been entered for each spring entered here. For the first cycle, the spring value used will be the support spring value (not the zero displacement value here). Use a realistic and stable value.

b. All directions that have been defined with an initial spring stiffness in the `SUPPORT` command will become multilinear with this one curve.

c. This command can be continued to up to 11 lines by ending all but last with a hyphen. The semi-colons and the XRANGE, YRANGE, ZRANGE list items may not be used.

d. This command needs a minimum of two displacement and spring constant pairs.

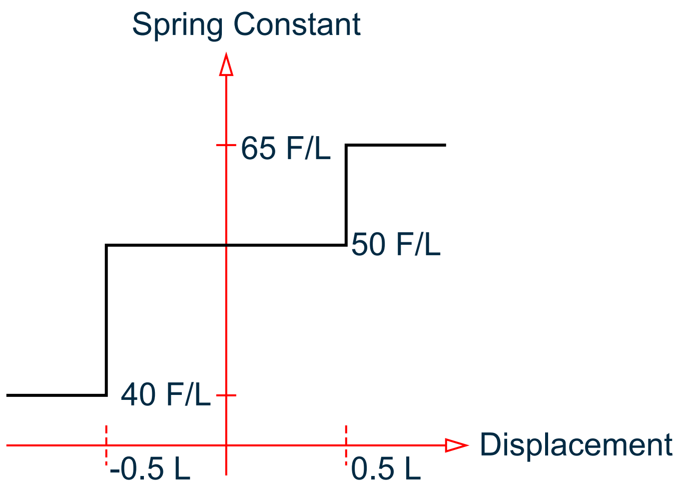


Figure 308: Spring constant is always positive or zero. F = Force Units, L = Length Units

e. Multilinear springs should *not* be used in the following conditions:

Modal dynamics
Buckling analysis
Imperfection analysis
`PDELTA` analysis
`NONLINEAR` analysis
Advanced cable analysis
Direct analysis
Models with tension/compression members and/or supports
Models with inclined supports

Related Links

- *G.17.2.5 Multilinear Analysis* (on page 2148)

TR.27.5 Spring Tension/Compression Specification

This command may be used to designate certain support springs as Tension-only or Compression-only springs.

General Format

```
SPRING TENSION
joint-list  (spring-spec )
SPRING COMPRESSION
joint-list  (spring-spec )
```

where:

```
spring-spec  = *{ KFX | KFY | KFZ | ALL }
```

Description

Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them. Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

If no spring spec is entered or ALL is entered then all translational springs at that joint will be tension (or compression) only. This input command does not create a spring, only that if a support spring exists at the joint in the specified direction then it will also be tension (or compression) only. Refer to TR.27 Support Specifications (on page 2315) to define springs.

For compression only springs the `ALL` parameter has special meaning. The compression only spring is in the Y direction; the X and Z direction springs are bi-directional. However when the Y direction spring goes slack, the X and Z springs at the same joint are made inactive as well.

Note: The procedure for analysis of Tension-only or Compression-only springs requires iterations for every load case and therefore may be quite involved.

Since this command does not specify whether the spring is in the positive or negative direction from the joint, it is assumed in STAAD.Pro to be in the negative direction. For negative displacement the spring is in compression and for positive the spring is in tension.

If a `CHANGE` command is used (because of a change in list of tension springs, supports, etc.), then the `SET NL` command must be used to convey to STAAD.Pro that multiple analyses and multiple structural conditions are involved.

1. Refer to TR.5 Set Command Specification (on page 2206) for explanation of the `SET NL` command. The number that follows this command is an upper bound on the total number of primary load cases in the file.
2. STAAD.Pro performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a `SET ITERLIM i` command (with `i` > 10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high. If not converged, a message will be in the output.
3. The principle used in the analysis is the following.
   - The program reads the list of springs declared as `SPRING TENSION` and/or `COMPRESSION`.
   - The analysis is performed for the entire structure and the spring forces are computed.
   - For the springs declared as `SPRING TENSION` / `COMPRESSION`, the program checks the axial force to determine whether it is tensile or compressive. Positive displacement is `TENSION`. If the spring cannot take that load, the spring is "switched off" from the structure.
   - The analysis is performed again without the switched off springs.
   - Up to `ITERLIM` iterations of the above steps are made for each load case.
   - This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable. You may need to include some support in each global direction that is not tension (or compression) only to be stable on every iteration.
4. A revised `SPRING TENSION` / `COMPRESSION` command and its accompanying list of joints may be provided after a `CHANGE` command. If entered, the new `SPRING` commands replace all prior `SPRING` commands. If not entered after a CHANGE, then the previous spring definitions are used.
5. The `SPRING TENSION` command should not be used if the following load cases are present: Response Spectrum load case, Time History Load case, Moving Load case. If used, the `SPRING TENSION` / `COMPRESSION` will be ignored in all load cases.
6. If the `SPRING TENSION` / `COMPRESSION` command is used in a model with UBC, IBC or other such seismic load cases, each such load case must be followed by an `ANALYSIS` and `CHANGE` command.

Notes

a. A spring declared as tension-only or compression-only will carry axial forces only. It will not carry moments.

b. The `SPRING TENSION` / `COMPRESSION` commands should not be specified if the `INACTIVE MEMBER` command is specified.

c. Do not use Load Combination to combine these cases. Tension/Compression cases are non-linear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.

Example

```
SPRING TENSION
12 17 19 TO 37 65
SPRING COMPRESSION
5 13 46 TO 53 87 KFY
```

The following is the general sequence of commands in the input file if the `SPRING TENSION` or `COMPRESSION` command is used. This example is for the `SPRING TENSION` command. Similar rules are applicable for the `SPRING COMPRESSION` command. The dots indicate other input data items.

```
STAAD …
SET NL …
UNITS …
JOINT COORDINATES
…
MEMBER INCIDENCES
…
ELEMENT INCIDENCES
…
CONSTANTS
…
MEMBER PROPERTY
…
element property
…
SUPPORTS
…
spring TENSION
…
LOAD 1
…
LOAD 2
…
LOAD 3
REPEAT LOAD
…
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT …
PRINT …
PARAMETER
…
CHECK CODE …
FINISH
```

Related Links

- *M. To assign support springs as a one-way* (on page 823)
- *G.13.1 Tension- and Compression- Only Springs* (on page 2127)

TR.28 Rigid Diaphragm Modeling

STAAD.Pro has two methods for defining rigid floor diaphragms. Both are functionally equivalent but the Control-Dependent joint requires that an analytical node be located and specified as the control node. For most models, the simpler Floor Diaphragm option is preferred.

TR.28.1 Control/Dependent Specification

This set of commands may be used to model specialized linkages (displacement tying, rigid links) through the specification of “control” and “dependent” joints. Please read the notes for restrictions.

Note: Control / Dependent was formerly referred to as master / slave.

General Format

```
Dependent *{ XY | YZ | ZX | RIGID | FX | FY | FZ | MX | MY | MZ } CONTROL j JOINT
joint-spec
```

where:

```
joint-spec = { joint-list  | *{ XRANGE | YRANGE | ZRANGE } f1 f2 }
```

Description

The control/dependent option provided in STAAD.Pro allows you to model specialized linkages (displacement tying, rigid links) in the system. For example, `DEPENDENT FY` … connects the two joints such that the Y displacement at the dependent will be the sum of Y displacement at the control plus the rigid rotation,

$$R \sin \theta$$

Note that instead of providing a joint list for the dependent joints, a range of coordinate values (in global system) may be used. All joints whose coordinates are within the range are assumed to be dependent joints. For convenience, the coordinate range specified for dependent joints in this entry may include the control joint for this entry. However, control and dependent joints of other entries must *not* be included in the coordinate range. All 2 or 3 ranges can be entered to form a “tube” or “box” for selecting joints in the tube or box region.

Fx, Fy etc. are the directions in which they are dependent (any combination may be entered).

If two or more entries have the same control, the dependent lists will be merged. Please ensure that the same direction specs are used.

The direction specifiers (XY, YZ, ZX) are combinations of the basic directions, XY is the same as entering FX, FY, MZ; etc. Any combination of direction specifiers may be entered. An example of the use of this format is: a rigid diaphragm floor that still retains bending flexibility entered as `SLA ZX` … .

If `RIGID` or if all directions are provided, the joints are assumed to be rigidly connected as if `SLA DIA RIG` were entered, even if `DIA` is omitted.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Restrictions

- Solid elements may not be connected to dependent joints.
- Control joints may not be control or dependent in another entry.
- Dependent joints may not be control or dependent in another entry.
- Dependent directions at joints may not be supported directions or have displacements imposed.
- Control and/or dependent joints may not be inclined supports.
- The control/dependent specification is only intended for linear static and dynamic analysis.
- Multilinear springs are not permitted.
- Control/dependent joints may not be specified when rigid floor diaphragms are used (refer to TR.28.2 Floor Diaphragm (on page 2328)).
- If a TR.38 Change Specification (on page 2660) is specified, then any control/dependent joints must be re-specified if they are intended to be used after the change.

The internal processing of any control/dependent command includes an automatic bandwidth reduction.

Example

Fully Rigid and Rigid Floor Diaphragm

```
DEPENDENT RIGID CONTROL 22 JOINT 10 TO 45
DEPENDENT RIGID CONTROL 70 JOINT YRANGE 25.5 27.5
DEP ZX CON 80 JOINT YR 34.5 35.5
```

Related Links

- *G.14 Rigid Diaphragms* (on page 2128)
- *M. To assign a rigid link between nodes* (on page 817)
- *G.14 Rigid Diaphragms* (on page 2128)

TR.28.2 Floor Diaphragm

This command is used to create rigid floor diaphragms without the need to specify a control joint at each. When specified, this command directs the engine to perform the following:

a. Calculate the center of mass for each rigid diaphragm (where control joint is to be located) considering the mass model of the structure. The mass must be modeled using mass reference load. See TR.31.6 Defining Reference Load Types (on page 2453)

b. Create, internally, an analytical node at the center of mass location to be included during analysis (unless a control node is specified) if an existing analytical node exists at this point, then the existing joint is used in lieu of creating a new joint.

Tip: The center of mass of each diaphragm is included in the preprocessing output. To include the center of rigidity in the post-processing output, you must include the `PRINT DIAPHRAGM CR` command. See TR.42 Print Specifications (on page 2666)

c. Search all nodes available within a diaphragm and add them as dependent nodes; with the control node located at the center of mass for the diaphragm (or at the specified control node)

General Format

```
FLOOR DIAPHRAGM
DIAPHRAGM i1 TYPE RIGID diaphragm-spec
...
( BASE b1 )
```

After all diaphragms are defined, the following optional commands can be added:

```
( CHECK SOFT STORY soft-story-code )
( CHECK IRREGULARITIES CODE irregularities-code )
```

where:

```
diaphragm-spec = HEIGHT f1 ( CONTROL i2 ) (JOINT joint-list )
```

or

```
diaphragm-spec = YRANGE f2 f3 ( CONTROL i2 ) ( JOINT XRANGE f4 f5 ZRANGE f6 f7 )
```

or

```
diaphragm-spec = ( CONTROL i2 ) JOINT joint-list
```

DIAPHRAGM *i1*	Diaphragm identification number
BASE *b1*	base/ground floor level of the structure when not at the minimum Y coordinate defined in the model
HEIGHT *f1*	Global coordinate value, in Y direction, to specify the floor level
YRANGE *f2 f3*	Global coordinate values to specify a Y range, where *f2* is the lower bound and *f3* is the upper bound. The diaphragm is considered to be located at that floor height.
XRANGE *f4 f5*	Global coordinate values to specify an X range. The diaphragm is considered to be located between this X range. If full floor is to be considered as only one diaphragm there is no need to define X range.
ZRANGE *f6 f7*	Global coordinate values to specify Z range. The diaphragm is considered to be located between this Z range. If full floor is to be considered as only one diaphragm there is no need to define Z range.
CONTROL *i2*	User-specified control joint number at the specified floor level. If not defined, the program will automatically calculate this joint as the diaphragm center of mass.

Instead of providing height or Y-range, joint lists can be provided to indicate the number of joints present at a particular floor level which will be connected to a control joint (either specified or calculated by the program).

Note: A single diaphragm may be defined at a given level in STAAD.Pro.

Soft Story and Irregularities Checks

Refer to TR.28.2.1 Soft Story Checking (on page 2331) for details on *soft-story-code* and to TR.28.2.2 Check Irregularities (on page 2333) for details on *irregularities-code*.

Notes

a. One full diaphragm definition should be provided per line. However, if there is joint-list, the list can extend to the second line with a continuation sign ("-").

```
DIA f1 TYP RIG YR f2 f3 JOI XR f4 f5 ZR f6 f7
DIA f11 TYP RIG YR f21 f31 JOI XR f41 f51 ZR f61 f71
DIA f12 TYP RIG JOI 35 45 51 TO 57 59 TO 83 -
90 TO 110
```

where *f1*, *f11*, and *f12* are three rigid diaphragms located at floor height ranging between *f2* and *f3*, *f21* and *f31*, and the joints lying in the plane as indicated by their global Y coordinates respectively.

b. Diaphragms should be specified in ascending order (i.e., diaphragms at first floor level should be specified first before specifying that on 2nd floor level and so on).

c. If a user-defined control joint is specified in one diaphragm, then user-defined control joints should be specified for *all* diaphragms. Combination of user-defined control joint for one diaphragm and program calculated control joint for another diaphragm is not supported.

d. The mass model (in terms of reference load) must be specified before specifying floor diaphragm .

e. Floor diaphragms can be specified only once in an input file.

f. Floor diaphragm cannot be specified along with the `FLOOR HEIGHT` command (refer to TR.31.2.9.1 Identification of Floor Level (on page 2392)).

g. Floor diaphragm cannot be specified along with the `CONTROL-DEPENDENT` command (refer to TR.28.1 Control/Dependent Specification (on page 2327)).

h. Floor diaphragm cannot be specified with the `SET Z UP` command.

i. Sloped diaphragms are not supported.

j. Base level (or ground floor level or support level) is taken as the minimum of Y coordinates defined. Different base level can be specified using the `BASE` *b1* option in the command. If used, this option must be the last line of the floor diaphragm system.

k. The maximum number of diaphragms allowed by the program (default value) is 150. If more than 150 diaphragms need to be specified, then `SET RIGID DIAPHRAGM` *n* must be specified before specifying joint incidence, where n = total number of diaphragms in the structure.

Example

```
***********************************************************
UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
* MASS MODEL
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 2.5 FY 2.5 FZ 2.5
49 TO 64 FX 1.25 FY 1.25 FZ 1.25
*
LOAD R2
SELFWEIGHT Y -1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEF REF LOAD
```

```
*****************************************************************
FLOOR DIAPHRAGM
DIAPHRAGM 1 TYPE RIGID YR 4.1 4.3 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 2 TYPE RIGID YR 8.3 8.5 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 3 TYPE RIGID YR 12.5 12.7 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 4 TYPE RIGID YR 16.7 16.9 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 5 TYPE RIGID YR 20.9 21.1 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 6 TYPE RIGID YR 25.1 25.3 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 7 TYPE RIGID YR 29.3 29.5 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 8 TYPE RIGID YR 33.5 33.7 JOI XR -0.1 21.3 ZR -0.1 31.9
DIAPHRAGM 9 TYPE RIGID YR 37.7 37.9 JOI XR -0.1 21.3 ZR -0.1 31.9
BASE 0.5
************************************************************
```

Related Links

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

TR.31.2.16 IBC 2015 Seismic Load Definition

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *M. To assign nodes to a floor diaphragm* (on page 818)
- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *G.17.3.2 Mass Modeling* (on page 2157)

TR.28.2.1 Soft Story Checking

STAAD.Pro will perform this check when the option `CHECK SOFT STORY` command is used following a floor diaphragm. If omitted, no soft story check is performed. Soft story checking is only valid for structures having vertical elements in the form of columns and shear wall (without opening). When used, the program will exclude the effect of any other forms of lateral force resisting structural elements.

General Format

Note: This command is inserted after any pre-analysis `PRINT` commands.

```
CHECK SOFT STORY { 1893 | 1893 2016 | ASCE7 }
```

Tip: Include the `PRINT STORY STIFFNESS` command in the post-analysis print commands in order to review the story stiffness values.

Description

The option check for soft story can be performed per the IS 1893:2002, IS 1893-2016, or ASCE 7-95 codes.

A soft story building is a multi-story building where one or more floors are soft due to structural design. These floors can be dangerous in earthquakes, because they cannot cope with the lateral forces caused by the swaying of the building during a quake. As a result, the soft story may fail, causing what is known as a soft story collapse.

Soft story buildings are characterized by having a story which has a lot of open space. Parking garages, for example, are often soft stories, as are large retail spaces or floors with a lot of windows. While the unobstructed space of the soft story might be aesthetically or commercially desirable, it also means that there are less opportunities to install shear walls, specialized walls which are designed to distribute lateral forces so that a building can cope with the swaying characteristic of an earthquake.

If a building has a floor which is 70% less stiff than the floor above it, it is considered a soft story building. This soft story creates a major weak point in an earthquake, and since soft stories are mostly associated with retail spaces and parking garages, they are often on the lower stories of a building, which means that when they collapse, they can take the whole building down with them, causing serious structural damage which may render the structure totally unusable.

When used, the program checks for soft stories per Clause 7.1 Table 5 i)a and i)b for IS 1893 and per Clause 12.3.3 Table 12.3-2 1a and 1b for ASCE 7-05 (used for seismic design categories D, E, and F).

Stiffness Irregularities

- **Stiffness Irregularities: Soft Story** – As per this provision of the code, a soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 80 percent of the average lateral stiffness of the three story above.
- **Stiffness Irregularities: Extreme Soft Story** – As per this provision of the code, a extreme soft story is one in which the lateral stiffness is less than 60 percent of that in the story above or less than 70 percent of the average lateral stiffness of the three story above.

  Thus, if any story of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

Example

```
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
CHECK SOFT STORY 1893 2016
```

Example Output

```
SOFT STORY CHECK

   VERTICAL STRUCTURAL IRREGULARITIES : SOFT STORY CHECK - IS 1893:2016
   -------- ---------- --------------

   STORY           FL. LEVEL IN METE          S  T  A  T  U  S
   -----           -----------------        --------------------
                                            X                  Z

     1                   3.00               OK                 OK
     2                   6.00               OK                 OK
     3                   9.00               OK                 OK

   NOTE : NO SOFT STOREY IS DETECTED.
```

Related Links

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

TR.31.2.16 IBC 2015 Seismic Load Definition

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

TR.28.2.2 Check Irregularities

STAAD.Pro can check irregularities per the IS 1893 2016 and ASCE 7-05/10/16 seismic codes.

For IS 1893 2016, the program can check horizontal irregularities (torsional and reentrant corners) per Table 5 and vertical irregularities (mass irregularities and irregular modes of oscillation) per Table 6.

For ASCE 7-05/10/16, the program can check horizontal irregularities (torsional and reentrant corners) and vertical irregulares (mass). Irregular modes of oscillation are not considered for this code.

General Format

Note: This command is inserted after any pre-analysis PRINT commands.

```
CHECK IRREGULARITIES CODE [ ASCE7 | IS1893 2016 ]
```

Note: Soft story checks per IS 1893 2016, Table 6(i) can also be performed by the program. Refer to TR.28.2.1 Soft Story Checking (on page 2331). For ASCE 7, soft story checking is performed per figure C12-3, Type-1.

Note: Only one code may be checked per irregularities in a single model.

Torsion Check per ASCE 7

Torsion checks per Cl. 12.3.2.1 of ASCE 7 2016 (ref. fig C12.3-1 Type 1) are preformed by applying unit loads in each orthogonal direction to the control node of the diaphragm.

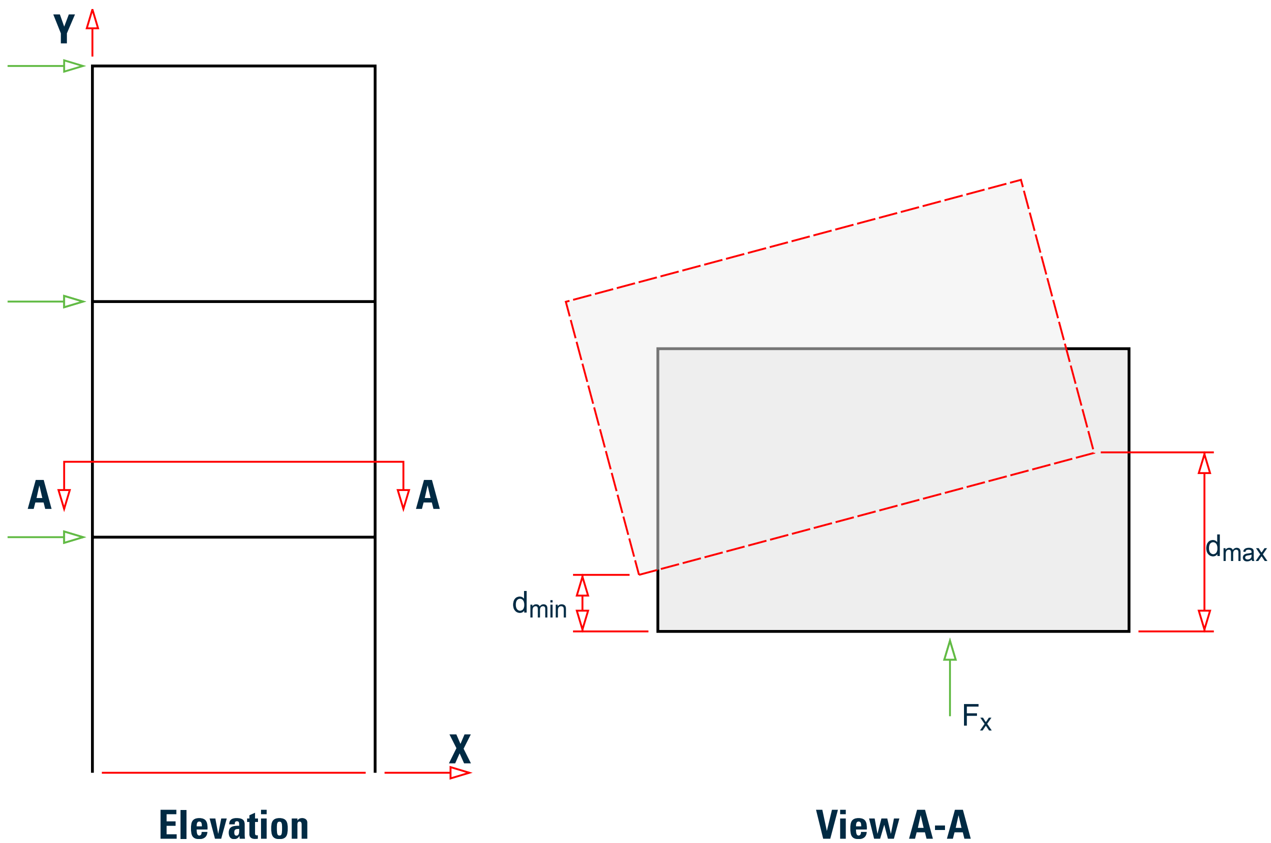


Figure 309: Example floor diaphragm torsion evaluation

The unit load, F_x, is applied at each diaphragm in the X-direction. After analysis, the program locates the extreme nodes of the diaphragm and then d_{min} and d_{max} are calculated. The ratio of d_{max}/d_{min} is evaluated and compared to the code limit of 1.5. This process is then repeated in the Z direction for this diaphragm. Then the similar process is applied to the other diaphragms in the structure.

Torsion Check per IS 1893

Torsion checks per Table 5(i)-a IS 1893 2016 are performed by applying a unit force in each orthogonal direction at two different distances (the design eccentricity, e_{di}) as per Cl. 7.8.2. This results in evaluating four different conditions for IS 1893 2016. The analysis is run and the displacements of the two extreme ends of the diaphragm are extracted from the analysis results. This ratio of these displacements is then reported with a status based on the following limits:

Ratio	Status	Description
$\Delta_{max}/\ \Delta_{avg} < 1.2$	OK	Indicates that the floor is regular and passes the irregularity checks.
$1.2 \leq \Delta_{max}/\ \Delta_{avg} \leq 1.4$	Warning	Indicates that the floor is irregular and requires a full 3D dynamic analysis in order to justify the structural configuration *or* requires a change in structural configuration as per Cl. 7.1, Table 5, Sl No. (i) sec-i.a and sec-i.b.

Ratio	Status	Description
$\Delta_{max}/\ \Delta_{avg} > 1.4$	Fails	Indicates that the floor is irregular and fails the irregularity checks. The structural arrangement must be changed as per Cl. 7.1, Table 5, Sl No. (i) sec-ii.

$$e_{di} = \begin{cases} 1.5e_{si} + 0.05b_i \\ e_{si} - 0.05b_i \end{cases} \qquad \text{(Design eccentricity per IS 1893 2016 Cl. 7.8.2)}$$

where

e_{si} = the static eccentricity at floor i, which is equal to the distance between the center of mass and the center of rigidity.

b_i = the floor plan dimension of floor i perpendicular to the direction of force

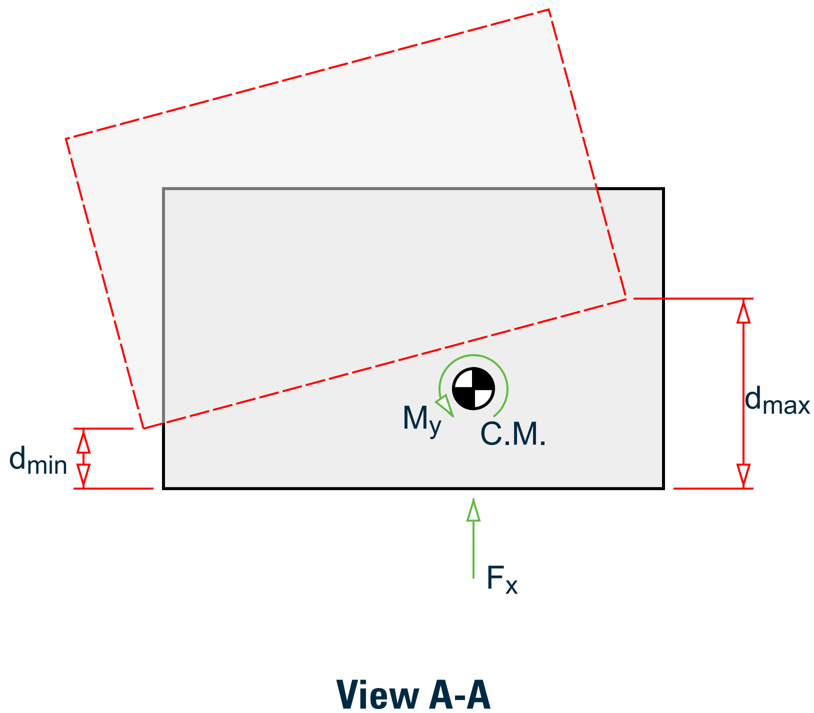


Figure 310: Example floor diaphragm torsion for IS 1893

The unit load, F_x is applied at each diaphragm in the X-direction in combination with two cases of $M_y = F_x \times (e_{di} - e_{si})$ representing the to cases for e_{di}. After analysis, the program locates the extreme nodes of the diaphragm and then Δ_{min}, Δ_{max}, and Δ_{avg} values are calculated. The ratio of $\Delta_{max}/\Delta_{avg}$ is evaluated per the table above. This process is then repeated in the Z direction for this diaphragm for a total of four cases at each diaphragm. Then the similar process is applied to the other diaphragms in the structure.

Note: Checks mandated by IS 1893 2016, Table 5(i)-b are *not* implemented as the prerequisite modelling information to determine the torsional mode is not present in STAAD.Pro.

Reentrant Corners

The program will check for reentrant corners per Table 5(ii) of IS 1893 2016 and Cl. 12.3.2.1 of ASCE 7 2016 (ref. fig C12.3-1 Type 2). The program first automatically identifies the boundary topology of the diaphragm. That is, the order in which analytical members and nodes of the boundary of the diaphragm are joined together to for a closed polygon. The program then determines the reentrant, or concave, corners of the diaphragm. The two analytical members joining the reentrant node is determined, their lengths are calculated, and then the projections of those lengths are calculated. To determine the ratios, the following are calculated:

$$L_i \times \cos(\alpha)/L_x \leq 0.15$$

$$L_i \times \cos(\alpha)/L_z \leq 0.15$$

If both members from the reentrant node are orthogonal, then for each member, only 1 ratio is calculated as L_i/L_x for members oriented along the x-direction and L_i/L_z for members oriented along the z-direction.

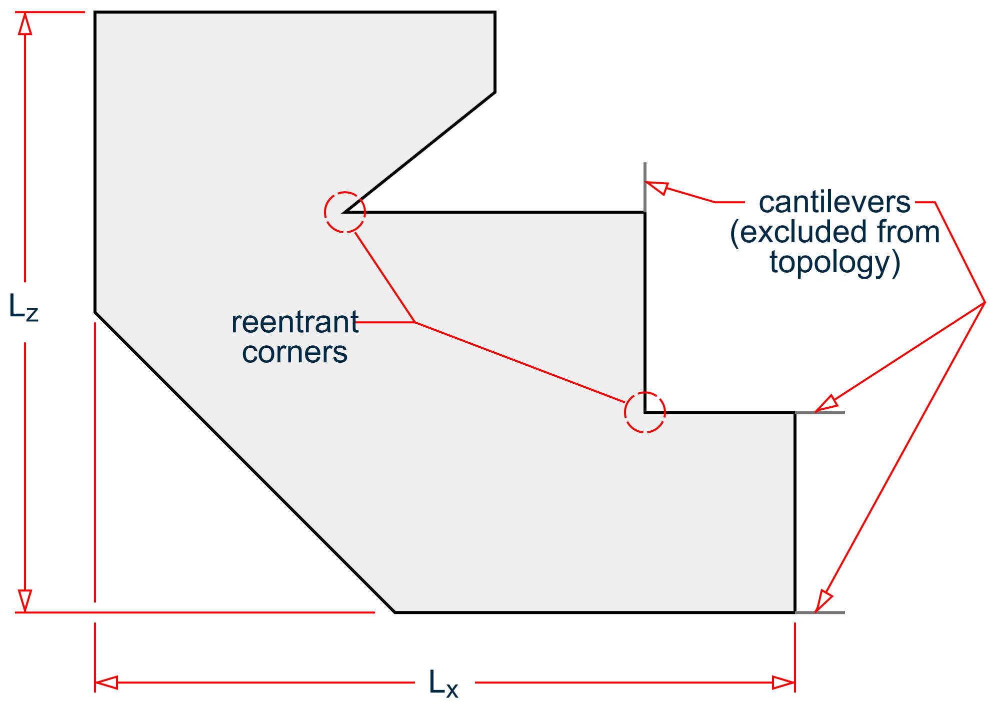


Figure 311: Example reentrant corner topology

Note: Cantilever members that do not form part a closed polygon are identified. These members are ignored by the program when evaluating reentrant corners.

All dependent nodes that form part of a diaphragm must lie within the same plane. Further, the boundary dependent nodes must form a closed polygon. Improper modeling of diaphragms may yield incorrect irregularity status results.

Note: Member offsets both in plan as well as in elevation may result in the reentrant corner check to be incorrect. You must exercise caution while using offsets and should check such joints manually.

Mass Irregularities

The program will check for mass irregularities per Table 6(ii) of IS 1893 2016 and Cl. 12.3.2.2 of ASCE 7 2016 (ref. fig C12.3-2 Type 2). The mass of each floor is calculated as the total of all floor diaphragm masses at the same level. The ratio of each floor mass to the floor above and below is calculated. These ratios are compared to the code stipulated values.

Irregular Modes of Oscillation

Note: The program will check for irregular modes of oscillation per the IS 1893 2016 code. This check is performed only if you have include the static seismic IS 1893 2016 definition using zone 4 or 5.

Per Table 6(vii)-a, the sum of the percentage of mass participation for the first 3 lateral translational modes should contribute at least 65% in each principle plan direction. This is checked in both orthogonal directions.

Per Table 6(vii)-b, the time period for the fundamental modes in one direction should differ by at least 10%.

Note: Irregular modes of oscillation are not considered the ASCE 7 codes.

IS 1893 2016 Example

An example using IS 1893 2016:

```
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
CHECK IRREGULARITIES CODE IS1893 2016
```

ASCE 7 Example

An example using ASCE 7:

```
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 0
DIA 2 TYPE RIG HEI 5
CHECK IRREGULARITIES CODE ASCE7
```

IS 1893 2016 Example Output

An example output section of an IS 1893 2016 seismic irregularities check:

```
-IRREGULARITY CHECKS

                    STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                    *********************************************************

                         Including Amendment no. 2 November 2020
                    *********************************************************

--TORSION IRREGULARITY CHECKS

 Torsion Irregularity Check
 Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
 --------------------------------------------------------------------
 edi : Design Eccentricity
```

```
esi : Static Eccentricity
bi  : Floor/Diaphragm plan dimension perpendicular to force direction
For Details Refer Clause 7.8 IS1893:2016-Part-1
---------------------------------------------------------------------


Using edi = 1.5esi + 0.05bi
---------------------------
Displacement of extreme points of diaphragm(dia.) in X dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      7    0.4984      2    0.4662    0.4823    1.0333     PASS
   2     12    2.4871      9    2.3990    2.4430    1.0180     PASS
   3     16    6.1637     13    6.0157    6.0897    1.0122     PASS

Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in X dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      2    0.4984      7    0.4662    0.4823    1.0333     PASS
   2      9    2.4871     12    2.3990    2.4430    1.0180     PASS
   3     13    6.1637     16    6.0157    6.0897    1.0122     PASS

Using edi = 1.5esi + 0.05bi
---------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      3    0.2699      2    0.2383    0.2541    1.0622     PASS
   2     10    1.0087      9    0.9347    0.9717    1.0381     PASS
   3     14    2.0463     13    1.9160    1.9812    1.0329     PASS

Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      3    0.3916      2    0.2319    0.3118    1.2562   WARNING*
   2     10    1.3253      9    0.9131    1.1192    1.1841     PASS
   3     14    2.5782     13    1.8732    2.2257    1.1584     PASS

  *** WARNING: The floor is irregular.  Please ensure conformance
   with Cl. 7.1, Table 5, Sl No. (i) sec-i.a or sec-i.b.
```

ASCE 7 Example Output

An example output section of an ASCE 7-2016 seismic irregularities check:

```
                   STAAD.PRO IRREGULARITIES CHECK - (  ASCE7-2016   )   v1.0
                   ****************************************************

--TORSION IRREGULARITY CHECKS

Torsion Irregularity Check
Ref: Fig. C12.3-1 T1- Ratio Limit(s): 1.20, 1.40
------------------------------------------------

Dia.   Extreme Points of Dia in X        Extreme Points of Dia in Z
      Node     Disp.     Node     Disp.     Node     Disp.     Node     Disp.
               (mm)               (mm)               (mm)                (mm)
------------------------------------------------------------------------
   1      3    0.09130      1    0.09993      4    0.09484      1    0.10559
   2     15    0.29636     13    0.30993     16    0.29819     13    0.34410

            Diaphragm ΔX-max/avg ΔZ-max/avg Status
            --------------------------------------
                  1     1.0451   1.0537        OK
                  2     1.0224   1.0715        OK
--GEOMETRY IRREGULARITY CHECKS

Re-Entrant Corner Check.
(Ref: Fig. C12.3-1 T2- Ratio Limit: 0.15 )
------------------------------------------

    Node      Re-Entrant X-Proj  X-Proj/Lx Z-Proj  Z-Proj/Lz  Status
 Connectivity     Node    ( m)              ( m)
----------------------------------------------------------------------
        6->         5      0.0000  0.0000    7.0000  0.7778 Re-Entrant
        4                  1.0000  0.2000    0.0000  0.0000
       18->        17      0.0000  0.0000    7.0000  0.7778 Re-Entrant
       16                  1.0000  0.2000    0.0000  0.0000

                  Diaphragm:    Lx:       Lz:
                                ( m)     ( m)
                  ----------------------------
                       1      5.0000    9.0000
                       2      5.0000    9.0000
--MASS IRREGULARITY CHECKS

Mass Irregularity Check
Ref: Fig. C12.3-2 T2- Ratio Limit: 1.50
---------------------------------------

Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     0.000    341.643    253.287     Base      1.349   N/A     OK
   2     5.000    253.287    Top         341.643    N/A   0.741    OK
```

Related Links

- *A. To check for soft stories and seismic code irregularities* (on page 962)

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *V. IS 1893 2016 GL Calculation* (on page 3509)
- *V. IS 1893 2016 Irregular Modes of Oscillation* (on page 3521)
- *V. IS 1893 2016 Mass Irregularity* (on page 3533)
- *V. IS 1893 2016 Re entrant Corners* (on page 3543)
- *V. IS 1893 2016 Response Spectrum Vertical* (on page 3560)
- *V. IS 1893 2016 Torsion Irregularity* (on page 3567)

TR.29 Definition of Member Attributes

Used to contain additional member attributes typically inserted from the STAAD.Pro User Interface.

General Format

Note: This section immediately follows `MEMBER RELEASE` data.

```
DEFINE MEMBER ATTRIBUTE
"attr-name " "attr-value " LIST member-list
...
END DEFINE
```

Note: The attribute name and attribute value are surrounded in quote mark (") characters.

Where

attr-name	The name of the attribute being used.
attr-value	The value of the current attribute. Values are specifically defined for each attribute name. Refer to the attribute types for explanations of appropriate values.
member-list	A list of member numbers.

Example

An example of some member attribute definitions in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"STRUCLINK" "BREAK_MEMBER_ACCROSS" LIST 6 8
"CONTAG" "START:SHEAR:SS" LIST 4
END DEFINE
```

TR.29.1 Struclink Member Attribute

Used to provide identify continuity which analytical model members in the STAAD input file should be made continuous physical members across a joint passing data through the Struclink utility.

Caution: Though it is possible to manually enter Struclink data in the STAAD input file, it is *strongly* recommended that the user interface be used to add this information.

General Format

Note: A `STRUCLINK` attribute must be inserted in the `DEFINE MEMBER ATTRIBUTE` section.

```
"STRUCLINK" "{ USE_DEFAULT | BREAK_MEMBER_ACROSS }" LIST member-list
```

Where

USE_DEFAULT	This attribute value indicates that two adjacent members which meet the requirements for merging into a physical member will do so when exported via Struclink.
BREAK_MEMBER_ACROSS	This attribute value indicates that two adjacent members will *not* be merged into a single physical member when exported via Struclink, even if they would meet the criteria to do so. This allows you to define where physical members should be broken at a joint. For example, where two identical beams frame into either side of a multi-story column.
member-list	A list of member numbers.

Example

An example of some physical member formation attributes for Struclink in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"STRUCLINK" "USE_DEFAULT" LIST 1 TO 5 7
"STRUCLINK" "BREAK_MEMBER_ACCROSS" LIST 6 8
END DEFINE
```

TR.29.2 Connection Tag Member Attribute

Used to assign members connection tag data from an associated Connection Tag XML file which is used for storing connection data and checking connection capacities.

Caution: Though it is possible to manually enter connection tag data in the STAAD input file, it is *strongly* recommended that the user interface be used to add this information as to ensure compatibility with the associated Connection Tags XML file. Refer to D. Connection Tags sub menu (on page 1066) for additional information on using this feature.

General Format

Note: A `CONTAG` attribute must be inserted in the `DEFINE MEMBER ATTRIBUTE` section.

The attribute value for the `CONTAG` attribute is delimited into three parts: the end of the member, the tag category, and the tag name to use for this tag assignment. The later two parts are used to look up the connection tag releases and capacities in the Connection Tag XML file.

```
"CONTAG" "{ START | END }:tag-category :tag-name " LIST member-list
```

Where:

tag-category One of the Category Names (`//Categories/Category/@CategoryName`) defined in the Connection Tag XML file.

tag-name One of the Tag Names (`//Tags/Tag/@TagName`) defined in the Connection Tag XML file.

member-list A list of member numbers.

Example

An example of a pair of connection tags in a STAAD.Pro input file:

```
DEFINE MEMBER ATTRIBUTE
"CONTAG" "START:MOMENT:EM" LIST 6
"CONTAG" "START:SHEAR:SS" LIST 4
END DEFINE
```

The corresponding Connection Tag XML file would have —as a minimum— the following data:

```
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile xmlns="http://tempuri.org/XMLSchema.xsd">
 <FileVersion value="1.0" />
  <Categories>
   <Category CategoryName="MOMENT">
    <CategoryDesc>End Moment Connection</CategoryDesc>
   </Category>
   <Category CategoryName="SHEAR">
    <CategoryDesc>Single Shear Connection</CategoryDesc>
   </Category>
  </Categories>
  <Equations>
   <Equation EquationID="Eq1" Equation="Abs([MZ])/[MZ.CAP]+Abs([FX])/
[FX.CAP]" Condition="LT" Limit="1.0" />
   <Equation EquationID="Eq2" Equation="Abs([FZ])/[FZ.CAP]" Condition="LT"
Limit="1.0" />
  </Equations>
  <Tags>
   <Tag TagName="EM" CategoryName="MOMENT">
    <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
    <Capacities UnitSystem="METRIC">
     <Beam Name="UB203x102x23">
      <BeamOrCol Name="UC152x152x23" Mz.cap="207.345" Fx.cap="35.915"
Fz.cap="205.095" />
     </Beam>
    </Capacities>
    <Checks>
     <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
     <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
    </Checks>
   </Tag>
   <Tag TagName="SS" CategoryName="SHEAR">
    <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
    <Capacities UnitSystem="METRIC">
     <Beam  Name="UB203x102x23">
      <BeamOrCol Name="UC152x152x23" Mz.cap="207.345" Fx.cap="35.915"
Fz.cap="205.095" />
     </Beam>
```

```
    </Capacities>
    <Checks>
     <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
    </Checks>
   </Tag>
  </Tags>
</ConnectionTagFile>
```

Note: Refer to D. Connection Tags XML File Schema (on page 1052) for additional information on the required structure of this XML file.

Related Links

- *G.9 Connection Tags* (on page 2125)
- *G.9 Connection Tags* (on page 2125)

TR.29.3 Member Type Attribute

Used to specify member types for connection design performed in Connection Design workflow.

General Format

```
"MEMBTYPE" "member-type" LIST member-list
```

Where:

member-type One of the following predefined member types:

- column
- primary beam
- brace
- rafter
- girt
- purlin
- eave-strut
- secondary-beam
- tertiary-beam
- chord
- branch

member-list A list of member numbers.

These member types are utilized by RAM Connection in order to identify members for connection design.

Example

```
DEFINE MEMBER ATTRIBUTE
"MEMBTYPE" "BRACE" LIST 14 15 20 21
END DEFINE
```

Related Links

- *D. To assign member type attributes* (on page 1031)

TR.30 Miscellaneous Settings for Dynamic Analysis

When dynamic analysis such as frequency and mode shape calculation, response spectrum analysis and time history analysis is performed, it involves eigenvalue extraction and usage of a certain number of modes during the analysis process. These operations are built around certain default values. This section explains the commands required to override those defaults.

Related Links

- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)

TR.30.1 Cut-Off Frequency, Mode Shapes, or Time

These commands are used in conjunction with dynamic analysis. They may be used to specify the highest frequency or the number of mode shapes that need to be considered.

General Format

```
CUT (OFF) { FREQUENCY f1 | MODE SHAPE i1 ( SHIFT MODE i2 ) (FREQ f2 )  | TIME t1 }
```

Where:

FREQUENCY *f1*	Highest frequency (cycle/sec) to be considered for dynamic analysis.
MODE SHAPE *i1*	Number of mode shapes to be considered for dynamic analysis. If the cut off frequency command is not provided, the cut off frequency will default to 108 cps. If the cut off mode shape command is not provided, the first six modes will be calculated. These commands should be provided prior to the loading specifications.
SHIFT MODE *i2*	Number of eigen vectors per shift. This input may be used only when the solver fails to extract *i1* number of modes due to insufficient memory.
FREQUENCY *f2*	Frequency (in Hz) to be used as initial shift in Arnoldi/Lanczos method (i.e., it is not applicable to the basic solver). This is an optional input. If provided, the solver looks for eigenvalues close to the shift. The eigen values found may not necessarily be the smallest eigenvalues (i.e., closest to zero). If a full scale eigen solution is required, this value should not be defined. The engine default value in 0.0 (zero)
TIME *t1*	Ending time for a time history analysis. If zero (default), the time history will end when the last forcing function ends.

A maximum of *i1* mode shapes will be computed regard-less of *f1*. If during convergence testing, the 0 through *f1* frequencies are converged, then the modal calculation will be completed before *i1* mode shapes are calculated.

A maximum of *i1* mode shapes will be computed regard-less of *f1*. If the `CUT OFF FREQ` *f1* and `CUT OFF MODE` *i1* commands are both entered, the program will report only those modes that lie within *f1* frequency.

Related Links

- *M. To add a SNiP wind load definition* (on page 847)
- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.30.2 Mode Selection

This command allows specification of a reduced set of active dynamic modes. All modes selected by this command remain selected until a new `MODE SELECT` is specified.

This command is used to limit the modes used in dynamic analysis to the modes listed in this command and deactivate all other modes that were calculated but not listed in this command. If this command is not entered, then all modes calculated are used in the dynamic analysis.

General Format

```
MODE SELECT mode-list
```

Example

```
CUT OFF MODES 10
MODE SELECT 1 TO 3
```

In this example, 10 modes will be calculated but only modes 1 and 3 will be used in dynamic analysis.

Notes

a. Do not enter this command within the loads data (from the first Load command in an analysis set down to the associated Analysis command).

b. The advantage of this command is that you may find the amount of structural response generated from a specific mode or a set of modes. For example, if 50 modes are extracted, but the effect of just the 40th to the 50th mode in a response spectrum analysis is to be determined, you may set the active modes to be 40 through 50. The results will then be devoid of any contribution from modes 1 through 39.

Related Links

- *G.17.3 Dynamic Analysis* (on page 2154)

TR.31 Definition of Load Systems

This section describes the specifications necessary for defining various load systems, for automatic generation of Moving loads, UBC Seismic loads and Wind loads. In addition, this section also describes the specification of Time History load for Time History analysis.

STAAD.Pro has built-in algorithms to generate moving loads, lateral seismic loads, and wind loads on a structure. Use of the load generation facility consists of two parts:

1. Definition of the load system(s).
2. Generation of primary load cases using previously defined load system(s).

Definition of the load systems must be provided before any primary load case is specified. This section describes the specification of load systems. Information on how to generate primary load cases using the defined load systems is available in TR.32.12 Generation of Loads (on page 2596).

Note: UBC loads do not fully consider the effects of forces at inclined support directions or at dependent joint directions. Applying forces at these locations may introduce errors that are generally small.

Related Links

- *G.16 Load Generator* (on page 2134)

TR.31.1 Definition of Moving Load System

This set of commands may be used to define the moving load system. Enter the DEFINE command only once with up to 100 TYPE commands.

The `MOVING LOAD` system may be defined in two possible ways: directly within the input file or using an external file.

General Format

```
DEFINE MOVING LOAD (FILE file-name)
TYPE j { LOAD f1 f2 … fn ( DISTANCE d1 d2 … dn-1 (WIDTH w) ) | load-name (f) }
( DISTANCE d1 d2 … dn-1 (WIDTH w) ) optionally as 2nd set
```

The FILE option should be used only in the second case when the data is to be read from an external file. The file name should be limited to 24 characters.

Note: Moving Loads can be generated for frame members only. They will not be generated for finite elements.

Note: All loads and distances are in current unit system.

Define Moving Load Within the Input File

Use the first TYPE specification. Input Data must be all in one line (as shown above) or two sets of lines. If two sets, then the second set must begin with DIS as shown above. If two sets, then Load and Dist lines may end each line but last of each set with a hyphen (See example).

```
TYPE j  LOAD f1 f2 … fn
DISTANCE d1 d2 … dn-1 (WIDTH w)
```

Where:

TYPE *j*	moving load system type number (integer limit of 200 types)
n	number of loads (eg, axles), 2 to 200.
LOAD *fi*	value of concentrated i^{th} load
DISTANCE *di*	distance between the $(i+1)^{th}$ load and the i^{th} load in the direction of movement
WIDTH *w*	spacing between loads perpendicular to the direction of movement (e.g., the width of vehicle). If left out, one dimensional loading is assumed. This parameter will double the total load since the f_i is applied to each wheel.

Note: For a single moving load use: `TYPE j LOAD f1 DIST 0`

Define Moving Load Using an External File

Use the second TYPE specification.

```
TYPE j load-name (f)
```

Where:

TYPE *j*	moving load system type no. (integer).
load-name	the name of the moving load system (maximum of 24 characters).
f	Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)

Following is a typical file containing the data.

```
CS200
50. 80. 90. 100.
7. 7. 9.
6.5
```

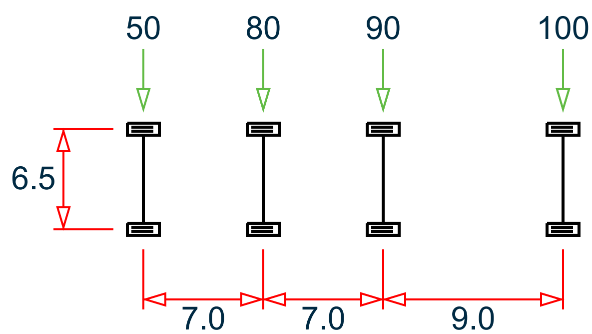


Figure 312: Graphical representation of the previous load system

Several load systems may be repeated within the same file.

The STAAD moving load generator assumes:

1. All positive loads are acting in the negative global vertical (Y or Z) direction. The user is advised to set up the structure model accordingly.
2. Resultant direction of movement is determined from the X and Z (or Y if Z is up)increments of movements as provided by the user.

Reference Load

The first specified concentrated load in the moving load system is designated as the reference load. While generating subsequent primary load cases, the initial position of the load system and the direction of movement are defined with respect to the reference load location. Also, when selecting the reference load location with a positive value of Width specified, then the following two views define the reference load location.

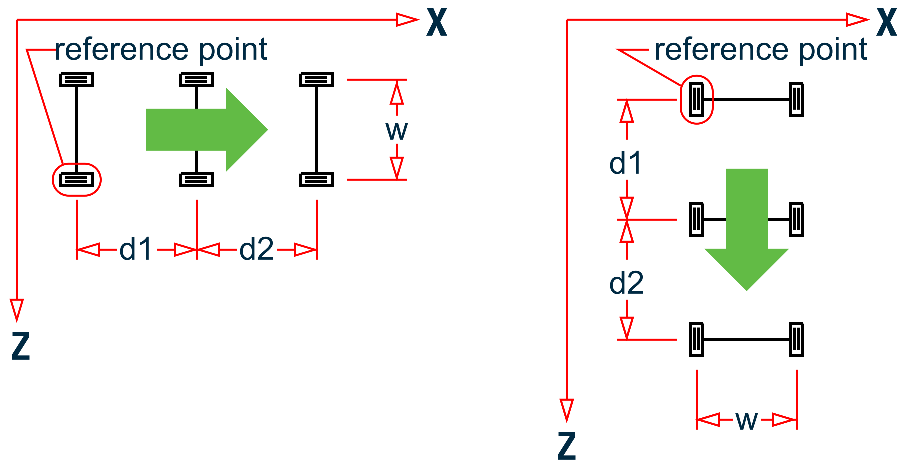


Figure 313: Movement parallel to global X axis; Movement parallel to global Z axis

Notice that in the left view, the reference point is on the positive Z wheel track side; whereas in the right view, the reference point is on the least positive X wheel track side.

Specifying Standard AASHTO Loadings

Truck loads specified in AASHTO specifications are also built in to STAAD.

```
TYPE j { HS20 | HS15 | H20 | H15 } (f) ( vs )
```

Where:

TYPE *j*	moving load system type no. (integer).
f	Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)
vs	variable spacing as defined by AASHTO, for HS series trucks (default = 14 ft.)

Example 1

```
DEFINE MOVING LOAD
TYPE 1 LOAD 10.0 20.0 -
15.0 10.0
DISTANCE 5.0 7.5 -
6.5 WIDTH 6.0
TYPE 2 HS20 0.80 22.0
```

Example 2

When data is provided through an external file called `MOVLOAD`

Data in STAAD input file

```
UNIT …
DEFINE MOVING LOAD FILE MOVLOAD
TYPE 1 AXLTYP1
TYPE 2 AXLTYP2 1.25
```

Data in external file MOVLOAD

```
AXLTYP1
10 20 15
5.0 7.5
6.0
AXLTYP2
20 20
10
7.5
```

Related Links

- *G.16.1 Moving Load Generator* (on page 2134)
- *M. To define a vehicle for loading* (on page 873)
- *G.16.1 Moving Load Generator* (on page 2134)
- *M. Moving Loads* (on page 873)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 6398)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 6684)
- *V. Moving Load Generator* (on page 3597)

TR.31.2 Definitions for Static Force Procedures for Seismic Analysis

STAAD.Pro offers facilities for determining the lateral loads acting on structures due to seismic forces, using the rules available in several national codes and widely accepted publications. The codes and publications allow for so called equivalent static force methods to be used in place of more complex methods like response spectrum and time history analysis.

Once the lateral loads are generated, the program can then analyze the structure for those loads using the applicable rules explained in the code documents.

Table 239: Codes available in STAAD.Pro with Seismic loads

Country	Code	Title
Algeria	TR.31.2.1 RPA (Algerian) Seismic Load (on page 2353)	*Règles Parasismiques Algériennes*
Canada	TR.31.2.2 Canadian Seismic Code (NRC) - 1995 (on page 2355)	National Building Code (NRC/ CNRC) of Canada
	TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1 (on page 2358)	National Building Code (NRC/ CNRC) of Canada
	TR.31.2.4 Canadian Seismic Code (NRC) - 2010 (on page 2362)	National Building Code (NRC/ CNRC) of Canada
China	TR.31.2.5 Chinese Static Seismic per GB50011-2001 (on page 2369)	*Code for Seismic Design of Buildings* GB50011-2001

Country	Code	Title
	TR.31.2.6 Chinese Static Seismic per GB50011-2010 (on page 2375)	*Code for Seismic Design of Buildings* GB50011-2010 (2016 Edition)
Colombia	TR.31.2.7 Colombian NSR-98 Seismic Load (on page 2382)	*Reglamento Colombiano de Construcción Sismo Resistente* (NSR-98), *Normas Colombianas de Diseño y Construcción*, 1998, Asociación Colombiana de Ingeniería Sísmica
	TR.31.2.8 Colombian NSR-10 Seismic Load (on page 2384)	NSR-10 *Reglamento Colombiano Sismo Resistente*
India	TR.31.2.9 IS:1893 - 1984 Code - Lateral Seismic Load (on page 2386)	Criteria for Earthquake Resistant Design of Structures
	TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)	Criteria for Earthquake Resistant Design of Structures - Part 1 : General Provisions and Buildings
	TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)	Criteria for Earthquake Resistant Design of Structures - Part 4 : Industrial Structures Including Stack-Like Structures
	TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393)	Criteria for Earthquake Resistant Design of Structures - Part 1 : General Provisions and Buildings
	TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load (on page 2398)	Criteria for Earthquake Resistant Design of Structures Part 4 Industrial Structures Including Stack-Like Structures
Japan	TR.31.2.18 Japanese Seismic Load (on page 2418)	Building Codes Enforcement Ordinance 2006
Mexico	TR.31.2.19 CFE (Comisión Federal De Electricidad) Seismic Load (on page 2420)	*Manual de Diseño por Sismo* - Comisión Federal de Electricidad (*Seismic Design Handbook* - Electric Power Federal Commission)
	TR.31.2.20 NTC (Normas Técnicas Complementarias) Seismic Load (on page 2423)	Reglamento de Construcciones del Distrito Federal de México (Mexico Federal District)

Country	Code	Title
Turkey	TR.31.2.21 Turkish Seismic Code (on page 2426)	"Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention" Amended on 2.7.1998, Official Gazette No. 23390
US	TR.31.2.22 UBC 1994 or 1985 Load Definition (on page 2429)	Uniform Building Code, 1985 edition
	TR.31.2.22 UBC 1994 or 1985 Load Definition (on page 2429)	Uniform Building Code, 1994 edition
	TR.31.2.23 UBC 1997 Load Definition (on page 2432)	Uniform Building Code, 1997 edition
	TR.31.2.13 IBC 2000/2003 Load Definition (on page 2401)	International Building Code, 2000 & 2003 editions
	TR.31.2.14 IBC 2006/2009 Seismic Load Definition (on page 2405)	International Building Code, 2006 & 2009 editions
	TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409)	International Building Code, 2012 edition
	TR.31.2.16 IBC 2015 Seismic Load Definition (on page 2413)	International Building Code, 2015 edition
	TR.31.2.17 IBC 2018 Seismic Load Definition (on page 2416)	International Building Code, 2018 edition

Common Weight Data

```
SELFWEIGHT
JOINT WEIGHT
joint-list
WEIGHT w
MEMBER WEIGHT
mem-list { UNI v1 v2 v3 | CON
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
floor-weight-spec
ONEWAY WEIGHT
oneway-weight-spec
REFERENCE LOAD { X | Y | Z }
Ri1 f11
```

floor-weight-spec See TR.32.4.3 Floor Load Specification (on page 2483) for floor weight specification.

oneway-weight-spec See TR.32.4.2 One-way Load Specification (on page 2476) for One-way load specification.

Note: The weight definitions must be in the order specified above. That is, selfweight, joint weight, member weight, element weight, and then floor weights. If one or more is not present, it can be skipped as long as the general order is preserved.

WEIGHT *w*	The joint weight associated with list
UNI *v1 v2 v3*	Used when specifying a uniformly distributed load with a value of *v1* starting at a distance of *v2* from the start of the member and ending at a distance of *v3* from the start of the member. If *v2*and v $_3$ are omitted, the load is assumed to cover the entire length of the member.
CON *v4 v5*	Used when specifying a concentrated force with a value of *v4* applied at a distance of *v5* from the start of the member. If *v5* is omitted, the load is assumed to act at the center of the member.
PRESS *p1*	The weight per unit area for the plates selected. Assumed to be uniform over the entire plate. Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.
Ri1	Identification number of a previously defined reference load case. See TR.31.6 Defining Reference Load Types (on page 2453)
f11	Magnification factor (required for reference loads).

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which you would use `FLOOR LOADS` (See TR.32.4.3 Floor Load Specification (on page 2483) for details). Similarly, you can use the Oneway Weight command to specify a load path direction for the pressure on a region.

Note: If both mass table data (`SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` options) and a `REFERENCE LOAD` are specified, these will be added algebraically for a combined mass.

Wall Area Definitions

Wall width and length data for the first story of the structure must be specified in order to calcluate the natural period of the structure per TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393) when `ST 4` (i.e., reinforced concrete buildings with structural walls).

```
WALL AREA { X | Z }
wall-data-pairs
```

where

```
wall-data-pairs = w1, l1; w2, l2; w3, l3, …; wn, ln;
```

w1, l1; w2, l2; w3, l3, ...; wn, ln	Used to specify wall dimensions for calculating effective cross section area of wall in the first story of the building. These should specify the walls which resist seismic force along the global direction (X or Z) of the seismic load only. *w* is the width of the wall and *l* is the length of the wall.

Related Links

- *M. To add a seismic load definition* (on page 857)
- *TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load* (on page 2393)
- *Add New Seismic Definitions dialog* (on page 2895)
- *IS:1893 Seismic Parameters dialog box* (on page 2898)
- *M. To add wall data area to an IS1893 2016 seismic definition* (on page 858)

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.1 RPA (Algerian) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per RPA specifications using a static equivalent approach similar to those outlined by RPA. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE RPA (ACCIDENTAL) LOAD
```

```
 rpa-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
rpa-spec = { A f1 Q f2 RX f3 RZ f4 STYPE f5 CT f6 CRDAMP f7 (PX f8) (PZ f9) }
```

A *f1* Seismic zone coefficient. Use a fractional value such as 0.08, 0.15, 0.2, 0.3, 0.05, etc., instead of an integer value.

Q *f2* Importance factor

RX *f3* Coefficient R for lateral load in X direction – table 4.3

RZ *f4* Coefficient R for lateral load in Z direction – table 4.3

STYPE *f5* Soil Profile Type

CT *f6* Coefficient from table 4.6 of RPA 99

CRDAMP *f7* Critical damping factor

PX *f8* Optional Period of structure (in sec) in X direction

PZ *f9* Optional Period of structure (in sec) in Z direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh method.

Used for Y if the `SET Z UP` command is used.

Generation of RPA Seismic Load

General format to provide RPA Seismic load in any load case:

```
LOAD i
```

```
RPA LOAD {X | Y | Z} (f10) (ACC f11)
```

Where:

LOAD *i*	the load case number
RPA LOAD {X | Y | Z} *f10*	factor to multiply horizontal seismic load.
ACC *f11*	multiplying factor for Accidental Torsion, to be used to multiply the RPA accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Note: If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the RPA specifications. The value of the accidental torsion is based on the "center of mass" for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`s and `MEMBER WEIGHT`s you have specified.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The design base shear is computed in accordance with section 4.2.3 of the RPA 99 code. The primary equation, namely 4-1, as shown below, is checked.

$$V = (A\ D\ Q)W / R$$

Where:

W = total weight on the structure
A = zone coefficient
D = average dynamic amplification factor
R = lateral R factor
Q = structural quality factor

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE RPA LOAD` command.

Program calculates the natural period of building T utilizing clause 4.2.4 of RPA 99.

Design spectral coefficient (D) is calculated utilizing T as,

$$D = 2.5\eta \text{ when } 0 \leq T \leq T_2$$

$$= 2.5\eta(T_2/T)^{2/3} \text{ when } T_2 \leq T \leq 0.3 \text{ sec.}$$

$$= 2.5\eta(T_2/3)^{2/3}(3/T)^{5/3} \text{ when } T > 0.3 \text{ sec.}$$

Where:

η = factor of damping adjustment (Eq. 4.3)
T_2 = specific period (Table 4.7)

Total lateral seismic load, V is distributed by the program among different levels.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE RPA LOAD command.

Example

```
DEFINE RPA LOAD
A 0.15 Q 1.36 STYP 2 RX 3 RZ 4 CT 0.0032 -
```

```
CRDAMP 30 PX .027 PZ 0.025
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
RPA LOAD X 1.0
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.2 Canadian Seismic Code (NRC) - 1995

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 1995 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE NRC LOAD` command.

```
DEFINE NRC LOAD
```

```
nrc-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
nrc-spec = *{ V f1 ZA f2 ZV f3 RX f4 RZ f5 I f6 F f7 (CT f8) (PX f9) (PZ f10) }
```

V ***f1***	Zonal velocity ratio per Appendix C
ZA ***f2***	Factor for acceleration related seismic zone per Appendix C
ZV ***f3***	Factor for velocity related seismic zone per Appendix C
RX ***f4***	Force modification factor along X-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B
RZ ***f5***	Force modification factor along Z-direction that reflects the capability of a structure to dissipate energy through inelastic behavior. Please refer Table 4.1.9.1B
I ***f6***	Seismic importance factor per sentence 10 section 4.1.9.1
F ***f7***	Foundation factor conforming to Table 4.1.9.1C and sentence 11 section 4.1.9.1
CT ***f8***	Factor to be used to calculate the fundamental period of structure .This is an optional parameter.
PX ***f9***	Period of structure (in seconds) in the X- direction. This is an optional parameter.

PZ *f10* Period of structure (in seconds) in the Z- direction. This is an optional parameter.

Used for Y if the `SET Z UP` command is used.

w joint weight associated with list

v1*, *v2*, *v3 Used when specifying a uniformly distributed load with a value of *v1* starting at a distance of *v2* from the start of the member and ending at a distance of v_3 from the start of the member. If *v2*and v_3 are omitted, the load is assumed to cover the entire length of the member.

v4*, *v5 Used when specifying a concentrated force with a value of *v4* applied at a distance of *v5* from the start of the member. If *v5* is omitted, the load is assumed to act at the center of the member.

p1 weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see TR.32.4 Area, One-way, and Floor Load Specifications (on page 2475) for details of the Floor Load input).

The weights have to be input in the order shown.

Generation of NRC Load

The load so defined as above is applied on the structure in the NRC loadcases. These loadcases have to be the first loadcases in the input file. Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications.

The following general format should be used to generate loads in a particular direction.

```
LOAD i
```

```
NRC LOAD { X | Y | Z } (f1)
```

Where:

LOAD *i*	load case number
NRC LOAD { X | Y | Z } *f1*	factor to be used to multiply the NRC Load (default = 1.0). May be negative.

Notes

1. By providing either PX or PZ or both, you may override the period calculated by STAAD using Rayleigh method. If you do not define PX or PZ, the period for Method 2(b) above will be calculated by the program using Rayleigh method and the stipulations of sentence 7(c) of section 4.1.9.1
2. Some of the items in the output for the NRC analysis are explained below.

   T_a = Time period calculated per sentence 7(a) or 7(b) of section 4.1.9.1
   T_c = Time period calculated per sentence 7(c) of section 4.1.9.1

   CALC / USED PERIOD

   The CALC PERIOD is the period calculated using the Rayleigh method. For NRC in the x-direction, the USED PERIOD is PX. For the NRC in the z-direction (or Y direction if SET Z UP is used), the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one utilized to find out the value of S.

3. In the analysis for NRC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD using the appropriate equation(s); namely sentence 4, section 4.1.9.1 of NRC.

$$V = 0.6 \cdot V_e/R$$

Where:

V_e ,the equivalent lateral seismic force representing elastic response (per sentence 5,section 4.1.9.1) is given by:

$$V_e = v \cdot S \cdot I \cdot W$$

Where:

v = Zonal velocity ratio per appendix C
S = Seismic Response Factor per table 4.9.1.A
I = Seismic importance factor per sentence 10 section 4.1.9.1
F = Foundation factor conforming to Table 4.9.1.C and sentence 11 section 4.1.9.1
W = Total load lumped as weight per sentence 2 section 4.1.9.1
R = Force modification factor conforming to Table 4.9.1.B that reflects the capability of a structure to dissipate energy through inelastic behavior.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. User provides seismic zone co-efficient and desired "nrc-spec" (1995) through the DEFINE NRC LOAD command.
2. The program calculates the fundamental period(T) of the structure by
   a. finding out whether the structure being analysed is a moment resisting frame made primarily of steel or of concrete or it is a structure of any other type. Alternatively, the software uses the optional parameter CT if provided. The calculation is done per sentence 7(a) & 7(b) of section 4.1.9.1.
   b. using the Rayleigh method or using the optional parameters PX , PZ –if provided. The stipulations of sentence 7(c) of section 4.1.9.1 are also considered while calculating.
   c. taking the conservative value of T between those calculated by methods (a) and (b) above.
3. The program finds out the value of Seismic Response Factor(S) per table 4.9.1.A utilizing the values of T as calculated above and the values of ZA & ZV input by the user.
4. The program calculates V per sentence 4 section 4.1.9.1. W is obtained from the weight data (SELFWEIGHT, JOINT WEIGHT(s), etc.) provided by the user through the DEFINE NRC LOAD command. The weight data must be in the order shown.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per applicable NRC guidelines like sentence 13(a) section 4.1.9.1.

Example

```
DEFINE NRC LOAD
V 0.2 ZA 4 ZV 4 RX 4 RZ 4 I 1.3 F 1.3 CT 0.35 PX 2 PZ 2
SELFWEIGHT
JOINT WEIGHT
```

```
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

See also Example Problem `Examp14_NRC.std`

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 2005 Volume 1 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

STAAD.Pro utilizes the following format to generate the lateral seismic loads.

```
DEFINE NRC 2005 (ACCIDENTAL) LOAD
```

```
nrc-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
nrc-spec = { SA1 f1 SA2 f2 SA3 f3 SA4 f4 MVX f5 MVZ f6 JX f7 JZ f8 IE f9 RDX f10 ROX
f11 RDZ f12 ROZ f13 SCLASS f14 ( FA f15 ) ( FV f16 ) ( CT f17 ) ( PX f18 ) ( PZ f19) }
```

where:

SA1 *f1*	Seismic Data, Sa(0.2), as per Table C-2.
SA2 *f2*	Seismic Data, Sa(0.5), as per Table C-2.
SA3 *f3*	Seismic Data, Sa(1.0), as per Table C-2.
SA4 *f4*	Seismic Data, Sa(2), as per Table C-2.
MVX *f5*	The higher mode factor along the X direction. Refer NRC Table 4.1.8.11.
MVZ *f6*	The higher mode factor along the Z direction. Refer NRC Table 4.1.8.11.
JX *f7*	The numerical reduction coefficient for base overturning moment along the X direction. Refer to NRC Table 4.1.8.11.
JZ *f8*	The numerical reduction coefficient for base overturning moment along the Z direction. Refer to NRC Table 4.1.8.11.

IE *f9* The earthquake importance factor of the structure. This input is based on Importance Category and ULS / SLS. Refer to NRC Table 4.1.8.11.

RDX *f10* The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

ROX *f11* The over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

RDZ *f12* The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

ROZ *f13* The over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

SCLASS *f14* an integer corresponding to site classes A through E(where 1 = A and 6 = F). F_a and F_v are determined based on Site Class as per NRC Table 4.1.8.4.B and Table 4.1.8.4.C.

FA *f15* Optional Short-Period site coefficient at 0.2s. Value must be provided if `SCLASS` set to F (i.e., *f14* = 6).

FV *f16* Optional Long-Period site coefficient at 1.0s. Value must be provided if `SCLASS` set to F (i.e., *f14* = 6).

CT *f17* Optional CT value used in calculating the period of the structure based on the empirical method.

> **Note:** The CT parameter entered is used directly in the calculation of the fundamental period, Ta. Therefore, care should be taken to convert the CT value to imperial units as the program internally uses the height of the building in feet for this calculation.

PX *f18* Optional period of the structure (in sec) in the X direction, to be used as fundamental period of the structure. If not entered the value is calculated from the code.

PZ *f19* Optional period of the structure (in sec) in the Z direction, to be used as fundamental period of the structure. If not entered the value is calculated from the code.

If the `ACCIDENTAL` option is specified, the program calculates the accidental torsion per the NRC 2005 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands.

The `ACC` option along with accidental eccentricity factor (default 0.1 as per NRC 2005) needs to be provided in the NRC seismic primary load case (i.e., `NRC LOAD X / Z f1 ACC f3`). f3 can be negative.

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the `ACCIDENTAL` option should *not* be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the NRC seismic primary load case (i.e., `NRC LOAD X / Z f1 DEC f2 ACC f3`). For equivalent seismic analysis, *f2* is 1 and *f3* is 0.1 as per NRC 2005 code. *f1* is always positive or zero, however *f2* can be negative. If *f2* is 0.0, only accidental torsion will be considered for this particular load case.

> **Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD.Pro using the appropriate equation (s).

$$V = \frac{S(T_a) M_v I_E W}{R_d R_o}$$

as per section 4.1.8.11(2) of NBC of Canada 2005, Volume 1

Except that V shall not be less than:

$$V_{\min} = \frac{S(2.0) M_v I_E W}{R_d R_o}$$

and for an R_d= 1.5, V need not be greater than:

$$V_{\max} = \frac{2}{3} \frac{S(0.2) I_E W}{R_d R_o}$$

(i.e., the upper limit of V)

Description of the terms of the equation to calculate V:

- T_a is the fundamental lateral period in the direction under consideration and is determined as:
  - **a.** For moment-resisting frames that resist 100% of the required lateral forces and where the frame is not enclosed by or adjoined by more rigid elements that would tend to prevent the frame from resisting lateral forces, and is calculated by the empirical formulae as described below provided h_n is in meter
    - **i.** $0.085(h_n)^{3/4}$ for steel moment frames,
    - **ii.** $0.075(h_n)^{3/4}$ for concrete moment frames, or
  - **b.** The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

    If design spectral acceleration, $S(T_a)$,calculated considering structural time period calculated based on method (b) is greater than 0.8 time the same calculated considering structural time period calculated based on method (a), the former is used for further calculation. Otherwise, the later time period is used.
  - **c.** Other established methods of mechanics using a structural model that complies with the requirements of Sentence 4.1.8.3.(8), except that
    - **i.** for moment resisting frames, Ta shall not be greater than 1.5 times that determined in Clause (a).
    - **ii.** for braced frames, Ta shall not be greater than 2.0 times that determined in Clause (b).
    - **iii.** for shear wall structures, Ta shall not be greater than 2.0 times that determined in Clause (c), and
    - **iv.** for the purpose of calculating the deflections, the period without the upper limit specified is referred from Appendix A.
- $S(T_a)$ is the design spectral acceleration and is determined as follows, using linear interpolation for intermediate values of T_a:

$$S(T_a) = F_a S_a(0.2) \text{ for } T_a \le 0.2\text{s}$$

= $F_v S_a(0.5)$ or $F_a S_a(0.2)$, whichever is smaller for T_a = 0.5s
= $F_v S_a(1.0)$ for T_a = 1.0s
= $F_v S_a(2.0)$ for T_a = 2.0s
= $F_v S_a(2.0)/2$ for $T_a \ge$ 4.0s

The above terms $S_a(0.2)$, $S_a(0.5)$, $S_a(1.0)$ and $S_a(2.0)$ are the Seismic Data and are obtained as user input from the *Table C-2.*

Based on the above values of $S_a(T_a)$, F_a and$F_{v,}$ the acceleration- and velocity based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of $S_a(0.2)$ and $S_a(1.0)$. It is to be mentioned that, these are the user inputs based on the site classes from A to E and the desired $S_a(0.2)$ and $S_a(1.0)$ values as required as per the above equations.

- M_v is the factor to account for higher mode effect on base shear and the associated base overturning moment reduction factor is J which are obtained as user input from the Table 4.1.8.11. To get this higher mode factor (M_v) and numerical reduction coefficient for base overturning moment (J), you must get the ratios of $S_a(0.2)/S_a(2.0)$ as also the "Type of Lateral Resisting System."

  For values of Mv between fundamental lateral periods, Ta of 1.0 and 2.0 s, the product S(Ta). Mv shall be obtained by linear interpolation.

  Values of J between fundamental lateral periods, Ta of 0.5 and 2.0 s shall be obtained by linear interpolation.
- I_E is the earthquake importance factor of the structure and is determined from the Table 4.1.8.5. This is a user input depending on Importance Category and ULS / SLS
- W is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^{n} W_i$$

  Where W_i is the portion of W that is located at or assigned to level i.
- R_d is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9.
- R_o is the over-strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9.

  These R^d and R_o values are the user inputs depending on the type of SFRS

As per 4.1.8.11(6), the total lateral seismic force, V, shall be distributed such that a portion, F_t shall be concentrated at the top of the building, where,

$F_t = 0.07T_aV$

but F_t is not greater than 0.25V and $F_t = 0$ when T_a is not greater than 0.7s.

The remainder (V- F_t), shall be distributed along the height of the building, including the top level, in accordance with the following formula [as per section 4.1.8.11(6)]:

$$F_x = \frac{(V - F_t)W_x h_x}{\sum_{i=1}^{n} W_i h_i}$$

where

F_x	=	the lateral force applied to level x
F_t	=	the portion of V to be concentrated at the top of the structure
W_i, W_x	=	the portion of W that is located at or assigned to level i or x, respectively
h_i, h_x	=	the height above the base (i=0) to level i or x, respectively
i	=	is any level in the building (e.g., i = 1 for the first level above the base)
n	=	is the uppermost level in the main portion of the structure

Example

```
DEFINE NRC 2005 ACC LOAD
SA1 .33 SA2 .25 SA3 .16 SA4 .091 MVX 1.2 MVZ 1.5 JX .7 JZ .5 IE 1.3 -
RDX 4.0 ROX 1.5 RDZ 3.0 ROZ 1.3 SCLASS 4
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)
- *V. NRC 2005 Static Seismic* (on page 3620)

TR.31.2.4 Canadian Seismic Code (NRC) - 2010

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code (NRC/CNRC) of Canada- 2010 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

STAAD.Pro utilizes the following format to generate the lateral seismic loads.

```
DEFINE NRC 2010 (ACCIDENTAL) LOAD
```

```
nrc-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
nrc-spec = { SA1 f1 SA2 f2 SA3 f3 SA4 f4 MVX f5 MVZ f6 I f7 RDX f8 ROX f9 RDZ f10 ROZ
f11 SCLASS f12 STX f13 STZ f14 MD f15 ( CTX f16 ) ( CTZ f17 ) ( PX f18 ) ( PZ f19 )
( FA f20 ) ( FV f21 ) }
```

where:

SA1 *f1*	Seismic Data, Sa(0.2), as per Table C-2.
SA2 *f2*	Seismic Data, Sa(0.5), as per Table C-2.
SA3 *f3*	Seismic Data, Sa(1.0), as per Table C-2.
SA4 *f4*	Seismic Data, Sa(2), as per Table C-2.

MVX *f5* The higher mode factor along the X and Z direction. Refer to NRC Table 4.1.8.11.

MVZ *f6* The higher mode factor along the Z direction. Refer to NRC Table 4.1.8.11.

I *f7* Earthquake Importance Factor, I_e, of the structure as determined from table 4.1.8.5 in the code. This dependent on the Importance Category and ULS/SLS.

RDX *f8* The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

RDZ *f10* The ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

ROX *f9* the over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the X direction. Refer to NRC Table 4.1.8.9.

ROZ *f11* the over strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along the Z direction. Refer to NRC Table 4.1.8.9.

SCLASS *f12* an integer corresponding to site classes A through E (where 1 = A and 6 = F). F_a and F_v are determined based on Site Class as per Table 4.1.8.4.B and Table 4.1.8.4.C.

STX *f13* Type of lateral resisting system along the X direction. The parameters can take values from 1 to 5:

1 = Moment Resisting Frames
2 = Coupled Walls
3 = Braced Frames
4 = Walls, wall-frame systems
5 = Other systems

STZ *f14* Type of lateral resisting system along the Z direction. The parameters can take values from 1 to 5:

1 = Moment Resisting Frames
2 = Coupled Walls
3 = Braced Frames
4 = Walls, wall-frame systems
5 = Other systems

MD *f15* Command to check if the time period calculated is for the purpose of Member Strength or Deflection as per Clause 4.1.8.11.(3).(v)

1 = Member strength
2 = Deflection

CTX *f16* Optional CT value along the X direction to calculate time period.

CTZ *f17* Optional CT value along the Z direction to calculate time period.

PX *f18* Optional Periods of structure (in sec) in the X direction to be used as fundamental period of the structure. If not entered the value is calculated from the code.

PZ *f19* Optional Periods of structure (in sec) in the Z direction to be used as fundamental period of the structure. If not entered the value is calculated from the code.

FA *f20* Optional Short-Period site coefficient at 0.2s.

> **Note:** Value must be provided if SCLASS set to F (i.e., *f12* = 6).

FV *f21* Optional Long-Period site coefficient at 1.0s.

> **Note:** Value must be provided if SCLASS set to F (i.e., *f12* = 6).

If the `ACCIDENTAL` option is specified, the program calculates the accidental torsion per the NRC 2010 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` commands.

The `ACCIDENTAL` option along with accidental eccentricity factor (default 0.1 as per NRC 2010) needs to be provided in the NRC seismic primary load case (i.e., NRC LOAD X / Z *f1* ACC *f3*). *f3* can be negative.

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the `ACCIDENTAL` option should not be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the NRC seismic primary load case (i.e., `NRC LOAD X / Z f1 DEC f2 ACC f3`). For equivalent seismic analysis, *f2* is 1 and *f3* is 0.1 as per NRC 2005 code. *f1* is always positive or zero, however *f2* can be negative. If *f2* is 0.0, only accidental torsion will be considered for this particular load case.

> **Note:** For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The equivalent static force procedure to obtain the base shear is implemented according to section 4.1.8.11, Division B of NBCC 2010.

Seismic Base Shear

The minimum lateral earthquake force is calculated according to the following equation (see 4.1.8.11.2)

$$V = \frac{S(T_a)M_v I_E W}{R_d R_o}$$

- · For walls, coupled walls and wall-frame systems, V shall not be less than

  $$S(2.0)M_v I_E W/R_d R_o)$$

  M_v must be calculated with $T \geq 4.0$
- For moment-resisting frames, braced-frames and other systems, V shall not be less than

  $$S(2.0)M_v I_E W/R_d R_o)$$

  M_v must be calculated with $T \geq 2.0$
- And for $V > 1.5$, V need not be greater than

  $$\frac{2}{3}S(0.2)I_E W / \left(R_d R_o\right)$$

Fundament Period, T_a

The fundamental period, Ta, is based on one of the following choices:

- Clause (a): Use 4.1.18.11.3a:
  i. $0.085(h_n)^{3/4}$ for steel moment frames
  ii. $0.075(h_n)^{3/4}$ for concrete moment frames
- Clause (b):

  $0.025h_n$ for braced frames
- Clause (c):

  $0.05(h_n)^{3/4}$ for shear walls and other structures

where

h_n = the height of the building in meters

In the preceding equations, clauses (a), (b) and (c) are implemented except $T_a = 0.1N$. The period is also calculated in accordance with the Rayleigh method

You may also specify the time period using parameters PX an PX. In this case the program checks the following limits:

- Clause d-i: For moment resisting frames: $T_a \leq 1.5\times$that determined in clause (a)
- Clause d-ii: For braced frames: $T_a \leq 2.0\times$that determined in clause (b)
- Clause d-iii: For shear wall structures: $T_a \leq 2.0\times$that determined in clause (c)
- Clause d-iv: For other structures: $T_a \leq$ that determined in clause (c)

However, if the load case is created for drift provisions, (i.e., calculating drift/deflections), the above limits are *not* checked but instead the following limits are enforced (see Clause d-v):

- $T_a \leq 2.0$ if it is a moment-resisting frame, braced frame, or other system
- $T_a \leq 4.0$ for all others (i.e., walls, coupled walls, and wall frame system)

Also, it is stated in code that these upper limits specified may not be checked for deflection and period calculations.

Design Spectral Response Acceleration, $S(T)$

The design spectral acceleration, $S(T)$, is determined as follows, using linear interpolation for intermediate values of T:

$$S(T) = \begin{array}{ll} F_a S_a(0.2) & \text{for} \quad T \leq 0.2s \\ F_v S_a(0.5) \text{or} F_a S_a(0.2) \text{whichever is smaller} & \text{for} \quad T = 0.5s \\ F_v S_a(1.0) & \text{for} \quad T = 1.0s \\ F_v S_a(2.0) & \text{for} \quad T = 2.0s \\ \frac{1}{2} F_v S_a(2.0) & \text{for} \quad T \geq 4.0s \end{array}$$

The above data Sa(0.2), Sa(0.5), Sa(1.0) and Sa(2) are the seismic data and are provided in the parameters SA1, SA2, SA3, and SA4, respectively, from the table C-2.

Based on the above values of Sa(Ta), Fa and Fv, the acceleration- and velocity- based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of Sa(0.2) and Sa(1.0). It is to be mentioned that, these are the user inputs based on the site classes from A to E and the desired Sa(0.2) and Sa(1.0) values as required as per the above equations.

Higher Mode Factor, M_v

M_v is the factor to account for higher mode effect on base shear which is obtained from the Table 4.1.8.11 based on Lateral Resisting System, Sa(0.2), and Sa(2.0). To get this higher mode factor (M_v), you must evaluate the ratios of $Sa(0.2)/Sa(2.0)$ as also the "Type of Lateral Resisting System." You may alternatively directly specify M_v using the `MVX` and `MVZ` parameters.

Force Modification Factors, R_d and R_o

The ductility related force modification factor, R_d, reflects the capability of a structure to dissipate energy through inelastic behavior. The over-strength-related force modification factor, R_o , accounts for the dependable portion of reserve strength in a structure designed according to the provision of Article 4.1.8.9. These values are specified using the `RDX`, `ROX`, `RDZ`, and `ROZ` parameters and depend on the type of SFRS.

Seismic Weight of the Building, W

Calculated by the program as:

$$W = \sum_{i=1}^{n} W_i$$

where

W_i = the portion of W that is located at or assigned to level i

Distribution of Lateral Earthquake Force

The calculated base shear, V, is distributed over the height of the building by the following equation:

$$F_x = \frac{(V - F_t) W_x h_x}{\sum_{i=1}^{n} W_i h_i}$$

where

F_t	=	Concentrated force applied on the top of the building and it accounts for effects of higher order modes. As per Clause 4.1.8.11(6) Ft is equal to $0.07T_aV$, but F_t is not greater than 0.25V and Ft = 0 when Ta is not greater than 0.7s
W_i, W_x	=	the portion of W that is located at or assigned to level i or x respectively
h_i, h_x	=	height above the base($i=0$) to level i or x respectively
i	=	any level of the building, $i=1$ for the first floor above the base and $i=n$ for the uppermost level in the main portion of the structure

Torsional Effect

Torsional effects are accounted for according to the Clause 4.1.8.11(10):

$$T_x = F_x(e_x + 0.10\ D_{nx})$$

$$T_x = F_x(e_x - 0.10\ D_{nx})$$

where

e_x	=	natural eccentricity due to center of rigidity and center of mass being at different positions
D_{nx}	=	plan dimension normal to the direction of ground motion

Note: The overturning moment calculation as given in 4.1.8.11 is *not* evaluated by the program.

Example

This example input file demonstrates a seismic load using the equivalent force method and a seismic response spectrum analysis per NRC 2010.

```
STAAD SPACE EXAMPLE PROBLEM FOR NRC LOAD
START JOB INFORMATION
ENGINEER DATE 15-Jan-16
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3.5 0 0; 3 7 0 0; 4 13.5 0 0; 5 0 0 3.5; 6 3.5 0 3.5;
7 7 0 3.5; 8 13.5 0 3.5; 9 0 0 7; 10 3.5 0 7; 11 7 0 7; 12 13.5 0 7;
13 0 0 12.5; 14 3.5 0 12.5; 15 7 0 12.5; 16 13.5 0 12.5; 17 0 3.5 0;
18 3.5 3.5 0; 19 7 3.5 0; 20 13.5 3.5 0; 21 0 3.5 3.5; 22 3.5 3.5 3.5;
23 7 3.5 3.5; 24 13.5 3.5 3.5; 25 0 3.5 7; 26 3.5 3.5 7; 27 7 3.5 7;
28 13.5 3.5 7; 29 0 3.5 12.5; 30 3.5 3.5 12.5; 31 7 3.5 12.5;
32 13.5 3.5 12.5; 33 0 7 0; 34 3.5 7 0; 35 7 7 0; 36 13.5 7 0;
37 0 7 3.5; 38 3.5 7 3.5; 39 7 7 3.5; 40 13.5 7 3.5; 41 0 7 7;
42 3.5 7 7; 43 7 7 7; 44 13.5 7 7; 45 0 7 12.5; 46 3.5 7 12.5;
47 7 7 12.5; 48 13.5 7 12.5; 49 0 10.5 0; 50 3.5 10.5 0; 51 7 10.5 0;
52 13.5 10.5 0; 53 0 10.5 3.5; 54 3.5 10.5 3.5; 55 7 10.5 3.5;
56 13.5 10.5 3.5; 57 0 10.5 7; 58 3.5 10.5 7; 59 7 10.5 7;
60 13.5 10.5 7; 61 0 10.5 10.5; 62 3.5 10.5 10.5; 63 7 10.5 10.5;
64 13.5 10.5 10.5;
MEMBER INCIDENCES
101 17 18; 102 18 19; 103 19 20; 104 21 22; 105 22 23; 106 23 24;
107 25 26; 108 26 27; 109 27 28; 110 29 30; 111 30 31; 112 31 32;
113 33 34; 114 34 35; 115 35 36; 116 37 38; 117 38 39; 118 39 40;
119 41 42; 120 42 43; 121 43 44; 122 45 46; 123 46 47; 124 47 48;
125 49 50; 126 50 51; 127 51 52; 128 53 54; 129 54 55; 130 55 56;
131 57 58; 132 58 59; 133 59 60; 134 61 62; 135 62 63; 136 63 64;
201 17 21; 202 18 22; 203 19 23; 204 20 24; 205 21 25; 206 22 26;
207 23 27; 208 24 28; 209 25 29; 210 26 30; 211 27 31; 212 28 32;
213 33 37; 214 34 38; 215 35 39; 216 36 40; 217 37 41; 218 38 42;
219 39 43; 220 40 44; 221 41 45; 222 42 46; 223 43 47; 224 44 48;
225 49 53; 226 50 54; 227 51 55; 228 52 56; 229 53 57; 230 54 58;
231 55 59; 232 56 60; 233 57 61; 234 58 62; 235 59 63; 236 60 64;
301 1 17; 302 2 18; 303 3 19; 304 4 20; 305 5 21; 306 6 22; 307 7 23;
308 8 24; 309 9 25; 310 10 26; 311 11 27; 312 12 28; 313 13 29;
314 14 30; 315 15 31; 316 16 32; 317 17 33; 318 18 34; 319 19 35;
320 20 36; 321 21 37; 322 22 38; 323 23 39; 324 24 40; 325 25 41;
326 26 42; 327 27 43; 328 28 44; 329 29 45; 330 30 46; 331 31 47;
332 32 48; 333 33 49; 334 34 50; 335 35 51; 336 36 52; 337 37 53;
338 38 54; 339 39 55; 340 40 56; 341 41 57; 342 42 58; 343 43 59;
344 44 60; 345 45 61; 346 46 62; 347 47 63; 348 48 64;
START GROUP DEFINITION
MEMBER
_B1 301 TO 303 305 TO 307 309 TO 311 317 TO 319 321 TO 323 325 TO 327 -
333 TO 335 337 TO 339 341 TO 343 345 TO 347
END GROUP DEFINITION
MEMBER PROPERTY CANADIAN
101 TO 136 201 TO 236 PRIS YD 0.4 ZD 0.3
301 TO 303 305 TO 307 309 TO 311 317 TO 319 321 TO 323 325 TO 327 333 -
334 TO 335 337 TO 339 341 TO 343 345 TO 347 TABLE ST W460X52
304 308 312 TO 316 320 324 328 TO 332 336 340 344 348 TABLE ST W530X85
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
```

```
E 2.5e+007
POISSON 0.17
DENSITY 24
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 2.17185e+007
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-005
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 101 TO 136 201 TO 236
MATERIAL STEEL MEMB 301 TO 348
SUPPORTS
1 TO 16 FIXED
CUT OFF MODE SHAPE 10
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 7
49 TO 64 FX 3.5
17 TO 48 FY 7
49 TO 64 FY 3.5
17 TO 48 FZ 7
49 TO 64 FZ 3.5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3.5
DIA 2 TYPE RIG HEI 7
DIA 3 TYPE RIG HEI 10.5
*** Equivelant Lateral Force Definition ***
DEFINE NRC 2010 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1.2 MVZ 1.2 -
 RDX 1.4 RDZ 3 ROX 1.5 ROZ 1.5 STX 3 STZ 4 MD 1
*****************************************************
*** X-DIRECTION
LOAD 1 FX+TX
NRC LOAD X 1 DEC 1 ACC 0.1
LOAD 2 FX-TX
NRC LOAD X 1 DEC 1 ACC -0.1
LOAD 3 -FX-TX
NRC LOAD X -1 DEC 1 ACC -0.1
LOAD 4 -FX+TX
NRC LOAD X -1 DEC 1 ACC 0.1
*** Z-DIRECTION
```

```
LOAD 5 FZ+TZ
NRC LOAD Z 1 DEC 1 ACC 0.1
LOAD 6 FZ-TZ
NRC LOAD Z 1 DEC 1 ACC -0.1
LOAD 7 -FZ-TZ
NRC LOAD Z -1 DEC 1 ACC -0.1
LOAD 8 -FZ+TZ
NRC LOAD Z -1 DEC 1 ACC 0.1
*****************************************************
**** RESPONSE SPECTRUM ****
*** X-DIRECTION
LOAD 9
SPECTRUM CQC NRC 2010 TOR X 1 ACC DAMP 0.05
0.03 1; 0.05 1.35; 0.1 1.95; 0.2 2.8; 0.5 2.8; 1 1.6;
LOAD 10
*** Z-DIRECTION
SPECTRUM CQC NRC 2010 TOR Z 1 ACC DAMP 0.05
0.03 1; 0.05 1.35; 0.1 1.95; 0.2 2.8; 0.5 2.8; 1 1.6;
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
PRINT DIA CR
FINISH
```

Related Links

- *V. NRC 2010 Static Seismic* (on page 3610)

TR.31.2.5 Chinese Static Seismic per GB50011-2001

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2001. This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

Note: As GB50011-2010 static seismic was added in CONNECT Edition V22 Update 6, in order to use the 2001 edition you must directly specify the code year. Otherwise, the program will default to the latest code edition as is typical for STAAD.Pro.

General Format

The following general format should be used to generate loads in a particular direction.

```
DEFINE GB50011 2001 (ACCIDENTAL) LOAD
```

```
INTENSITY s1 { FREQUENT | RARE } GROUP i1 SCLASS i2 (DAMP f1) GFACTOR { 0.85 | 1.0 }
(DELN f2) (SF f3) (PX f4) (PZ f5)
```

Where:

INTENSITY *s1*	the Fortification Intensity (ref. table 5.1.4-1). Acceptable values are 6, 7, 7A, 8, 8A, or 9. 7A represents 7 (0.15g), 8A represents 8 (0.30g).
FREQUENT | RARE	Frequency of seismic action, as specified by FREQUENT or RARE (ref. table 5.1.4-1).
GROUP *i1*	Design Seismic Group (ref. table 5.1.4-2). Acceptable values are 1,2, or 3.
SCLASS *i2*	Site-Class (ref. table 5.1.4-2). Acceptable values are 1, 2, 3, or 4.
DAMP *f1*	Damping ratio (default = 0.05 for 5% damping)

GFACTOR	Equivalent factor of gravity load of horizontal seismic action, as specified by 0.85 or 1.0 (ref. clause 5.2.1). The default value is 0.85.
DELN *f2*	δ_n , Additional seismic action factor at the top of the building (default as calculated from Table 5.2.1)
SF *f3*	Shear Factor, λ, Minimum seismic shear factor of the floor (default as calculated from Table 5.2.5)
PX *f4*	optional time period along the X direction
PZ *f5*	optional time period along the Z direction

Generation of GB50011 Seismic Load

To apply the load in any load case, following command would be used

```
LOAD CASE i
```

```
GB LOAD { X | Y | Z } (f6) (ACC f7)
```

Where:

LOAD *i*	load case number
GB LOAD { X | Y | Z } *f6*	An optional factor to multiply horizontal seismic load.
ACC *f7*	The multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Note: If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the GB specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Notes:

1. When the seismic load is defined as per topic TR.32.12.2 Generation of Seismic Loads (on page 2598), then currently only the accidental torsion `ACC` is supported, `DEC` is *not* supported.
2. If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the GB 50011-2001 specifications. The value of the accidental torsion is based on the “center of mass” for each level. The “center of mass” is calculated from the selfweight, joint weights and member weights you have specified.
3. The seismic load generator can be used to generate lateral loads in the X and Z directions for Y up and the X and Y directions for Z up; where Y up or Z up is the vertical axis parallel to the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).
4. All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.
5. This method of seismic load generation is limited in use to buildings not taller than 40 meters, with deformations predominantly due to shear, and a rather uniform distribution of mass and stiffness in elevation. Alternately, for buildings modeled as a single-mass system, a simplified method such as this base shear method, may be used.

Gravity Loads for Design

In the computation of seismic action, the representative value of gravity load of the building shall be taken as the sum of characteristic values of the weight of the structure and members plus the combination values of variable loads on the structure. The combination coefficients for different variable loads shall be taken from the following table.

Table 240: Combinations of different load effects per GB50011-2001

Type of Variable	land Combination coefficient	
Snow load	0.5	
Dust load on roof	0.5	
Live load on roof	Not considering	
Live load on the floor, calculated according to actual state	1.0	
Live load on the floor, calculated according to equivalent uniform state	Library, archives	0.8
	Other civil buildings	0.5
Gravity for hanging object of crane	Hard hooks	0.3
	Soft hooks	Not considering

Seismic Influence Coefficient

This shall be determined for building structures according to the Intensity, Site-class, Design seismic group, and natural period and damping ratio of the structure. The maximum value of horizontal seismic influence coefficient shall be taken from Table 2.2; the characteristic period shall be taken as Table 2.3 according to Site-class and Design seismic group, that shall be increased 0.05s for rarely earthquake of Intensity 8 and 9.

Table 241: Earthquake influence per GB50011-2001

Earthquake influence	Intensity 6	Intensity 7	Intensity 8	Intensity 9
Frequent earthquake	0.04	0.08 (0.12)	0.16(0.24)	0.32
Rarely earthquake	-	0.50(0.72)	0.90(1.20)	1.40

Note: The values in parenthesis are separately used for where the design basic seismic acceleration is 0.15g and 0.30g.

Table 242: Earthquake group per GB50011-2001

Earthquake Group	Site class			
	I	II	III	IV
1	0.25	0.35	0.45	0.65
2	0.30	0.40	0.55	0.75
3	0.35	0.45	0.65	0.90

Calculation of Seismic Influence Coefficient

The design base shear is computed in accordance with the equations shown below.

The damping adjusting and forming parameters on the building seismic influence coefficient curve (Fig.2.1) shall comply with the following requirements:

1. The damping ratio of building structures shall select 0.05 except otherwise provided, the damping adjusting coefficient of the seismic influence coefficient curve shall select 1.0, and the coefficient of shape shall conform to the following provisions:
   a. Linear increase section, whose period (T) is less than 0.1 s;
   b. Horizontal section, whose period form 0. is thought to characteristic period, shall select the maximum value (α_{max});
   c. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index (γ) shall choose 0.9.
   d. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope (η_1) shall choose 0.02.

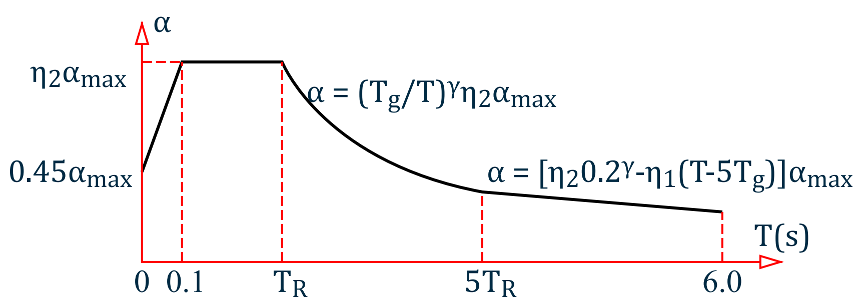


Figure 314: Seismic influence coefficient curve

2. When the damping adjusting and forming parameters on the seismic influence coefficient curve shall comply with the following requirements:
   a. The power index of the curvilinear decreased section shall be determined according to the following equation E2.1

$$\gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} \tag{E2.1}$$

where

γ	=	the power index of the curvilinear decrease section
ξ	=	the damping ratio

b. The adjusting factor of slope for the linear decrease section shall be determined from following equation:

$$\eta_1 = 0.02 + \frac{0.05 - \zeta}{8} \quad \text{E2.2}$$

where

η_1 = the adjusting factor of slope for the linear decrease section, when it is less than 0, shall equal 0.

c. The damping adjustment factor shall be determined according to the following equation:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} \quad \text{E2.3}$$

where

η_2 = the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

Calculation of Horizontal Seismic Action

Characteristic Value of Horizontal Seismic Action

When the base shear force method is used, only one degree of freedom may be considered for each story; the characteristic value of horizontal seismic action of the structure shall be determined by the following equations:

$$F_{Ek} = a_1 G_{eq} \quad \text{E2.4}$$

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n) \quad \text{E2.5}$$

$$\Delta F_n = \delta_n F_{Ek} \quad \text{E2.6}$$

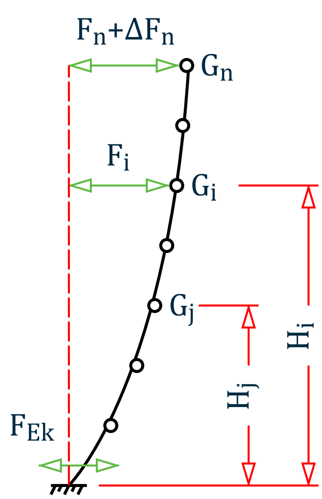


Figure 315: Calculation of horizontal seismic action

where

F_{Ek} = characteristic value of the total horizontal seismic action of the structure

a_1 = horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 2.3. For multistory masonry buildings and multi-story

		brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.
G_{eq}	=	equivalent total gravity load of a structure. When the structure is modeled as a single-mass system, the representative value of the total gravity load shall be used; and when the structure is modeled as a multi-mass system, the 85% of the representative value of the total gravity load may be used.
F_i	=	characteristic value of horizontal seismic action applied on mass i^{th} level.
G_i, G_j	=	representative values of gravity load concentrated at the masses of i^{th} and j^{th} respectively, which shall be determined by Clause 2.1.
H_i, H_j	=	calculated height of i^{th} and j^{th} from the base of the building respectively.
δ_n	=	additional seismic action factors at the top of the building; for multi-story reinforced concrete buildings, it may be taken using Table 2.4; for multi-story brick buildings with inner-frames, a value of 0.2 may be used; no need to consider for other buildings
ΔF_n	=	additional horizontal seismic action applied at top of the building.

Table 243: Additional seismic action factors at top of the building (Table 5.2.1 from GB50011)

T_g (s)	$T_1 > 1.4T_g$	$T_1 \le 1.4T_g$
$T_g \le 0.35$	$0.08T_1 + 0.07$	0
$0.35 < T_g \le 0.55$	$0.08T_1 + 0.01$	
$T_g > 0.55$	$0.08T_1 - 0.02$	

Note: T_1 is the fundamental period of the structure.

Horizontal Seismic Shear Force Verification

The horizontal seismic shear force at each floor level of the structure shall comply with the requirement of the following equation:

$$V_{Eki} > \lambda \sum_{j=1}^{n} G_j \quad \text{E2.7}$$

where

V_{Eki}	=	the floor i^{th} shear corresponding to horizontal seismic action characteristic value.
λ	=	Shear factor, it shall not be less than values in Table 2.5; for the weak location of vertical irregular structure, these values shall be multiplied by the amplifying factor of 1.15.
G_j	=	the representative value of gravity load in floor j^{th} of the structure.

Table 244: Minimum seismic shear factor value of the floor level per GB50011-2001

Structures	Intensity 7	Intensity 8	Intensity 9
structures with obvious torsion effect or fundamental period is less than 3.5s	0.16 (0.024)	0.032 (0.048)	0.064
Structures with fundamental period greater than 5.0s	0.012 (0.018)	0.024 (0.032)	0.040

Note:

1. The values may be selected through interpolation method for structures whose fundamental period is between 3. 5s and 5s.
2. Values in the brackets are used at the regions with basic seismic acceleration as 0.15g and 0.30g respectively.

Notes

1. Structures having the oblique direction lateral-force-resisting members and the oblique angel to major orthogonal axes is greater than 150, the horizontal seismic action along the direction of each lateral-force-resisting member shall he considered respectively. So we could consider this though the item, the action of the oblique member could be multiplied by this factor as design force.
2. Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be $e_i = \pm 0.05L_i$,

   where

e_i	=	Eccentricity value of gravity center on ith floor.
L_i	=	maximum width of calculated story of the building.

3. Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

Related Links

- *M. To add a seismic load definition* (on page 857)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *V. GB 50011-2001 Static Seismic - Case1* (on page 3380)
- *V. GB 50011-2001 Static Seismic - Case2* (on page 3388)

TR.31.2.6 Chinese Static Seismic per GB50011-2010

This set of commands may be used to define and generate static equivalent seismic loads as per Chinese specifications GB50011-2010 (2016 edition). This load uses a static equivalent approach, similar to that found in the UBC. Depending on this definition, equivalent lateral loads will be generated in the horizontal direction(s).

General Format

The following general format should be used to generate loads in a particular direction.

```
DEFINE GB50011 ( 2010 ) (ACCIDENTAL) LOAD
```

```
INTENSITY s1 { FREQUENT | FORTIFIED | RARE } GROUP i1 SCLASS i2 (DAMP f1) GFACTOR
{ 0.85 | 1.0 } (DELN f2) (SF f3) (PX f4) (PZ f5)
```

Where:

INTENSITY *s1*	the Fortification Intensity (ref. table 5.1.4-1). Acceptable values are 6, 7, 7A, 8, 8A, or 9. 7A represents 7 (0.15g), 8A represents 8 (0.30g).
FREQUENT | FORTIFIED | RARE	Frequency of seismic action, as specified by `FREQUENT`, `FORTIFIED`, or `RARE` (ref. table 5.1.4-1 and clause 3.10.3).
GROUP *i1*	Design Seismic Group (ref. table 5.1.4-2). Acceptable values are 1,2, or 3.
SCLASS *i2*	Site-Class (ref. table 5.1.4-2). Acceptable values are 0, 1, 2, 3, or 4. 0 represents I0, 1 represents I1, 2 represents II, 3 represents III, 4 represents IV
DAMP *f1*	Damping ratio (default = 0.05 for 5% damping)
GFACTOR	Equivalent factor of gravity load of horizontal seismic action, as specified by 0.85 or 1.0 (ref. clause 5.2.1). The default value is 0.85.
DELN *f2*	δ_n , Additional seismic action factor at the top of the building (default as calculated from Table 5.2.1)
SF *f3*	Shear Factor λ, Minimum seismic shear factor of the floor (default as calculated from Table 5.2.5)
PX *f4*	optional time period along the X direction
PZ *f5*	optional time period along the Z direction

Generation of GB50011 Seismic Load

To apply the load in any load case, following command would be used

```
LOAD CASE i
```

```
GB LOAD { X | Y | Z } (f6) (ACC f7)
```

Where:

LOAD *i*	load case number
GB LOAD { X | Y | Z } *f6*	An optional factor to multiply horizontal seismic load.
ACC *f7*	The multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Note: If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the GB specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Notes:

1. When the seismic load is defined as per topic TR.32.12.2 Generation of Seismic Loads (on page 2598), then currently only the accidental torsion ACC is supported, DEC is *not* supported.

2. If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the GB 50011-2001 specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the selfweight, joint weights and member weights you have specified.
3. The seismic load generator can be used to generate lateral loads in the X and Z directions for Y up and the X and Y directions for Z up; where Y up or Z up is the vertical axis parallel to the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).
4. All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.
5. This method of seismic load generation is limited in use to buildings not taller than 40 meters, with deformations predominantly due to shear, and a rather uniform distribution of mass and stiffness in elevation. Alternately, for buildings modeled as a single-mass system, a simplified method such as this base shear method, may be used.

Gravity Loads for Design

In the computation of seismic action, the representative value of gravity load of the building shall be taken as the sum of characteristic values of the weight of the structure and members plus the combination values of variable loads on the structure. The combination coefficients for different variable loads shall be taken from the following table.

Table 245: Combinations of different load effects per GB50011-2010

Type of Variable	land Combination coefficient	
Snow load	0.5	
Dust load on roof	0.5	
Live load on roof	Not considering	
Live load on the floor, calculated according to actual state	1.0	
Live load on the floor, calculated according to equivalent uniform state	Library, archives	0.8
	Other civil buildings	0.5
Gravity for hanging object of crane	Hard hooks	0.3
	Soft hooks	Not considering

Seismic Influence Coefficient

This shall be determined for building structures according to the Intensity, Site-class, Design seismic group, and natural period and damping ratio of the structure. The maximum value of horizontal seismic influence coefficient shall be taken from Table 5.1.4-1; the characteristic period shall be taken from Table 5.1.4-2 according to Site-class and Design seismic group, that shall be increased 0.05s for rarely earthquake.

Table 246: Earthquake influence per GB50011-2010

Earthquake influence	Intensity 6	Intensity 7	Intensity 8	Intensity 9
Frequent earthquake	0.04	0.08 (0.12)	0.16(0.24)	0.32
Fortified earthquake (per clause 3.10.3)	0.12	0.23 (0.34)	0.45 (0.68)	0.90
Rarely earthquake	0.28	0.50(0.72)	0.90(1.20)	1.40

Note: The values in parenthesis are separately used for where the design basic seismic acceleration is 0.15g (equal to 7A in the command) and 0.30g (equal to 8A in the command).

Table 247: Characteristic Period per GB50011-2010

Earthquake Group	Site class				
	I0	I	II	III	IV
1	0.20	0.25	0.35	0.45	0.65
2	0.25	0.30	0.40	0.55	0.75
3	0.30	0.35	0.45	0.65	0.90

Calculation of Seismic Influence Coefficient

The design base shear is computed in accordance with the equations shown below.

The damping adjusting and forming parameters on the building seismic influence coefficient curve (Fig.2.1) shall comply with the following requirements:

1. The damping ratio of building structures shall select 0.05 except otherwise provided, the damping adjusting coefficient of the seismic influence coefficient curve shall select 1.0, and the coefficient of shape shall conform to the following provisions:
   a. Linear increase section, whose period (T) is less than 0.1 s;
   b. Horizontal section, whose period form 0. is thought to characteristic period, shall select the maximum value (α_{max});
   c. Curvilinear decrease section, whose period from characteristic period thought to 5 times of the characteristic period, the power index (γ) shall choose 0.9.
   d. Linear decrease section, whose period from 5 times characteristic period thought to 6s, the adjusting factor of slope (η_1) shall choose 0.02.

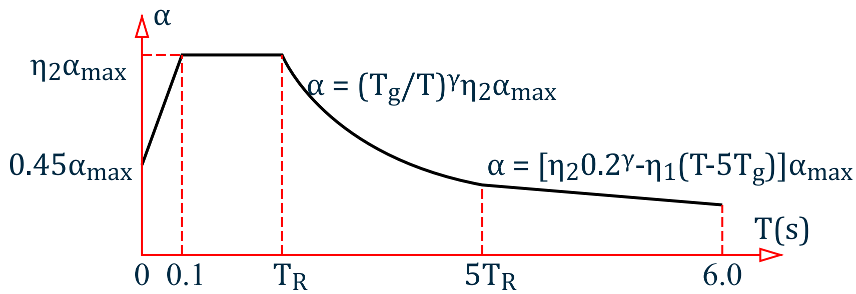


Figure 316: Seismic influence coefficient curve

2. When the damping adjusting and forming parameters on the seismic influence coefficient curve shall comply with the following requirements:

   a. The power index of the curvilinear decreased section shall be determined according to the following equation:

   $$\gamma = 0.9 + \frac{0.05 - \zeta}{0.3 + 6\zeta}$$

   where

γ	=	the power index of the curvilinear decrease section
ξ	=	the damping ratio

   b. The adjusting factor of slope for the linear decrease section shall be determined from following equation:

   $$\eta_1 = 0.02 + \frac{0.05 - \zeta}{4 + 32\zeta}$$

   where

η_1	=	the adjusting factor of slope for the linear decrease section, when it is less than 0, shall equal 0.

   c. The damping adjustment factor shall be determined according to the following equation:

   $$\eta_2 = 1 + \frac{0.05 - \zeta}{0.08 + 1.6\zeta}$$

   where

η_2	=	the damping adjustment factor, when it is smaller than 0.55 shall equal 0.55.

Calculation of Horizontal Seismic Action

<u>Characteristic Value of Horizontal Seismic Action</u>

When the base shear force method is used, only one degree of freedom may be considered for each story; the characteristic value of horizontal seismic action of the structure shall be determined by the following equations:

$$F_{Ek} = a_1 G_{eq} \qquad \text{E2.4}$$

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n) \qquad \text{E2.5}$$

$$\Delta F_n = \delta_n F_{Ek} \qquad \text{E2.6}$$

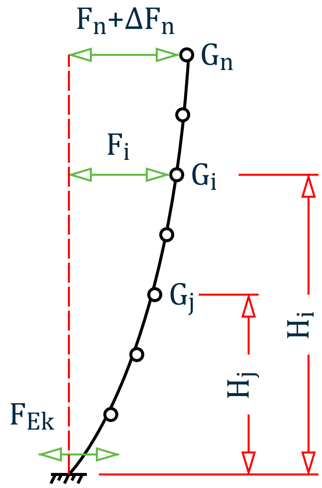


Figure 317: Calculation of horizontal seismic action

where

F_{Ek}	=	characteristic value of the total horizontal seismic action of the structure
a_1	=	horizontal seismic influence coefficient corresponding to the fundamental period of the structure, which shall be determined by using Clause 5.1.4, 5.1.5. For multistory masonry buildings and multi-story brick buildings with bottom-frames or inner-frames, the maximum value of horizontal seismic influence coefficient should be taken.
G_{eq}	=	equivalent total gravity load of a structure. When the structure is modeled as a single-mass system, the representative value of the total gravity load shall be used; and when the structure is modeled as a multi-mass system, the 85% of the representative value of the total gravity load may be used.
F_i	=	characteristic value of horizontal seismic action applied on mass i^{th} level.
G_i, G_j	=	representative values of gravity load concentrated at the masses of i^{th} and j^{th} respectively, which shall be determined by Clause 2.1.
H_i, H_j	=	calculated height of i^{th} and j^{th} from the base of the building respectively.
δ_n	=	additional seismic action factors at the top of the building; for multi-story reinforced concrete buildings, it may be taken using Table 2.4; no need to consider for other buildings
ΔF_n	=	additional horizontal seismic action applied at top of the building.

Table 248: Additional seismic action factors at top of the building (Table 5.2.1 from GB50011)

T_g (s)	$T_1 > 1.4T_g$	$T_1 \leq 1.4T_g$
$T_g \leq 0.35$	$0.08T_1 + 0.07$	0
$0.35 < T_g \leq 0.55$	$0.08T_1 + 0.01$	

$T_g > 0.55$	$0.08T_1 - 0.02$	

Note: T_1 is the fundamental period of the structure.

Horizontal Seismic Shear Force Verification

The horizontal seismic shear force at each floor level of the structure shall comply with the requirement of the following equation:

$$V_{Eki} > \lambda \sum_{j=1}^{n} G_j \qquad \text{E2.7}$$

where

V_{Eki} = the floor ith shear corresponding to horizontal seismic action characteristic value.

λ = Shear factor, it shall not be less than values in Table 2.5; for the weak location of vertical irregular structure, these values shall be multiplied by the amplifying factor of 1.15.

G_j = the representative value of gravity load in floor jth of the structure.

Table 249: Minimum seismic shear factor value of the floor level per GB50011-2001

Structures	Intensity 6	Intensity 7	Intensity 8	Intensity 9
structures with obvious torsion effect or fundamental period is less than 3.5s	0.008	0.16 (0.024)	0.032 (0.048)	0.064
Structures with fundamental period greater than 5.0s	0.006	0.012 (0.018)	0.024 (0.032)	0.040

Note:

1. The values may be selected through interpolation method for structures whose fundamental period is between 3. 5s and 5s.
2. Values in the brackets are used at the regions with basic seismic acceleration as 0.15g and 0.30g respectively.

Notes

1. Structures having the oblique direction lateral-force-resisting members and the oblique angel to major orthogonal axes is greater than 150, the horizontal seismic action along the direction of each lateral-force-resisting member shall he considered respectively. So we could consider this though the item, the action of the oblique member could be multiplied by this factor as design force.
2. Eccentricity: similar to UBC code. The eccentricity value of gravity center on each floor should be $e_i = \pm 0.05L_i$,

   where

   e_i = Eccentricity value of gravity center on ith floor.

   L_i = maximum width of calculated story of the building.
3. Structures having obviously asymmetric mass and stiffness distribution, the torsion effects caused by both two orthogonal horizontal direction seismic action shall be considered; and other structures, it is permitted

that a simplified method, such as adjusting the seismic effects method, to consider their seismic torsion effects.

Example Input

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE DL
JOINT LOAD
2 3 5 6 8 9 11 TO 16 FY -70
END DEFINE REFERENCE LOADS
DEFINE GB50011 2010 LOAD
INTENSITY 6 RARE GROUP 3 SCLASS 0 GFA 0.85 PX 0.4
REFERENCE LOAD Y
R1 1.0
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
GB50011 LOAD X 1
```

Related Links

- *V. GB 50011-2010 Static Seismic - Case1* (on page 3395)
- *V. GB 50011-2010 Static Seismic - Case2* (on page 3403)

TR.31.2.7 Colombian NSR-98 Seismic Load

Used to define and generate static equivalent seismic loads as the 1998 edition of the Colombian seismic code, *Reglamento Colombiano de Construcción Sismo Resistente* (NSR-98),*Normas Colombianas de Diseño y Construcción*, 1998, Asociación Colombiana de Ingeniería Sísmica.

This implementation uses a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). The load is based on

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE COLOMBIAN (ACCIDENTAL) LOAD
```

```
ZONE f1 ( I f2 S f3)
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

ZONE *f1*	Seismic Risk factor
I *f2*	Soil Site Coefficient
S *f3*	Coefficient of Importance

Generation of NSR Seismic Load

General format to provide Colombian Seismic load in any load case:

```
LOAD i
```

```
COLOMBIAN LOAD {X | Y | Z} (f4) (ACCIDENTAL f5)
```

Where:

LOAD *i*	load case number
COLUMBIAN LOAD { X | Y | Z } *f4*	factor to multiply horizontal seismic load
ACCIDENTAL *f5*	multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` commands specified for this load definition.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Example

Using NSR-98:

```
DEFINE COLOMBIAN LOAD
ZONE 0.38  1.0  S  1.5
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
COLOMBIAN  LOAD  X
```

Methodology for NSR-98

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE COLOMBIAN LOAD` command.

The program calculates the natural period of building T utilizing clause 1628.2.2 of UBC 1994.

Design spectral coefficient, S_a, is calculated utilizing T as:

$$S_a = \begin{cases} A_a I(1.0 + 5.0T) \text{ when } 0 \le T \le 0.3 \text{ sec} \\ 2.5 A_a I \text{ when } 0.3 < T \le 0.48 \text{ sec} \\ 1.2 A_a S \frac{I}{T} \text{ when } 0.48 < T < 2.4 \text{ sec} \\ A_a I / 2 \text{ when } 2.4\text{sec} < T \end{cases}$$

where

A_a = Seismic Risk factor

S	=	Soil Site Coefficient
I	=	Coefficient of Importance

Base Shear, Vs is calculated as:

$$V_s = W \times S_a$$

where

W	=	Total weight on the structur

The total lateral seismic load, V_s, is then distributed by the program among different levels as:

$$F_x = C_{vx} \times V_s$$

where

C_{vx}	=	$\dfrac{W_x h_x^K}{\sum_{i=1}^{n} \left(W_x h_x^K\right)}$
W_x	=	Weight at the particular level
h_x	=	Height of that particular level
K	=	1.0 when $T \leq 0.5$ sec $0.75 + 0.5T$ when $0.5 < T \leq 2.5$ sec 2.0 when $T > 2.5$ sec

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.8 Colombian NSR-10 Seismic Load

Used to define and generate static equivalent seismic loads as the 2010 edition of the Colombian seismic code, NSR-10 *Reglamento Colombiano Sismo Resistente*.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE COLOMBIAN 2010 (ACCIDENTAL) LOAD
AA f1 AV f2 FA f3 FV f4 I f5 ( CT f6) ( PX f7) ( PZ f8) ( ALPHA f9 )
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

AA *f1* A coefficient representing effective peak horizontal acceleration for design (Table A.2.4-3 NSR-10). Must be within the range of 0 to 0.5 (inclusive) or the output will indicate an error.

AV *f2* A coefficient representing the effective peak horizontal velocity for design (Table A.2.4-4 NSR-10). Must be within the range of 0 to 0.5 (inclusive) or the output will indicate an error.

FA *f3* The dimensionless amplification factor that affects the acceleration in the short periods due to the effects site (Table A.2.4-3 NSR-10). Must be within the range of 0 to 3.5 (inclusive) or the output will indicate an error.

FV *f4* The dimensionless amplification factor that affects the acceleration in the intermediate periods due to site effects (Table A.2.4-4 NSR-10). Must be within the range of 0 to 3.5 (inclusive) or the output will indicate an error.

I *f5* The importance factor (Table A.2.5-1 NSR-10). Must be within the range of 1 to 1.5 (inclusive) or the output will indicate an error.

CT *f6* Optional CT value to calculate time period. (Table A.4.2-1 NSR-10)

PX *f7* Optional Period of structure (in sec) in X-direction to be in lieu of fundamental period of the structure

PZ *f8* Optional Period of structure (in sec) in Z-direction to be in lieu of fundamental period of the structure

ALPHA *f9* Exponent used in calculating the approximate period, T_a (Table A.4.2-1 NSR-10).

Generation of NSR Seismic Load

General format to provide Colombian Seismic load in any load case:

```
LOAD i
```

```
COLOMBIAN LOAD {X | Y | Z} (f10) (ACCIDENTAL f11)
```

Where:

LOAD *i*	load case number
COLOMBIAN LOAD { X | Y | Z } *f10*	factor to multiply horizontal seismic load
ACCIDENTAL *f11*	multiplying factor for Accidental Torsion, to be used to multiply the accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` commands specified for this load definition.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Example

Using NSR-10:

```
DEFINE COLOMBIAN 2010 LOAD
AA 0.2 AV 0.15 FA 0.85 FV 1.01 I 1.12
JOINT WEIGHT
6 TO 15 21 TO 30 36 TO 45 WEIGHT 10
LOAD 1
COLOMBIAN LOAD X 1.2
```

Methodology for NSR-10

The time period, T, is determined using the Rayleigh method (see section TR.34.1 Rayleigh Frequency Calculation (on page 2614)) or specified directly in the X and Z directions with the optional parameters PX and PZ, is used to calculate spectral coefficient, S_a, as (refer to Fig. A.2.6-1 in the code):

$$S_a = \begin{cases} 2.5A_aF_aI\left(0.4 + 0.6\frac{T}{T_0}\right) \text{ when } T < T_0 \\ 2.5A_aF_aI \text{ when } T_0 \leq T < T_C \\ 1.2A_vF_v\frac{I}{T} \text{ when } T_C \leq T < T_L \\ 1.2A_vF_vT_L\frac{I}{T^2} \text{ when } T > T_L \end{cases}$$

where

T_0	=	$0.1\frac{A_vF_v}{A_aF_a}$
T_C	=	$0.48\frac{A_vF_v}{A_aF_a}$
T_L	=	$2.4F_v$
A_a	=	Seismic Risk factor
S	=	Soil Site Coefficient
I	=	Coefficient of Importance

Base Shear, Vs is calculated as

$$V_s = S_aW$$

where

W	=	Total weight on the structure

The horizontal seismic force, F_x, at any level x is calculated by the program as:

$$F_x = V_sC_{vx}$$

where

C_{vx}	=	$\frac{W_xF_x{}^k}{\Sigma\left(W_xF_x{}^k\right)}$
W_x	=	Weight at the particular level
k	=	the exponent related to the fundamental period, T, of the building;

$$k = \begin{cases} 1.0 \text{ when } T \leq 0.5\text{sec} \\ 0.75 + 0.5T \text{ when } 0.5 \leq T < 2.5\text{sec} \\ 2.0 \text{ when } 2.5\text{sec} \geq T \end{cases}$$

TR.31.2.9 IS:1893 - 1984 Code - Lateral Seismic Load

This feature enables you to generate seismic loads using a static equivalent approach per the IS:1893 - 1984 specifications, *Earthquake Resistant Design of Structures*.

General Format

```
DEFINE 1893 ( ACCIDENTAL ) LOAD
ZONE f1 K f2 I f3 B f4 PX f5 PZ f6
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

where:

ZONE *f1*	Seismic zone coefficient.
K *f2*	Performance factor.
I *f3*	Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance.
B *f4*	Soil interaction factor.
PX *f5*	Optional period in the X direction.
PZ *f6*	Optional period in the Z direction.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2 Generation of Seismic Loads (on page 2598).

Example

```
DEFINE 1893 LOAD
ZONE 0.36 K 2 I B 1
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
```

TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach. Both Part 1 (2002) for building structures and Part 4 (2005) for industrial and stack-like structures are available.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).

General Format

```
DEFINE 1893 ( ACCIDENTAL ) LOAD ( PART4 )
ZONE f1 { 1893-spec-part1  | 1893-spec-part4  }
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

```
(CHECK SOFT STORY)
```

where

```
1893-spec-part1 =    RF f2 I f3 { SS f4 | SA f11 } (ST f5) ( { DM f6 | DF f12 } ) (PX
f7) (PZ f8) ( { DT f9 | GL f10 } )
```

```
1893-spec-part4 =    RF f2 I f3 { SS f4 | SA f11 } ST f5 ( { DM f6 | DF f12 } ) (PX
f7) (PZ f8) ( { DT f9 | GL f10 } ) ( CS f13 ) ( AX f14 ) ( ES f15 ) CV f16 DV f17
```

where

ZONE *f1* Seismic zone coefficient. Refer to Table 2 of IS:1893 (Part 1)-2002.

RF *f2* Response reduction factor. Refer Table 7 of IS: 1893 (Part 1) -2002 or Table 3 of IS: 1893 (Part 4) -2005.

I *f3* Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer Table 6 of IS: 1893(Part 1)-2002 or Table 2 of IS: 1893 (Part 4)-2005.

SS *f4* Rock or soil sites factor. Depending on type of soil, average response acceleration coefficient S_a/g is calculated corresponding to 5% damping. Refer Clause 6.4.5 of IS: 1893 (Part 1) -2002 or Clause 8.3.2 of IS: 1893 (Part 4) -2005.

1 = hard soil
2 = medium soil
3 = soft soil

Note: Use either SS *or*SA to specify site conditions. If both parameters are specified, SS is ignored.

ST *f5* For IS 1893 Part 1, the program will calculate natural period as per Clause 7.6 of IS:1893(Part 1)-2002.

1 = RC frame building
2 = Steel frame building
3 = All other buildings

For IS1893 Part 4, the program will calculate period as per Clause 14.1 of IS:1893(Part 4)-2005 for stack-like structures. For Category 1 Industrial Structures base shear is calculated as twice the base shear of other structures as per Clause 8.3 of IS:1893(Part 4)-2005.

1 = Category 1 industrial structure (Part 4)
3 = All other industrial structures
5 = stack-like structures

Note: This parameter is optional for Part 1, but is required for Part 4.

DM *f6* Damping ratio to obtain multiplying factor for calculating S_a/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer Table 3 of IS:1893(Part 1)-2002.

Note: Use either `DM` *or* `DF` to specify damping. If both parameters are specified, `DM` is ignored.

This should be a value between 0 (zero) and 1.0, inclusive. For example, 7% damping should be specified as `0.07`.

PX *f7* Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along X direction.

Note: See Note 'b' below.

PZ *f8* Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate Sa/g for generation of seismic load along Z direction.

Note: See Note 'b' below.

DT *f9* Depth of foundation below ground level. It should be defined in current units. If the depth of foundation is 30 m or more, the value of A_h is taken as half the value obtained. If the foundation is placed between the ground level and 30 m depth, this value is linearly interpolated between A_h and $0.5A_h$.

Note: Use either `DT` *or* `GL` to specify foundation depth. If both parameters are specified, `DT` is ignored.

GL *f10* Y coordinate of ground level. A reduced lateral force is applied to levels below this height, per Clause 6.4.4.

Used for Y if the `SET Z UP` command is used.

SA *f11* Average response spectral acceleration coefficient corresponding to site specific spectra.

DF *f12* Multiplying factor for calculating S_a/g.

CS *f13* Coefficient as given in Table 6 of IS:1893(Part 4)-2005. Valid only for stack-like structures for calculating fundamental time period per Cl. 14.1.

AX *f14* Area of cross-section at the base of stack-like structures for calculating fundamental time period per Cl. 14.1 of IS:1893(Part 4)-2005.

ES *f15* Modulus of elasticity of material of stack-like structures for calculating fundamental time period per Cl. 14.1 of IS:1893(Part 4)-2005.

CV *f16* Coefficient of shear force given in Table 6 of IS:1893(Part 4)-2005. Required for stack-like structures (`ST 5`).

DV *f17* Distribution factor for shear force given in Table 11 of IS:1893(Part 4)-2005. Required for stack-like structures (`ST 5`).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Notes

a. If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` commands you have specified.

The `ACC` option along with accidental eccentricity factor (generally 0.05 as per IS 1893 code) needs to be provided in the 1893 seismic primary load case (i.e., `1893 LOAD X / Z f1 ACC f3`). *f2* can be negative. See TR.32.12.2 Generation of Seismic Loads (on page 2598)

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the `ACCIDENTAL` option should *not* be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the 1893 seismic primary load case (i.e., `1893 LOAD X / Z f1 DEC f2 ACC f3`). For equivalent seismic analysis, *f2* is 1.5 and *f3* is 0.05 as per IS 1893 code. *f1* is always positive or zero, however *f2* can be negative. If *f2* is 0.0, only accidental torsion will be considered for this particular load case.

b. By default STAAD calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If `PX` and `PZ` are included, the program will consider these values for calculation of average response acceleration coefficient. If `ST` is used instead of `PX` and `PZ` values, then the program will calculate natural period depending upon the empirical expression given in IS: 1893 (Part 1)-2002 or IS: 1893 (Part 4)-2005.

c. In the case where no rigid floor diaphragm is present, STAAD identifies columns and shear walls (without openings) as vertical components for the purpose of computing lateral stiffness of the story.

The lateral stiffness of a column is calculated as:

$$12EI / L^3$$

where

E	=	Young's modulus
I	=	moment of inertia
L	=	length of the column

The lateral stiffness for a shear wall (without opening) is calculated as:

$$\frac{1}{\frac{Ph^3}{12EI} + \frac{1.2Ph}{AG}}$$

Which is the summation of inverse of flexural stiffness and inverse of shear stiffness, obtained as deflection of a cantilever wall under a single lateral load, P, at its top.

where

h	=	height
A	=	cross-sectional area
G	=	shear modulus of the wall

The summation of lateral stiffnesses of all columns and shear walls at a particular floor level constitutes the total lateral stiffness of that particular story or floor level. The program checks for a soft story of a building along both global X and Z directions respectively. This computation is valid only for those structures whose floors are treated as rigid diaphragm

Example

```
DEFINE 1893 LOAD
ZONE 0.36 RF 5 I 1 SS 1 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
```

Methodology

The design base shear is computed by STAAD.Pro for building structures as per IS: 1893 (Part 1) 2002 equation 7.5.3 or for industrial structures as per (Part 4) 2005:

$$V = A_{h.}W$$

Where:

$$A_h = \frac{Z}{2}\frac{I}{R}\frac{S_a}{g}$$

When site specific spectra is used per IS 1893 (Part 4) 2005, then:

$$A_h = \frac{I}{R}\frac{S_a}{g}$$

For stack-like structures, the design base shear is computed as per IS 1893 (Part 4) 2005 as:

$$V = C_v A_{h.} W \cdot D_v$$

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1) 2002 and IS: 1893 (Part 4) 2005.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired 1893 specs through the `DEFINE 1893 LOAD` command. Use the `PART 4` command option to specify using IS: 1893 (Part 4) 2005.
2. The program calculates the structure period, T.
3. The program calculates S_a/g utilizing T.
4. The program calculates V from the above equation. W is obtained from mass table data entered via `SELFWEIGHT`, `JOINT WEIGHT`(s), `MEMBER WEIGHT`(S), and/or `REFERENCE LOAD` you provide through the `DEFINE 1893 LOAD` command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

See TR.32.12 Generation of Loads (on page 2596) for additional information.

Soft Story Checking

Tip: When a TR.28.2 Floor Diaphragm (on page 2328) is used, a soft story check may be initiated within that command without some of the limitations imposed on soft story checking in a seismic load.

As per the IS1893-2002 code Clause 7.1, to perform well during an earthquake a building must have simple and regular configuration, adequate lateral strength, stiffness and ductility. This is because a building with simple regular geometry and uniformly distributed mass and stiffness in plan as well as in elevation, will suffer much less damage than buildings with irregular configurations.

According to this standard, a building can be considered irregular, if at least one of the conditions given in Table 4 - Plan Irregularities and Table 5 - Vertical Irregularities, of IS1893-2002 is applicable.

For IS 1893 2002, STAAD.Pro has implemented the methodology to find vertical stiffness irregularities, as given in IS 1893-2002 Table 5 Sl No. (1) i) a) and Sl No. (1) i) b), in the form of soft story checking.

- Stiffness Irregularities: Soft Story – As per this provision of the code, a soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 80 percent of the average lateral stiffness of the three story above.

- Stiffness Irregularities: Extreme Soft Story – As per this provision of the code, a extreme soft story is one in which the lateral stiffness is less than 60 percent of that in the story above or less than 70 percent of the average lateral stiffness of the three story above.

  Thus, if any story of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

Note: STAAD.Pro identifies column and shear wall (without opening) as vertical component for the purpose of computing lateral stiffness of the story. The vertical stiffness of a column is calculated as $12EI / L^3$ where E is the Young's modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as $Ph^3/3EI + 1.2Ph/AG$ (i.e., summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area, and G is the shear modulus of the wall (E and I are Young's modulus of elasticity and moment of inertia, respectively). The summation of lateral stiffness of all columns and shear walls at a particular floor level constitute the total lateral stiffness of that particular story or floor level. The program checks soft story of a building along both global X and Z directions respectively. This computation is valid *only* for those structures whose floors are treated as rigid diaphragm.

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)
- *V. IS 1893 2002 Static Seismic* (on page 3593)

TR.31.2.9.1 Identification of Floor Level

The following two ways can identify floor level:

- Program calculated
- User defined

For regular building which has well defined floors (i.e., does not contain shear wall, staggered flooring, etc), STAAD.Pro can identify floor level on its own. However, if floor level is not so well defined, it is better to define floor height to have more accurate result for the purpose of torsion and soft story checking.

Program calculated

In general, STAAD.Pro identifies floor levels in order of increasing magnitude of Y-coordinates of joints. The program sorts different values of Y-coordinates, from minimum to maximum values, in ascending order and consider each Y-coordinate value as each floor level. This is the method used by the `DEFINE UBC` or similar load generation features.

This feature has been enhanced to identify the beam-column junctions at each floor level, as identified by the method above. If no beam-column junctions are identified at that level, a floor level will not be considered at that level in the structure. Where beam-column junctions are found, the program identifies two beams, at the same level which span in two different directions from the same beam-column junction. If this true, this identified floor level will be considered as truly existing floor level in the structure.

The enhanced feature for finding floor level is being used by lateral load generation feature for response spectrum analysis and soft story checking as per IS1893-2002 code. For response spectrum analysis, story drift and soft story checking is performed only when floor heights are specified.

User defined

Floor heights should be defined before using the `DEFINE` command for any primary response spectrum load case. The following general format is used for a user-defined floor height:

```
FLOOR HEIGHT
    h1; h2; h3; …;hi -
    hi+1; …;hn
(BASE hb)
```

Where:

h1 … hn	The different floor heights in current length unit and **n** is the number of floor levels.
BASE *hb*	The base level with respect to which 1st story height will be calculated. If *hb* is not defined, the minimum Y-coordinate value present in the model will be taken as base level.

User-defined floor heights are used by lateral load generation for response spectrum analysis and soft story checking as per IS1893-2002 code.

See TR.32.12.2 Generation of Seismic Loads (on page 2598) for an example of the correct usage of this command.

Note: Floor height values *cannot* be used when the `FLOOR DIAPHRAGM` is used. Refer to TR.28.2 Floor Diaphragm (on page 2328) for details.

TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach per Part 1 (2016) for building structures.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).

General Format

```
DEFINE IS1893 2016 ( ACCIDENTAL ) LOAD
1893-2016-spec
wall-definitions
```

Refer to Wall Area Definitions (on page 2352) for information on defining walls.

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

where

```
1893-2016-spec =    ZONE f1 RF f2 I f3 { SS f4 | SA f11 } (ST f5) ( { DM f6 | DF f12 } ) (PX f7) (PZ f8) ( { DT f9 | GL f10 } ) (HT f13) (DX f14) (DZ f15)
```

ZONE *f1* Seismic zone factor. Refer to Table 3 (Clause 6.4.2) of IS:1893 (Part 1)-2016.

RF *f2* Response reduction factor. Refer Table 9 (Clause 7.2.6) of IS: 1893 (Part 1) -2016.

I *f3* Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer Table 8 (Clause 7.2.3) of IS:1893 (Part 1)-2016.

SS *f4* Rock or soil sites factor. Depending on type of soil, average response acceleration coefficient S_a/g is calculated corresponding to 5% damping. Refer to Table 4 (Clause 6.4.2.1) of IS:1893 (Part 1) -2016.

1 = hard soil
2 = medium soil
3 = soft soil

Note: Use either `SS` *or*`SA` to specify site conditions. If both parameters are specified, `SS` is ignored.

ST *f5* Structure type of the seismic-resisting system. The program will calculate natural period as per Clause 7.6.2 of IS:1893(Part 1)-2016.

1 = RC moment-resisting frame building
2 = RC-Steel composite moment-resisting frame building
3 = Steel moment-resisting frame building
4 = Building with RC structural walls*
5 = All other buildings

Note: *ST 4 requires that wall data be entered in order to determine the natural period.

DM *f6* Damping ratio to obtain multiplying factor for calculating S_a/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer to Clause 7.2.4 of IS:1893(Part 1)-2016.

Note: Use either `DM` *or*`DF` to specify damping. If both parameters are specified, `DM` is ignored.

This should be a value between 0 (zero) and 1.0, inclusive. For example, 7% damping should be specified as `0.07`.

PX *f7* Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along X direction.

PZ *f8* Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate Sa/g for generation of seismic load along Z direction.

DT *f9* Depth of foundation below ground level. It should be defined in current units. If the depth of foundation is 30 m or more, the value of A_h is taken as half the value obtained. If the foundation is placed between the ground level and 30 m depth, this value is linearly interpolated between A_h and $0.5A_h$. Refer to Clause 6.4.5 of IS:1893(Part 1)-2016.

Note: Use either `DT` *or* `GL` to specify foundation depth. If both parameters are specified, `DT` is ignored. See Node d below.

GL *f10* Y coordinate of ground level (or global Z coordinate for `SET Z UP`). A reduced lateral force is applied to levels below this height, per Clause 6.4.5.

Note: Use either DT *or* GL to specify foundation depth. If both parameters are specified, DT is ignored. See Node d below.

SA *f11* Average response spectral acceleration coefficient corresponding to site specific spectra. Refer to Clause 6.4.7 of IS:1893(Part 1)-2016.

DF *f12* Multiplying factor for calculating S_a/g.

HT *f13* Height of the building. Refer Clause 7.6.2 (a) and Fig. 5 of IS 1893 2016

DX *f14* Base dimension of the building in X direction at the plinth level. Refer Clause 7.6.2(b) or (c)

DZ *f15* Base dimension of the building in Z direction at the plinth level. Refer Clause 7.6.2(b) or (c)

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Notes

a. If the ACCIDENTAL option is specified, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the SELFWEIGHT, JOINT WEIGHT, and MEMBER WEIGHT commands you have specified.

The ACC option along with accidental eccentricity factor (generally 0.05 as per IS 1893 code) needs to be provided in the 1893 seismic primary load case (i.e., 1893 LOAD X / Z *f1* ACC *f3*). *f2* can be negative. See TR.32.12.2 Generation of Seismic Loads (on page 2598)

To consider horizontal torsion in cases where a floor diaphragm is present in the model, the ACCIDENTAL option should *not* be specified. Instead, dynamic eccentricity along with accidental eccentricity should be provided in the 1893 seismic primary load case (i.e., 1893 LOAD X / Z *f1* DEC *f2* ACC *f3*). For equivalent seismic analysis, *f2* is 1.5 and *f3* is 0.05 as per IS 1893 code. *f1* is always positive or zero, however *f2* can be negative. If *f2* is 0.0, only accidental torsion will be considered for this particular load case.

b. By default, STAAD.Pro calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If PX and PZ are included, the program will consider these values for calculation of average response acceleration coefficient. If ST is used instead of PX and PZ values, then the program will calculate natural period depending upon the empirical expression given in IS: 1893 (Part 1)-2016.

c. In the case where no rigid floor diaphragm is present, STAAD.Pro identifies columns and shear walls (without openings) as vertical components for the purpose of computing lateral stiffness of the story.

The lateral stiffness of a column is calculated as:

$$12EI / L^3$$

where

E	=	Young's modulus
I	=	moment of inertia
L	=	length of the column

The lateral stiffness for a shear wall (without opening) is calculated as:

$$\frac{1}{\frac{Ph^3}{12EI} + \frac{1.2Ph}{AG}}$$

Which is the summation of inverse of flexural stiffness and inverse of shear stiffness, obtained as deflection of a cantilever wall under a single lateral load, P, at its top.

where

h	=	height
A	=	cross-sectional area
G	=	shear modulus of the wall

The summation of lateral stiffnesses of all columns and shear walls at a particular floor level constitutes the total lateral stiffness of that particular story or floor level. The program checks for a soft story of a building along both global X and Z directions respectively. This computation is valid only for those structures whose floors are treated as rigid diaphragm

d. Clause 6.4.5 of IS:1893 part-I -2016 stipulates that for underground structures and foundation at a depth 30m or below, the design horizontal spectrum (A_h or A_k) value should be taken as half of the actual one for structures placed between ground level and 30 m depth the design horizontal acceleration spectrum must be interpolated between A_h and ($0.5\ A_h$). The reduction of A_h should be done on the potion of the structure (mass situated below GL) located below ground.

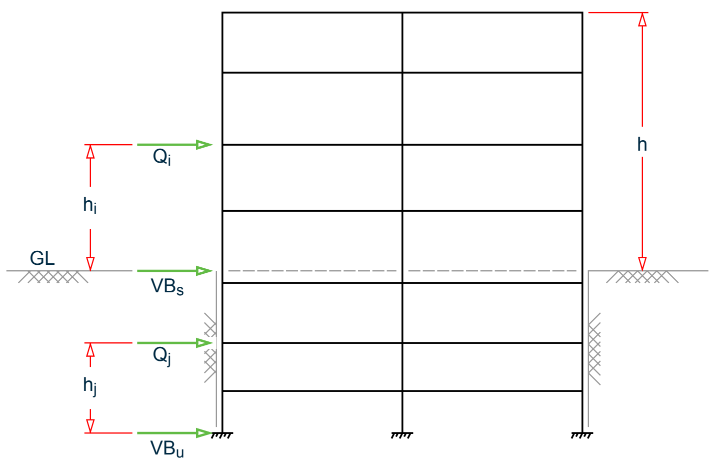


Figure 318: Base shear, VB, calculation above and below ground level, GL

You can provide DT *or* GL parameter to tell the program what your actual depth of foundation below the ground level.

Note: The parameter DT should not be used to reduce A_h. Only GL should be used.

The program will then evaluate the multiplication factor on A_h and calculate the base shear. This reduces the actual base shear for underground portion and the base shear, V_B is distributed into story shear of that portion (for static analysis).

- For the portion of the structure above the ground, the design lateral force at the i^th floor, Q_i:

$$Q_i = \left(\frac{W_i h_i^2}{\sum W_i h_i^2} \right) VB_S$$

where

W_i	=	seismic weight of ith floor above the ground
h_i	=	height of ith floor above the ground
VB_s	=	horizontal base shear = $A_h \cdot W_s$
W_s	=	seismic weight of the portion which is above the ground

- For the portion of the structure below the ground, the design lateral force at the jth floor, Q_j:

$$Q_j = \left(\frac{W_j h_j^2}{\sum W_j h_j^2} \right) VB_S$$

where

W_j	=	seismic weight of jth floor below the ground
h_j	=	height of jth floor below the ground
VB_u	=	horizontal base shear = $Ah_R \cdot W_u$
W_u	=	seismic weight of the portion which is below the ground

Example

```
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
1893 LOAD Y 1
```

Methodology

The design base shear is computed by STAAD.Pro for building structures as per IS: 1893 (Part 1) 2016:

$$V = A_{h.}W$$

where

A_h	=	the design spectral acceleration based on Clause 6.4.2. This value is calculated for each mode.
		$= \frac{Z}{2}\frac{I}{R}\frac{S_a}{g}$
W	=	Seismic weight of the building which is determined from the loads. Iideally, this is defined in a reference load case of type Mass. Alternately, weights can be added in this definition.

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1) 2016.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired 1893 specs through the `DEFINE 1893 LOAD` command.
2. The program calculates the structure period, T.
3. The program calculates Sa/g utilizing T. For the Y direction, Sa/g = 2.5 per clause 6.4.6.
4. The program calculates V from the above equation. W is obtained from mass table data entered via `SELFWEIGHT`, `JOINT WEIGHT`(s), `MEMBER WEIGHT`(S), and/or `REFERENCE LOAD` you provide through the `DEFINE 1893 LOAD` command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

See TR.32.12 Generation of Loads (on page 2596) for additional information.

IS 1893 2016 Implementation

Note: STAAD.Pro can automatically comply with the reduced section property requirements of clause 6.4.3 for seismic loads with use of the `MEMBER CRACKED CODE IS1893 2016` command. Refer to TR.20.10 Member Property Reduction Factors (on page 2282) for details.

7.2 Lateral Force - The design spectral acceleration, A_h, is based on Clause 6.4.2. This value is calculated for each mode and is multiplied by the seismic weight at each degree of freedom.

- 7.2.2 - A minimum of lateral base shear which needs to be distributed to each floor (or each node of each floor) in a building is calculated per Table 7 and Clause 7.2.2. This is used to determine a minimum base shear value.
- 7.2.3 - An additional importance factor has been added.
- 7.2.4 - 5% damping should be used for all structures, regardless of the material. The program will accept other values in the `DM` parameter, but a warning will be issued in the output.
- 7.2.6 - The user interface includes a list of response reduction factors taken from Table 10 of IS 1893 2016.

7.6 Equivalent Static Method

- 7.6.2 - Calculation of the approximate time-period based on height of the building

  For point a, the time period is calculated as follows for reinforced-concrete and steel composite MRF builds: $T_a = 0.08^{0.75}$.
  For point b and point c, the time period is a function of the wall area. Therefore, the *`wall-data-pairs`* must provided to correctly calculate the time period for ST 4 or 5 (reinfroced concrete structural walls or all “other” buildings).

Related Links
TR.31.2.16 IBC 2015 Seismic Load Definition
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *M. To add wall data area to an IS1893 2016 seismic definition* (on page 858)
- *V. IS 1893 2016 Static Seismic* (on page 3504)

TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach per Part 4 (2015) for industrial structures.

All industrial (plant and auxiliary) structures like main plants, process plants, water systems, and maintenance workshops are recognized as industrial structures.

The seismic load generator can be used to generate lateral loads in the X, Y, and Z directions. This facility has not been developed for cases where the Z axis is set to be the vertical direction (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).

Note: This implementation is only applicable to industrial structures. It is *not* valid for stack-like structures.

General Format

```
DEFINE IS1893 2015 ( ACCIDENTAL )  LOAD PART4
```

```
1893-2015-P4-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

where

```
1893-2015-P4-spec =    ZONE f1 RF f2 I f3 { SS f4 | SA f11 } (ST f5) ( { DM f6 | DF f12 } ) (PX f7) (PZ f8)
```

Note: The `ACCIDENTAL` option should only be used for `ST 4` structures.

ZONE *f1* Zone factor based on Table 14 (Clause 7.3.2) FOR 5% damping and Table 15 for Other damping Ratios.

RF *f2* Response Reduction factor. Refer Table 4 (Clause 8.6).

I *f3* Importance factor as per Table 3 (Clause 8.5).

SS *f4* Soil Type as per Annex B. (Clause 16) (Use if the `SA` parameter is not specified).

1 = Type I, rock or hard soil
2 = Type II, stiff soil
3 = Type III, soft soil

Note: Use either `SS` *or*`SA` to specify site conditions. If both parameters are specified, `SS` is ignored.

ST *f5* Structure Type as per Clause 8.1..

1 = Category 1, Failure can cause loss of life
2 = Category 2, Failure can lead to serious fire hazard
3 = Category 3, Failure will not be a serious hazard
4 = Category 4, All other structures

DM *f6* Damping ratio to obtain multiplying factor for calculating S_a/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Damping is considered to be 5% for RC structures and 2% for Steel structures (Refer Table 2). Refer Clause 9.4 of IS:1893(Part 5) -2015.

Note: Use either `DM` *or*`DF` to specify damping. If both parameters are specified, `DM` is ignored.

This should be a value between 0 (zero) and 1.0, inclusive. For example, 7% damping should be specified as `0.07`.

PX *f7* Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along X direction.

PZ *f8* Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along Z direction.

SA *f11* Average response spectral acceleration coefficient corresponding to site specific spectra. Refer to Clause 7.3. Use if SS is not specified.

DF *f12* Multiplying factor for calculating S_a/g.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2 Generation of Seismic Loads (on page 2598).

Notes

a. If the `ACCIDENTAL` option is specified for an `ST 4` structure, the accidental torsion will be calculated per the IS 1893 specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, and `MEMBER WEIGHT` commands you have specified. The `ACCIDENTAL` option is not applicable to other structure types.

The `ACCIDENTAL` option along with accidental eccentricity factor (generally 0.05 as per IS 1893 code) needs to be provided in the 1893 seismic primary load case (i.e., `1893 LOAD X / Z f1 ACC f3`). *f2* can be negative. See TR.32.12.2 Generation of Seismic Loads (on page 2598)

b. By default, STAAD.Pro calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for seismic force at each node. If PX and PZ are included, the program will consider these values for calculation of average response acceleration coefficient. If ST is used instead of PX and PZ values, then the program will calculate natural period using the Rayleigh method.

Example

```
DEFINE IS1893 2015 LOAD PART4
ZONE 0.36 RF 5 I 1.25 SS 1 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
1893 LOAD Y 1
```

Methodology

The seismic force at each node is computed by STAAD.Pro for building structures as per IS: 1893 (Part 4) 2015:

$$V = A_h . W$$

where

A_h = the design seismic coefficient based on either clause 7.3.1 or 7.3.2. This value is calculated for each node.

For site-specific spectra, $A_h = {}^{S_a}/_{g} \times {}^{I}/_{R}$

For standard specific spectra, $A_h = {}^{Z}/_{2} \times {}^{I}/_{R} \times {}^{S_a}/_{g}$

W = Seismic weight of at each node.

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1) 2016.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads:

1. You provide seismic zone coefficient and desired 1893 specs through the `DEFINE 1893 2015 LOAD PART4` command.
2. The program calculates the structure period, T using the Rayleigh method.
3. The program calculates S_a/g utilizing T.
4. Calculate Ah from section 7.3.
5. The program calculates V from the above equation at each node corresponding to the product of the seismic coefficient and mass at each node. W is obtained from mass table data entered via `SELFWEIGHT`, `JOINT WEIGHT`(s), `MEMBER WEIGHT`(S), and/or `REFERENCE LOAD` you provide through the `DEFINE 1893 2015 LOAD PART4` command.

See TR.32.12 Generation of Loads (on page 2596) for additional information.

Related Links

- *V. IS 1893 2015 Static Seismic* (on page 3582)

TR.31.2.13 IBC 2000/2003 Load Definition

The specifications of the IBC 2000, and 2003 codes for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

General Format

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2000 or 2003 LOAD command.

```
DEFINE IBC ( { 2000 | 2003 } ) (ACCIDENTAL) LOAD
 ibc-spec
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
ibc-spec = { SDS f1 SD1 f2 S1 f3 I  f4 RX f5 RZ f6 SCLASS f7 (CT f8) (PX f9) (PZ f10) }
```

SDS *f1*	Design spectral response acceleration at short periods. See equation 16-18, Section 1615.1.3 of IBC 2000 and equation 9.4.1.2.5-1 of ASCE7-02
SD1 *f2*	Design spectral response acceleration at 1-second period. See equation 16-19, Section 1615.1.3. of IBC 2000 and equation 9.4.1.2.5-2 of ASCE7-02
S1 *f3*	Mapped spectral acceleration for a 1-second period. See equation 16-17 of IBC 2000, and 9.4.1.2.4-2 of ASCE 7-02
I *f4*	Occupancy importance factor determined in accordance with Section 1616.2 of IBC 2000 and 2003, and section 9.1.4 (page 96) of ASCE 7-02
RX *f5*	The response modification factor for lateral load along the X direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000

RZ *f6* The response modification factor for lateral load along the Z direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000.

SCLASS *f7* Site class as defined in Section 1615.1.1 of IBC 2000 (page 350) & 2003 (page 322).

Table 250: Values of IBC soil class (SCLASS) used in STAAD

STAAD Value	IBC Value
1	A
2	B
3	C
4	D
5	E
6	F

CT *f8* Optional C_t value used to calculate time period, T_a. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02. If specified, it is your responsibility to provide the value in the correct system of units. Refer to Table 9.5.5.3.2 of AISC 7-02 for values.

If the value of C_t is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where M is the number of members) and uses this to determine the structure type:

i. $E_{avg} < 4{,}000\ ksi$, the program uses a C_t for a moment-resisting concrete frame.
ii. $E_{avg} > 10{,}000\ ksi$, the program uses a C_t for a moment-resisting steel frame.
iii. $4{,}000\ ksi \le E_{avg} \le 10{,}000\ ksi$, the program uses a C_t value for "all other structural systems".

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when C_t not specified. Refer to the IBC/ ASCE 7 code for detailed descriptions.

Table 9.5.5.3.2 of ASCE 7-02 also includes "Eccentrically braced steel frames". STAAD.Pro does not select this value automatically. For this structure type, you must specify C_t.

PX *f9* Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

PZ *f10* Optional Period of structure (in sec) in Z or Y direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The implementation details of the respective codes are as follows:

IBC 2000

On a broad basis, the rules described in section 1617.4 of the IBC 2000 code document have been implemented. These are described in pages 359 thru 362 of that document. The specific section numbers, those which are implemented, and those which are not implemented, are as follows:

Table 251: Sections of IBC 2000 implemented and omitted in the program

Implemented sections of IBC 2000	Omitted sections of IBC 2000
1617.4.1	1617.4.4.1
1617.4.1.1	1617.4.4.2
1617.4.2	1617.4.4.3
1617.4.2.1	1617.4.4.5
1617.4.3	1617.4.5
1617.4.4	1617.4.6
1617.4.4.4	

IBC 2003

On a broad basis, the rules described in section 1617.4 of the IBC 2003 code document have been implemented. This section directs the engineer to Section 9.5.5 of the ASCE 7 code. The specific section numbers of ASCE 7-2002, those which are implemented, and those which are not implemented, are shown in the table below. The associated pages of the ASCE 7-2002 code are 146 thru 149.

Table 252: Sections of IBC 2003 (ASCE 7-02) implemented and omitted in the program

Implemented sections of IBC 2003 (ASCE 7-02)	Omitted sections of IBC 2003 (ASCE 7-02)
9.5.5.2	9.5.5.5.1
9.5.5.2.1	9.5.5.5.2
9.5.5.3	9.5.5.6
9.5.5.3.1	9.5.5.7
9.5.5.3.2	
9.5.5.4	
9.5.5.5	
Portions of 9.5.5.5.2	

Methodology

The design base shear is computed in accordance with Eqn. 16-34 of IBC 2000 and Eqn. 9.5.5.2-1 of ASCE 7-02:

$$V = C_s W$$

The seismic response coefficient, C_s, is determined in accordance with Eqn. 16-35 of IBC 2000 / Eqn. 9.5.5.2.1-1 of ASCE 7-02:

$$C_s = \frac{S_{DS}}{R / I_E}$$

C_s need not exceed the limit given in Eqn. 16-36 of IBC 2000 / Eqn. 9.5.5.2.1-2 of ASCE 7-02:

$$C_s = \frac{S_{D1}}{(R / I_E)T}$$

C_s shall not be taken less than the lower limit given in Eqn. 16-37 of IBC 2000 / Eqn. 9.5.5.2.1-3 of ASCE 7-0:

$$C_s = 0.044\ S_{DS} I_E$$

In addition, for structures for which the 1-second spectral response, S_1, is equal to or greater than 0.6g, the value of the seismic response coefficient, C_s, shall not be taken less than the limit given in Eqn. 16-38 of IBC 2000 / Eqn. 9.5.5.2.1-4 of ASCE 7-02:

$$C_s = \frac{0.5 S_1}{R / I_E}$$

For an explanation of the terms used in the above equations, please refer to the relevant IBC and ASCE 7-02 codes.

Procedure Used by the Program

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02 (IBC 2003) This is reported in the output as T_a.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. you may override the Rayleigh based period by specifying a value for `PX` or `PZ` depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in clause 1617.4.2 of IBC 2000, section 9.5.5.3 of ASCE 7-02 (IBC 2003) or section 12.8.2.1 of ASCE 7-05 (IBC 2006). The resulting value is reported as “Time Period used.”
5. The Design Base Shear is calculated based on equation 16-34 of IBC 2000, equation 9.5.5.2-1 of ASCE 7-02 (IBC 2003) or equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 1617.4.3, equations 16-41 and 16-42 of IBC 2000. For IBC 2003, using clause 9.5.5.4, equations 9.5.5.4-1 & 9.5.5.4-2 of ASCE 7-02.
6. If the `ACCIDENTAL` option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (clause 1617.4.4.4 of IBC 2000, and section 9.5.5.5.2 of ASCE 7-02 for IBC 2003). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

Example 1

```
DEFINE IBC 2003 LOAD
SDS 0.6 SD1 0.36 S1 0.31 I 1.0 RX 3 RZ 4 SCL 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2596) for details.

```
LOAD 1 ( Seismic Load in X Direction )
IBC LOAD X 0.75
LOAD 2 ( Seismic Load in Z Direction )
IBC LOAD Z 0.75
```

The Examples manual contains examples illustrating load generation involving IBC and UBC load types.

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.14 IBC 2006/2009 Seismic Load Definition

The specifications of the seismic loading chapters of the International Code Council 2006 & 2009 code and the ASCE 7-05 (including Supplement #2) code for seismic analysis of a building using a static equivalent approach are available in the program. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2006 LOAD` command.

```
DEFINE IBC 2006 (ACCIDENTAL) LOAD
```

```
map-spec ibc06-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

map-spec = { ZIP f_1 | LAT f_2 LONG f_3 | SS f_4 S1 f_5 }

Where:

ZIP *f1* The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-05 Chapter 22).

LAT ***f2*** The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22).

LONG ***f3*** The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22).

SS ***f4*** The mapped MCE for 0.2s spectral response acceleration. (IBC 2006/2009 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1).

S1 ***f5*** The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-05

```
ibc06-spec = { RX f6 RZ f7 I f8 TL f9 SCLASS f10 (CT f11) (PX f12) (PZ f13) (K f14) (FA f15) (FV f16) }
```

Where:

RX ***f6*** The response modification factor, R, for lateral load along the X direction, (ASCE Table 12.2.1). This is the value used for calculating C_s.

RZ ***f7*** The response modification factor, R, for lateral load along the Z direction, (ASCE Table 12.2.1) This is the value used for calculating C_s.

I ***f8*** Occupancy importance factor. (IBC 2006/2009 Clause 1604.5, ASCE 7-05 Table 11.5-1)

TL ***f9*** Long-Period transition period in seconds. (ASCE 7-05 Clause 11.4.5 and Chapter 22).

SCLASS ***f10*** Site class. Enter 1 through 6 in place of A through F, see table below. (IBC 2006/2009 Table 1613.5.2, ASCE 7-05 Section 20.3)

CTX ***f11*** Optional C_t value in X-direction to calculate time period. (ASCE 7-05 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-05 for values.

If the value of C_t is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where M is the number of members) and uses this to determine the structure type:

i. E_{avg} < *4,000 ksi*, the program uses a C_t for a moment-resisting concrete frame.
ii. E_{avg} > *10,000 ksi*, the program uses a C_t for a moment-resisting steel frame.
iii. *4,000 ksi* ≤ E_{avg} ≤ *10,000 ksi*, the program uses a C_t value for "all other structural systems".

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when C_t not specified. Refer to the IBC/ ASCE 7 code for detailed descriptions.

ASCE 7-05 also includes "Eccentrically braced steel frames". STAAD.Pro does not select this value automatically. For this structure type, you must specify C_t.

CTZ ***f12*** Optional C_t value in Z-direction to calculate time period. (ASCE 7-05 Table 12.8-2). Refer to CTX for details.

PX ***f13*** Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

PZ ***f14*** Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-05 Table 12.8-2).

XX *f15* Optional exponent value, x, in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-05 table 12.8-2 p129). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

XZ *f16* Optional exponent value, x, in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-05 table 12.8-2 p129). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

FA *f17* Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

FV *f18* Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006/2009 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Implementation in STAAD.Pro

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The rules described in section 1613 of the ICC IBC-2006 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2005 code. The specific section numbers of ASCE 7— those which are implemented, and those which are not implemented—are shown in the table below.

Table 253: Sections of IBC 2006 implemented and omitted in the program

Implemented sections of IBC 2006/2009 (ASCE 7-2005)	Omitted sections of IBC 2006/2009 (ASCE 7-2005)
11.4	12.8.4.1
11.5	12.8.4.3 and onwards
12.8	

Additionally, Supplement #2 of ASCE 7-05—as referenced by IBC 2009—specifies a different equation to be used for the lower bound of the seismic response coefficient, which has also been implemented.

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.2.1 of ASCE 7-05 (IBC 2006/2009). This is reported in the output as T_a.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. you may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2 of ASCE 7-05 (IBC 2006). The resulting value is reported as “Time Period used” in the output file.

5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 12.83, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-05.
6. If the `ACCIDENTAL` option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (section 12.8.4.2 of ASCE 7-05 for IBC 2006). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-05 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.Pro.

Methodology

The design base shear is computed in accordance with the following equation (equation 12.8-1 of ASCE 7-05):

$$V = C_s W$$

The seismic response coefficient, C_s, is determined in accordance with the following equation (equation 12.8-2 of ASCE 7-05):

$$C_s = S_{DS}/[R/I_E]$$

For IBC 2006, C_s need not exceed the following limits defined in ASCE 7-05 (equations 12.8-3 and 12.8-4):

$C_s = S_{D1}/[T \cdot (R/I)]$ for $T \leq T_L$
$C_s = S_{D1} \cdot T_L/[T^2(R/I)]$ for $T > T_L$

However, C_s shall not be less than (equation 12.8-5 of ASCE 7-05, supplement #2):

$$C_s = 0.044 \cdot S_{DS} \cdot I \geq 0.01$$

In addition, per equation 12.8-6 of ASCE 7-05, for structures located where S_1 is equal to or greater than 0.6g, C_s shall not be less than

$$C_s = 0.5 \cdot S_1/(R/I)$$

For an explanation of the terms used in the above equations, please refer to the IBC 2006/2009 and ASCE 7-05 codes.

Example 1

```
DEFINE IBC 2006
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2596) for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.15 IBC 2012 Seismic Load Definition

The specifications of the seismic loading chapters of the International Code Council 2012 code and the ASCE 7-10 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

General Format

Lateral loads generated by seismic loading are specified in two stages. The first is to define the IBC 2012 loading, as detailed here. The second is to include that specification in one or more load cases.

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2012 LOAD` command.

```
DEFINE IBC 2012 (ACCIDENTAL) LOAD
map-spec ibc12-spec
(optional-weight-specs)
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

map-spec This identifies the mapped spectral accelerations at a given location by either ZIP (postal) code, latitude and longitude, or direct input. The ZIP code and latitude & longitude options are for use in mainland US only:

```
{ ZIP f1 | LAT f2 LONG f3 | SS f4 S1 f5 }
```

ibc12-spec This identifies the parameters needed to determine the lateral loading to be applied in a load case which references this definition.

```
{ RX f6 RZ f7 I f8 TL f9 SCLASS f10 (CTX f11) (CTZ f12) (PX f13) (PZ f14)
(XX f15) (XZ f16) (FA f17) (FV f18) }
```

The optional seismic weight option is provided for old files. It is recommended that instead the weights are defined using a one or more reference load cases of type `MASS`, (refer to TR.31.6 Defining Reference Load Types (on page 2453)). If the older format is used, refer to TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2349) for the range of available weight commands that can be used.

The `ACCIDENTAL` option is used to include an additional torsional moment taken as the lateral load is applied at a horizontal eccentricity as 5% of the building dimension at each level. This option should *not* be used if the option to include the natural torsion in the application of seismic load (refer to TR.32.12.2 Generation of Seismic Loads (on page 2598) for additional details).

map-spec parameters:

ZIP *f1* The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-10 Chapter 22).

LAT *f2* The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

LONG *f3* The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

SS *f4* The mapped MCE for 0.2s spectral response acceleration. (IBC 2012 Clause 1613.5.1, ASCE 7-10 Clause 11.4.1).

S1 *f5* The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-10.

ibc12-spec parameters:

RX *f6* The response modification factor, R, for lateral load along the X direction, (ASCE 7-10 Table 12.2.1). This is the value used for calculating C_S.

RZ *f7* The response modification factor, R, for lateral load along the Z direction, (ASCE 7-10 Table 12.2.1) This is the value used for calculating C_S.

I *f8* Occupancy importance factor (IBC 2012 Clause 1604.5, ASCE 7-10 Table 11.5-1).

TL *f9* Long-Period transition period in seconds (ASCE 7-10 Clause 11.4.5 and Chapter 22).

SCLASS *f10* Site class. Enter 1 through 6 in place of A through F, see table below (IBC 2012 clause 1613.3.2, ASCE 7-10 Section 20.3)

CTX *f11* Optional C_t value in X-direction to calculate time period. (ASCE 7-10 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-10 for values.

If the value of C_t is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where M is the number of members) and uses this to determine the structure type:

i. $E_{avg} < 4{,}000\ ksi$, the program uses a C_t for a moment-resisting concrete frame.
ii. $E_{avg} > 10{,}000\ ksi$, the program uses a C_t for a moment-resisting steel frame.
iii. $4{,}000\ ksi \le E_{avg} \le 10{,}000\ ksi$, the program uses a C_t value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when C_t not specified. Refer to the IBC/ ASCE 7 code for detailed descriptions.

ASCE 7-10 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify C_t.

CTZ *f12* Optional C_t value in Z-direction to calculate time period. (ASCE 7-10 Table 12.8-2).

Refer to CTX for details.

PX *f13* Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

PZ *f14* Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

XX *f15* Optional exponent value, x, in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of *x* is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

XZ *f16* Optional exponent value, x, in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of *x* is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

FA *f17* Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2012 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

FV *f18* Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2012 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Implementation in STAAD.Pro

Note: Refer to AD.2007-11.3.8 IBC 2012 / ASCE 7-10 Seismic Loads (on page 157) for additional information on using this feature.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The rules described in section 1613 of the ICC IBC-2012 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2010 code. The specific section numbers of ASCE 7 —those which are implemented, and those which are not implemented— are shown in the table below.

Table 254: Sections of IBC 2012 implemented and omitted in the program

Implemented sections of IBC 2012 (ASCE 7-10)	Omitted sections of IBC 2012 (ASCE 7-10)
11.4	12.8.4.1
11.5	12.8.4.3 and onwards
12.8	

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.2.1 of ASCE 7-10 (IBC 2012). This is reported in the output as Ta.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. You may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2 of ASCE 7-10 (IBC 2012). The resulting value is reported as "Time Period used" in the output file.
5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-10 (IBC 2012). It is then distributed at each floor using the rules of clause 12.83, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-10.
6. If the `ACCIDENTAL` option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (section 12.8.4.2 of ASCE 7-10 for IBC 2012). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-10 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-10 code is not implemented in STAAD.

Methodology

The design base shear is computed in accordance with the following equation (equation 12.8-1 of ASCE 7-10):

$$V = C_s W$$

The seismic response coefficient, C_s, is determined in accordance with the following equation (equation 12.8-2 of ASCE 7-10):

$$C_s = S_{DS}/[R/I_E]$$

For IBC 2012, C_s need not exceed the following limits defined in ASCE 7-10 (equations 12.8-3 and 12.8-4):

$C_s = S_{D1}/[T \cdot (R/I)]$ for $T \le T_L$
$C_s = S_{D1} \cdot T_L/[T^2(R/I)]$ for $T > T_L$

However, C_s shall not be less than (equation 12.8-5 of ASCE 7-10):

$$C_s = 0.044 \cdot S_{DS} \cdot I \geq 0.01$$

In addition, per equation 12.8-6 of ASCE 7-10, for structures located where S_1 is equal to or greater than 0.6g, C_s shall not be less than

$$C_s = 0.5 \cdot S_1/(R/I)$$

For an explanation of the terms used in the above equations, please refer to the IBC 2012 and ASCE 7-10 codes.

Example 1

```
DEFINE IBC 2012
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
```

```
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2596) for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

Using SS and S1 for map-spec

```
DEFINE IBC 2018
SS 2.451 S1 0.882 –
I 1.25 RX 2.5 RZ 2.5 SCLASS 4 TL 12 FA 1 FV 1.5
```

Using Latitude and Longitude

```
DEFINE IBC 2018
LAT 34.0998 LONG -118.4128 -
I 1.25 RX 2.5 RZ 2.5 SCLASS 4 TL 12 FA 1 FV 1.5
```

Using ZIP Code

```
DEFINE IBC 2018
ZIP 90210 –
I 1.25 RX 2.5 RZ 2.5 SCLASS 4 TL 12 FA 1 FV 1.5
```

Related Links

- *V. IBC 2012 Static Seismic* (on page 3477)

TR.31.2.16 IBC 2015 Seismic Load Definition

The specifications of the seismic loading chapters of the International Code Council 2015 code and the ASCE 7-10 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2015 LOAD` command.

```
DEFINE IBC 2015 (ACCIDENTAL) LOAD
map-spec  ibc15-spec
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
map-spec  = { ZIP f1 | LAT f2 LONG f3 | SS f4 S1 f5 }
```

Where:

ZIP *f1* The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-10 Chapter 22).

LAT *f2* The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

LONG *f3* The latitude and longitude, respectively, of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-10 Chapter 22).

SS *f4* The mapped MCE for 0.2s spectral response acceleration. (IBC 2015 Clause 1613.5.1, ASCE 7-10 Clause 11.4.1).

S1 *f5* The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-10.

```
ibc15-spec  = { RX f6 RZ f7 I f8 TL f9 SCLASS f10 (CTX f11) (CTZ f12) (PX f13) (PZ
f14) (XX f15) (XZ f16) (FA f17) (FV f18) }
```

Where:

RX *f6* The response modification factor, R, for lateral load along the X direction, (ASCE 7-10 Table 12.2.1). This is the value used for calculating C_s.

RZ *f7* The response modification factor, R, for lateral load along the Z direction, (ASCE 7-10 Table 12.2.1) This is the value used for calculating C_s.

I *f8* Occupancy importance factor (IBC 2015 Clause 1604.5, ASCE 7-10 Table 11.5-1).

TL *f9* Long-Period transition period in seconds (ASCE 7-10 Clause 11.4.5 and Chapter 22).

SCLASS *f10* Site class. Enter 1 through 6 in place of A through F, see table below (IBC 2015 clause 1613.3.2, ASCE 7-10 Section 20.3)

CTX *f11* Optional CT value in X-direction to calculate time period. (ASCE 7-10 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-10 for values.

If the value of C_t is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where M is the number of members) and uses this to determine the structure type:

i. $E_{avg} < 4{,}000\ ksi$, the program uses a C_t for a moment-resisting concrete frame.
ii. $E_{avg} > 10{,}000\ ksi$, the program uses a C_t for a moment-resisting steel frame.
iii. $4{,}000\ ksi \le E_{avg} \le 10{,}000\ ksi$, the program uses a C_t value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when C_t not specified. Refer to the IBC/ ASCE 7 code for detailed descriptions.

ASCE 7-10 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify C_t.

CTZ *f12* Optional CT value in Z-direction to calculate time period. (ASCE 7-10 Table 12.8-2).

Refer to CTX for details.

PX *f13* Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

PZ *f14* Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-10 Table 12.8-2).

XX *f15* Optional exponent value, x, in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

XZ *f16* Optional exponent value, x, in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-10 table 12.8-2). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

FA *f17* Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2015 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

FV *f18* Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2015 Clause 1613.3.3, ASCE 7-10 Section 11.4.3).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Implementation and Methodology

Refer to TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409) for details on the implementation of the IBC 2015 / IBC 2012 / ASCE 7-10 static seismic method.

Example 1

```
DEFINE IBC 2015
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2596) for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

Related Links

- *V. IBC 2015 Static Seismic* (on page 3467)

TR.31.2.17 IBC 2018 Seismic Load Definition

The specifications of the seismic loading chapters of the International Code Council 2018 code and the ASCE 7-16 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

General Format

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE IBC 2018 LOAD` command.

```
DEFINE IBC 2018 (ACCIDENTAL) LOAD
```

```
map-spec  ibc18-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
map-spec  = { ZIP f1 | LAT f2 LONG f3 | SS f4 S1 f5 }
```

Where:

ZIP *f1* The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-16 Chapter 22).

LAT *f2* The latitude of the site used to determine the Ss and S1 factors. (ASCE 7-16 Chapter 22).

LONG *f3* The longitude of the site used to determine the Ss and S1 factors. (ASCE 7-16 Chapter 22).

SS *f4* The mapped MCE for 0.2s spectral response acceleration. (ASCE 7-16 Clause 11.4.2).

S1 *f5* The mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.2 ASCE7-16.

Note: The mapped spectral response acceleration values for IBC 2018 are retrieved from dynamic data provided by the USGS when `ZIP` code or `LAT`/`LONG` methods are used. An internet connection is required for this. If an internet connection is not available, then the values should be input using the `SS` and `S1` parameters.

```
ibc18-spec  = { RX f6 RZ f7 I f8 TL f9 SCLASS f10 (CTX f11) (CTZ f12) (PX f13) (PZ
f14) (XX f15) (XZ f16) (FA f17) (FV f18) }
```

Where:

RX *f6* The response modification factor, R, for lateral load along the X direction, (ASCE Table 12.2.1). This is the value used for calculating C_S.

RZ *f7* The response modification factor, R, for lateral load along the Z direction, (ASCE Table 12.2.1) This is the value used for calculating C_S.

I *f8* Occupancy importance factor. (IBC 2018 Clause 1604.5, ASCE 7-16 Table 11.5-1)

TL *f9* Long-Period transition period in seconds. (ASCE 7-16 Clause 11.4.5 and Chapter 22).

SCLASS *f10* Site class. Enter 1 through 6 in place of A through F, see table below. (IBC 2018 clause 1613.3.2, ASCE 7-16 Section 20.3)

CTX *f11* Optional CT value in X-direction to calculate time period. (ASCE 7-16 Table 12.8-2). If specified, it is your responsibility to provide the value in the correct system of units. Refer to AISC 7-16 for values.

If the value of C_t is not provided, then the program computes the average value of the modulus of elasticity of the model, $E_{avg} = \sum E / M$ (where M is the number of members) and uses this to determine the structure type:

i. $E_{avg} < 4{,}000\ ksi$, the program uses a C_t for a moment-resisting concrete frame.
ii. $E_{avg} > 10{,}000\ ksi$, the program uses a C_t for a moment-resisting steel frame.
iii. $4{,}000\ ksi \le E_{avg} \le 10{,}000\ ksi$, the program uses a C_t value for “all other structural systems”.

Note: It is your responsibility to ensure that the structure type used actually matches the description for the automatically determined structure when C_t not specified. Refer to the IBC/ ASCE 7 code for detailed descriptions.

ASCE 7-16 also includes “Eccentrically braced steel frames”. STAAD.Pro does not select this value automatically. For this structure type, you must specify C_t.

CTZ *f12* Optional CT value in Z-direction to calculate time period. (ASCE 7-16 Table 12.8-2).

Refer to CTX for details.

PX *f13* Optional period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-15 Table 12.8-2).

PZ *f14* Optional period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. (ASCE 7-16 Table 12.8-2).

XX *f15* Optional exponent value, x, in X-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-16 table 12.8-2). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

XZ *f16* Optional exponent value, x, in Z-direction, used in equation 12.8-7, ASCE 7. (ASCE 7-16 table 12.8-2). If the value of x is not provided, then the program computes the average value of the modulus of elasticity of the model to determine the structure type. Refer to CTX for details.

FA *f17* Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2018 Clause 1613.3.3, ASCE 7-16 Section 11.4.3).

FV *f18* Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2018 Clause 1613.3.3, ASCE 7-16 Section 11.4.3).

Implementation and Methodology

Refer to TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409) for details on the implementation of the IBC 2018 / IBC 2015 / IBC 2012 / ASCE 7-16 / ASCE 7-10 static seismic method.

Note: Exceptions 1 and 2 in Clause 11.4.8 of ASCE 7-16 are implemented.

In exception 1, for Site class E and $S_s > 0.75$, Fa value is taking the same for site class C. In this case, there is slight reduction in the seismic force generated.

In exception 2, for site class D, and $S_1 > 0.1$, the program checks if the time period, $T > 1.5T_s$. If that is the case then C_s value is taken as $1.5 \times C_s$ obtained from eqn. 12.8-3 or 12.8-4 (based on the time period). In this case the seismic force increases. In this case but when $T \leq 1.5T_s$, Cs as calculated by eqn 12.8-2 is used.

Example 1

```
DEFINE IBC 2018
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 -
TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. Refer to TR.32.12 Generation of Loads (on page 2596) for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

Related Links

- *V.IBC 2018 Static Seismic T 1.2* (on page 3423)
- *V.IBC 2018 Static Seismic T Greater Than 2.5* (on page 3431)
- *V.IBC 2018 Static Seismic T Less Than 0.5* (on page 3449)

TR.31.2.18 Japanese Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Japanese specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). The implementation is as per Article 88 in the “Building Codes Enforcement Ordinance 2006”.

General Format

```
DEFINE  AIJ (ACCIDENTAL) LOAD
```

```
AIJ-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
AIJ-spec
 = { ZONE f1 CO f2 TC f3 ALPHA f4 }
```

where:

ZONE *f1*	Zone factor (0.7, 0.8, 0.9 or 1.0)
CO *f2*	Normal coefficient of shear force (0.2 or 1.0)
TC *f3*	Period defined by ground support specification (0.4, 0.6 or 0.8 sec)
ALPHA *f4*	Ratio of steel height to overall building height, which are used in the calculation of Rt.

Generation of AIJ Seismic Load

General format to provide Japanese Seismic load in a primary load case:

```
LOAD i
```

```
AIJ LOAD {X | Y | Z} (f5) (ACC f6)
```

Where:

LOAD *i*	load case number
AIJ LOAD { X | Y | Z } *f5*	optional factor to multiply horizontal seismic load.
ACC *f6*	multiplying factor for Accidental Torsion, to be used to multiply the AIJ accidental torsion load (default = 1.0). May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces). **Note:** Choose horizontal directions only.

If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the AIJ specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE AIJ LOAD` command.

Program calculates the natural period of building *T* utilizing the following equation:

$$T = h(0.02 + 0.01\alpha)$$

where

h	=	height of building (m)
a	=	Alpha, ratio of steel height for overall height

Design spectral coefficient, R_t, is calculated utilizing T and T_c as follows:

$$R_t = \begin{cases} 1.0 \text{ when } T < T_c \\ 1 - 0.2(T / T_c - 1)^2 \text{ when } T_c \le T \le 2T_c \\ 1.6T_c / T \text{ when } 2T_c < T \end{cases}$$

where

T_c = Period defined by ground support specification for Rt.

a_i is calculated from the weight values provided in the Define AIJ Load command.

$$\alpha_i = W_i / W$$

where

W_i = sum of weight from a top to floor i.
W = all weight

The seismic coefficient of floor, C_i, is calculated using:

$$C_i = Z R_t A_i C_o$$

where

Z = zone factor
C_o = normal coefficient of shear force

$$A_i = 1 + \frac{2T\left(1/\sqrt{a_i} - a_i\right)}{1 + 3T}$$

Seismic load shear force, Q_i, of each floor is calculated by C_i and W_i:

$$Q_i = C_i W_i$$

where

W_i = sum of weight from a top to i floor

Load value of each floor Pi is calculated by seismic load shear force Qi.

$$P_i = Q_i - Q_{i+1}$$

The total lateral seismic load is distributed by the program among different levels.

Example

```
DEFINE AIJ LOAD
ZONE 0.8 CO 0.2 TC 0.6 ALPHA 1.0
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X)
AIJ LOAD X
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.19 CFE (Comisión Federal De Electricidad) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Manuel de Diseño por Sismo - Seismic Design Handbook Comisión Federal De Electricidad - Electric Power Federal Comission - October 1993 (Chapters 3.1, 3.2, 3.3 and 3.4) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). This is a code used in the country of Mexico.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE CFE (ACCIDENTAL) LOAD
```

```
cfe-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
cfe-spec = { ZONE f1 QX f2 QZ f3 GROUP f4 STYP f5 (REGULAR) (TS f6) (PX f7) (PZ f8) }
```

ZONE *f1*	Zone number specified in number such as 1, 2, 3 or 4
QX *f2*	seismic behavior factor of the structure along X direction as a parameter according 3.2.4.
QZ *f3*	seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.
GROUP *f4*	Group of structure entered as A or B
STYPE *f5*	Soil type entered as 1 or 2 or 3
TS *f6*	site characteristic period
PX *f7*	Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method
PZ *f8*	Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

The optional parameter `REGULAR` is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.

Generation of CFE Seismic Load

To provide a CFE Seismic load in any load case:

```
LOAD i
```

```
CFE LOAD {X | Y | Z} (f)
```

Where:

LOAD *i*	load case number
CFE LOAD { X | Y | Z | *f*	factor to multiply horizontal seismic load. Choose horizontal directions only

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the `DEFINE CFE LOAD` command.

Program calculates the natural period of building T utilizing Rayleigh-Quotient method. If time period is provided in the input file, that is used in stead of calculated period.

The acceleration a is calculated according to the following:

$$a = \begin{cases} a_0 + \left(c - a_0\right)\frac{T}{T_a} \text{when} T < T_a \\ c \text{when} T_a \le T \le T_b \\ c\left(T_b / T\right)^r \text{when} T_b < T \end{cases}$$

Where:

c = Seismic coefficient is extracted from table 3.1
a_0, T_a, T_b, and r are obtained form table 3.1

The ductility reduction factor Q' is calculated according to section 3.2.5.

$$Q' = \begin{cases} Q \text{when} T \ge T_a \\ 1 + \left(\frac{T}{T_a}\right)\left(Q - 1\right) \text{when} T < T_a \end{cases}$$

If not regular, then $Q' = Q' x\ 0.8$

If the period T_s of the soil is known and the soil type II or III T_a and T_b will be modified according to section 3.3.2.

Lateral loads for each direction are calculated for:

When $T \le T_b$, Eq. 4.5. Section 3.4.4.2 is used:

$$P_n = W_n h_n \frac{\sum_{n=1}^{N}\left(W_n\right)a}{\sum_{n=1}^{N}\left(W_n h_n\right)Q'}$$

When $T > T_b$, Eq. 4.6/7/8. Section 3.4.4.2 is used:

$$P_n = W_n\left(a / Q\right)\left(K_1 h_i + K_2 h_i^2\right)$$

Where:

$$K_1 = \frac{q\left[1 - r\left(1 - q\right)\right]\Sigma W_i}{\Sigma\left(W_i / h_i\right)}$$

$$K_2 = \frac{1.5rq\left(1 - q\right)\Sigma W_i}{\Sigma\left(W_i / h_i^2\right)}$$

q =

The base shear are distributed proportionally to the height if $T \le T_b$ or with the quadratic equation mentioned if $T > T_b$.

The distributed base shears are subsequently applied as lateral loads on the structure.

Example

```
UNIT KGS METER
DEFINE CFE LOAD
ZONE 2 QX .5 QZ 0.9 STYP 2 GROUP B TS 0.2
SELFWEIGHT
MEMBER WEIGHT
1 TO 36 41 TO 50 UNI 300
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
FLOOR WEIGHT
YRA 11.8 12.2 FLOAD 400 -
XRA -1 11 ZRA -1 21
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
CFE  LOAD  X 1.0
LOAD 2 ( SEISMIC LOAD IN -Z DIRECTION )
CFE  LOAD  Z -1.0
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.20 NTC (Normas Técnicas Complementarias) Seismic Load

The purpose of this command is to define and generate static equivalent seismic loads as per Code of the México Federal District (Reglamento de Construcciones del Distrito Federal de México) and Complementary Technical Standards for Seismic Design (y Normas Técnicas Complementarias (NTC) para Diseño por Sismo -Nov. 1987) (Chapters 8.1 8.2 8.6 and 8.8) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE NTC LOAD
```

`ntc-spec`

`weight-data`

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
ntc-spec = { ZONE f1 QX f2 QZ f3 GROUP f4 (SHADOWED) (REGULAR) (REDUCE) (PX f5) (PZ f6) }
```

where:

ZONE *f1*	Zone number specified in number such as 1, 2, 3 or 4
QX *f2*	seismic behavior factor of the structure along X direction as a parameter according 3.2.4.

QZ *f3* seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.

GROUP *f4* Group of structure entered as A or B

PX *f5* Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

PZ *f6* Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value calculated by the program using Rayleigh-Quotient method

`REGULAR` is an optional parameter which is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.

`SHADOWED` is an optional parameter which is used to define the shaded zone II as the site of the structure. By default regular zone II is used.

`REDUCE` is an optional parameter which allows to reduce the seismic factors as described above. Otherwise the following formula is used to calculate base shear, $V = \frac{c}{Q'} \sum_{n=1}^{N} W_n$

Generation of NTC Seismic Load

To provide NTC Seismic load in any load case:

```
LOAD i
```

```
NTC LOAD {X/Y/Z} (f)
```

where:

LOAD *i*	load case number
NTC LOAD { X | Y | Z } *f*	factor to multiply horizontal seismic load

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The design base shear is computed in accordance with Sections 8.1 or 8.2 of the NTC as decided by the user.

A. Base Shear is given as

$$V_o / W_o = c / Q$$

where

c = Seismic Coefficient, which is obtained by the program from the following table

Table 255: Seismic coefficient per NTC

Seismic Coefficient, c	Group A	Group B
I	0.24	0.16
II not shaded	0.48	0.32

Seismic Coefficient, c	Group A	Group B
III (and II where shaded)	0.60	0.40

Q = is entered by the user as a parameter

B. Base shear is given as

Vo / Wo = a / Q'

Where Reduction of Shear Forces are requested

Time Period T of the structure is:calculated by the program based on using Rayleigh quotient technique.

you may override the period that the program calculates by specifying these in the input

a and Q' are calculated according to the sections 3 and 4 of the NTC, that is to say:

$$a = \begin{array}{l} \left(1 + 3\frac{T}{T_a}\right)\frac{c}{4} \text{when} T < T_a \\ c \text{when} T_a \le T \le T_b \\ q \cdot c \text{when} T_b < T \end{array}$$

Where:

$q = (T_b/T)^r$

$$Q' = \begin{array}{l} Q \text{when} T \ge T_a \\ 1 + \left(\frac{T}{T_a}\right)\left(Q - 1\right) \text{when} T < T_a \end{array}$$

If not regular, then $Q' = Q' \times 0.8$

T_a, T_b and r are taken from table 5-13 (Table 3.1 in the NTC).

Table 256: Values of T_a, T_b and r per NTC

Zone	T_a	T_b	r
I	0.2	0.6	1/2
II not shaded	0.3	1.5	2/3
III (and II where shaded)	0.6	3.9	1.0

a shall not be less than c/4

V_o for each direction is calculated:

$$V_o = \begin{array}{l} W_o a / Q' \text{when} T \le T_b \\ \frac{\Sigma W_i a}{Q'\left(K_1 h_i + K_2 h_i^2\right)} \text{when} T > T_b \end{array}$$

Where:

$$K_1 = \frac{q\left[1 - r\left(1 - q\right)\right]\Sigma W_i}{\Sigma\left(W_i / h_i\right)}$$

$$K_2 = \frac{1.5rq\left(1 - q\right)\Sigma W_i}{\Sigma\left(W_i / h_i^2\right)}$$

W_i and h_i the weight and the height of the i^{th} mass over the soil or embedment level.

The base shear are distributed proportionally to the height if $T \le T_b$ or with the quadratic equation mentioned if $T > T_b$. The distributed base shears are subsequently applied as lateral loads on the structure.

Example

```
UNIT KGS METER
DEFINE NTC LOAD
ZONE 2 QX .5 QZ 0.9 GROUP B
SELFWEIGHT
ELEMENT  WEIGHT
1577 TO 1619 PRESSURE 275
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
NTC  LOAD  X 1.0
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
NTC  LOAD  Z 1.0
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.21 Turkish Seismic Code

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per the specifications laid out in *Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention* Amended on 2.7.1998, Official Gazette No. 23390 (English Translation). This is referred to as the Turkish Seismic Provisions.

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

STAAD utilizes the following format to generate the lateral seismic loads.

```
DEFINE TURKISH LOAD
```

```
tur-spec
```

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
tur-spec = { A f1 TA f2 TB f3 I f4 RX f5 RZ f6 (CT f7) (PX f8) (PZ f9) }
```

A *f1*	Effective Ground Acceleration Coefficient, Ao. Refer table 6.2

TA *f2*	Spectrum Characteristic Periods, TA and TB. These are user input and found in Table 6.4
TB *f3*	Spectrum Characteristic Periods, TA and TB. These are user input and found in Table 6.4
I *f4*	the earthquake importance factor of the structure.
RX *f5*	Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5
RZ *f6*	Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5
CT *f7*	Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.
PX *f8*	Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.
PZ *f9*	Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.
WEIGHT *w*	joint weight associated with list
UNI *v1 v2 v3*	Used when specifying a uniformly distributed load with a value of *v1* starting at a distance of *v2* from the start of the member and ending at a distance of *v3* from the start of the member. If *v2* and *v3* are omitted, the load is assumed to cover the entire length of the member.
CON *v4 v5*	Used when specifying a concentrated force with a value of *v4* applied at a distance of *v5* from the start of the member. If *v5* is omitted, the load is assumed to act at the center of the member.
PRESSURE *p1*	weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (See TR.32.4.3 Floor Load Specification (on page 2483) for details).

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR.32.12.2 Generation of Seismic Loads (on page 2598).

Base Shear

The minimum lateral seismic force or base shear, V_t, is automatically calculated per equation 6.4 in section 6.7.1:

$$V_t = \frac{WA(T_1)}{R_a(T_1)}$$

where

T_1 = the fundamental time period of the structure

Except that V_t shall not be less than:

$$V_{t,\min} = 0.10 A_0 I W$$

Seismic Load Reduction Factor, R_a (T_1), in above equation is determined based on the following equations 6.3a and 6.3b in the code:

$$R_a(T_1) = \begin{cases} 1.5 + (R - 1.5)\dfrac{T_1}{T_a} \text{when} 0 \le T_1 \le T_a \\ R \text{when} T_1 > T_a \end{cases}$$

The Structural Behavior Factor in either direction, RX and RZ, are provided through user input (variables *f5* and *f5*) along the direction of calculation. Spectrum Characteristics Period, TA and TB, are also provided by user through the parameters (variables *f2* and *f3*).

where

R_a = the Structural Behavior Factor in either direction, RX and RZ, are provided through user input (variables *f5* and *f5*) along the direction of calculation

T_a, T_b = the Spectrum Characteristics Period are provided by user through the parameters (variables *f2* and *f3*)

T_1 = the fundamental Lateral period in the direction under consideration and is determined as:

a. Calculated by the empirical formulae as described below provided hn is in meter:

$$T_1 = C_T \, [hn]^{3/4}$$

Where C_T is assumed to be 0.075 for steel moment frames, 0.085 for concrete moment frames, or any user specified value.

b. The period is also calculated in accordance with the Rayleigh method but could be overridden by user specified time period (PX, PZ).

The time period calculated based on method (a) is used in further calculation unless it is greater than 1.0 sec and 1.3 times of this is greater than the same calculated based on method (b). In that case time period calculated based on method (b) is used.

$A(T_1)$ = the Spectral Acceleration Coefficient is determined as follows as per eq. 6.1,

$A(T_1) = Ao \, I \, S(T_1)$

A_o and I = in above equation are Effective Ground Acceleration Coefficient and Building Importance Factor are provided by the user through the load definition parameter and could be found in table 6.2 and 6.3 respectively in the code

$S(T_1)$ = the Spectrum Coefficient is found by following equations, could be found in eq. 6.2a, 6.2b and 6.3c in original code

$$S(T_1) = \begin{cases} 1 + 1.5\dfrac{T_1}{T_A} \text{when} 0 \le T < T_A \\ 2.5 \text{when} T_A \le T \le T_B \\ 2.5\left(\dfrac{T_B}{T_1}\right)^{0.8} \text{when} T_B < T \end{cases}$$

W = the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^{n} W_i$$

W_i	=	the portion of W that is located at or assigned to level i

Vertical Distribution

As per 4.1.8.11(6), the total lateral seismic force, Vt, shall be distributed such that a portion, Ft shall be concentrated at the top of the building, where,

$$\Delta F_N = 0.07\, T_1\, V_t$$

but ΔF_N is not greater than $0.20 V_t$
and $\Delta F_N = 0$ when $H_N \le 25$ m.

The remainder (V- ΔF_N), shall be distributed along the height of the building, including the top level, in accordance with the following formula (per equation 6.9:

$$Fi = (Vt - \Delta FN)\ wi\ Hi / \Sigma\ wj\ Hj$$

where

F_i	=	the lateral force applied to level i
ΔF_N	=	the portion of Vt to be concentrated at the top of the structure
Wi, Wj	=	the portion of W that is located at or assigned to level i or j respectively
i	=	level i is any level in the building, i = 1 for first level above the base.
N	=	level N is uppermost in the main portion of the structure

Example

```
DEFINE TUR LOAD
A 0.40 TA 0.10 TB 0.30 I 1.4 RX 3.0 RZ 3.0
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
TUR LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.22 UBC 1994 or 1985 Load Definition

This set of commands may be used to define the parameters for generation of UBC-type equivalent static lateral loads for seismic analysis. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

General Format

```
DEFINE UBC (ACCIDENTAL) LOAD
{ ubc-1994-spec | ubc-1985-spec }
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Where:

```
ubc-1994-spec  = { ZONE f1 I f2 RWX f3 RWZ f4 S f5 (CT f8) (PX f9) (PZ f10) }
```

```
ubc-1985-spec = { ZONE f1 K f6 I f2 (TS f7) }
```

Where:

ZONE *f1*	The seismic zone coefficient (0.2, 0.3 etc.). Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.
I *f2*	The importance factor. Used for both 1985 and 1994 specifications.
RWX *f3*	(UBC 1994 spec only) Numerical co-efficient R_w for lateral load in Z-directions
RWZ *f4*	(UBC 1994 spec only) Numerical co-efficient R_w for lateral load in Z-directions
S *f5*	(UBC 1994 spec only) Site co-efficient for soil characteristics
K *f6*	(UBC 1985 spec only) Horizontal force factor
TS *f7*	(UBC 1985 spec only) Period of structure (in seconds) in the X- direction.
CT *f8*	(UBC 1994 spec only) Value of the term C_t which appears in the equation of the period of the structure per Method A. See note f (on page 2431) for details.
PX *f9*	(UBC 1994 spec only) Period of structure (in seconds) in the X direction.
PZ *f10*	(UBC 1994 spec only) Period of structure (in seconds) in the Z direction. Used for Y if the `SET Z UP` command is used.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Notes

a. If the option `ACCIDENTAL` is used, the accidental torsion will be calculated per UBC specifications. The value of the accidental torsion is based on the center of mass for each level. The center of mass is calculated from the `SELFWEIGHT`, `JOINT WEIGHT`, `MEMBER WEIGHT`, `ELEMENT WEIGHT`, `FLOOR WEIGHT`, and `ONEWAY WEIGHT` commands you have specified.

b. In *ubc-spec* for 1985 code, specification of TS is optional. If TS is specified, resonance co-efficient S is determined from the building period T and user provided TS using UBC equations. If TS is not specified, the default value of 0.5 is assumed.

c. By providing either PX or PZ or both, you may override the period calculated by STAAD for Method B of the UBC Code. The user defined value will then be used instead of the one recommended by UBC per equation 28.5 of UBC 94. If you do not define PX or PZ, the period for Method B will be calculated by the program per equation 28.5.

d. Some of the items in the output for the UBC analysis are explained below.

CALC / USED PERIOD

The CALC PERIOD is the period calculated using the Rayleigh method (Method B as per UBC code). For UBC in the x-direction, the USED PERIOD is PX. For the UBC in the z-direction, the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one substituted into the critical equation of the UBC code to calculate the value of C.

- **e.** In the analysis for UBC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.
- **f.** If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10,000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. Ct is in units of seconds/feet$^{3/4}$ or in units of seconds/meter$^{3/4}$. Ct < 0.42 if the units are in feet, and Ct > 0.42 if the units are in meter.

Philosophy

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate UBC equations (All symbols and notations are per UBC).

UBC 1994: Equation 1

$$V = \frac{ZIC}{Rw} W$$

UBC 1984: Equation 2

$$V = ZIK \cdot C_S \cdot W$$

Base shear V may be calculated by STAAD using either the 1994 procedure (equation 1) or the 1985 procedure (equation 2). The user should use the appropriate "ubc-spec" (see General Format (on page 2429)) to instruct the program accordingly.

Procedure Used by the Program

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. You must specify seismic zone co-efficient and desired *`ubc-spec`* (1985 or 1994) following the `DEFINE UBC LOAD` command.
2. Program calculates the structure period T.
3. Program calculates C from appropriate UBC equation(s) utilizing T.
4. Program calculates V from appropriate equation(s). W is obtained from `SELFWEIGHT`, `JOINT WEIGHT`, `MEMBER WEIGHT`, `ELEMENT WEIGHT`, `FLOOR WEIGHT`, and `ONEWAY WEIGHT` commands specified following the `DEFINE UBC LOAD` command. The weight data must be in the order shown.

   Note: If both mass table data (`SELFWEIGHT`, `JOINT WEIGHT`, `MEMBER WEIGHT`, etc. options) and a `REFERENCE LOAD` are specified, these will be added algebraically for a combined mass.

5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per UBC procedures.

Example

```
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
```

```
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
FLOOR WEIGHT
_SLAB1 FLOAD 0.045
ONEWAY LOAD
_ROOF ONE 0.035 GY
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.2.23 UBC 1997 Load Definition

This feature enables one to generate horizontal seismic loads per the UBC 97 specifications using a static equivalent approach. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

General Format

```
DEFINE UBC (ACCIDENTAL) LOAD
```

```
ubc-1997-spec
seismic-weights
```

Where:

```
ubc-1997-spec = { ZONE f1 I f2 RWX f3 RWZ f4 STYPE f5 NA f6 NV f7 (CT f8) (PX f9) (PZ f10)}
```

Where:

ZONE *f1* Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.

I *f2* Importance factor

RWX *f3* Numerical coefficient R for lateral load in X direction

RWZ *f4* Numerical coefficient R for lateral load in Z direction

STYPE *f5* Soil Profile type. Valid range of values are integers 1 through 5. These are related to the values shown in Table 16-J of the UBC 1997 code in the following manner:

1. S_A
2. S_B
3. S_C
4. S_D

5. S_E

Note: The soil profile type S_F is not supported.

NA *f6* Near source factor Na

NV *f7* Near source factor Nv

CT *f8* Optional CT value to calculate time period based on Method A (see Note 7 (on page 2434))

PX *f9* Optional Period of structure (in sec) in X-direction to be used in Method B

PZ *f10* Optional Period of structure (in sec) in Z-direction to be used in Method B

The seismic zone factor (`ZONE`) in conjunction with the soil profile type (`STYPE`), Near source factor (`NA`), and the Near source factor (`NV`), is used to determine the values of seismic coefficients Ca and Cv from Tables 16-Q and 16-R of the UBC 1997 code.

If the `ACCIDENTAL` option is specified, the accidental torsion will be calculated per the UBC specifications. The value of the accidental torsion is based on the “center of mass” for each level. The “center of mass” is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs you have specified.

seismic-weights =

```
weight-data
```

Refer to Common Weight Data (on page 2351) for information on how to specify structure weight for seismic loads.

Note: For additional details on the application of a seismic load definition used to generate loads, refer to TR. 32.12.2 Generation of Seismic Loads (on page 2598).

Methodology

The design base shear is computed in accordance with Section 1630.2.1 of the UBC 1997 code. The primary equation, namely, 30-4 of UBC 1997, as shown below, is checked.

$$V = C_v I/(RT) \cdot W$$

In addition, the following equations are checked :

Equation 30-5 – The total design base shear shall not exceed

$$V = 2.5 \cdot C_a I/R \cdot W$$

Equation 30-6 - The total design base shear shall not be less than

$$V = 0.11 \cdot C_a IW$$

Equation 30-7 – In addition, for Seismic Zone 4, the total base shear shall also not be less than

$$V = 0.8 \cdot ZN_v I/R \cdot W$$

For an explanation of the terms used in the above equations, please refer to the UBC 1997 code.

There are two stages of command specification for generating lateral loads. This is the first stage and is activated through the `DEFINE UBC LOAD` command.

Procedure Used by the Program

Steps to calculate base shear are as follows:

1. Time Period of the structure is calculated based on clause 1630.2.2.1 (Method A) and 1630.2.2.2 (Method B).
2. You may override the period that the program calculates using Method B by specifying a value for PX or PZ (Items f9 and f10) depending on the direction of the UBC load. The specified value will be used in place of the one calculated using Method B.
3. The governing Time Period of the structure is then chosen between the above-mentioned two periods on the basis of the guidance provided in clause 1630.2.2.2.
4. From Table 16-Q and 16-R, Ca and Cv coefficients are calculated.
5. The Design Base Shear is calculated based on clause 1630.2.1 and distributed at each floor using the rules of clause 1630.5.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the UBC load (clause 1630.6). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10,000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. Ct is in units of seconds/feet$^{3/4}$ or in units of seconds/meter$^{3/4}$. Ct < 0.42 if the units are in feet, and Ct > 0.42 if the units are in meter.
8. Due to the abstractness of the expression “Height above foundation,” in STAAD, height, h, is measured above supports. If supports are staggered all over the vertical elevations of the structure, it is not possible to calculate "h" if one doesn't have a clear elevation level from where to measure "h". Also, the code deals with distributing the forces only on regions above the foundation. If there are lumped weights below the foundation, it is not clear as to how one should determine the lateral forces for those regions.

Example 1

```
DEFINE UBC LOAD
ZONE 0.38 I 1.0 STYP 2 RWX 5.6 RWZ 5.6 NA 1.3 NV 1.6 CT 0.037
SELFWEIGHT
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
MEMBER WEIGHT
12 17 24 UNI 25.7
FLOOR WEIGHT
YRA 9 11 FLOAD 200 XRA -1 21 ZR -1 41
ELEMENT WEIGHT
234 TO 432 PR 150
```

Example 2

The following example shows the commands required to enable the program to generate the lateral loads. See TR.32.12 Generation of Loads (on page 2596) for this information.

```
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
UBC  LOAD  X 0.75
```

```
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
UBC LOAD  Z 0.75
```

The UBC / IBC input can be provided in two or more lines using the continuation mark (hyphen) as shown in the following example :

```
DEFINE UBC ACCIDENTAL LOAD
ZONE 3.000 -
I 1.00 RWX 1.100 -
RWZ 1.200 STYP 5.000 NA 1.40 NV 1.50 CT -
1.300 PX 2.100 PZ 2.200
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)

TR.31.3 Definition of Wind Load

This set of commands may be used to define some of the parameters for generation of wind loads on the structure. See TR.32.12 Generation of Loads (on page 2596) for the definition of wind direction and the possible surfaces to be loaded. G.16.3 Wind Load Generator (on page 2136) describes the two types of structures on which this load generation can be performed.

The general wind load generator can be used to generate lateral loads in the horizontal –X and Z (or Y if Z up)–directions only.

Tip: The graphical user interface can be used to automatically generate the appropriate intensity values via the ASCE 7 Wind Load dialog box (on page 2882). See Persistence of Parameters used to Generate ASCE Wind Loads (on page 2441)

General Format

```
DEFINE WIND LOAD
TYPE j  ( optional_comment  )
{ intensity-definition  | code-parameters  }
EXPOSURE e1 { JOINT joint-list | YRANGE f1 f2 | ZRANGE f1 f2 }
```

Repeat EXPOSURE command up to 98 times.

Where:

Parameter	Description
TYPE *j* *optional_comment*	wind load system type number (integer) The optional comment is a text string comment or description used to help identify the wind load type.
intensity-definition or *code-parameters*	data is entered based on either custom or Russian code wind definitions. See Wind Intensity Definition (on page 2436) or See Russian Wind Loads (on page 2438)

Parameter	Description
EXPOSURE *e1*, *e2* … *em*	exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors. If the command `EXPOSURE` is not specified or if a joint is not listed in an Exposure, the exposure factor for those joints is chosen as 1.0.
JOINT *joint-list*	Joint list associated with Exposure Factor (joint numbers or `TO` or `BY`) or enter only a group name.
YRANGE | ZRANGE *f1 f2*	global coordinate values to specify Y (or Z if Z UP) vertical range for Exposure Factor. Use `YRANGE` when Y is Up and `ZRANGE` when Z is Up (See the `SET Z UP` command in TR.5 Set Command Specification (on page 2206)).

Wind Intensity Definition

For custom (including for ASCE 7) wind load definitions, the wind intensity at heights above ground are defined as follows:

```
intensity-definition  = INTENSITY p1 p2 p3 … pn HEIGHT h1 h2 h3 … hn
```

Note: These values are automatically generated for ASCE 7 wind loads when the ASCE-7: Wind Load dialog is used.

Where:

Parameter	Description
p_1,p_2,p_3… p_n	wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.
h_1,h_2,h_3… h_n	corresponding heights in global vertical direction, measured in terms of actual Y (or Z for Z UP) coordinates up to which the corresponding intensities occur.

All intensities and heights are in current unit system. The heights specified are in terms of actual Y coordinate (or Z coordinates for Z UP) and not measured relative to the base of the structure. The first value of intensity (p_1) will be applied to any part of the structure for which the Y coordinate (or Z coordinate for Z UP) is equal to or less than h_1. The second intensity (p_2) will be applied to any part of the structure that has vertical coordinates between the first two heights (h_1 and h_2) and so on. Any part of the structure that has vertical coordinates greater than h_n will be loaded with intensity p_n.

Only exposed surfaces bounded by members (not by plates or solids) will be used. The joint influence areas are computed based on surface member selection data entered in TR.32.12.3 Generation of Wind Loads (on page

2603) and based on the wind direction for a load case. Only joints actually exposed to the wind and connected to members will be loaded. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Exposure factor (e) is the fraction of the influence area associated with the joint(s) on which the load may act if it is also exposed to the wind load. Total load on a particular joint is calculated as follows.

$$\text{Joint load} = (\text{Exposure Factor}) \times (\text{Influence Area}) \times (\text{Wind Intensity})$$

The exposure factor may be specified by a joint-list or by giving a vertical range within which all joints will have the same exposure. If an exposure factor is not entered or not specified for a joint, then it defaults to 1.0 for those joints; in which case the entire influence area associated with the joint(s) will be considered.

For load generation on a closed type structure defined as a PLANE FRAME, influence area for each joint is calculated considering unit width perpendicular to the plane of the structure. You can accommodate the actual width by incorporating it in the Exposure Factor as follows.

$$\text{Exposure Factor (User Specified)} = (\text{Fraction of influence area}) \times (\text{influence width for joint})$$

Notes:

a. All intensities, heights and ranges must be provided in the current unit system.

b. If necessary, the `INTENSITY` and `EXPOSURE` command lines can be continued on to additional lines by ending all but last line with a space and hyphen (-). Use up to 11 lines for a command.

Example

```
UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOITN 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
```

Note: For additional examples, see TR.32.12 Generation of Loads (on page 2596) and EX. US-15 Wind and Floor Load Generation on a Space Frame (on page 6423).

The Intensity line can be continued in up to 12 lines.

So the following

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 HEIG
15 20 25 30 40 50 60 70 80 90 100 120
```

could be split as

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 –
HEIG 15 20 25 30 40 50 60 70 80 90 100 120
```

or

```
INT 0.008 0.009 0.009 0.009 0.01 0.01-
0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20 25 -
30 40 50 60 70 80 90 100 120
```

etc.

Russian Wind Loads

This specifies the definition of a wind load to the Russian wind code which will need to be referenced in a wind load command included in a primary load case. For wind loads per Russian codes SNiP 85 or SP20 2016, the code parameters are defined as follows:

```
code-parameters  =  SNIP 1985 PRESSURE f3 TERRAIN { A | B | C }
```

or

```
code-parameters  = { SP20 } PRESSURE f3 TERRAIN { A | B | C } REGION f4 LOG f5
```

Where:

`SNIP 1985` — design according to previous design Code SNiP 2.01.07-85
`SP20 2016` — design according to renewed design Code SP 20.13330.2016

Parameter	Description
PRESSURE *f3*	the characteristic value of wind pressure, always positive
TERRAIN	terrain roughness category: **A.** Coastal Zone. **B.** Urban Zone. **C.** Large City.
REGION *f4*	Wind region as per clause 11.5 of SNiP 2.01.07-85* 2016. This is used to determine the wind pressure in determine the dynamic wind component. 0 = region 1a 1 = region 1 2 = region 2 3 = region 3 4 = region 4 5 = region 5 6 = region 6 7 = region 7

Parameter	Description
LOG *f5*	logarithmic decrement of oscillations, Delta (see table 11.5 - Section 11.1.8 for the definition). Typically values are: 0.15 for steel towers, masts, lined chimneys, column means including the ones on the reinforced concrete pedestals 0.3 for reinforced concrete and stone structures as well as for buildings with steel framework if there are walling structures

Notes:

a. A structured modeled as a set of vertical members –such as is typically used to define a cylindrical stack or chimney structure, and thus doe snot define a closed panel such as defined by a frame formed from a number of columns and beams– is considered to be a "Stick Structure." Refer to TR.32.12.3 Generation of Wind Loads (on page 2603) for additional details.

b. The loading has two components: static and dynamic. The dynamic effect is determined by the number of modes included in the dynamic load case and changing the number of modes considered using the `CUT OFF MODE SHAPE` command may result in a change in the resulting wind force. Refer to TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) for details.

   As the analysis will requires extraction of eigen solutions to determine the dynamic effects of the wind loading, the number of modes to be used will also affect the results. Thus setting the command `CUT OFF FREQUENCY` or `CUT OFF MODE` should be considered and specified as required prior to the definition of the load cases.

   If the cut-off command is omitted, six mode shapes are computed by default.

c. If the `PRINT STATICS CHECK` option is included in the analysis command, then the output file will include a section on the SNiP wind load. This output will indicate both the static and dynamic contributions to the wind load as well as the total applied at each node.

d. This command *cannot* be used with models that have been defined with the `SET Z UP` command.

e. The building classification for wind load definitions per SNiP 2.01.07-85 and SP 20.13330.2016 is limited to prismatic building structures. General type concrete and steel framed structures are not supported.

The General Format for Russian SNiP 1985 Wind Loads (on page 2608) must also be used in conjunction with the SNiP wind load definition.

STAAD.Pro can generate both static and dynamic wind loads per the SP 20.13330.2016 code.

Example

An example with various static wind loads from different directions that use a SNiP 1985 definition:

```
DEFINE WIND LOAD
TYPE 1
SNIP 1985 PRESSURE 0.38 TERRAIN A
EXP 0.5 JOINT 1 3 5 7 9 11
*
LOAD 1 LOADTYPE WIND TITLE Wind load in the +ve X direction
```

```
WIND LOAD X 1 CONFIG 0 NU 1 TYPE 1
* Mass model required in first wind load case
JOINT LOAD
3 TO 6 FZ 62.223
9 TO 12 FZ 62.223
*
LOAD 2 LOADTYPE WIND TITLE Wind load in the –ve X direction
WIND LOAD X -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 3 LOADTYPE WIND TITLE Wind load in the +ve Z direction
WIND LOAD Z 1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 4 LOADTYPE WIND TITLE Wind load in the –ve Z direction
WIND LOAD Z -1 CONFIG 0 NU 1 TYPE 1
* No mass model or additional loads in this load case
*
LOAD 10 LOADTYPE DEAD TITLE Selfweight load case
SELF Y -1 ALL
*
LOAD COMBINATION 100 Wind plus selfweight
1 1.0 10 1.0
```

The following example uses SP 20.13330.2016 with both static and dynamic load cases:

```
DEFINE WIND LOAD
TYPE 1
SP20 2016 PRESSURE 0.38 TERRAIN A REGION 0 LOG 0.3
*
* Reference Mass Definition for Modal Analysis
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass TITLE REF LOAD CASE 1
JOINT LOAD
3 TO 6 FX 62.223
3 TO 6 FZ 62.223
9 TO 12 FX 62.223
9 TO 12 FZ 62.223
END DEFINE REFERENCE LOADS
*
* Request mode shapes
CUT OFF MODE SHAPE 3
*
* Static Joint Load Required for Dynamic Wind Load
LOAD 1 LOADTYPE None TITLE LOAD CASE 1
JOINT LOAD
3 5 9 11 FX -100
*
* Modal Analysis
LOAD 2 LOADTYPE None TITLE LOAD CASE 2
MODAL CALCULATION REQUESTED
*
* Dynamic Wind Load Command
LOAD 3 LOADTYPE None TITLE SNiP Dynamic
WIND LOAD X 1.0 TYPE 1 DYN 1
PERFORM ANALYSIS PRINT LOAD DATA
```

Persistence of Parameters used to Generate ASCE Wind Loads

In the Analytical Modeling workflow, in the Create Wind Type Definition dialog (on page 2880), you can click the **Calculate as per ASCE-7** button to generate the pressure versus height table per the ASCE 7 wind load specifications per the 1995, 2002, or 2010 editions. The parameters which go into the derivation of this table are not retained by the graphical environment but rather added into the STAAD input file so they may be edited as needed. These values are not read by the STAAD engine directly and, therefore, are not directly processed as a load but are rather used to generate the wind intensity values which are used by the engine. An example of it is shown below.

Example

```
DEFINE WIND LOAD
TYPE 1
This entire section of the input file must not be edited.<! STAAD PRO
GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2002:PARAMS 85.000 MPH 0 1 0 0 0.000 FT 0.000 FT 0.000 FT 1 -
1 40.000 FT 30.000 FT 25.000 FT 2.000 0.010 0 -
0 0 0 0 0.761 1.000 0.870 0.850 0 -
0 0 0 0.866 0.800 0.550
!> END GENERATED DATA BLOCK
INT 0.0111667 0.0111667 0.0113576 0.0115336 0.0116972 0.0118503 0.0119944 -
0.0121307 0.0122601 0.0123834 0.0125012 0.0126141 0.0127226 0.012827
0.0129277 -
HEIG 0 15 16.9231 18.8461 20.7692 22.6923 24.6154 26.5385 28.4615 -
30.3846 32.3077 34.2308 36.1538 38.0769 40
```

Related Links

- *M. To add a SNiP wind load definition* (on page 847)
- *M. To apply a dynamic wind load per SP 20.13330.2016* (on page 856)
- *M. To add a SNiP wind load definition* (on page 847)
- *M. To add an ASCE 7 wind load definition* (on page 845)
- *G.16.3 Wind Load Generator* (on page 2136)
- *M. To add a wind load definition* (on page 844)
- *M. To add a GB50009 wind load definition* (on page 848)
- *G.16.3 Wind Load Generator* (on page 2136)

TR.31.4 Definition of Time History Load

This set of commands may be used to define parameters for time history loading on the structure. The time history data may be specified using either explicit definition, function specification, a spectrum specification, or time history data provided in an external file.

General Format

```
DEFINE TIME HISTORY (DT x) ( MISSING MASS )

TYPE i { ACCELERATION | FORCE |  MOMENT } (SCALE f7) (SAVE)

{ t1 p1 t2 p2 … tn pn | function-spec  | spectrum-spec  | READ filename (f8) }
```

Repeat TYPE and Amplitude vs Time sets until all are entered, then:

```
ARRIVAL TIME
```

```
a1 a2 a3 … an
```

```
{ DAMPING d | CDAMP | MDAMP }
```

Entering the MISSING MASS parameter will include the missing mass procedure in the time history analysis.

The time history data can be explicitly defined using pairs of time and values of acceleration, force, or moment, where:

- ACCELERATION indicates that the time varying load type is a ground motion.
- FORCE indicates that it is a forcing function.
- MOMENT indicates that it is a moment forcing function.

Table 257: Parameters for explicitly defined time history load

Variable or Command	Default Value	Description
DT *x*	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
TYPE *i*	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE *f7*	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: TYPE 1 FORCE SAVE

Variable or Command	Default Value	Description
t_1 p_1 t_2 p_2	-	Values of time (in sec.) and corresponding force (current force unit) or acceleration (current length unit/sec^2) depending on whether the time varying load is a forcing function or a ground motion. If the first point is not at zero time, then the forces before the first time (but after the arrival time) will be determined by extrapolation using the first two points entered. If the first point has a nonzero force, there will be a sudden application of that force over a single integration step (DT) at that time. Zero force will be assumed for all times after the last data point.
a_1 a_2 a_3 ... a_n	-	Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.
DAMPING *d*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

The *function-spec* option can be used to specify harmonic loads. Both sine and cosine harmonic functions may be specified. The program will automatically calculate the harmonic load time history based on the following specifications.

For the function and amplitude option (:

```
function-spec =
{ SINE | COSINE }
AMPLITUDE f0 { FREQUENCY | RPM } f2 (PHASE f3) CYCLES f4 { SUBDIV f5 | STEP f6 }
```

Note: Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (DT) at that time.

Where:

Table 258: Parameters for use with function time history load

Variable or Command	Default Value	Description
DT *x*	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
TYPE *i*	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE *f7*	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with .TIM and .FRC file extensions). The .TIM file contains the history of the displacements of every node. The .FRC file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: `TYPE 1 FORCE SAVE`
AMPLITUTDE *f0*	-	Max. Amplitude of the forcing function in current units.
FREQ | RPM *f2*	-	If FREQUENCY, then cyclic frequency (cycles / sec.) If RPM, then revolutions per minute.
PHASE *f3*	0	Phase Angle in degrees.
CYCLES *f4*	-	No. of cycles of loading.

Variable or Command	Default Value	Description
SUBDIV *f5*	3	Used to subdivide a ¼ cycle into this many integer time steps. **Note:** Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions will make the digitized force curve more closely match a sine wave. The default is usually adequate.
STEP *f6*	(p/12)	Time step of loading. Default is equal to one twelfth of the period corresponding to the frequency of the harmonic loading. (It is best to use the default) **Note:** Only used to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions or smaller step size will make the digitized force curve more closely match a sine wave.
a_1 a_2 a_3 ... a_n	-	Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.
DAMPING *d*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

The *spectrum-spec* option can be used to specify a synthetic ground motion acceleration time history based statistically on a user supplied acceleration spectrum.

The program will automatically calculate the acceleration time history based on the following specifications. Enter *f12*, *f13*, and *f14* to indicate the rise, steady, & decay times, respectively.

For the spectrum option:

```
spectrum-spec =
```

```
SPECTRUM (TMAX f9) (DTI f10) (DAMP f11) (T1 f12) (T2 f13) (T3 f14) (SEED f15)
```

```
OPTIONS NF f16 NITR f17 ( THPRINT f18 ) ( SPRINT f19 ) ( FREQ )
```

Starting on the next line, enter Spectra in the following input form:

```
P1, V1; P2, V2; … ; Pn, Vn
```

Where:

Table 259: Parameters used with the spectrum time history load

Variable or Command	Default Value	Description
DT *x*	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
TYPE *i*	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.
SCALE *f7*	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with `.TIM` and `.FRC` file extensions). The `.TIM` file contains the history of the displacements of every node. The `.FRC` file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: `TYPE 1 FORCE SAVE`
TMAX *f9*	20 seconds	The Max. time (in seconds) in the generated time history. This value must be greater than *f14* (T3).
DTI *f10*	0.2	Delta time step (in seconds) in the generated time history.
DAMP *f11*	0.05	Damping ratio (5% is entered as 0.05) associated with the input spectrum.
T1 *f12*	4 seconds	Ending time of the acceleration rise time. This value must be greater than zero.

Variable or Command	Default Value	Description
T2 *f13*	9 seconds	Ending time of the steady acceleration. This value must be greater than zero and greater than *f12* (T1).
T3 *f14*	14 seconds	Ending time of the acceleration decay. This value must be greater than *f13* (T2).
SEED *f15*		Optional random Seed. Enter a positive integer (in the range of 1 to 2,147,483,647) to be used as a unique random number generation "seed." A unique time history will be produced for each seed value. Change this value when you want to produce a "different (from the time history generated with the prior seed value)" but statistically equivalent time history. Omit this entry to get the default value (normal option).
NF *f16*		The input shock spectrum will be re-digitized at `NF` equally spaced frequencies by interpolation. Default is the greater of 35 or the number of points in the input spectrum.
NITR *f17*	10	The number of iterations which will be used to perfect the computed time history.
THPRINT *f18*	1	Print the time history that is generated. Omit the `THPRINT` parameter to avoid printing. 1 = print beginning 54 values and last 54 values 2 = Print entire curve. >10 = print beginning *f18* values and last *f18* values
SPRINT *f19*	1	Print the spectrum generated from the time history that is generated. Omit the `SPRINT` parameter to avoid printing
FREQ		If entered, then frequency-spectra pairs are entered rather than period-spectra pairs.
P1, *V1* *P2*, *V2* … *Pn*, *Vn*		Data is part of input, immediately following `SPECTRUM` command. Period (or frequency if `FREQ` option entered above) Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Enter the period in seconds (or frequency in Hz.) and the corresponding Value is in acceleration (current length unit/sec^2) units. Continue the curve data onto as many lines as needed (up to 999 spectrum pairs). Spectrum pairs must be in ascending or descending order of period (or frequency). **Note:** If data is in g acceleration units, then set `SCALE` to a conversion factor to the current length unit (9.807, 386.1, etc.). Also note, do not end these lines with a hyphen. Commas and semi-colons are optional.

Variable or Command	Default Value	Description
a_1 a_2 a_3 ... a_n	-	Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.
DAMPING *d*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Note: Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (DT) at that time.

The time history data can also be defined in an external file, where:

Table 260: Parameters for a time history load defined in an external file

Variable or Command	Default Value	Description
DT *x*	-	Solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.
TYPE *i*	-	Type number of time varying load (integer). Up to 136 types may be provided. This number should be sequential.

Variable or Command	Default Value	Description
SCALE *f7*	1.0	The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.).
SAVE	-	The save option results in the creation of two files (input file name with `.TIM` and `.FRC` file extensions). The `.TIM` file contains the history of the displacements of every node. The `.FRC` file contains the history of the 12 end forces of every member of the structure at every time step, and the 6 reactions at each support at every step. Syntax: `TYPE 1 FORCE SAVE`
READ *filename*		Filename for an external file containing time varying load history data.
f8		The optional delta time spacing used for the external file data
a_1 a_2 a_3 ... a_n	-	Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593)). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to TR.32.10.2 Time Varying Load for Response History Analysis (on page 2593) for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.

Variable or Command	Default Value	Description
DAMPING *d*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Examples

Using Force and Acceleration options:

```
UNIT …
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
TYPE 2 ACCELERATION SCALE 9.80665
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Using the Spectrum option:

```
UNIT …
DEFINE TIME HISTORY
TYPE 1 ACCELERATION SCALE 9.80665
SPECTRUM TMAX 19 DTI 0.01 DAMP 0.03
OPTIONS NF 40
0.03 1.00 ; 0.05 1.35
0.1 1.95 ; 0.2 2.80
0.5 2.80 ; 1.0 1.60
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Using the Harmonic Loading Generator:

```
UNIT …
DEFINE TIME HISTORY
TYPE 1 FORCE
*Following lines for Harmonic Loading Generator
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100 STEP 0.02
ARRIVAL TIME
0.0
DAMPING 0.075
```

To define more than one sinusoidal load, the input specification is as follows:

```
DEFINE TIME HISTORY
TYPE 1 FORCE
FUNCTION SINE
AMPLITUDE 1.925 RPM 10794.0 CYCLES 1000
TYPE 2 FORCE
FUNCTION SINE
AMPLITUDE 1.511 RPM 9794.0 CYCLES 1000
TYPE 3 FORCE
FUNCTION SINE
AMPLITUDE 1.488 RPM 1785.0 CYCLES 1000
ARRIVAL TIME
0.0 0.0013897 0.0084034
DAMPING 0.04
```

The data in the external file must be provided as one or more time-force pairs per line as shown in the following example.

Data in Input file

```
UNIT …
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
0.0
DAMPING 0.075
```

Data in the External file `THFILE`:

```
0.0 1.0 1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6
```

Notes

a. By default the response (displacements, forces etc.) will contain the contribution of only those modes whose frequency is less than or equal to 108 cps. Use the `CUT OFF FREQUENCY` command to change this limit. Contributions of modes with frequency greater than the Cut Off Frequency are not considered.

b. Results are the individual maximums over the time period. Thus, derived quantities such as section forces and stresses, plate surface stresses and principal stresses should not be used.

c. Results from harmonic input are the maximum over the time period including the start-up transient period. These results are not the steady-state results.

d. By default, the results do not include the time period after the time loads end. Use the `CUT OFF TIME` command to lengthen (or shorten) the time period. If an intense short-term loading is used, the loading should be continued until after the expected peak response is reached.

e. The `READ` *`filename`* command is to be provided only if the history of the time varying load is to be read from an external file. *`filename`* is the file name and may be up to 72 characters long. If the data on the file consists only of amplitudes, then enter *`f8`* as the delta time spacing.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *M. To define a time history type from tabular data* (on page 875)
- *M. To define a time history type from a function* (on page 876)

- *M. To define a time history type by spectrum* (on page 877)
- *M. To define a time history type by external file* (on page 879)
- *M. To generate output for time history spectrum* (on page 878)
- *M. To use frequency-spectra pairs in a time history load* (on page 878)
- *G.17.3.5 Response Time History* (on page 2165)
- *M. To define time history parameters* (on page 880)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *G.17.3.5 Response Time History* (on page 2165)

TR.31.5 Definition of Snow Load

This set of commands may be used to define some of the parameters for generation of snow loads on the structure.

Refer to TR.32.12.4 Generation of Snow Loads (on page 2609) or the definition of additional parameters and the surfaces to be loaded.

General Format

```
DEFINE SNOW LOAD
```

```
TYPE f1 PG f2 CE f3 CT f4 IM f5
```

Where:

TYPE *f1* Type No: (limit of 100). The “Type No.” is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model. Include as many types as needed.

PG *f2* Ground Snow Load (Default = 0.0). The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.

CE *f3* Exposure Factor (Default = 1.0). Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

CT *f4* Thermal Factor (Default = 0.0). Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

IM *f5* Importance Factor (Default = 1.0). Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Example

```
START GROUP DEFINITION
FLOOR
_ROOFSNOW 102 TO 153 159 160 TO 170 179 195 TO 197
END GROUP DEFINITION
UNIT FEET POUND
```

```
DEFINE SNOW LOAD
TYPE 1 PG 50 CE 0.7 CT 1.1 IM 1.1
```

Related Links

- *M. To add an ASCE 7-02 snow load* (on page 870)
- *G.16.4 Snow Load* (on page 2138)

TR.31.6 Defining Reference Load Types

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. Thus Reference Loads may be defined for this purpose. This is similar to a REPEAT LOAD command (See TR.32.11 Repeat Load Specification (on page 2595)), but has the added benefit of not being solved in its own right.

This converts a real load case to something similar to a load case definition. A reference load case is solved only when it is later called in a load case. The benefit is that it enables you to define as many load cases as you wish, but instruct the program to actually solve only a limited number of "real" load cases, thus limiting the amount of results to be examined.

See TR.33 Reference Load Cases - Application (on page 2613) for a description of the procedure for specifying the reference load information in active load cases.

General Format

```
DEFINE REFERENCE LOADS

LOAD R(i) LOADTYPE (type) TITLE load_title

(Load items)

...

END DEFINE REFERENCE LOADS
```

Example

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead TITLE REF DEAD
SELFWEIGHT Y -1
JOINT LOAD
4071 4083 4245 4257 FY -4.04
4090 FY -0.64
ELEMENT LOAD
378 TO 379 406 TO 410 422 TO 426 PR GY -1.44
MEMBER LOAD
5006 TO 5229 UNI GY -0.64
PMEMBER LOAD
1 TRAP GY -0.347 -0.254 35.5 42
LOAD R2 LOADTYPE Live TITLE REF LIVE
JOINT LOAD
4209 FY -6.63
4071 4083 4245 4257 FY -1.71
LOAD R3 LOADTYPE Snow TITLE REF SNOW
JOINT LOAD
4109 FY -8.69
```

```
4071 4083 4245 4257 FY -3.29
LOAD R4 LOADTYPE Soil TITLE REF SOIL
ELEMENT LOAD
1367 TO 1394 1396 1398 1522 1539 TO 1574 -
1575 TRAP JT -0.78 -0.78 -0.719167 -0.719167
LOAD R4 LOADTYPE mass TITLE Mass Model
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEFINE REFERENCE LOADS
```

Mass Modeling Using Reference Loads

A reference load case of type `MASS` can be created which can then be used to define the structure mass used for all dynamic analyses (i.e., seismic, response spectrum, time history, etc.). Some analysis methods require you to create separate weight tables in the form of `SELFWEIGHT`, `MEMBER WEIGHT`, `JOINT WEIGHT`, etc. for each analysis, thus resulting in repetition of the same information. Using a `LOADTYPE MASS` reduces the repetitive data entry and the need for manually creating a weight table.

A mass model using this method is defined once and then used for all dynamic analyses.

If the `LOADTYPE MASS` is missing and no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing. If a mass table is provided in a seismic load definition, response spectrum definition, or time history loading definition, then that mass table will be used for analysis under those loads only instead of a mass model generated by a `REF LOAD TYPE MASS`. However, if no masses are defined within the individual seismic, response spectrum, or time history load definitions, then the program uses the mass reference load case when analyzing that seismic or dynamic load case.

Related Links

- *M. To add a mass model reference load* (on page 900)
- *M. To add mass loads to the mass model reference load* (on page 900)
- *M. To add weight by a reference load to a seismic load definition* (on page 859)
- *M. To create a reference load* (on page 893)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *G.17.3.2 Mass Modeling* (on page 2157)

TR.31.7 Definition of Direct Analysis Members

This set of commands may be used to define the members whose flexural stiffness or axial stiffness is considered to contribute to the lateral stability of the structure.

These parameters are then used by the program to analyze the structure as per guidelines provided in Chapter C of the AISC 360-10/16 specifications (Appendix 7 of AISC 360-05).

Note: The yield strength of the steel is taken from the TR.26.1 Define Material (on page 2303). If the material definition is not provided for a member, then a default value of $F_y = 36\ ksi$ is used.

General Format

```
DEFINE DIRECT
```

```
FLEX (f1) list-spec
```

```
AXIAL list-spec
```

Where:

```
list-spec = * { XR f4 f5 | YR f4 f5 | ZR f4 f5 | MEMBERS mem-list | LIST memb-list | ALL }
```

Parameter	Default Value	Description
FLEX *f1*	1.0	The initial τ_b value (See Chapter C of AISC 360-10/16 or Appendix 7 AISC 360-05). Members listed with FLEX will have their EI factored by 0.80 times τ_b while performing the global solution.
AXIAL	-	Members listed with AXIAL will have their EA factored by 0.80 while performing the global solution. Although reduced stiffness is considered for analysis, the program considers the full value of E for both flexural and axial design.
XR | YR | ZR *f4*, *f5*	-	Upper and lower range values, respectively, used when specifying a range for FLEX.

For specifying NOTIONAL LOADs, please see TR.32.13 Notional Loads (on page 2610). The notional loads and the factor used is specified entirely in the loading data.

Notes

τ_b is the value entered in the FLEX command. τ_b defaults to 1.0 if not entered.

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *M. To define direct analysis parameters* (on page 871)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

TR.31.8 Mass Modeling

Each load case consisting of a response spectrum requires a mass definition. The following methods may be used to define masses for use with seismic loads.STAAD.Pro.

TR.31.8.1 Explicitly Defined Weights

This method uses different load items within the load case to define the weights for use with that specific response spectrum. Combinations of selfweight, joint weights, member weights, element weights, and floor weights can defined prior to the response spectrum parameters.

Static Seismic Weight Definition Table

```
seismic-weights =
SELFWEIGHT (f1)
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT
member-list { UNI v1 v2 v3 | CON v4 v5 }
ELEMENT WEIGHT
plate-list PRESS p1
FLOOR WEIGHT
floor-weight-spec
ONEWAY WEIGHT
oneway-weight-spec
REFERENCE LOAD { X | Y | Z }
Ri1 f2
```

Response Spectrum Weight Table

```
SELFWEIGHT { X | Y | Z } f1
JOINT LOADS
joint-loads
MEMBER LOADS
member-loads
```

Example

TR.31.8.2 Reference Load Mass Tables

This method uses loads defined in a reference load case as the weights for the response spectrum. You may also define in which global direction the loading is considered for base shear calculation. By default, the program will consider Global Y if no load direction is specified. If no applied loads are found in the Y direction, then it consider the maximum of the total loads in either X or Z directions.

General Format

Multiple reference load cases can be used in a single response spectrum, each with a separate load factor applied. This allows you to factor the reference loads for use with weights. For example, if a code-prescribed seismic load requires that 100% of the dead load mass but only 50% of the live load mass be used for seismic loads, then you can use factors of 1.0 for the reference dead load case and 0.5 for the reference live load case.

This has the advantage of allowing you to define a single mass model for use with both static seismic loading as well as response spectra. In order to accomplish this, simply define the response spectrum load case as a reference load and then utilize this reference load in both the static seismic and response spectrum primary load case.

Tip: You may also use the same reference load cases for modal analysis and calculating Rayleigh's frequency.

Related Links

- *G.17.3.2 Mass Modeling* (on page 2157)
- *M. To generate a mass model* (on page 899)

TR.31.8.3 Mass Model Using Reference Load

Mass type reference loads can be used to create mass model which will be used for all types of analysis, including seismic, response spectrum, time history, and any other dynamic analysis.

When a rigid floor diaphragm is present in the model, the program follows the following logic:

a. If a reference load type MASS is present, then the mass model is formed by combining all MASS reference loads.
b. If reference load type MASS is not present, then the mass model is formed by combining all gravity reference loads. Warnings are displayed in the analysis output to alert you to this fact.
c. If neither reference load types MASS nor gravity are present, then the mass model is formed by combining all DEAD reference loads. If any LIVE reference loads are present, then they will also be combined to this mass model. Warnings are displayed in the analysis output to alert you to this fact.

Mass model will be formed from Gravity or Dead/Live reference loads in case Mass reference load types are not present, only when rigid floor diaphragm is present in the model.

General Format

```
DEFINE REFERENCE LOADS
LOAD  Ri title
…
Load header and items
…
LOAD  Rj LOADTYPE MASS title
…
Load header and items
…
END DEFINE REFERENCE LOADS
DEFINE UBC/1893/… LOAD
ZONE …
LOAD 1 title
UBC/1893/… LOAD X f1
LOAD 2 title
UBC/1893/… LOAD Z f2
LOAD 3 title
SPECTRUM …
```

The mass model is defined using the LOADTYPE MASS command. This mass model will be used for all types of analysis, including static equivalent, response spectrum, time history, and simply Eigen solution. If the load type

MASS is missing and also no mass is defined in the corresponding seismic or dynamic analysis load cases, the program will report error as mass is missing

If a mass model using reference load type MASS is defined and also a seismic weight table is defined in DEFINE UBC/1893/… DEFINITION (or seismic mass is defined as part of a response spectrum or time history loading), the program will simply the later in place of the former for the seismic weight calculation. The program will issue a warning message in the analysis output. Care should be taken so that the mass model is not defined twice.

Example

```
UNIT FEET KIP
DEFINE REF LOAD
LOAD R1 LOADTYPE MASS
* MASS MODEL
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 2.5 FY 2.5 FZ 2.5
49 TO 64 FX 1.25 FY 1.25 FZ 1.25
*
LOAD R2
SELFWEIGHT Y -1
JOINT LOAD
17 TO 48 FY -2.5
49 TO 64 FY -1.25
END DEF REF LOAD
************************************************************
DEFINE UBC LOAD
ZONE 0.38 I 1 RWX 5.6 RWZ 5.6 STYP 2 CT 0.032 NA 1.3 NV 1.6
***********************************************************
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
TYPE 2 ACCELERATION
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.05
***********************************************************
LOAD 1
UBC LOAD X 0.75
***********************************************************
LOAD 2
BC LOAD Z 0.75
***********************************************************
LOAD 3
SPECTRUM CQC X 0.174075 ACC DAMP 0.05 SCALE 32.2
0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ; 1.0 1.60
***********************************************************
LOAD 4
SPECTRUM CQC Z 0.174075 ACC DAMP 0.05 SCALE 32.2
0.03 1.00 ; 0.05 1.35 ; 0.1 1.95 ; 0.2 2.80 ; 0.5 2.80 ; 1.0 1.60
**********************************************************
LOAD 5
TIME LOAD
53 57 37 41 21 25 FX 1 1
```

```
GROUND MOTION X 2 1
***********************************************************
```

Related Links

- *G.17.3.2 Mass Modeling* (on page 2157)
- *M. To generate a mass model* (on page 899)
- *G.17.3.2 Mass Modeling* (on page 2157)

TR.31.9 Defining Starting Load

This command is used to directly specify the starting load vectors for use with load dependent Ritz vector method for the eigen solution.

General Format

```
DEFINE STARTING LOAD
```

```
MASS { X | Y | Z | XY | YZ | ZX } NGEN n
```

or

```
REFERENCE LOAD Ri NGEN n
```

```
END
```

You may specify one of each direction or direction pair when using the `MASS` option. You may specify any number of reference loads.

where

n	=	The number of Ritz vectors to be extracted corresponding to each starting load.
Ri	=	The reference load number to be used as a starting load.

Program Versus User-Generated Start Vectors

Using the program generated method, the starting load vector is generated using the mass model of the structure. Assuming mass matrix only has translational components, the resulting load vector will have force components in the directions of all the translational degrees of freedom (DOFs) and will have zero values for all the rotational DOFs. This will guarantee a static deflection with components in many directions to start with the initial mode assumption. As long as the initial mode assumption has a component in the direction of interest, it will yield a correct set of Ritz vectors.

With the user-defined starting load, the load vector will have a force component in one translational degree of freedom in the direction of the dynamic load. This results in a static deflection mainly in the direction of interest. In some models where mass participation is predominant in one translational direction for the initial few modes this method can achieve 90% mass participation with only few modes.

Note: A single set of Ritz vectors can be derived from a single starting load. If all starting loads that participate in dynamic loads are defined, a single set of Ritz vectors can be extracted from all participating starting loads. The STAAD.Pro Advanced Analysis can extract a single set of Ritz vector only from *one* single starting load. Thus, the use of user-defined start vectors is best suited if the structural response is predominant in one translational direction. If responses in multiple translational DOF are predominant, it is recommended to use the program generated starting load vector.

Notes

Use of this command (and specifying the eigen solution in general) requires the Advanced Analysis.

This command requires that the `SET EIGEN METHOD RITZ` command also be used previously in the input file. Refer to TR.5 Set Command Specification (on page 2206) for details.

Examples

The following uses starting loads in each direction, with 10 Ritz vectors in each except for the Y direction only, which will extract 5.

```
DEFINE STARTING LOAD
* Load components in X direction only
MASS X NGEN 10
* Load components in Y direction only
MASS Y NGEN 5
* Load components in Z direction only
MASS Z NGEN 10
* Load components in X & Y directions
MASS XY NGEN 10
* Load components in Y & Z directions
MASS YZ NGEN 10
* Load components in Z & X directions
MASS ZX NGEN 10
END
```

The following example contains four reference loads:

1. R1: all 3 translational degrees of freedom (X, Y, & Z)
2. R2: in X d.o.f. only
3. R3: in Y d.o.f. only
4. R4: in Z d.o.f. only

Twenty five Ritz vectors will be generated for each of the starting loads.

```
DEFINE STARTING LOAD
* Load components in all directions
REF LOAD R1 NGEN 25
* Load components in X direction only
REF LOAD R2 NGEN 25
* Load components in Y direction only
REF LOAD R3 NGEN 25
* Load components in Z direction only
REF LOAD R4 NGEN 25
END
```

Related Links

- *M. To use starting vectors with load-dependent Ritz vectors* (on page 885)
- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *M. To use starting vectors with load-dependent Ritz vectors* (on page 885)

TR.32 Loading Specifications

This section describes the various loading options available in STAAD.Pro. The following command may be used to initiate a new load case.

General Format

```
LOADING i1 ( LOADTYPE a1 ) ( REDUCIBLE ) ( TITLE any_load_title )
```

Where:

LOADING *i1* any unique integer number (up to five digits) to identify the load case. This number need not be sequential with the previous load number.

LOADTYPE *a1* one of the following:

Dead	Soil	Ice
Live	Rain Water/ Ice	Wind on Ice
Roof Live	Ponding	Crane Hook
Wind	Dust	Mass (Notes (on page 2461))
Seismic-H (horizontal, Notes (on page 2461))	Traffic	Gravity
Seismic-V (vertical, Notes (on page 2461))	Temperature	Push
Snow	Accidental	None
Fluids	Flood	

The words `LOADTYPE a` are necessary only if you intend to use the Automatic Load Combination generation tool in the graphic interface. For details, refer to the Auto Load Combination dialog (on page 2872).

The keyword `REDUCIBLE` should be used only when the load type is `Live` or `Roof Live`. It instructs the program to reduce –according to the provisions of UBC 1997, IBC 2000, or IBC 2003 codes– a floor live load specified using `FLOOR LOAD` or `ONEWAY LOAD` commands. For details, see TR.32.4 Area, One-way, and Floor Load Specifications (on page 2475).

Under this heading, all different loads related to this loading number can be input.

Notes

a. For Mass Model Loading in Dynamics, it is strongly recommended that you read G.17.3.2 Mass Modeling (on page 2157) and TR.31.8.3 Mass Model Using Reference Load (on page 2457). For the purpose of entering the mass distribution for the first dynamic load case, use the following sections:

A reference load can be used to create a mass model for all dynamic analyses.

The purpose of the mass modeling step is to create lumped masses at the joints that the eigensolution can use. The member/element loading is only a convenience in generating the joint masses. Analytically the masses are not in the elements but are lumped at the joints.

b. The absolute value of joint loads or loads distributed to joints from member/element loadings will be treated as weights. The moments applied to member/elements or computed at joints as a result of member/element loadings will be ignored. Only moments (actually weight moment of inertia, force-length^2 units) applied in the Joint.

c. The load command will be used in defining the weight moment of inertias at joints. For dependent joint directions, the associated joint weight or weight moment of inertia will be moved to the control. In addition, the translational weights at dependent joint directions will be multiplied by the square of the distance to the control to get the additional weight moment of inertia at the control. Cross-product weight moment of inertias at the control will be ignored.

d. When Y is set as "Up" (default), then the horizontal seismic loads act in the global X and Y directions. The vertical seismic loads act in the global Y direction.

When `SET Z UP` is used, then the horizontal seismic loads act in the global X and Z directions. The vertical seismic loads then act in the global Z direction.

Related Links

- *M. To create a new primary load case* (on page 830)
- *M. To create a new primary load case* (on page 830)
- *M. To define primary load type* (on page 886)
- *G.17.3.2 Mass Modeling* (on page 2157)

TR.32.1 Joint Load Specification

This set of commands may be used to specify joint loads on the structure. For dynamic mass modeling see TR. 32.10 Dynamic Loading Specification (on page 2498) and G.17.3 Dynamic Analysis (on page 2154).

General Format

```
JOINT LOAD
```

```
joint-list (inclined-spec) *{ FX f8 | FY f9 | FZ f10 | MX f11 | MY f12 | MZ f13 }
```

Where:

```
inclined-spec = INCLINED { f1 f2 f3 | REF f4 f5 f6 | REFJT f7 }
```

Use the optional *inclined-spec* to specify a joint load in a inclined load axis. See Inclined Load Axis System (on page 2463) below for details.

f1, f2, f3	x, y, z relative distances from the joint in the global directions to the reference point
f4, f5, f6	x, y, z global coordinates of an arbitrary reference point

f7	a joint number whose x, y, z global coordinates are the reference point
f8, f9, f10	force values in the corresponding global direction (even at inclined support joints).
f11, f12, f13	moment values in the corresponding global direction.

Example

```
UNIT FEET KIP
*…
JOINT LOAD
3  TO  7  9  11  FY  -17.2  MZ  180.0
5  8  FX  15.1
UNIT INCH KIP
12  MX  180.0  FZ  6.3
```

Notes

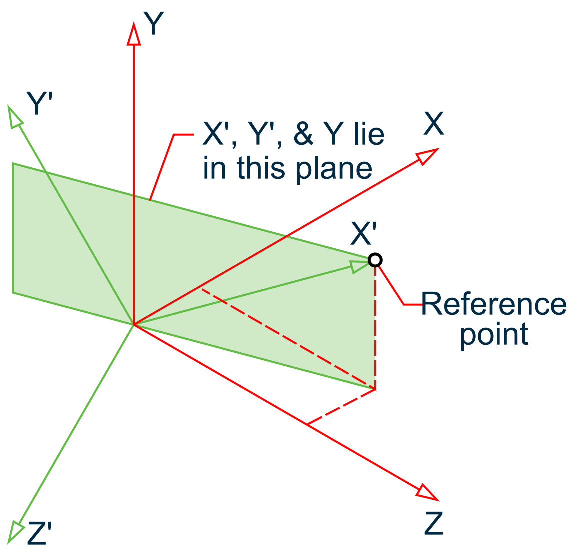
a. Joint numbers may be repeated where loads are meant to be additive in the joint.

b. A `UNIT` command may be on lines in between joint-list lines.

c. If moments are for dynamic mass, then the units are assumed to be force-length^2.

Inclined Load Axis System

If the joint loading is specified using the option `INCLINED`, then the axes in which the direct forces (and about which for moment forces) are applied are realigned from the global directions of X, Y, and Z to the directions x', y', and z' as shown below.



The reference point –whether it is specified using absolute coordinates, relative distances, or a reference joint– defines the direction of the x' axes from the loaded node to the reference point. The direction of the Y' axes is then defined as the direction perpendicular to x' that lies in the plane of X' and Y. In the special case of when X' is in the direction of Y, Y' is taken to be the direction of Z. The direction of Z' is then defined as the direction perpendicular to the plane X' and Y' and follows the right-hand rule as for all other axes systems used in STAAD.Pro.

Example

```
JOINT LOAD
4 INCLINED 1.0 -1.0 0.0 FX 10

JOINT LOAD
3 INCLINED REFJT 7 FZ -30
```

Related Links

- *G.15.1 Joint Loads* (on page 2128)
- *M. To add a nodal load* (on page 832)
- *G.15.1 Joint Loads* (on page 2128)

TR.32.2 Member Load Specification

This set of commands may be used to specify member loads on frame members

General Format

```
MEMBER LOAD
```

member-list { { UNI | UMOM } dir-spec f1 f2 f3 f4| { CON | CMOM } dir-spec f5 f6 f4| LIN dir-spec f7 f8 f9 | TRAP dir-spec f_{10} f_{11} f_{12} f_{13} }

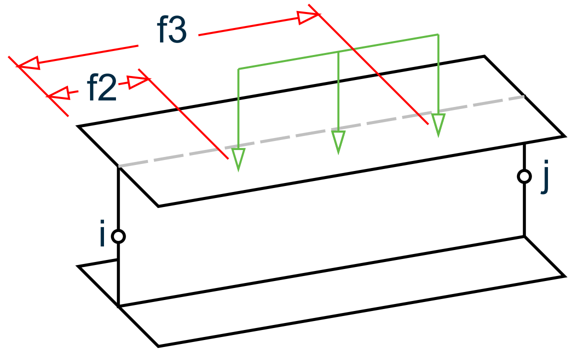
Where:

```
dir-spec = { X | Y | Z | GX | GY | GZ | PX | PY | PZ }
```

X, Y, & Z specify the direction of the load in the local (member) x, y and z-axes.
GX, GY, & GZ specify the direction of the load in the global X, Y, and Z-axes.
PX, PY, & PZ may be used if the load is to be along the projected length of the member in the corresponding global direction.

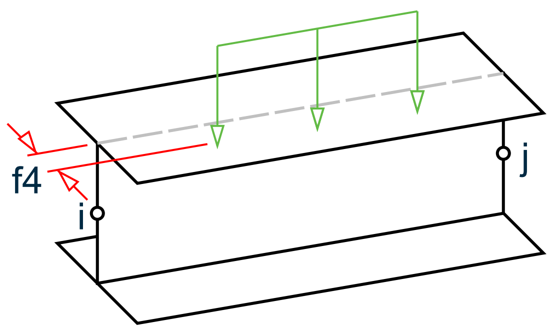
Note: Load start and end distances are measured along the member length and *not* the projected length.

f1	value of uniformly distributed load (UNI) or moment (UMOM).
f2, f3	distance of from the start of the member to the start of the load, and distance from the start of the member to the end of the load, respectively. The load is assumed to cover the full member length if f2 and f3 are omitted.



Note: Uniformly distributed moments can not be assigned to tapered members for analysis.

f4 Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. If global or projected load is selected, then the local Y component of load is offset the `f4` distance; the local Z component is offset the `f4` distance; and the local X component is not offset.



Note: The local x component of force is not offset (i.e., no secondary moment is caused by axial load).

f5 value of concentrated force (`CON`) or moment (`CMOM`)

f6 distance of from the start of the member to concentrated force or moment. `f6` will default to half the member length if omitted.

f7, f8 `LIN` specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then `f7` is the value at the start of the member and `f8` is the value at the end.

f9 If the load is triangular, then `f7` and `f8` are input as zero and `f9` is the value of the load in the middle of the member.

f10, f11 The starting and ending load value for a trapezoidal linearly varying load (`TRAP`), respectively. The trapezoidal load may act over the full or partial length of a member and in a local, global or projected direction.

f12, f13 the loading starting point and stopping point, respectively. Both are measured from the start of the member. If `f12` and `f13` are not given, the load is assumed to cover the full member length.

Notes

a. If the member being loaded has offset distances (refer to TR.25.1 Member Offset Specification (on page 2296)), the location of the load is measured from the initial end offset distance instead of the coordinates of the starting node .

b. Trapezoidal loads are converted into a uniform load and 8 or more concentrated loads.

c. A `UNIT` command may be on lines in between member-list lines.

d. If a load location is less than zero (i.e., occurs off the starting end of the member), then it is reset to 0.0.

e. If a load location is greater than the length, then it is reset to the length.

Example

```
MEMBER LOAD
619 CON GY -2.35 5.827
68 TO 72 UNI GX -0.088 3.17 10.0
186 TRAP GY -0.24 -0.35 0.0 7.96
3212 LIN X -5.431 -3.335
41016 UNI PZ -0.075
3724 LIN GY -6.2 -7.8
```

Projected Loads

If a projected load direction is specified, then the length of the member considered is that projected on the plane perpendicular to the load direction. This is always less than or equal to the actual member length. Projected loads reduce the scale of the load proportional to the projected length.

For example, if an inclined beam of total length, L, with a uniform load, w, is given the `GY` (global Y) load direction, then the resulting reactions are $R = 1/2 \times L \times w$. However, if the `PY` (projected Y) load direction is used, then the projected component of the length, L_x, is used as the member length. The resulting reactions are $R = 1/2 \times L_x \times w$ in this case.

The projected load direction, PY, uses the component length in the XZ plane for the load effect.

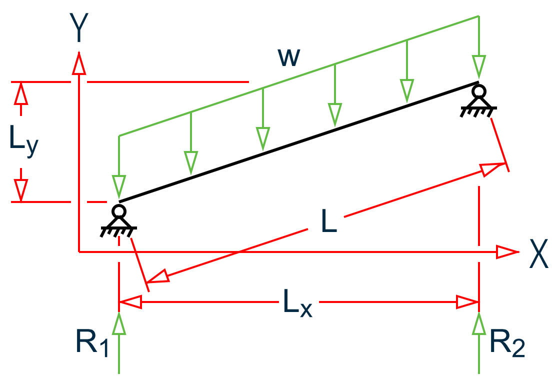


Figure 319: Example inclined beam parallel to the XY plane

```
JOINT COORDINATES
1 0 0 0; 2 12 3 0;
MEMBER INCIDENCES
```

```
1 1 2;
…
SUPPORTS
1 3 PINNED
…
LOAD 1
MEMBER LOAD
1 UNI PY -1
```

Related Links

- *G.15.2 Member Load* (on page 2129)
- *M. To add a concentrated force or moment on members* (on page 834)
- *M. To add a uniform load to members* (on page 834)
- *M. To add a linear varying load to members* (on page 835)
- *M. To add a hydrostatic load to objects* (on page 842)
- *G.15.2 Member Load* (on page 2129)

TR.32.2.1 PMember Load Specification

This set of commands may be used to specify member loads on a physical member (pmember).

General Format

```
PMEMBER LOAD
```

```
pmember-list { { UNI | UMOM } dir-spec f1 f2 f3 f4 | { CON | CMOM } dir-spec f5 f6 f4 |
 TRAP dir-spec f7 f8 f9 f10 }
```

Where:

```
dir-spec = { X | Y | Z | GX | GY | GZ | PX | PY | PZ }
```

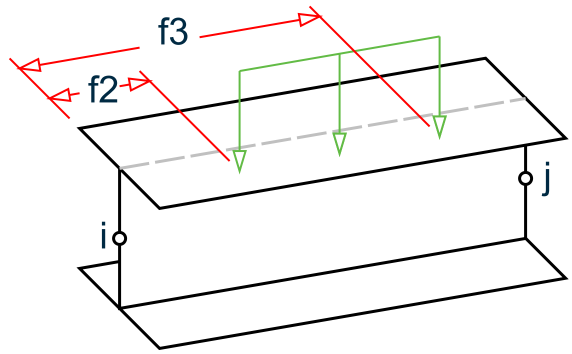
X, Y, & Z specify the direction of the load in the local (member) x, y and z-axes.

GX, GY, & GZ specify the direction of the load in the global X, Y, and Z-axes.

PX, PY, & PZ may be used if the load is to be along the projected length of the member in the corresponding global direction.

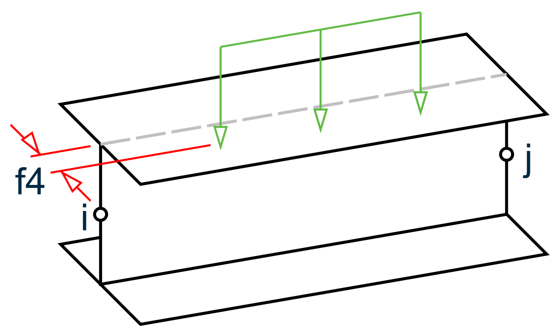
Note: Load start and end distances are measured along the member length and *not* the projected length.

f1	value of uniformly distributed load (UNI) or moment (UMOM).
f2, f3	distance of from the start of the member to the start of the load, and distance from the start of the member to the end of the load, respectively. The load is assumed to cover the full member length if *f2* and *f3* are omitted.



Note: Uniformly distributed moments can not be assigned to tapered members for analysis.

f4 Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. If global or projected load is selected, then the local Y component of load is offset the *f4* distance; the local Z component is offset the *f4* distance; and the local X component is not offset.



Note: The local x component of force is not offset (i.e., no secondary moment is caused by axial load).

f5 value of concentrated force (`CON`) or moment (`CMOM`)

f6 distance of from the start of the member to concentrated force or moment. *f6* will default to half the member length if omitted.

f7*, *f8 The starting and ending load value for a trapezoidal linearly varying load (`TRAP`), respectively. The TRAP load may act over the full or partial length of a member and in a local, global or projected direction.

f9*, *f10 the loading starting point and stopping point, respectively. Both are measured from the start of the member. If *f9* and *f10* are not given, the load is assumed to cover the full member length.

TR.32.3 Element Load Specifications

This set of commands may be used to specify various types of loads on plate and solid elements.

Related Links

- *G.15.8 Loading on Elements* (on page 2133)
- *G.5 Finite Element Information* (on page 2099)

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.32.3.1 Element Load Specification - Plates

This command may be used to specify various types of `ELEMENT LOADS` for plates.

Plate element loads must be applied following the expression

```
ELEMENT LOAD (PLATE)
```

using the format explained under the following options.

Option 1 - Load Applied in a Global Direction

```
element-list  PRESSURE {GX | GY | GZ} p1 (x1 y1) (x2 y2)
```

This is for specifying a pressure of magnitude *p1* in one of the global axis directions on the full element or a small rectangular part of an element.

- Partial Pressure - If applied on a small part, (x1,y1,x2 and y2) define the corners of the rectangular region where the load is applied.
- Concentrated Load - If only x1, y1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x1,y1).
- Full Plate Pressure - If (x1,y1,x2,y2) is not provided, the load is assumed to act over the full area of the element.

(x1,y1, x2 and y2) are measured from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

p1 has units of force per square of length for pressure and units of force for concentrated load.

GX, GY and GZ represent the global axis directions.

Option 2 - Load Applied Normal to Plate (Local z)

```
element-list PRESSURE p1 (x1 y1) (x2 y2)
```

This is for specifying a constant pressure of magnitude *p1* acting perpendicular to the plane of the element on the full element or a small rectangular part of an element. This coincides with the element's local Z axis.

- Partial Pressure - If applied on a small area, x1,y1,x2, and y2 define the corners of the rectangular region where the load is applied.
- Concentrated Load - If only x1, y1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x1,y1).
- Full Plate Pressure - If x1, y1, x2, and y2 are not provided, then the load is assumed to act over the full area of the element.

(x1,y1, x2 and y2) are measured from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

p1 has units of force per square of length for pressure and units of force for concentrated load.

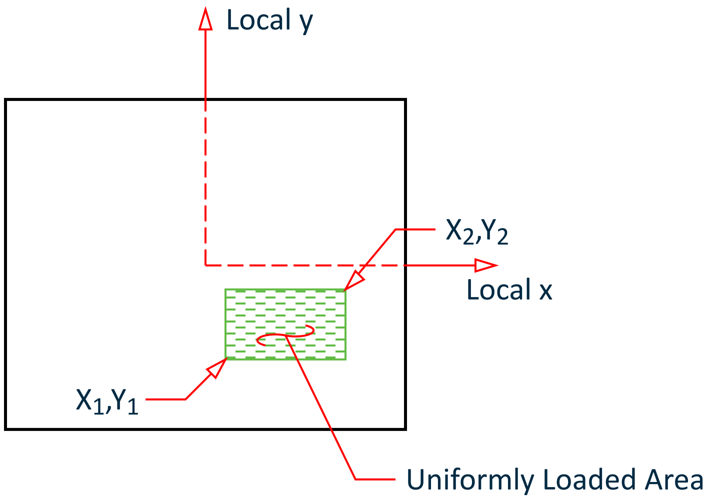


Figure 320: Coordinate values, x1,y1 & x2,y2, in the local coordinate system used in options 1 and 2

Option 3 - Pressure on Full Plate in Local X or Y or Projection

```
element-list PRESSURE {LX | LY | PX | PY | PZ} p2
```

This is for specifying a constant pressure of magnitude *p2* along the local X (LX) or Y (LY) axis of the element (parallel to the element surface) *or* along a global axis on the projected area of the plate. For example, in the case of a friction load you can use the LX or LY options.

A projected pressure is scaled down by a factor of the projected area onto the global plane perpendicular to the load direction to the element area. This is always equal to or less than the pressure in the global direction.

The resulting pressure, $p' = p\frac{A_{xz}}{A_{xyz}}$

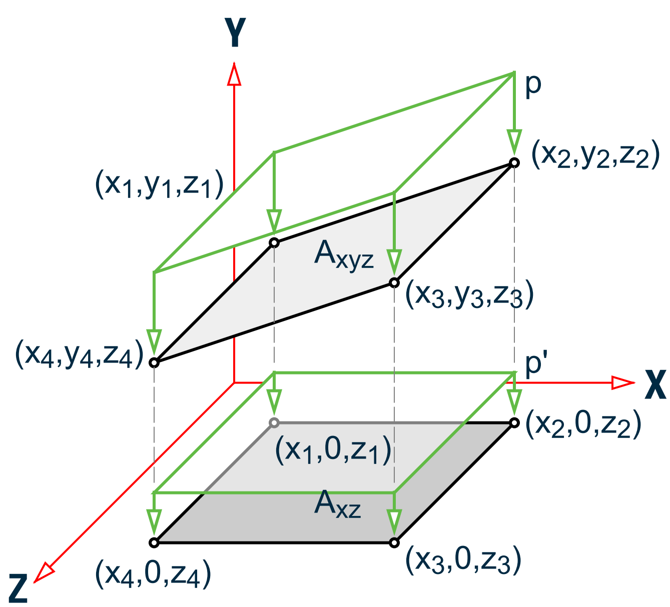


Figure 321: Projected pressure, p, in the PY direction

> **Note:** With this option, a load can be applied only on the full area of the element.

p_2 has units of force per square of length.

Option 4 - Trapezoidal Plate Load

```
element-list TRAP {GX | GY | GZ} {X | Y} f1 f2
```

This is for applying a trapezoidally varying load with the following characteristics:

a. The direction of action of the load is global (GX, GY or GZ), parallel to the surface (LX or LY as in friction type loads), or normal to the element surface (local Z). The last becomes the automatic direction of action if the global or tangential directions are not specified).

b. The load varies along the local X or Y directions (imagine the wall of a tank with hydrostatic pressure where pressure at the lower nodes is higher than at the upper nodes, and hence the load varies as one travels from the bottom edge of the elements to the top edge.)

This type of load has to be applied over the full area of the element. *f1* is the intensity at the I-J (or J-K) edge and *f2* is the intensity at the K-L (or L-I) edge depending on whether the load varies along "X" or "Y".

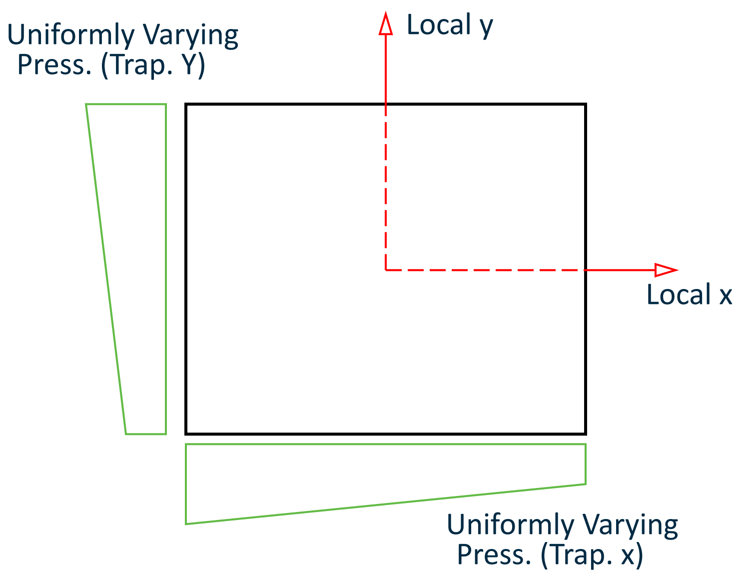


Figure 322: Center of the element is the origin defining the rectangular area on which the pressure is applied in option 4

The TRAP option should be used when a linearly varying pressure needs to be specified. The variation must be provided over the entire element.

X or Y - Direction of variation of element pressure. The TRAP X/Y option indicates that the variation of the Trapezoid is in the local X or in the local Y direction. The load acts in the global or local direction if selected, otherwise along the local Z axis.

f1 - Pressure intensity at start.
f2 - Pressure intensity at end.

Option 5 - Trapezoidal Loading Using Nodes

element-list TRAP {GX | GY | GZ | LX | LY} JT *f3 f4 f5 f6*

This is for specifying a trapezoidally varying load over the full area of the element where one happens to know the intensity at the joints (JT) of the element. The load is defined by intensities of *f3*, *f4*, *f5*, and *f6* at the 4 corners of a 4-noded element. For triangular elements, *f6* is not applicable. The load can act along the global directions (GX, GY and GZ) or along the local X and Y directions (LX and LY, like a friction load).

Notes

a. “Start” and “end” defined above are based on positive directions of the local X or Y axis.
b. Pressure intensities at the joints allows linear variation of pressure in both the X and Y local directions simultaneously.
c. The TRAP load with global directions may be used to apply a volumetric type of pressure. For example, consider a grain silo with a sloping wall. In the event of modeling it using non-uniform elements, by which we mean elements whose 3 or 4 nodes are all at different elevations, the grain height at each node will depend on the elevation of the node. One can apply the pressure by specifying the intensity at each node of each element.

Example

```
LOAD 4
ELEMENT LOAD
1 7 TO 10 PR 2.5
11 12 PR 2.5 1.5 2.5 5.5 4.5
15 TO 25 TRAP X 1.5 4.5
15 TO 20 TRAP GY JT 1.5 4.5 2.5 5.5
34 PR 5.0 2.5 2.5
35 TO 45 PR -2.5
15 25 TRAP GX Y 1.5 4.5
29 95 TRAP LX Y 3.7 8.7
```

Related Links

- *G.15.8 Loading on Elements* (on page 2133)
- *M. To add pressure load on a plate* (on page 837)
- *M. To add a concentrated load on a plate* (on page 838)
- *G.5 Finite Element Information* (on page 2099)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *G.5.2 Solid Elements* (on page 2110)
- *G.5.3 Surface Elements (Deprecated)* (on page 2113)

TR.32.3.2 Element Load Specification - Solids

The following types of loads can be assigned on the individual faces of solid elements:

- A uniform pressure
- A volumetric type of pressure on a face where the intensity at one node of the face can be different from that at another node on the same face.

  An example of such a load is the weight of water on the sloping face of a dam. If the dam is modeled using solids, for the individual elements, the water height at the lower elevation nodes will be larger than those at the higher elevation nodes.

General Format

The syntax is as follows.

```
ELEMENT LOAD SOLIDS
```

```
elem-list FACE i1 PRESSURE { GX | GY | GZ } f1 f2 f3 f4
```

Where:

f_1 f_2 f_3 f_4 = Pressure values at the joints for each 3 or 4 joint face defined. Only f_1 needs to be specified for uniform pressure. In any case the pressure is provided over the entire face.

i_1= one of six face numbers to receive the pressure for the solids selected. See the figure in G.5.2 Solid Elements (on page 2110) for the following face definitions. Enter a pressure on all 4 joints even if the face is collapsed to 3 points.

Description

The first option above loads the solid by specifying one or more of the 6 faces to receive pressure.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be normal to the face and a positive pressure is into the solid. The loads are proportional to the area, not the projected area.

Face Number	Surface Joints			
	f1	f2	f3	f4
1 front	Jt 1	Jt 4	Jt 3	Jt 2
2 bottom	Jt 1	Jt 2	Jt 6	Jt 5
3 left	Jt 1	Jt 5	Jt 8	Jt 4
4 top	Jt 4	Jt 8	Jt 7	Jt 3
5 right	Jt 2	Jt 3	Jt 7	Jt 6
6 back	Jt 5	Jt 6	Jt 7	Jt 8

Example

```
LOAD 4
ELEMENT LOAD SOLIDS
11 12 FACE 3 PR 2.5 1.5 2.5 5.5
```

Related Links

- *G.5.2 Solid Elements* (on page 2110)

TR.32.3.3 Element Load Specification - Joints

This command may be used to specify various types of element like loads for joints. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads. This command may be used as an alternative or supplement for the Area Load, Floor Load, and the other Element Load commands.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be in the local Z direction as if the joints defined a plate. The loads are proportional to the area, not the projected area.

General Format

```
ELEMENT LOAD JOINTS
```

i_1 (BY i_2) i_3 (BY i_4) i_5 (BY i_6) i_7 (BY i_8) -

FACETS j_1 PRESSURE { GX | GY | GZ } f_1 f_2 f_3 f_4

Note: If this data is on more than one line, the hyphens must be within the joint data.

Where:

f_1 f_2 f_3 f_4 = Pressure values at the joints for each 3 or 4 joint facet defined. Only f_1 needs to be specified for uniform pressure. In any case the pressure is provided over the entire element.
j_1 = number of facets loaded.
i_1, i_3, i_5, i_7 = Joint numbers that define the first facet.
i_2, i_4, i_6, i_8 = each joint number is incremented by the BY value (1 if omitted).

Example

```
LOAD 4
ELEMENT LOAD JOINT
1 BY 1 2 BY 1 32 BY 1 31 BY 1 -
FACETS 5 PR GY 10 10 15 15
```

The above data is equivalent to the following:

```
LOAD 4
ELEMENT LOAD JOINT
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
2 3 33 32 FACETS 1 PRESSURE GY 10 10 15 15
3 4 34 33 FACETS 1 PRESSURE GY 10 10 15 15
4 5 35 34 FACETS 1 PRESSURE GY 10 10 15 15
5 6 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

So, the value following the word FACETS is like a counter for generation, indicating how many element faces the load command must be created for. Thus a value of 5 for facets means, a total of 5 imaginary element faces have been loaded.

BY is the value by which the individual corner node number is being incremented during the generation. In this example, the value is 1, which is same as the default. Instead, if it had been say,

```
1 BY 22 BY 2 32 BY 1 31 BY 1 -
FACETS 5 PRESSURE GY 10 10 15 15
```

we would have obtained

```
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
3 5 33 32 FACETS 1 PRESSURE GY 10 10 15 15
5 8 34 33 FACETS 1 PRESSURE GY 10 10 15 15
```

```
8 11 35 34 FACETS 1 PRESSURE GY 10 10 15 15
9 14 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

Notes

If a pressure or volumetric load is acting on a region or surface, and the entity which makes up the surface, like a slab, is not part of the structural model, one can apply the pressure load using this facility. The load is defined in terms of the pressure intensity at the 3 or 4 nodes which can be treated as the corners of the triangular or quadrilateral plane area which makes up the region. This command may be used as an alternative or supplement for the Area Load, Floor Load, Wind Load, and other pressure load situations.

In other words, the element pressure load can be applied along a global direction on any surface, without actually having elements to model that surface. Thus, for a sloping face of a building, if one wants to apply a wind pressure on the sloping face, one can do so by specifying the joints which make up the boundary of that face. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads.

TR.32.4 Area, One-way, and Floor Load Specifications

These commands may be used to specify loading over an area enclosed by beam members and the program will distribute the loading onto the perimeter beams as either a one way or two-way system. They are used mostly when the entity transmitting the load, such as a slab, is not part of the structural model. The `AREA LOAD` or `ONEWAY LOAD` may be used for modeling one-way distribution and the `FLOOR LOAD` may be used for modeling two-way distribution. There are three commands which should be used in the following way:

Area Load	The program will establish the direction of the shorter span and load the beams in that direction. This command is used for distributing a pressure load onto the beams that define a closed loop.
Floor Load	This command is used for distributing a pressure load onto all beams that define a closed loop assuming a two way distribution of load.
One-way	This may appear similar to the `AREA LOAD` command, but this command defines an area from which the program will search out closed loops of beams, similar to the `FLOOR LOAD` command and also has the option to define the direction of span. This command is an development of the principals defined in the `FLOOR LOAD` command, but the load is defined to span in a single direction.

Live Load Reduction

Floor loads and one-way loads can be made reducible according to Section 1607 of IBC 2000 if included in a Live Load case which is specified as Reducible. The following rules are implemented:

- Reduction only applies to live loads on members with a tributary area of greater than 150 ft^2.
- Live loads over 100 psf are not reduced.
- The reduction in live load (in percent) is = $0.08\times(A_{trib} - 150)$, with a maximum reduction amount of 40% for members carrying load from one level only.

Note: Live load reduction is *not* not available for area loads.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)

TR.32.4.1 Area Load Specification

Used for distributing a pressure load onto the beams that define a closed loop. The program will establish the direction of the shorter span and load the beams in that direction.

Note: The `AREA LOAD` command has been deprecated in favor of the `ONEWAY LOAD` or `FLOOR LOAD` commands.

General Format

```
AREA LOAD
```

```
memb-list ALOAD f1  { GX | GY | GZ }
```

Where:

f1 The value of the area load (units of weight over square length unit). If Global direction is omitted, then this load acts along the positive local y-axis [for the members of a floor analysis, the local y direction will coincide with global vertical axis in most cases]. If Global direction is included, then the load acts in that direction. The magnitude of the loads calculated is the same as if the positive local y axis option was selected. (For detailed description, refer to G.15.3 Area, One-way, and Floor Loads (on page 2130).)

Note: Area load should not be specified on members declared as `MEMBER CABLE`, `MEMBER TRUSS` or `MEMBER TENSION`.

Example

```
AREA LOAD
2 4 TO 8 A LOAD - 0.250
12 16 ALOAD -.0500
```

Notes

a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).

b. For the `FLOOR LOAD` specification, a two-way distribution of the load is considered. For the `ONEWAY` and `AREA LOAD` specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the `FLOOR LOAD` type.

c. The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADSs, and FLOOR LOADs for mass matrix for frequency calculations.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *M. To add an area load* (on page 839)
- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)

TR.32.4.2 One-way Load Specification

Defines an area from which the program will search out closed loops of beams, similar to the TR.32.4.3 Floor Load Specification (on page 2483) command and also has the option to define the direction of span. This command is a development of the principals defined in the `FLOOR LOAD` command, but the load is defined to span in a single direction (similar to `AREA LOAD`).

The One-way Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

General Format

```
ONEWAY LOAD (SAVE { LOAD })
```

```
YRANGE  f1 f2 ONELOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (TOWARDS f8) (PRINT
{MEMBER | LOAD})
```

or

```
XRANGE f1 f2 ONELOAD f3 (YRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (TOWARDS f8) (PRINT
{MEMBER | LOAD})
```

or

```
ZRANGE f1 f2 ONELOAD f3 (XRA f4 f5 YRA f6 f7) { GX | GY | GZ } (TOWARDS f8) (PRINT
{MEMBER | LOAD})
```

or

```
_FloorGroupName ONELOAD f3 { GX | GY | GZ } (INCLINED) (TOWARDS f8)
```

Where:

f1 f2	Global coordinate values to specify Y, X, or Z range. The load will be calculated for all members lying in that global plane within the first specified global coordinate range.
f3	The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.
f4-f7	Global coordinate values to define the corner points of the area on which the specified floor load (*f3*) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.
GX,GY,GZ	If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.
FloorGroupName	See TR.16.1 Listing of Entities by Specifying Groups (on page 2235) for the procedure for creating FLOORGROUPs. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.
f8	Defines a member onto which the loading is directed and defines the span direction for the one way loading. If the `TOWARDS` option is not used, the program will default to distributing a one-way load to the longest side. See Note b below for square panels.

`INCLINED` - This option must be used when a `ONEWAY LOAD` is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ, or ZX planes.

`PRINT` = This option is used when floor panel information printout is required. The total number of panels identified, total area of all panels and total load generated will be printed in the output file.

`PRINT MEMBER` = This option is used to print panel member numbers on which loads are generated along with the total number of panels identified, total area of all panels and total load generated.

`PRINT LOAD` = This option is used to print panel loading information. All other information available with `PRINT MEMBER` will also be printed in the output file.

`SAVE` = This option will give panel information in an external text file named *(filename)* `_FLD.TXT`. The information corresponding to `PRINT MEMBER` will be printed in this file.

`SAVE LOAD` = This option will give panel information in an external text file named *(filename)* `_FLD.TXT`. The information corresponding to `PRINT LOAD` will be printed in this file.

Note: It is recommended to use `SAVE` or `SAVE LOAD` option which will give same output printed in an external text file. If detailed output is printed in output (`.ANL` file) file using `PRINT MEMBER` or `PRINT LOAD` option, the size of the output file may increase to a large extent.

Tip: The `SET FLOOR LOAD TOLERANCE` command may be used to specify a specify the tolerance for out-of-plane nodes to be included in a floor load. See TR.5 Set Command Specification (on page 2206)

Notes

a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).

b. For the `FLOOR LOAD` specification, a two-way distribution of the load is considered. For the `ONEWAY` and `AREA LOAD` specification, a one-way action is considered. For `ONE WAY` loads, the program attempts to find the shorter direction within panels for load generation purposes.

In V8i SELECTseries 6 build 20.07.11.50 and later: for one-way loads applied to square panels (i.e., span direction is the same in either direction), triangular loads are generated on the members forming the four sides of the square. The intensity of the loads, which are equal for all four members, is calculated on the basis of four triangles formed by the two intersecting diagonals of the square.

Alternately, you may use the `FLOOR LOAD` type.

c. The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.

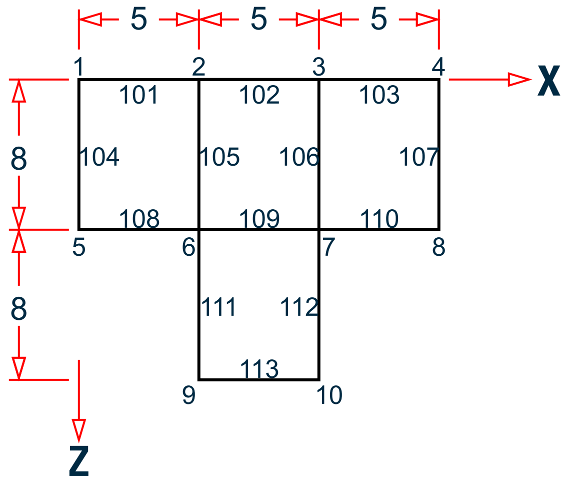
d. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

The attached example illustrates a case where the floor has to be sub-divided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command:

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35 XRA -.01 15.1 ZRA -0.1 8.1
YRANGE 11.9 12.1 FLOAD -0.35 XRA 4.9 10.1 ZRA 7.9 16.1
```



e. At least one quadrilateral panel bounded on at least 3 sides by "complete" members has to be present within the bounds of the user-defined range of coordinates (XRANGE, YRANGE and ZRANGE) in order for the program to successfully generate member loads from the FLOOR/ONEWAY LOAD specification. A "complete" member is defined as one whose entire length between its start and end coordinates borders the specified panel.

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.

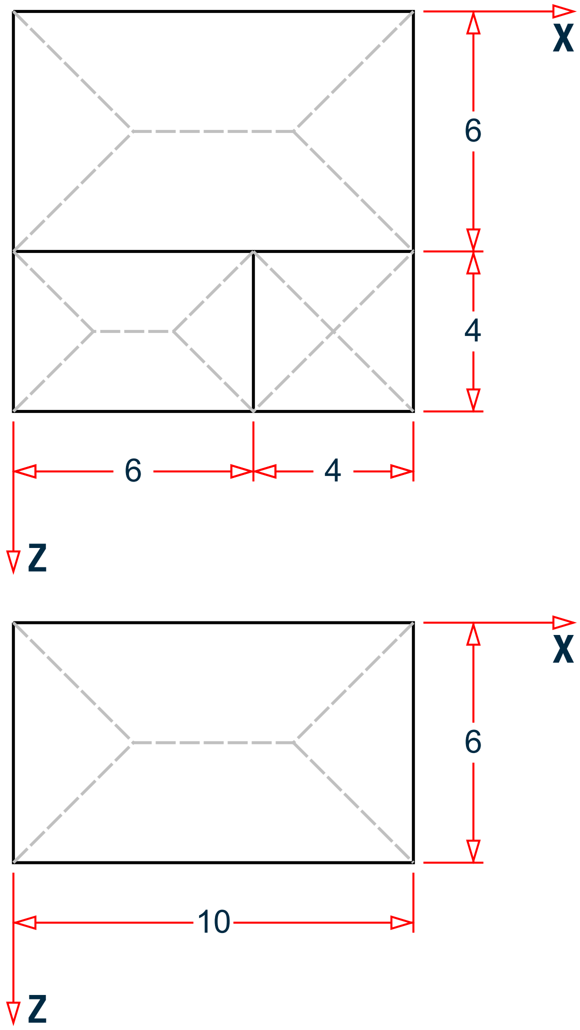


Figure 323: Trapezoidal and triangular load distribution for rectangular panels

First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.

Live Load Reduction per UBC and IBC Codes

The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD.Pro model.

i. The live load must be applied using the `FLOOR LOAD` or `ONEWAY LOAD` option. This option is described above, and an example of its usage may be found in EX. US-15 Wind and Floor Load Generation on a Space Frame (on page 6423).

ii. As shown in TR.32 Loading Specifications (on page 2461), the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called **Reducible**, also has to be specified as shown.

```
LOAD n LOADTYPE Live REDUCIBLE
```

Where:

n is the load case number

The following figures show the load generated on members for the two situations.

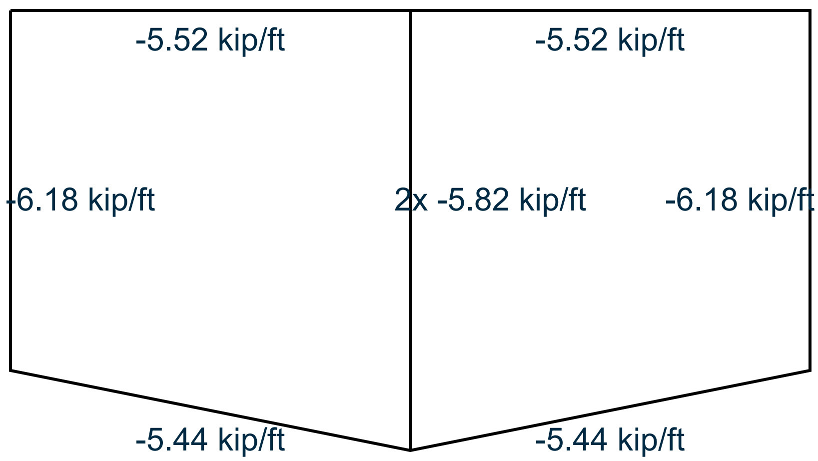


Figure 324: Load generated for the previously described cases

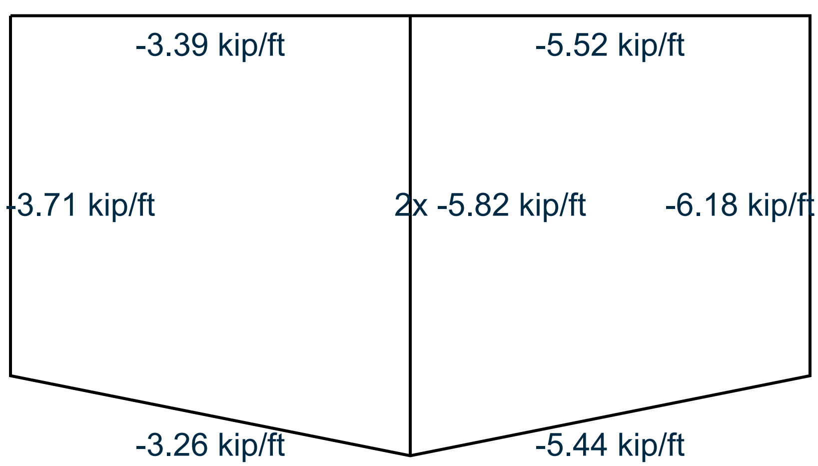


Table 261: Table - Details of the code implementation

Code Name	Section of code which has been implemented	Applicable Equations
UBC 1997	1607.5, page	Equation 7-1 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2000	1607.9.2, page 302	Equation 16-2 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2003	1607.9.2, page 277	Equation 16-22 R = r(A-150) for FPS units R = r(A-13.94) for SI units

where

A = area of floor supported by the member
R = reduction in percentage
= rate of reduction equal to 0.08 for floors.

Notes

a. Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.
b. Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.
c. Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.
d. In the IBC 2000 and 2003 codes, the first note says "A reduction shall not be permitted in Group A occupancies." In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user's responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.
e. In the UBC 97 code, the last paragraph of section 1607.5 states that "The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle." Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user's responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.
f. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *M. To add a floor load or one-way load* (on page 840)
- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)

TR.32.4.3 Floor Load Specification

Used to distribute a pressure load onto all beams that define a closed loop assuming a two way distribution of load.

The Floor Load specification be applied to groups and can also use live load reduction per IBC or UBC codes.

General Format

```
FLOOR LOAD (SAVE { LOAD })
```

```
YRANGE  f1 f2 FLOAD f3 (XRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
XRANGE f1 f2 FLOAD f3 (YRA f4 f5 ZRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
ZRANGE f1 f2 FLOAD f3 (XRA f4 f5 YRA f6 f7) { GX | GY | GZ } (PRINT {MEMBER | LOAD})
```

or

```
_FloorGroupNameFLOAD f3 { GX | GY | GZ } (INCLINED) }
```

Where:

f1 f2	Global coordinate values to specify Y, X, or Z range. The load will be calculated for all members lying in that global plane within the first specified global coordinate range.
f3	The value of the load (unit weight over square length unit). If the global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.
f4 - f7	Global coordinate values to define the corner points of the area on which the specified floor load (*f3*) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.
GX,GY,GZ	If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.
FloorGroupName	See TR.16.1 Listing of Entities by Specifying Groups (on page 2235) for the procedure for creating FLOORGROUPs. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.

`INCLINED` - This option must be used when a `FLOOR LOAD` is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ, or ZX planes.

`PRINT` = This option is used when floor panel information printout is required. The total number of panels identified, total area of all panels and total load generated will be printed in the output file.

PRINT MEMBER = This option is used to print panel member numbers on which loads are generated along with the total number of panels identified, total area of all panels and total load generated.

PRINT LOAD = This option is used to print panel loading information. All other information available with PRINT MEMBER will also be printed in the output file.

SAVE = This option will give panel information in an external text file named *(filename)* _FLD.TXT. The information corresponding to PRINT MEMBER will be printed in this file.

SAVE LOAD = This option will give panel information in an external text file named *(filename)* _FLD.TXT. The information corresponding to PRINT LOAD will be printed in this file.

Note: It is recommended to use SAVE or SAVE LOAD option which will give same output printed in an external text file. If detailed output is printed in output (.ANL file) file using PRINT MEMBER or PRINT LOAD option, the size of the output file may increase to a large extent.

Tip: The SET FLOOR LOAD TOLERANCE command may be used to specify a specify the tolerance for out-of-plane nodes to be included in a floor load. See TR.5 Set Command Specification (on page 2206)

Notes

a. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
b. For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the FLOOR LOAD type.
c. FLOOR LOAD from a slab is distributed on the adjoining members as trapezoidal and triangular loads depending on the length of the sides as shown in the diagram. Internally, these loads are converted to multiple point loads.

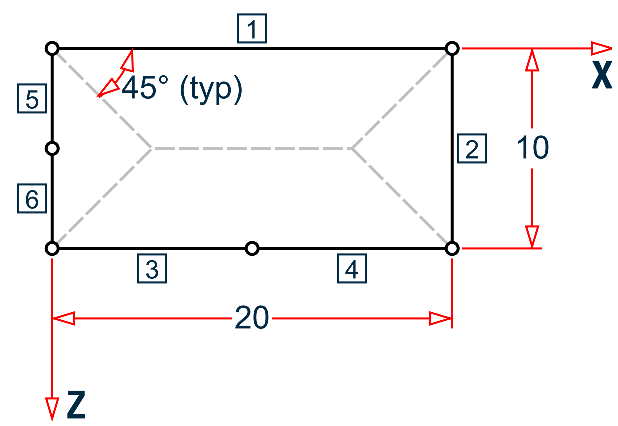


Figure 325: Members 1 and 2 get full trapezoidal and triangular loads respectively. Members 3 and 4 get partial trapezoidal loads and 5 and 6 get partial triangular load.

d. The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
e. The FLOOR LOAD facility is not available if the SET Z UP command is used (See Section 5.5.)

f. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

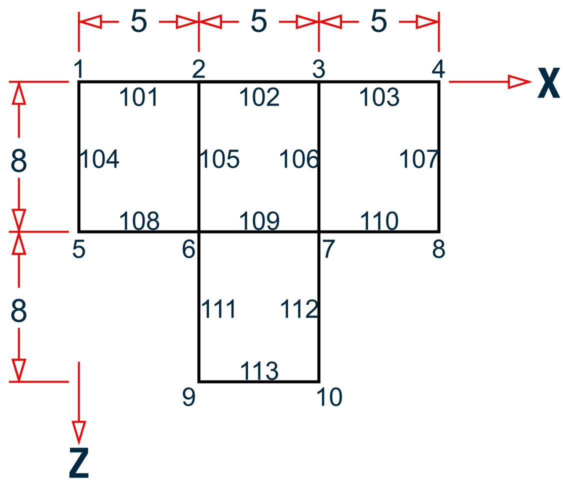
If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better.

The attached example illustrates a case where the floor has to be sub-divided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command:

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.35 XRA -.01 15.1 ZRA -0.1 8.1
YRANGE 11.9 12.1 FLOAD -0.35 XRA 4.9 10.1 ZRA 7.9 16.1
```



g. The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADSs and FLOOR LOADs for mass matrix for frequency calculations.

h. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated on the members circumscribing those panels. In such cases, one ought to use the FLOOR LOAD type.

Applying FLOOR LOAD onto a Floor Group

When applying a floor load using XRANGE, YRANGE and ZRANGE, there are two limitations that one may encounter:

a. If panels consist of members whose longitudinal axis cross each other in an X type, and if the members are not connected to each other at the point of crossing, the panel identification and hence the load generation in that panel may fail. A typical such situation is shown in the plan drawing shown in the next figure.

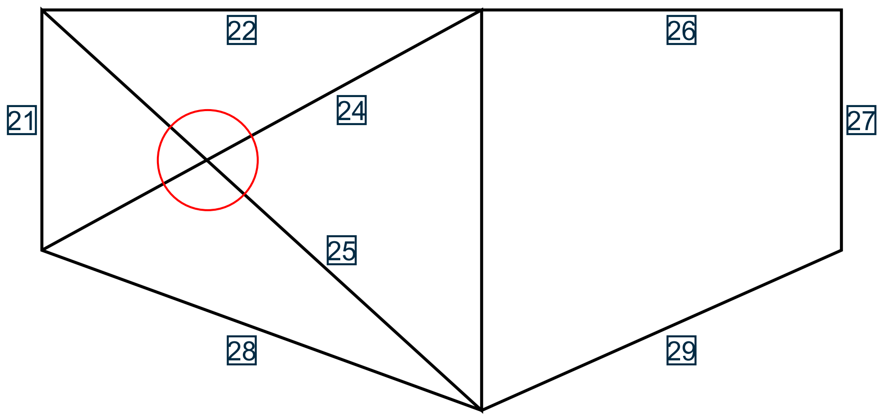


Figure 326: Load generation on a panel with members not connected at a point of crossing (members 24 and 24)

b. After the load is specified, if the user decides to change the geometry of the structure (X, Y or Z coordinates of the nodes of the regions over which the floor load is applied), she/he has to go back to the load and modify its data too, such as the XRANGE, YRANGE and ZRANGE values. In other words, the 2 sets of data are not automatically linked.

The above limitations may be overcome using a FLOOR GROUP. A GROUP name is a facility which enables us to cluster a set of entities – nodes, members, plates, solids, etc. into a single moniker through which one can address them. Details of this are available in section 5.16 of this manual.

The syntax of this command, as explained earlier in this section is:

```
FLOOR LOAD
```

```
Floor-group-name FLOAD  f3 { GX | GY | GZ }
```

Where:

f3 = pressure on the floor

To create equal loads in all 3 global directions for mass definition or other reasons, then enter direction labels for each direction desired; GY first then GX and/or GZ.

Example

```
START GROUP DEFINITION
FLOOR
_PNL5A  21 22 23 28
END GROUP DEFINITION
LOAD 2 FLOOR LOAD on intermediate panel @ Y = 10 ft
FLOOR LOAD
_PNL5A  FLOAD -0.45 GY
_PNL5A FLOAD -0.45 GY GX GZ
```

```
LOAD 5 LOAD ON SLOPING ROOF
FLOOR LOAD
_SLOPINGROOF FLOAD -0.5 GY INCLINED
```

Live Load Reduction per UBC and IBC Codes

The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD.Pro model.

i. The live load must be applied using the `FLOOR LOAD` or `ONEWAY LOAD` option. This option is described above, and an example of its usage may be found in EX. US-15 Wind and Floor Load Generation on a Space Frame (on page 6423).
ii. As shown in TR.32 Loading Specifications (on page 2461), the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called **Reducible**, also has to be specified as shown.

```
LOAD n LOADTYPE Live REDUCIBLE
```

Where:

n is the load case number

The following figures show the load generated on members for the two situations.

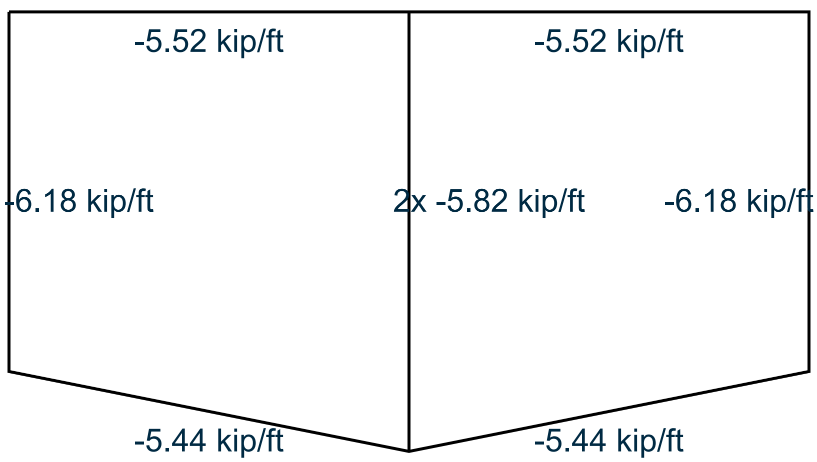


Figure 327: Load generated for the previously described cases

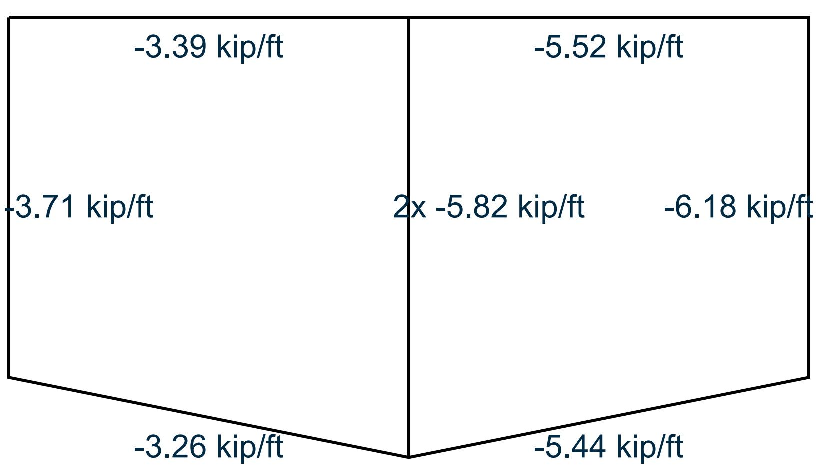


Table 262: Table - Details of the code implementation

Code Name	Section of code which has been implemented	Applicable Equations
UBC 1997	1607.5, page	Equation 7-1 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2000	1607.9.2, page 302	Equation 16-2 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2003	1607.9.2, page 277	Equation 16-22 R = r(A-150) for FPS units R = r(A-13.94) for SI units

where

A	=	area of floor supported by the member
R	=	reduction in percentage
		= rate of reduction equal to 0.08 for floors.

Notes

a. Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.

b. Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.

c. Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.

d. In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

e. In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

f. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *M. To add a floor load or one-way load* (on page 840)
- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)

TR.32.5 Prestress Load Specification

This command may be used to specify PRESTRESS loads on members of the structure.

General Format

```
MEMBER {PRESTRESS | POSTSTRESS } (LOAD)
```

```
memb-list FORCE f1 *{ ES f2 | EM f3 | EE f4 }
```

Where:

FORCE *f1*	Prestressing force. A positive value indicates precompression in the direction of the local x-axis. A negative value indicates pretension.
ES *f2*	specifies eccentricity (from the centroid) of the prestress force at the start of the member.
EM *f3*	specifies eccentricity (from the centroid) of the prestress force at the mid-point of the member.
EE *f4*	specifies eccentricity (from the centroid) of the prestress force at the end of the member.

Description

The first option, (MEMBER PRESTRESS LOAD), considers the effect of the prestressing force during its application. Thus, transverse shear generated at the ends of the member(s) subject to the prestressing force is transferred to the adjacent members.

The second option, (MEMBER POSTSTRESS LOAD), considers the effect of the existing prestress load after the prestressing operation. Thus, transverse shear at the ends of the member(s) subject to the prestressing force is not transferred to the adjacent members.

Example

```
MEMBER PRESTRESS
2 TO 7 11 FORCE 50.0
MEMBER POSTSTRESS
8 FORCE 30.0 ES 3.0 EM -6.0 EE 3.0
```

In the first example, a prestressing force of 50 force units is applied through the centroid (i.e., no eccentricity) of members 2 to 7 and 11. In the second example, a poststressing force of 30 force units is applied with an eccentricity of 3 length units at the start, -6.0 at the middle, and 3.0 at the end of member 8.

One of the limitations in using this command is that under any one load case, on any given member, a prestress or poststress load may be applied only once. If the given member carries multiple stressed cables or has a PRESTRESS and POSTSTRESS load condition, such a situation will have to be specified through multiple load cases for that member. See example below.

Incorrect Input:

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
6 FORCE 150 ES 3 EM -6 EE 3
PERFORM ANALYSIS
```

Correct Input:

```
LOAD 1 MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
6 FORCE 150 ES 3 EM -6 EE 3
LOAD COMBINATION 3
1 1.0 2 1.0
PERFORM ANALYSIS
```

Examples for Modeling Techniques

The following examples describe the partial input data for the members and cable profiles shown below.

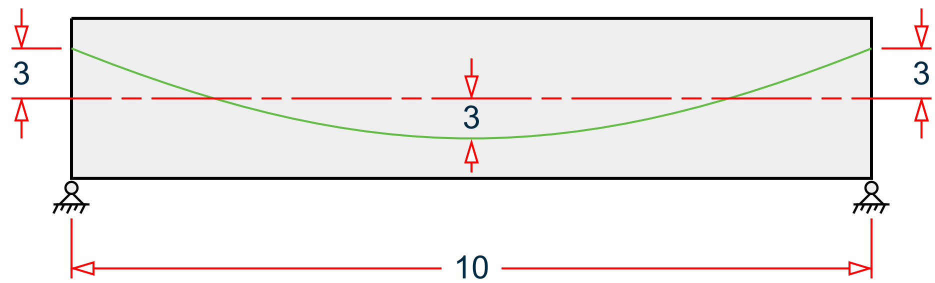


Figure 328: Example 1

```
JOINT COORD
1 0 0 ; 2 10 0
MEMBER INCI
1 1 2
…
UNIT …
LOAD 1
MEMBER POSTSTRESS
1 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```

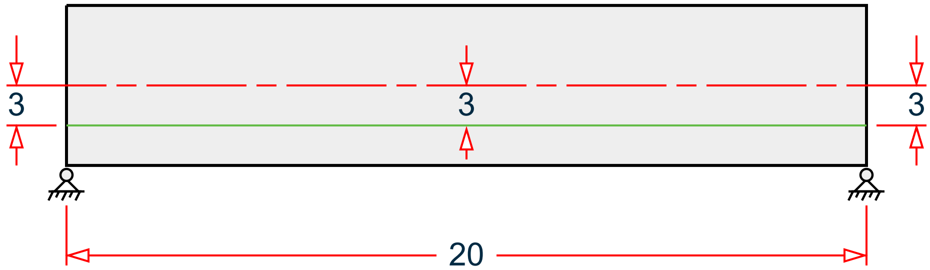


Figure 329: Example 2

```
JOINT COORD
1 0 0 ; 2 20 0
MEMBER INCI
1 1 2
…
UNIT …
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES -3 EM -3 EE -3
PERFORM ANALYSIS
```

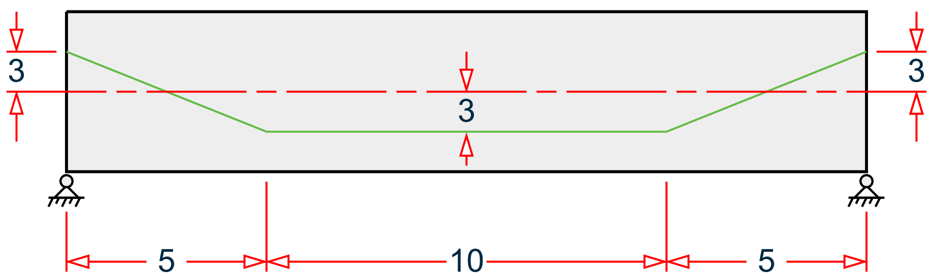


Figure 330: Example 3

```
JOINT COORD
1 0 0 ; 2 5 0 ; 3 15 0 0 ; 4 20 0
MEMBER INCI
1 1 2 ; 2 2 3 ; 3 3 4
…
UNIT …
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM -3 EE -3
3 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS
```

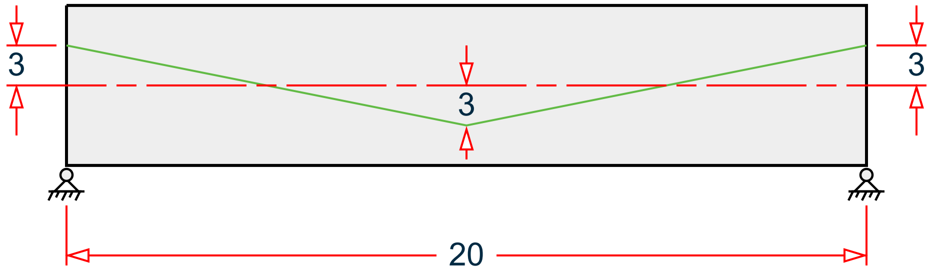


Figure 331: Example 4

```
JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMBER INCI
1 1 2 ; 2 2 3
…
UNIT …
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS
```

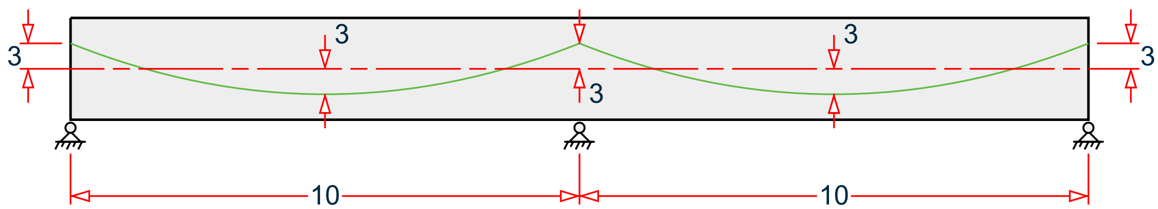


Figure 332: Example 5

```
JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMBER INCI
1 1 2 ; 2 2 3
…
UNIT …
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM -3 EE 3
2 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```

Related Links

- *G.15.5 Prestress and Poststress Member Load* (on page 2131)
- *EX. US-6 Prestress and Poststress Loading* (on page 6351)
- *EX. UK-6 Prestress and Poststress Loading* (on page 6635)
- *M. To add a prestress or post-tension load to members* (on page 836)
- *G.15.5 Prestress and Poststress Member Load* (on page 2131)

TR.32.6 Temperature Load Specification for Members, Plates, and Solids

This command may be used to specify TEMPERATURE loads or strain loads on members, plates, and solids; or strain loads on members.

General Format

```
TEMPERATURE LOAD
```

```
memb/elem-list { TEMP f1 f2 f4 | STRAIN f3 | STRAINRATE f5 }
```

Where:

f1 The change in temperature which will cause axial elongation in the members or uniform volume expansion in plates and solids. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command. (Members/Plates/Solids)

f2 The temperature differential from the top to the bottom of the member or plate ($T_{top\ surface}$-$T_{bottom\ surface}$). If *f2* is omitted, no bending will be considered. (Local Y axis) (Members/Plates). Section depth must be entered for prismatic.

f3 Initial axial elongation (+)/ shrinkage (-) in member due to misfit, etc. in length unit (Members only).

f4 The temperature differential from side to side of the member. (Local Z axis) (Members). Section or flange width must be entered for prismatic.

f5 Initial axial elongation (+)/ shrinkage (-) per unit length, of members only.

Example

```
UNIT MMS
TEMP LOAD
1 TO 9 15 17 TEMP 70.0
18 TO 23 TEMP 90.0 66.0
8 TO 13 STRAIN 3.0
15 27 STRAINRATE 0.4E-4
```

Notes

a. It is not necessary or possible to specify the units for temperature or for ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for the temperature load. (See TR.26 Specifying and Assigning Material Constants (on page 2303)).

If ALPHA was provided by a material name (STEEL, CONCRETE, ALUMINUM) then the temperature here must be in degree Fahrenheit units (if English length units were in effect when Alpha was defined) or degree Celsius units (if metric length units were in effect when Alpha was defined).

Related Links

- *G.15.6 Temperature and Strain Load* (on page 2133)

TR.32.7 Fixed-End Load Specification

This command may be used to specify `FIXED END` loads on members (beams only) of the structure.

General Format

```
FIXED ( END ) LOAD
```

```
memb-list FXLOAD f1, f2, … f12
```

Where:

member_list	normal STAAD.Pro member list rules (TO and BY for generation; and - to continue list to next line).
f1 … f6	Force-x, shear-y, shear-z, torsion, moment-y, moment-z (all in local coordinates) at the start of the member.
f7 … f12	Same as above except at the end of the member.

If less than 12 load values are entered, zero values will be added to the end. The loads may be extended to one additional line by ending the first load line with a - (hyphen).

These loads are given in the member local coordinate system and the directions are opposite to the actual load on the member.

Related Links

- *G.15.4 Fixed End Member Load* (on page 2131)
- *M. To add fixed end member loads* (on page 837)
- *G.15.4 Fixed End Member Load* (on page 2131)

TR.32.8 Support Joint Displacement Specification

This command may be used to specify displacements (or generate loads to induce specified displacements) in supported directions (pinned, fixed, enforced, or spring).

General Format

```
SUPPORT DISPLACMENT
```

```
support-joint-list { FX | FY | FZ | MX | MY | MZ } f1
```

Where:

f1	Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.

FX, FY, FZ specify translational displacements in X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in X, Y, and Z directions.

Description

There are two distinct modes of usage for this command. If any `ENFORCED` specifications were used in the support command then the “displacement mode” is used; otherwise the “load mode” is used. Despite the name of this command, if displacements are specified in spring directions, the displacement is at the joint not at the grounded end of the support spring. Displacement cannot be specified in a direction that does not have a support or a spring.

Displacement Mode

With this mode, the support joint displacement is modeled as an imposed joint displacement. The joint directions where displacement may be specified must be defined (same for all cases) in the `SUPPORT` command, see TR.27.1 Global Support Specification (on page 2316). Any beam members, springs or finite elements will be considered in the analysis. Other loading, inclined supports, and control/dependent are all considered. Any number of cases may have displacements entered. However, all cases will have zero displacements at the enforced directions if no displacement values are entered for that case. Displacements occur along global directions, even at inclined supports. Displacements may not be specified at dependent directions.

If some cases are to have spring supports and others enforced displacements at the same joint directions, then two `PERFORM ANALYSES` must be used with the `CHANGE` command in between. The first perform analysis could have the `SUPPORTS` with springs, no enforced directions, and with the load cases without displacements. The second perform analysis would then have `SUPPORTS` without springs but with enforced directions and the cases with displacements.

Displacement Mode Restrictions:

- The Support Displacement command may be entered only once per case.
- Spring directions and Enforced directions may not both be specified at the same joint direction in the same Perform Analysis step.

Load Mode

With this mode, the support joint displacement is modeled as a load. Only beam members (not springs or finite elements) are considered in computing the joint load distribution necessary to cause the displacement. Other loading, inclined supports, and control/ dependent specifications are also not considered. These unconsidered factors, if entered, will result in displacements other than those entered (results are superimposed). Only those cases with displacements entered will be affected.

Load Mode Restrictions:

- Support Displacements can be applied in up to four load cases only.
- The Support Displacement command may be entered only once per case.
- Finite elements should not be entered.
- Inclined supports must not be entered.
- Spring supports are not considered in calculating the load so their use will lead to displacements different from the input values.

Example

```
UNIT …
SUPPORT DISPL
5 TO 11 13 FY -0.25
19 21 TO 25 MX 15.0
```

In this example, the joints of the first support list will be displaced by 0.25 units in the negative global Y direction. The joints of the second support list will be rotated by 15 degrees about the global X-axis.

Related Links

- *G.15.7 Support Displacement Loads* (on page 2133)
- *M. To add a support displacement* (on page 833)
- *G.15.7 Support Displacement Loads* (on page 2133)

TR.32.9 Selfweight

Selfweight commands are used to calculate and apply the self weight of structural elements for analysis.

TR.32.9.1 Selfweight Loads

Used to calculate and apply the self weight of members, plates, and solids in the structure for analysis.

General Format

```
SELFWEIGHT ( { X | Y | Z } f1) ( LIST member-list | ALL )
```

Where:

X, Y, & Z	represent the global direction in which the selfweight acts. The default direction is along the Y axis.
f1	The factor to be used to multiply the selfweight. The default value is -1.0.
member-list	member, plate and/or solid list or group. All list specifications such as explicit list including TO and BY, range (XR, YR, ZR), parallel to (X, Y, Z) and groups are supported. If a group is used, the selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by INACTIVE command specification or internally set due to TENSION/COMPRESSION ONLY specification.

Description

This command is used if the self-weight of the members, plates, and solids within the structure is to be considered. The self-weight of every active element is calculated and applied as a uniformly distributed member load.

Note: Surfaces are not included in the selfweight command. The SSELFWT must be used to include the selfweight for surface elements. See TR.32.9.2 Surface Selfweight Load (on page 2497)

This command may also be used without any direction and factor specification. Thus, if specified as SELFWEIGHT, loads will be applied in the negative global Y direction with a factor of unity.

Notes

a. Density must be provided for calculation of the self weight.
b. The selfweight of finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.
c. The selfweight of a plate is placed at the joints, regardless of plate releases.

d. The `SELFWEIGHT` specification for definition of static method of seismic load generator also is capable of accepting a list. Only there is no direction specification for this command.

e. Similarly, the `SELFWEIGHT` specification when used in a Reference Load is also capable of handling a list.

Example

```
LOAD 1 DEAD AND LIVE LOAD
* includes the selfweight of all members, plates, and solids.  The following
two commands are functionally identical
SELF
SELFWEIGHT y -1.0 all
*
* Factored weight for members, plates, and solids 4 through 10 along global x
direction
SELFWEIGHT X 1.4 LIST 4 TO 10
* Includes weight of al members, plates, and solids associated with the group
_PLATEGRP1
SELF X 1.0 _PLATEGRP1
```

Related Links

- *M. To add selfweight load* (on page 831)

TR.32.9.2 Surface Selfweight Load

Used to calculate and apply the self weight of surface elements in the structure for analysis.

Note: This feature requires STAAD.Pro V8i (SELECTseries 3) or higher.

General Format

```
SSELFWEIGHT ( { X | Y | Z } f1) ( LIST surface-list | ALL )
```

Where:

X, Y, & Z	represent the global direction in which the selfweight acts. The default direction is along the Y axis.
f1	The factor to be used to multiply the selfweight. The default value is -1.0.
surface-list	surface element list or group. All list specifications such as explicit list including `TO` and `BY`, range (XR, YR, ZR), parallel to (X, Y, Z) and groups are supported. If a group is used, the selfweight command must be repeated for each group. If list is not provided, all structural components are used for body weight calculation unless those are inactive either by `INACTIVE` command specification or internally set due to TENSION/COMPRESSION ONLY specification.

See TR.32.9.1 Selfweight Loads (on page 2496) for Description and Notes pertaining to self weight.

Example

```
LOAD 1 LOADTYPE none TITLE DEAD LOAD
SURFACE LOAD
1 PR 1 0 0
SSELFWEIGHT Y -1
```

Related Links

- *M. To add a surface selfweight load* (on page 841)

TR.32.10 Dynamic Loading Specification

The command specification needed to perform response spectrum analysis and time-history analysis is explained in the following sections.

Note: STAAD.Pro is also capable of generating floor spectrum responses for a time history analysis. Refer to TR.37.10 Floor Spectrum Command (on page 2657) for details on adding this to the analysis commands.

Related Links

- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *G.15.1 Joint Loads* (on page 2128)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.32.10.1 Response Spectrum Analysis

Various methods for performing response spectrum analysis have been implemented in STAAD.Pro. They include a generic method that is described in most text books, as well as code based methods like those required by the IBC, Eurocode 8, IS 1893, etc. These are described in the following sections.

Table 263: Codes available in STAAD.Pro with Response Spectrum loads

Country	Code	Title
Canada	TR.32.10.1.2 Response Spectrum Specification per NRC 2005 (on page 2506)	National Building Code(NRC/CNRC) of Canada
	TR.32.10.1.3 Response Spectrum Specification per NRC 2010 (on page 2512)	National Building Code(NRC/CNRC) of Canada
China	TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010 (on page 2531)	*Code for seismic design of buildings* GB50011-2010 (2016 Edition)
India	TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534)	Criteria for Earthquake Resistant Design of Structures - Part 1: General Provisions and Buildings and Part 4: Industrial structures including Stack-like structures
	TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016 (on page 2544)	Criteria for Earthquake Resistant Design of Structures - Part 1: General Provisions

Country	Code	Title
	TR.32.10.1.9 Response Spectrum Specification per IS: 1893 (Part 4)-2015 (on page 2555)	Criteria for Earthquake Resistant Design of Structures Part 4 Industrial Structures Including Stack-Like Structures
Europe	TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2520)	Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings (published 1996)
	TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2525)	Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings (published 2004)
Russia	TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81 (on page 2584)	Строительство в сейсмических районах (Construction in Seismic Regions)
	TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011 (on page 2589)	Строительство в сейсмических районах (Construction in Seismic Regions)
US	TR.32.10.1.10 Response Spectrum Specification per IBC 2006 (on page 2561)	International Building Code, 2006 edition
	TR.32.10.1.11 Response Spectrum Specification per IBC 2012 (on page 2566)	International Building Code, 2012 edition
	TR.32.10.1.12 Response Spectrum Specification per IBC 2015 (on page 2573)	International Building Code, 2015 edition
	TR.32.10.1.13 Response Spectrum Specification per IBC 2018 (on page 2578)	International Building Code, 2018 edition

Related Links

- *G.17.3.4 Response Spectrum* (on page 2163)

TR.32.10.1.1 Response Spectrum Specification - Custom

This command may be used to specify and apply a custom (i.e., “generic” method) `RESPONSE SPECTRUM` loading for dynamic analysis.

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. The maximum number of response spectrum load cases allowed in one run is 50.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, refer to TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154). An illustration of mass modeling is available, with explanatory comments, in Example Problem No.11.

General Format

```
SPECTRUM comb-method *{ X f1 | Y f2 | Z f3 } { ACCELERATION | DISPLACEMENT } (SCALE f4)
```

```
{DAMP f5 | CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7) (FF1 f8)
(FF2 f9) ( { DOMINANT f10 | SIGN } ) (SAVE) (IMR f11) (STARTCASE f12)
```

Note: The data from SPECTRUM through SCALE above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, enter Spectra in one of these two input forms (i.e., explicit values or an external file):

```
{ p1 v1; p2 v2; p3 v3; … | FILE filename }
```

Where:

Table 264: Parameters used for generic model response spectrum

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
SCALE *f4*	1.0	Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Parameter	Default Value	Description
MISSING *f6*	0	Optional parameter to use the "Missing Mass" method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec^2 for this missing mass mode is the *f6* value entered in length per second squared units (this value is not multiplied by `SCALE`). If *f6* is zero, then the spectral acceleration at the `ZPA` *f7* frequency is used. If *f7* is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. For SRSS, CQC, and TEN the results of this calculation are SRSSed with the modal combination results. For ABS, missing mass is ignored. For ASCE, the missing mass result is algebraically added with the rigid parts of the extracted modes. For ASCE, the MIS option is assumed to be on. If any of *f6*, *f7*, *f8*, or *f9* are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.
FF1 *f8*	2 [Hz]	The *f1* parameter defined in the ASCE 4-98 standard in Hz units. For `ASCE` option only.
FF2 *f9*	33 [Hz]	The f2 parameter defined in the ASCE 4-98 standard in Hz units. For `ASCE` option only.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \le 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`ACCELERATION` or `DISPLACEMENT` indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1/\omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used

- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

LINEAR or LOGARITHMIC — Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When FILE *filename* is entered, the interpolation along the damping axis will be linear.

> **Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SAVE — This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

SIGN — This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

> **Caution:** Do *not* enter DOMINANT parameter with this option.

Spectra data is input in one of these two input forms:

1. p1, v1; p2, v2; …. ; pn, vn. Data is part of input, immediately following the SPECTRUM command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec^2) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.81, 386.4, etc.). Also note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.
2. FILE *filename* data is in a separate file, using the format described in File Format for Spectra Data (on page 2504).

   When the File *filename* command has been provided, then you must have the spectra curve data on a file named filename prior to starting the analysis. The format of the FILE spectra data allows spectra as a function of damping as well as period.

> **Note:** If the FILE *filename* command is entered, it must be with the first spectrum case and will be used for all spectrum cases.

No File *filename* command needs to be entered with the remaining spectrum cases. The filename may not be more than 72 characters in length.

Examples

An example using joint loads and the SRSS combination method:

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
```

```
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
```

An example using member loads and the CQC combination method:

```
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
MEMBER LOADS
5  CON  GX  5.0  6.0
5  CON  GY  5.0  6.0
5  CON  GX  7.5  10.0
5  CON  GY  7.5  10.0
5  CON  GX  5.0  14.0
5  CON  GY  5.0  14.0
SPECTRUM CQC X 1.0 ACC DAMP 0.05 SCALE 32.2
0.03  1.00  ;  0.05  1.35
0.1  1.95  ;  0.2  2.80
0.5  2.80  ;  1.0  1.60
```

Multiple Response Spectra

If there is more than one response spectrum defined in the input file, the load data (representing the dynamic weight) should accompany the first set of spectrum data only. In the subsequent load cases, only the spectra should be defined. See example below.

```
LOAD 1 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2  IMR 2 STARTCASE 11
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
PERFORM ANALYSIS
CHANGE
*
LOAD 2 SPECTRUM IN Y-DIRECTION
SPECTRUM SRSS Y 1.0 ACC SCALE 32.2
0.20 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48
1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42
```

File Format for Spectra Data

The format of the `FILE` spectra data allows spectra as a function of damping as well as period. The format is:

```
   Dataset 1      MDAMPCV NPOINTCV                      (no of values = 2)
   Dataset 2      Damping Values in ascending order     (no of values = Mdampcv)
```

```
   Dataset 3a     Periods                                (no of values = Npointcv)
           3b     Spectra                                (no of values = Npointcv)
```

For ASCE, the MIS option is assumed to be on. If any of f6, f7, f8, f9 are not entered the defaults will be used.

Repeat Data set 3 *Mdampcv* times (3a,3b , 3a,3b , 3a,3b , etc.) (i.e., for each damping value).

Data sets 2, 3a and 3b must have exactly *Npointcv* values each. Blanks or commas separate the values. The data may extend to several lines. Do not end lines with a hyphen (-). No comment lines (*) or semi-colons. Multiple values may be entered per line.

where

MDAMPCV	=	Number of damping values for which there will be separate Spectra vs. Period curves.
NPOINTCV	=	Number of points in each Spectra vs. Period curve. If NPOINTCV is negative, then the period-spectra values are entered as pairs.

Examples of Spectra Data files

An example of spectral data for use in the X direction:

```
1,-10
0.05
0.20 0.2 0.40 0.25 0.60 0.35 0.80 0.43 1.0 0.47
1.2 0.5 1.4 0.65 1.6 0.67 1.8 0.55 2.0 0.43
```

An example of spectral data for use in the Z direction:

```
1 10
0.05
0.20 0.40 0.60 0.80 1.0 1.2 1.4 1.6 1.8 2.0
0.1 0.15 0.33 0.45 0.48 0.51 0.63 0.67 0.54 0.42
```

Note: It is important that any STAAD plain text file be encoded with ANSI/UTF-8. If you use a text editor in a non-English user interface, ensure that the correct encoding is used when saving the file.

Individual Modal Response Case Generation

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

Related Links

- *M. To add a generic response spectrum* (on page 861)
- *G.17.3.4 Response Spectrum* (on page 2163)
- *M. To add an IBC 2000 response spectrum* (on page 862)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.32.10.1.2 Response Spectrum Specification per NRC 2005

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2005 edition of the National Research Council specification *National Building Code of Canada* (NBC) , for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method NRC 2005 ( TORSION (DECCENTRICITY f20) (ECCENTRICITY f21) ) *{ X
f1 | Y f2| Z f3} { ACC | DIS } ( SCALE f4)
```

```
{ DAMP f5| CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7)
({ DOMINANT f10| SIGN }) (SAVE)  (IMR f11) (STARTCASE f12)
```

Note: The data from `SPECTRUM` through `SCALE` must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following spectrum data which must be started on a new line:

```
{ p1 v1; p2 v2; p3 v3; … pn vn | FILE filename  }
```

Where:

Table 265: Parameters used for NRC 2005 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f20*	1.0	Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).
ECCENTRICITY *f21*	0.05	Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

Parameter	Default Value	Description
SCALE *f4*	1.0	Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.

Parameter	Default Value	Description
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.

comb-method = { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \le 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

TORSION indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Inherent and Accidental Torsion (on page 2510) for additional information.

Note: If TORSION is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

ACCELERATION or DISPLACEMENT indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1/\omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

DAMP, MDAMP, and CDAMP select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

LINEAR or LOGARITHMIC Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When `FILE` *filename* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SIGN This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

Caution: Do *not* enter `DOMINANT` parameter with this option.

SAVE This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Spectra data is input in one of these two input forms:

1. `p1, v1; p2, v2; …. ; pn, vn.` Data is part of input, immediately following the `SPECTRUM` command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec^2) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.81, 386.4, etc.). Also note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.
2. FILE *filename* data is in a separate file, using the format described in File Format for Spectra Data (on page 2504).

When the `File filename` command has been provided, then you must have the spectra curve data on a file named `filename` prior to starting the analysis. The format of the FILE spectra data allows spectra as a function of damping as well as period.

Note: If the `FILE filename` command is entered, it must be with the first spectrum case and will be used for all spectrum cases.

No `File filename` command needs to be entered with the remaining spectrum cases. The `filename` may not be more than 72 characters in length.

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Inherent and Accidental Torsion

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity e_{di} = *f20*×e_{si} + *f12*×b_i where *f20* = 1.0 and *f21* = (±) 0.05

   where

e_{si}	=	static eccentricity between center of mass and center of rigidity at floor i
b_i	=	floor plan dimension in the direction of earthquake loading
3. Torsional moment is calculated at each floor.

   $M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.
7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = DEC \times e_{si} + ECC \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
ECC and DEC	=	Factors to determine the design eccentricity. These are input parameters.

Example

```
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS NRC 2005 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3
24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61436; 1.08 9.08023; 1.14 8.60233; 1.2 8.17221;
1.26 7.78306; 1.32 7.42928; 1.38 7.10627; 1.44 6.81018; 1.5 6.53777;
1.56 6.28632; 1.62 6.05349; 1.68 5.83729; 1.74 5.63601; 1.8 5.44814;
1.86 5.2724; 1.92 5.10763; 1.98 4.95286; 2.04 4.80718; 2.1 4.66984;
2.16 4.54012; 2.22 4.41741; 2.28 4.30116; 2.34 4.19088; 2.4 4.08611;
2.46 3.98645; 2.52 3.89153; 2.58 3.80103; 2.64 3.71464; 2.7 3.63209;
2.76 3.55314; 2.82 3.47754; 2.88 3.40509; 2.94 3.3356; 3 3.26889; 3.06
3.20479;
3.12 3.14316; 3.18 3.08385; 3.24 3.02675; 3.3 2.97171; 3.36 2.91865;
3.42 2.86744; 3.48 2.818; 3.54 2.77024; 3.6 2.72407; 3.66 2.67941; 3.72
2.6362;
3.78 2.59435; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643; 4.02 2.45166;
4.08 2.45166; 4.14 2.45166; 4.2 2.45166; 4.26 2.45166; 4.32 2.45166;
4.38 2.45166; 4.44 2.45166; 4.5 2.45166; 4.56 2.45166; 4.62 2.45166;
4.68 2.45166; 4.74 2.45166; 4.8 2.45166; 4.86 2.45166; 4.92 2.45166;
4.98 2.45166; 5.04 2.45166; 5.1 2.45166; 5.16 2.45166; 5.22 2.45166;
5.28 2.45166; 5.34 2.45166; 5.4 2.45166; 5.46 2.45166; 5.52 2.45166;
5.58 2.45166; 5.64 2.45166; 5.7 2.45166; 5.76 2.45166; 5.82 2.45166;
5.88 2.45166; 5.94 2.45166;
```

Related Links

- *V. NRC 2005 Response Spectrum* (on page 3614)

TR.32.10.1.3 Response Spectrum Specification per NRC 2010

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2010 edition of the National Research Council specification *National Building Code of Canada* (NBC) , for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method NRC 2010 ( TORSION  (DECCENTRICITY f20) (ECCENTRICITY f21) ) *{ X
f1 | Y f2| Z f3} { ACCELERARTION | DISCPLACEMENT } ( SCALE f4)
```

```
{ DAMP f5| CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC } ) (MISSSING f6) (ZPA f7)
({ DOMINANT f10| SIGN }) (SAVE)  (IMR f11) (STARTCASE f12)
```

Note: The data from `SPECTRUM` through `SCALE` must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following spectrum data which must be started on a new line:

```
{ p1 v1; p2 v2; p3 v3; … pn vn | FILE filename  }
```

Table 266: Parameters used for NRC 2010 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f20*	1.0	Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).
ECCENTRICITY *f21*	0.05	Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
SCALE *f4*	1.0	Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.

Parameter	Default Value	Description
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.

comb-method = `{ SRSS | ABS | CQC | ASCE | TEN | CSM | GRP }` are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`TORSION` indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Inherent and Accidental Torsion (on page 2510) for additional information.

Note: If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

`ACCELERATION` or `DISPLACEMENT` indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1 / \omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *`f2`* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`LINEAR` or `LOGARITHMIC` Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When `FILE` *`filename`* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

`SIGN` This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

Caution: Do *not* enter `DOMINANT` parameter with this option.

SAVE	This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Inherent and Accidental Torsion

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity $e_{di} = f20 \times e_{si} + f12 \times b_i$ where *f20* = 1.0 and *f21* = (±) 0.05

   where

e_{si}	=	static eccentricity between center of mass and center of rigidity at floor i
b_i	=	floor plan dimension in the direction of earthquake loading

3. Torsional moment is calculated at each floor.

   $M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.
7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = DEC \times e_{si} + ECC \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
ECC and DEC	=	Factors to determine the design eccentricity. These are input parameters.

Example

This example input file demonstrates a seismic load using the equivalent force method and a seismic response spectrum analysis per NRC 2010.

```
STAAD SPACE EXAMPLE PROBLEM FOR NRC LOAD
START JOB INFORMATION
ENGINEER DATE 15-Jan-16
END JOB INFORMATION
UNIT METER KN
```

```
JOINT COORDINATES
1 0 0 0; 2 3.5 0 0; 3 7 0 0; 4 13.5 0 0; 5 0 0 3.5; 6 3.5 0 3.5;
7 7 0 3.5; 8 13.5 0 3.5; 9 0 0 7; 10 3.5 0 7; 11 7 0 7; 12 13.5 0 7;
13 0 0 12.5; 14 3.5 0 12.5; 15 7 0 12.5; 16 13.5 0 12.5; 17 0 3.5 0;
18 3.5 3.5 0; 19 7 3.5 0; 20 13.5 3.5 0; 21 0 3.5 3.5; 22 3.5 3.5 3.5;
23 7 3.5 3.5; 24 13.5 3.5 3.5; 25 0 3.5 7; 26 3.5 3.5 7; 27 7 3.5 7;
28 13.5 3.5 7; 29 0 3.5 12.5; 30 3.5 3.5 12.5; 31 7 3.5 12.5;
32 13.5 3.5 12.5; 33 0 7 0; 34 3.5 7 0; 35 7 7 0; 36 13.5 7 0;
37 0 7 3.5; 38 3.5 7 3.5; 39 7 7 3.5; 40 13.5 7 3.5; 41 0 7 7;
42 3.5 7 7; 43 7 7 7; 44 13.5 7 7; 45 0 7 12.5; 46 3.5 7 12.5;
47 7 7 12.5; 48 13.5 7 12.5; 49 0 10.5 0; 50 3.5 10.5 0; 51 7 10.5 0;
52 13.5 10.5 0; 53 0 10.5 3.5; 54 3.5 10.5 3.5; 55 7 10.5 3.5;
56 13.5 10.5 3.5; 57 0 10.5 7; 58 3.5 10.5 7; 59 7 10.5 7;
60 13.5 10.5 7; 61 0 10.5 10.5; 62 3.5 10.5 10.5; 63 7 10.5 10.5;
64 13.5 10.5 10.5;
MEMBER INCIDENCES
101 17 18; 102 18 19; 103 19 20; 104 21 22; 105 22 23; 106 23 24;
107 25 26; 108 26 27; 109 27 28; 110 29 30; 111 30 31; 112 31 32;
113 33 34; 114 34 35; 115 35 36; 116 37 38; 117 38 39; 118 39 40;
119 41 42; 120 42 43; 121 43 44; 122 45 46; 123 46 47; 124 47 48;
125 49 50; 126 50 51; 127 51 52; 128 53 54; 129 54 55; 130 55 56;
131 57 58; 132 58 59; 133 59 60; 134 61 62; 135 62 63; 136 63 64;
201 17 21; 202 18 22; 203 19 23; 204 20 24; 205 21 25; 206 22 26;
207 23 27; 208 24 28; 209 25 29; 210 26 30; 211 27 31; 212 28 32;
213 33 37; 214 34 38; 215 35 39; 216 36 40; 217 37 41; 218 38 42;
219 39 43; 220 40 44; 221 41 45; 222 42 46; 223 43 47; 224 44 48;
225 49 53; 226 50 54; 227 51 55; 228 52 56; 229 53 57; 230 54 58;
231 55 59; 232 56 60; 233 57 61; 234 58 62; 235 59 63; 236 60 64;
301 1 17; 302 2 18; 303 3 19; 304 4 20; 305 5 21; 306 6 22; 307 7 23;
308 8 24; 309 9 25; 310 10 26; 311 11 27; 312 12 28; 313 13 29;
314 14 30; 315 15 31; 316 16 32; 317 17 33; 318 18 34; 319 19 35;
320 20 36; 321 21 37; 322 22 38; 323 23 39; 324 24 40; 325 25 41;
326 26 42; 327 27 43; 328 28 44; 329 29 45; 330 30 46; 331 31 47;
332 32 48; 333 33 49; 334 34 50; 335 35 51; 336 36 52; 337 37 53;
338 38 54; 339 39 55; 340 40 56; 341 41 57; 342 42 58; 343 43 59;
344 44 60; 345 45 61; 346 46 62; 347 47 63; 348 48 64;
START GROUP DEFINITION
MEMBER
_B1 301 TO 303 305 TO 307 309 TO 311 317 TO 319 321 TO 323 325 TO 327 -
333 TO 335 337 TO 339 341 TO 343 345 TO 347
END GROUP DEFINITION
MEMBER PROPERTY CANADIAN
101 TO 136 201 TO 236 PRIS YD 0.4 ZD 0.3
301 TO 303 305 TO 307 309 TO 311 317 TO 319 321 TO 323 325 TO 327 333 -
334 TO 335 337 TO 339 341 TO 343 345 TO 347 TABLE ST W460X52
304 308 312 TO 316 320 324 328 TO 332 336 340 344 348 TABLE ST W530X85
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.5e+007
POISSON 0.17
DENSITY 24
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 2.17185e+007
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-005
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 101 TO 136 201 TO 236
MATERIAL STEEL MEMB 301 TO 348
SUPPORTS
1 TO 16 FIXED
CUT OFF MODE SHAPE 10
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
17 TO 48 FX 7
49 TO 64 FX 3.5
17 TO 48 FY 7
49 TO 64 FY 3.5
17 TO 48 FZ 7
49 TO 64 FZ 3.5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3.5
DIA 2 TYPE RIG HEI 7
DIA 3 TYPE RIG HEI 10.5
*** Equivelant Lateral Force Definition ***
DEFINE NRC 2010 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1.2 MVZ 1.2 -
 RDX 1.4 RDZ 3 ROX 1.5 ROZ 1.5 STX 3 STZ 4 MD 1
*****************************************************
*** X-DIRECTION
LOAD 1 FX+TX
NRC LOAD X 1 DEC 1 ACC 0.1
LOAD 2 FX-TX
NRC LOAD X 1 DEC 1 ACC -0.1
LOAD 3 -FX-TX
NRC LOAD X -1 DEC 1 ACC -0.1
LOAD 4 -FX+TX
NRC LOAD X -1 DEC 1 ACC 0.1
*** Z-DIRECTION
LOAD 5 FZ+TZ
NRC LOAD Z 1 DEC 1 ACC 0.1
LOAD 6 FZ-TZ
NRC LOAD Z 1 DEC 1 ACC -0.1
LOAD 7 -FZ-TZ
NRC LOAD Z -1 DEC 1 ACC -0.1
LOAD 8 -FZ+TZ
NRC LOAD Z -1 DEC 1 ACC 0.1
*****************************************************
**** RESPONSE SPECTRUM ****
```

```
*** X-DIRECTION
LOAD 9
SPECTRUM CQC NRC 2010 TOR X 1 ACC DAMP 0.05
0.03 1; 0.05 1.35; 0.1 1.95; 0.2 2.8; 0.5 2.8; 1 1.6;
LOAD 10
*** Z-DIRECTION
SPECTRUM CQC NRC 2010 TOR Z 1 ACC DAMP 0.05
0.03 1; 0.05 1.35; 0.1 1.95; 0.2 2.8; 0.5 2.8; 1 1.6;
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
PRINT DIA CR
FINISH
```

Related Links

- *V. NRC 2010 Response Spectrum* (on page 3604)

TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 1994 edition of Eurocode 8 (EC8) for dynamic analysis.

General Format

```
SPECTRUM comb-method  EURO {ELASTIC | DESIGN} *{ X f1| Y f2| Z f3} ACCELERATION
```

```
{DAMP f5| CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC} ) (MISSING f6) (ZPA f7) ({ DOMINANT
f10| SIGN }) (SAVE)  (IMR f11) (STARTCASE f12)
```

Note: The data from `SPECTRUM` through `ACC` must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, the response spectra is input using following standard input parameters:

```
SOIL TYPE { A | B | C} ALPHA f8 Q f9
```

Unlike the custom defined response spectra, the EC 8 response spectra does not input use frequency-acceleration pairs. Based on the type of Response Spectra (Elastic/Design), Soil Type, Alpha, and Q, the program generates the applicable response spectra curve using the guidelines of section 4.2.2 or 4.2.4 of Eurocode 8 as applicable.

Where:

Table 267: Parameters used for Eurocode 8 1994 response spectrum

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

Parameter	Default Value	Description
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the ZPA *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING *f6* is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.

Parameter	Default Value	Description
ALPHA *f8*		The design ground acceleration expressed in terms of acceleration due to gravity(g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.
Q *f9*		The behavior factor used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2 (1+r) r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

ELASTIC or DESIGN	The response spectrum loading can be based on either **Elastic** or **Design** response spectra. Refer to Eurocode 8. The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed as "Design Response Spectrum." STAAD.Pro generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8. So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.
ACCELERATOIN	indicates that the Acceleration spectra will be entered. **Note:** Eurocode 8 does not have provisions displacement response spectra.
DAMP, MDAMP, and CDAMP	select source of damping input: • DAMP indicates to use the *f2* value for all modes • MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used • CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification
LINEAR or LOGARITHMIC	Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. When FILE *filename* is entered, the interpolation along the damping axis will be linear. **Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.
SIGN	This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option. **Caution:** Do *not* enter DOMINANT parameter with this option.
SAVE	This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

`SOIL TYPE`	This parameter is used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds • Type A: for Rock or stiff deposits of sand • Type B: for deep deposits of medium dense sand ,gravel or medium stiff clays. • Type C: Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil. Refer to section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Description

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154).

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area are generated by sources varying widely in location and other parameters. In those cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

Example

```
LOAD 2 SPECTRUM X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP 0.05 -
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2004 edition of Eurocode 8, "General Rules, seismic actions and rules for buildings", BS EN 1998-1:2004. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method EURO 2004 {ELASTIC | DESIGN} {RS1 | RS2} *{ X f1| Y f2| Z f3}
ACCELERATION
```

```
{DAMP f5| CDAMP | MDAMP } ( {LINEAR | LOGARITHMIC} ) (MISSING f6) (ZPA f7) ({ DOMINANT
f10 | SIGN }) (SAVE)  (IMR f11) (STARTCASE f12)
```

Note: The data SPECTRUM through ACC must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Starting on the next line, the response spectra is input using following standard input parameters:

```
SOIL TYPE { A | B | C | D | E } ALPHA f8 Q f9
```

Unlike the custom defined response spectra the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra(Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 3.2.2.2 or 3.2.2.3 or 3.2.2.5 of Eurocode 8: 2004 as applicable.

Where:

Table 268: Parameters used for Eurocode 8 2004 response spectrum

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.

Parameter	Default Value	Description
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.
ALPHA *f10*		Alpha is the “Ground Acceleration On Type A Ground” and defined in Eurocode 8 as a_g, as used in equations 3.2 – 3.5. Refer to the Eurocode for further information. **Note:** The specified alpha factor in STAAD.Pro is given as the *ratio* of the ground acceleration (a_g) to acceleration due to gravity. That is, the value for *f10*should be a factor of G, not an actual acceleration (e.g., if the value of $a_g = 1.5\ m/s^2$, then enter the ALPHA value of $1.5\ m/s^2\ /\ 9.81\ m/s^2 = 0.15$).
Q *f11*		Q is the 'Behaviour Factor' and is an approximation of the ratio of the seismic forces that the structure would experience if its response was completely elastic with 5% viscous damping, to the seismic forces that may be used in design, with a conventional elastic analysis model, still ensuring a satisfactory response of the structure.

comb-method = { SRSS | ABS | CQC | ASCE | TEN | CSM | GRP } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n/\omega_m \leq 1.0$$

> **Note:** The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

> **Note:** If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

See G.17.3 Dynamic Analysis (on page 2154), TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2344), and TR.34 Frequency Calculation (on page 2614)

The keywords EURO 2004 is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

ELASTIC or DESIGN The response spectrum loading can be based on either **Elastic** or **Design** response spectra. Refer to Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed as “Design Response Spectrum.” STAAD.Pro generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8.

So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

RS1 or RS2 Two types of response spectra curve can be generated based on either response spectra type 1 curve (RS1) or response spectra type 2 curve (RS2).

ACCELERATOIN indicates that the Acceleration spectra will be entered.

> **Note:** Eurocode 8 does not have provisions displacement response spectra.

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the $f2$ value for all modes
- MDAMP indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

LINEAR or LOGARITHMIC	Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. When `FILE` *`filename`* is entered, the interpolation along the damping axis will be linear. **Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.
SIGN	This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option. **Caution:** Do *not* enter `DOMINANT` parameter with this option.
SAVE	This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).
SOIL TYPE	This parameter is used to define the subsoil conditions based on which the response spectra will be generated as defined in Table 3.1 Ground Types. Based on the subsoil conditions the soil types may be of five kinds: • Type `A`: rock or other rock-like geographical formation. • Type `B`: very dense sand, gravel or very stiff clay. • Type `C`: Deep deposits of dense or medium dense sand, gravel or stiff clay. • Type `D`: Deposits of loose-to-medium cohesionless soil or of predominantly soft to firm cohesive soil. • Type `E`: Surface alluvium layer Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Description

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154).

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area is generated by sources varying widely in location and other parameters. In such cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

Example

```
LOAD 2 SPECTRUMIN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP 0.05 -
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Related Links

- *M. To add an EC8 response spectrum* (on page 867)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2016 edition of the Chinese specification *Code for seismic design of building*, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM { CQC | SRSS } GB50011 (2010) (TORSION) (DECCENTRICITY f8) (ECCENTRICITY f9)
*{ X f1 | Y f2| Z f3 } ALPHA-
```

```
{DAMP f4 | CDAMP | MDAMP }  ( { LINEAR | LOGARITHMIC } ) (MISSING f5) ( ZPA f6 )
({ DOMINANT f10| SIGN }) (SAVE)  (IMR f11) (STARTCASE f12) -
```

```
{ ( INTENSITY f9 ) ( FREQUENT | FORTIFIED | RARE ) ( GROUP f11 ) ( SCLASS f12 ) }
```

Note: The data from `SPECTRUM` through `ALPHA` must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:

Table 269: Parameters used for GB 50011-2010 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f8*	0	(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. **Note:** The torsion arising due to dynamic eccentricity (i.e,. static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion. The dynamic eccentricity is automatically calculated by the program while you can specify the amount of accidental eccentricity (if not specified, the default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).
ECCENTRICITY *f9*	0.05	A factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If the value is positive, it indicates clockwise torsion whereas a negative value indicates counterclockwise torsion.

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	-	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero if the global axis is not specified. If the direction is specified, then the default factor for X and Z is 1.0 where as the default factor for Y is 0.65. At least one direction must be specified. Any directions that is not specified will default to zero.
DAMP *f4*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f5*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2. If the "missing mass" option is used, then this value must be provided. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f6*	33 (Hz)	The zero-period acceleration value used with the MISSING option only.
DOMINANT *f7*	1 (1st Mode)	Dominant mode method. All results will have the same sign as mode number *f7* alone would have if it were excited then the results were used as a static displacements result. Defaults to mode 1 if no value is entered. If a value of 0 is entered, then the mode with the greatest % participation in the excitation direction will be used (only 1 direction factor may be nonzero).
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through *f11* will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.

Parameter	Default Value	Description
INTENSITY *f9*	7	Fortification seismic intensity. Permitted values are: 6, 7, 7A, 8, 8A, or 9
GROUP *f10*	2	Design seismic group.
SCLASS *f11*	2	Site class.

`{ SRSS | CQC }` are methods of combining the responses from each mode into a total response. The CQC methods requires damping. SRSS method do not use damping unless spectra-period curves are made a function of damping. CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic method and is recommended.

SRSS Square Root of Summation of Squares method.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \le 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

DAMP, MDAMP, and CDAMP select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

LINEAR | LOGARITHMIC Specifies the type of interpolation of the input seismic coefficient versus time period curves for determining the seismic coefficient value for a mode given its period. Linear is the default if not specified.

SIGN This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign.

Warning: Do *not* enter `DOMINANT` parameter with this option.

SAVE This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

FREQUENT | FORTIFIED | RARE	The earthquake type.

Inherent and Accidental Torsion

Note: STAAD.Pro does not support the coupled torsion methodology as per GB50011-2010. This implementation enables the general "inherent and accidental torsion" for GB50011 response spectrum analysis. Note that this implementation does *not* comply with Cl. 5.2.3 of GB50011-2010.

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either CQC or SRSS) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to the positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. For example, consider two joints J2 and J3 whose maximum joint displacement in the global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X-direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, the torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from the response spectrum along with torsion for all modes are combined using a specified modal combination method to get the final maximum possible joint displacements.

Related Links

- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *V. GB 50011 2010 Response Spectrum* (on page 3376)

TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per IS: 1893 (Part 1)-2002 for dynamic analysis.

The seismic load generator can be used to generate lateral loads in the X, Y, and Z directions.

Note: This facility has *not* been developed for cases where the Z axis is set to be the vertical direction using the `SET Z UP` command.

General Format

The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the ACC and DOMINANT or SIGN commands, respectively.

```
SPECTRUM comb-method 1893 ( TORSION (DECCENTRICITY f8) (ECCENTRICITY f9) ) *{ X f1 | Y
f2 | Z f3 }
```

```
{ ACCELERATION | DISPLACEMENT } (SCALE f4) {DAMP f5 | CDAMP | MDAMP } (MISSING f6)
(ZPA f7) (IGNNORE f13)
({ DOMINANT f10 | SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

The following command (SOIL TYPE parameter *or* response spectra data pairs) must be in a separate line.

```
{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;…PN,VN } }
```

The following command, if present, must be on a separate line. This performs the option al soft story check.

```
( CHECK SOFT STORY )
```

The following command, if present, must be on a separate line. This performs the story drift check.

```
( CHECK STORY DRIFT ) (RF f14)
```

Where:

Table 270: Parameters used for IS: 1893 (Part 1) 2002 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f8*	When ECC > 0, DEC defaults to 1.5 When ECC < 0, DEC defaults to 1.0	(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. **Note:** The torsion arising due to dynamic eccentricity (i.e,. static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.8.2 of the code. The dynamic eccentricity is automatically calculated by the program while you can specify the amount of accidental eccentricity (if not specified, the default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered). For details See Torsion Methodology (on page 2542).

Parameter	Default Value	Description
ECCENTRICITY *f9*	0.05	It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If the *f9* value is positive, it indicates clockwise torsion whereas a negative value indicates counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
SCALE *f4*	1.0	Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.

Parameter	Default Value	Description
IGNORE *f13*	0.009	(Optional input) It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases. **Note:** If the value of *f14* is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using `IGNORE` command. If the `MODE SELECT` command is provided along with the `IGNORE` command, the number of modes excluded from the analysis will be those deselected by the `MODE SELECT` command and *also* those deselected by the `IGNORE` command.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.

Parameter	Default Value	Description
SOIL TYPE *f11*		The soil type present. Depending upon time period, types of soil and damping, average response acceleration coefficient, S_a/g is calculated. 1 = for rocky or hard soil 2 = medium soil 3 = soft soil sites
custom P1,V1; P2,V2; P3,V3; … Pn,Vn		Data is part of input immediately following spectrum command for a "custom" response spectrum. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/ sec^2). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc). **Note:** Do not enter if a SOIL TYPE *f11* value is specified.
RF *f14*		The response reduction factor. If not specified, the program will look for the factor defined under `DEFINE 1893 LOAD` (refer to TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)). If none is provided there either, a factor of 1.0 is assumed. The response reduction factor represents ratio of maximum seismic force on a structure during specified ground motion if it were to remain elastic to the design seismic force. Actual seismic force is reduced by a factor `RF` to obtain design force.)

`1893` indicates the analysis as per IS:1893(Part 1)-2002 procedures.

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

Note: CQC, SRSS, and CSM Grouping methods are recommended by IS:1893 (Part 1) –2002.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1 + r)r^{2/3}}{(1 - r^2)^2 + 4\zeta^2 r(1 + r)^2}$$

r = $\omega_n / \omega_m \leq 1.0$

> **Note:** The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

`TORSION` indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Torsion (on page 2542) for additional information.

> **Note:** If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

`ACCELERATION or DISPLACEMENT` indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1/\omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`SIGN` This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

> **Caution:** Do *not* enter `DOMINANT` parameter with this option.

`SAVE` This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

`CHECK SOFT STORY` indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking. Refer to TR.28.2.1 Soft Story Checking (on page 2331) for details.

`CHECK STORY DRIFT` indicates that a story drift check is to be performed.

Tip: This is done in place of post-analysis story drift checks for IS 1893-2002.

Methodology

The design lateral shear force at each floor in each mode is computed by STAAD.Pro in accordance with the Indian IS: 1893 (Part 1)-2002 equations 7.8.4.5c and 7.8.4.5d.

$$Q_{ik} = A_k \cdot \phi_{ik} \cdot P_k \cdot W_k$$

and

$$V_{ik} = \sum_{i=i+1}^{n} Q_{ik}$$

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1)-2002.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads.

1. You provide the value for $Z/2 \cdot I/R$ as factors for input spectrum. You calculate the expression $Z/2 \cdot I/R$ and provide these values using the terms `f1`, `f2`, and `f3` and applicable, where these terms an explained in the table below.
2. The program calculates time periods for first six modes or as specified.
3. The program calculates Sa/g for each mode utilizing time period and damping for each mode.
4. The program calculates the design horizontal acceleration spectrum value A_k for each mode.
5. The program then calculates mode participation factor for each mode.
6. The peak lateral seismic force at each floor in each mode is calculated.
7. All response quantities for each mode are calculated.
8. The peak response quantities are then combined as per the specified method (`SRSS, CQC, ABS, CSM` or `TEN`) to get the final results.

Note: For vertical motions (i.e., when forces are applied in the Y direction), the resulting forces are taken as 2/3 of the horizontal forces as per 6.4.5 of the IS 1893 2002 code.

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

Refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Notes

a. The design base shear V_B, calculated from the Response Spectrum method, is compared with the base shear V_b, calculated by empirical formula for the fundamental time period. If V_B is less than V_b, all of the response quantities are multiplied by V_b / V_B as per Clause 7.8.2.

For this, the following input is necessary before defining any primary load case.

```
DEFINE  1893  LOAD
ZONE f1 1893-spec
SELFWEIGHT
JOINT WEIGHT
joint-list   WEIGHT w
MEMBER  WEIGHT
...
UNI   v1 v2 v3
mem-list
CON v4  v5
...
CHECK SOFT STORY

1893-Spec  = {RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)}
```

Refer to TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387) for full details on this command structure.

Note: STAAD.Pro does not calculate the fundamental frequency of the structure needed for the empirical base shear V_b. calculation; so you must enter either the ST parameter or the PX and PZ parameters in the `DEFINE 1893 LOAD` data.

b. The following interpolation formula is adopted for interpolation between damping values as given in Table 3.

Interpolation and/or extrapolation of ground response acceleration for a particular mode has been made for determining the spectrum ordinates corresponding to the modal damping value for use in Response Spectrum analysis. The relationship that shall be used for this purpose is defined by:

$$Sa = Ae^{-\xi} + B/\xi$$

where

Sa = Spectrum ordinate

ξ = damping ratio

Constants A and B are determined using two known spectrum ordinates Sa_1 and Sa_2 corresponding to damping rations ξ_1 and ξ_2, respectively, for a particular time period and are as follows:

$$A = \frac{Sa_1\xi_1 - Sa_2\xi_2}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

$$B = \frac{\xi_1\xi_2\left(Sa_2 e^{-\xi_1} - Sa_1 e^{-\xi_2}\right)}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

Where:

$\xi_1 < \xi < \xi_2$

c. The story drift in any story shall not exceed 0.004 times the story height as per Clause 7.11.1. To check this, the following command should be given after the analysis command.

```
PRINT STOREY DRIFT
```

A warning message will be printed if story drift exceeds this limitation.

d. If any soft story (as per definition in Table 5 of IS:1893-2002) is detected, a warning message will be printed in the output.

Torsion

The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.9.2 of the IS 1893 code. The dynamic eccentricity is automatically calculated by the program (in both cases of `TOR` and `TOR OPP` options), while the amount of accidental eccentricity can be specified through the `ECC` option (if not specified, default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).

Non-symmetric or torsionally unbalanced buildings are prone to earthquake damage due to coupled lateral and torsional movements (i.e., the translational vibration of the building couples with its torsional vibrations within elastic range). The level of coupling between lateral and torsional vibrations of the building can be larger, thus leading to significantly higher lateral-torsional coupling than that predicted by elastic analysis.

- Cl. 7.8.4.5 of IS 1893 (Part 1) : 2002 is valid for buildings with *regular* or *nominally irregular* plan configurations. For buildings which are *irregular* in plan, it is better to consider torsion from dynamic eccentricity into analysis; even if torsionally coupled vibration is considered during response spectrum analysis.
- Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 states that, in the case that a 3D dynamic analysis is carried out, the dynamic amplification factor 1.5—as given by Cl. 7.9.2—can be replaced by 1.0. This implies that the code also recommends to use Cl. 7.9.2 for all types of buildings by including torsion from both dynamic and accidental eccentricity in the response spectrum analysis.

Torsion Methodology

As per IS1893-2002 code, provision shall be made in all buildings for increase in shear forces on the lateral force resisting elements resulting from the horizontal torsional moment arising due to eccentricity between the center of mass and the center of rigidity.

In response spectrum analysis all the response quantities (i.e. joint displacements, member forces, support reactions, plate stresses, etc) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated. Refer Cl. 7.8.4.5c of IS 1893 (Part 1) : 2002. (Q_{ik} at floor i for mode k)
2. At each floor design eccentricity is calculated. Refer Cl. 7.9.2 and Cl. 7.9.2 Note 2 of Amendment No. 1 January 2005 of IS 1893 (Part 1) : 2002.

   Thus, design eccentricity $e_{di} = f15 \times e_{si} + f12 \times b_i$ where f15 = 1.0 and f12 = (±) 0.05

   Where:

   e_{si} = dynamic eccentricity arising due to center of mass and center of rigidity at floor i (static eccentricity multiplied by dynamic amplification factor 1.0 for response spectrum analysis),
   b_i = floor plan dimension in the direction of earthquake loading.
3. Torsional moment is calculated at each floor. ($M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k)
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

Modal Combination

Steps 1 to 6 are performed for all modes considered and missing mass correction (if any). Finally the peak response quantities from different modal response are combined as per `CQC` or `SRSS` or `TEN PERCENT` or `CSM` method.

Notes

After the analysis is complete following files are generated.

a. Story shear for each mode for each load case is given in the file `<filename>_RESP1893.txt`.
b. Rotational stiffness of each floor is given in the file `<filename>_ROT1893.txt`.
c. Center of mass, center of rigidity, design eccentricity at each floor level and additional shear due to torsion at each floor level for each mode for each load case is given in the file `<filename>_TOR1893.txt`.

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = DEC \times e_{si} + ECC \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
ECC and DEC	=	Factors to determine the design eccentricity. These are input parameters.

IS 1893-2002 clause 7.8.2 defines two equations:

$$e_{di} = 1.5 \times e_{si} + 0.05 \times b_i$$

$$e_{di} = 1.0 \times e_{si} - 0.05 \times b_i$$

By including the optional command `TORSION`, the first equation is used by default. To account for the second equation, you may simply specify `DEC -0.05` (at which point the default for ECC is then 1.0).

Example

Refer to V. IS 1893 2002 Response Spectrum (on page 3588) for a detailed explanation of this example.

```
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS 1893 X 0.036 ACC DAMP 0.05
SOIL TYPE 1
```

Related Links

- *M. To add an IS 1893 response spectrum* (on page 864)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *V. IS 1893 2002 Response Spectrum* (on page 3588)

TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per IS: 1893 (Part 1)-2016 for dynamic analysis.

The seismic load generator can be used to generate lateral loads in the X, Y, and Z directions.

Note: This facility has *not* been developed for cases where the Z axis is set to be the vertical direction using the `SET Z UP` command.

General Format

The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the `ACC` and `DOMINANT` or `SIGN` commands.

```
SPECTRUM comp-method IS1893 2016 ( TORSION (DECCENTRICITY f8) (ECCENTRICITY f9) ) *{ X
f1 | Y f2  | Z f3 }
```

```
{ ACCELERATION | DISPLACEMENT } (SCALE f4) {DAMP f5 | CDAMP | MDAMP } ( { LINEAR |
LOGARITHMIC } ) (MISSING f6) (ZPA f7) (IGNORE f13)
({ DOMINANT f10 | SIGN }) (IMR f11) (STARTCASE f12)
```

The following command (`SOIL TYPE` parameter *or* response spectra data pairs) must be in a separate line.

```
{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;…PN,VN } }
```

Note: The spectrum type options `ACCELERATION` or `DISPLACEMENT` should only be used with custom soil types for IS 1893 2016.

The following command, if present, must be on a separate line. This performs the story drift check.

```
( CHECK STORY DRIFT ) (RF f14)
```

Where:

Table 271: Parameters used for IS: 1893 (Part 1) 2016 response spectrum

Parameter	**Default Value**	**Description**
DECCENTRICITY *f8*	When ECC > 0, DEC defaults to 1.5 When ECC < 0, DEC defaults to 1.0	(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. **Note:** The torsion arising due to dynamic eccentricity (i.e,. static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.8.2 of the code. The dynamic eccentricity is automatically calculated by the program while you can specify the amount of accidental eccentricity (if not specified, the default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered). For details See Torsion Methodology (on page 2553).

Parameter	Default Value	Description
ECCENTRICITY *f9*	0.05	It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If the *f9* value is positive, it indicates clockwise torsion whereas a negative value indicates counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
SCALE *f4*	1.0	Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING` *f6* is entered.

Parameter	Default Value	Description
IGNORE *f13*	0.009	(Optional input) It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases. **Note:** If the value of *f14* is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using `IGNORE` command. If the `MODE SELECT` command is provided along with the `IGNORE` command, the number of modes excluded from the analysis will be those deselected by the `MODE SELECT` command and *also* those deselected by the `IGNORE` command.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.

Parameter	Default Value	Description
SOIL TYPE *f11*		The soil type present. Depending upon time period, types of soil and damping, average response acceleration coefficient, S_a/g is calculated. 1 = for rocky or hard soil 2 = medium soil 3 = soft soil sites
custom P1,V1; P2,V2; P3,V3; … Pn,Vn		Data is part of input immediately following spectrum command for a "custom" response spectrum. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/ sec^2). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc). **Note:** Do not enter if a SOIL TYPE *f11* value is specified.
RF *f14*		The response reduction factor. If not specified, the program will look for the factor defined under `DEFINE 1893 2016 LOAD` (refer to TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393)). If none is provided there either, a factor of 1.0 is assumed. The response reduction factor represents ratio of maximum seismic force on a structure during specified ground motion if it were to remain elastic to the design seismic force. Actual seismic force is reduced by a factor `RF` to obtain design force.)

1893 indicates the analysis as per IS:1893(Part 1)-2016 procedures.

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

Note: CQC, SRSS, and CSM Grouping methods are recommended by IS:1893 (Part 1) –2016.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

`TORSION` indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Torsion (on page 2552) for additional information.

Note: If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

`ACCELERATION` or `DISPLACEMENT` indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1 / \omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the $f2$ value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`LINEAR` or `LOGARITHMIC` Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

Note: The `LINEAR` or `LOGARITHMIC` option can only be used for custom soil types (e.g., when response spectra data pairs are specified). Do not use these commands when the `SOIL TYPE` command is used.

`SIGN` This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

Caution: Do *not* enter DOMINANT parameter with this option.

SAVE This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Note: In order to perform a soft story check per IS 1893 2016, use the TR.28.2.1 Soft Story Checking (on page 2331) command.

Methodology

The design lateral shear force at each floor in each mode is computed by STAAD.Pro in accordance with the Indian IS: 1893 (Part 1)-2016 equations 7.8.4.5c and 7.8.4.5d.

$$Q_{ik} = A_k \cdot \phi_{ik} \cdot P_k \cdot W_k$$

and

$$V_{ik} = \sum_{i=i+1}^{n} Q_{ik}$$

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1)-2016.

STAAD.Pro utilizes the following procedure to generate the lateral seismic loads.

1. You provide the value for $Z/2 \cdot I/R$ as factors for input spectrum. You calculate the expression $Z/2 \cdot I/R$ and provide these values using the terms f1, f2, and f3 and applicable, where these terms an explained in the table below.
2. The program calculates time periods for first six modes or as specified.
3. The program calculates Sa/g for each mode utilizing time period and damping for each mode, based on IS 1893 2016 Clause 6.4.2 and Fig. 2(b).

   For the Y direction, Sa/g is always equal to 2.5 per Clause 6.4.6.
4. The program calculates the design horizontal acceleration spectrum value A_k for each mode.
5. The program then calculates mode participation factor for each mode.
6. The peak lateral seismic force at each floor in each mode is calculated.
7. All response quantities for each mode are calculated.
8. The peak response quantities are then combined as per the specified method (SRSS, CQC, ABS, CSM or TEN) to get the final results.

Note: For vertical motions (i.e., when forces are applied in the Y direction), the resulting forces are taken as 2/3 of the horizontal forces as per 6.4.6 of the IS 1893 Part 1 2016 code.

According to clause 7.7.3b, the scale factor for the earthquake in the vertical direction shall be take as the maximum of the scale factors in both the X and Z direction (see also Notes below). There are broadly three cases which the program considers to evaluate this maximum. These are load lists defined containing one of the following cases

i. response spectra in X, response spectra in Y, and response spectra in Z separately in load cases in any order

ii. response spectra in XY, response spectra in YZ, and response spectra in XYZ in individual load cases in any order
iii. a mix of the first two cases in any order

For a scale factor for any load case with a vertical component (that is, response spectra of Y, XY, YZ, or XYZ types), the program will use the maximum of the scale factor for all preceding load cases that are listed*after* a previous load case with a Y component. For example, for model with the following 11 load cases, the following scale factors will be used:

Load Case No.	Type	Scale Factor	Load Case Used for Scale Factor
1	X	2.18	1
2	XY	2.18	2
3	YZ	1	3
4	XZ	2.18	4
5	Y	2.18	max(3,4) = 4
6	X	2.18	6
7	Z	1	7
8	Y	2.18	max(6,7) = 6
9	XY	2.18	9
10	XZ	2.18	10
11	Y	2.18	max(9 ,10) = 9 = 10

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

Refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Notes

a. The design base shear, VB_RS, calculated from the response spectrum method, is compared with the base shear, VB_SS, calculated by the empirical formula for the fundamental time period based on Clause 7.2.1. If VB_RS is less than VB_RS, all of the response quantities are multiplied by VB_SS/VB_RS as per Clause 7.7.3(a) for each of the orthogonal directions and by $_{max}\left[\left(\frac{VB\_SS}{VB\_RS}\right)_X, \left(\frac{VB\_SS}{VB\_RS}\right)_Z\right]$ for response amplification when considering response spectrum load in the vertical direction based on Clause 7.7.3(b).

For this, the following input is necessary before defining any primary load case.

```
DEFINE IS1893 2016 LOAD
...
...
```

Refer to TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393) for full details on this command structure.

> **Note:** STAAD.Pro does not calculate the fundamental frequency of the structure needed for the empirical base shear VB_SS. calculation; so you must enter either the ST parameter or the PX and PZ parameters in the DEFINE IS1893 2016 LOAD data.

b. Although IS 1893 2016 recommends a damping ratio of 0.05, you may use any other damping and the values will be calculated based on Table 3. A warning will be printed in the output.

c. The story drift in any story shall not exceed 0.004 times the story height as per Clause 7.11.1. To check this, the following command should be given after the analysis command.

```
PRINT STORY DRIFT
```

A warning message will be printed if the story drift exceeds this limitation.

d. If any soft story (as per definition in Table 5 of IS:1893-2016) is detected, a warning message will be printed in the output.

Torsion

The torsion arising due to dynamic eccentricity (i.e., static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.8 of the IS 1893 2016 code. The dynamic eccentricity is automatically calculated by the program (in both cases of TOR and TOR OPP options), while the amount of accidental eccentricity can be specified through the ECC option (if not specified, default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered).

Non-symmetric or torsionally unbalanced buildings are prone to earthquake damage due to coupled lateral and torsional movements (i.e., the translational vibration of the building couples with its torsional vibrations within elastic range). The level of coupling between lateral and torsional vibrations of the building can be larger, thus leading to significantly higher lateral-torsional coupling than that predicted by elastic analysis.

- Cl. 7.7.5.4 of IS 1893 (Part 1) : 2016 is valid for buildings with *regular* or *nominally irregular* plan configurations. For buildings which are *irregular* in plan, it is better to consider torsion from dynamic eccentricity into analysis; even if torsionally coupled vibration is considered during response spectrum analysis.

- Cl. 7.8 states that, in the case that a 3D dynamic analysis is carried out, the dynamic amplification factor 1.5—as given by Cl. 7.8—can be replaced by 1.0. This implies that the code also recommends to use Cl. 7.8 for all types of buildings by including torsion from both dynamic and accidental eccentricity in the response spectrum analysis.

Torsion Methodology

As per IS1893-2016 code, provision shall be made in all buildings for increase in shear forces on the lateral force resisting elements resulting from the horizontal torsional moment arising due to eccentricity between the center of mass and the center of rigidity.

In response spectrum analysis all the response quantities (i.e. joint displacements, member forces, support reactions, plate stresses, etc) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated. Refer Cl. 7.7.5.4(c) of IS 1893 (Part 1) : 2016. (Q_{ik} at floor i for mode k)
2. At each floor design eccentricity is calculated. Refer Cl. 7.8.

   Thus, design eccentricity e_{di} = *f15*×e_{si} + *f12*×b_i. See Dynamic Eccentricity (on page 2554) for details.
3. Torsional moment is calculated at each floor. ($M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k)
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

Modal Combination

Steps 1 to 6 are performed for all modes considered and missing mass correction (if any). Finally the peak response quantities from different modal response are combined as per `CQC` or `SRSS` or `TEN PERCENT` or `CSM` method.

Notes

After the analysis is complete following files are generated.

a. Story shear for each mode for each load case is given in the file `<filename>_RESP1893.txt`.
b. Rotational stiffness of each floor is given in the file `<filename>_ROT1893.txt`.
c. Center of mass, center of rigidity, design eccentricity at each floor level and additional shear due to torsion at each floor level for each mode for each load case is given in the file `<filename>_TOR1893.txt`.

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = \mathrm{DEC} \times e_{si} + \mathrm{ECC} \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
ECC and DEC	=	Factors to determine the design eccentricity. These are input parameters.

IS 1893-2016 clause 7.8.2 defines two equations:

$$e_{di} = 1.5 \times e_{si} + 0.05 \times b_i$$

$$e_{di} = 1.0 \times e_{si} - 0.05 \times b_i$$

By including the optional command `TORSION`, the first equation is used by default. To account for the second equation, you may simply specify `ECC -0.05` (at which point the default for `DEC` is then 1.0). The defaults for `ECC` and `DEC` provided to meet the requirements but the parameters may be specified manually.

Example

Refer to V. IS 1893 2016 Response Spectrum (on page 3498) for a detailed explanation of this example.

```
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IS1893 2016 X 0.0432 DAMP 0.05
SOIL TYPE 1
```

Related Links
TR.31.2.16 IBC 2015 Seismic Load Definition

- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)

- *V. IS 1893 2016 Response Spectrum* (on page 3498)

TR.32.10.1.9 Response Spectrum Specification per IS: 1893 (Part 4)-2015

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per IS: 1893 (Part 4)-2015 for dynamic analysis of industrial and stack-like structures.

The seismic load generator can be used to generate lateral loads in the X, Y, and Z directions.

> **Note:** This facility has *not* been developed for cases where the Z axis is set to be the vertical direction using the `SET Z UP` command.

> **Note:** For vertical motions (i.e., when forces are applied in the Y direction), the resulting forces are taken as 2/3 of the horizontal forces as per 6.4.6 of the IS 1893 2016 Part 1 code.

General Format

The data in the following format can be contained all on a single line or broken into two or three lines, so long as the second and third lines start with the `ACC` and `DOMINANT` or `SIGN` commands.

```
SPECTRUM comp-method IS1893 2015-P4 LOAD ( TORSION (DECCENTRICITY f8) (ECCENTRICITY
f9) ) *{ X f1 | Y f2  | Z f3 }
```

```
{ ACCELERATION | DISPLACEMENT } (SCALE f4) {DAMP f5 | CDAMP | MDAMP } (MISSING f6)
(ZPA f7) (IGNORE f13)
({ DOMINANT f10 | SIGN }) (IMR f11) (STARTCASE f12) (RSMIN f14 )
```

The following command (`SOIL TYPE` parameter *or* response spectra data pairs) must be in a separate line.

```
{ SOIL TYPE f11 | *{ P1,V1; P2,V2; P3,V3;…PN,VN } }
```

> **Note:** The spectrum type options ACCELERATION or `DISPLACEMENT` should only be used with custom soil types for IS 1893 2015 (Part 4).

Where:

Table 272: Parameters used for IS: 1893 (Part 4) 2015 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f8*	When ECC > 0, DEC defaults to 1.5 When ECC < 0, DEC defaults to 1.0	(Optional input) It is a factor which when multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity) gives dynamic eccentricity. Since the applied load is acting at the center of mass, the effect of inherent torsion arising due to static eccentricity is included in the analysis. **Note:** The torsion arising due to dynamic eccentricity (i.e,. static eccentricity multiplied by dynamic amplification factor) between center of mass and center of rigidity is applied along with accidental torsion, as per the recommendations of Cl. 7.8.2 of the IS 1893 Part 1 specification. The dynamic eccentricity is automatically calculated by the program while you can specify the amount of accidental eccentricity (if not specified, the default of 5% of lateral dimension of the floor in the direction of the earthquake will be considered). For details See Torsion Methodology (on page 2553).
ECCENTRICITY *f9*	0.05	It is a factor which indicates the extent of accidental eccentricity. For all buildings this factor is to be provided as 0.05. However, for highly irregular buildings this factor may be increased to 0.10. This factor is to be externally provided to calculate design eccentricity. Since accidental eccentricity can be on either side, you must consider lateral force acting at a floor level to be accompanied by a clockwise or a counterclockwise accidental torsion moment. If the *f9* value is positive, it indicates clockwise torsion whereas a negative value indicates counterclockwise torsion.
X *f1* | Y *f2* | Z *f3*	0	Factors for the input spectrum to be applied in X, Y, & Z directions. These must be entered as the product of $[(Z/2)\times(I/R)]$ for standard specific spectra. Any one or all directions can be input. Directions not provided will default to zero. Based on Clause7.3 of IS 1893(Part 4): 2015.
SCALE *f4*	1	Linear scale factor by which design horizontal acceleration spectrum will be multiplied. This factor signifies that the structures and foundations, at which level base shear will be calculated, are placed below the ground level. **Note:** If site specific spectra curve is used then f4 value is to be multiplied by the scale factor by which spectra data will be multiplied. Usually to factor g's to length/sec^2 units.

Parameter	Default Value	Description
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the ZPA *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING *f6* is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.

Parameter	Default Value	Description
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.
SOIL TYPE *f11*	2	The soil type present. Depending upon time-period, types of soil and damping, average response acceleration coefficient, Sa/ g is calculated from reference to Fig.1 of this code where: 1 = Type 1 - for rocky or hard soil 2 = Type 2 - medium soil 3 = Type 3 - soft soil sites
custom P1,V1; P2,V2; P3,V3; ... Pn,Vn		Data is part of input immediately following spectrum command for a “custom” response spectrum. Period - Value pairs (pairs separated by semicolons) are entered to describe the spectrum curve. Period is in seconds and the corresponding Value is acceleration (current length unit/ sec^2). If data is in g acceleration units then the factor by which spectra data will be multiplied is g to the current length unit (9.81, 386.4, etc). **Note:** Do not enter if a SOIL TYPE *f11* value is specified.
IGNORE *f13*	0.009	(Optional input) It indicates the mass participation (in percent) of those modes to be excluded while considering torsion provision of IS-1893. Depending upon the model it may be found that there are many local modes and torsional modes whose mass participation is practically negligible. These modes can be excluded without much change in the final analysis result. If not provided all modes will be considered. If none provided the default value of 0.009% will be considered. If IGN is entered on any one spectrum case it will be used for all spectrum cases. **Note:** If the value of *f14* is considerable it may lead to considerable variation of analysis result from the actual one. Hence caution must be taken while using `IGNORE` command. If the `MODE SELECT` command is provided along with the `IGNORE` command, the number of modes excluded from the analysis will be those deselected by the `MODE SELECT` command and *also* those deselected by the `IGNORE` command.

Parameter	Default Value	Description
RSMIN *f14*	2.0	Used to enter a factor, given in percent, to multiply by the total seismic weight of the structure to obtain a minimum base shear value. This percentage is taken from Table 2 of IS:1839(Part 4)-2015. This percentage must match a value in this table. The default value is taken for a structure category 1 and zone 2. If the minimum base shear value exceed the calculated response spectrum base shear, then all analysis results of the response spectrum load case are scaled by the ratio of the minimum base shear to the calculated response spectrum base shear. This multiplying factor will be noted in the output.

`1893 2015-P4` indicates the analysis as per IS:1893(Part 4)-2015 procedures.

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

Note: CQC, SRSS, and CSM Grouping methods are recommended by IS:1893 (Part 4) –2015.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

`TORSION` indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See Torsion (on page 2552) for additional information.

Note: If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

The Torsion methodology as per Clause 10.4 of IS 1893 (Part4): 2015 remains same as Torsion (on page 2552) and is only applicable for ST=4.

ACCELERATION or DISPLACEMENT indicates whether **Acceleration** or **Displacement** spectra will be entered. The relationship between acceleration and displacement values in response spectra data is:

$$\text{Displacement} = \text{Acceleration} \times (1/\omega)^2$$

where

ω = 2π/Period (period given in seconds; ω in cycles per second)

DAMP, MDAMP, and CDAMP select source of damping input:

- `DAMP` indicates to use the $f2$ value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

For response spectra analysis, the damping ratio coefficients for steel and concrete that are recommended (other than the default value of 5%) for industrial structures as per Clause 9.4 (Table 5) are as follows:

Material	DBE	MCE
Steel	0.02	0.04
Reinforced concrete	0.05	0.07
Prestressed concrete	0.03	0.05

Note: For combined material structures, the damping ratio coefficient shall be determined based on well established procedures. If a composite damping ratio coefficient is not evaluated, it shall be taken as that corresponding to material having lower value of damping.

LINEAR or LOGARITHMIC Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

Note: The `LINEAR` or `LOGARITHMIC` option can only be used for custom soil types (e.g., when response spectra data pairs are specified). Do not use these commands when the `SOIL TYPE` command is used.

SIGN This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

Caution: Do *not* enter `DOMINANT` parameter with this option.

SAVE This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Individual Modal Response Case Generation

Individual modal response (`IMR`) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

Refer to TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Example

```
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IS1893 2015-P4 LOAD X 0.0432 DAMP 0.05
SOIL TYPE 1
```

Related Links

- *V. IS 1893 2015 Response Spectrum* (on page 3575)

TR.32.10.1.10 Response Spectrum Specification per IBC 2006

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2006 edition of the ICC specification *International Building Code* (IBC), for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method IBC 2006 *{ X f1| Y f2| Z f3} ACCELERATION
```

```
{DAMP f5| CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f6) (ZPA f7)
({ DOMINANT f10 | SIGN }) (SAVE)  (IMR f11) (STARTCASE f12)
```

Note: The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8| LAT f9 LONG f13| SS f14S1 f15} SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

Table 273: Parameters used for IBC 2006 response spectrum

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use “Missing Mass” method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the ZPA *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING *f6* is entered.

Parameter	Default Value	Description
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.
ZIP *f8*		The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
LAT *f9*		The latitude of the site used with the longitude to determine the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
LONG *f13*		The longitude of the site used with the latitude to determine the S_s and S_1 factors. (IBC 2006, ASCE 7-02 Chapter 22)
SS *f14*		Mapped MCE for 0.2s spectral response acceleration. (IBC 2006, ASCE 7-02 Chapter 22)
S1 *f15*		Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)
SITE CLASS *f16*		Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)
FA *f17*		Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
FV *f18*		Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
TL *f19*		Long-Period transition period in seconds. (IBC 2006, ASCE 7-02 Chapter 22)

`IBC 2006` indicates that the spectrum should be calculated as defined in the IBC 2006 specification.

comb-method = { `SRSS | ABS | CQC | ASCE | TEN | CSM | GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n/\omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`ACCELERATOIN` indicates that the Acceleration spectra will be entered.

Note: IBC / ASCE 7 does not have provisions for displacement response spectra.

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used

- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

LINEAR or LOGARITHMIC

Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When FILE *filename* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SIGN

This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for ABS option.

Caution: Do *not* enter DOMINANT parameter with this option.

SAVE

This option results in the creation of a acceleration data file (with the model file name and an .acc file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Methodology

The methodology for calculating the response spectra is defined in ASCE7-05, section 11.4. The following is a quick summary:

a. Input S_s and S_1 (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \text{ x } S_s$$

and

$$S_{m1} = F_v \text{ x } S_1$$

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

c. Calculate

$$S_{ds} = (2/3)\ S_{ms}$$

and

$$S_{d1} = (2/3)\ S_{m1}$$

The spectrum is generated as per section 11.4.5.

Individual Modal Response Case Generation

Individual modal response (IMR) cases are simply the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. If the IMR parameter is entered, then STAAD.Pro will create load cases for the first specified number of modes for this response spectrum case (i.e., if five is specified then five load cases are generated, one for each of the first five modes). Each case will be created in a form like any other primary load case.

The results from an IMR case can be viewed graphically or through the print facilities. Each mode can therefore be assessed as to its significance to the results in various portions of the structure. Perhaps one or two modes could be used to design one area/floor and others elsewhere.

You can use subsequent load cases with TR.32.11 Repeat Load Specification (on page 2595) combinations of these scaled modes and the static live and dead loads to form results that are all with internally consistent signs (unlike the usual response spectrum solutions). The modal applied loads vector will be omega squared times mass times the scaled mode shape. Reactions will be applied loads minus stiffness matrix times the scaled mode shape.

With the Repeat Load capability, you can combine the modal applied loads vector with the static loadings and solve statically with P-Delta or tension only.

Note: When the IMR option is entered for a Spectrum case, then a TR.37 Analysis Specification (on page 2620) & TR.38 Change Specification (on page 2660) must be entered after each such Spectrum case.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Example

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
…
LOAD 1 LOADTYPE Seismic  TITLE RS_X
SPECTRUM SRSS IBC 2006 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
```

Related Links

- *M. To add an IBC 2006 response spectrum* (on page 866)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.32.10.1.11 Response Spectrum Specification per IBC 2012

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2012 edition of the ICC specification *International Building Code* (IBC) and ASCE 7-10, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method IBC 2012 ( TORSION (DECCENTRICITY f20) (ECCENTRICITY f21) ) *{ X
f1 | Y f2 | Z f3 } ACCELERATION
```

```
{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7) ({ DOMINANT f10 | SIGN })
(SAVE) (IMR f11) (STARTCASE f12)
```

Note: The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15 } SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

Table 274: Parameters used for IBC 2012 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f20*	1.0	Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).
ECCENTRICITY *f21*	0.05	Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If CDAMP is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If MDAMP is specified, then modal damping is calculated using the method defined in a DEFINE DAMPING INFORMATION command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Parameter	Default Value	Description
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING` *f6* is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.
ZIP *f11*		The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (IBC 2012, ASCE 7-10 Chapter 22)
LAT *f12*		The latitude of the site used with the longitude to determine the S_s and S_1 factors. (IBC 2012, ASCE 7-10 Chapter 22)
LONG *f13*		The longitude of the site used with the latitude to determine the S_s and S_1 factors. (IBC 2012, ASCE 7-10 Chapter 22)

Parameter	Default Value	Description
SS *f14*		Mapped MCE for 0.2s spectral response acceleration. (IBC 2012, ASCE 7-10 Chapter 22)
S1 *f15*		Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)
SITE CLASS *f16*		Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)
FA *f17*		Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
FV *f18*		Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
TL *f19*		Long-Period transition period in seconds. (IBC 2012, ASCE 7-10 Chapter 22)

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`IBC 2012` indicates that the spectrum should be calculated as defined in the IBC 2012 specification.

TORSION indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See "Torsion" for additional information.

Note: If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

ACCELERATOIN indicates that the Acceleration spectra will be entered.

Note: IBC / ASCE 7 does not have provisions for displacement response spectra.

DAMP, MDAMP, and CDAMP select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

LINEAR or LOGARITHMIC Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When `FILE` *filename* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SIGN This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

Caution: Do *not* enter `DOMINANT` parameter with this option.

SAVE	This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Methodology

The methodology for calculating the response spectra is defined in ASCE 7-2010, section 11.4. The following is a quick summary:

a. Input S_s and S_1 (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \times S_s$$

and

$$S_{m1} = F_v \times S_1$$

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

c. Calculate

$$S_{ds} = (2/3)\ S_{ms}$$

and

$$S_{d1} = (2/3)\ S_{m1}$$

The spectrum is generated as per section 11.4.5.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Torsion

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity $e_{di} = f20 \times e_{si} + f12 \times b_i$ where f20 = 1.0 and f21 = (±) 0.05

   Where:

   e_{si} = static eccentricity between center of mass and center of rigidity at floor i.
   b_i = floor plan dimension orthogonal to the direction of earthquake loading.
3. Torsional moment is calculated at each floor.

   $M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.
7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Example

Refer to V. IBC 2012 Response Spectrum (on page 3472) for a detailed explanation of this example.

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
...
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IBC 2012 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
```

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)
- *V. IBC 2012 Response Spectrum* (on page 3472)

TR.32.10.1.12 Response Spectrum Specification per IBC 2015

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per the 2015 edition of the ICC specification *International Building Code* (IBC) and ASCE 7-10, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method IBC 2015 ( TORSION (DECCENTRICITY f20) (ECCENTRICITY f21) ) *{ X f1 | Y f2 | Z f3 } ACCELERATION
```

```
{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7) ({ DOMINANT f10 | SIGN }) (SAVE) (IMR f11) (STARTCASE f12)
```

Note: The data from `SPECTRUM` through `ACC` must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15 } SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

Table 275: Parameters used for IBC 2015 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f20*	1.0	Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).
ECCENTRICITY *f21*	0.05	Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Parameter	Default Value	Description
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the ZPA *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the MISSING parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with MISSING option only. Defaults to 33 Hz if not entered. The value is printed but not used if MISSING *f6* is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the SIGN parameter with this option. Ignored for the ABS method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (NM), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the IMR parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through NM, the load case number is the prior case number plus one.
ZIP *f11*		The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (IBC 2015, ASCE 7-10 Chapter 22)
LAT *f12*		The latitude of the site used with the longitude to determine the S_s and S_1 factors. (IBC 2015, ASCE 7-10 Chapter 22)
LONG *f13*		The longitude of the site used with the latitude to determine the S_s and S_1 factors. (IBC 2015, ASCE 7-10 Chapter 22)

Parameter	Default Value	Description
SS *f14*		Mapped MCE for 0.2s spectral response acceleration. (IBC 2015, ASCE 7-10 Chapter 22)
S1 *f15*		Mapped spectral acceleration for a 1-second period. (IBC 2000, equation 16-17. IBC 2003, ASCE 7-02 section 9.4.1.2.4-2. IBC 2006, ASCE 7-05 Section 11.4.1)
SITE CLASS *f16*		Enter A through F for the Site Class as defined in the IBC code. (IBC 2000, Section 1615.1.1 page 350. IBC 2003, Section 1615.1.1 page 322. IBC 2006 ASCE 7-05 Section 20.3)
FA *f17*		Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
FV *f18*		Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2006, ASCE 7-05 Section 11.4.3)
TL *f19*		Long-Period transition period in seconds. (IBC 2015, ASCE 7-10 Chapter 22)

comb-method = { `SRSS` | `ABS` | `CQC` | `ASCE` | `TEN` | `CSM` | `GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1 + r) r^{2/3}}{(1 - r^2)^2 + 4\zeta^2 r(1 + r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`IBC 2015` indicates that the spectrum should be calculated as defined in the IBC 2015 specification.

TORSION	indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See "Torsion" for additional information. **Note:** If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases. Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.
ACCELERATOIN	indicates that the Acceleration spectra will be entered. **Note:** IBC / ASCE 7 does not have provisions for displacement response spectra.
DAMP, MDAMP, and CDAMP	select source of damping input: • `DAMP` indicates to use the *`f2`* value for all modes • `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used • `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification
LINEAR or LOGARITHMIC	Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. When `FILE` *`filename`* is entered, the interpolation along the damping axis will be linear. **Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.
SIGN	This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option. **Caution:** Do *not* enter `DOMINANT` parameter with this option.

SAVE	This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Methodology

The methodology for calculating the response spectra is defined in ASCE 7-2010, section 11.4. The following is a quick summary:

a. Input S_s and S_1 (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \times S_s$$

and

$$S_{m1} = F_v \times S_1$$

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

c. Calculate

$$S_{ds} = (2/3)\ S_{ms}$$

and

$$S_{d1} = (2/3)\ S_{m1}$$

The spectrum is generated as per section 11.4.5.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity $e_{di} = f20 \times e_{si} + f12 \times b_i$ where f20 = 1.0 and f21 = (±) 0.05

   Where:

   e_{si} = static eccentricity between center of mass and center of rigidity at floor i.
   b_i = floor plan dimension orthogonal to the direction of earthquake loading.
3. Torsional moment is calculated at each floor.

   $M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k
4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.

7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Example

Refer to V. IBC 2015 Response Spectrum (on page 3462) for a detailed explanation of this example.

```
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
…
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IBC 2015 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
```

Related Links

- *V. IBC 2015 Response Spectrum* (on page 3462)

TR.32.10.1.13 Response Spectrum Specification per IBC 2018

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2018 edition of the ICC specification *International Building Code* (IBC) and ASCE 7-16, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
SPECTRUM comb-method IBC 2018 (TORSION) (DECCENTRICITY f20) (ECCENTRICITY f21) *{ X f1
| Y f2 | Z f3 } ACCELERATION
```

```
{DAMP f5 | CDAMP | MDAMP } ( {LIN | LOG} ) (MIS f6) (ZPA f7) ({ DOMINANT f10 | SIGN })
(SAVE) (IMR f11) (STARTCASE f12)
```

Note: The data from SPECTRUM through ACC must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of four lines per spectrum).

The command is completed with the following data which must be started on a new line:

```
{ZIP f8 | LAT f9 LONG f13 | SS f14 S1 f15 } SITE CLASS (f16) (FA f17 FV f18) TL f19
```

Where:

Table 276: Parameters used for IBC 2015 response spectrum

Parameter	Default Value	Description
DECCENTRICITY *f20*	1.0	Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity).
ECCENTRICITY *f21*	0.05	Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion.

Parameter	Default Value	Description
X *f1*, Y *f2*, Z *f3*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.
DAMP *f5*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f6*		Optional parameter to use "Missing Mass" method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the *f6*value entered in length/sec^2 (this value is not multiplied by SCALE). If *f6*is zero, then the spectral acceleration at the `ZPA` *f7*frequency is used. If *f7*is zero or not entered, the spectral acceleration at 33Hz (Zero Period Acceleration, ZPA) is used. The results of this calculation are SRSSed with the modal combination results. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f7*	33 [Hz]	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING f6` is entered.
DOMINANT *f10*	1 (1st Mode)	The dominant mode method. All results will have the same sign as mode number *f10* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option. Ignored for the `ABS` method of combining spectral responses from each mode.
IMR *f11*	1	The number of individual modal responses (scaled modes) to be copied into load cases. Defaults to one. If greater than the actual number of modes extracted (`NM`), then it will be reset to NM. Modes one through f11 will be used. Missing Mass modes are not output.

Parameter	Default Value	Description
STARTCASE *f12*	Highest Load Case No. + 1	The primary load case number of mode 1 in the `IMR` parameter. Defaults to the highest load case number used so far plus one. If *f12* is not higher than all prior load case numbers, then the default will be used. For modes 2 through `NM`, the load case number is the prior case number plus one.
ZIP *f11*		The zip code of the site location to determine the latitude and longitude and consequently the S_s and S_1 factors. (IBC 2018, ASCE 7-16 Chapter 22)
LAT *f12*		The latitude of the site used with the longitude to determine the S_s and S_1 factors. (IBC 2018, ASCE 7-16 Chapter 22)
LONG *f13*		The longitude of the site used with the latitude to determine the S_s and S_1 factors. (IBC 2018, ASCE 7-16 Chapter 22)
SS *f14*		Mapped MCE for 0.2s spectral response acceleration. (IBC 2018, ASCE 7-16 Section 11.4.1) This is obtained using the USGS web service.
S1 *f15*		Mapped spectral acceleration for a 1-second period. (IBC 2018, ASCE 7-2016 Section 11.4.1) This is obtained using the USGS web service.
CLASS *f16*		Enter A through F for the Site Class as defined in the IBC code. (IBC 2018 ASCE 7-2016 Section 20.3)
FA *f17*		Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2018, ASCE 7-2016 Section 11.4.3)
FV *f18*		Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e., 6). (IBC 2018, ASCE 7-2016 Section 11.4.3)
TL *f19*		Long-Period transition period in seconds. (IBC 2018, ASCE 7-16 Chapter 22)

comb-method = { `SRSS` \| `ABS` \| `CQC` \| `ASCE` \| `TEN` \| `CSM` \| `GRP` } are methods of combining the responses from each mode into a total response.

The CQC and ASCE4-98 methods require damping. ABS, SRSS, CRM, GRP, and TEN methods do not use damping unless spectra-period curves are made a function of damping (see File option below). CQC, ASCE, CRM, GRP, and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE and CQC are more sophisticated and realistic methods and are recommended.

SRSS Square Root of Summation of Squares method.

ABS Absolute sum. This method is very conservative and represents a worst case combination.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2 (1 + r) r^{2/3}}{(1 - r^2)^2 + 4\zeta^2 r (1 + r)^2}$$

$$r = \omega_n / \omega_m \le 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

ASCE NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes.

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (Rev. 1.2.2, 1976).

CSM Closely Spaced Method as per IS:1893 (Part 1)-2002 procedures.

GRP Closely Spaced Modes Grouping Method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Note: If SRSS is selected, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

`IBC 2018` indicates that the spectrum should be calculated as defined in the IBC 2018 specification.

`TORSION` indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the center of mass and center of rigidity needs to be considered. See "Torsion" for additional information.

Note: If `TORSION` is entered on any one spectrum case it will be used for all spectrum cases.

Lateral shears at story levels are calculated in global X and Z directions. For global Y direction the effect of torsion will not be considered.

`ACCELERATOIN` indicates that the Acceleration spectra will be entered.

Note: IBC / ASCE 7 does not have provisions for displacement response spectra.

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`LINEAR` or `LOGARITHMIC`	Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. When `FILE` *`filename`* is entered, the interpolation along the damping axis will be linear. **Note:** The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.
`SIGN`	This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option. **Caution:** Do *not* enter `DOMINANT` parameter with this option.
`SAVE`	This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Methodology

The methodology for calculating the response spectra is defined in ASCE 7-2016, section 11.4. The following is a quick summary:

a. Input S_s and S_1 (this could have been searched from database or entered explicitly)

b. Calculate

$$S_{ms} = F_a \times S_s$$

and

$$S_{m1} = F_v \times S_1$$

Where:

F_a and F_v are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied. These are required to be provided by the user. You may also specify values for F_a and F_v in lieu of table values.

c. Calculate

$$S_{ds} = (2/3)\ S_{ms}$$

and

$$S_{d1} = (2/3)\ S_{m1}$$

The spectrum is generated as per section 11.4.5.

Notes:

- Exceptions 1 in Clause 11.4.8 of ASCE 7-16 is implemented. In exception 1, for Site class E and S_s > 1.0, Fa value is taking the same for site class C. In this case, there is slight reduction in the seismic force generated.
- The exception of Clause 2 in 11.4.8 of ASCE 7-16 is *not* implemented in STAAD.Pro for response spectra.

- The vertical ground motion for seismic design as described in clause 11.9 is *not* implemented as the program cannot determine the vertical time period, T_v.
- ASCE 7-2016 Clause 12.9.1.4 stipulates that if the combined response for modal base shear, V, is less than that of the calculated base shear from the equivalent lateral force procedure, V_t, then the response spectra values should be scaled up by V/V_t. STAAD.Pro does not automatically check this nor increase the response quantities in this situation. You can check this by simply analyzing TR.31.2.17 IBC 2018 Seismic Load Definition (on page 2416) and comparing the base shear values. The X, Y, or Z factors for the response spectrum can be then increased accordingly if required.

Inherent and Accidental Torsion

In response spectrum analysis all the response quantities (i.e., joint displacements, member forces, support reactions, plate stresses, etc.) are calculated for each mode of vibration considered in the analysis. These response quantities from each mode are combined using a modal combination method (either by CQC, SRSS, ABS, TEN PERCENT, etc.) to produce a single positive result for the given direction of acceleration. This computed result represents a maximum magnitude of the response quantity that is likely to occur during seismic loading. The actual response is expected to vary from a range of negative to positive value of this maximum computed quantity.

No information is available from response spectrum analysis as to when this maximum value occurs during the seismic loading and what will be the value of other response quantities at that time. As for example, consider two joints J2 and J3 whose maximum joint displacement in global X direction come out to be X1 and X2 respectively. This implies that during seismic loading joint J1 will have X direction displacement that is expected to vary from -X1 to +X1 and that for joint J2 from -X2 to +X2. However, this does not necessarily mean that the point of time at which the X displacement of joint J1 is X1, the X displacement of joint J2 will also be X2.

For the reason stated above, torsional moment at each floor arising due to dynamic eccentricity along with accidental eccentricity (if any) is calculated for each mode. Lateral story shear from this torsion is calculated forming global load vectors for each mode. Static analysis is carried out with this global load vector to produce global joint displacement vectors for each mode due to torsion. These joint displacements from torsion for each mode are algebraically added to the global joint displacement vectors from response spectrum analysis for each mode. The final joint displacements from response spectrum along with torsion for all modes are combined using specified modal combination method to get final maximum possible joint displacements. Refer to the steps explained below.

See TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) for additional details on IMR load case generation.

Steps

For each mode following steps are performed to include Torsion provision.

1. Lateral story force at each floor is calculated.
2. At each floor design eccentricity is calculated.

   Thus, design eccentricity $e_{di} = f20 \times e_{si} + f12 \times b_i$ where f20 = 1.0 and f21 = (±) 0.05

   Where:

   e_{si} = static eccentricity between center of mass and center of rigidity at floor i.
   b_i = floor plan dimension orthogonal to the direction of earthquake loading.
3. Torsional moment is calculated at each floor.

   $M_{ik} = Q_{ik} \times e_{di}$ at floor i for mode k

4. The lateral nodal forces corresponding to torsional moment are calculated at each floor. These forces represent the additional story shear due to torsion.
5. Static analysis of the structure is performed with these nodal forces.
6. The analysis results (i.e., member force, joint displacement, support reaction, etc) from torsion are algebraically added to the corresponding modal response quantities from response spectrum analysis.
7. Steps 1 through 6 are performed for all modes considered and missing mass correction (if any). Finally, the peak response quantities from the different modal responses are combined as per the specified combination method (e.g., SRSS, CQC, TEN, etc.)

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = DEC \times e_{si} + ECC \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
ECC and DEC	=	Factors to determine the design eccentricity. These are input parameters.

Refer to Cl. 12.9.2.2.2 for the requirements of accidental torsion per the ASCE 7-16 code.

Example

```
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
...
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IBC 2018 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
```

TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per SNiP II-7-81 for dynamic analysis.

General Format

```
SPECTRUM comb-method SNIP A f1 *{ {X f2 | KWX f3 KX1 f4} | { Y f5 | KWY f6 KY1 f7 } |
{ Z f8 | KWZ f9 KZ1 f10 } ACCELERATION (SCALE f11)
```

```
{DAMP f12 | CDAMP | MDAMP } ( { LINEAR | LOGARITHMIC } ) (MISSING f13) (ZPA f14)
({ DOMINANT f15 | SIGN }) SOIL { 1 | 2 | 3} (SAVE)
```

The data from `SPECTRUM` through `SCALE` above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:

Table 277: Parameters for SNiP II-7-81 response spectrum

Parameter	Default Value	Description
A *f1*		Zoning factor, A, which is based on maximum acceleration factor for the seismic zone. This factor must be modified for `SOIL` types other than 2. The exact zone factor value used for a specific location requires engineering judgment. The following table serves as a guide for accelerations and corresponding zone factors which would be used.
X *f2* Y *f5* Z *f8*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero. Alternatively, you may input individual parameters such as `KWX`, `KX1` the product of which would be used as the factor along that direction.
SCALE *f11*		Linear scale factor by which the spectra data will be multiplied. Usually used to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system (thus, 9.81 m/s^2 or 32.2 ft/s^2).
DAMP *f12*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)

Parameter	Default Value	Description
MISSING *f13*		Optional parameter to use the "Missing Mass" method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec^2 for this missing mass mode is the *f13* value entered in length per second squared units (this value is not multiplied by SCALE). If *f13* is zero, then the spectral acceleration at the `ZPA` *f14* frequency is used. If *f14* is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. For SRSS and CQC, the results of this calculation are SRSSed with the modal combination results. If either *f13* or *f14* are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f14*		The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING` *f13* is entered.
DOMINANT *f15*		The dominant mode method. All results will have the same sign as mode number *f15* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option.

comb-method = `{ SRSS | CQC }` are methods of combining the responses from each mode into a total response.

The CQC method requires damping. The SRSS method does not use damping unless spectra-period curves are made a function of damping (see File option below). CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic methods and is recommended.

SRSS Square root of summation of squares method as prescribed by the SNiP II-7-81 code.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2 (1+r) r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r (1+r)^2}$$

$$r = \omega_n / \omega_m \leq 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

The specifier `SNIP` is mandatory to denote that the applied loading is as per the guidelines of SNiP II-7-81.

ACCELERATOIN
indicates that the Acceleration spectra will be entered.

Note: SNiP II-7-81 / SP 14.13330.2011 do not have provisions for displacement response spectra.

DAMP, MDAMP, and CDAMP
select source of damping input:

- `DAMP` indicates to use the *f2* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

LINEAR or LOGARITHMIC
Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When `FILE` *filename* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

SIGN
This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

Caution: Do *not* enter `DOMINANT` parameter with this option.

SOIL
Defines the subsoil conditions on which the response spectrum will be generated.

Note: The Zoning Factor, `A (f1)`, must be adjusted for soil types other than type 2.

1. Non-weathered rock and rocklike geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

SAVE
This option results in the creation of a acceleration data file (with the model file name and an `.acc` file extension) containing the joint accelerations in g's and radians/sec^2. These files are plain text and may be opened and viewed with any text editor (e.g., Notepad).

Description

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154). An illustration of mass modeling is available, with explanatory comments, in the sample file
C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Rus\Seismic_Russ.STD.

Example 1

The definition of a SNiP response spectrum in the X direction on a structure built on weather rock and where the Zoning Factor is 0.7071. As this is the first load case with a response spectrum, then the masses are modeled as loads.

```
LOAD 2 LOADTYPE Seismic TITLE SPECTRUM IN X-DIRECTION
*Masses
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
*SNiP Spectrum definition
SPECTRUM SRSS SNIP A 0.7071 X 1.0 ACC DAMP 0.05 SCALE 1.0 LIN MIS 0 ZPA 40
SOIL 2
...
```

Note: The maximum response spectrum load cases allowed in one run is 50.

For full details on Response Spectrum refer to section TR.32.10.1 Response Spectrum Analysis (on page 2498).

Example 2

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
START JOB INFORMATION
JOB NAME Plane Russian example
ENGINEER DATE 12-Feb-08
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.2 0 0; 3 0 4.5 0; 4 7.2 4.5 0; 5 0 9.0 0; 6 7.2 9.0 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY RUSSIAN
1 TO 4 PRIS AY 10000 YD 0.6 ZD 0.6
5 TO 6 PRIS YD 0.9 ZD 0.3
SUPPORTS
```

```
1 TO 2 FIXED BUT MZ
3 TO 6 FIXED BUT FX
CONSTANTS
E 30.0e+006 ALL
POISSON 0.2 ALL
CUT OFF MODE SHAPE 2
*NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
*WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 1 SEISMIC LOADING
JOINT LOAD
3 4 FX 310.586
3 4 FY 310.586
5 6 FX 310.586
5 6 FY 310.586
SPECTRUM CQC SNIP A 1.0 X 1.0 ACC MIS SOIL 2
PERFORM ANALYSIS
PRINT MODE SHAPES
PRINT STORY DRIFT
PRINT ANALYSIS RESULTS
FINISH
```

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.32.10.1.15 Response Spectrum Specification per SP 14.13330.2011

This command may be used to specify and apply the `RESPONSE SPECTRUM` loading as per SP 14.13330.2011 for dynamic analysis.

General Format

```
SPECTRUM { SRSS | CQC } SP11 (ECCENTRICITY) A f1 *{ {X f2 | Y f3 | Z f4} ACCELERATION
(SCALE f5)
(DAMP f6 ) (LOG) (MIS f7) (ZPA f8) ({ DOMINANT f9 | SIGN }) SOIL { 1 | 2 | 3}
```

The data from `SPECTRUM` through `SCALE` above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of four lines per spectrum).

Where:

Table 278: Parameters for SP 14.13330.2011 response spectrum

Parameter	Default Value	Description
A *f1*		Zoning factor, A, which is based on maximum acceleration factor for the seismic zone. This factor must be modified for SOIL types other than 2.
X *f2* Y *f3* Z *f5*	0.0	Factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

Parameter	Default Value	Description
SCALE *f5*		A linear scaling factor by which the spectrum data will be multiplied according to the provisions of SP 14.13330.2011.
DAMP *f6*	0.05	The damping ratio. Specify a value of exactly 0.0000011 to ignore damping. If `CDAMP` is specified, then composite damping is used as determined by the values for material damping (and spring damping, if specified). Refer to TR.26.2 Specifying Constants for Members and Elements (on page 2305) If `MDAMP` is specified, then modal damping is calculated using the method defined in a `DEFINE DAMPING INFORMATION` command, which must be included in the input file. Refer to TR.26.4 Modal Damping Information (on page 2312)
MISSING *f7*		Optional parameter to use the "Missing Mass" method to include the static effect of the masses not represented in the modes. The spectral acceleration length/sec^2 for this missing mass mode is the *f7* value entered in length per second squared units (this value is not multiplied by `SCALE`). If *f7* is zero, then the spectral acceleration at the `ZPA` *f8* frequency is used. If *f8* is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. For SRSS and CQC, the results of this calculation are SRSSed with the modal combination results. If either of *f7* or *f8* are not entered, the defaults will be used. Missing mass does not include the effect of masses lumped at the supports unless the support is a stiff spring or an Enforced support. **Note:** If the `MISSING` parameter is entered on any spectrum case it will be used for all spectrum cases.
ZPA *f8*	33.0 Hz	The zero period acceleration value for use with `MISSING` option only. Defaults to 33 Hz if not entered. The value is printed but not used if `MISSING` *f7* is entered.
DOMINANT *f9*	1	The dominant mode method. All results will have the same sign as mode number *f9* alone would have if it were excited then the scaled results were used as a static displacements result. Defaults to mode 1 if no value entered. If a 0 value entered, then the mode with the greatest % participation in the excitation direction will be used (only one direction factor may be nonzero). **Note:** Do not enter the `SIGN` parameter with this option.

comb-method = `{ SRSS | CQC }` are methods of combining the responses from each mode into a total response.

The CQC method requires damping. The SRSS method does not use damping unless spectra-period curves are made a function of damping (see File option below). CQC includes the effect of response magnification due to closely spaced modal frequencies. CQC is a more sophisticated and realistic methods and is recommended.

SRSS Square root of summation of squares method as prescribed by the SNiP II-7-81 code.

CQC Complete Quadratic Combination method (Default). This method is recommended for closely spaced modes instead of SRSS.

Resultants are calculated as:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m}$$

where

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{2/3}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

$$r = \omega_n / \omega_m \le 1.0$$

Note: The cross-modal coefficient array is symmetric and all terms are positive.

The specifier `SP11` is mandatory to denote that the applied loading is as per the guidelines of SP 14.13330.2011.

`ECCENTRICITY` automatic introduction of artificial mass eccentricity according to SP 14.13330.2011.

`ACCELERATOIN` indicates that the Acceleration spectra will be entered.

Note: SNiP II-7-81 / SP 14.13330.2011 do not have provisions for displacement response spectra.

`DAMP`, `MDAMP`, and `CDAMP` select source of damping input:

- `DAMP` indicates to use the *`f2`* value for all modes
- `MDAMP` indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- `CDAMP` indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`LINEAR` or `LOGARITHMIC` Select **Linear** or **Logarithmic** interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. Linear is the default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

When `FILE` *`filename`* is entered, the interpolation along the damping axis will be linear.

Note: The last interpolation parameter entered on the last of all of the spectrum cases will be used for all spectrum cases.

`SIGN` This option results in the creation of signed values for all results. The sum of squares of positive values from the modes are compared to sum of squares of negative values from the modes. If the negative values are larger, the result is given a negative sign. This command is ignored for `ABS` option.

> **Caution:** Do *not* enter `DOMINANT` parameter with this option.

SOIL Defines the subsoil conditions on which the response spectrum will be generated.

> **Note:** The Zoning Factor, `A (f1)`, must be adjusted for soil types other than type 2.

1. Non-weathered rock and rocklike geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

Description

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For more information on dynamic mass modeling, refer to sections TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154)

Example

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
START JOB INFORMATION
JOB NAME Plane Russian example
ENGINEER DATE 12-Feb-08
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.2 0 0; 3 0 4.5 0; 4 7.2 4.5 0; 5 0 9.0 0; 6 7.2 9.0 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY RUSSIAN
1 TO 4 PRIS AY 10000 YD 0.6 ZD 0.6
5 TO 6 PRIS YD 0.9 ZD 0.3
SUPPORTS
1 TO 2 FIXED BUT MZ
3 TO 6 FIXED BUT FX
CONSTANTS
E 30.0e+006 ALL
POISSON 0.2 ALL
CUT OFF MODE SHAPE 1
*NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
*WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 1 SEISMIC LOADING
JOINT LOAD
3 4 FX 310.586
3 4 FY 310.586
5 6 FX 310.586
5 6 FY 310.586
SPECTRUM CQC SP11 ECC A 1.0 X 1.0 ACC MIS SOIL 2
PERFORM ANALYSIS
PRINT MODE SHAPES
PRINT STORY DRIFT
PRINT ANALYSIS RESULTS
FINISH
```

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.32.10.2 Time Varying Load for Response History Analysis

Used to apply loads which are defined with a changing magnitude of acceleration, force, or moment (refer to TR. 31.4 Definition of Time History Load (on page 2441)). These can either be assigned to specific nodes (as a FORCE or MOMENT type) and/or globally to the all the supports on the model as a ground motion (as an ACCELERATION type). In addition to the data produced from a general modal analysis, a load case which includes time history loading will produce a set of graphs in the post processing mode to indicate how the displacement, velocity or acceleration of a selected node changes in each of the three global directions over the time period of the applied dynamic loading.

General Format

The following set of commands may be used to model time history loading on the structure for response time history analysis. Nodal time histories and ground motion time histories may both be provided within one load case.

```
TIME LOAD
```

```
joint-list *{ FX | FY  | FZ | MX | MY | MZ } It Ia f2
```

```
GROUND MOTION { ABSOLUTE | (RELATIVE ) } { X | Y | Z } It Ia f2
```

Where:

Parameter	Description
ABSOLUTE	nodal results are absolute (elastic response + motion of ground). If entered on any ground command, all results will be absolute.
RELATIVE	nodal results are relative (elastic response). (Default if neither ABS or REL are specified)
It	sequential position in the input data of type number of time varying load. To refer to first type number entered, use a 1 here regardless of actual type number entered. Ground Motion must have an Acceleration Type; Time Load forces must have a Force type; and Time Load moments must have a Moment Type (refer to TR. 31.4 Definition of Time History Load (on page 2441)).
Ia	Arrival time number (integer). This is the sequential number of the arrival time in the list explained in TR.31.4 Definition of Time History Load (on page 2441). Thus the arrival time number of a_3 is 3 and of a_n is n.
f2	The Force, Moment, or Acceleration Amplitude at this joint and direction will be multiplied by this factor (default = 1.0). For accelerations, if the amplitude-time curve was in g’s, please use the Scale Factor in the Define Time History command to convert g’s to the acceleration units used in that command. This is recommended due to possible unit changes between that command and this command.

Note: Multiple loads at a joint-direction pair for a particular (I_t I_a) pair will be summed. However there can be no more than four (I_t I_a) pairs associated with a particular joint-direction pair, the first four such entries will be used. Loads at dependent joint directions will be ignored.

Either `TIME LOAD`, `GROUND MOTION`, or both may be specified under one load case. More than one load case for time history analysis is *not* permitted.

For `TIME LOAD` data, multiple direction specifiers can be in one entry as follows (the direction specifiers must be on one line and missing values are assumed to be 1):

```
TIME LOAD
2  3  FX  1  FZ  1  4  -2.1  MX  2  2
6  7  FX  FY  FZ
```

Notes

a. A Time History analysis requires the mode shapes. These are calculated using the mass matrix determined from the loading specified in the first dynamic load case. (refer to TR.34.2 Modal Calculation Command (on page 2615))

b. The Node Displacement table reports the maximum displacement that occurs at each node over the entire time range.

c. The displacement of the model at a specific time instance can be displayed by using the time slider bar on the Results toolbar when displaying a load case with time history loading.

d. If any Node Groups are defined, the time history graphs can be set to display the average results for the group. The name of the selected node or group being displayed is given in the graph title bar.

e. A model can include only one load case with time history loads.

Note: STAAD.Pro is also capable of generating floor spectrum responses for a time history analysis. Refer to TR. 37.10 Floor Spectrum Command (on page 2657) for details on adding this to the analysis commands.

Example

```
LOAD 1
SELFWEIGHT X  1.0
SELFWEIGHT Y  1.0
SELFWEIGHT Z  1.0
MEMBER LOADS
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GZ 7.5 10.0
TIME LOAD
2  3  FX  1  3
5  7  FX  1  6
GROUND MOTION REL X  2 1
```

In this example, the permanent masses in the structure are provided in the form of selfweight and member loads (refer to TR.32 Loading Specifications (on page 2461) and G.17.3 Dynamic Analysis (on page 2154)) for obtaining the mode shapes and frequencies. The mass model can also be created using a Reference Load case with the Load type of Mass (refer to TR.31.6 Defining Reference Load Types (on page 2453)). The rest of the data is the input for application of the time varying loads on the structure. Forcing function type 1 is applied at joints 2 and 3 starting at arrival time number 3. (Arrival time number 3 is 1.8 seconds in example shown in TR.31.4 Definition of Time History Load (on page 2441)). Similarly, forcing function type 1 is applied at joints 5 and 7 starting at arrival time number 6 (4.4 seconds). A ground motion (type 2) acts on the structure in the x-direction starting at arrival time number 1 (0.0 seconds).

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *M. To add a time history load* (on page 880)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *G.17.3.5 Response Time History* (on page 2165)

TR.32.11 Repeat Load Specification

This command is used to create a primary load case using combinations of previously defined primary load cases.

General Format

```
REPEAT LOAD
```

```
i1, f1, i2, f2 … in, fn
```

Where:

i1, i2 … in	primary load case numbers
f1, f2 … fn	corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. Limit of 550 prior cases may be factored. Prior cases to be factored may also contain the `REPEAT LOAD` command.

Description

This command may be used to create a primary load case using combinations of previously defined primary load case(s). The `REPEAT LOAD` differs from the load `COMBINATION` command (See TR.35 Load Combination Specification (on page 2616)) in two ways:

1. A `REPEAT LOAD` is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. (`LOAD COMBINATIONS`, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary loadings evaluated independently).
2. In addition to previously defined primary loads, you can also add new loading conditions within a load case in which the `REPEAT LOAD` is used.
3. The `REPEAT LOAD` option is available to factor prior load case data and add those forces into the current load case.

   The load case data types that will be factored include Joint Loads, Member loads, Element loads, Inertia Loads, Fixed End loads, Selfweight loads, Displacements, Area loads, Prestress loading, and Temperature loads. Floor loads, Wind loads Snow loads, and UBC loads are first converted to equivalent member loads and then factored.

   Modal dynamic analysis load cases (Response Spectrum, Time History, Steady State) should not be used in `REPEAT LOAD`. It is also not available for loads generated using some of the program's load generation facilities such as `MOVING LOAD` Generation. However load cases with `WIND LOAD` may be used in Repeat Load.

UBC cases may only be used in `REPEAT LOAD` if there is a `PERFORM ANALYSIS` and `CHANGE` command after each UBC case. See notes with UBC LOAD command.

Prestress on a given member from 2 or more load cases cannot be combined.

Example:

```
LOAD 1 DL + LL
SELFWEIGHT Y -1.4
MEMBER LOAD
1 TO 7 UNIFORM Y -3.5
LOAD 2 DL + LL + WL
REPEAT LOAD
1 1.10
```

4. For a load case that is defined using the `REPEAT LOAD` attribute, the constituent load cases themselves can also be `REPEAT LOAD` cases. See load case 4 below.

```
LOAD 1
SELFWEIGHT Y -1.0
LOAD 2
MEMBER LOAD
2 UNI GY -1.5
LOAD 3
REPEAT LOAD
1 1.5
LOAD 4
REPEAT LOAD
2 1.2 3 1.25
```

Related Links

- *M. To add a repeat load case* (on page 892)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

TR.32.12 Generation of Loads

This command is used to generate Moving Loads, UBC Seismic loads and Wind Loads using previously specified load definitions.

Primary load cases may be generated using previously defined load systems. The following sections describe generation of moving loads, UBC seismic loads and Wind Loads.

TR.32.12.1 Generation of Moving Loads

This command is used to generate Moving Loads using previously specified load definitions.

Predefined moving load system types may be used to generate the desired number of primary load cases, each representing a particular position of the moving load system on the structure. This procedure will simulate the movement of a vehicle in a specified direction on a specified plane on the structure.

General Format

```
LOAD GENERATION n (ADD LOAD i )
```

```
TYPE j x1 y1 z1 *{ XINC f1 | YINC f2 | ZINC f3 } ( { YRANGE | ZRANGE } r)
```

Where:

n total number of primary load cases to be generated

i load case number for the previously defined load case to be added to the generated loads

j type number of previously defined load system

x1, y1, z1 x, y and z coordinates (global) of the initial position of the reference load

f1, f2, f3 x, y, or z (global) increments of position of load system to be used for generation of subsequent load cases.

Use only XINC & ZINC if Y up; Use only XINC & YINC if Z up.

r (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is added and subtracted to the reference vertical coordinate (y_1 or z_1)in the global vertical direction to form a range. The moving load will be externally distributed among all members within the vertical range thus generated. r always should be a positive number. In other words, the program always looks for members lying in the range *y1* and *y1*+ABS(*r*) or *z1* and *z1*+ABS(*r*). The default *r* value is very small, so entering *r* is recommended.

The `ADD LOAD` specification may be used to add a previously defined load case to all the load cases generated by the `LOAD GENERATION` command. In the example below, the `SELFWEIGHT` specified in load case 1 is added to all the generated load cases.

Sequential load case numbers will be assigned to the series of generated primary load cases. Numbering will begin at one plus the immediate previous load case number. Allow for these when specifying load cases after load case generation.

Notes

a. Primary load cases can be generated from Moving Load systems for frame members only. This feature does not work on finite elements.

b. This facility works best when the roadway, as well as the movement of the vehicle are along one of the global horizontal (X or Z) or (X or Y) directions.

For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the M. Bridge Deck workflow (on page 908) is recommended is recommended. The Bridge Deck workflow works on the influence line/influence surface method, and is considerably superior to the moving load generator. It also has the advantage of being able to calculate the critical load positions on decks modeled using plate elements.

c. The x_1, y_1, z_1 values of the starting position of the reference wheel must be provided bearing in mind that the reference wheel has to be at the elevation of the deck. An improper set of values of these parameters may result in the wheels being positioned incorrectly, and consequently, no load may be generated at all.

Example

```
LOAD 1 DL Only
SELFWEIGHT
LOAD GENERATION 20 ADD LOAD 1
TYPE 1 0. 5. 10. Xi 10.
TYPE 2 0. 10. 10. Zi 15.
LOAD 22 Live Load on Pavement
MEMBER LOAD
10 TO 20 30 TO 40 UNI GY -5.0
```

```
LOAD COMBINATION 31 10 0.75 22 0.75
PERFORM ANALYSIS
```

Related Links

- *M. Moving Loads* (on page 873)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 6398)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 6684)
- *M. To add vehicles to the load generation* (on page 874)
- *G.16.1 Moving Load Generator* (on page 2134)
- *M. To generate moving load cases* (on page 874)
- *G.16.1 Moving Load Generator* (on page 2134)

TR.32.12.2 Generation of Seismic Loads

This command is used to generate seismic loads using previously specified load definitions.

Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications. The following general format should be used to generate loads in a particular direction.

General Format

```
LOAD i
```

```
code LOAD { X | Y | Z } (f1) (DECCENTRICITY f2) (ACCIDENTAL f3)
```

Where:

```
code = { 1893 | AIJ | COL | CFE | GB | IBC | NRC | NTC | RPA | TUR | UBC}
```

Note: The specified code should correspond to the seismic definition defined earlier in the input file. Refer to TR. 31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2349) for details.

Parameter	Default Value	Description
LOAD *i*	-	Load case number.
X | Y | Z *f1*	1.0	Optional factor to be used to multiply the seismic Load. May be negative. **Note:** Only apply in the horizontal directions (i.e., X and Z for Y-up or X and Y for Z-up models).
DEC *f2*	0.0	Multiplying factor for natural torsion –arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm– to be used to multiply the seismic horizontal torsion load. Must be a positive value (greater than 1.0) or exactly 0.0.
ACC *f3*	1.0	Multiplying factor for accidental torsion, to be used to multiply the seismic accidental torsion load. May be negative (otherwise, the default sign for MY is used based on the direction of the generated lateral forces).

Use only horizontal directions. This means that seismic loads should only be applied in the X and Z directions with Y up (or X and Y directions with Z up).

To include horizontal torsional moment arising due to static eccentricity for a rigid floor diaphragm, the following conditions must be satisfied:

a. The floor must be modeled as a rigid diaphragm.

b. A positive value (greater than 1.0) for `DEC` must be provided. Seismic load is applied at center of mass instead of center of rigidity which incorporates the effect that a value less than or equal to 1.0 will yield. Placing seismic load at center of mass of a rigid diaphragm automatically includes inherent torsion in analysis corresponding to static eccentricity (the difference between center of mass and center of rigidity). Providing `DEC` parameter as 0.0 for a model having rigid diaphragm to ignore inherent torsion is not possible.

c. The `ACC` command must not be present in seismic definition (i.e., in the `DEFINE *code* LOAD` command). If present, the natural torsion factor will be ignored and only the accidental torsion for all seismic loads will be considered.

The design eccentricity for calculating horizontal torsion is the `DEC` + `ACC` values. When `ACC` is negative, it becomes `DEC` - `ACC` (i.e., the torsion magnitudes are always additive).

Note: Refer to Notes (on page 2389) for additional information on using this command in conjunction with IS 1893 static seismic loads.

Dynamic Eccentricity

The static eccentricity is generally defined as the distance between the center of mass (CM) and the center of rigidity (CR) at respective floors levels. Accidental eccentricity generally accounts for factors such as:

- the rotational component of ground motion about the vertical axis,
- the difference between computed and actual values of the mass, stiffness, or strength, and
- uneven live mass distribution.

The provision for design eccentricity e_{di} at ith floor of a building is given by the following equation:

$$e_{di} = DEC \times e_{si} + ECC \times b_i$$

where

e_{si}	=	static eccentricity at ith floor
b_i	=	plan dimension of the ith floor normal to the direction of ground motion
`ECC` and `DEC`	=	Factors to determine the design eccentricity. These are input parameters.

Only `TOR ECC 0.05` or `TOR ECC -0.05` can also be defined without specifying `DEC 1.0` since it is the default that is included in the analysis.

Notes

a. The static seismic load cases should be provided as the first set of load cases. Other (non-seismic) primary load case specified before a seismic load case is not acceptable. Additional loads such as `MEMBER LOADS` and `JOINT LOADS` may be specified along with the seismic load under the same load case.

Example of *Incorrect* Usage: The error here is that the UBC cases appear as the 3rd and 4th cases, when they should be the 1st and 2nd cases.

```
⋮
LOAD 1
```

```
SELFWEIGHT Y -1
LOAD 2
JOINT LOAD
3 FX 45
LOAD 3
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 4
UBC LOAD Z 1.2
MEMEBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
```

Example of Correct Usage

```
⋮
SET NL 10
⋮
LOAD 1
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 2
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 3
SELFWEIGHT Y -1
LOAD 4
JOINT LOAD
3 FX 45
PERFORM ANALYSIS
LOAD LIST ALL
```

b. If the static seismic cases

are to be factored later in a `REPEAT LOAD` command;
or if the static seismic case is to be used in a tension/compression analysis;
or if re-analysis (i.e., two analysis commands without a `CHANGE` or new load case in between);

then each seismic case should be followed by `PERFORM ANALYSIS` then `CHANGE` commands as shown in the example above. Otherwise the `PERFORM ANALYSIS` then `CHANGE` can be omitted. Using the `CHANGE` command will require the `SET NL` command to define the maximum number of load cases being entered in the analysis. Also `LOAD LIST ALL` should be entered after the last `PERFORM ANALYSIS` command.

Example of *Incorrect* Usage: The error here is that the `CHANGE` command is missing before Load Case 2.

```
⋮
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
```

```
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
```

Example of Correct Usage

```
⋮
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
Change
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
CHANGE
```

c. Up to 8 sesimic load cases may be entered.
d. The `REPEAT LOAD` specification cannot be used for load cases involving seismic load generation unless each seismic case is followed by an analysis command then `CHANGE`.

   Example of repeat load using a seismic load case:

```
⋮
LOAD 1
UBC LOAD X 1.0
PDELTA ANALYSIS
CHANGE
LOAD 2
SELFWEIGHT Y -1
PDELTA ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.4 2 1.2
PDELTA ANALYSIS
```

e. If seismic load generation is performed for the X and the Z (or Y if Z up) directions, the command for the X direction must precede the command for the Z (or Y if Z up) direction.

UBC Example

In the following example, notice that the first three load cases are UBC load cases. They are specified before any other load cases.

```
DEFINE UBC LOAD
ZONE 0.2 K 1.0 I 1.5 TS 0.5
SELFWEIGHT
JOINT WEIGHT
1 TO 100 WEIGHT 5.0
101 TO 200 WEIGHT 7.5
LOAD 1 UBC in X-Direction
UBC LOAD X DEC 1.0 ACC 0.05
```

```
JOINT LOAD
5 25 30 FY -17.5
PERFORM ANALYSIS
CHANGE
LOAD 2 UBC in X-Direction
UBC LOAD X DEC 1.0 ACC -0.05
JOINT LOAD
5 25 30 FY -17.5
PERFORM ANALYSIS
CHANGE
LOAD 3 UBC in Z-Direction
UBC LOAD Z DEC 0.0 ACC 0.05
PERFORM ANALYSIS
CHANGE
LOAD 4 Dead load
SELFWEIGHT
LOAD COMBINATION 4
1 0.75 2 0.75 3 1.0
```

IS 1893 Example

In the following example, the first two load cases are the 1893 load cases. They are specified before any other load case.

```
DEFINE 1893 Load
ZONE 0.05 RF 1.0 I 1.5 SS 1.0
SELFWEIGHT
JOINT WEIGHT
7 TO 12 WEIGHT 17.5
13 TO 20 WEIGHT 18.0
MEMEBER WEIGHT
1 TO 20 UNI 2.0
LOAD 1 1893 Load in X-Direction
1893 LOAD X
JOINT LOAD
5 25 30 FY -17.5
LOAD 2 1893 Load in Z-Direction
1893 LOAD Z
LOAD 3 Dead Load
SELFWEIGHT
LOAD COMBINATION 4
1 0.75 2 0.75 3 10
```

Related Links

- *G.16.2 Seismic Load Generator* (on page 2135)
- *M. To add a seismic load* (on page 860)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)

- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)
- *G.16.2 Seismic Load Generator* (on page 2135)

TR.32.12.3 Generation of Wind Loads

This command is used to generate Wind Loads using previously specified load definitions. This command should be part of a load case.

The built-in wind load generation facility can be used to calculate the wind loads based on the parameters defined in TR.31.3 Definition of Wind Load (on page 2435). The following general format should be used to perform the wind load generation. See G.16.3 Wind Load Generator (on page 2136) for the two types of structures on which the load can be generated. For closed type structures, the vertical panel areas bounded by beam members only (and ground), and exposed to the wind, are used to define loaded areas (plates and solids are ignored). The loads generated are applied only at the joints at vertices of the bounded areas. For open type structures also, generation is done considering only the members in the model.

The automated load generator should only be used for vertical panels. Panels not parallel to the global Y axis (for Y UP) should be loaded separately.

General Format

```
LOAD i

WIND LOAD (-){ X | Y | Z } (f) TYPE j (OPEN) { { XR f1 f2  | YR f3 f4 ZR | f5 f6 }* |
LIST memb-list | ALL }
```

Where:

LOAD *i*	load case number
(-){ X | Y | Z }	Direction of wind in global axis system. Use horizontal directions only.
f	The factor to be used to multiply the wind loads. Negative signs may be used to indicate opposite direction of resulting load (default=1.0). Using X, -X, Z or –Z and the *f* factor. With respect to the axis, a minus sign indicates that suction occurs on the other side of the selected structure. If all of the members are selected and X (or Z) is used and the factor is positive, then the exposed surfaces facing in the –x (or –z) direction will be loaded in the positive x (or z) direction (normal wind in positive direction). See diagrams that follow. If X and a negative factor is used, then the exposed surfaces facing in the +x direction will be loaded in the negative x direction (normal wind in negative direction). [If –X is entered and a negative factor, then the exposed surfaces facing in the -x direction will be loaded in the negative x direction (suction). If –X is entered and a positive factor, then the exposed surfaces facing in the +x direction will be loaded in the positive x direction (suction).]

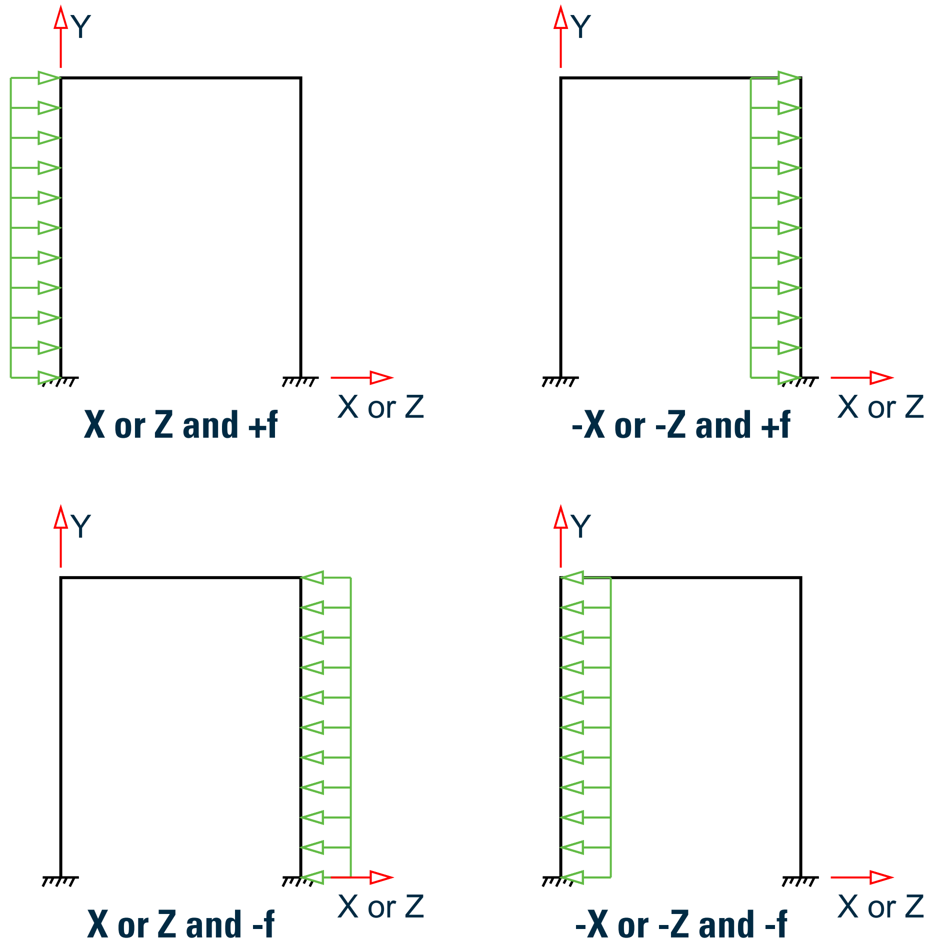


Figure 333: Sign convention for internal and external pressure

TYPE ***j***	Type number of previously defined systems
OPEN	optional word to be used if loading is to be generated on “open” type of structures. If this not specified, load will be generated assuming the panels are “closed”.
XR ***f1 f2***	global coordinate values to specify X or Y or Z range for member selection

A member list or a range of coordinate values (in global system) may be used. All members which have both end coordinates within the range are assumed to be candidates (for closed type structures) for defining a surface which may be loaded if the surface is exposed to the wind. The loading will be in the form of joint loads (not member loads). 1, 2, or 3 ranges can be entered to form a “layer”, “tube”, or “box” for selecting members in the combined ranges. Use ranges to speed up the calculations on larger models. Using multiple, overlapping ranges (in a sequence of `WIND LOAD` entries within the same load case) may be used to further control the automatic panel identification.

Example

```
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.12 HEIGHT 100 200
EXP 0.6 JOI 1 TO 25 BY 7 29 TO 37 BY 4 22 23
```

```
TYPE 2
INT 0.1 0.12 HEIGHT 100 900
EXP 0.3 YR 0 500
LOAD 1
SELF Y -1.0
LOAD 2
WIND LOAD Z 1.2 TYPE 2 ZR 10 11
LOAD 3
WIND LOAD X TYPE 1 XR 7 8 ZR 14 16
LOAD 4 SUCTION ON LEEWARD SIDE
WIND LOAD -X 1.2 LIST 21 22 42
```

Example

For open structures

```
LOAD 1 WIND LOAD IN Z DIRECTION
WIND LOAD 2 -1.2 TYPE 1 OPEN
```

Notes

a. For closed type structures, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is then calculated. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Hence, one should make sure that the members/joints that are exposed to the wind make up a closed surface (ground may form an edge of the closed surface). Without a proper closed surface, the area calculated for the region may be indeterminate and the joint force values may be erroneous. Consequently, the number of exposed joints should be at least three.

b. Plates and solids are not considered for wind load generation. On such entities, wind must be applied using pressure loading facilities for plates and solids.

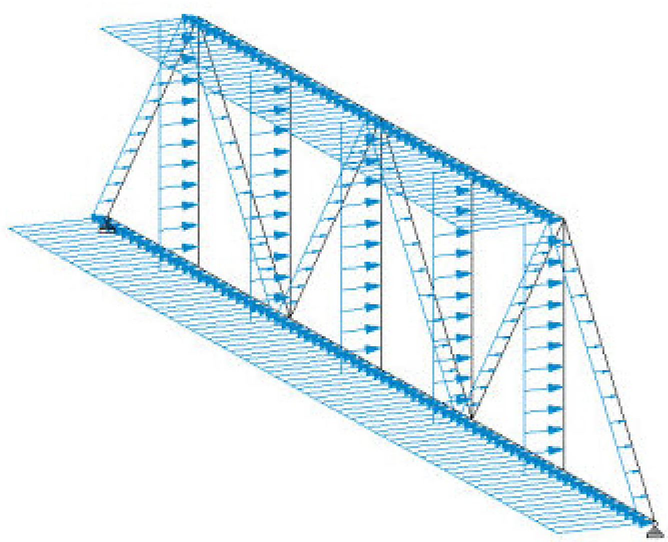
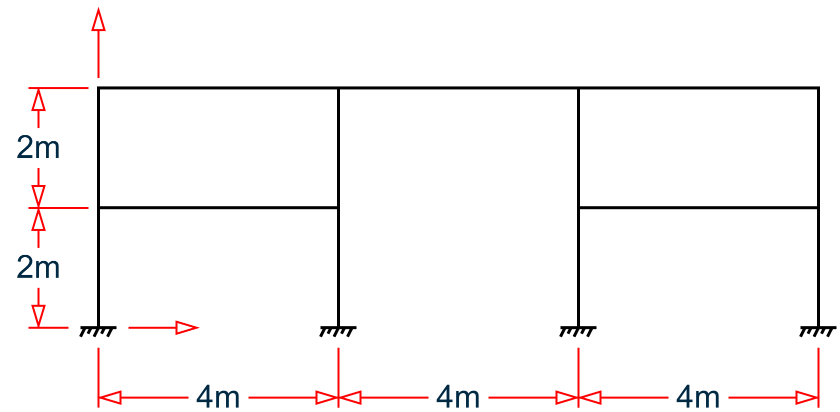


Figure 334: Load diagram for wind on open structures

c. The program identifies the panels for wind load generation based on the geometry of the structure. However, the panel identification routine may fail to identify some open-ended panels (i.e., panels bound by the "ground" at the bottom rather than a member). In this case, you may use multiple, overlapping ranges or member lists to further refine the panel identification.

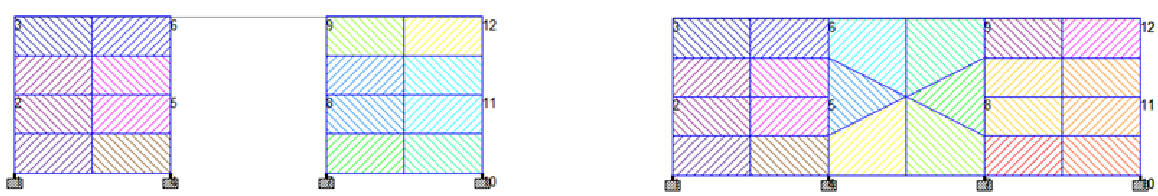
Controlling Open-Ended Panel Identification

The following structure is a wall with a large door opening. The wind is intended to applied across the entire wall. When a single range is used (Case A) to identify the wind load, the center panel is not correctly identified.




```
DEFINE WIND LOAD
TYPE 1 WIND 1
INT 0.1 HEIG  12
EXP 1 JOINT 1 TO 3 10 12
LOAD 1 LOADTYPE Wind  TITLE Wind Using Single Range
WIND LOAD X -1 TYPE 1 YR 0 4 ZR 0 12
```

A series of three ranges which overlap along the global Z direction (Case B) can be used to force the program to evaluate each bay individually.



Case A Case B

```
DEFINE WIND LOAD
…
LOAD 2 LOADTYPE Wind  TITLE Wind Using Overlapping Ranges
WIND LOAD X -1 TYPE 1 YR 0 4 ZR -0.1 4.1
WIND LOAD X -1 TYPE 1 YR 0 4 ZR 3.9 8.1
WIND LOAD X -1 TYPE 1 YR 0 4 ZR 7.9 12.1
```

Similarly, a series of three wind loads could be added to a load case, each with a member list (or a predefined group of those members) that represent each bay.

```
DEFINE WIND LOAD
…
LOAD 3 LOADTYPE Wind  TITLE Wind Using Member Lists
```

```
WIND LOAD X -1 TYPE 1 LIST 1 TO 4 9 10
WIND LOAD X -1 TYPE 1 LIST 3 TO 6 11
WIND LOAD X -1 TYPE 1 LIST 5 TO 8 12 13
```

Tip: Alternatively, you could specify a pair of uniform loads along the columns either side of the middle bay. There are situations where this may, in fact, be a closer approximately of the load distribution path.

General Format for SP 20.13330.2016 Wind Loads

STAAD.Pro can generate both static and dynamic wind loads per the SP 20.13330.2016 code.

Format for the static SP 20.13330.2016 wind load:

```
LOAD i
WIND LOAD { X | Z | -X | -Z } f1 CONFIGURATION k ( NU fNU )
TYPE j { XRANGE f1 f2 | YRANGE f1 f2 | ZRANGE f1 f2 | LIST member-list | ALL }
```

Format for the dynamic SP 20.13330.2016 wind load:

```
LOAD i
(mass-data)
WIND LOAD { X | Z} f1 WALONG f2 WACROSS f3 ( GL f4 ) ( FRQ ) ( ORT ) TYPE j ( DYN k )
```

Where:

mass-data	Mass load defined within the dynamic wind load when a separate load case for modal analysis is not previously defined in the STAAD input file.
f1	Correction factor along specified direction. This needs to be used in combination with the WIND LOAD setting.
CONFIGURATION *k*	The configuration used (from 0 to 12)
NU *fNU*	Wind pressure coefficient.
WALONG *f2*	Effective length of the structure parallel to wind direction.
WACROSS *f3*	Effective projection of the structure facing the wind direction.
GL *f4*	Vertical coordinate of ground level. It should be a positive value (zero or greater) defined in current input units. If not provided ground level is automatically calculated from the lowest Y coordinate.
FRQ	Optional parameter which results in computation of dynamic wind load vector for all mode shapes extracted from Modal Analysis provided it is within code stipulated frequency limit. Absence of the parameter will result in calculation taking the 1st mode shape only.
ORT	Optional parameter to include orthogonal directions in the load case.
TYPE *j*	Wind definition type number. This definition must be of type `SP20 2016`. Both static and dyanmic wind loads per the 2016 Russian code use this wind definition.
DYN *k*	Optional parameter that indicates that the wind loading should include the dynamic component determined from the modal response and the static wind load defined in

primary load case *k*. If this option is omitted, then only the static loads from the load definition will be calculated.

General Format for Russian SNiP 1985 Wind Loads

The format for applying wind loads per the older editions of the Russian codes are as follows:

```
LOAD i
WIND LOAD { X | Z | -X | -Z} f1 CONFIGURATION k ( NU fNU )
TYPE j { XRANGE f1 f2 | YRANGE f1 f2 | ZRANGE f1 f2 | LIST member-list  | ALL }
```

The value X or Z defines the wind direction. Use of a negative sign defines a leeward wind load.

Where:

f the wind pressure factor. A negative value defines a reverse wind loading.

k For a SNiP 1985 wind load definition, a configuration parameter defined for prismatic buildings with a valid range of values is -1 to 12, where these values represent:

-1. Prismatic building structure - rectangular building. Outstanding architectural details on the left facade.
0. Prismatic building structure - rectangular building. Both facade faces are smooth (no outstanding architectural details).
1. Prismatic building structure - rectangular building. Outstanding architectural details on the right facade.
2. Prismatic building structure - rectangular building. Outstanding architectural details on both side facades.
3. Prismatic building structure - triangular building.
4. Prismatic building structure - rhombic building.
5. to 12. Prismatic building structure - number of polygon vertices in building from 5 to 12.

fNU the wind pressure correlation coefficient. If the parameter is 1, a computed value is used. For rectangular buildings, the calculated value is always used (any input is ignored).

Note: The Russian wind load routines establish the applied wind forces from a dynamic response of the structure. Either a dynamic load case should be defined before any wind load cases or loads that represent masses should also be included in the first lad case that includes Russian wind load assignments. This in effect becomes a dynamic load case. Thus, if the effects of these forces are to be combined to the effects of static load cases, then this should be done using a `LOAD COMBINATION` command.

Dynamic Wind Load per Russian SP 20.13330.2016

In order to perform dynamic wind load generation in STAAD.Pro you must previously define a static load case. The static load case could be any primary static load case which is defined before the Russian dynamic load case, including a static wind loading per the same code. This static load case will provide static load vector to the dynamic wind load module.

As the Russian dynamic wind load component requires modal masses and eigen vectors to calculate the dynamic wind load component at nodes, modal analysis must be performed before the dynamic wind load definition. Therefore, you must also include a separate load case for modal analysis with reference mass defined before the load case. See G.17.3.2 Mass Modeling (on page 2157) for details. Alternatively, if the mass loads are not needed for use with other load cases (that is, a reference load case is not needed for other loads), then you may define the mass loads within the dynamic load case prior to the `WIND LOAD` command.

The overall wind load effect will be determined by the sum of effects of the static load and an SRSS of all of the contributing modes of the dynamic wind effect which will be used to increase the magnitude of the static load effect.

$$w = w_m + w_p$$

where

w_m	=	mean wind pressure
w_p	=	quasi-static wind fluctuation component

STAAD.Pro uses the static wind load and the modal results to create a combined dynamic load from which modal loads are calculated. The modal loads are analyzed and the overall dynamic effect is determined by combining all these with an SRSS combination which is used to increase the effect of the static wind component.

Related Links

- *M. To add a SNiP wind load definition* (on page 847)
- *M. To apply a dynamic wind load per SP 20.13330.2016* (on page 856)
- *M. To add a SNiP wind load definition* (on page 847)
- *M. To add an ASCE 7 wind load definition* (on page 845)
- *G.16.3 Wind Load Generator* (on page 2136)
- *G.16.3 Wind Load Generator* (on page 2136)
- *V.Wind On Closed Structure 1* (on page 3711)
- *GUID-6A46B9D1-50EA-417D-B730-A8877E54DA2C-low.svg*
- *V.Wind On Closed Structure 2* (on page 3718)
- *V. Wind On Open Structure* (on page 3726)

TR.32.12.4 Generation of Snow Loads

This command is used to generate Snow Loads using previously specified Snow load definitions. This input should be a part of a load case.

General Format

```
SNOW LOAD
```

```
_flr_groupTYPE j CS f1 { BALANCED | UNBALANCED } { OBSTRUCTED | UNOBSTRUCTED } { MONO |
 HIP | GABLE }
```

Where:

Parameter	Default Value	Description
_flr_group	-	The members that form the roof and that are to be loaded by snow load must be listed in a floor group (See TR.16 Entities as Single Objects (on page 2235)).
TYPE *j*	-	The type number of previously defined snow load system.
CS *f1*	0.0	Roof Slope factor (CS). For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

Parameter	Default Value	Description
BALA or UNBA	BALA	Balanced or unbalanced snow load. Default is balanced. These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.
OBST or UNOB	UNOB	Obstructed or unobstructed.
MONO or HIP or GABLE	NONO	Roof geometry type.

Use as many floor groups and types as necessary in each load case.

Related Links

- *G.16.4 Snow Load* (on page 2138)
- *M. To add an ASCE 7-02 snow load* (on page 870)
- *G.16.4 Snow Load* (on page 2138)

TR.32.13 Notional Loads

A notional load is a lateral load (horizontal load) which is derived from an existing vertical load case. This load type has been introduced to accommodate a requirement in design codes. The AISC 360-05 specification for example defines notional loads as lateral loads that are applied at each framing level and are specified in terms of gravity loads.

Description

Both Principal and Reference load cases can be selected and moved into the Notional Load Definition where the required factor and direction can be specified. The notional loads are calculated and applied as joint loads.

Note: The actual values of the applied loading are not displayed in the User Interface until *after* the analysis has been performed.

General Format

```
LOAD nLoad LOADTYPE type TITLE title
...
Load items
...
NOTIONAL LOAD
n { X | Z } (f1)
```

Where:

Parameter	Default Value	Description
n	-	Load case number of the primary load case or reference load case which contains the vertical load items.

Parameter	Default Value	Description
f1	0.002 (or user-specified)	Factor by which the contents of LN are to be multiplied. Typically, codes recommend 0.2 to 0.3 % (0.002 to 0.003). The default is 0.002 or the factor specified in `DIRECT ANALYSIS` definition block.

Multiple load items can be specified under any `NOTIONAL LOAD` and there can be one or more blocks of `NOTIONAL LOAD` command(s) in any load case.

Note: Notional Loads can be derived from gravity load cases specified under the `DEFINE REFERENCE LOAD` table which are described in TR.31.6 Defining Reference Load Types (on page 2453).

Example 1

```
LOAD 1 : DEAD
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100
LOAD 2: DEAD NOTIONAL LOAD
NOTIONAL LOAD
1 X 0.002
…
LOAD 10: IMPOSED
JOINT LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -50
LOAD 11:IMPOSED NOTIONAL LOAD
NOTIONAL LOAD
10 X 0.002
```

Example 2

If we want to combine the load cases in Example 1, two methods are available:

```
LOAD 3 : DEAD LOAD + IMPOSED LOAD + Notional Loads
REPEAT LOAD
1 1.0 10 1.0
NOTIONAL LOAD
1 X 0.002 10 X 0.002
```

As an alternative, the following could be used:

```
LOAD 3 : DEAD LOAD + IMPOSED LOAD + Notional Loads
REPEAT LOAD
1 1.0 2 1.0 10 1.0 11 1.0
```

Example 3

Similarly,

```
Load 1 LOADTYPE None TITLE DEAD
SELFWEIGHT Y -1.15
```

```
MEMBER LOAD
143 145 CON GY -16
*
Load 2 LOADTYPE None TITLE DEAD-NOTIONAL
NOTIONAL LOAD
1 X 1.0
*
Load 3 LOADTYPE None TITLE DEAD AND NOTIONAL
REPEAT LOAD
1 1.0
NOTIONAL LOAD
1 X 1.0
```

Alternatively the following could be used to define load case 3:

```
Load 3 LOADTYPE None TITLE DEAD AND NOTIONAL
REPEAT LOAD
1 1.0 2 1.0
```

Example 4

Using Notional loads with REFERENCE loads

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead TITLE REF DEAD LOAD
SELFWEIGHT Y -1 LIST 1 TO 120
FLOOR LOAD
YRANGE 19 21 FLOAD -0.05 GY
YRANGE 39 41 FLOAD -0.05 GY
LOAD R2 LOADTYPE Live TITLE REF LIVE LOAD
FLOOR LOAD
YRANGE 19 21 FLOAD -0.03 GY
YRANGE 39 41 FLOAD -0.03 GY
END DEFINE REFERENCE LOADS
**********************************************
* LOAD COMBINATIONS INCLUDING NOTIONAL LOADS *
**********************************************
*
*DEAD + LIVE +/- NOTIONAL LOAD
*
LOAD 1
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X 0.0024 R2 X 0.0032
LOAD 2
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 X -0.0024 R2 X -0.0032
LOAD 3
REFERENCE LOAD
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z 0.0024 R2 Z 0.0032
LOAD 4
REFERENCE LOAD
```

```
R1 1.2 R2 1.6
NOTIONAL LOAD
R1 Z -0.0024 R2 Z -0.0032
```

If a `PRINT STATIC CHECK` or `PRINT LOAD DATA` command is added with any analysis specification in the input file, the generated NOTIONAL LOADs would be printed as in the following:

```
   NOTIONAL LOAD - (POUN,FEET)
    LOADING    4  DL+NDLX
      JOINT          DIRECTION                   LOAD
…
        6117           X     -0.00200 X      74124.266 =        -148.249
        6118           X     -0.00200 X     115504.569 =        -231.009
        6119           X     -0.00200 X      38397.157 =        -76.794
        6120           X     -0.00200 X      15551.155 =        -31.102
        6121           X     -0.00200 X      17347.718 =        -34.695
        6122           X     -0.00200 X      29341.723 =        -58.683
        6123           X     -0.00200 X      39771.147 =        -79.542
        6124           X     -0.00200 X      39712.750 =        -79.426
        6125           X     -0.00200 X      34006.254 =        -68.013
                             ==============================================
48554642.872     -97109.290
```

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *M. To add a notional load case* (on page 872)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

TR.33 Reference Load Cases - Application

Describes how you can call the data specified under those types in actual load cases. Reference Load types are described in TR.31.6 Defining Reference Load Types (on page 2453).

General Format

The format of a reference to a Reference Load in a primary load (j) case is thus:

```
LOAD j LOADTYPE (type) load_title
REFERENCE LOAD
R(i) 1.0
…
```

Example

```
LOAD 1 LOADTYPE None TITLE D+L
REFERENCE  LOAD
R1  1.0 R2 1.0
LOAD  2 LOADTYPE None TITLE DEAD+SNOW
REFERENCE  LOAD
R1  1.0 R3 1.0
LOAD  3 LOADTYPE None TITLE D+H
```

```
REFERENCE  LOAD
R1  1.0 R4 1.0
ELEMENT  LOAD
1212  1267 TRAP GY JT -0.54 -0.44 -0.44 -0.54.
```

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

TR.34 Frequency Calculation

There are two methods available in STAAD for calculating the frequencies of a structure:

1. an approximate method called the Rayleigh method and
2. a more exact method which involves the solution of an eigenvalue problem.

Both methods are explained in the following sections.

Related Links

- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.34.1 Rayleigh Frequency Calculation

This command may be used to calculate the Rayleigh method approximate frequency of the structure for vibration corresponding to the general direction of deflection generated by the load case that precedes this command. Thus, this command typically follows a load case.

General Format

```
CALCULATE RAYLEIGH (FREQUENCY)
```

Description

This command is specified after all other load specifications of any primary load case for which the Rayleigh frequency is calculated. This Rayleigh frequency calculation is based on the Rayleigh iteration method using 1 iteration. If a more accurate, full-scale eigensolution is required, the `MODAL CALCULATION` command (see next section) may be used. A full eigensolution is automatically performed if a `RESPONSE SPECTRUM` or `TIME HISTORY` is specified in any load case.

Example

```
LOADING 1
SELFWEIGHT X 1.0
CALCULATE RAYLEIGH FREQUENCY
LOADING 2
SELFWEIGHT Z 1.0
```

```
CALCULATE RAYLEIGH FREQUENCY
LOADING 3 WIND LOAD
```

In this example, the Rayleigh frequency is calculated for the X direction mode of vibration in load case 1, and the Z direction in case 2.

In case 1, the structure is being displaced by `SELFWEIGHT` applied statically along global X. For most frames which are supported only at their base, this produces a deflected shape resembling the lowest mode shape along global X. Hence, this frequency will resemble that for the lowest X direction mode.

Similarly, the frequency calculated for load case 2 ought to be similar to that for the lowest Z direction mode because the selfweight is applied statically along global Z.

Tip: If you want a mode or frequency to be for lateral motion (X or Z), then enter the loads in the X or Z directions. Entering loads in the vertical direction when the lowest modes are in the lateral direction is a common mistake.

The output will consist of the value of the frequency in cycles per second (cps), the maximum deflection along with that global direction and the joint number where that maximum occurs.

Notes

This command is based on the Rayleigh method of iteration using 1 iteration. The frequency calculated estimates the frequency as if the structure were constrained to vibrate in the static deflected shape generated by the loads in the load case.

In many instances, the forces should be in one global direction to get the mode and frequency associated with that direction.

Related Links

- *M. To calculate the structure frequency* (on page 843)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.34.2 Modal Calculation Command

This command may be used to obtain a full scale eigensolution to calculate relevant frequencies and mode shapes. It should not be entered if this case or any other case is a `TIME LOAD` or `RESPONSE SPECTRUM` case. For Steady State/Harmonic analysis this command must be included in the load case that defines the weights and weight moment of inertias for eigensolutions

General Format

```
MODAL CALCULATION ( REQUESTED ) ( MISSING MASS )
```

This command is typically used in a load case after all loads are specified. The loads will be treated as weights and weight moment of inertias for eigensolutions (see G.17.3 Dynamic Analysis (on page 2154) and TR.32 Loading Specifications (on page 2461)). You are advised to specify the loads keeping this in mind.

This case will be independently solved statically and dynamically. Static results using the loads will include joint displacements, member forces, support reactions, and other outputs computed from a normal static analysis without any dynamic effects included.

STAAD.Pro can include the stress stiffening effect (geometric stiffness) based on the axial member forces/plate in-plane stresses from a selected load case when calculating the modes & frequencies of a structure.

In addition, the Dynamic results (using loads as masses) will include Mode Shapes and Frequencies.

Note: The `MODAL CALCULATION` command can be included in any of the primary load cases, but only in one of them.

For Steady State/Harmonic analysis, enter the `MISSING MASS` parameter to include the missing mass procedure in the analysis.

Related Links

- *M. To calculate the structure frequency* (on page 843)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *G.17.3 Dynamic Analysis* (on page 2154)
- *EX. US-28 Calculation of Modes and Frequencies of a Bridge* (on page 6536)
- *EX. UK-28 Calculation of Modes and Frequencies of a Bridge* (on page 6823)
- *M. To calculate the structure frequency* (on page 843)

TR.35 Load Combination Specification

This command may be used to combine the results of the analysis. The structure will *not* be analyzed for the combined loading. The combination of results may be algebraic, SRSS, a combination of both algebraic and SRSS, or Absolute.

Note: The `LOAD COMBINATION` specification is not appropriate for obtaining secondary effects for combined loads. Refer to notes below.

General Format

```
LOAD COMBINATION ( { SRSS | ABS } ) ( GENERATE ) i a1
```

```
i1 f1 i2 f2 … (fSRSS)
```

Where:

i	Load combination number. This can be any integer smaller than 100,000 that is not the same as any previously defined primary load case or load combination number.
a1	Any title for the load combination.
***i1*, *i2*, …**	The load case or load combination numbers which are to be combined. For the SRSS option, a minus sign may be used to indicate that this load case should be combined algebraically rather than as an SRSS case.
***f1*, *f2*, …**	The corresponding factors to be applied to loadings. Values greater than or equal to 0.0 are allowed.
fSRSS	An optional factor to be applied as a multiplying factor on the combined result of the SRSS load combination (see examples below).
GENERATE	Optional command for `SRSS` or `ABS` load combinations which will generate load combinations representing all possible positive or negative actions in each load degree of freedom. When used, a single set of absolute value analysis results is replaced by these 64 load combination analysis results.

A hyphen may be used at the end of a line to continue this command onto the next line.

Description

If the load combination option is left out (i.e., neither ABS or SRSS is included), then the results from analyses will be combined algebraically.

```
LOAD COMBINATION 6 DL+LL+WL
1 0.75 2 0.75 3 1.33
```

If the ABS load combination method is included, then the absolute value of results from the analyses will be combined.

Note: If GEN is included this single set of absolute values is replaced by 64 sets of load combinations.

```
LOAD COMBINATION ABS 7 DL+LL+WL
1 0.85 2 0.65 3 2.12
```

If the SRSS load combination method is included, then the results from analyses may be combined both algebraically and using the SRSS (Square Root of Summation of Squares) method. The combination scheme may be mixed if required. For example, in the same load combination case, results from load cases may be combined in the SRSS manner and then combined algebraically with other load cases.

Note: The case factor is not squared.

Note: If GEN is included this single set of absolute values is replaced by 64 sets of load combinations

Notes

a. In the LOAD COMBINATION SRSS option, if the minus sign precedes any load case no., then that load case will be combined algebraically with the SRSS combination of the rest.

b. The LOAD COMBINATION command should *not* be used to obtain the secondary effects of combined load cases through a P-Delta, Member/Spring Tension/Compression, Multilinear Spring, or Nonlinear Analysis. See the REPEAT LOAD command (TR.32.11 Repeat Load Specification (on page 2595)) for details.

c. In a load combination specification, a value of 0 (zero) as a load factor is permitted. In other words, a specification such as

```
LOAD COMB 7
1 1.35 2 0.0 3 1.2 4 0.0 5 1.7
```

is permitted. This is the same as

```
LOAD COMB 7
1 1.35 3 1.2 5 1.7
```

d. All combination load cases must be provided immediately after the last primary load case.

e. The total number of primary and combination load cases combined cannot exceed the limit described in TR.2 Problem Initiation and Model Title (on page 2202).

f. A maximum of 550 load cases can be combined using a LOAD COMBINATION command.

g. Load combinations can refer to previously defined load combination numbers.

Generate Load Combinations

If the optional `GENERATE` command is included, this single set of absolute values is replaced by 64 sets of load combinations. Otherwise, the program will follow the usual method of `ABS` or `SRSS` load combination.

Note: The generate load combination feature is implemented in AISC 360 05/10 and IS800 2007 only. For other design codes, the program will not design for these 64 generated load cases.

For a particular model, all 13 section forces for each of the members for the internally generated 64 load combination cases can be printed in an external text file. To do this, use the following `SET` command in the input file before joint incidences are specified.

```
SET CG TXT
```

This command will instruct the program to print the section forces for each member for 1 to 64 generated load combinations to a file named *`filename`* `_CG.txt` located in the same location as the STAAD input file.

Caution: For models with significantly larger number of members or load combination cases generated, this file can be come extremely large. This output option should only be used when the individual cases require review and *not* used in normal circumstance.

Response Spectra in Load Combinations

The following steps are followed in order to derive member section forces for a response spectrum case.

a. Forces at start and end joints of a member are calculated for each mode.
b. Since masses are lumped at two member ends, it is assumed that the section forces due each of the modes are varying linearly between member ends. Thus at any section the section force is linearly interpolated for each mode.
c. The forces at each section for all modes are combined using modal combination method to arrive at the final result.

The section forces for each mode for any response spectrum load case can be printed in an external text file. To do this, use the following `SET` command in the input file before joint incidences are specified.

```
SET RS TXT
```

This command will instruct the program to print the section forces for each member to a file named *`filename`* `_RESP.txt` located in the same location as the STAAD input file.

Caution: For models with significantly larger number of members or load combination cases generated, this file can be come extremely large. This output option should only be used when the individual cases require review and *not* used in normal circumstance.

Once the section forces are available it can be combined with other static load cases using SRSS/ABS and GENERATE options.

Design for Load Combinations per ASME NF

Steel design is performed for active load cases that may include primary loads and load combinations. If any load combination is an internal load generation case, instead of designing for one particular load combination case, the design is performed for all 64 load combinations. Thus, the steel design results may vary from what if it had been designed for only one combination case. If any load combination generation case becomes a critical load case for design, it is noted as such in the design output.

All section forces for the 64 generated load cases –whose parent load combination case becomes the critical design load– can be printed in an external text file. To do this, use the following `SET` command in the input file before joint incidences are specified.

```
SET NF TXT
```

This command will instruct the program to print the section forces for each member to a file named *`filename`* `_RESP.txt` located in the same location as the STAAD input file.

Caution: For models with significantly larger number of members or load combination cases generated, this file can be come extremely large. This output option should only be used when the individual cases require review and *not* used in normal circumstance.

Example of Simple SRSS Combination

Several combination examples are provided to illustrate the possible combination schemes

```
LOAD COMBINATION SRSS 8 DL+SEISMIC
1 1.0 2 0.4 3 0.4
```

This (LOAD COMBINATION SRSS 8) illustrates a pure SRSS load combination with a default SRSS factor of one. The following combination scheme will be used:

$$v = 1.0 \cdot (1 \cdot L1^2 + 0.4 \cdot L2^2 + 0.4 \cdot L3^2)^{1/2}$$

where

v	=	the combined value
L1, L2, and L3	=	values from load cases 1, 2 and 3.

Since an SRSS factor is not provided, the default value of 1.0 is being used.

Examples of Algebraic & SRSS Combination in the Same Load Combination Case

Example 1

```
LOAD COMBINATION SRSS 9
-1 0.75 2 1.3 3 2.42 0.75
```

The combination formula will be as follows:

$$v = 0.75 \times L\ 1 + 0.75 \times \sqrt{1.3(L\ 2)^2 + 2.42(L\ 3)^2}$$

where

v	=	the combined value
L2 and L3	=	values from load cases 2 and 3

In the above specification, a minus sign precedes load case 1. Thus, Load 1 is combined algebraically with the result obtained from combining load cases 2 and 3 in the SRSS manner. The SRSS factor of 0.75 is applied on the SRSS combination of 2 and 3.

Example 2

```
LOAD COMBINATION SRSS 10
-1 0.75 -2 0.572 3 1.2 4 1.7 0.63
```

Here, both load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4. Note the SRSS factor of 0.63. The combination formula will be as follows:

$$v = 0.75 \times L\ 1 + + 0.572 \times L\ 2 + 0.63 \times \sqrt{1.2(L\ 3)^2 + 1.7(L\ 4)^2}$$

Related Links

- *M. To define a new load combination* (on page 886)
- *M. To automatically generate load combinations* (on page 888)

TR.36 Calculation of Problem Statistics

This item has been removed. Please contact the Bentley Technical Support Group for further details.

TR.37 Analysis Specification

STAAD analysis options include linear static analysis, P-Delta (or second order analysis), Nonlinear analysis, and several types of Dynamic analysis.

This command is used to specify the analysis request. In addition, this command may be used to request that various analysis related data, like load info, statics check info, etc. be printed.

Related Links

- *G.17.1 Stiffness Analysis* (on page 2138)

TR.37.1 Linear Elastic Analysis

Used to perform a static, linear elastic analysis on the structure.

General Format

```
PERFORM  ANALYSIS (PRINT { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL } )
```

Without one of these analysis commands, no analysis will be performed. These `ANALYSIS` commands can be repeated if multiple analyses are needed at different phases.

The following `PRINT` options are available:

- If the `PRINT LOAD DATA` command is specified, the program will print an interpretation of all the load data.
- `PRINT STATICS CHECK` will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- `PRINT STATICS LOAD` prints everything that `PRINT STATICS CHECK` does, plus it prints a summation of all internal and external forces at each joint. This option generates a large volume of output.

Note: Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- `PRINT BOTH` is equivalent to `PRINT LOAD DATA` plus `PRINT STATICS CHECK`.
- `PRINT ALL` is equivalent to `PRINT LOAD DATA` plus `PRINT STATICS LOAD`.
- The `PRINT MODE SHAPES` command may be added separately after the Analysis command if mode shapes are desired. Refer to TR.42 Print Specifications (on page 2666) for using additional print specifications.

Notes

This command directs the program to perform the analysis that includes:

a. Checking whether all information is provided for the analysis;
b. Forming the joint stiffness matrix;
c. Checking the stability of the structure;
d. Solving simultaneous equations, and
e. Computing the member forces and displacements.
f. If a `RESPONSE SPECTRUM`, `TIME LOAD`, or `GROUND MOTION` is specified within a load case or the `MODAL CALCULATION` command is used, a dynamic analysis is performed.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

General Analysis Comments

STAAD.Pro allows multiple analyses in the same run. Multiple analyses may be used for the following purposes:

a. Successive analysis and design cycles in the same run result in optimized design. STAAD.Pro automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.

b. Multiple analyses may be used for load-dependent structures. For example, structures with bracing members are analyzed in several steps. The bracing members are assumed to take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD.Pro run using multiple `PERFORM ANALYSIS` commands. STAAD.Pro is capable of performing a design based on the load combinations provided.

Refer to Example 4 in the Getting Started & Examples manual for detailed illustration.

c. You may also use Multiple Analyses to model change in other characteristics like supports, releases, section properties, etc.
d. Multiple Analyses may require use of additional commands like the `SET NL` command (TR.5 Set Command Specification (on page 2206)) and the `CHANGE` command.
e. Analysis and `CHANGE` are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Related Links

- *A. To specify a linear elastic analysis* (on page 952)
- *G.17 Analysis Facilities* (on page 2138)

TR.37.2 P-Delta Analysis Options

Used to perform a second-order analysis, considering either large- or small-delta effects—or both—on the structure.

General Format

The following options are available in STAAD.Pro when performing a P-Delta analysis:

1. P-Delta analysis with Small & Large Delta effects or Large Delta effects only. With this option the global forces are adjusted with every iteration

   ```
   PDELTA (n) ANALYSIS ( { LARGEDELTA | SMALLDELTA } ) (PRINT  print-options)
   ```

   Where:

PDELTA *n*	no. of iterations desired (default value of n = 1).
LARGEDELTA | SMALLDELTA	`SMALLDELTA` is the default

2. P-Delta analysis including stress stiffening effect of the KG matrix. With this option the global stiffness is adjusted with every iteration

   ```
   PDELTA KG ANALYSIS (PRINT  print-options)
   ```

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }
```

See TR.37.1 Linear Elastic Analysis (on page 2620) for details.

Without one of these P-Delta analysis commands, no P-Delta analysis will be performed.

These `ANALYSIS` commands can be repeated if multiple analyses are needed at different phases.

A `PDELTA ANALYSIS` will correctly reflect the secondary effects of a combination of load cases only if they are defined using the `REPEAT LOAD` specification (TR.32.11 Repeat Load Specification (on page 2595)). Secondary effects will not be evaluated correctly for load combinations.

P-Delta effects are computed for frame members and plate elements only. They are not calculated for solid elements or curved beams.

Notes for Small or Large Delta (Option 1)

a. This command directs the program to perform the analysis that includes:
   a. Checking whether all information is provided for the analysis;
   b. Forming the joint stiffness matrix
   c. Checking the stability of the structure;
   d. Solving simultaneous equations, and
   e. Computing the member forces and displacements.
   f. For P-Delta analysis, forces and displacements are recalculated, taking into consideration the chosen P-Delta effect.
   g. If a `RESPONSE SPECTRUM`, `TIME LOAD`, or `GROUND MOTION` is specified within a load case or the `MODAL CALCULATION` command is used, a dynamic analysis is performed.

      Note: Computing P-Delta effects for dynamic load cases is not recommended since such effects are not considered.

   h. In each of the iterations of the `PDELTA ANALYSIS`, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous iterations.
b. The default procedure of option 1 is based on “P-small & large Delta” effects.(sometimes referred to as P-δ & P-Δ). Enter the LargeDelta parameter to only include the “PlargeDelta” effects (P-Δ only). SmallDelta is recommended.

c. This `PDELTA (n) ANALYSIS` command (option 1) should specify 3 to 30 iterations to properly incorporate the P-Delta effect. With this many iterations, the `PDELTA (n) ANALYSIS SMALLDELTA` command results are as good as or better than the `PDELTA KG` (option 2) command results for static analysis. The advantage of this `PDELTA (n) ANALYSIS` command comes from not having to re-form and then triangular factorize the stiffness matrix for every iteration within every case. Also this command allows tension/compression.
d. Be aware that global buckling can occur in P-Delta analysis, resulting in large or infinite or NaN values for displacement. Do not use the results of such a case. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Example

Following are some examples on use of the command for P-Delta analysis as described in TR.37.2 P-Delta Analysis Options (on page 2621).

```
PDELTA ANALYSIS
PDELTA 5 ANALYSIS
PDELTA 20 ANALYSIS SMALLDELTA PRINT STATICS CHECK
```

STAAD.Pro allows multiple P-Delta analyses in the same run (see the General Comments section of TR.37.1 Linear Elastic Analysis (on page 2620) for details).

Notes for Stress Stiffening Matrix (Option 2)

The P-Delta analysis also provides the option of including the stress stiffening effect of the Kg matrix into the member / plate stiffness.

a. A regular STAAD P-Delta Analysis (option 1) performs a first order linear analysis and obtains a set of joint forces from member/plates based on both large and small P-Delta effects. In contrast, the P-Delta KG Analysis (that is, with the Kg option selected) includes the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included in this KG option.
b. This command directs the program to perform the analysis that includes:
   a. Solving the static case.
   b. Reforming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces.
   c. Solving simultaneous equations for displacements;
   d. If a `RESPONSE SPECTRUM`, `TIME LOAD`, or `GROUND MOTION` is specified within a load case or the `MODAL CALCULATION` command is used, a dynamic analysis is performed. The static cases solved for a `PDELTA KG` analysis command will be solved first then the dynamic analysis cases.

   Note: The stiffness matrix used in the dynamic analysis will be the K+Kg matrix used in the last iteration for the last static case. This is a stress stiffened dynamic analysis, sometimes known as a PDELTA Dynamic analysis.

c. A `PDELTA KG ANALYSIS` will correctly reflect the secondary effects of a combination of load cases only if they are defined using the `REPEAT LOAD` specification (TR.32.11 Repeat Load Specification (on page 2595)). Secondary effects will not be evaluated correctly for load combinations since only final results are combined.
d. P-Delta KG effects are computed for frame members and plate elements only. They are not calculated for solid elements. The results are based on “P-large & small Delta” effects.
e. For static analysis, the other P-Delta command [`PDELTA (n) ANALYSIS (SMALL)`] with 20 or more iterations (option 1) is preferred.
f. Tension/compression only members are *not* allowed with this `PDELTA KG` command. You must use the `PDELTA (n) ANALYSIS (SMALL)` command instead.
g. Be aware that global buckling can occur in a `PDELTA KG ANALYSIS`. This condition is usually detected by STAAD.Pro. A message is issued and the results for that case are set to zero. STAAD.Pro will continue with the next load case.
h. Global buckling may not be detected which could result in a solution with large or infinite or NaN values for displacement or negative L-matrix diagonals or stability errors. Do not use the results of such cases. This condition may require a nonlinear analysis. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for P-Delta analyses.

Example

```
PDELTA KG ANALYSIS PRINT BOTH
PDELTA KG 2 ANALYSIS
```

Related Links

- *G.17.2.1 P-Delta Analysis* (on page 2141)
- *A. To specify a P-Delta analysis* (on page 952)
- *G.17.2.1.1 P-Delta Analysis – Large Delta and Small Delta* (on page 2141)
- *G.17.2.1.2 P-Delta Kg Analysis* (on page 2142)
- *G.17.2.1.3 P-Delta K+Kg Dynamic Analysis* (on page 2143)

TR.37.3 Nonlinear Cable Analysis

General Format

For basic cable analysis:

```
PERFORM CABLE ANALYSIS BASIC (STEPS f1) (EQITERATIONS f2) (EQTOLERANCE f3) (SAGMINIMUM f4) (STABILITY f5 f6) (KSMALL f7) (PRINT print-options)
```

Note: Use of the Basic cable analysis requires including the `BASIC` option in the command.

For advanced cable analysis:

```
PERFORM CABLE ANALYSIS ( ADVANCED ) (STEPS f8) (EQITERATIONS f9) (EQTOLERANCE f10) (REFORM f11) (KGEOM f12) (PRINT print-options)
```

Note: Use of the Advanced Cable analysis feature requires the STAAD.Pro Advanced License. The `ADVANCED` option is the default if you have the license.

Where:

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }
```

See TR.37.1 Linear Elastic Analysis (on page 2620) for details.

See TR.42 Print Specifications (on page 2666) for details on including the cable sag in the output.

This command may be continued to the next line by ending with a hyphen.

Table 279: Parameters for basic cable analysis

Parameter	Default Value	Description
STEPS *f1*	145	The number of load steps for basic cable analysis. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. If entered, the value should be in the range 5 to 145.
EQITERATIONS *f2*	300	Maximum number of iterations permitted in each load step. Should be in the range of 10 to 500.
EQTOLERANCE *f3*	0.0001	The convergence tolerance for the above iterations.
SAGMINIMUM *f4*	0.0	Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction). As soon as SAGMIN becomes less than 0.95, the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The Eq iterations may need to be 300 or more. The Eq tolerance may need to be greater or smaller.

Parameter	Default Value	Description
STABILITY *f5*	1.0	A stiffness matrix value (f5) that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first f6 load steps. The amount added linearly decreases with each of the f6 load steps. If f5 is entered, use 0.0 to 1000.0. This parameter alters the stiffness of the structure.
f6	1	The number of load steps over which f_5 is gradually applied. The default is one step.
KSMALL *f7*	0.0	A stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. The range for f_7 is between 0.0 and 1.0. This parameter alters the stiffness of the structure.

Table 280: Parameters for advanced cable analysis

Parameter	Default Value	Description
STEPS *f8*	1	Used to divide the loading into small increments. Any positive integer is valid. Generally, a value larger than one (1) may be beneficial for solution convergence, because the loading is applied gradually. However, with cable elements having fixed prestressing, a single step typical converges faster.

Parameter	Default Value	Description
EQITERATIONS *f9*	300	Maximum number of iterations permitted in each load step. During each loading increment, the analysis will iterate to find the converging solution. Once the iteration number reaches the maximum value, the analysis will be stopped even though converged solution is not achieved. Any positive integer is valid. Values too small may prevent the solution to achieve convergence. However, excessively large values may cost unnecessary running time when the problem diverges.
EQTOLERANCE *f10*	1.0E-6	The convergence tolerance for the iterations specified in `EQITERATIONS`. The residual force norm is used for this convergence. This error threshold indicates when the nonlinear solver will stop iterating and consider the ongoing step as converged once the computed error is equal or smaller than this value. A value that is too small may prevent the solution to be considered as converged. However, value that is too larger may result in inaccurate results. The tolerance printed in the output file is the norm of the current out-of-balance forces, divided by the norm of all the externally applied forces. This calculated tolerance must be less than f_{10} for the current iteration to converge.
REFORM *f11*	1	Used to specify if full Newton-Raphson method or modified Newton- Raphson method will be used. Theoretically, the full Newton-Raphson method can approach converge with less iterations but may require more computation effort than modified Newton-Raphson method.

Parameter	Default Value	Description
KGEOM *f12*	0	Used to specify whether or not geometric stiffness will be used.

Notes

a. STAAD allows multiple analysis in the same run (see the General Comments in TR.37.1 Linear Elastic Analysis (on page 2620)).
b. Multiple analyses may require use of additional commands like the `SET NL` command and the `CHANGE` command.
c. Analysis and `CHANGE` are required between primary cases for `PERFORM CABLE ANALYSIS BASIC` or `PERFORM CABLE ANALYSIS ADVANCED`.
d. A cable element cannot be loaded with either a non-uniformly distributed load or a concentrated load (using a `MEMBER LOAD` command).

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Related Links

- *A. To specify a nonlinear cable analysis* (on page 955)
- *G.17.2.7 Nonlinear Cable or Truss Analysis* (on page 2149)
- *G.8.2.1 Linearized Cable Members* (on page 2122)
- *G.8.2.2 Nonlinear Cable and Truss Members* (on page 2123)
- *G.17.2.7 Nonlinear Cable or Truss Analysis* (on page 2149)
- *G.8.2.3 Nonlinear Cable Members for Advanced Cable Analysis* (on page 2124)
- *G.17.2.8 Advanced Nonlinear Cable Analysis* (on page 2150)

TR.37.4 Buckling Analysis

General Format

```
PERFORM BUCKLING { ANALYSIS (MAXSTEPS f1) | EIGEN } (PRINT print-options)
```

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }
```

See TR.37.1 Linear Elastic Analysis (on page 2620) for details.

Without this command (or any of the analysis commands described earlier), no analysis will be performed. These `ANALYSIS` commands can be repeated if multiple analyses are needed at different phases.

Where:

EIGEN if specified the analysis will perform the eigen method, otherwise the iterative method will be used

Note: This requires a STAAD.Pro Advanced license.

MAXSTEPS *f1* the maximum no. of iterations desired (default value of n = 20). 15 is recommended. For use with the iterative (`ANALYSIS`) method only.

There are two procedures available:

i. the iterative method: `PERFORM BUCKLING ANALYSIS`
ii. the eigen method: `PERFORM BUCKLING EIGEN`

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Iterative Method

This command directs the program to perform an analysis that includes:

a. Solving the static case.
b. Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces and inplane plate stresses..
c. Solving simultaneous equations for displacements;
d. Repeat b) and c) for the number of required additional iterations; either until convergence or until MAXSTEPS is reached.

If the loads must be in the opposite direction, STAAD.Pro will stop solving that case at 1 iteration. The results for the case will be outputted; then STAAD.Pro will continue with the next case.

Convergence occurs when two consecutive buckling factors in the iteration are within 0.1% of each other.

Results are based on the highest successful Buckling Factor estimate that was calculated; as if the original applied loads times the buckling factor had been entered.

Eigen Method

Note: `MAXSTEPS` input is ignored when using the eigen method.

This command directs the program to perform the analysis that includes:

a. Solving the static case.
b. Reforming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces & inplane plate stresses..
c. Solving an eigenvalue problem for up to 4 buckling factors and buckling shapes

A Buckling Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the `REPEAT LOAD` specification (TR.32.11 Repeat Load Specification (on page 2595)). Buckling will not be performed for `LOAD COMBINATIONS` cases.

Buckling Kg matrices are computed for frame members and plate elements only. They are not calculated for solid elements or curved members. The results are based on “P-large & small Delta” effects (refer to Option 1 in TR.37.2 P-Delta Analysis Options (on page 2621)).

Buckling Analysis solves for Buckling Factors. These are the amounts by which the load case must be factored for the buckling shape to occur. Only the last buckling case will be presented in the post processing. However, the buckling factors of all buckling cases will be written to the output.

If the loads must be in the opposite direction, STAAD.Pro will compute negative buckling factors.

Results are for the normalized buckling shape not as if the original applied loads times the buckling factor had been entered.

Example with eigen method:

```
PERFORM BUCKLING  EIGEN
```

Related Links

- *G.17.2.2 Buckling Analysis* (on page 2145)
- *A. To specify buckling analysis* (on page 957)
- *G.17.2.2 Buckling Analysis* (on page 2145)

TR.37.5 Direct Analysis

General Format

```
PERFORM DIRECT ANALYSIS ( {LRFD | ASD} TAUTOL f1 DISPTOL f2 ITERDIRECT i3 (REDUCEDEI
i4) (PDITER i5) (TBITER) PRINT print-options )
```

```
print-options = { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH | ALL }
```

This command directs the program to perform the analysis that includes:

a. Reduce Axial & Flexure stiffness to 80% for members selected in the Define Direct input. The 80% applies only to analysis.
b. Solving the static case which has notional loads included.
c. Perform iterations of the iterative PDelta with SmallDelta analysis procedure (default 15 iterations).
d. Solving simultaneous equations for displacements;
e. Compute Tau-b of AISC 05 Direct Analysis Appendix 7 based on required strength versus yield strength.
f. Reforming the global joint stiffness matrix.
g. Solving simultaneous equations for displacements;
h. Repeat steps c) through g) until converged or `ITERDIRECT` iterations are reached.

A Direct Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the `REPEAT LOAD` specification (TR.32.11 Repeat Load Specification (on page 2595)) and/or `REFERENCE LOAD` specification (TR.33 Reference Load Cases - Application (on page 2613)). Direct analysis will not be performed for `LOAD COMBINATIONS` cases.

Notional loads must be defined using a `DEFINE NOTIONAL` table.

A list of members which will have their initial Tau-b value set and/or have their Axial stiffness reduced and/or their flexural stiffness reduced must be entered using a `DEFINE DIRECT` table.

For information on `NOTIONAL LOADS`, see sections TR.31.7 Definition of Direct Analysis Members (on page 2454) and TR.32.13 Notional Loads (on page 2610).

PDELTA iterative load adjustments are computed for frame members only. They are not calculated for plate or solid elements. The results are based on “P-large&small Delta” effects (refer to Option 1 in TR.37.2 P-Delta Analysis Options (on page 2621)).

Convergence occurs when 2 consecutive iterations have all member tau-b values the same within a tolerance, `TAUTOL`, and displacements & rotations the same within a tolerance, `DISPTOL`.

LRFD is the default (all generated loads are factored by 1.0). If ASD entered then loads are factored by 1.6 for the Pdelta and Tau-b calculations. ASD final results are based on the final displacements divided by 1.6

If resulting displacements are diverging, then the P-Delta iterations will be terminated and the current iteration results will be used as the final results for that load case.

Table 281: Direct analysis parameters

Parameter Name	Default Value	Description
TAUTOL *f1*	0.01	Tau-b tolerance *f1* is normally 0.001 to 1.0.
DISPTOL *f2*	0.01 inch (displacement) 0.01 radians (rotation)	Displacement tolerance *f2* should not be too tight. The value is in current length units.
ITERDIRECT *i3*	1	Limits the number of iterations. A value for *i3* between 1 to 10 is typically sufficient.
REDUCEDEI *i4*	1	Integer, *i4*, specifies whether to use the reduced EI (Tau-b × 0.8 × EI) for member section moment and section displacement. 1) uses the reduced EI (Tau-b × 0.8 × EI) for member section moment & section displacement calculations 2) uses the full EI for member section moment & section displacement calculations
PDiter *i5*	15	The number of iterations, *i5*, used in the iterative PDelta with SmallDelta analysis procedure within Direct Analysis; 5 to 25 iterations is the normal range. The default is recommended.
TBITER	-	If this command is present, then the analysis procedure will iterate Tau-b.

Note: You can use the `SET NOPRINT DIRECT` command to turn off the tau-b details in the output file when running a Direct Analysis. This can greatly reduce the volume of output content for models with many load cases.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Example

```
PERFORM DIRECT ANALYSIS LRFD TAUTOL 0.01 -
DISPTOL 0.01 ITERDIRECT 2 TBITER -
PRINT LOAD DATA
```

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *A. To specify a direct analysis* (on page 953)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

TR.37.6 Steady State and Harmonic Analysis

This section describes how to define steady state analysis in STAAD.Pro.

Note: This requires a STAAD.Pro Advanced license.

Related Links

- *G.17.3.6 Steady State and Harmonic Response* (on page 2166)
- *G.17.3 Dynamic Analysis* (on page 2154)

TR.37.6.1 Purpose

This analysis type is used to model steady, harmonically varying load on a structure to solve for the steady harmonic response after the initial transient response has damped out to zero. STAAD.Pro Steady State analysis options include results for one forcing frequency or for a set of frequencies. You may specify ground motion or a distributed joint loading in one load case. Damping is required either in this input or from the Modal Damp input or from the Composite Damping input.

This command is used to specify the analysis request, specify that the load case with the `MODAL CALCULATION` command (which must be prior to this analysis command) be used as the definition of the mass distribution, and to begin a block of data input describing the steady state forcing functions, the output frequencies, and the printing of the joint responses.

All of the input and output frequencies are in Hertz (Hz or CPS).

Related topics can be found in the following sections:

G.17.3 Dynamic Analysis (on page 2154)
TR.30 Miscellaneous Settings for Dynamic Analysis (on page 2344)
TR.34 Frequency Calculation (on page 2614)
TR.26.4 Modal Damping Information (on page 2312)
TR.26.5 Composite Damping for Springs (on page 2313)
TR.37 Analysis Specification (on page 2620)
TR.34 Frequency Calculation (on page 2614)

Note: The Modal Calculation command is required in the weight/mass definition load case.

General Format

```
PERFORM STEADY STATE ANALYSIS
```

This command directs the program to perform the analysis that includes:

a. Checking whether all information is provided for the analysis;
b. Forming the joint stiffness matrix;
c. Solving simultaneous equations;
d. Solving for modes and frequencies;

e. Computing for the steady state joint displacements, velocities & accelerations and phase angles;
f. Computing the above quantities versus frequency and displaying the results graphically.
g. Member & element forces & stresses and support reactions are not currently computed.

The first input after the Perform Steady State Analysis command is:

```
BEGIN ( { STEADY | HARMONIC } ) ( { FORCE | GROUND } )
```

Steady or Harmonic Steady = the analysis is at one forcing frequency

Harmonic = the analysis is at several frequencies

Force or Ground Choose whether the loading is a distributed joint force load or a ground motion.

This command selects which of the 4 load/analysis types, that are available, will be used in this analysis. These four are described in sections TR.37.6.4 Steady Ground Motion Loading (on page 2634) through TR.37.6.7 Harmonic Force Loading (on page 2637).

This block of data should be terminated with the `END STEADY` command as mentioned in TR.37.6.9 Last Line of this Steady State/Harmonic Analysis (on page 2641).

The steady state/harmonic analysis will calculate the maximum displacement and the associated phase angle for each of 6 joint directions, relative to the ground motion, for each frequency defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2633).

In `PRINT JOINT DISP` and in Post processor displayed results, the load case displacement for a given joint and direction will be the maximum value over all of the frequencies (without the phase angles) for a Steady State load case.

In post-processing for harmonic analysis, Log-Log graphs of any joint's relative translational displacement or velocity or acceleration versus frequency may be selected.

See TR.37.6.8 Print Steady State/Harmonic Results (on page 2639) for printing displacements with phase angles by frequency.

TR.37.6.2 Define Harmonic Output Frequencies

If Harmonic is requested above, then optionally include the next input.

```
FREQUENCY (FLO f1 FHI f2 NPTS f3 ( MODAL ) FLIST freqs  )
```

Where:

FLO *f1* Lowest frequency to be included in Harmonic output. Default to half the first natural frequency.

FHI *f2* Highest frequency to be included in Harmonic output. Default to highest frequency plus largest difference between two consecutive natural frequencies.

NPTS *f3* Number of plot frequencies to be included between natural frequencies. Defaults to 5 (7 if fewer than 10 modes) (3 if more than 50 modes). These points are added to improve the graphic display of responses versus frequency.

The natural and forcing frequencies are automatically included in the plot frequencies.

MODAL This option causes the natural frequencies between `FLO` and `FHI` to be added to the list of forcing frequencies.

FLIST *freqs* List of forcing frequencies to be included in the Harmonic analysis. Continue *freqs* input to additional lines by ending each line except the last with a hyphen.

Only forcing frequencies will be used to create load case results and print results.

TR.37.6.3 Define Load Case Number

The load case which contains the `MODAL CALC` command is automatically used for the load case number.

TR.37.6.4 Steady Ground Motion Loading

This set of commands may be used to specify steady ground motion loading on the structure, the ground motion frequency, the modal damping, and the phase relationship of ground motions in each of the global directions.

General Format

This command specifies the ground motion frequency and damping.

```
STEADY GROUND FREQUENCY f1 { DAMP f2 | CDAMP | MDAMP } { ABSOLUTE | RELATIVE }
```

Where:

FREQ *f1* the steady state frequency in cycles/sec at which the joint loads below will oscillate.

DAMP *f2* Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the *f2* value for all modes
- MDAMP indicates to use the damping entered or computed with the `DEFINE DAMP` command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the `CONSTANT` specification

`ABSOLUTE` or `RELATIVE`. Ground motion results in output file will be relative to the ground unless `ABSOLUTE` is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

Enter the direction of the ground motion, the acceleration magnitude, and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

```
GROUND MOTION { X | Y | Z } { ACCELERATION | DISPLACEMENT } f3 ( PHASE f4 )
```

Where:

f3 Ground acceleration as multiple of ‘g’ (i.e. acceleration due to gravity 9.81 m/s^2 or 32.17 ft/s^2) *or* displacement in length units.

PHASE *f4* Phase angle in degrees. If not specified, this is take as zero (i.e., no shifting).

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.37.6.5 Steady Force Loading

This set of commands may be used to specify JOINT loads on the structure, the forcing frequency, the modal damping, and the phase relationship of loads in each of the global directions.

General Format

This command specifies the forcing frequency and damping for a case of steady forces.

```
STEADY FORCE FREQ f1 { DAMP f2 | CDAMP | MDAMP }
```

Where:

FREQ *f1* the steady state frequency in cycles/sec at which the joint loads below will oscillate.

DAMP *f2* Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the *f2* value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

Joint Loads

Refer to TR.32.1 Joint Load Specification (on page 2462) for additional information on Joint Loads

```
JOINT LOAD ( [ PHASE *{ X | Y | Z } f7] )
```

Where:

PHASE *f7* Phase angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

```
joint-list *{ FX f1 | FY f2 | FZ f3 | MX f4 | MY f5 | MZ f6 }
```

Where:

FX *f1*, FY *f3*, FZ *f3* specify a force in the corresponding global direction.

MX *f4*, MY *f4*, MZ *f6* specify a moment in the corresponding global direction.

Notes

a. Joint numbers may be repeated where loads are meant to be additive in the joint.
b. UNIT command may be on lines in between joint-list lines.
c. Forces applied at a dependent degree of freedom will be ignored.

Copy Loads

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

```
COPY LOAD
```

```
i1, f1, i2, f2 ... in, fn
```

Where:

i_1, i_2 ... i_n	prior primary load case numbers that are in this analysis set.
f_1, f_2 ... f_n	corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.37.6.6 Harmonic Ground Motion Loading

This set of commands may be used to specify harmonic ground motion loading on the structure, the modal damping, and the phase relationship of ground motions in each of the global directions.

Response at all of the frequencies defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2633) will be calculated.

General Format

```
HARMONIC GROUND { DAMP f2 | CDAMP | MDAMP } { ABSOLUTE | RELATIVE }
```

Where:

DAMP *f2* Damping ratio for all modes when DAMP is selected. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping. The steady state response will be calculated for each specified output frequency entered or generated, see TR.37.6.2 Define Harmonic Output Frequencies (on page 2633).

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the *f2* value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

ABSOLUTE or RELATIVE. Ground motion results in output file will be relative to the ground unless ABSOLUTE is specified. Graphical results are relative. This option has no effect for the force loading cases.

General Format

```
GROUND MOTION { X | Y | Z } { ACCELERATION | DISPLACEMEBNT } f3 ( PHASE f4 )
```

Enter the direction of the ground motion, the acceleration and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

Where:

f3	Ground acceleration as multiple of 'g' (i.e. acceleration due to gravity 9.81 m/s^2 or 32.17 ft/s^2) *or* displacement in length units.
PHASE *f4*	Phase angle in degrees. If not specified, this is take as zero (i.e., no shifting).

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration *f3* entered above. If no amplitude data is entered for a direction then the acceleration is *f3* for that direction.

```
AMPLITUDE ( A a B b C c )
```

where

Amplitude	=	$a \times \omega^2 + b \times \omega + c$
ω	=	forcing frequency in rad/sec.
a, b, c	=	Constants. a and b default to 0.0 and c defaults to 1.0.

Or

```
AMPLITUDE
```

```
(f1 a1  f2 a2  … fn an )
```

f1 a1 f2 a2* … *fn an	Frequency - Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

One Ground Motion and Amplitude command set can be entered for each global direction.

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.37.6.7 Harmonic Force Loading

This set of commands may be used to specify `JOINT` loads on the structure, the modal damping, and the phase relationship of loads in each of the global directions.

Response at all of the frequencies defined in TR.37.6.2 Define Harmonic Output Frequencies (on page 2633) will be calculated.

General Format

```
HARMONIC FORCE { DAMP f2 | CDAMP | MDAMP }
```

Where:

DAMP *f2*	Damping ratio for all modes when `DAMP` is selected. Default value is 0.05 (5% damping if 0 or blank entered).

This command specifies the damping for a case of harmonic forces.

DAMP, MDAMP, and CDAMP select source of damping input:

- DAMP indicates to use the *f2* value for all modes
- MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used
- CDAMP indicates to use the composite damping of the structure calculated for each mode. You must specify damping for different materials under the CONSTANT specification

Joint Loads

Refer to TR.32.1 Joint Load Specification (on page 2462) for additional information on Joint Loads

```
JOINT LOAD ( [ PHASE *{ X | Y | Z } f7] )
```

Where:

PHASE *f7*	Phase angle in degrees. One phase angle per global direction.

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

Next are the joint forces, if any. Repeat as many lines of joint force data as needed.

```
joint-list *{ FX f1 | FY f2 | FZ f3 | MX f4 | MY f5 | MZ f6 }
```

Where:

FX *f1*, FY *f3*, FZ *f3*	specify a force in the corresponding global direction.
MX *f4*, MY *f4*, MZ *f6*	specify a moment in the corresponding global direction.

Notes

a. Joint numbers may be repeated where loads are meant to be additive in the joint.
b. UNIT command may be on lines in between joint-list lines.
c. Forces applied at a dependent degree of freedom will be ignored.

Copy Loads

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

```
COPY LOAD
```

$i_1, f_1, i_2, f_2 \ldots i_n, f_n$

Where:

$i_1, i_2 \ldots i_n$	prior primary load case numbers that are in this analysis set.
$f_1, f_2 \ldots f_n$	corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration *f3* entered above. If no amplitude data is entered for a direction then the acceleration is *f3* for that direction.

```
AMPLITUDE ( A a B b C c )
```

where

Amplitude	=	$a \times \omega^2 + b \times \omega + c$
ω	=	forcing frequency in rad/sec.
a, b, c	=	Constants. a and b default to 0.0 and c defaults to 1.0.

Or

```
AMPLITUDE
```

```
(f1 a1  f2 a2  … fn an )
```

f1 a1 f2 a2 … fn an	Frequency - Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

Leaving the direction field blank or inserting `ALL` will use the same Frequency versus Amplitude for all 3 force directions.

Enter amplitudes for up to three directions. For directions without amplitude input, including moment directions, the amplitude will be set to 1.0.

The first form of the amplitude input is converted to the second form by creating 199 equally spaced frequencies, then computing the amplitude for each frequency.

TR.37.6.8 Print Steady State/Harmonic Results

General Format

```
PRINT HARMONIC DISPLACEMENTS list-spec
```

```
list-spec = { (ALL) | LIST list of items-joints }
```

This command must be after all steady state/harmonic loadings and before the `END STEADY` command. For each harmonic frequency of TR.37.6.2 Define Harmonic Output Frequencies (on page 2633) the following will be printed:

1. Modal responses.
2. Phase angles with 1 line per selected joint containing the phase angle for each of the 6 directions of motion.
3. Displacements table with 1 line per selected joint containing the maximum displacements for each of the 6 directions of motion.
4. Velocities.
5. Accelerations.

Steady State Notes

For members, the final results at the joints will be complete but the section results will be as if the member loads were applied as statically equivalent loads at the member ends.

Steady State Examples

Example 1

```
BEGIN STEADY GROUND
STEADY GROUND FREQ 22.4 DAMP .033 ABS
GROUND MOTION X ACC .11 PHASE 0.0
GROUND MOTION Y ACC .21 PHASE 10.0
GROUND MOTION Z ACC .15 PHASE 20.0
PRINT HARMONIC DISP ALL
END
```

Example 2

```
BEGIN HARMONIC GROUND
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC GROUND DAMP .033 REL
GROUND MOTION X ACC .11 PHASE 0.0
AMPLIT A 0.10 B .21 C 0.03
GROUND MOTION Y DIS .21 PHASE 10.0
AMPLITUDE
3 5 5 4 10 6 -
35 3
GROUND MOTION Z ACC .15 PHASE 20.0
AMPLIT A 0.10 B .21 C 0.03
PRINT HARMONIC DISP ALL
END
```

Example 3

```
BEGIN STEADY FORCE
STEADY FORCE FREQ 11.2 DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
PRINT HARMONIC DISP ALL
END
```

Example 4

```
BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT X A 0.10 B .21
```

```
AMPLITUDE Y
3 5 5 4 10 6 -
35 3
AMPLIT Z A 0.10 C 0.03
PRINT HARMONIC DISP ALL
END
```

Example 5

```
BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT ALL A 0.10 B .21
PRINT HARMONIC DISP ALL
END
```

Note: For full examples, refer to the files `SVM33.std`, `SVM32.std`, `SS-beam2.std`, `SS-beam3.std`, `Exam07.std`, and `Exam14b.std` in the folder `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples \SteadyState\`.

Related Links

- *G.17.3.3 Damping Modeling* (on page 2158)
- *G.17.3.3 Damping Modeling* (on page 2158)

TR.37.6.9 Last Line of this Steady State/Harmonic Analysis

Used to end a Steady State or Harmonic Analysis.

General Format

```
END STEADY
```

TR.37.6.10 Steady State Examples

Example 1: Steady Ground Motion

```
BEGIN STEADY GROUND
STEADY GROUND FREQ 22.4 DAMP .033 ABS
GROUND MOTION X ACC .11 PHASE 0.0
GROUND MOTION Y ACC .21 PHASE 10.0
GROUND MOTION Z ACC .15 PHASE 20.0
PRINT HARMONIC DISP ALL
END
```

Example 2: Harmonic Ground Motion

```
BEGIN HARMONIC GROUND
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC GROUND DAMP .033 REL
GROUND MOTION X ACC .11 PHASE 0.0
AMPLIT A 0.10 B .21 C 0.03
GROUND MOTION Y DIS .21 PHASE 10.0
AMPLITUDE
3 5 5 4 10 6 -
35 3
GROUND MOTION Z ACC .15 PHASE 20.0
AMPLIT A 0.10 B .21 C 0.03
PRINT HARMONIC DISP ALL
END
```

Example 3: Steady Forcing Frequency

```
BEGIN STEADY FORCE
STEADY FORCE FREQ 11.2 DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
PRINT HARMONIC DISP ALL
END
```

Example 4: Harmonic Forcing Frequency with Directional Amplitudes

```
BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT X A 0.10 B .21
AMPLITUDE Y
3 5 5 4 10 6 -
35 3
AMPLIT Z A 0.10 C 0.03
```

```
PRINT HARMONIC DISP ALL
END
```

Example 5: Harmonic Forcing Frequency with Same Amplitudes

```
BEGIN HARMONIC FORCE
FREQ FLO 3.5 FHI 33 NPTS 5 MODAL FLIST 4 5 10 -
17 21 30
HARMONIC FORCE DAMP .033
JOINT LOAD PHASE X 0.0 PHASE Y 10.0 PHASE Z 15.0
UNIT KIP
10 5 TO 7 BY 2 88 FX 10.0 FY 5.0
UNIT POUND
10 5 TO 7 BY 2 -
88 FX 10.0 FY 5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT ALL A 0.10 B .21
PRINT HARMONIC DISP ALL
END
```

TR.37.7 Pushover Analysis

This section describes how to define a pushover analysis in STAAD.Pro.

Note: This requires a STAAD.Pro Advanced license.

Related Links

- *Perform Pushover Analysis tab* (on page 2926)
- *A. To specify a pushover analysis* (on page 957)

TR.37.7.1 Beginning of Pushover Data

All Pushover analysis related data are to be entered after entering this command.

General Format

```
DEFINE PUSHOVER DATA
```

TR.37.7.2 Define Input

The following sections describe the input parameters used for a pushover definition.

TR.37.7.2.1 Type of Frame

FEMA recognizes only three types of frames: concentric braced frame (CBF), eccentric braced frame (EBF), and moment frame. Only CBF and fully restrained (FR) moment frame are used in the program.

General Format

```
FRAME i1
```

where:

i1 Type of frame:

1. for a Concentric Braced frame
2. for a Moment frame (default)

Related Links

- *G.17.4.2.1.1 Define Steel Moment and Braced Frames* (on page 2178)
- *M. To define general pushover data* (on page 881)

TR.37.7.2.2 Expected Yield Stress

Expected yield stress may be specified.

General Format

```
FYE f1 MEMBER { <list>  | ALL }
```

where:

f1 User-defined expected yield stress (default = 50 ksi). Refer to Section 5.3.2.3 and Tables 5-1 and 5-2 of FEMA 356 for additional information on values used.

<list> List of members associated with the user defined yield stress

Related Links

- *M. To define member specific pushover data* (on page 882)

TR.37.7.2.3 Effective Length Factor of Member

The effective length factor of any member can be defined. This value is used in finding Euler Buckling load of a member. The minimum of these values two will be taken in calculation.

General Format

```
KY f2 MEMBER { <list>  | ALL }
```

```
KZ f3 MEMBER { <list>  | ALL }
```

where:

f2 User-defined K value in local y-axis (default = 1.0)

f3 User-defined K value in local z-axis (default = 1.0)

<list> List of members associated with the user-defined effective length factor

Related Links

- *M. To define member specific pushover data* (on page 882)

TR.37.7.2.4 Consideration of Geometric Nonlinearity Effect

The following command is used to optionally consider geometric nonlinear effects.

You may specify either (or both) the `DISPTOL` or `GEOCYCLE` commands to optionally check for convergence.

General Format

```
GNONL  i2
( DISPTOL f4  )
( GEOCYCLE i3  )
```

i2 Specifies if geometric nonlinear effects are to be considered

0. = Include the effect of large displacements (Default)
1. = Ignore the effect of large displacements

f4 Convergence displacement tolerance for convergence of Geometric Nonlinearity, in current units of length. The default value is the maximum span of the structure divided by 120.

See Note a below.

i3 Number of iterations to be performed for convergence of Geometric Nonlinearity. Default value is one.

See Note b below.

Notes

a. he member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained.

Refer `SET DISPLACEMENT` command in TR.5 Set Command Specification (on page 2206) for additional details. If any displacement on any iteration exceeds the `DISPTOL` limit, the solution is diverging and is terminated.

b. If the number of nonlinear iterations exceeds `GEOCYCLE` limit the analysis is terminated. Please refer `NONLINEAR ANALYSIS` in STAAD for more details. However, maximum allowable iteration is 50. If the number of iterations exceeds 50 the analysis is terminated.

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.2.5 KG Matrix Iteration

Optional control to limit the number of iterations performed on the geometric stiffness, KG, matrix.

General Format

```
IKGITER i4
```

where:

i4 The number of iterations to be performed for KG matrix iterations. If no value is specified, the program will assume the value of TR.37.7.2.4 Consideration of Geometric Nonlinearity Effect (on page 2645).

If the value is one (1), the program will continue iteration until the solution converges. However, maximum allowable iteration is 50. If the number of iterations exceeds 50, the analysis is terminated.

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.2.6 Maximum number of Analysis cycle

Used to specify a maximum number of cycles to be considered during the strain hardening stage for the load increment stage.

Whenever there is load increment on the structure, a new analysis cycle starts. This consists of one analysis cycle for gravity loading and the rest consists of the sum total of number of cycles required in the linear stage plus the number of cycles required to pass the structure from the linear to the nonlinear and strain hardening stage plus the number of cycles required in the strength degraded stage. The strength degraded stage consists of both analysis cycles during the load increment and load decrement stages. You can specify a lower number of analysis cycles to be considered during the strain hardening stage for the load increment stage only, which will be considered after pushover analysis enters into the nonlinear stage.

Note: The sum total of number of cycles specified by user plus number of analysis cycles required in load decrement stage plus number of cycles required in linear stage including analysis cycle for gravity loading will be limited to 10001 cycles.

General Format

```
MAXCYCLE i5
```

where:

i5 Number of analysis cycles.

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.2.7 Print Output Result

Only the final state is saved for nonlinear static analysis. You may instruct the program to print analysis result (joint displacements, member end forces, and support reactions) for the final state of nonlinear static analysis in the output file.

General Format

```
PRINT RESULT ( OUTPUT f5  )
```

where:

f5 Optionally used to limit the output to displacements, member end forces, or reactions.

1. Joint displacements only
2. Member end forces only

3. Support reactions only

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.2.8 Save Output Results for Multiple Steps

By default the intermediate analysis results are saved for positive increments only at 0.1-inch incremental value of displacement at roof or at control joint. However, you instruct the program to save results at specific intermediate steps. This will create binary post-processing files that include load step results at user-defined points.

General Format

```
SAVE LOADSTEP RESULT { DISP | BSHEAR } f6
```

f6 If `DISP` is defined, this is the incremental value of displacement at roof or at control joint in current units. If no value is given 0.1 inch is taken as default.

If `BSHEAR` is defined, this is the incremental value of base shear in current units. If no value is given, 5.0 kip is taken as default

If the value of base shear defined is B and incremental value of `BSHEAR` is f, the number of intermediate results that can be saved in B / f. The maximum allowable value is 500. Similarly, if the value of allowable displacement at control joint is D and incremental value of `DISP` is n, the number of intermediate results that can be saved in D / n. The maximum allowable value is 500.

Besides saving user-defined intermediate results, the other cases where intermediate results will be saved are as follows:

- A frame hinge is formed
- A frame hinge is trying to unload
- A frame hinge has failed

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.3 Define Loading Pattern

The pattern of the push load distribution on the nodes of the structure to be entered.

General Format

```
LOADING PATTERN i1
```

where:

i1 Method of loading pattern used:

0. = STAAD calculates internally the push load based on the specific gravity load and the first modal displacement in the direction of push load (Default)
1. = user-defined push load pattern

Note: You can define the Push load pattern explicitly when the `LOADING PATTERN` command value is one. The external push load is defined as primary load case. See TR.37.7.8 Pushover Loading Input (on page 2653).

Related Links

- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2179)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2177)
- *M. To add a pushover loading* (on page 884)

TR.37.7.3.1 Program Defined Push Load Distribution Pattern

If `LOADING PATTERN` is 0 (i.e., STAAD.Pro internally calculates the lateral push load), you are required to define following two inputs.

For all analyses, at least two vertical distributions of lateral load shall be applied. One pattern shall be selected from each of the following two groups. Therefore, two different input files are to be generated. One analysis is to be performed by selecting any one method from Group 1. Another separate analysis is to be performed by selecting Method 3 from Group 2.

Group 1

Method 1 A vertical distribution proportional to the value of C_{vx} given in Equation (3-12) of FEMA 356 : 2000 is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

Method 2 A vertical distribution proportional to the shape of the fundamental mode in the direction under consideration is performed. Use of this distribution shall be permitted only when more than 75% of the total mass participates in the fundamental mode in the direction under consideration, and the uniform distribution is also used.

Group 2

Method 3 A vertical distribution is performed consisting of lateral forces at each level proportional to the total mass at each level.

General Format

```
VDB i2
```

i2 Method used for load distribution pattern:

1. Method 1
2. Method 2
3. Method 3

Related Links

- *M. To add a pushover loading* (on page 884)

TR.37.7.3.2 Total Base Shear to be Distributed

Optionally used to define base shear that will be distributed vertically along the height of the structure at each floor level.

Note: If base shear to be distributed is not defined, the program distributes 10 percent of gravity loading as lateral load.

General Format

```
DISTRIBUTE BASE SHEAR { X | Z } f1
```

where:

f1 Total base shear to be distributed in current units of force.

Related Links

- *M. To add a pushover loading* (on page 884)

TR.37.7.3.3 Number of Push Load Steps

This command is used to specify number of steps for the push load.

The lateral load at each floor divided by the number of load steps gives the push load increment at that floor level.

General Format

```
LDSTEP i3
```

i3 Number of load steps. Default is 100.

Related Links

- *M. To add a pushover loading* (on page 884)

TR.37.7.4 Define Solution Control

One of the two following methods must be specified to define the limit of the pushover analysis.

Note: Both method may be specified in the same input file. Upon exceeding either of the limits, the analysis will stop.

Related Links

- *G.17.4.2.1.6 Define Pushover Analysis Solution Control* (on page 2182)
- *M. To define solution control* (on page 884)

TR.37.7.4.1 Push Up to Defined Base Shear

The pushover analysis will continue until the cumulative base shear is less than or equal to the base shear specified by this command or the structure has additional strength.

General Format

```
BASE SHEAR { X | Z } DEFINED f1
```

f1 Explicitly defined base shear in current units of force.

TR.37.7.4.2 Push Up to Defined Displacement at Control Joint

The pushover analysis will continue until the displacement at the specified joint at the specified direction exceeds specified displacement.

General Format

```
DISP { X | Z } f2 JOINT i1
```

f2 Explicitly defined joint displacement value in current units of length.

i1 Node number of control joint.

TR.37.7.5 Define Hinge Property

If the automatic hinge calculation per FEMA 356 is not used, then the hinge properties must be defined and these hinges must be assigned to members.

TR.37.7.5.1 User-Defined Hinge Property

Several hinge property may be specified with specific type identifier. User may later refer this type identifier while assigning specific hinge properties to members (See "TR.37.7.9 Validation of Commands other than Input Parameters (on page 2654)"). If hinge property is not specified, STAAD assumes the FEMA specified hinge properties.

To specify uncoupled moment hinge property of the member following input is required.

General Format

```
HINGE PROPERTY MOMENT
TYPE n1
A xa1 ya1 B xb1 yb1 C xc1 yc1 D xd1 yd1 E xe1 ye1 YM f11 YR f21
IO f31 LS f41 CP f51
...
TYPE ni
A xai yai B xbi ybi C xci yci D xdi ydi E xei yei YM f1i YR f2i
IO f3i LS f4i CP f5i
```

n1,..., ni	Type identifier (Hinge property number)
A, B, C, D, E	Points on the load deformation curve at points A, B, C, D, and E as specified in FEMA
xa1 ya1 ... xe1 ye1,..., xai yai ... xei yei	Coordinates of A, B, C, D, E respectively

xa1, ..., xe1, ..., xai, ..., xei	Deformation Ratio, θ/θ_y
ya1 ..., ye1, ..., yai, ...,yei	Moment Ratio, Q/Q_y
f11,...,f1i	Yield moment in current force units
f21,...,f2i	Yield rotation in radians
f31,...,f3i	Immediate occupancy in deformation ratio
f41,...,f4i	Life safety in deformation ratio
f51,...,f5i	Collapse prevention in deformation ratio

The coordinates of point A should be always (0, 0) and that of point B is (1.0, 1.0). Both the coordinates of point C must be greater than 1.0. The X coordinate of point D must be same as that of point C and Y coordinate must be less than 1.0. The X coordinate of point E must be greater than that of point D and Y coordinate should be same as that of point D.

Note: The hinge property will be applied to hinge formed due to moment about local z-axis for beam. For column, hinge formation will be considered either about local y-axis or about local z-axis depending upon whichever moment is dominating and also the orientation of the column with respect to the push load direction. The hinge property will be applied to the hinge formed due to this moment.

See "TR.37.7.5.2 Assignment of Hinge Property to the Members (on page 2651)" for example input.

Related Links

- *G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria* (on page 2180)
- *G.17.4.1.4 Types of Nonlinearity* (on page 2173)
- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *M. To manually define and assign hinges* (on page 882)

TR.37.7.5.2 Assignment of Hinge Property to the Members

This command is used to define the method by which the plastic hinges are formed. If no assignment of hinge property is done, the program will assign built-in FEMA 356 hinge property to the members.

```
HINGE { TYPE n1  | FEMA | IGNORE } { MEMBER <list>  | ALL }
```

n1	Hinge type identifier
<list>	List of members associated the current hinge command

Notes

a. Once `HINGE PROPERTIES` are specified (See "TR.37.7.5.1 User-Defined Hinge Property (on page 2650)"), you can assign these to the members using the `HINGE TYPE` command.

Note: Hinge properties must be defined prior to assigning them to members in the input file.

b. If a user-defined hinge property (i.e., the `HINGE TYPE` command) is assigned to some members, the program will consider hinge formation in these members only. The `HINGE FEMA` hinge property must be assigned to other members. Thus, you must take care to assign hinge property to all members if hinge formation is to be considered in all members.

In this case, the `HINGE FEMA` command must follow after the `HINGE TYPE` command in the input.

c. The `HINGE IGNORE` command can be used to ignore hinge formation in some members.

Note: The ALL option should not be used with HINGE IGNORE as this will prevent a successful pushover analysis.

Example

```
DEFINE PUSHOVER DATA
HINGE PROPERTY MOMENT
TYPE 1
A 0.0 0.0 B 1.0 1.0 C 10.0 1.2 D 10.0 0.6 E 12.0 0.6 YM 3325 YR 0.00899
IO 1.001 LS 6.0 CP 8.0
…
HINGE IGNORE MEMBER 16
HINGE TYPE 1 MEMBER 1 3 17
HINGE FEMA MEMBER 2 4 to 15 18 to 20
…
END PUSHOVER DATA
```

Related Links

- *G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria* (on page 2180)
- *G.17.4.1.4 Types of Nonlinearity* (on page 2173)
- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *M. To manually define and assign hinges* (on page 882)

TR.37.7.6 Define Spectral Parameters

The parameters of this section are used to construct the response spectrum according to FEMA 356:2000.

General Format

```
SPECTRUM PARAMETERS
DAMPING f1  ( f2 f3 f4  )
SC f5
SS f6
S1 f7
```

f1 Percentage of critical damping for the 1st response spectrum (Default 5%)

f2 Optional percentage of critical damping for the 2nd response spectrum (Default is zero)

f3 Optional percentage of critical damping for the 3rd response spectrum (Default is zero)

f4 Optional percentage of critical damping for the 4th response spectrum (Default is zero)

f5 An integer corresponding to the site class as described in the following table. The default value is 4 (i.e., site class D):

Site Class	Value of f_5
A	1
B	2

Site Class	Value of f_5
C	3
D	4
E	5

f6 Spectral acceleration at short-period, S_S

f7 Spectral acceleration at one-second period, S_1

Related Links

- *G.17.4.2.1.7 Define Input for Demand Spectrum* (on page 2182)
- *M. To define pushover spectral data* (on page 883)

TR.37.7.7 End Pushover Data

All Pushover data are to be entered before this command.

General Format

```
END PUSHOVER DATA
```

Tip: This command is added automatically when the `DEFINE PUSHOVER DATA` command is added through the **Create New Definitions / Load Cases / Load Items** dialog in the graphical environment.

Related Links

- *M. To define general pushover data* (on page 881)

TR.37.7.8 Pushover Loading Input

Only two types of loading are accepted in pushover analysis.

Refer to TR.32 Loading Specifications (on page 2461) for additional information on primary load types.

Gravity Load

Syntax for gravity loading is as follows:

```
LOAD i1 LOADTYPE GRAVITY ( TITLE any_load_title  )
...
```

i1	gravity loading number

Push Load

Syntax for push loading (user-defined external incremental push loading pattern on the structure) is as follows:

```
LOAD i2 LOADTYPE PUSH  ( TITLE any_load_title  )
...
```

`i2` user-defined push loading number

Note: For user-defined incremental push load, the loading has to be applied only in global X or Z directions. Combinations of X and Z directions is not accepted. Also, the lateral load in either global X or Z directions is to be applied in form of joint loads.

Related Links

- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2179)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2177)
- *M. To add a pushover loading* (on page 884)

TR.37.7.9 Validation of Commands other than Input Parameters

1. The `LOADTYPE` is required and must be either `GRAVITY` or `PUSH`. See "TR.37.7.8 Pushover Loading Input (on page 2653)"
2. There can be more than one gravity load case. STAAD internally combines all gravity load cases into one. Analysis uses this combined gravity loading only.
3. A mass vector for eigen solution is formed from this combined gravity loading. There is no need to provide mass modeling.
4. Only one push load case (either in the form of user defined external push load or generated by the program) is allowed. The direction of loading is either global X or global Z. Combination of these directions are not allowed. For user defined incremental push load pattern, only `JOINT LOAD`s in `FX` or `FZ` directions may be specified.

Limitations on Pushover Analysis Input

The following inputs are not allowed in Pushover Analysis:

- `PERFORM PUSHOVER ANALYSIS` cannot be repeated more than once.
- `PERFORM PUSHOVER ANALYSIS PRINT` is not accepted.
- `CHANGE` command is not accepted.
- `LOAD COMBINATION` command is not accepted.
- `SET Z UP` command is not accepted.
- `SET RESTART 1` command will not work (i.e., *<filename>* `.L17` file will not be created).
- `SURFACE` element is not accepted.
- Material concrete is not considered.
- Non-prismatic section is not considered.
- Curved beam is not considered.
- Cable member is not considered.
- Plate and solid elements is not considered.
- Beta angle other than 0 and 90 degree is not accepted.

TR.37.8 Geometric Nonlinear Analysis

Note: This requires a STAAD.Pro Advanced license.

General Format

```
PERFORM NONLINEAR ANALYSIS (ARC f1) (ITERATION i1) (TOLERANCE f2) (STEPS i2) (REBUILD
i3) (KG (i4)) (JOINT_TARGET  i5 (i6) DISPL_TARGET f3) (PRINT print-specs)
```

The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.

The following table describes the parameters available for nonlinear analysis:

Table 282: Geometric Nonlinear Analysis parameters

Parameter	Value	Default Value	Description
ARC	f_1	0.0	Displacement control. Value is the absolute displacement limit for the first analysis step. If max. displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for STEPS. Value should be in current length units. ARC = 0 indicates no displacement control.
ITERATION	i_1	100	Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.
TOLERANCE	f_2	0.0001 inch	For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.
STEPS	i_2	1	Number of load steps. Load is applied in stages if entered. One means that all of the load is applied in the first step.
REBUILD	i_3	1	Frequency of rebuilds of the tangent K matrix per load step & iteration.
KG	i_4	0	This parameter controls whether the geometric stiffness, KG, is added to the stiffness matrix, K. • `KG` or `KG 1` Use K+KG for the stiffness matrix. (Default) • `KG 0` (or KG omitted) Do not use KG.
JOINT_TARGET	i_5	none	Joint being monitored in a displacement target analysis.

Parameter	Value	Default Value	Description
	i_6	1	Global degree of freedom (1 through 6): **1.** Global X **2.** Global Y **3.** Global Z **4.** Moment about Global X **5.** Moment about Global Y **6.** Moment about Global Z
DISPL_TARGET	f_3	none	Displacement target value in current length units.
print-spec		none	Standard STAAD analysis print options. See TR.37.1 Linear Elastic Analysis (on page 2620) for details.

If a target displacement is used, then *both* the `JOINT_TARGET` and `DISPL_TARGET` must be defined. If the degree of freedom is not specified, then the Global X direction is assumed.

If Joint target and Displacement target are entered and the `STEPS` parameter is greater than two; then the analysis will proceed step by step until the targeted joint degree of freedom has displaced `DISPL_TARGET` length units or more.

Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.

Caution: The deprecated `NONLINEAR n ANALYSIS` command will adopt the new procedure (unless a `SET NONLINEAR OLD` command is entered before the joint coordinates to invoke the old procedure for backward compatibility). This uses *n* iterations. It is strongly recommended to use the new procedure.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Temperature loads are *not* supported for geometric nonlinear analysis.

Example

```
PERFORM NONLINEAR ANALYSIS ARC 0.01 ITER 200 STEPS 100 TOL 0.00001 -
REBUILD 1 KG joint 11 2 DISPL_TARGET 0.5
```

Related Links

- *G.17.2.3 Static Geometrically Nonlinear Analysis* (on page 2147)
- *A. To specify a nonlinear analysis* (on page 954)
- *A. To specify a nonlinear analysis* (on page 954)
- *A. To specify a nonlinear analysis* (on page 954)
- *G.17.2.3 Static Geometrically Nonlinear Analysis* (on page 2147)

TR.37.9 Imperfection Analysis

This performs a modified linear elastic analysis using Member Imperfection Specifications (See TR.26.6 Member Imperfection Information (on page 2313)) defined on beam and column members.

An Imperfection analysis will reflect the secondary effects only if the camber and/or drift is specified in a `DEFINE IMPERFECTIONS` specification (TR.26.6 Member Imperfection Information (on page 2313)). For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

General Format

```
PERFORM IMPERFECTION ANALYSIS (PRINT { LOAD DATA | STATICS CHECK | STATICS LOAD | BOTH
| ALL } )
```

See TR.37.1 Linear Elastic Analysis (on page 2620) for details.

Without one of these analysis commands, no analysis will be performed. These `ANALYSIS` commands can be repeated if multiple analyses are needed at different phases.

Note: Due to the mechanisms used to include control/dependent systems, if the reactions on control nodes are not included in a statics check then an out of balance report may result. This can be avoided by adding a short stiff member from a control node to the support.

Related Links

- *G.17.2.4 Imperfection Analysis* (on page 2148)
- *A. To specify an imperfection analysis* (on page 956)
- *G.17.2.4 Imperfection Analysis* (on page 2148)

TR.37.10 Floor Spectrum Command

This command is used to specify the calculation of floor spectra from time history acceleration results. The Floor Response Spectrum command must immediately follow an analysis command associated with the time history load case. The acceleration used may be either the absolute acceleration or the relative to ground acceleration.

Note: This data should only be entered if there is a time history case being solved.

Note: This requires a STAAD.Pro Advanced license.

General Format

First line of this Floor Spectrum Data.

```
GENERATE FLOOR SPECTRUM
```

Specify Floor Groups

Each new floor definition starts with the next command:

```
BEGIN  FLOOR  DIRECTION   { GX | GY | GZ }* {  TITLE  }
```

GX, GY, and GZ specify up to 3 global directions for which acceleration vs frequency spectra will be generated for this floor. Optionally enter a Title/Description (up to 50 characters) for this floor that will be displayed on the graphs in post processing.

The next one or more lines will identify the floors which will have spectrum curves generated either by referencing a NODE GROUP (see TR.16.1 Listing of Entities by Specifying Groups (on page 2235)) or explicitly listing the joints that constitute the floor. For multiple groups, enter each on a separate line.

```
{ _jointgroup | jointlist }
```

Enter as many lines as necessary to specify all of the groups needed to define this floor.

If more floors are to be defined, then repeat the `BEGIN FLOOR DIRECTION` command followed by the *_jointgroup_* name data. Enter as many floors as desired.

After the last floor definition, enter the following parameters that will be used in all of the Floor Spectrum calculations.

```
OPTIONS ( { FLOW f1 | FHIGH f2 | FDELTA f3 | DAMP f4 (… fn)| RELATIVE } ) print-options
```

This command may be continued to the next line by ending the line with a hyphen.

Where:

FLOW *f1*	Lowest frequency to be in the calculated spectrum. The value for FLOW should be at least 0.01 Hz.
FHIGH *f2*	Highest frequency to be in the calculated spectrum.
FDELTA *f3*	The spectrum will be calculated at *f3* intervals from the `FLOW` value (*f1*) to `FHIGH` (*f2*).
DAMP *f4* ... *fn*	Up to 10 damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios.
RELATIVE	If there is ground motion defined and you want the spectra to be based on the relative acceleration of the floor to the ground acceleration, then enter the relative parameter. Default is absolute.

The *print-options* are:

THPRINT *i1*	Option to print the time history acceleration being used in each spectrum calculation. 0 - No print 2 - Print the time history accelerations.
SPRINT	Include this parameter to print the calculated spectrum.

Tip: Omitting these options is recommended for most analysis runs.

Last line of the floor spectrum data:

```
END FLOOR SPECTRUM
```

Example 1

This partial input utilizes direct input data from a forcing function.

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0 -20 0.5 100 1 200 1.5 500 2 800 2.5 500 3 70 16 0
ARRIVAL TIME
0
DAMPING 0.075
*
LOAD 1 LOADTYPE SEISMIC TITLE Time History case
* Mass model required
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
JOINT LOAD
1 TO 6 FX 62.223 FY 62.223 FZ 62.223
* Time loads
TIME LOAD
2 FX 1 1
PERFORM ANALYSIS
GENERATE FLOOR SPECTRUM
BEGIN FLOOR DIRECTION GX GZ Ground Motion
_FL1
_FL17
BEGIN FLOOR DIRECTION GX GZ Floor 18 A/C Unit 36
_FL18
OPTIONS FLO 0.5 FHI 35.0 FDEL 0.1 –
DAMP 0.03 0.05 0.07
END FLOOR SPECTRUM
```

Example 2

This complete input example uses an external seismic data file.

```
STAAD SPACE INTRODUCTORY PROBLEM
START JOB INFORMATION
ENGINEER DATE 27-Oct-08
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 5 3 0; 4 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X26
2 TABLE ST W14X34
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-006
```

```
DAMP 0.03
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 PINNED
CUT OFF MODE SHAPE 30
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
START GROUP DEFINITION
JOINT
_FLOORNODES 2
_BASENODES 1
END GROUP DEFINITION
UNIT FEET KIP
LOAD 1 DEAD + LIVE
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MEMBER LOAD
2 UNI GX 1
2 UNI GY 1
2 UNI GZ 1
GROUND MOTION X 1 1 9.806000
PERFORM ANALYSIS
GENERATE FLOOR SPECTRUM
BEGIN FLOOR DIRCTION GX Nodes 2
_FLOORNODES
BEGIN FLOOR DIRCTION GX Nodes 1
_BASENODES
OPTIONS FLO 0.5 FHI 60.0 FDEL 0.1 DAMP 0.05 REL
END FLOOR SPECTRUM
FINISH
```

Note: The seismic data file included with the US and UK example sets is used here and can be found at `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Samples\Sample Models\US\EQDATA.TXT` (typical location).

Related Links

- *A. To generate a floor spectrum* (on page 959)
- *A. To generate a floor spectrum* (on page 959)

TR.38 Change Specification

This command is used to reset the stiffness matrix. Typically, this command is used when multiple analyses are required in the same run.

General Format

```
CHANGE
```

This command indicates that input, which will change the stiffness matrix, will follow. This command should only be used when an analysis has already been performed. The `CHANGE` command does or requires the following:

- sets the stiffness matrix to zero
- makes members active if they had been made inactive by a previous `INACTIVE` command
- allows the re-specification of the supports with another `SUPPORT` command that causes the old supports to be ignored. The `SUPPORT` specification must be such that the number of joint directions that are free to move (DOF or "releases") before the `CHANGE` must be *greater than or equal to* the number of "releases" after the `CHANGE`.
- the supports must be specified in the same order before and after the `CHANGE` command. To accomplish this when some cases have more supports than others do, you can enter unrestrained joints into the `SUPPORT` command list using `FIXED BUT FX FY FZ MX MY MZ`. It is best to put every joint that will be supported in any case into every `SUPPORT` list.
- `CHANGE`, if used, should be after `PERFORM ANALYSIS` and *before* the next set of `SUPPORT` or `LOADS`.
- Only active cases are solved after the `CHANGE` command.
- Analysis and `CHANGE` are required between primary cases for `PERFORM CABLE ANALYSIS`.
- Analysis and `CHANGE` are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Example

Before `CHANGE` is specified:

```
1 PINNED
2 FIXED BUT FX MY MZ
3 FIXED BUT FX MX MY MZ
```

After `CHANGE`is specified:

```
1 PINNED
2 FIXED
3 FIXED BUT FX MZ
```

The `CHANGE` command is not necessary when only member properties are revised to perform a new analysis. This is typically the case in which the user has asked for a member selection and then uses the PERFORM ANALYSIS command to reanalyze the structure based on the new member properties.

Notes

a. If new load cases are specified after the CHANGE command such as in a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option (see TR.5 Set Command Specification (on page 2206) and Example 4).

b. Multiple Analyses using the CHANGE command should not be performed if the input file contains load cases involving dynamic analysis or Moving Load Generation.

c. Section forces and moments, stress and other results for postprocessing will use the last entered data for supports and member properties regardless of what was used to compute the displacements, end forces and reactions. So beware of changing member properties and releases after a CHANGE command.

Related Links

- *G.19 Multiple Analyses* (on page 2194)
- *A. To add a change command* (on page 958)

TR.39 Load List Specification

This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. When the `PERFORM ANALYSIS` command is used, the program internally uses all load cases, regardless of `LOAD LIST` command, except after a `CHANGE` command. In these two cases, the `LOAD LIST` command allows the program to perform analysis only on those loads in the list. If the `LOAD LIST` command is never used, the program will assume all load cases to be active.

General Format

```
LOAD LIST { load-list | ALL }
```

Example

```
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 3
PRINT SUPPORT REACTIONS
CHECK CODE ALL
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions and code-checking of all members.

Notes

a. The `LOAD LIST` command may be used for multiple analyses situations when an analysis needs to be performed with a selected set of load cases only. All load cases are automatically active before a first `CHANGE` command is used.

b. After a `CHANGE` command has been used anywhere in the data, it is good practice to specify the `LOAD LIST` command after an `ANALYSIS` command and before the next command; otherwise only the last case analyzed may be used in the design.

c. Do not enter this command within the loads data (from the first Load command in an analysis set to the associated Analysis command).

d. Additionally, load cases can also be short-listed using load envelopes. This may be needed when operations such as steel design need to distinguish between service load combinations and factored load combination cases.

Related Links

- *A. To create a load list* (on page 961)

TR.40 Load Envelope

Load Envelopes are a means for clustering a set of load cases under a single moniker (number). If one or more tasks have to be performed for a set of load cases, such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc., this feature is convenient.

It is an alternative to the `LOAD LIST` command described in TR.39 Load List Specification (on page 2662).

General Format

Load envelopes are defined within a definition block. Each envelope can be assigned an optional type to specify the qualitative nature of the load or load combination cases in the envelope definition.

```
DEFINE ENVELOPE
load-list ENVELOPE i  ( TYPE { STRESS | SERVICEABILITY | COLUMN | CONNECTION |
 STRENGTH | TEMPORARY} )
...
END DEFINE ENVELOPE
```

Where:

i = the load envelope number.

Two of the load envelope types —`SERVICEABILITY` and `STRENGTH`— have specific meaning from the standpoint of steel design to certain codes. See Using Load Envelopes in Designing Steel Members for Strength and Serviceability (on page 2664). Other types are used for annotating the load envelope use in reports and post-processing.

A load case or combination may only be included in *one* load envelope. If a load case or combination is included in more than one load envelope, that case or combination will only be included in the last load envelope that includes it.

Example

The first line within the `DEFINE ENVELOPE` command means that load cases numbered 1 to 8 make up the `CONNECTION` type load envelope 1. Similarly load case 9 to 15 define the `SERVICEABILITY` type load envelope 2.

```
DEFINE ENVELOPE
1 TO 8 ENVELOPE 1 TYPE CONNECTION
9 TO 15 ENVELOPE 2 TYPE SERVICEABILITY
16 TO 28 ENVELOPE 4 TYPE STRESS
END DEFINE ENVELOPE
```

Load Lists for Envelopes

For operations and calculations which are going to be based on the load cases contained in the envelopes, the command

```
LOAD LIST ENVELOPE load-env-list
```

must be specified *prior* to those commands.

For example, to print out the support reactions corresponding to load envelope 1, the following commands should be provided in the input file

```
LOAD LIST ENV 1
PRINT SUPPORT REACTIONS
```

Using Load Envelopes in Designing Steel Members for Strength and Serviceability

Most design codes require two types of checks to be performed

a. deflection
b. strength (capacity)

In codes that are based on the strength design method —like the AISC 360-16 / 10 / 05, AISI 2016, IS 800 2007 / 1984, Canadian S16-14 / S16-09 / S16-01, NZS3404 1997, and Eurocode 3— deflection checks are normally required to be done on the serviceability load cases and strength checks on a different set of load cases which are the factored load cases.

Hence, it is necessary to convey to the program which load cases are to be considered for the deflection checks, and which ones for the strength related checks.

Using Load Envelopes, you can convey to the program this information using the type keywords `SERVICEABILITY` and `STRENGTH`. Additionally, prior to the design command, you must also specify the command `LOAD LIST ENV` *`load-env-list`* .

```
DEFINE ENVELOPE
101 TO 110 ENVELOPE 1 TYPE SERVICEABILITY
201 to 225 ENVELOPE 2 TYPE STRENGTH
END DEFINE ENVELOPE

LOAD LIST ENVELOPE 1 2

PARAMETER 1
CODE AISC UNIFIED
METHOD LRFD
FYLD 46 MEMB 27 37 67 TO 89
DFF 240 ALL
DJ1 …
DJ2 …
…
…
CHECK CODE ALL
```

Note that if only one type of check is required —say for example, strength only— then the separation of load cases into the different categories is not needed which means these envelopes do not have to be created. In that event, the simple `LOAD LIST` command described in TR.39 Load List Specification (on page 2662) will suffice instead of `LOAD LIST ENV` *`load-env-list`* .

Related Links

- *M. To create a load envelope* (on page 901)
- *M. To create a load envelope* (on page 901)
- *G.18 Member End Forces* (on page 2187)
- *G.18.1.1 Member Forces at Intermediate Sections* (on page 2192)
- *G.18.1.3 Member Stresses at Specified Sections* (on page 2193)

TR.41 Section Specification

This command is used to specify sections along the length of frame member for which forces and moments are required.

This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

General Format

```
SECTION f1( f2) ( f3) { MEMBER memb-list | (ALL) }
```

Where:

f1, f2 , f3 Section (in terms of the fraction of the member length) provided for the members. The maximum number of sections is three, and only values between 0.0 and 1.0 will be considered. In other words, no more than three intermediate sections are permissible per `SECTION` command.

Example

```
SECTION 0.17 0.48 0.72 MEMB 1 2
SECTION 0.25 0.75 MEMB 3 TO 7
SECTION 0.6 MEMB 8
```

In the above example, first, section locations of 0.17, 0.48, and 0.72 are set for members 1 and 2. In the next `SECTION` command, sections 0.25 and 0.75 are set for members 3 to 7. In the third `SECTION` command, member 8 has its section specified at 0.6. The remainder of the members will have no sections provided for them. As mentioned earlier, no more than three intermediate sections are allowed per `SECTION` command. However, if more than three intermediate sections are desired, they can be examined by repeating the `SECTION` command after completing the required calculations. The following example will clarify.

Example

```
SECTION  0.2  0.4  0.5  ALL
PRINT  SECTION  FORCES
SECTION  0.6  0.75  0.9  ALL
PRINT  SECTION  FORCES
```

In this example, forces at three intermediate sections (namely 0.2, 0.4 and 0.5) are printed. Then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the ability to obtain section forces at more than three intermediate sections.

Notes

a. The `SECTION` command just specifies the sections. Use the `PRINT SECTION FORCES` command after this command to print out the forces and moments at the specified sections.
b. This is a secondary analysis command. The analysis must be performed before this command may be used.
c. To obtain values at member ends (`START` and `END`), use the `PRINT MEMBER FORCES` command.
d. If this command is specified before a steel design operation *and* if the `BEAM` parameter is set to zero, the section locations specified by this command will also be designed for, in addition to the `BEAM` ends. This can be used to specify three locations along the beam for design when the default sections are not sufficient. See D1.A.6 Design Parameters (on page 1100).

Related Links

- *G.18 Member End Forces* (on page 2187)
- *G.18.1.1 Member Forces at Intermediate Sections* (on page 2192)
- *G.18.1.3 Member Stresses at Specified Sections* (on page 2193)
- *G.18.1 Secondary Analysis* (on page 2192)
- *G.18.1.2 Member Displacements at Intermediate Sections* (on page 2193)

TR.42 Print Specifications

This command is used to direct the program to print various modeling information and analysis results. STAAD.Pro offers a number of versatile print commands that can be used to customize the output.

General Format

Data Related Print Commands

```
PRINT { PROBLEM STATISTICS | JOINT COORDINATES | MEMBER PROPERTIES | ELEMENT
INFORMATION (SOLID) | MATERIAL PROPERTIES | SUPPORT INFORMATION | ALL } { (ALL) | LIST
item/joint/member-list }
```

Print Location of CG

```
PRINT CG (_group_name)
```

Print Analysis Results

General:

```
PRINT { (JOINT) DISPLACEMENTS | ANALYSIS RESULTS | MODE SHAPES } list-spec
```

Members:

```
PRINT { MEMBER FORCES (GLOBAL) | PMEMBER FORCES | MEMBER SECTION FORCES | MEMBER
STRESSES } list-spec
```

Elements:

```
PRINT { ELEMENT (JOINT) STRESSES (AT f1 f2) | ELEMENT FORCES | ELEMENT (JOINT)
STRESSES SOLID } list-spec
```

where:

```
list-spec = { (ALL) | LIST joint/member/elements-list }
```

Print Support Reactions

```
PRINT SUPPORT REACTIONS
```

Print Story Drifts or Stiffness

```
PRINT STORY DRIFT (f3)
```

```
PRINT STORY STIFFNESS
```

Print Location of Center of Rigity at each Floor

```
PRINT DIAPHRAGM CR
```

Print the deflected (sag) shape of a cable after an advanced cable analysis:

```
PRINT CABLE SAG
```

Description

The list of items is not applicable for `PRINT ANALYSIS RESULTS` and `PRINT MODE SHAPES` commands.

The `PRINT JOINT COORDINATES` command prints all interpreted coordinates of joints.

The `PRINT MEMBER INFORMATION` command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1 = released, 0 = not released).

The `PRINT ELEMENT INFORMATION` command prints all incident joints, element thicknesses, and Poisson ratios for Plate/Shell elements. The `PRINT ELEMENT INFORMATION SOLID` command prints similar information for Solid elements.

The `PRINT MEMBER PROPERTIES` command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on FPS or METRIC) regardless of the unit specified in UNIT command.

The following designation is used for member property names:

AX Cross section area

AY Area used to adjust shear/bending stiffness in local Y axis to account for pure shear in addition to the classical bending stiffness

AZ Area used to adjust shear/bending stiffness in local Z axis to account for pure shear in addition to the classical bending stiffness.

IZ Moment of Inertia about the local Z-axis

IY Moment of Inertia about the local Y-axis

IX Torsional constant

SY Smallest section modulus about the local Y-axis

SZ Smallest section modulus about the local Z-axis

The `PRINT MATERIAL PROPERTIES` command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members. This command is available for members only. G may be listed as zero if command is before load cases and Poisson ratio was entered but G was not entered.

The `PRINT SUPPORT INFORMATION` command prints all support information regarding their fixity, releases and spring constant values, if any. The `LIST` option is not available for this command.

The `PRINT ALL` command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

The `PRINT CG` command prints out the coordinates of the center of gravity and the total weight of the structure or of a single group of member/elements. If the CG of a portion of the structure is desired, the members and elements of that portion must be assigned using a group name (see TR.16 Entities as Single Objects (on page 2235) for details on using group-names). Only the selfweight of the structure is used to calculate the C.G. User defined joint loads, member loads etc. are not considered in the calculation of C.G.

The `PRINT (JOINT) DISPLACEMENTS` command prints joint displacements in a tabulated form. The displacements for all six directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on FPS or METRIC unit) regardless of the unit specified in `UNIT` command.

The `PRINT (MEMBER) FORCES` command prints member forces (i.e., Axial force (AXIAL), Shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), Torsional Moment (TORSION), Moments about local Y and Z axes (MOM-Y and MOM-Z)) in a tabulated form for the listed members, for all specified load cases. The `GLOBAL` option will output forces in the global coordinate system rather than the member local coordinate system for each member.

The `PRINT SUPPORT REACTIONS` command prints global support reactions in a tabulated form, by support, for all specified load cases. Use LIST option for selected joints.

The `PRINT ANALYSIS RESULTS` command is equivalent to the above three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

The `PRINT (MEMBER) SECTION FORCES` command prints axial force, shear forces, & bending moments at the intermediate sections specified with a previously input TR.41 Section Specification (on page 2665) The printing is done in a tabulated form for all specified cases for the first requested member, then for the next member, etc.

Note: An asterisk following a critical load case number within `PRINT JOINT DISPLACEMENT`, `PRINT MEMBER FORCES`, `PRINT SECTION FORCES`, `PRINT MEMBER STRESS`, or `PRINT SUPPORT REACTION` outputs indicates that this load case is a generated load combination. See TR.35 Load Combination Specification (on page 2616) for additional information.

The `PRINT (MEMBER) STRESSES` command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e., axial force over the area), bending-y (i.e., moment-y over section modulus in local y-axis), bending-z (i.e., moment-z over section modulus in local z-axis), shear stresses in both local y and z directions (shear flow, q, over the shear area), and combined (absolute combination of axial, bending-y and bending-z) stresses.

- For `PRISMATIC` sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be used.
- For `TAPERED` sections, the values of AY and AZ are those for the location where the stress is printed. Hence at the location 0.0, the AY and AZ are based on the dimensions of the member at the start node.

The `PRINT ELEMENT STRESSES` command must be used to print plate stresses (SX, SY, SXY, SQX, SQY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) for plate/shell elements. Typically, the stresses and moments per unit width at the centroid will be printed. The Von Mises stresses (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.

The variables that appear in the output are the following. Refer to G.5.1 Plate and Shell Elements (on page 2099) for more information regarding these variables.

SQX	Shear stress on the local X face in the Z direction
SQY	Shear stress on the local Y face in the Z direction
MX	Moment per unit width about the local X face
MY	Moment per unit width about the local Y face
MXY	Torsional Moment per unit width in the local X-Y plane
SX	Axial stress in the local X direction
SY	Axial stress in the local Y direction
SXY	Shear stress in the local XY plane
VONT	Von Mises stress on the top surface of the element
VONB	Von Mises stress on the bottom surface of the element
TrescaT	Tresca stress on the top surface of the element
TrescaB	Tresca stress on the bottom surface of the element
SMAX	Maximum in-plane Principal stress
SMIN	Minimum in-plane Principal stress
TMAX	Maximum in-plane Shear stress
ANGLE	Angle which determines direction of maximum principal stress with respect to local X axis

Note: If the `JOINT` option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

The `AT` option may be used to print element forces at any specified point within the element. The AT option must be accompanied by f_1 and f_2. f_1 and f_2 are local X and Y coordinates (in current units) of the point where the stresses and moments are required. For detailed description of the local coordinate system of the elements, refer to G.5 Finite Element Information (on page 2099).

The `PRINT ELEMENT FORCES` command enables printing of plate "corner forces" *[$Fp = Kp \bullet Dp$]* in global axis directions.

The `PRINT ELEMENT (JOINT) STRESS SOLID` command enables printing of stresses at the center of the SOLID elements. The variables that appear in the output are the following.

Normal Stresses	SXX, SYY and SZZ
Shear Stresses	SXY, SYZ and SZX
Principal Stresses	S1, S2 and S3.
Von Mises Stresses	SE
Direction cosines	Six direction cosines are printed following the expression DC, corresponding to the first two principal stress directions.

Note: The `JOINT` option will print out the stresses at the nodes of the solid elements.

The `PRINT MODE SHAPES` command prints the relative joint motions of each of the modes that were calculated. The maximum motion is arbitrary and has no significance. Dynamic analysis will scale and combine the mode shapes to achieve the final dynamic results.

Example

```
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESS
PRINT ELEMENT STRESS AT 0.5 0.5 LIST 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124
```

Example

Printing the Center of Gravity (CG)

```
PRINT CG
PRINT CG _RAFTERBEAMS
PRINT CG _RIDGEBEAMS
```

Notes

1. The output generated by these commands is based on the current unit system. The user may wish to verify the current unit system and change it if necessary.
2. Results may be printed for all joints/members/elements or based on a specified list.

Printing the Story Drift and Stiffness

The `PRINT STORY DRIFT` command may be used to obtain a print-out of the average lateral displacement of all joints at each horizontal level along the height of the structure.

The procedure used in STAAD.Pro for calculating story drift is independent of any code. For example, the story drift determination as explained in section 12.8.6 of the ASCE 7-05 code is *not* implemented in STAAD.Pro.

The method implemented in STAAD.Pro involves:

a. Find all the distinct Y coordinates in the model. Those are what STAAD.Pro calls as stories.
b. For each of those distinct stories, find all the nodes at that story elevation.
c. For each story, find the average displacement along the horizontal directions (X and Z) by adding up corresponding displacement for all the nodes at that story, and dividing by the number of nodes for that story. Thus, even if there is only a single node representing a story, a drift is calculated for that story too.

In STAAD.Pro if `PRINT STORY DRIFT` command is issued, the program prints the average of horizontal displacements of all the joints present at the particular floor level.

However, to check inter-story drift, the following commands needs to be issued after the `PERFORM ANALYSIS` command.

```
LOAD LIST i1
```

```
PRINT STORY DRIFT f3
```

Where:

i_1 = the primary load number for which inter-story drift check is required
f_3 = The allowable drift factor, as per the code provision

The program will calculate the relative horizontal displacement between two adjacent floors. This calculated value is checked against the allowable limit. The result is reported as either "PASS" or "FAIL" in the output.

Note:

There is only one exception to this format. For IS 1893: 2002 static seismic load case, even if this factor is not provided the program internally checks if the loading is IS 1893 static seismic loading or not. If yes, it automatically calculates the inter-story drift and checks against the code provisions.

For dynamic IS 1893: 2002 response spectrum analysis the above format for inter-story drift check does not hold true. The reason is that in response spectrum analysis the joint displacements represents the maximum magnitude of the response quantity that is likely to occur during seismic loading. Any response quantity like story drift should be calculated from actual displacements of each mode considered during analysis. The inter-story drift from each mode is combined using modal combination to get the maximum magnitude of this response quantity. In order to compute story drift for IS 1893 response spectrum the load case command format described in TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534) must be used..

The `PRINT STORY STIFFNESS` command may be used to include the calculated lateral stiffness of each story used in determining the drift. In STAAD.Pro lateral stiffness is calculated only when the floor is modeled as rigid floor diaphragm since it functions as transferring story shears and torsional moments to lateral force-resisting members during earthquake.

Printing the Center of Rigidity & Center of Mass

The `PRINT DIAPHRAGM CR` command may be used to obtain a print-out of the center of rigidity and center of mass at each rigid diaphragm in the model. The lateral force at each floor, as generated by earthquake and wind loading, acts at the center of rigidity of each floor which is modeled as rigid floor diaphragm. The center of mass of each floor is defined as the mean location of the mass system of each floor. The mass of the floor is assumed to be concentrated at this point when the floor is modeled as rigid diaphragm. The distance between these two is the lever arm for the natural torsion moment for seismic loads when that option is used.

Related Links

- *A. To specify post-analysis print commands* (on page 963)
- *A. To specify pre-analysis commands* (on page 960)
- *A. To output the center of rigidity* (on page 964)
- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *A. To check for inter-story drift* (on page 965)
- *G.22 Miscellaneous Facilities* (on page 2195)
- *G.18 Member End Forces* (on page 2187)
- *G.18.1.1 Member Forces at Intermediate Sections* (on page 2192)

- *G.18.1.3 Member Stresses at Specified Sections* (on page 2193)
- *G.18.1 Secondary Analysis* (on page 2192)
- *G.18.1.2 Member Displacements at Intermediate Sections* (on page 2193)

TR.43 Stress/Force Output Printing for Surface Entities

Note: Surface elements have been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow them but their use is not recommended.

Related Links

- *G.18.1.4 Force Envelopes* (on page 2193)

TR.44 Printing Section Displacements for Members

This command is used to calculate and print displacements at sections (intermediate points) of frame members. This provides the user with deflection data between the joints.

General Format

```
PRINT SECTION (MAX) DISPLACEMENTS (NSECT i) (SAVE a) { NOPRINT | ALL | LIST memb-list }
```

Where:

i = number of sections to be taken. Defaults to 12 if NSECT is not used and also if SAVE is used (max=24, min=2).
a = File name where displacement values can be stored and used by the STAADPL graphics program. If the NOPRINT command is used in conjunction with the SAVE command, the program writes the data to file only and does not print them in the output.
This option is not necessary in STAAD.Pro.

Description

This command prints displacements at intermediate points between two joints of a member. These displacements are in global coordinate directions (see figure). If the MAX command is used, the program prints only the maximum local displacements among all load cases.

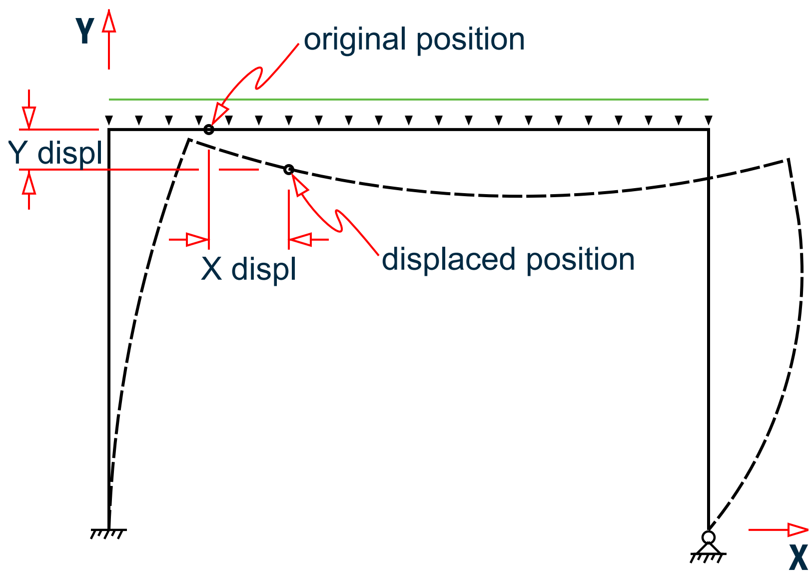


Figure 335: Displacements in the global coordinate directions

Example

```
PRINT SECTION DISPL SAVE
PRINT SECTION MAX DISP
```

`SECTION DISPLACEMENTS` are measured in `GLOBAL COORDINATES`. The values are measured from the original (undeflected) position to the deflected position. See figure above.

The maximum local displacement is also printed. First, the location is determined and then the value is measured from this location to the line joining start and end joints of the deflected member.

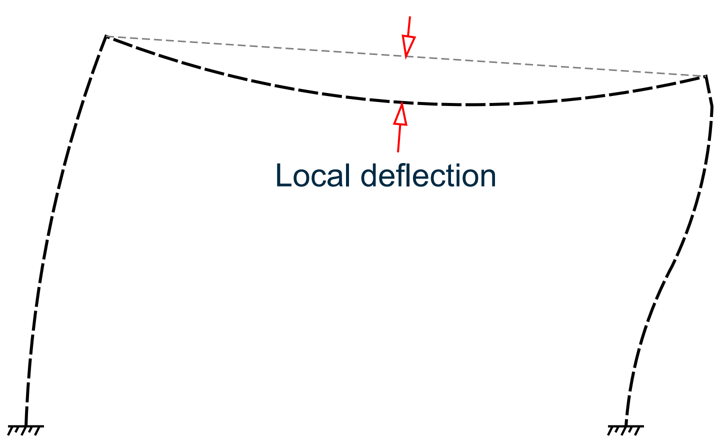


Figure 336: Local deflection

Notes

a. The section displacement values are available in Global Coordinates. The undeflected position is used as the datum for calculating the deflections.
b. This is a secondary analysis command. An analysis must be performed before this command may be used.

Related Links

- *G.21 Printing Facilities* (on page 2195)
- *G.18.1 Secondary Analysis* (on page 2192)
- *G.18.1.2 Member Displacements at Intermediate Sections* (on page 2193)

TR.45 Printing the Force Envelope

This command is used to calculate and print force/moment envelopes for frame members. This command is not available for finite elements.

General Format

```
PRINT { FORCE | MAXFORCE } ENVELOPE (NSECTION i) list-spec
```

```
list-spec = { LIST memb-list | (ALL) }
```

Description

Where:

NSECTION *i*	the number of equally spaced sections to be considered in printing maximum and minimum force envelopes. i is an integer up to 96, though not recommended to be less than 12. If the NSECTION i command is omitted, i defaults to 12.

MAXFORCE command produces maximum/minimum force values only of all sections, whereas the FORCE command prints maximum/minimum force values at every section as well as the max/min force values of all sections. The force components include FY, MZ, FZ, and MY. The SECTION command (as described in TR.40 Load Envelope (on page 2663)) does not define the number of sections for force envelopes. For the sign convention of force values, refer to G.18 Member End Forces (on page 2187)

Note: This is a secondary analysis command and should be used after analysis specification.

Example

```
PRINT FORCE ENV
PRINT MAXF ENV NSE 24
PRINT FORCE ENV NSE 24 LIST 3 TO 15
```

Related Links

- *G.21 Printing Facilities* (on page 2195)
- *G.18.1.4 Force Envelopes* (on page 2193)
- *G.18.1 Secondary Analysis* (on page 2192)

- *G.18.1.2 Member Displacements at Intermediate Sections* (on page 2193)

TR.46 Post Analysis Printer Plot Specifications

This command has been discontinued in STAAD.Pro. Please use the facilities of the Graphical User Interface (GUI) for screen and hard copy graphics.

Related Links

- *G.21 Printing Facilities* (on page 2195)

TR.47 Size Specification

This feature has been deprecated.

TR.48 Steel and Aluminum Design Specifications

This section describes the specifications necessary for structural steel & aluminum design.

The specific details of the implementation of these codes may be found in:

- D. Available Steel Design Codes (on page 969)
- D. Available Aluminum Design Codes (on page 1081)

Additional related information:

- TR.48.1 Parameter Specifications (on page 2676) discusses specification of the parameters that may be used to control the design.
- TR.49 Code Checking Specification (on page 2677) and TR.49.1 Member Selection Specification (on page 2678) describe the `CODE CHECKING` and `MEMBER SELECTION` options respectively.
- TR.49.2 Member Selection by Optimization (on page 2679) discusses member selection by optimization.

Shortlisting Load Cases Used for Design

There are two options available to instruct STAAD.Pro to use only specific load cases for design:

1. The `LOAD LIST` command.
2. The `LOAD LIST ENVELOPE` command, which can only be used if load envelopes have been created (using the `DEFINE ENVELOPE` command). Once envelopes have been created, this command must be used or the design will be based on all active load cases or those specified in a `LOAD LIST` command, rather than only the envelopes.

Related Links

- *Design Commands dialog* (on page 2935)
- *Steel Design - Whole Structure dialog* (on page 2932)
- *D. To design steel members in groups* (on page 977)
- *D. To specify steel design commands* (on page 977)

TR.48.1 Parameter Specifications

This set of commands may be used to specify the parameters required for steel and aluminum design.

Each design code supported by STAAD.Pro has an associated list of control parameters that allow the attributes associated with the members to be defined. The default values for the parameters associated with a design code are documented in the section associated with that code. Refer to D. Available Steel Design Codes (on page 969) for the list of available steel design codes. The section for each design code contains a titled "Design Parameters" which documents the available parameters and the default values that will be used unless alternatives are specified.

Parameters are defined in a collection that is initiated with the command PARAMETER and followed by a design code. This collection of parameters is referred to as a "parameter block."

Note: The code specified in the parameter block will determine the design code that will be used in any subsequent design command (e.g., CHECK CODE or SELECT).

General Format

```
PARAMETER (i)
```

```
CODE design-code
```

```
parameter-name f1  { MEMBER memb-list  | member-group-name  | ALL| PMEMBER psychical-member-list }
```

where:

PARAMETER (*i*)	Optional value used to identify the parameter block.
CODE *design-code*	The CODE parameter is used to specify the steel or aluminum design code and code edition to be checked for design. The default code is AISC ASD. Refer to D. Available Steel Design Codes (on page 969) and D. Available Aluminum Design Codes (on page 1081) for the lists of available steel and aluminum design codes, respectively.
***parameter-name* f1**	refers to the Parameter Names listed in the parameter table contained in the Steel Design and Aluminum Design sections. f1 refers to the value of the parameter. The details of the parameters available for specific codes may be found in D. Design Codes (on page 1086). You can control the design through specification of proper parameters.
MEMBER *memb-list*	a list of member numbers that the parameter value applies to. This can be a simple list of member numbers or can include a range defined by two numbers separated by the keyword TO.
member-group-name	Refer to TR.16 Entities as Single Objects (on page 2235) for the definition of member group names.
ALL	Indicates that the assigned value of the given parameter is assigned to all the members in the model.

PMEMBER ***psychical-member-list***	This is used only with parameters associated with a physical member and lists the physical members to which the parameter value applies.

Example

```
PARAMETERS
CODE AISC
KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
RATIO 0.9 ALL
```

Notes

- All unit-sensitive values should be the current input unit system.
- For default values of parameters, refer to the appropriate parameter table.
- Where a parameter is assigned to a member multiple times, the *last* value given before a design command is used.

Related Links

- *D.Steel Design Overview* (on page 975)
- *D. To specify steel design code and parameters* (on page 976)
- *D. To specify aluminum design code and parameters* (on page 1083)

TR.49 Code Checking Specification

Used to perform the `CODE CHECKING` operation for steel and aluminum members.

This command checks the specified members against the specification of the desired code. Refer to D. Steel Design (on page 969) for detailed information.

General Format

```
CHECK CODE { MEMBER memb-list | ALL | member-group-name  | deck-name | PMEMB pmember-list }
```

Example

```
CHECK CODE MEMB 22 TO 35
CHECK CODE PMEMB 1 3
CHECK CODE MEMB _BEAMS
CHECK CODE ALL
CHECK CODE PMEMB ALL
```

Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2676) for more information on the TRACK parameter.

b. Member group names and deck names are explained in TR.16 Entities as Single Objects (on page 2235) and TR.20.7 Composite Decks (on page 2270) respectively.

c. The PMEMB list is explained in TR.16.2 Physical Members (on page 2238).

Related Links

- *D. To specify steel design commands* (on page 977)
- *D.Aluminum Design Overview* (on page 1082)
- *D. To specify aluminum design commands* (on page 1083)
- *D1.B.1.3 Code Checking* (on page 1141)
- *D1.C.7 Code Checking and Member Selection* (on page 1171)

TR.49.1 Member Selection Specification

This command may be used to perform the MEMBER SELECTION operation.

This command instructs STAAD.Pro to select specified members based on the parameter value restrictions and specified code. The selection is done using the results from the most recent analysis and iterating on sections until a least weight size is obtained. Refer to D. Steel Design (on page 969) for more details.

The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members. If the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in D1.A.4 Built-in Steel Section Library (on page 1088). Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in that table.

Member selection *cannot* be performed on members whose section properties are input as prismatic.

General Format

```
SELECT { MEMBER memb-list | ALL | member-group-name  | deck-name | PMEMB pmember-list }
```

It is important that the keywords MEMBER, ALL, or PMEMB be provided. Thus, the keyword SELECT by itself is not sufficient.

Example

```
SELECT MEMB 22 TO 35
SELECT PMEMB 1 3
SELECT _COLUMNS
SELECT ALL
SELECT PMEMB ALL
```

Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2676) for more information on the TRACK parameter.
b. Member selection can be done only after an analysis has been performed. Consequently, the command to perform the analysis has to be specified before the SELECT MEMBER command can be specified.
c. This command does not cause the program to re-analyze for results based on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.
d. Member group names and deck names are explained in TR.16 Entities as Single Objects (on page 2235) and TR.20.7 Composite Decks (on page 2270) respectively.
e. The PMEMB list is explained in TR.16.2 Physical Members (on page 2238).

Related Links

- *D. To specify steel design commands* (on page 977)
- *D.Aluminum Design Overview* (on page 1082)
- *D. To specify aluminum design commands* (on page 1083)
- *D.Aluminum Design Overview* (on page 1082)
- *D. To specify aluminum design commands* (on page 1083)
- *D1.B.1.4 Member Selection* (on page 1142)
- *D1.B.1.3 Code Checking* (on page 1141)
- *D1.C.7 Code Checking and Member Selection* (on page 1171)

TR.49.2 Member Selection by Optimization

This command performs member selection using an optimization technique based on multiple analysis/design iterations.

The program selects all members based on an optimization technique. This method performs 2 analyses as well as iteration of sizes to reduce the overall structure weight. This command should be used with caution since it will require longer processing time.

The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members. If the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in D1.A.4 Built-in Steel Section Library (on page 1088). Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in that table.

Member selection *cannot* be performed on members whose section properties are input as prismatic.

General Format

```
SELECT OPTIMIZED
```

Notes

a. The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in TR.48.1 Parameter Specifications (on page 2676) for more information on the TRACK parameter.

b. This command will perform one additional analysis and design cycle and therefore may be time consuming. Steps taken are: CHECK CODE ALL; then modify ratios; then SELECT ALL; then PERFORM ANALYSIS; then SELECT ALL. See TR.50 Group Specification (on page 2680) for other options used with this command. You may want to repeat this command for further optimization.

c. This command does not cause the program to re-analyze for results based on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.

Related Links

- *D. To specify steel design commands* (on page 977)
- *D.Aluminum Design Overview* (on page 1082)
- *D. To specify aluminum design commands* (on page 1083)
- *D1.B.1.3 Code Checking* (on page 1141)
- *D1.C.7 Code Checking and Member Selection* (on page 1171)

TR.50 Group Specification

This command may be used to group members together for analysis and steel design.

General Format

```
(FIXED GROUP)
```

```
GROUP (prop-spec) MEMBER memb-list (SAME AS i1)
```

```
prop-spec = { AX | SY | SZ }
```

Where:

SAME AS *i1* member number used in the `SAME AS` command. If provided, the program will group the members based on the properties of *i1*.

Description

This command enables the program to group specified members together for analysis based on their largest property specification.

When `FIXED GROUP` is omitted, the `GROUP` command is usually entered after the member selection command, and the selected members will be grouped immediately and the new member properties will be used in any further operations. After the grouping is completed, the `GROUP` commands are discarded and will not be used again. Further grouping will be done only if a new `GROUP` command is encountered later.

If the `FIXED GROUP` option precedes the group data, the specified grouping will be retained in memory by the program and will be used in subsequent `SELECT` commands. No grouping will occur unless a `SELECT` (MEMBER or ALL or OPTIMIZED) command is performed. However, grouping will be performed with every subsequent `SELECT` command.

Example 1

```
SELECT ALL
GROUP SZ MEMB 1 3 7 TO 12 15
GROUP MEMB 17 TO 23 27 SAME AS 30
```

In this example, the members 1, 3, 7 to 12, and 15 are assigned the same properties based on which of these members has the largest section modulus. Members 17 to 23 and 27 are assigned the same properties as member 30, regardless of whether member 30 has a smaller or larger cross-sectional area. AX is the default property upon which grouping is based.

Example 2

```
FIXED GROUP
GROUP MEMB 1 TO 5
SELECT OPTIMIZED
```

In the above example, the usage of the `FIXED GROUP` command is illustrated. In this example, the `SELECT OPTIMIZED` command involves the six stage process of

1. CHECK CODE ALL followed by modification of RATIO
2. SELECT ALL
3. GROUPING MEMBERS 1 TO 5
4. PERFORM ANALYSIS
5. SELECT ALL
6. GROUPING MEMBERS 1 TO 5

The `FIXED GROUP` command (and the `GROUP` commands that follow it) is required for execution of steps 3 and 6 in the cycle. You may want to repeat this data for further optimization.

Notes

The `FIXED GROUP` + `GROUP` commands are typically entered before the member selection for further analysis and design. This facility may be effectively utilized to develop a practically oriented design where several members need to be of the same size.

All the members in a list for a specific `GROUP` command should have the same cross section type. Thus, if the command reads

```
GROUP MEMB 1 TO 10
```

and member 3 is a W shape and member 7 is a Channel, grouping will not be done. The 10 members must be either all W shapes or all channels.

Also, refer to Note 'C' in TR.49.1 Member Selection Specification (on page 2678).

Related Links

- *D. To design steel members in groups* (on page 977)

TR.51 Steel and Aluminum Take Off Specification

This command may be used to obtain a summary of all steel sections and aluminum sections being used along with their lengths and weights (quantity estimates).

This command provides a listing of the different steel and aluminum table sections used in the members selected. For each section name, the total length and total weight of all members which have been assigned that section will be listed in a tabular form. This can be helpful in estimating steel and aluminum quantities.

General Format

```
{ STEEL | ALUMINUM } (MEMBER) TAKE (OFF) ( { LIST memb-list | LIST membergroupname |
ALL } )
```

If the `MEMBER` option is specified, the length and weight of each member and the section name it is assigned will be reported.

Example

```
STEEL TAKE OFF LIST 71 TO 85
ALUMINUM TAKE OFF LIST PLGNO3
```

Related Links

- *D. To generate aluminum take off* (on page 1084)
- *D. To generate steel take off* (on page 978)

TR.52 Timber Design Specifications

This section describes the specifications required for timber design.
Related Links

- *D. Available Timber Design Codes* (on page 1084)
- *Timber Design - Whole Structure dialog* (on page 2939)
- *Design Parameters dialog* (on page 2934)
- *D. To specify timber design code and parameters* (on page 1085)
- *D1.G.1 Design Operations* (on page 1248)
- *D1.G.2 Allowable Stress per AITC Code* (on page 1249)
- *D1.G.3 Input Specification* (on page 1251)

TR.52.1 Timber Design Parameter Specifications

This set of commands may be used for specification of parameters for timber design.

General Format

```
PARAMETER
CODE design-code
parameter-name f1 { MEMBER member-list | ALL }
```

Where:

parameter-name f1 = the name and corresponding value of the parameter.

Code	*design-code* Value	Parameters List
AITC 1994	AITC 1994 or AITC	D1.G.5 Design Parameters (on page 1254)
AITC 1984	AITC 1984 or TIMBER	D1.G.5 Design Parameters (on page 1254)
CAN/CSA -086-01	TIMBER CANADIAN	D4.D.6 Design Parameters (on page 1444)
EC 5: Part 1-1	TIMBER EC5	D5.E.3 Design Parameters (on page 1589)

Note: All values must be provided in the current unit system.

Related Links

- *D1.G.1 Design Operations* (on page 1248)
- *D1.G.2 Allowable Stress per AITC Code* (on page 1249)
- *D1.G.3 Input Specification* (on page 1251)

TR.52.2 Code Checking Specification

This command performs code checking operation on specified members based on the American Institute of Timber Construction (AITC), Canadian Standards Agency (CSA), or Eurocode (EC5) codes.

The results of the code checking are summarized in a tabular format. Examples and detailed explanations of the tabular format are available in D1.G. American Codes - Timber Design per AITC Code (on page 1248).

General Format

```
CHECK CODE { MEMBER member-list | ALL }
```

Note: For the EC5 code, the output of this command may be controlled by the TRACK parameter.

Related Links

- *D1.G.6 Member Design Capabilities* (on page 1259)

TR.52.3 Member Selection Specification

This command performs member selection operation on specified members based on the American Institute of Timber Construction (AITC 1984) code.

This command may be used to perform member selection according to the AITC 1984 code. The selection is based on the results of the latest analysis and iterations are performed until the least weight member satisfying all the applicable code requirements is obtained. Parameters may be used to control the design and the results are available in a tabular format. Detailed explanations of the selection process and the output are available in D1.G. American Codes - Timber Design per AITC Code (on page 1248).

General Format

```
SELECT { MEMBER memb-list | ALL }
```

TR.53 Concrete Design Specifications

This section describes the specifications for concrete design for beams, columns and individual plate elements. The concrete design procedure implemented in STAAD consists of the following steps:

TR.53.1 Design Initiation

This command is used to initiate concrete design for beams, columns and individual plate elements.

This command initiates the concrete design specification. Without this command, none of the following concrete design commands will be recognized.

Note: This command must be present before any concrete design command is used.

General Format

```
START CONCRETE DESIGN
```

Related Links

- *D. To specify concrete design code and parameters* (on page 1069)

TR.53.2 Concrete Design-Parameter Specification

This set of commands may be used to specify parameters to control concrete design for beams, columns and individual plate elements.

General Format

```
CODE design-code
```

```
parameter-name f1 { MEMBER member-list | ALL }
```

Where:

design-code = the concrete design code name, which is described in the concrete chapters of the D. Design Codes (on page 1086).

parameter-name f1 = is the concrete design parameter and corresponding value. Wherever applicable, this value is input in the current units. The UNIT command is also accepted during any phase of concrete design.

parameter-name = refers to the concrete parameters described in D1.F.3 Design Parameters (on page 1201) for the ACI code and in various corresponding concrete chapters of the D. Design Codes (on page 1086) for all other codes.

Note: All values must be provided in the current unit system.

Related Links

- *D. To specify concrete design code and parameters* (on page 1069)
- *D1.F.3 Design Parameters* (on page 1201)

TR.53.3 Concrete Design Command

This command may be used to specify the type of design required. Members may be designed as BEAM, COLUMN or ELEMENT.

Members to be designed must be specified as BEAM, COLUMN, or ELEMENT. Members, once designed as a beam, cannot be redesigned as a column again, or vice versa.

General Format

```
DESIGN { BEAM | COLUMN | ELEMENT } { memb-list | (ALL) }
```

Notes

a. Only plate elements may be designed as ELEMENT.
b. Enter this command after the parameters needed for this command have been entered.
c. The DESIGN ELEMENT command designs individual plate elements using the procedure explained in D1.F.6 Slab Design (on page 1243) for the ACI code. For theoretical information on designing individual plate elements per other design codes, please refer to D. Design Codes (on page 1086).

Related Links

- *D. To specify concrete beam design command* (on page 1070)

TR.53.4 Concrete Take Off Command

This command may be used to obtain an estimate of the total volume of the concrete, reinforcement bars used and their respective weights.

This command can be issued to print the total volume of concrete and the bar numbers and their respective weight for the members designed.

Tip: This command may be used effectively for quick quantity estimates.

General Format

```
CONCRETE TAKE OFF
```

Sample Output

```
          **************  CONCRETE  TAKE OFF  **************
                (FOR BEAMS, COLUMNS AND PLATES DESIGNED ABOVE)
     NOTE: CONCRETE QUANTITY REPRESENTS VOLUME OF CONCRETE IN BEAMS, COLUMNS,
AND PLATES DESIGNED ABOVE.
           REINFORCING STEEL QUANTITY REPRESENTS REINFORCING STEEL IN BEAMS
AND COLUMNS DESIGNED ABOVE.
           REINFORCING STEEL IN PLATES IS NOT INCLUDED IN THE REPORTED
QUANTITY.

          TOTAL VOLUME OF CONCRETE =         4.4 CU. YARD


           BAR SIZE         WEIGHT
            NUMBER         (in lbs)
           --------        --------
               4               261
               5                87
               6                99
               8               161
               9               272
                       ------------
             *** TOTAL=        880
```

Related Links

- *D. To generate concrete take off* (on page 1071)

TR.53.5 Concrete Design Terminator

This command must be used to terminate the concrete design.

This command terminates the concrete design, after which normal STAAD commands resume.

Note: Without this command, further STAAD commands will not be recognized.

General Format

```
END CONCRETE DESIGN
```

Example

```
START CONCRETE DESIGN
CODE ACI
FYMAIN 40.0 ALL
FC 3.0 ALL
DESIGN BEAM 1 TO 4 7
DESIGN COLUMN 9 12 TO 16
DESIGN ELEMENT 20 TO 30
END
```

Related Links

- *D. To specify concrete design code and parameters* (on page 1069)

TR.54 Footing Design Specifications

This feature has been removed from batch mode design. Please contact the Technical Support department for further information.

Footing design may be performed using STAAD Foundation Advanced through the D. Foundation Design (on page 1078) in the graphical interface.

TR.55 Shear Wall Design

Note: Shearwall design has been deprecated in STAAD.Pro CONNECT Edition. The analysis and design engine will allow it but its use is not recommended.

TR.56 End Run Specification

This command must be used to terminate the STAAD run.

This command should be provided as the last input command. This terminates a STAAD run.

General Format

```
FINISH
```

Index of Commands

The following is an alphabetic list of all STAAD.Pro commands in the STAAD Command Reference.

A, B

C

D

E

F

G

H

I

J, K

L

M

T

U, V, W, X, Y, Z

A
Ribbon Control Reference

This section describes the tools found in the ribbon toolbars.

Note: The ribbon toolbar dynamically updates for the current workflow selection.

File tab

The **File** tab opens the STAAD.Pro Backstage view, which is used for file operations.

Info tab

Used to specify general information about the structure, including a job description, job number, persons responsible for creating, checking, and approving the structure, etc. This page also provides general information about the structure such as the total numbers of nodes, members, elements as well as the STAAD commands present in the structure file.

Job Information

Job Name	Type a short description of the job. This description appears in the reports.
Client	Type the client name, if any.
Job Number / Revision / Part / Reference	Type the job, part and reference numbers, if any.
Engineer / Checker / Approved	These items are used to track modifications in the structure file. The names and dates of the changes may be entered.
Last Review	Type the date of the last review provided by respective engineer, checker, or approver.

CONNECT Project Information	When your STAAD.Pro model has a CONNECTED Project associated with it, the CONNECTED Project is displayed here. Click **[...]** to open the **CONNECTED Project Information** dialog which contains CONNECTED Project details and can be used to change the associated CONNECTED Project.

File Information

This group contains several items for information only. These items include the Filename, Directory, Date / Time of first creation, and the **File size**.

Model Information

Displays the total numbers of nodes, beams, and plates present in the structure. The Highest # column shows the highest Node / Beam / Plate numbers in the structure. In case the structure has discontinuous Node/Member/ Element number, the Highest # value will be different from the Count value.

The **Commands in File** group shows the STAAD commands which have been used in the structure.

New tab

Used to create a new STAAD project. Options are available to start with an analytical model, a physical model, or in STAAD Building Planner.

The **Additional License** options are available here to allow you to select which license options you want to use before creating a file.

Open tab

Used to open an existing STAAD input file or Archive. You can also open STAAD projects from a ProjectWise data source.

Select the **Recent** tab to view recent files by type. Hover your mouse pointer over any file to display the file and CONNECT Project properties.

The **Additional License** options are available here to allow you to select which license options you want to use before opening a file.

Save / Save As tabs

Used to save changes to the current model or to save the current model as a different file.

Save

Click this entry to save any changes made to the current STAAD input file.

Save As

This tab allows you to enter a new **File Name** and **Location** to save the current input file as a different file. Click **Browse** to open a Windows open dialog to select a folder.

Click **Save** to save the new file.

Backup/Restore tab

Used to create a backup of your STAAD project or to restore a STAAD project to a previously created restore point.

Menu item	Description
Restore Model	Used to restore the open STAAD project to a previously created restore point. Select a restore point in the list and then click the **Restore file** tool that appears when hovering over the right-hand side of an entry.
Compare Models	Used to compare two STAAD project versions. When one restore point is selected, you can compare this input file with the current model state. Selecting to compare points (by holding the **<Ctrl>** key and clicking the restore point labels) can be used to compare any two points. Click Compare Models or Compare with Current Model and a window opens with the input files compared.
Create Restore Point	Used to create a restore point for your entire STAAD project. Type an optional Name and Description for the resotre point (a generated name will be created for the restore point if you don't). You can also select the option to **Include Analysis Data** to the restore point as well.

Related Links

- *I. To create a restore point* (on page 2065)
- *I. To restore a model from a backup* (on page 2066)
- *I. To compare backups* (on page 2066)

Print tab

Used to print input, output, and reports for the current STAAD project.

Tip: Press **<Ctrl+P>** to open this **File** menu tab.

Tool name	Description
Input Command File	Used to print the contents of the STAAD input file (file extension .std) from the STAAD Editor.
Output File	Used to print the contents of the STAAD output file (file extension .anl) from the STAAD Output Viewer.
Report	Used to print the report for the STAAD input file. **Note:** You can set up the report contents in the **Postprocessing** workflow.
Error Report	Print error file, if any.

Report tab

Used to configure, review, and export STAAD reports.

Tool name	Description
Setup	Opens the **Report Setup** dialog, which is used to specify the contents of the customized report.
Preview Report	Opens the **Report Print Preview** window, which is used to review your project report before creating a hard copy. **Note:** You can also preview the report in the **Report** page in the **Postprocessing** worfklow.
Export to Text File	Opens a **Save As** dialog, which is used to save the current report to a plain text file (`.txt` file extension).
Export to MS Word File	Opens a **Save As** dialog, which is used to save the current report to a Microsoft Office Word document (`.doc` file extension).

Report Setup dialog

Used to specify the contents of the customized report

Opens when either:

- the **Setup** tool is selected from the **Report** tab on the **File** ribbon tab, or
- **Setup Report** is clicked on the **Reports** page in the **Postprocessing** workflow

Items tab

The *Items* tab is used for specifying the items to be included in the report.

Available list Displays items that may be included in the report. The Available list is divided into the following categories using the drop-down list:

- **Input -** lists all input related data that may be included in a report.
- **Output -** lists all output related data that may be included in a report.
- **Pictures -** lists all the pictures included in the **Picture Album** tab.
- **Reports -** lists all reports that have been previously created and saved. Thus, reports may contain sub-reports.
- **Saved Views -** lists all of the named (i.e., saved) views from the **View Management** dialog
- **RAM Connection Summary Report -** lists the “RCNX Report” after steel Connection Design has been performed on one or more joints.
- **Earthquake Check Reports -** lists the reports for a Eurocode 8 response spectrum check after these have been performed in the Earthequake workflow.

Selected list Displays items that have been selected for the report. Items in the **Available** list may be mixed and matched in any order or category within the **Selected** list. When this dialog box opens, a default item, “Job Information” appears in the **Selected** list.

Click this button…	to…
>	Add the currently selected item or items in the **Available** list to the **Selected** list. **Tip:** Double-clicking an item in the **Available** list has the same effect.
>>	Add all items from the current drop-down list category to the **Selected** list.
<	Remove the currently selected item or items the **Selected** list from that list. **Tip:** Double-clicking an item in the **Selected** list has the same effect.
<<	Remove all items from the **Selected** list.

Tip: To select multiple items in a list, hold down the **<Ctrl>** key while selecting items.

Items in the **Selected** list can be rearranged in any desired order by selecting an item and then using the up or down arrow to the right of the **Selected** list.

Report Details Increments — Used to specify the number of segments into which a member would be broken up for printing section forces.

Load Cases tab

Used specify which load case for which results are to be included in the report.

The results can be grouped by node and beam numbers or by load case. Specify the method of *Grouping* using the radio buttons. Load cases are moved to and from the *Available* and *Selected* lists as described before under the *Items* tab.

Available list	Displays all avaialble load cases that may be included in the report which have not already been included.
Selected list	Displays load cases that have been selected for the report.
Grouping for Load Tables	Check the option to group load tables either **by Load Type** or **by Load Case**.
Grouping for Result Tables	Check the option to group results either **by Node/Beam** or **by Load Case**.

Modes tab

Used to select mode shape numbers for which the results will be included for dynamic analyses (modal calculation, response spectrum, time history, etc.).

Note: A successful dynamic analysis must first be performed for mode shapes to be made available.

Grouping — Select to group modal results either **by Node** number or **by Mode** number.

Available list Displays available modes in the mode.

Selected list Displays all modes in the model. All mode shapes will be added to the **Selected** list by default.

Ranges tab

Used to select the members and elements to be included in the report. By default, the report includes all members, elements, and nodes.

All Select this option to include all model entities in the report (default).

View Select this option to select a previously saved view in the drop-down list. Only model entities included in the selected view will be included in the report.

Group Select this option to select a previously defined group name in the drop-down list. Only model entities included in the selected group will be included in the report.

Property Select this option to select a property tag in the drop-down list. Only model entities included in the selected property will be included in the report.

Ranges Select this option to specify a range of **Nodes** (by node number) and/or **Beams/Plates/Solids** (by entity number). Separate individual entries in either list using a comma. Ranges of values can be represented using a dash (e.g., 5-10).

Steel Design tab

Used to format the steel design results included in the report when steel design detail output has been added to the report on the **Items** tab.

Check Results Options for results generated by the code check facility. Use the radio buttons to specify whether to include no results, all results, the results of the first check or the results of the final check.

Select Results Options for results generated by the member selection facility. Use the radio buttons to specify whether to include no results, all results, the results of the first select or the results of the final check.

Summary Only (Track 0) Set this option to include a brief design summary for a BS 5950 steel design. Otherwise, the specified level of details (`TRACK` command option) is used.

Print Failed checks Only Set this option to include only steel design results for members which have failed one or more checks. Otherwise, the specified level of details (`TRACK` command option) is used for both passing and failing members.

Font Opens an Font dialog, which is used to change the typeface parameters used for steel design results. The current font settings are displayed adjacent to this button.

Picture Album tab

Used to control the display of pictures of the structure that have been taken using the **Take Picture** tool.

Note: To include pictures in the report, they must be added in the **Selected** list on the **Items** tab. Pictures are available in the Pictures category.

Name	A list of all available pictures appears in the drop down list.
Delete Picture	Removes the currently selected picture from the picture album. **Caution:** This action cannot be undone.
Preview	radio buttons allow the selection of results generated by the code check facility. Use the radio buttons to specify whether to include no results, all results, the results of the first check or the results of the final check.
Caption	Type or edit the caption to be included with the drawing in the report. **Tip:** If the model has changed since the picture was taken, a note is appended to the caption letting you know that changes have been made.
Full Page	Set this option to specify values of Height and Width equal to the full size of the default paper size. The picture will then take up an entire page of the report.
Height / Width	Specify printed dimensions for the selected picture. The aspect ration of the picture will remain constant, so updating one dimension will update the other. Click **in/mm** beside either dimension to toggle between the two units of length.
Width	Opens an Font dialog, which is used to change the typeface parameters used for steel design results. The current font settings are displayed adjacent to this button.

Options tab

Used for setting various report display options.

Header	Set this option to include the header on each page of the report. This includes custom name and logo, sheet numbering, job information, file name, and file modification information.
Page Outline	Set this option to add a solid, black border around the extents of the page content.
Footer	Set this option to include a footer on each page of the report. The footer contains a print time stamp, program version information, and print run information.
Prefix / Suffix	Specify optional sheet number prefix or suffix content in the respective field. This is useful if the STAAD.Pro report is to be included in an appendix or some other format where a special numbering format is used.
No. pages from	Specify the initial page number used for the report. This is useful if you have other content to include in a larger report before the STAAD.Pro output.
Reverse page order	Set this option to reverse the order in which pages are sent to the printer. The contents of the report, including sheet number, are *not* affected by this option.
Grid	Set this option to include grid lines around and between table elements.
Start each table on a new page	Set this option to add a page break after each table, even when a table does not fill an entire page alone.

Font	Opens a Font dialog, which is used to select the type face properties for the **Column Heading** or **Table** contents. The current font face and size are displayed for each setting.

Name and Logo tab

Used to add the your company name and/or logo to the report. This tab contains a viewing area to preview how the company name and logo will appear on the report.

Note: The **Remove Bentley (B) logo from Report** must be set on the **Configure** dialog File Options tab to remove the Bentley logo from the report heading.

Preview area	To use a text header, simply type in the preview area. Use the **Font** and **Alignment** controls to edit the appearance of the text. To use a graphic logo, use the **File** and **Position** controls.
File	Opens a Windows file open dialog, which is used to select a Windows Bitmap image (file extension `.bmp`) to use graphic for inclusion in report headings.
Paste	Adds an image file from the Windows clipboard.
Delete	Removes the graphic file from the report header.
Position	Select the horizontal position in the company identification portion of the report header.
Font	Opens a Font dialog, which is used to select the type face properties for the logo. **Tip:** In order to prevent clipping of the text block, a 14 point font size is the largest size for two lines and a 9 point font size is the largest size for three lines.

Load / Save tab

Used to save the contents of the customized report and to load previously saved reports.

Report	Previously saved reports are available in the drop-down list. Select one to load the report details for editing, printing, or deletion.
Save As	Opens the **Save Report** dialog, which is used to provide a name for the current report settings. Click **OK** in this dialog to save the named report in the **Report** list.
Delete	Removes the selected report name in the **Report** list. **Caution:** This action cannot be undone.

OK	Saves any changes to the Report setup and closes the dialog.
Cancel	Closes the dialog with out saving any changes.
Print	Prints the report with the current settings.
Help	Opens the STAAD.Pro Help window.

Related Links

- *P. To setup report contents* (on page 2017)

- *P. To add a custom header and logo to reports* (on page 2018)

ISM tab

Contains tools for working with Integrated Structural Modeling repositories.

Note: ISM has some limited functionality when working with analytical models. It is recommended to use the Physical Modeling workflow in order to exchange physical model data with an ISM repository through the STAAD.Pro Physical Modeler.

Menu item	Description
 **Create Repository**	Transfers the current model opened in STAAD.Pro and generates a new ISM Repository. This is the most common way in which an ISM Repository is initially created.
 **New from Repository**	Creates a new STAAD.Pro input file from an existing ISM Repository. This is used to transfer model data into other tools used for your workflow.
 **Update Repository**	Coordinates changes made to the model in the client application and coordinate some or all of those changes with an existing ISM Repository.
 **Update from Repository**	Updates the current STAAD.Pro input file with some or all of the changes which have been made to the ISM Repository.

Related Links

- *I. ISM Sync Tools Overview* (on page 2063)

Import/Export tab

Import

Used to import data from other formats. Opens a Windows Open dialog, which is used to select a file name and location for importing the selected file format.

DXF (*.dxf)	Used to import an Autodesk AutoCAD® 3D DXF file.
QSE ASA (*.asa)	Import an ASA format file.
CIS/2 (*.stp)	Options are provided for creating new models (from Start page only) or updating the current model (when a STAAD project is open) from a CIS/2 model.

Export

Used to export data to other formats, such as DXF, the CIMsteel STEP format, etc.

3D DXF (*.dxf)	Exports the entire model into a 3D DXF file.
2D DXF (*.dxf)	Exports a selected plane of the model into a 2D DXF file. This option opens the Export 2D DXF dialog. **Note:** The Export 2D DXF dialog is used to indicate the section of the model to use to generate a plan as well as the model objects to include in the drawing.
QSE ASA (*.asa)	Export to an ASA format file.
CIS/2 (*.stp)	Exports the entire model into an analysis model CIS/2 file through the **Cis2Link** dialog.

DXF Import dialog

Used to specify the vertical axis for data imported from a DXF file into a STAAD input file.

Opens when a DXF file is selected from the **Import** dialog.

Structure Convention	Select an option for defining the global vertical ("up") axis for the structure. • **No Change -** Use the vertical axis as defined in the 3D DXF file. • **Y Up -** Adds a `SET Y UP` command to the STAAD input file. • **Z Up -** Adds a `SET Z UP` command to the STAAD input file.
OK	Imports the selected DXF file contents into the current STAAD input file.
Cancel	Closes the dialog without importing any model information.
Help	Opens the STAAD.Pro help window.

Cloud Services tab

Contains tools for using Bentley Cloud Services.

Menu item	Description
Access CONNECTION Center Port	Opens the CONNECT Personal Portal in your web browser.
ProjectWise Projects	Opens the Project Portal for your ProjectWise Project.

Menu item	Description
Download Results	Click to download the cloud analysis results for the current project.

The **CONNECT Project Information** associated with the current model is displayed here. If the model is not associated with a ProjectWise project, then you can click **Associate Project** to do so (required for a Cloud Analysis). If you want to disassociate the current ProjectWise project, then click **Disassociate Project.**

Settings tab

Used to control some of the application settings.

Tool name	Description
 **Display Options**	Opens the **Options** dialog, which is used to change the length and force units for various values which can be displayed in the graphics window, such as member properties, material constants, load magnitudes, plate stresses, etc.
 **Set Structure Colors**	Opens the **Color Manager** dialog, which is used to set colors for highlight, entities, and results..
 **Structural Tooltip Options**	Opens the **Tool Tip Options** dialog, which is used to set tooltip options for structural elements.

Help tab

Contains links for getting additional assistance and information about STAAD.Pro.

Table 284: Help menu items

Tool name	Description
Contents	Opens the online help in your web browser.
OpenSTAAD Help	Opens the online help in your web browser to the OpenSTAAD reference section.
Technical Support	Opens the Worldwide Technical Support Resources dialog, which offers a map of worldwide technical support contacts for STAAD.Pro.

Tool name	Description
ReadMe	Opens the STAAD.Pro Read Me file in your web browser.
Knowledge Base and FAQs	Opens the STAAD.Pro Support Solutions page on Bentley Communities in your web browser.
Discussion Group	Opens the RAM | STAAD Forum on Bentley Communities in your web browser.
Tutorials	Opens the Bentley LEARNserver in your web browser. Here you can access training and learning paths for STAAD.Pro and other Bentley products.

Table 285: About menu items

Tool name	Description
About STAAD.Pro	Opens the About STAAD.Pro CONNECT Edtion dialog, which contains version, licensing, and legal information about the product. Any Technical Preview items contained in the current version will also be listed here.
Product News	Opens the STAAD.Pro product page at Bentley.com in your web browser.
Home Page	Opens the Bentley.com home page in your web browser.
Ideas	Provide feedback with any ideas you have to improve STAAD.Pro.

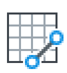
Geometry tab

Table 287: Clipboard group

Tool name	Description	Shortcut
Undo	Negates the last operation. **Note:** You cannot undo settings changes. After you undo an operation, the operation previous to the negated operation can be undone. You can, therefore, undo a series of previous operations by repeatedly choosing Undo. There is no limit to the number of undos that can be performed. **Note:** Only geometry, property, support and member/plate specification operations support Undo/Redo.	**<Ctrl+Z>**
Redo	Negates the last undo operation. You can redo a series of negated operations by repeatedly choosing Redo. Each undo operation is a single re-doable operation regardless of the number of negated operations.	**<Ctrl+Y>**
Delete	Used to delete selected object(s).	**<Del(ete)>**
Cut	Used to cut selected object(s) (delete and copy to clipboard). The deleted objects may then be pasted. The object(s) remain on the Clipboard until another Cut or Copy is performed.	**<Ctrl+X>**

Tool name	Description	Shortcut
 **Copy**	Used to copy selected object(s) to clipboard for subsequent pasting. The element(s) remain on the Clipboard until another Copy or Cut is performed.	**<Ctrl+C>**
 **Paste**	Opens the Paste with Move dialog (on page 2715), which is used to specify the insertion point for pasting clipboard elements into the model. Because the elements remain on the Clipboard after pasting, you can paste repeatedly.	**<Ctrl+V>**

Table 288: Structure group

<table>
<tr><th colspan="2">Tool name</th><th>Description</th></tr>
<tr><td rowspan="4">Grids ></td><td>
Beam Grid</td><td>Opens the Snap Node/Beam dialog, which is used to specify the grid and snap settings as well as to create members and nodes automatically by snapping to grid points.</td></tr>
<tr><td>
Triangular Plate Grid</td><td rowspan="2">Opens the Snap Node/Plate dialog, which is used to specify the grid and snap settings as well as to create plates and nodes automatically by snapping to grid points.</td></tr>
<tr><td>
Quad Plate Grid</td></tr>
<tr><td>Solid Grid</td><td>Opens the Snap Node/Solid dialog, which is used to specify the grid and snap settings as well as to create solids and notes automatically by snapping to grid points.</td></tr>
</table>

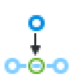
Tool name	Description
 **Structure Wizard**	Opens the M. Structure Wizard (on page 695), which offers a library of pre-defined structure prototypes, such as Pratt Truss, Northlight Truss, Cylindrical Frame, etc. The **Structure Wizard** may parametrically generate a structural model, and then transfer and superimpose it on the current structure.
 **Translational Repeat**	Opens the **Translational Repeat** dialog, which is used to make copies of portions of the structure at intervals along a linear path.
 **Circular Repeat**	Opens the **3D Circular** dialog, which is used to make copies of portions of the structure at intervals along a circular path.
 **Rotate**	Opens the **Rotate** dialog, which is used to rotate the selected portions of the structure about the specified axis through a specified angle. The selected portions may be copied or moved.
 **Mirror**	Opens the **Mirror** dialog, which is used to copy or move the selected portions of the structure by “mirroring” about any plane parallel to one of the three global Cartesian coordinates.
 **Wall/Slab Connection**	Used to select a wall which will have a common boundary with a slab consisting of a previously generated finite element mesh. This tool allows the consideration of boundary conditions at the interface of the panel and any other panel on whose surface one of its edge lies.
 **Input Units**	Opens a drop-down list used to set the current input units for length and force. Click **Apply** to set the unit selection.

Table 289: Node group

Tool name		Description
 **Move Node >**	**Move Joint**	Opens the **Move Entities** dialog, which is used to specify the translational offset for moving a selection of nodes.
	Move Origin	Opens the **Move Origin** dialog, which is used to specify the translational offset for moving a the model origin. All model geometry is updated relative to the new origin location.
 **Merge Selected Nodes**		Opens the **Select Node** dialog, which is used to select a node number you want to keep in the model. Other selected nodes will be merged to this node number.
 **Renumber Nodes**		Opens the **Renumber** dialog, which is used to renumber selected nodes starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.

Table 290: Beam group

Tool name		Description
 **Add Beam >**	 **Add Beam**	Used to add beam by clicking start and end nodes.
	 **Add Beam Between Midpoints**	Used to add a new beam connecting the midpoints of two existing beams. New nodes will be created at the exiting beam midpoints.

Tool name	Description
 **Add Beam by Perpindicular Intersection**	Used to add a beam from an existing node perpendicular to an existing beam. A new node will be created on the existing beam.
 **Add Curve Beam**	Used to define curved members. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees.
Create Colinear Beams	Used to create beams between selected nodes which lie along a straight path (i.e., colinear).
Set New Beam Attributes	Opens the **Define Member Attributes** dialog, which is used to predefine member attributes such as properties, materials, beta angles, or release specifications for member attribute sets. Members which are then added to the model will have these properties by default.
 **Beam Layout**	Opens the **Beams** table.
 **Insert Node**	Opens the **Insert Node into Beam #** dialog, which is used to insert one or more nodes at specified distances along selected members.
 **Stretch Members**	Opens the **Stretch Member(s)** dialog, which is used to increase the length of a member in various ways.

Tool name		Description
Connect Beams	**Connect Beams Along X | Y | Z axis**	Used for automatically creating beams connected between points which lie along an imaginary straight line parallel to the selected global axis. Members will only be created from selected nodes. The program will automatically detect which nodes are collinear and lie along the global axis selected. Beams will be generated along the nodes that satisfy those requirements. The following figure shows a group of nodes selected for this particular operation.
Move Beam		Opens the **Move Selected Beams** dialog, which is used to specify the translational offset for moving a selection of beams.
Intersect Beams >	**Highlight Intersect Beams**	Used to scan the model and locate members which cross each other in space, but are not necessarily connected to each other at the intersection point.
	Intersect Selected Beams	Splits members at detected intersection points and creates an addition node at this intersection point such that the members are now connected.
Merge Selected Beams		Opens the **Merge Selected Beams** dialog, which is used to join two collinear beams and replace them with one beam.
Renumber Beams		Opens the **Renumber** dialog, which is used to renumber selected beams starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.
Break Beams at Selected Nodes		Splits a beam into separate beams at any selected node(s) which are located along that beam. This is useful when some new nodes are created on the line of the member and you want to split the beams at these newly created nodes.

Table 291: Plate group

Tool name		Description
Add Plate >	 **Add Quad Plate**	Used to add Quadrilateral plates. To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated.
	 **Add Triangle Plate**	Used to add triangular plates. To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated.
	 **Create Infill Plates**	Used to automatically generate plates from a selection of beams bounding panels. Panels bounded by beams on all sides are filled with plates. Typically used for floor slabs, this method significantly reduces the modeling time for generation of floor slabs in multi-storied framed structures. If no closed polygon can be found of the enclosed shape is not planar, then an error message will be displayed. **Note:** Depending on the node order, some or several plates may be facing opposite directions. The **Plate Reference Point** dialog can be used to remedy this. This will orient all the plates in the same direction. Also, check for plates, whose boundary nodes do not align themselves with other connecting plates. The **Structure Tools** > **Check Beam/Plate Connectivity** on the **Utilities** ribbon tab can help identify these problems.
	Set New Plate Attributes	Opens the **Define Plate Object Property** dialog, which is used to define the property, material and releases to each new plate element as it is created.
 **Plate Layout**		Opens the **Plates** table.
 **Parametric Models**		Opens the **Parametric Models** dialog, which is used to create and edit wall, slab, and panel meshes.

Tool name		Description
Generate Mesh	 **Create Mesh**	Used to create a collection of plates by selecting node points and then defining mesh parameters.
	 **Plate Mesh**	Used to generate a finite element mesh for an existing plate element. The polygon can be meshed into quadrilateral or triangular elements. You have control over parameters like number of divisions along each side of the polygon. Polygonal holes can also be defined within the surface during the meshing process for polygonal meshing.
 **Move Plate**		Opens the **Move Selected Plates** dialog, which is used to specify the translational offset for moving a selection of plates.
 **Renumber Plates**		Opens the **Renumber** dialog, which is used to renumber selected plates starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.

Table 292: Solid group

<table>
<tr><th colspan="2">Tool name</th><th>Description</th></tr>
<tr><td>
Add Solid ></td><td>Add n Noded Solid




</td><td>Click to add a solid of 8, 7, 6, 5, or 4 nodes, respectively.</td></tr>
<tr><td colspan="2">
Solid Layout</td><td>Opens the Solids table.</td></tr>
<tr><td colspan="2">
Move Solid</td><td>Opens the Move Selected Solids dialog, which is used to specify the translational offset for moving a selection of solids.</td></tr>
<tr><td colspan="2">
Renumber Solids</td><td>Opens the Renumber dialog, which is used to renumber selected solids starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.</td></tr>
</table>

Table 293: Physical Member group

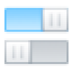
Tool name	Description
 **Physical Member Layout**	Opens the **Physical Members** table.
 **Toggle Physical Member Mode**	Sets the analytical modeling mode to assign sections, properties, and design parameters to physical members.
 **Form Members**	Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.
 **AutoForm Members**	Used to automatically generate multiple physical members from a group of selected analytical beams.

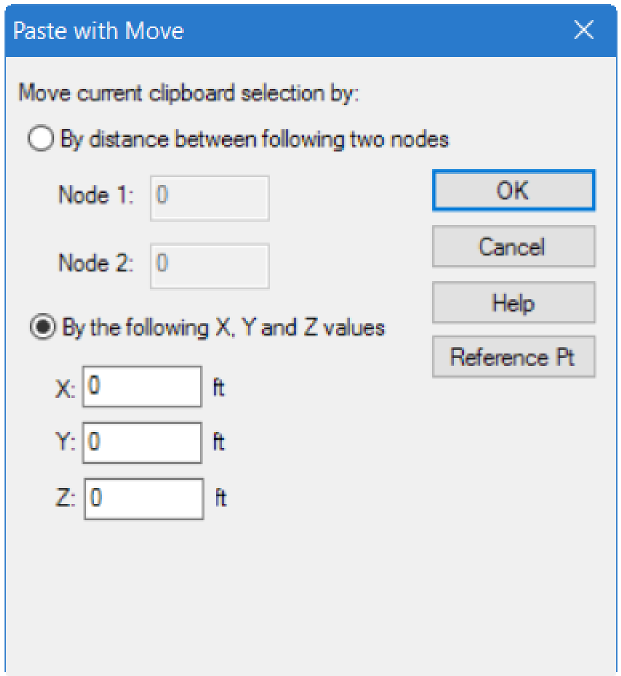
Table 294: Composite Deck group

Tool name	Description
 **Composite Deck Layout**	Opens the **Composite Deck** dialog, which is used to define and assign composite floor deck systems.

Paste with Move dialog

Used to specify the insertion point for pasting clipboard elements into the model.

Opens when the **Paste** tool is selected with one or more objects on the clipboard.

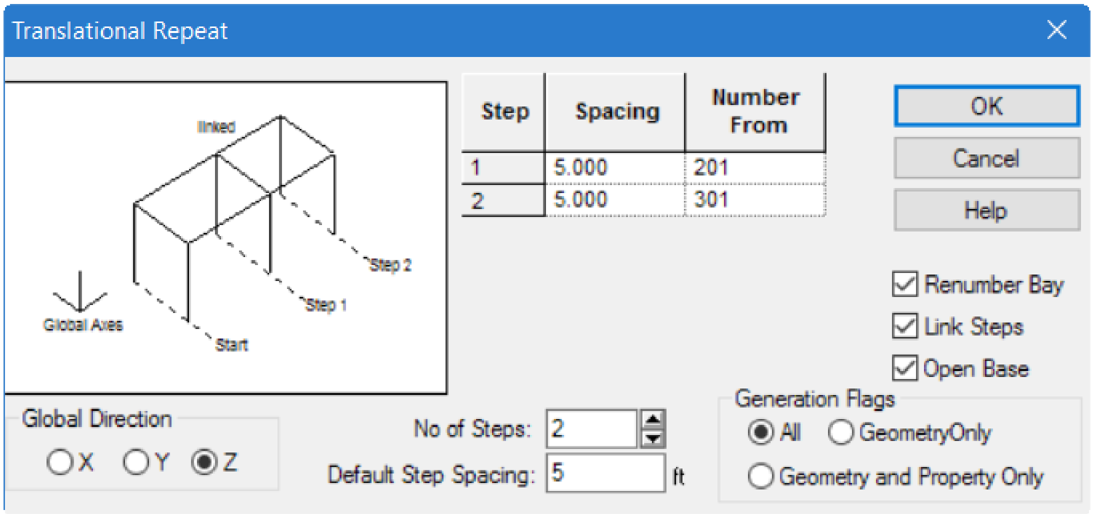


By distance between following two nodes	Specify two existing nodes. The vector distance between these two nodes will be used to insert a copy with respect to the original.
By the following X, Y, and Z values	Specify global axis distances to insert a copy to, with respect to the original.
OK	Accepts the paste settings and inserts a copy into the model.
Cancel	Closes the Paste with Move dialog without inserting a copy.
Help	Opens the Help window.
Reference Pt	Opens the Specify Reference Point dialog, which is used to select the insertion point which the paste command will be executed.

Translational Repeat dialog

Used to make copies of portions of the structure at intervals along a linear path

Opens when the **Repeat > Translational** tool is selected from the **Beam** group on the **Geometry** ribbon tab.



Global Direction — Choose any one of the three possible global directions along which the selected structural entities should be copied.

No of Steps — Specify the number of copies you want.

Default Step Spacing — Type the default spacing between steps (or copies) in the edit box in current length units. You can change the spacing between copies individually using the step table.

Step Spacing table — You can manually type the **Spacing** to each step as well as type the **Number From** to use when the **Renumber Bay** option is selected.

Renumber Bay — Instructs the program to use a user-specified starting number for the members generated in each step of the repeat.

Link Steps — Instructs the program to generate new members between each step in the direction of the repeat.

Open Base — This option instructs the program to *not* generate linking members at the base of the structure (i.e., the lowest nodes in the selection).

Generation Flags — Select the structural data that you want copied to the newly generated elements:

- **All** — Uses all properties assigned to the selected object (e.g., loads, properties, design parameters, member releases, etc.) are copied
- **Geometry Only** — Only geometry data is copied. That is, nodes, member and element incidences are generated but no properties, loads, etc. will be assigned to the new objects.
- **Geometry and Property Only** — Only geometry data and property assignments are copied. All other assignments (e.g., loads and design parameters) are not copied.

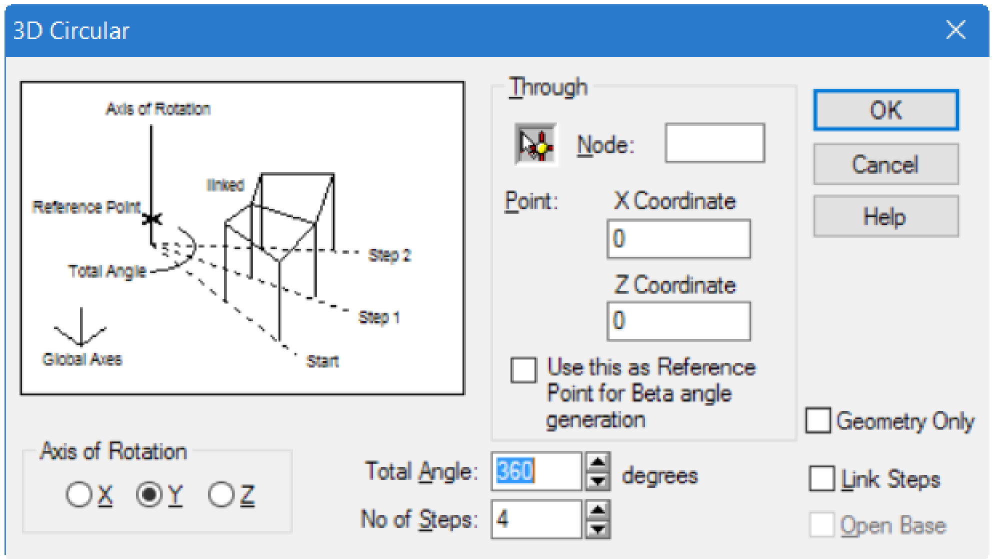
Related Links

- *M. To generate copies of geometry along a line* (on page 687)

3D Circular dialog

Used to make copies of portions of the structure at intervals along a circular path.

Opens when the **Repeat > Circular** tool is selected from the **Structure** group on the **Geometry** ribbon tab.



Axis of Rotation Select the global axis of rotation about which the selected components are repeated.

Through group

- **Node** The reference point through which the axis for circular repeat operation passes.

  Click the node selection tool and then click on a node in the view window to select a node in lieu of typing the node number.

- **Point** Type the X Coordinate and Y Coordinate of a an arbitrary reference point (i.e., when you want to use a axis of rotation that does not pass through an existing node).

- **Use this as Reference Point for Beta angle generation** The reference point used for the circular repeat will also be used as the reference point for the beta angle for each member. This allows you to orient all the members towards the central axis of rotation.

Total Angle Type he total sweep angle of rotation between the original structure and the last copied structure.

No of Steps Type the number of copies you want to generate over the specified angle.

Geometry Only Only geometry data is copied. That is, nodes, member and element incidences are generated but no properties, loads, etc. will be assigned to the new objects.

Link Steps Instructs the program to generate new members between each step in the direction of the repeat.

Open Base This option instructs the program to *not* generate linking members at the base of the structure (i.e., the lowest nodes in the selection).

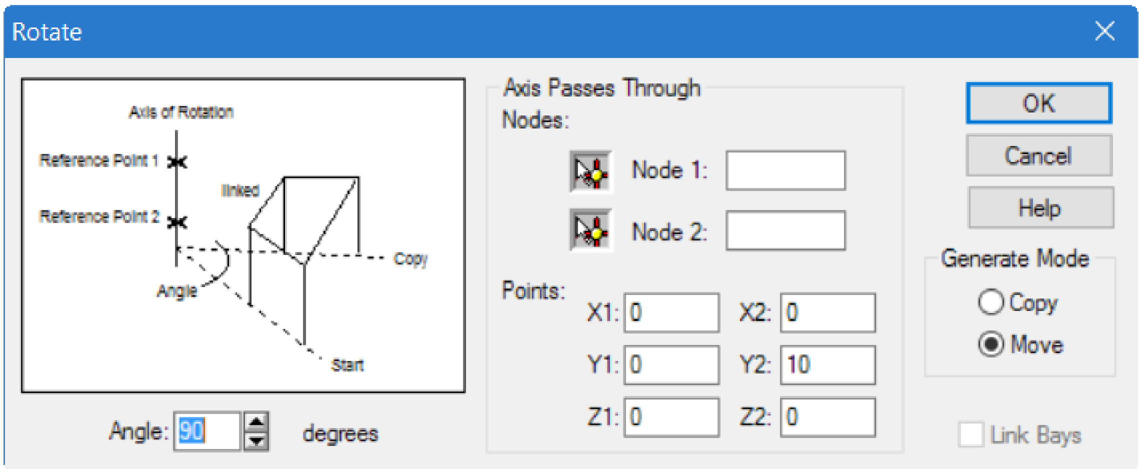
Related Links

- *M. To generate copies of geometry along an arc* (on page 688)

Rotate dialog

Used to rotate the selected portions of the structure about the specified axis through a specified distance. The selected portions may be copied or moved.

Opens when the **Rotate** tool is selected in the **Structure** group on the **Geometry** ribbon tab.



Note: Unlike the **Repeat** > **Circular** tool, the **Rotate** tool allows you to generate copies about any arbitrary axis instead of only one parallel to the global axes.

Angle	Type the angle of sweep.
Axis Passes Through	Define the axis of rotation by either a pair of existing nodes or two arbitrary points.
Generate Mode	The **Copy** option generates new structural elements. The **Move** option changes the coordinates of the selected structural elements.
Link Bays	(Available only for **Copy** option) This option connects the copy with the original geometry by creating new linking members between them.

Related Links

- *M. To rotate selected entities* (on page 687)

Snap Node-Beam dialog

Used to provide a construction grid which can then be used to easily draw beams, plates, or solid elements. Multiple grids may be created. Additionally, this dialog can be used to import a DXF file and use it as a template for a grid or import grid files created in another STAAD.Pro model.

Note: In STAAD.Pro 2007 (and later), grids are saved as separate files (with a `.grd`. file extension) for re-use in other models.

Opens when the **Grids** > **Snap Grid Beam** tool is selected in the **Structure** group on the **Geometry** ribbon tab.

Grid list	Lists all grid definitions in the current STAAD.Pro project.

The grid which is set to active has a check before its name. Only one grid may be active at a time.

Create Opens the **Grid Creation Method** pop-up dialog, which can be used to define either a Linear (on page 2721), Radial grid dialog (on page 2721), or Irregular grid dialog (on page 2722) grid pattern.

Edit When a grid definition is selected in the list, clicking this button opens the **Grid Creation Method** pop-up dialog for editing the definition.

Delete Delete a selected grid definition.

Copy Opens the **Provide Modified Grid Name** dialog, which is used to create a duplicate of the selected grid. Type the new grid name and Click **OK**. Click Cancel to close the dialog without creating a grid copy.

Rename Opens the **Provide Modified Grid** Name dialog, which is used to rename the selected grid. Type the new grid name and Click **OK**. Click Cancel to close the dialog without renaming the grid.

Import Opens the Import Options dialog, which is used to specify the imported grid file type. DXF files or STAAD.Pro Grid files (file extension `.grd`.) may be used.

- **DXF** — This option opens the DXF Import dialog (on page 2723), which is used to select elements from a `.DXF`. file for use as a construction grid in STAAD.Pro. Placement of an imported grid in the view window is controlled using the Imported Grid dialog (on page 2724) pattern.
- **STAAD Grid (.grd)** — Used to Import Grids previously defined in another STAAD.Pro model. Selecting this option opens a browse dialog box to identify a .GRD file created by the Snap Node Grid tool.

Active Grid Labels Setup Local Coordinate **-** this option allows us to view the grid line setting in the local coordinate mode. In this case, the construction grid lines intersecting the origin will be displayed as number zero construction lines and the other construction lines numbered accordingly irrespective of what was set in the global coordinate system.

Roaming grid labels help to identify the relative and exact gridline coordinates for the construction grid lines. This is an invaluable tool especially when the gridlines are skewed about a plane and the coordinates for the gridlines need to be displayed. These grid lines are used when adding nodes, beams, plates or solids by simply snapping on the intersection of gridlines.

Checking the **Rel Coords** box displays how far (either positive or negative) the grid line is from the origin. A “+” or “-” label will be drawn next to the gridline. By checking the **Axis Ids** box, the global axis id (X, Y or Z) which is parallel to the grid line being labeled will be drawn. The display font (size, color and type) can be changed through the **Font** button.

To display the actual coordinates for each grid lines in reference to the origin, choose which ends (Start, End or Both) of the grid line are to be labeled. Since the construction grid lines are only drawn in a plane, there will be a maximum of two directional gridline vectors that can be labeled (XY, XZ, or YZ). The frequency at which the grid labels are drawn can be controlled through the **Freq** spin button. If the frequency is set to 3, the grid labels will only be drawn at every third gridline. The coordinate values in the X, Y and Z directions can be displayed by pressing the appropriate coordinate labels next to the frequency spin button.

Snap to existing nodes too Select this option to select existing nodes for creating the members. If the mouse is clicked close to an existing node, the existing node is selected instead of creating a new node at that location.

Snap Node/ Beam Click to create members by snapping to grid and/or existing nodes. The button is depressed when this function is active. Click again to stop adding beams.

Related Links

- *M. To add a grid for drawing objects* (on page 653)

Linear

The *Linear* tab is meant for placing the construction lines perpendicular to one another along a “left to right - top to bottom” pattern, as in the lines of a chess board.

Related Links

- *M. To add a grid for drawing objects* (on page 653)

Radial grid dialog

Used to generate construction lines in transformed radial coordinates. This is used to create circular type models where members are modeled as piece-wise linear straight line segments.

Note: Though the parameters are entered in radial coordinate notation, the result coordinate values are in Cartesian coordinates. The grid is drawn with straight lines between points along an arc as well. Members drawn between two points on an arc will be straight line . Refer to the Curved Beam tool to add a beam with a radial curve.

Opens when Radial is selected as the type of new grid or an existing radial grid is edited in a Snap Node/<entity> dialog.

Name (Optional) Specify a name by which you can identify the grid in the Snap Node/<entity> dialog.

Plane Used when the grid is to be placed in a global plane. Select the global plane in which the gridlines will lie.

Angle of Plane Choose one of the three options (**X-X**, **Y-Y**, or **Z-Z**) and provide the angle of rotation (in degrees) of the grid plane about that axis.

Origin The (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin.

Type in the location of the origin of gridlines in global coordinate system, in the current length units, or click the  tool to change the location of the grid origin by graphically selecting a existing node.

Construction Lines Start Angle, is the angle in degrees about the orthogonal axis to the plane from the axis first referred to in the definition of the plane. E.g. if the selected plane is X-Y, then the angle is measured about the Z axis (using the right hand rule) from the axis parallel to the X axis.

Sweep is the angle in degrees measured from the start angle which is divided into the selected number of Bays, thus

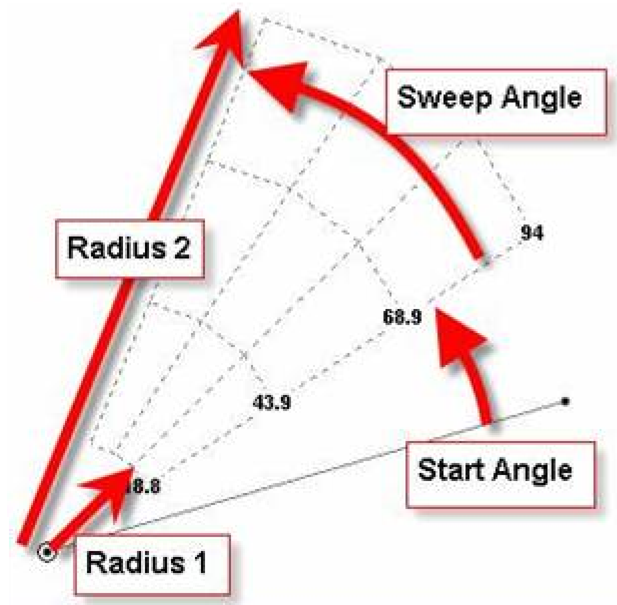


Figure 337: The nomenclature used for Radial grid construction lines

Related Links

- *M. To add a grid for drawing objects* (on page 653)

Irregular grid dialog

Used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane.

Opens when Irregular is selected as the type of new grid or an existing irregular grid is edited in a **Snap Node/<entity>** dialog.

Name	(Optional) Specify a name by which you can identify the grid in the **Snap Node/<entity>** dialog.
Origin	The (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin. Type in the location of the origin of gridlines in global coordinate system, in the current length units, or click the  tool to change the location of the grid origin by graphically selecting a existing node.
Use Arbitrary Plane	This option is used when the grid is to be placed in any arbitrary plane determined by a pair of vectors, which are specified by provided a pair of coordinates entered in global X, Y, and Z values. The vectors are then taken from the origin to each of nodes to define the plane's X and Y axis. **Tip:** These coordinates *do not* need to be existing nodes. Any arbitrary point may be type in.
Plane	Used when the grid is to be placed in a global plane. Select the global plane in which the gridlines will lie.

Angle of Plane	Choose one of the three options (**X-X**, **Y-Y**, or **Z-Z**) and provide the angle of rotation (in degrees) of the grid plane about that axis.

Relative gridline distances	Specify the distances between gridlines, with each distance separated by a space. Values are in the current input units of length. A gridline is always placed at zero in each direction, so there is no need to enter that value. Tabbing (or clicking) between fields updates the grids (if this is the active grid).
	For example, the following X and Y distances provide a grid for drawing `US.1. Plane Frame with Steel Design.std` (in units of feet, with zero angle and origin = 0,0,0).
	`5 2.5 2.5 5 5 2.5 2.5 5`
	`20 15 3 3 3`

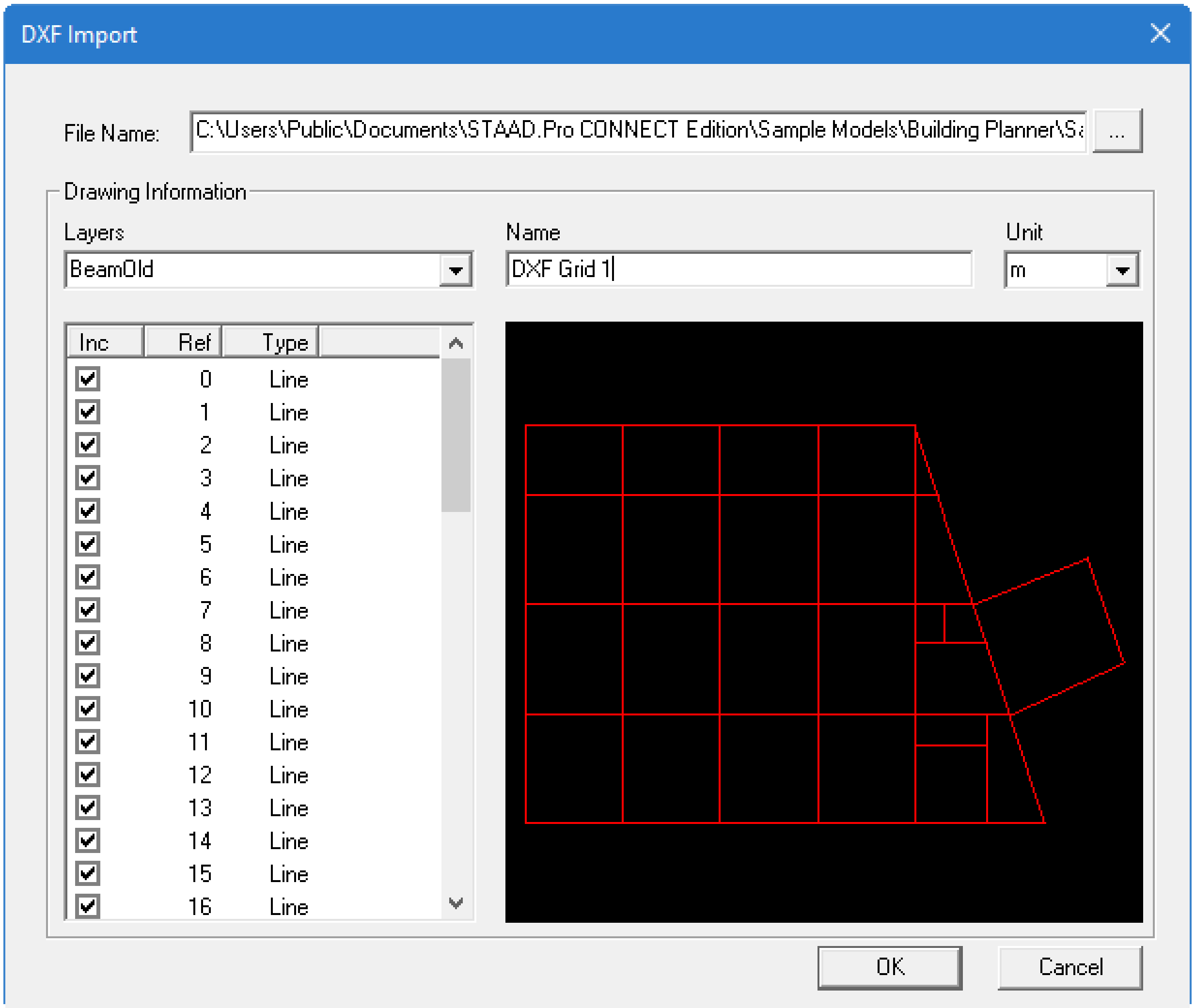
Related Links

- *M. To add a grid for drawing objects* (on page 653)

DXF Import dialog

Used to select drawing elements from a .DXF file for use as construction grids in STAAD.Pro.

Opens when the .DXF option is selected in the **Import Options** dialog.



To select a DXF file click on the **[...]** button and navigate to the required file.

The file will be opened and displayed in the preview window. Individual layers can be turned on and off from the Layers droplist. The individual entities in the selected layers are displayed and can be toggled on or off for import.

By clicking on an entity in the graphical window, the entity is highlighted in the table so that it can be turned off if required.

With the required entities selected, a suitable reference name supplied and unit selected, click on the [Import] button.

Related Links

- *M. To import a DXF file as a grid* (on page 655)

Imported Grid dialog

The data will be imported in the plane in which it was defined in the original DXF. However, if required this can be rotated about any of the global axes. Also, the origin of the grid can be located at any 3D coordinate.

The DXF grid is used just as a manually generated grid. Nodes can be created at the ends and intersections of grid lines.

Note: Curved lines are *not* imported.

Angle of Plane Choose one of the three options (**X-X, Y-Y, or Z-Z**) and provide the angle of rotation (in degrees) of the grid plane about that axis.

Grid Origin Provide the location of the origin of gridlines in global coordinate system in the current length units. One may do this by typing in the coordinates of the origin in the windows provided for the purpose.

The grid origin is the (0,0) position on the grid in the grid coordinate system. Usually, the grid origin coincides with the structure origin.

Click the  tool to change the location of the grid origin by graphically selecting a existing node.

Hide DXF text Select this option to toggle the display of grid labels if they start clashing with the rest of the model.

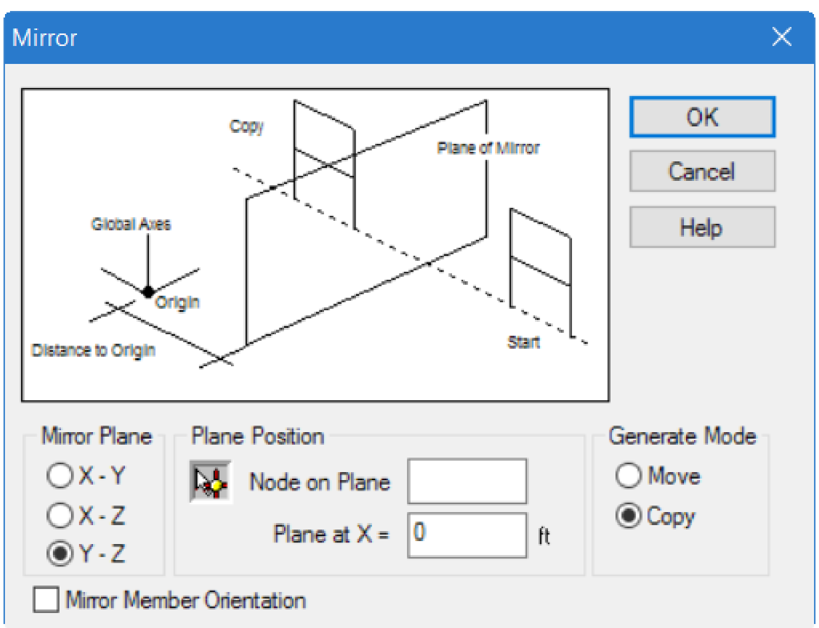
Related Links

- *M. To import a DXF file as a grid* (on page 655)

Mirror dialog

Used to copy or move the selected portions of the structure by “mirroring” about any plane parallel to one of the three global Cartesian coordinates.

Opens when the **Mirror** tool is selected in the **Structure** group on the **Geometry** ribbon tab.



Mirror Plane Choose one of the three global planes to mirror the selected geometry about.

Plane Position
: Used to establish the mirror plane position relative to the selected geometry. Select or type a **Node on Plane** to use an existing node to define the mirror plane.

  Alternately, type a distance from the global origin to the plane along the axis perpendicular to the selected Mirror Plane in the **Plane at X/Y/Z** field.

Mirror Member Orientation
: Select this option to consider the member orientation (Beta angle or member reference point) during mirroring. When this option is selected, the program will attempt to mirror the member orientation also, in addition to the member geometry.

Generate Mode
: The **Copy** option generates new structural elements. The **Move** option changes the coordinates of the selected structural elements.

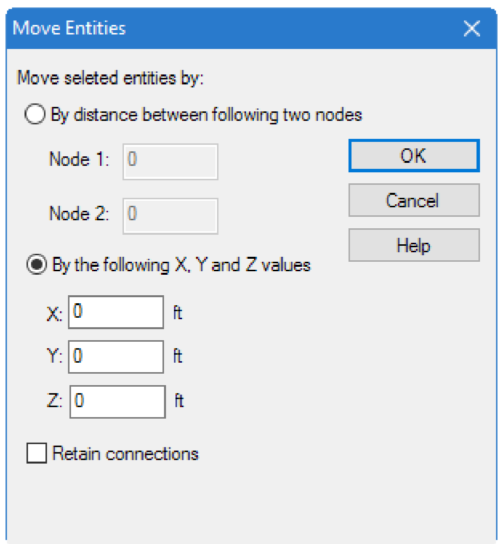
Related Links

- *M. To generate mirror copies of model entities* (on page 689)

Move Entities dialog

Used to specify the translational offset for moving a selection of model objects.

Opens when the **Move Node** tool is selected in the **Node** group on the **Geometry** ribbon tab.



Note: The controls in this dialog are analogous to the **Move Beams Selection** dialog (opens with the **Move Beam** tool is selected), the **Move Plates Selection** dialog, and the **Move Solids Selection** dialog.

By distance between following nodes
: uses two node numbers and determines the vector between them. The selected entities are moved along a parallel vector of the same distance

By following X, Y and Z values
: uses the Cartesian coordinates specified to create a vector

Retain connections
: Set this option to retain the connectivity between objects. The connection objects will distort by the resulting translation vector.

> **Note:** The model geometry will be adjusted to some degree, even if this is not selected, when nodes are the entities being moved as they member, plate, surface, and solid geometry is defined by node location. However, if you are moving a member, plate, surface, or solid, then selecting this option will move the associated nodes and thus retain the connection with adjacent entities (though their geometry will stretch, warp, or shift as a result of the move).

Related Links

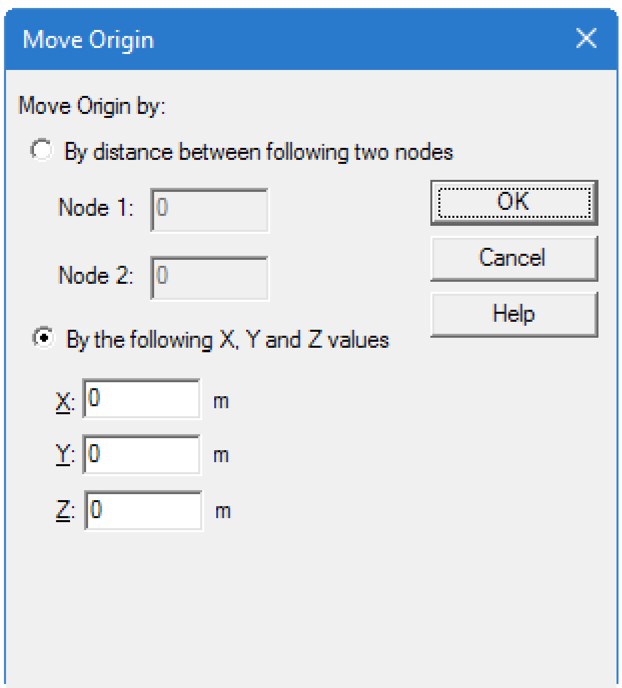
- *M. To move selected objects* (on page 685)

Move Origin dialog

Used to specify the translational offset for moving a the model origin.

Opens when the **Move Origin** tool is selected in the **Node** group on the **Geometry** ribbon tab.

> **Note:** All model geometry is updated relative to the new origin location.



> **Note:** The controls in this dialog are analogous to the **Move Beams Selection** dialog (opens with the **Move Beam** tool is selected), the **Move Plates Selection** dialog, and the **Move Solids Selection** dialog.

By distance between following nodes	Uses two node numbers and determines the vector between them. The origin is shifted along a parallel vector of the same distance.
By following X, Y and Z values	uses the Cartesian coordinates specified to create a vector

Related Links

- *M. To move the model origin* (on page 686)

Select Node dialog

Used to select a node number you want to keep in the model. .

Opens when the **Merge Nodes** tool is selected in the **Nodes** group on the **Geometry** ribbon tab.

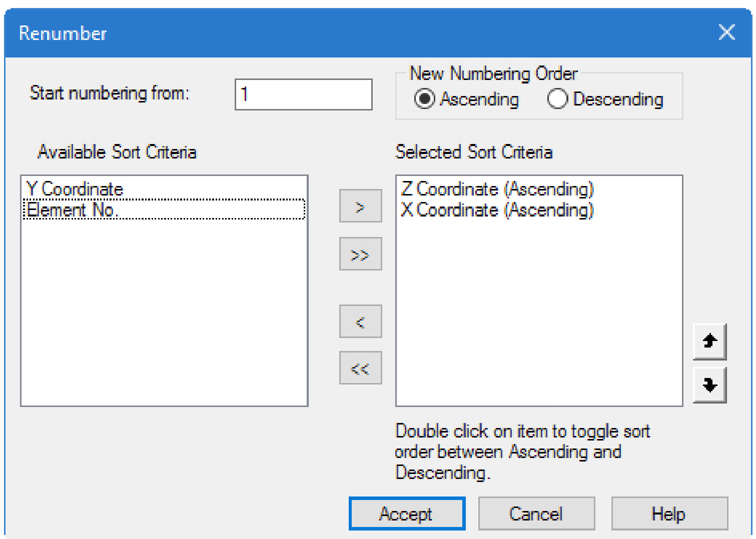
Node To Keep	The nodes in the selection set are listed. Select the node number you want to keep. Other selected nodes will be merged to this node number

Renumber dialog

Used to renumber selected entities starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.

Opens when one of the following tools is selected on the **Geometry** ribbon tab:

- the **Renumber Nodes** tool in the **Node** group
- the **Renumber Beams** tool in the **Structure** group
- the **Renumber Plates** tool in the **Plate** group
- the **Renumber Solids** tool in the **Solid** group



Start number from	Specify the starting number of the renumbered entities. Depending on the **New Numbering Order** option selected, the new entity numbers will ascend or descend from this number.
New Numbering Order	Select either an **Ascending** or **Descending** order to control the subsequent entity numbers.
Available Sort Criteria	Sort criteria can be used to control the order in which selected entities will be numbered once the procedure is complete.

List Operators

Click this button…	to…
>	Add the selected sort criteria to the Selected Sort Criteria list
>>	Add all sort criteria to the Selected Sort Criteria list.
<<	Remove all entries in the Selected Sort Criteria list.
<	Remove the selected entry from the Selected Sort Criteria list.

Selected Sort Criteria Items added to this list will be used as criteria for sorting the numbered items. The order of operations is that the entities are renumbered starting with the lowest criteria on the list. Once this is complete for a criteria, the next higher criteria in the list will be used.

Use the Move Up or Move Down buttons to change the order of the items in the list.

Note: You can change the numbering order (ascending or descending) of individual sort criteria by double-clicking its entry in this list.

Accept Renumbers the selected model entities based on the specified parameters.

Caution: This operation cannot be undone.

Cancel Closes the dialog without renumbering any model entities.

Help Opens the STAAD.Pro help window.

Related Links

- *M. To renumber selected beams* (on page 663)

Define Member Attributes dialog

Used to pre-define member attributes such as properties, materials, beta angles, or release specifications for member attribute sets. Members which are then added to the model will have these properties by default.

Opens when the **Set New Member Attributes** tool is selected from the **Add Beam** drop-down in the **Beam** group on the **Geometry** ribbon tab.

Select Template Contains a list of named templates, included user-defined templates created by clicking **New**.

New Click to create a new template. Type a name and click OK.

Save Save changes made to the currently selected template.

Rename Click to rename the currently selected template.

Delete Click to delete the currently selected template. You will be prompted to confirm this action.

Attributes group

Member Property
Select the Use Property option to predefine a Member profile. The Select Property: drop down list contains all the profiles currently being used in the STAAD.Pro command input file.

Beta Angle
Select the Use Beta Angle option to predefine a angle of rotation about the member's longitudinal axis. Options are available to use the Angle, the RAngle, or enter an Angle in Degrees.

Member Release
Select the Use Release at Start and/or Use Release at End options to predefine a member end release specification for either the start or end of the member(s). The Release at Start and Release at End drop-down lists contain all the member end releases currently being used at the respective member ends in the STAAD.Pro command input file.

Assign these attributes while creating new members
Select this option to ensure that these attributes will be assigned to any members which are created henceforth.

Related Links

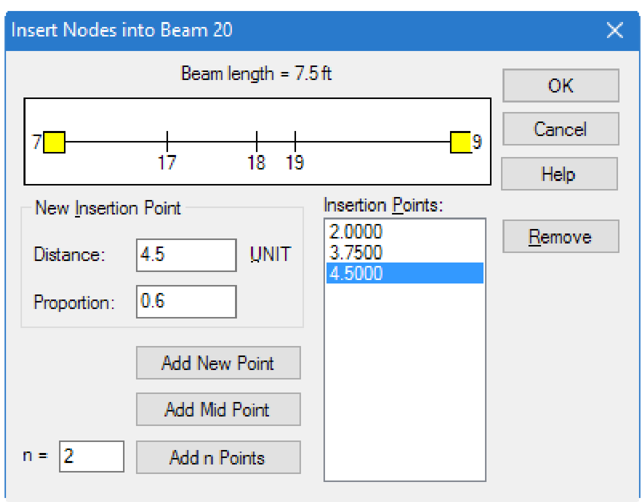
- *M. To set attributes for new beams* (on page 657)

Insert Nodes into Beam # dialog

Used to insert one or more nodes at specified distances along selected members.

Opens when the **Insert Node** tool is selected in the **Structure** group on the **Geometry** ribbon tab.

One or more beams must be selected for this tool be active.



Beam Length
Lists the distance from node A to node B along the beam to be split.

New Insertion Point
Provide the **Distance** from the start node (as shown in the associated graphics) of the member in current length units. Alternatively, provide the **Proportion** of the total length of the member to position the new node. Click **Add New Point** to add the node.

Add New Point	Adds the node indicated by the **Distance** or **Proportion** value.
Add Mid Point	Splits the member into two equal segments.
Add n Points	To divide the beam in a number of equal segments, provide the number of intermediate points in the **n =** field and click **Add n Points**. **Note:** This value should be an integer.
Insertion Points	A list of the locations of the newly created points is displayed here, shown as the distance from the start node of the member.
OK	Accept the added nodes and closed the dialog.
Cancel	Close the dialog without adding nodes.
Help	Opens the Help window.
Remove	To remove a node from the list of inserted nodes, highlight the desired node and click on this button.

Related Links

- *M. To insert a node in a single member* (on page 682)

Insert Node / Nodes dialog

Used to insert node(s) into multiple members simultaneously following the same prescribed method.

Opens when the **Insert Node** tool is selected in the Beam group on the Geometry ribbon tab (with multiple beams already selected).

New point by distance	Specify the distance in current length units at which the beam is to be split. The value for the distance is entered in the Distance edit box and is measured from the start node of the beam.
New point by proportion	This option allows the users to specify the distance in terms of a ratio. For example, to split a beam at the midpoint, enter 0.5 as the proportion. To split the beam at quarter points, use a proportion value of 0.25.
Add mid point	Splits the beams at their midpoints.
Add 'n' points	To split a beam by inserting **n** number of points, use this option. The beams are split up into n + 1 segments.
OK	Accept the added nodes and closed the dialog.
Cancel	Close the dialog without adding nodes.
Help	Opens the Help window.

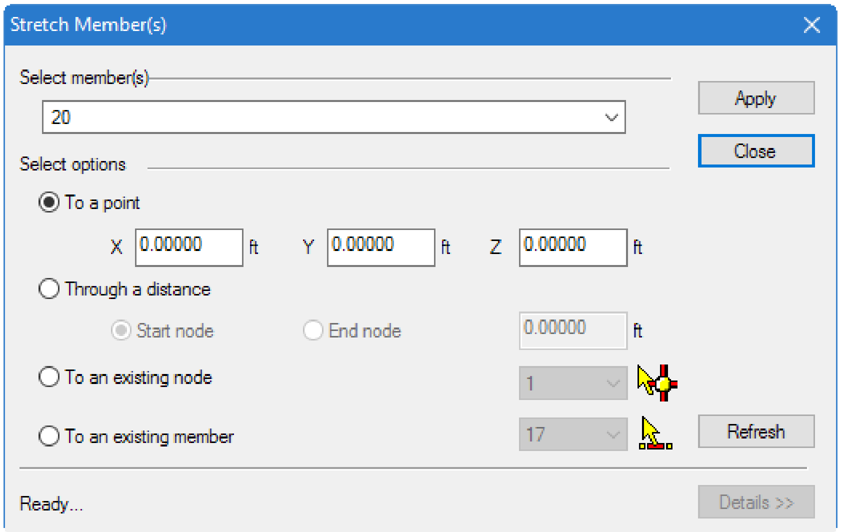
Related Links

- *M. To insert a node in multiple members* (on page 683)

Stretch Member(s) dialog

Used to increase the length of a member in various ways.

Opens when the **Stretch Members** tool is selected in the **Beams** group on the **Geometry** ribbon tab.



Select member(s)

Multiple members can be selected simultaneously for stretching. However, whether they will get stretched or not depends upon the type of method used in stretching as described later. If you wish to remove one or more members from the list, uncheck the corresponding boxes within the Select member(s) drop-down list.

Select options

Select the method to use for stretching the selected members:

To a point

Specify the coordinates in current length units of the point to which one of the ends of the member are to be moved. The point must lie on the axis of the member being stretched, or else, the stretching will not be performed. The program automatically determines which of the two ends of the member is to be moved. So, this method involves

- **i.** determining if the point lies on the axis of the member(s) being stretched
- **ii.** determining which end to move for the member(s) which satisfy criteria (i)
- **iii.** replacing the corresponding nodal coordinates with those of the desired point

Through a distance

Specify whether the start or end node will be stretched and the distance by which to stretch this end. In this method, all members selected for this operation will see the change in length.

You must create a node a resulting intersection point separately using the **Intersect Beams** tool.

To an existing node	This is very similar to the **To a point** method. The only difference is that instead of explicitly specifying the coordinates of the desired point, that point is already available for identification through its node number. A list of all nodes in the model is available in the drop-down list. Alternately, click the Node selection tool to select a node graphically.
To an existing member	If two members lie within a plane, this method may be used to stretch the first member to meet the second member. The second member will be automatically split at the intersection point into two segments. A list of all members in the model is available in the drop-down list. Alternately, click the Member selection tool to select a member graphically.

Details This button toggles the display of the detailed actions taken during the stretch operation.

Tip: If this button is disabled, then the selected stretch operation is invalid.

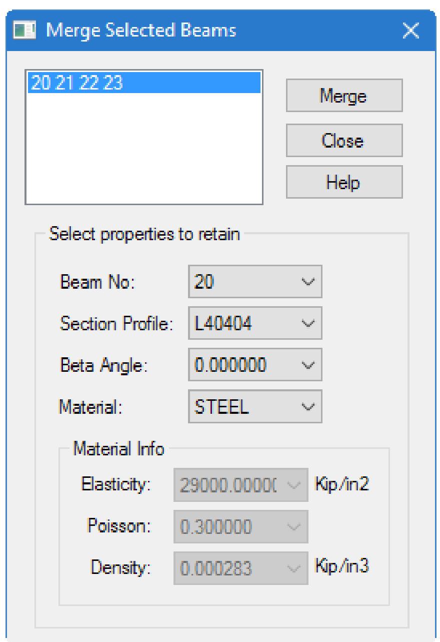
Related Links

- *M. To stretch a member* (on page 662)

Merge Selected Beams dialog

Used to join two collinear beams and replace them with one beam. The intermediate node(s) (common node) is removed from selected members and the incidences are redefined.

Opens when the **Merge Selected Beams** tool is selected in the **Beam** group on the **Geometry** ribbon tab.



Beam No Select the beam no to use for the merged beam.

Section Profile Select the section property to use for the merged beam.

Beta Angle	Select the local x axis rotation value to use for the merged beam.
Material	Select the material definition to use for the merged beam. Material data is listed below. **Note:** If the members are defined using material constants instead of material definitions, then the individual constants will be available to select for the merged beam instead.

Related Links

- *M. To merge two or more members* (on page 662)

Parametric Models dialog

Used to automatically generate finite element meshing for defined boundary areas.

Opens when the **Parametric Models** tool is selected in the **Plate** group on the **Geometry** ribbon tab.

Preview Models list

A list of mesh prototypes are listed here. Multiple settings may be experimented with here to obtain the desired surface mesh before merging with the base model. Selecting the meshed surface name displays the overall model parameters. Selecting a “child” item in the list (e.g., boundary, openings, etc.) displays the parameters for that item.

- model name

  Selecting the name of the model displays the overall model parameters, which were initially set in the **Add Parametric Model**
  dialog.

  - **Boundary Connectivity -** Two methods are available for parametric model boundaries:

    B - User Defined - this method uses the defined number of segments per boundary, which can be specified for each Boundary Point/Edge as needed
    C - Automatic - this method optimizes the boundary segmentation
  - **Target Element Size -** the target element size to use for meshing, in the given units of length
  - **Mesh the selected plate element -** the shape of the element used, which is restricted by the selected meshing method
  - **Meshing Method -**

    Standard - uses quadrilateral elements
    Basic - uses triangular elements
- **Boundary**

  Selecting a Boundary Point/Edge displays the points' 3D coordinates (which cannot be modified) and two additional parameters which can be edited:

  - **Division -** When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.
  - **Density -** When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the boundary point.

- **Openings**

  Openings can be added by selecting the Openings heading and clicking the **Add** button. Either polygonal or circular openings can be added as described above. Both opening types have options that can be edited.

  - **Polygonal**

    Selecting a Polygonal opening definition in the dialog displays the control point coordinates that form the opening (which cannot be edited). Each point includes two additional parameters which can be edited:

    Division - When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.

    Density - When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the opening point.
  - **Circular**

    Selecting a Circular opening definition in the dialog displays the circle center coordinates that form the opening (which cannot be edited). Each point includes three additional parameters which can be edited:

    Radius - This defines the size of the opening. This value is used by all Mesh Types.

    Division - This is used to define the number of segments that form the circle and is used by all Mesh Types.

    Density - When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around each point that forms the opening.
- **Regions**

  Selecting a Region definition in the dialog displays the control point coordinates that form the region (which cannot be edited). Each point includes two additional parameters which can be edited:

  - **Division -** When a Mesh Type B) has been selected, this value is used to identify the number of divisions along the boundary edge from the selected node to the next defined point on the boundary.
  - **Density -** When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the point on the region.

  Note: The control points for an `OPENING` or `REGION` cannot lie on the surface boundary.
- **Density Points**

  Selecting a Density Point definition in the dialog displays the coordinate of that point, which cannot be edited and a single parameters which can be edited:

  - **Density -** When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the density point.
- **Density Lines**

Density lines are defined by two points. Selecting a Density Line definition in the dialog displays the two coordinates of the line which cannot be edited and two additional parameters which can:

- **Division -** This is displayed on the first point that defines the density line, but applies to the line between both points. This value is used to identify the number of divisions along the density line and is used by *all* mesh types.
- **Density -** When a Mesh Type C) has been selected, this value is used as a biasing parameter. The larger the value, the denser the mesh around the point on the density line.

Parameters table	Displays the parameters for the selected mesh model, Opening, Region, Density Point, or Density Line. Changes made here can be saved using the **Apply** button.
Add...	Used to add a new mesh model to the list or to add a new Opening, Region, Density Point, or Density Line to the current mesh model.
Delete	Removes the selected mesh model from the Preview Models list or removes the selected Opening, Region, Density Point, or Density Line from the current mesh model. **Caution:** No confirmation is used to delete mesh models or their sub-components. This action cannot be undone.
Apply	Updates the selected mesh model with any changes made to the settings.
Merge Mesh	Commits the selected mesh model to the STAAD.Pro input file base model. **Caution:** Until this button has been clicked for one or more mesh models, no data from this page will be saved.

Related Links

- *TR.14.1 Parametric Mesh Models* (on page 2227)
- *M. To create a parametric surface model* (on page 671)

New Mesh Model dialog

Used to name a new mesh model.

Opens when **Add** is clicked to create a new mesh model in the Parametric Models dialog (on page 2734).

Name	Enter a unique name here to identify the mesh model in the Parametric Models dialog Preview Models list.
Use nodes and beams that occur on or inside the outer boundary as additional density points and lines?	Select this option to have the program auto-detect points on the outer boundary not explicitly clicked when defining the edge of the mesh boundary. The program will also used any beams and nodes inside the boundary as density lines and points, respectively.
OK	Closes the dialog and allows you to begin defining points around the mesh boundary.
Cancel	Closes the dialog without creating a new mesh model.

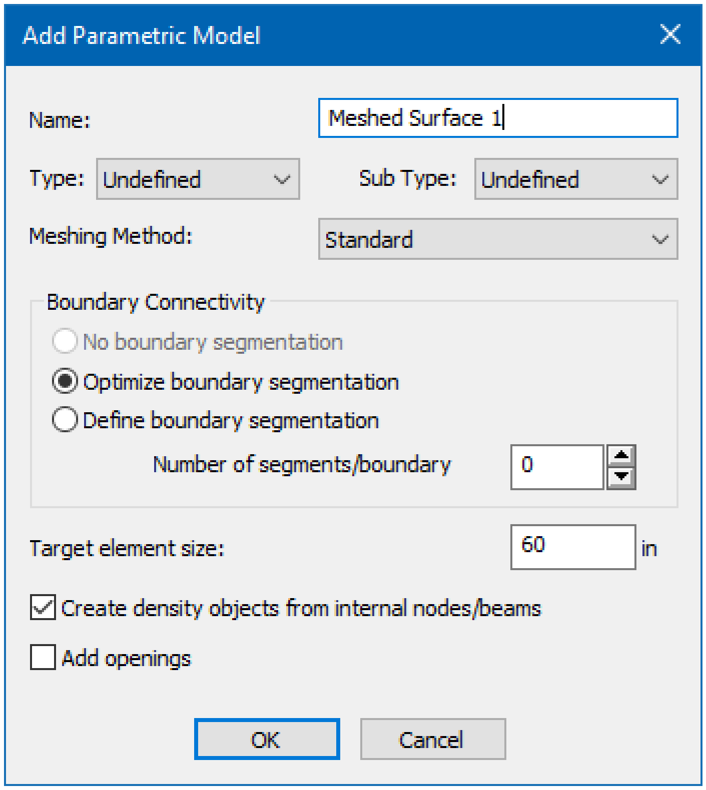
Related Links

- *M. To create a parametric surface model* (on page 671)

Add Parametric Model dialog

Used to specify the parameters used for automatically generating a finite element mesh over boundary area.

Opens when a new mesh boundary has been defined from the Parametric Models dialog (on page 2734).

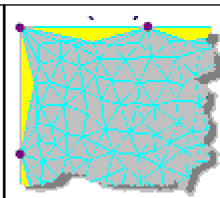
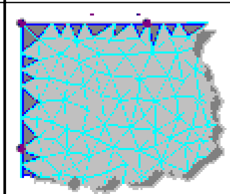
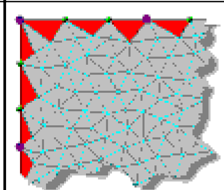


Name	Type a string used to identify the meshed surface.
Type	Select an optional description for the surface type: • Undefined • Wall • Slab
Sub Type	Select an optional description for the selected **Type**: • Wall_Tank (Wall only) • Slab_Tank (Slab only)
Meshing Method	Select one of the following mesh methods: • **Basic -** Can make use of any of the three **Boundary Connectivity** options and uses triangular elements.

- **Standard -** Can use type either optimized or defined **Boundary Connectivity** options and uses quadrilateral elements.

Boundary Connectivity

Select how the program should sub-divide the boundary edges between the selected nodes:

No boundary segmentation - Meshing will be done without creating any nodes along the edges of the boundary we defined. Thus, the entire distance between two sequential nodes will form the side of one element. In most cases, this can lead to poor quality elements, and is best avoided.	 No divisions
Optimize boundary segmentation - (Default, recommended) Automatic boundary divisions to the target element size.	 Optimized
Define boundary segmentation - Each boundary between the selected nodes will be sub-divided into the specified number of segments given in **Number of segments/ boundary**.	 Equal number of divisions

Target element size

Specify a target dimension to use as a guideline when automatically generating the mesh. This value is nominally used when dividing the boundary region into finite elements, but the element size is also determined by other factors such as the distance between boundary and other density points..

Create density objects from internal nodes/ beams

Check this option to have additional nodes created along any members or defined nodes that lie in the defined surface area.

Add openings

Check this option to specify openings within the surface after you click **OK**.

OK

Closes the dialog and generates the mesh.

Cancel

Closes the dialog without generating a meshed surface.

Related Links

- *M. To create a parametric surface model* (on page 671)
- *M. To create a parametric surface model* (on page 671)

Define Meshing Region dialog

Used to specify data such as the number of divisions along each side of the polygon, and whether any cutouts or holes are to be specified in the polygon.

Opens when a polygonal meshing is being defined.

Polygonal Plate tree

Entries in the tree represent the Boundary (polygonal surface edges) and the included Holes, if any.

Selecting the HOLES entry allows you to add holes or delete all holes either from the right-click pop-up menu or using the HOLES tools:

Table 295: HOLES tools in the Define Meshing Region dialog

Tool	Description
 Add New Hole	Adds a new hole for the current boundary region. Coordinates for the hole corners must be defined.
 Delete All Holes	Deletes all holes currently added in the boundary region.

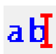
Table

Each row in the table represents a corner node of the selected Boundary or Hole, along with the number of edge divisions and bias for the edge between the node on this row and the previous row.

- X / Y / Z — Global coordinates of the Boundary or Hole corner.
- Div. — Number of elements will be created between the current node represented by the current row and that of the previous row (division).
- Bias — Used to control the variation in division length along the edge. Represents the ratio of length of the last element to the first element along a side. That is, the last element will be (bias value) times the length of the first element. A value of unity (1) results in equal divisions along a side.

Table 296: Table tools in the Define Meshing Region dialog

Tool	Description
 Add New Row	Adds a new row to the bottom of the table with default values.

Tool	Description
 Delete Row	Removes the currently selected row from the table.
 Rename Hole	(When hole is selected) Used to provide a user-defined name for the hole.
 Delete Hole	(When hole is selected) Removes the current selected hole from the polgyonal plate tree.

OK Generates a meshed surface with the specified parameters.

Cancel Closes the dialog without generating a meshed surface.

Reset Undoes any changes made to the dialog.

Select Meshing Parameters dialog

Used to name and specify parameters for a quadrilateral plate super-element.

Opens when a quadrilateral meshing is selected.

Model Name Specify an optional, user-defined name for the quadrilateral super-element.

Corners The X, Y, and Z coordinates of each corner (the selected nodes) are labeled A through D.

Length, Bias & Division For each side connecting two Corners, the straight line distance is provided. You may also edit the Bias and Divn. values for each side to control the meshing.

- Div. — Number of elements will be created between the current node represented by the current row and that of the previous row (division).
- Bias — Used to control the variation in division length along the edge. Represents the ratio of length of the last element to the first element along a side. That is, the last element will be (bias value) times the length of the first element. A value of unity (1) results in equal divisions along a side.

Element Type Select either a Triangle or Quadrilateral element for the finite elements which compose the super-element mesh.

Apply Closes the dialog and generates a quadrilateral mesh with the specified parameters.

Cancel Closes the dialog without creating a quadrilateral mesh.

Define Plate Object Property dialog

Used to define various attributes of the plate to be pre-defined.

Opens when the **Add Plate > Set New Plate Attributes** tool is selected in the **Plate** group on the **Geometry** ribbon tab.

Use Property	Select the Use Property option to predefine a Plate thickness. The **Select Property** drop down list contains all the thicknesses currently being used in the STAAD.Pro command input file. Additional thicknesses can be added using the Create New Property button.
Create New Property	Opens the **Plate Element/ Surface Property** dialog to the Plate Element Thickness tab, which is used to define the thickness of the plate at each corner node.
Use Material	Select the Use Material option to predefine a plate Material. The **Select Material** drop down list contains all the materials currently being used in the STAAD.Pro command input file. Additional materials can be added using the using the Create New Material button.
Create New Material	Opens the **Isotropic Material** dialog, which is used to define Material properties for a new isotropic material type or select a predefined, common isotropic material.
Use Release	Select the Use Release option to predefine a set of plate releases. The **Plate Release** drop-down list contain all the plate releases currently being used in the STAAD.Pro command input file. Additional releases can be added using the Create New Releases button.
Create New Release	Opens the Plate Specs dialog, which is used to define the degrees of freedom to be released at each node of the plate as well as define the plate to e plane stress, no in-plane rotation, or no stiffness.
Assign these attributes while creating new plates	Select this option for the program to recognize the pre-defined attributes. Any new plate element created from here on (whether created individually, through a mesh or Structure Wizard) will now possess these attributes.

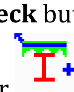
Related Links

- *M. To set new plate attributes* (on page 667)

Composite Deck dialog

Used to create and define composite decks.

Opens when the **Composite Deck Layout** tool is selected in the **Composite Deck** group on the **Geometry** ribbon tab.

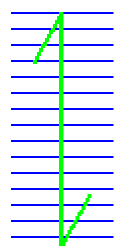
Composite Deck list	All composite deck groups and their constituent members are listed here.
New Deck options	• Clicking on Nodes - Used to define the deck by clicking on the nodal points that comprise the deck boundary. Click on the **Create New Deck** button to begin selecting the deck boundary using the composite boundary cursor . Once the deck is created, it will be shown with a hatch pattern. • Use Selected Beams - Used to define the deck by selecting the beams that comprise the deck boundary. Using the Beams cursor, select the beams which will define the outer boundary of the deck.

Create New Deck
Used to define the deck in by the selected method. Once the deck is identified by any one of the aforementioned methods, a dialog box as shown below will prompt for the deck name. Enter an appropriate deck name in the edit box provided.

The deck will then be defined and the deck name will appear in the **Composite Deck** box as shown below.

Select the deck in the tree control to define the properties of the deck. The various options in the **Composite Deck** box are explained as follows.

Create Direction
Used to define the deck direction (direction of the longitudinal flutes or ribs in the deck) by highlighting any two beams and clicking the **Create Direction** button. The deck direction will be defined to run perpendicular to the two beams that were selected. A symbol will appear to indicate the direction of the deck. This symbol will be placed at the centroid of the deck area.



Once the direction of the deck is specified, the beam numbers which comprise the deck will be displayed as child elements of the deck group.

Deck hash color
Click the color square (adjacent to the Create Direction button) to open a color picker dialog. Select a custom color to hash the selected deck or any newly created decks. Each deck can have a different color.

> **Tip:** Use different colors to quickly identify composite decks with different properties or directions.

Deck Properties

The following options appear in the Composite Deck dialog when a deck group is selected.

Update Deck Property
Synchronizes the graphical window and updates any changes made to the deck definition.

Concrete Properties
Specify the concrete thickness above the flutes, the unit weight of concrete and the grade of concrete in the respective edit boxes.

Rib Properties
Rib properties can now be defined either using the database or by entering the properties manually.

Use Database
A drop-down list of the various standards incorporated in STAAD.Pro. The program includes three standard catalogs: ASC™, Vulcraft™, and VERCO™. The rib properties will be used from the database provided.

Apart from using the available standards, you may also define custom rib properties by providing the **Rib Width** and **Rib Height**.

Rib Width
Width of the rib in this box. If a database deck is selected, this value is updated.

Rib Height The height of the rib in this box. If a database deck is selected, this value is updated.

Stud Diameter Specify a diameter for the Stud by choosing the **Custom** option or select from the three available diameters: 3/4", 5/8", 1/2".

Stud Length Specify the length (height above deck pan) of the stud in this box.

Shored Choose the appropriate radio button depending on whether or not shoring is to be used during construction.

Beam Properties

The following options appear in the Composite Deck dialog when a component beam is selected.

Current Property The current section and material of the selected beam is displayed.

Add / Change Property Opens the appropriate Section Profile Tables dialog (on page 2793) which is used to select a steel section and material.

Effective Width The effective width for each beam is automatically calculated by STAAD.Pro and is displayed in the **Effective Width** box. The user can modify the effective width for any individual beam by inputting the value in the **Effective Width** box and clicking on the update button.

Related Links

- *M. To create a new composite deck from perimeter beams* (on page 676)
- *M. To specify a direction for the composite deck ribs* (on page 677)
- *M. To assign composite deck properties* (on page 677)
- *M. To modify composite steel beam properties* (on page 678)
- *M. Composite Decks* (on page 676)
- *G.6.7 Composite Beams and Composite Decks* (on page 2119)
- *M. To add a floor load or one-way load* (on page 840)

View tab

Table 297: Labels group

Tool name	Description	Shortcut
 **Labels Settings**	Opens the **Diagrams** dialog to the **Labels** tab, which is used to customize the view of the structure by setting different view-related parameters.	

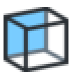
Tool name	Description	Shortcut
 **Node Labels**	Select any of the label types from the drop-down list to turn the display of that label on or off. **Tip:** Additional labels are controlled on the **Diagrams** dialog **Labels** tab.	**<Shift+N>**
 **Beam Labels**		**<Shift+B>**
 **Plate Labels**		**<Shift+P>**
 **Solid Labels**		**<Shift+C>**
 **Individual Node Labels**	When the object label type is activated, use these tools to show or hide the individual labels of objects. This can be useful for large models or views where you wish to display multiple types of labels simultaneously. **Note:** These tools require you to select the **Always Use Current Label Settings** option and then check the **Use Partial Labeling Mode** option on the **Diagrams** dialog **Labels** tab.	
 **Individual Beam Labels**		
 **Individual Plate Labels**		

Tool name	Description	Shortcut
Individual Solid Labels		

Table 298: Tools group

Tool name	Description	Shortcut
Zoom Window	Used to define the boundaries of a rectangular area of the active view to be displayed within the current view.	
Whole Structure	Fits the current view to display the entire structure limits. Resets view rotation to orthogonal.	
Zoom In	Zoom in the current view by a preset percentage magnification.	scroll wheel up
Zoom Out	Zoom out the current view by a preset percentage magnification.	scroll wheel down
Pan	Used to view a different part of the design without changing the view magnification. **Tip:** Press and hold the middle mouse button (typically the scroll wheel button) to quickly pan in the view window.	
Zoom Extents	Adjusts the view magnification so that the entire model is visible in the view.	

Tool name	Description	Shortcut
Zoom Factor	Opens the **Zoom Factor** dialog, which is used to zoom in or out of the structure by specifying a magnification factor. Factors less than one Zoom out.	
Zoom Previous	Undoes the last **Zoom Factor** or **Zoom Window** operation.	
Dynamic Zoom	Used to define the boundaries of a rectangular area of the active view to be displayed within a new view. The previous view window highlights the location of the zoomed portion.	
Magnifying Glass	Provides a magnified portion of the current view window when you click and drag the pointer.	
Isometric View	(Default view) View the model as an isometric projection.	
Front View	View the structure model from the positive Z axis.	
Left View	View the structure model from the negative X axis.	

Tool name	Description	Shortcut
 **Top View**	View the foundation mode in plan; from the positive Y axis.	
 **Back View**	View the structure model from the negative Z axis.	
 **Right View**	View the structure model from the positive X axis.	
 **Bottom View**	View the structure model from the negative Y axis.	
 **Rotate Up**	Rotate the structure model forward about the X axis.	**<↑>** (Up arrow key)
 **Rotate Left**	Rotate the structure model forward about the Y axis.	**<←>** (Left arrow key)
 **Spin Left**	Rotate the structure model forward about the Z axis.	**<Ctrl+←>** (Ctrl +Left arrow keys)

Tool name	Description	Shortcut
Rotate Down	Rotate the foundation mode backward about the X axis.	**<↓>** (Down arrow key)
Rotate Right	Rotate the structure model backward about the Y axis.	**<→>** (Right arrow key)
Spin Right	Rotate the structure model backward about the Z axis.	**<Ctrl+→>** (Ctrl +Right arrow keys)
Toggle View Rotation Mode	Used to select an node as the center of rotation. When toggled on, pressing **<Ctrl+Shift>** and clicking a node will set that node as the center of rotation.	
Orientation	Opens the **Orientation** dialog, which is used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.	**<F4>**
Always Fit in Current Window	Instructs the program on what guidelines to use when drawing a selected set of objects on the screen. It displays the selected portion of the model to a size governed by optimum usage of the dimensions of the current window. When you select various tools the program or switch workflows the size of the drawing window frequently changes. With this tool turned on, the model or selected portions of it, will be drawn in such a manner that all the entities will be drawn within the bounds of the drawing area. This means, the size to which the entities are drawn will correspondingly increase or decrease. With this tool turned off, the size of the entities will remain constant, but that means it may or may not fit within the bounds of the drawing window.	

Table 299: Views group

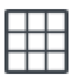
Tool name		Description
 **Open View**		Opens the **Open View** dialog, which is used to open a previously saved structural view window.
 **New View**		Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.
 **Selected Objects**		Hides all objects that are not part of the current selection.
 **View Management >**	**Detach View**	Used to delete any saved or active view window.
	Add to View	Used to add geometry to any saved or active view window.
	Save View	Opens the **Save View As** dialog, which is used to save the current view of the structure with a name. Type a name for the view. This view may be opened later for performing verification/visualization operations.
	Rename View	Opens the **Rename View** dialog, which is used to rename an existing view. Type a **New Name** for the view. The old name is shown for reference.

Table 300: Options group

Tool name	Description
 **Display Options**	Opens the **Options** dialog, which is used to change the length and force units for various values which can be displayed in the graphics window, such as member properties, material constants, load magnitudes, plate stresses, etc.
 **Set Structure Colors**	Opens the **Color Manager** dialog, which is used to set colors for highlight, entities, and results..
 **Structural Tooltip Options**	Opens the **Tool Tip Options** dialog, which is used to set tooltip options for structural elements.

Table 301: Windows group

Tool name	Description	Shortcut
 **Cascade**	Stacks view windows in order with the active window entirely visible and the title bar of each remaining window visible.	**<Shift+F5>**
 **Tile Horizontal**	Tiles view windows vertically, with each window in a landscape (horizontal layout). This is a quick way to clean up the screen.	**<Shift+F4>**
 **Tile Vertical**	Tiles view windows horizontally, with each window in a portrait (vertical layout). This is a quick way to clean up the screen.	**<Ctrl+Shift+F4>**

Tool name	Description	Shortcut
 **Structure Only**	Closes all open view windows, non-modal dialogs, and tables and opens a View window of the full structure expanded to the full program window.	
 **Window**	A list of all open and saved Views is displayed here. Select any number to make this the active view.	
 **Tables**	Opens the **Tables** dialog, which is used to display and close different tables, such as Node coordinates, Beam incidences, Node displacements, etc. irrespective of the current page.	
 **3D Rendering**	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.	**<Ctrl+4>**

Diagrams dialog

Used to customize the view of the structure by setting different view-related parameters.

Opens when either:

- the **Labels** tool is selected in the **Labels** group on the **View** ribbon tab

- when either **Labels** or **Structure Diagrams** is selected from the right-click pop-up menu in the View area

Structure tab

Used to set up structural view parameters.

3D Sections The options in this group control how the members are displayed (only one may be selected).

- **None** - Displays the structure without displaying the cross-sectional properties of the members and elements.
- **Full Sections** - Displays the 3D cross-sections of members, depending on the member properties.
- **Sections Outline** - Displays only the outline of the cross-sections of members.

View The options in this group allow additional view-related operations on the structure.

Fill Plates/Solids/Surface	Fills up the plate and solid and surface elements, if present.
Hide Plates/Solids	Hides all plate and solid elements from the view.
Hide Structure	Hides the entire structure from view. This option may be used to switch off the original structure view while displaying the deflected shape of the structure or the mode shapes.
Show Center Lines	Displays the centerlines of the members.
Shrink	Displays the individual structural elements detached from each other and helps to view their connectivity. The individual members are not drawn to full length or full width but shrunk by a percentage provided in the associated edit box.
Perspective	Change current view to perspective.
Sort Geometry	Changes the display of plate and solid element results so that different elements are in front.
Sort Nodes	Changes the display of node results so that different nodes are in front.
Draw Deck Hatch	Hatches in composite deck areas.

Colors Allows you to display and change the color of beams, plates, solids, section outline and selected entities.

Beams	Click Default to select the color used for beams.
Plates	Click Default to select the color used for plates.

		Plate elements can be color coded to distinguish the directions of their local z-axes. The face of the element in the positive direction of the z-axis is referred to as the "front." The opposite side is referred to as the "back."
	Solids	Click Default to select the color used for solids.
	Section Outline	Click select the color for the **Sections Outline** option in the 3D Sections group.
	Selected Entities	Click to select the color of selected entities in the View area.
	Show Color Legend	Selecting this option will display a color legend on the screen.
Margin around Structure	Represents the blank margin around the structure in percentage of the total view window.	
Solid – Fill Face	Select a face of the solid to be filled with color. After checking this option and clicking **Apply**, the selected face of solids are filled with color.	

Load and Results tab

Used to specify the scales to which the different diagrams for viewing input and results should be plotted.

Scales tab

Used to specify the scales to which the different diagrams for viewing input and results should be plotted.

Tip: In general, a larger scale number causes the diagram to shrink and a smaller scale number enlarges the diagram.

Save as Default	If the scale, to which an input item or result item is drawn, has been modified, one may instruct the program to keep that setting as the default, in place of the previous setting or the built-in default.	
Reset to Default	Resets all scales to the installed default values.	
Apply Immediately	When checked scale values in the view window are updated immediately as you change the value.	
Icon Size	**Support Icon**	The size to which the fixed-but support icons are plotted can be increased or decreased using this option. This applies to fixed but support icons only.
	Control Block	Controls the size of the control node icon when the **Control/ Dependent** option has been checked on the **Labels** tab.

Labels tab

Used to select various display labels for different components of the structure.

Tip: You can use the **Label Settings** tool in the **Labels** group on the **View** ribbon tab to quickly access this dialog tab directly.

Nodes

Node Numbers	displays the node numbers on screen.
Node Points	identifies the nodes with a small circle.
Supports	displays the support icons at supported nodes.
Dimension	displays the member lengths in current units.

Properties

References	displays the Property Tag number of the member/element properties.
Sections	displays the section name (such as W12x26).
None	removes the display of property information.

Physical Members

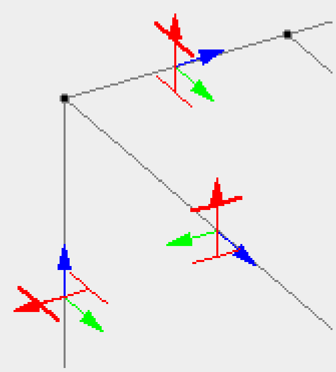
These options work only in the steel design mode. Please refer to the Miscellaneous section at the end of this manual for more details.

Beams

Beam Numbers displays the member numbers on the frame members.

Beam Orientation displays an icon showing the local axis of each member. The arrow indicates the positive direction of the local x-axis. The local y-axis is in the direction of the thicker flange. Please note that the I symbol is used regardless of the actual section type.

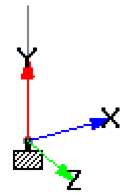
Note: The local x-axis is represented by the blue arrow. The local y-axis is represented by the red arrow. The local z-axis is represented by the green arrow.



Beam Spec option displays beam specifications which have been assigned, such as truss and tension only members.

Releases displays the member releases.

Beam Ends This switch, if turned on, paints the beams in the current view, with distinct colors to distinguish the start node of the member from its end node, thereby making it easy to identify the direction of the local X axis of the member.

	Start Color	Click on the little square adjacent to this option to set the color in which the zone of the member adjacent to its start node must be painted in.
	End Color	Click on the little square adjacent to this option to set the color in which the zone of the member adjacent to its end node must be painted in.
Plates	**Plate Numbers**	displays the plate element numbers.
	Plate Orientation	displays the local axis system for plates.
Solids	**Solid Numbers**	displays the solid element numbers.
Surfaces	**Surface Numbers**	displays the surface numbers
	Surface Orientation	displays the local axis system for surface object
Loading Display Options	**Load Values**	Check this box to display the intensity of the loads in the current unit system.
	Display Floor Loading	Check this box to enable/disable the display of the actual joint/ uniform load icons applied to the nodes or members.
	Display Floor Load Distribution	Check this box to display the tributary area diagrams for floor loads.
	Display Wind Load Contributory Area	Check this box to display the tributary area diagrams for wind loads.
	Display Wind Load	Check this box to enable/disable the display of the actual joint/ uniform load icons applied to the nodes or members.
General	**Show Axes Window**	Check this option to display a small icon at the bottom left corner of the drawing area which displays the directions of the global X, Y and Z axes are drawn with respect to the current view of the structure.
	Show Axes at Origin	Check this option to display an icon of the global axes system at the global X=0, Y=0, Z=0 position on the drawing. Axes are color-coded: X is blue, Y is red, and Z is green. 
	Material	Displays the name of any material assigned.
	Show Diagram Info	Through the various pages of the post processing mode, the program provides the facility for plotting diagrams such as the Node displacements, beam moments, plate stress contours, etc. Along with these diagrams, a caption is displayed along the bottom right corner of the drawing window indicating the type of diagram, load case, degree of

		freedom, length and force units, etc. In order to bring up those captions, this switch must be turned on. If you do not want those captions to come up, switch off this label.
	Control/ Dependent	Set this option to display control joints as green cubes.

Always Use STAAD.Pro Default Label Settings	As you navigates the various pages of the program, certain labels –meant for easy identification of the assigned attributes– will become switched on by default. Keeping this setting on enables that.
Always Use Current Label Settings	Setting this option will save the current label settings for the current STAAD input file. The labels can be unique to each page within the GS. Workflows in STAAD.Pro (on page 36).

Use Partial Labeling Mode (use labeling cursors to turn ON/OFF individual labels)	If a label is toggled on (e.g, node numbers or beam numbers), by default, STAAD.Pro will display those numbers for *all* the nodes and beams which appear in the current view. On large models or if several types of labels are turned on, this can clutter the drawing making it difficult to distinguish the individual entities. The "Partial Labeling Mode" can be used to display labels for only a select few nodes, members, etc. This switch must be used in conjunction with the individual labeling cursors found in the **Labels** group on the **View** ribbon tab.

Force Limits tab

Used to identify the members having the force values lying inside or outside the specified ranges.

Load Case	Select the **Load Case** for which you want to display the diagram.
Forces	Select the force type from the list of available options, such as **Axial**, **Shear YY**, **Bending ZZ**, etc. Provide the range in the associated edit box under **Minimum** and **Maximum.**
View Limits	The buttons under this group determine which members are going to be highlighted. The **Exceed Either** option highlights any member whose forces exceed either the **Maximum** or the **Minimum** values. **Exceed Maximum** option highlights only the members whose forces exceed the **Maximum** values. **Exceed Minimum** option highlights only the members whose forces exceed the **Minimum** values.
Color Within Limit	Allows you to change the color and width of the force diagram when the forces fall within the view limits.
Color Outside Limit	Allows you to change the color and width of the force diagram when the forces fall outside the view limits.

Animation tab

Used to graphically view deflections, section displacements, mode shapes and stresses in an animated mode. It is a way of visualizing the stress build-up or attainment of peak displacement using animated views.

Diagram Type	**No Animation -** select this option to turn off any current animation

Deflections
Section Displacements
Mode
Stress

Animation Setup	**Full Screen**	displays the animation in the full monitor screen rather than in a window. This option may use less memory than displaying the animation in a large window.
	Extra Frames	Choose the number of **Extra Frames** above the minimum needed to enhance the animation if it appears choppy.
	Target FPS	Choose the **Target FPS** (frames per second) to control the animation speed. To speed up an animation, show more frames per second. To slow down an animation, show fewer frames per second.
	Use Metafiles for offscreen data	Click the **Use Metafiles for offscreen data** to save the animated screens as Windows Metafiles. Speed may be slower if this option is used.

Design Results tab

Used to display code check and steel design information on the structure.

Show Diagram options

- **None -** Do not display the design results graphically
- **Show Diagram (Based on Actual Ratio) -** Displays the design results diagram in the active view window. Values and colors displayed are based on the ratio of the demand to capacity ratio for the selected load case.
- **Show Diagram (Based on Normalized Ratio) -** Displays the design results diagram in the active view window.

Diagram detail options

- **Basic Diagram -** Members are displayed in four distinct colors which indicate Not Designed, Pass or Safe, Fail and Extreme Failure respectively. The default values define a member as Safe if the ratio is less than 1, Fail if the ratio is between 1 and 1.5, and Extreme Fail if the ratio is greater than 1.5.

  Note: It is important to understand that in this page, the definition of Pass and Fail can be set to have a different meaning than the one the calculation engine used during the steel design process. In the calculation engine, Fail means exceedance of the value set for the `RATIO` parameter (whose default value is 1.0 usually).

  Note: The utilization ratio plot that was available in previous versions of STAAD.Pro can also be accessed by choosing the **Basic Diagram** option.

- **Detailed Diagram -** The table on the right hand side of the box displays the index for the color plot. The first column indicates the lower limit and the second column represents the upper limit for the unity ratio. You can accept the default index settings or can choose to define your own settings by selecting the Use Custom Limits option. The upper and lower limit for the ratio can be provided in the Maximum and Minimum fields, respectively. Users can control the number of divisions between the maximum and minimum values for the index through the No of values option. The values for the divisions or ranges can also be set by the user by checking the Use Custom Divisions check box which will allow the user to edit the ranges displayed in the table on the right.

Show values Select this option to display the ratio values on the contour plot.

Actual Ratio ranges table Displays the From and To range used for a color to graphically represent the design results status of a member. By default, members passing all design checks performed will be shown in green, members failing design checks by 50% or less shown in blue, and those failing by greater than 50% shown in red.

Tip: Double click the colored index cell to open a **Color** dialog, which is used to set a custom color for the selected range.

Plate Stress Contour tab

Used to display stress contours for plates for different types of stresses.

Tip: This tab is only present when plate elements are in the model.

Load Case Select the Load Case for which the stress is to be displayed. You can also select **Envelope** (or a previously defined envelope name) to display an envelope of stress results.

If **Envelope** is selected, select either the **+ve** (maximum positive) or **-ve** (maximum negative) cases for dislay.

Stress Type Select the type of stress gradients to display on the elements:

Max Absolute
Max Top, Min Top, Max Bottom, Min Bottom (Principal Major and Minor Stresses)
Tau Max Top, Tau Max Bottom
Max Von Mis
Von Mis Top, Von Mis Bottom
Max Tresca
Tresca Top, Tresca Bottom
SX, SY, SXY, MX, MY, MXY, SQX, SQY (local)
Global Moment, Global Stress
Base Pressure
Top Combined SX, Top Combined SY, Top Combined SXY, Bottom Combined SX, Bottom Combined SY, Bottom Combined SXY (local)

Contour Type The **Normal Fill** option provides contours based on the values of centre stress and corner stress as described above.

The **Enhanced Fill** option will determine the contour using the centre and corner stresses as used in the Normal Fill option, and an additional set of interpolated control points along the edges of each plate that will tend to smooth out the contour lines on plates that have large changes of stress over their surface.

Absolute Values causes the stress values to be compared based on the absolute values, rather than algebraic values. If this option is checked, stress values of +10 units and -10 units will be in the same range.

Index Based on Center Stress	The Plate Stress Contour maps are based on values from the calculated analysis results of the centre stress/force of each plate and corner stresses (which are averaged from all plates that share the same node). There are two ways that the corner stresses/forces can be determined. This can be either directly from the analysis results or estimated by extrapolating the centre stresses to the corners of each plate. These method to use is specified with the option: • When checked, the contour map will be drawn using only the values of centre stresses/ forces calculated by the analysis. Values for corner stresses/forces will be determined from each centre stress by extrapolating them to the corner and then averaged for all plates that share the node. • When unchecked, the contour map will be based on both the centre stresses/forces AND the values of corner stress/force calculated by the analysis. Again the value is averaged for all plates that share the node.
View Stress Index	displays the legends of the colors with stress values at the side of the screen.
Re-Index for new view	When individual elements are isolated from the full model then the stress distribution for the entire element may be displayed in a single color . This is because the variation in stresses across the element may not have been appreciable with respect to the entire model and the entire stress range may have been represented by a single color in the full model. This may create problem because the users in such cases will not be able to view the stress variation across the element. To take care of this **The re-index stress range for new view** option has been included which automatically regenerates the stress index for any newly created view which will allow the users to view the stress variation in much more detail.
Show Displaced Shape	Displays the plate stresses on a displaced model.
Contour Base on Visible Entities Only	Limits the stress contour range to the stresses contained in only visible entities.
Use Custom Limits	By default, 15 contours are plotted with the maximum and minimum values determined by the program based on the results for the selected load case and stress type. Set the Use Custom Limits option to specify **Minimum** and **Maximum** values as well as **No. of values** to use for equal divisions.
Use Custom Divisions	By default, the values of the stresses used for color gradation are obtained by taking the **Maximum** and **Minimum** and dividing the difference equally into as many parts as is defined **No. of values**. Setting the Use Custom Divisions allows you to directly type the cut-off values for each color. **Tip:** Take care to ensure that they are in ascending order.
Directions for Global Stress	(Global Stress or Global Moment only) Select the **Result Dir** of interest and the vector direction for **Up**.

Solid Stress Contour tab

Used to display stress contours for solids for different types of stresses.

Tip: This tab is only present when solid elements are in the model.

To display the solid stresses on a displaced model, select the **Solid Stress Contour** tool in the **View Results** group on the **Results** ribbon tab in the **Postprocessing** workflow.

Load Case	Select the Load Case for which the stress is to be displayed.
Stress Type	Select the **Stress Type** from the drop down list. Once we select these, the **Maximum** and the **Minimum** values of that stress under the selected load case are displayed on the dialog box.
Contour Type	The **Normal** and **Enhanced** buttons indicate how the Stress Contour is drawn. The **Normal** contour option uses the stress points at each corner of the solid along with the center stress to calculate the contour. The **Enhanced** contour option uses the same points as the Normal contour plus the interpolated stress at the mid point of the edges. The second option takes more time to generate but is more accurate.
Absolute Values	causes the stress values to be compared based on the absolute values, rather than algebraic values. If this option is checked, stress values of +10 units and -10 units will be in the same range.
Index Based on Center Stress	the stress range of the index is based on the stress values obtained at the center of the elements.
View Stress Index	displays the legends of the colors with stress values at the side of the screen.
Use Custom Limits	By default, 15 contours are plotted with the maximum and minimum values determined by the program based on the results for the selected load case and stress type. Set the Use Custom Limits option to specify **Minimum** and **Maximum** values as well as **No. of values** to use for equal divisions.
Use Custom Divisions	By default, the values of the stresses used for color gradation are obtained by taking the **Maximum** and **Minimum** and dividing the difference equally into as many parts as is defined **No. of values**. Setting the Use Custom Divisions allows you to directly type the cut-off values for each color. **Tip:** Take care to ensure that they are in ascending order.
Re-Index stress range for new view	When individual elements are isolated from the full model then the stress distribution for the entire element may be displayed in a single color . This is because the variation in stresses across the element may not have been appreciable with respect to the entire model and the entire stress range may have been represented by a single color in the full model. This may create problem because the users in such cases will not be able to view the stress variation across the element. To take care of this **The re-index stress range for new view** option has been included which automatically regenerates the stress index for any newly created view which will allow the users to view the stress variation in much more detail.
Show Displaced Shape	Displays the solid stresses on a displaced model.

Related Links

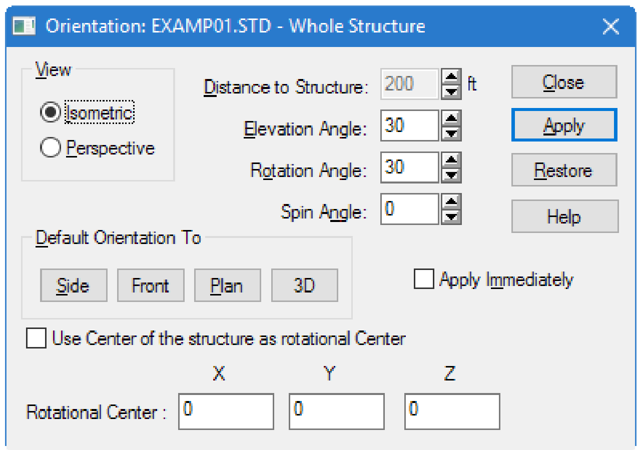
- *M. To display control nodes* (on page 652)
- *P. To display plate results contours* (on page 1988)
- *P. To display solid results contours* (on page 1990)

- *P. To display mode shapes* (on page 1991)

Orientation dialog

Used to modify the settings that define various view orientations of the structure, such as Plan view, Elevation view, Perspective view, etc.

Opens when the **Orientation** tool is selected in the **Rotation** group on the **View** ribbon tab.



View group

Isometric — This option will display the structure in default isometric view (with 70 degree Elevation angle and 330 degree Rotation Angle). An isometric view uses effectively an infinite distance to the structure.

Perspective — This option will display the perspective view of the structure. The viewing distance and angles can be changed by changing one of the three options: **Distance to Structure**, **Elevation Angle**, or **Rotation Angle**.

Distance to Structure

This value indicates the distance of the eye (or camera) from the structure in the perspective view. You may type this value or use the spin control to increase or decrease the current value.

Tip: This distance should be greater than the extents of a volume containing the structure in order to prevent distorting the view.

Elevation Angle

This value indicates the rotational angle of the eye about an axis which is lying horizontal (left to right) in the plane of the screen and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Rotation Angle

This value indicates the rotational angle of the eye about an axis which is lying horizontal (left to right) in the plane of the screen and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Spin Angle

This value indicates the rotational angle of the eye about an axis which is perpendicular to the plane of the screen (in to out) and passing through the center of the screen. You may type this value or use the spin control to increase or decrease the current value.

Default Orientation To

The orientation of the structure may be set to a view defined by one of the following options:

Side — View From + X (right)
Front — View From + Z (front)
Plan — View From + Y (top)
3D — isometric angle (default)

Apply Immediately Any change made through the **Orientation** dialog may be immediately reflected in the current view when this option is selected.

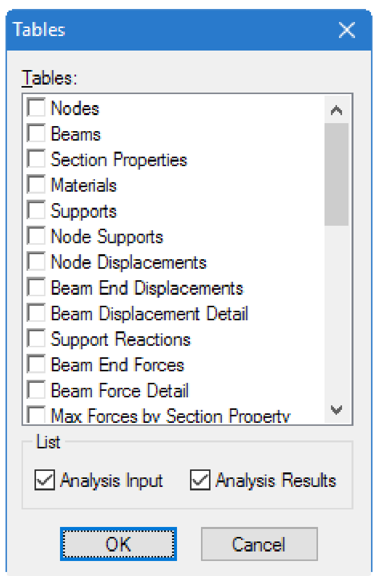
Use center of the structure as the rotational center When this options is selected, STAAD.Pro will consider the center of the structure as the rotational center. The axis of rotation can be manually input by clearing this box and then providing the **Rotational Center** values.

Rotational Center The axis of rotation can be manually input by checking this box and specifying the exact X, Y and Z coordinates in the Rotational Center boxes.

Tables dialog

Used to display and close different tables, such as Node coordinates, Beam incidences, Node displacements, etc. irrespective of the current page.

Opens when the **Tables** tool is selected in the **Windows** group on the **View** ribbon tab.



Tables Check the associated boxes for the tables you want to display. The boxes are automatically checked for the currently displayed tables. To close a table, leave the associated box blank.

List These check boxes control which tables are listed for selection from the **Tables** list. To display the list of available input related tables, set the **Analysis Input** option. To display the output related tables, set the **Analysis Results** option .

Open View dialog

Used to open a previously saved structural view window.

Opens when the **Open View** tool is selected in the **Views** group on the **View** ribbon tab.

Views list	A list of saved views is displayed here. Select the View you want to open.
Do you want to: options	Used to determine whether the selected view would be opened in a new window or whether it would replace the current view.
OK	Opens the selected view, using the selected option.
Cancel	Closes the Open View dialog without opening a view.
Help	Opens the Help window.

Options dialog

Used to control display items such as fonts, colors and styles for different labels, units, etc. The tabs of the dialog are explained in the following pages.

Opens when **Display options** is selected on the **Settings** tab in the Backstage view.

View Highlight tab

Controls the simultaneous display of several view windows on screen. STAAD allows us to create multiple "child" view windows from the default "Whole Structure" ("parent") view. Typically, the "parent" view would display the entire structure, while the "child" windows would display selected portions of the structure in various orientations.

When one or more "child" windows are displayed on screen along with the "parent" window, the parent window highlights the portion of the structure contained in a "child" window, whenever that child window is active (gets focus). The next figure shows that the *View 1* window is active and the portion of the structure contained in the *View 1* window is highlighted in the *Whole Structure* window.

The *View Highlight* tab controls how the Whole Structure window highlights the portion of the structure contained in a "child" window. In the above figure, the *Highlight View in Whole Structure Window* group is related to saved views, while the *Show Zoomed Area in Parent View* group is related to the view window created using the *Zoom Dynamic* option.

Highlight	When checked, the 'Parent" or Whole Structure window highlights the portion of the structure contained in the "child" window.
Width	Controls the width of the highlighted portion of the structure. The associated color button allows selection of the highlight color.

Tolerance tab

Tolerance	Controls the minimum allowable spacing between two adjacent nodes so that the program does not generate errors of duplicate or overlapping joints. Type a *Tolerance* in the associated edit box.
Split member if new node is added on the member	Among the various methods available for adding a beam in the Modeling mode, two of the methods are using **a.** The *Beam* sub-page of the *Geometry* page from the left side of the screen, and,

b. *Snap/Grid Node - Beam* option of the *Geometry* menu from the top of the screen.

In these methods, if the start or end point of a member being drawn (let us call this member B) happens to fall within the span point of an existing member (let us call this member A) , the setting chosen for this option will determine what happens to member A.

If the above option is switched on, the member A will be split into 2 members, and there will now be a total of 3 members incident from this common point.

If it is switched off, the member A will not be split, and the structure will be created as though the 2 members are NOT connected at the point in question.

One may test this by doing the following. Let us say that member A goes from (0,0,0) to (0,10,0), and that member B goes from (0,5,0) to (7,5,0).

If one were to use one of the methods (a) or (b) described above, it will be evident that leaving the option on will cause 3 members to be created, 5,5 and 6 units long. These 3 members will be connected at the common point (0,5,0).

Leaving the option off will cause only 2 members to be created, 10 and 7 units long respectively.

Here is another example of the relevance of this setting. Let us assume that a building is being assembled in two steps. First, the beam-column skeleton is created. Following that, the slabs and walls are created using the meshing capabilities of the *Geometry | Structure Wizard* option. If any of the nodes of the element mesh fall within the span lengths of the beams and columns, those beams and columns will either be split, or left unchanged, depending upon the settings here.

Tolerance for Warped Plate Element Detection

Used to so specify a tolerance for the plate warping check. By default, plate warping is reported when there is a 30 degree angle (or larger) between the normal to two triangles which are formed by splitting a quad element. This splitting can be done two ways so there are four triangles. If any have more than the tolerance angle (in degrees) between them, then an error is reported. M. To check for warped plates (on page 912) for how this value is used in the interface.

Set the **Save As Default** option to make this the default tolerance for warped plate checks.

> **Tip:** Alternatively, warped plates can be checked by the STAAD engine using the `SET PLATE FLATNESS TOLERANCE` command. See . However, checking for warped plates in the User Interface allows you to quickly correct any plates prior to running an analysis.

Tables tab

Used to choose the font used in various tables.

Sample Displays the current typeface settings used for Tables.

Font... Opens the Font (on page 2766), which is used to select font type, size, style, etc. used in tables.

Assign Dlgs tab

Used to control the display of labels for Property, Load, Support etc. while these are being assigned.

Show all labels when dialog is open	Displays labels for all assigned properties, loads, etc. when the corresponding dialog is opened.
Only show labels for selected items in dialog	Displays labels for only the selected items
Show all labels, with labels for selected items in the following color	Displays labels for the selected items in the specified color. Displays all other labels in default color.

Beam Labels/Plate Labels/Solid Labels/Load Labels/Load Icons/Annotation/Dimension tabs

These tabs offer options for labeling beams, plates, solids, etc. The *Beam Labels* tab controls how the labels for the beams (including the dimension of the members) appear on the screen, and is explained below.

Style	Select any of the pre-defined styles for displaying the beam numbers.
Separator, Reference	These options control how the property number reference is displayed with the member numbers.
Horizontal Alignment, Vertical Alignment	These items determine the relative position of the label with respect to the member.
Font...	Opens the Font (on page 2766), which is used to select font type, size, style etc. for the label.
Opaque	When checked, the labels behave as opaque objects, i.e. everything behind them become hidden.
Angle Text	When checked, displays the labels inclined along the members.

Structure Units/Section Units/Force Units tabs

The *Structure Units, Section Units* and *Force Units* tabs offer options for specifying units for the structure, sectional views, forces, etc. The *Section Units* tab is described below and is shown in the next figure.

Dimension

Used to set the length units in which dimension lines are drawn on the screen. This feature dictates the values that will be displayed for the *Dimension Beams* facility as well as the *Display Node to Node Dimension* facility of the *Tools* menu.

> **Note:** Changing units on these tabs does not change the input units in the Input Command File, the units are changed in the graphical display only. To change units in the Input Command File, see Tools | Set Current Input Unit...menu.

The *Section Units* tab allows you to specify the units to display different data items such as sectional Area, Moment of Inertia, etc. Select the appropriate units from the available drop down list and specify the number of digits to be displayed after the decimal point using the *Show* spin boxes.

Dimension

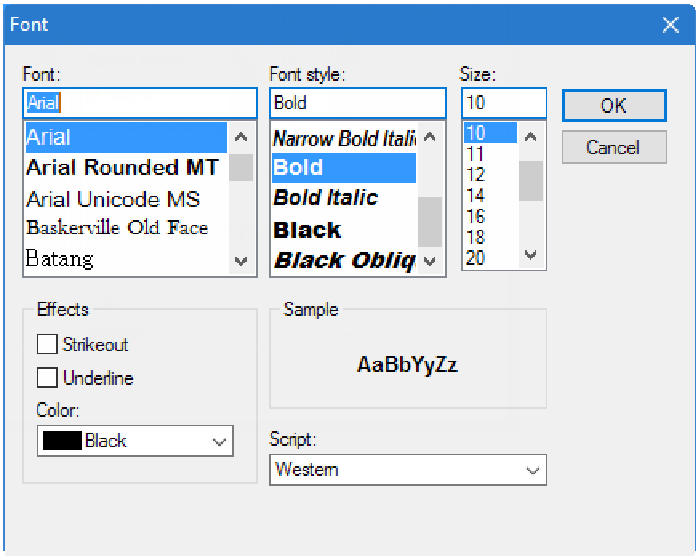
Used to set the length units in which certain property values are displayed on the screen. If you choose the *General* page followed by the *Property* page along the left side of the screen, a button called *Values* will be present along the right side of the screen. If properties have already been assigned to any members, you can examine the property values by clicking on the *Values* button. The terms *D, Bf, Tf* and *Tw* use the length settings of this facility. The same table may also be displayed by going to *View | Tables | Section Properties*.

OK	Applies all changes made and closes the dialog
Cancel	Closes the dialog without applying any changes.
Apply	Applies all changes made. Only active if changes have been made since the dialog was opened or the Apply button was last clicked.
Help	Opens the STAAD.Pro help window.

Font

Used to control the text display style.

Opens when the **Font** button is clicked in various dialogs.



Font	Sets the typeface. Select from the scroll list or simply type in the field to search for an installed font name.
Font style	Select font styles or combination of styles from the list.
Size	Sets the text size.
Effects	(Not available for all Font dialogs) Add **Strikeout** and **Underline** effects to the text.
Color	(Not available for all Font dialogs) Select a standard color to use for the font.
Script	Select the script set used.

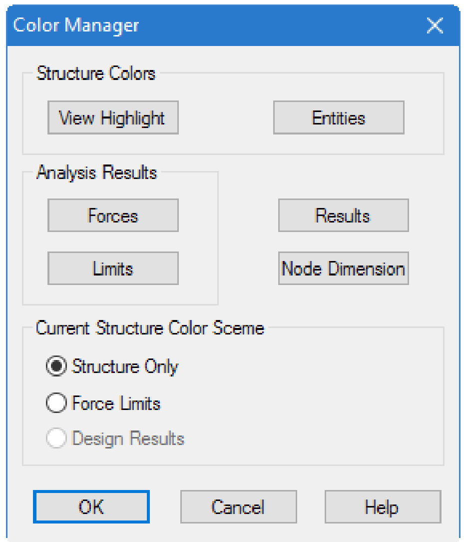
OK	Applies the font changes and closes the dialog.
Cancel	Closes the dialog without applying any changes.

Color Manager dialog

Used to specify colors of different items such as analysis results.

Opens when either:

- **Set Structure Colors** is selected on the **Settings** tab in the Backstage view, or
- the **Set Structure Colors** tool is selected in the **Options** group on the **View** ribbon tab.



View Highlights Opens a color selection dialog, which is used to specify a highlighting color for selected objects.

Entities Opens the **Define Colors** dialog, which is used to color code structural entities by specific attributes.

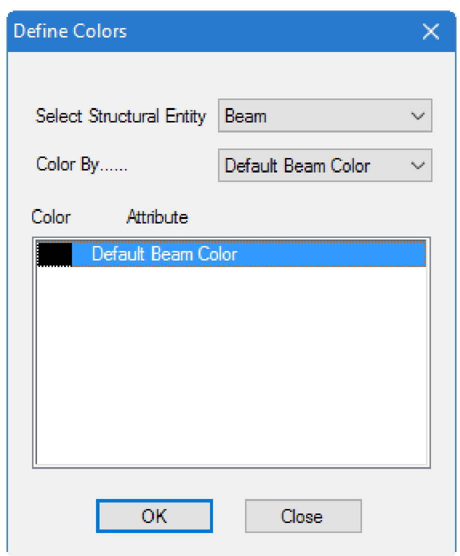


Figure 338: Define Colors dialog

	Specify the type of entity (Beam, Plate or Solid) whose colors are to be changed under the Select **Structural Entity** list box. Under the **Color By** option, select the type of attribute for which the colors will be arranged according to. To change the color of a particular attribute (for example, if Beam Properties option is selected, all the properties for that model will be listed), double-click on the color to bring up the color palette dialog box. Select a new color for that particular attribute. **Note:** The colors of the entities can also be changed through the Structure tab of the **Diagrams** dialog .
Forces	Opens a color selection dialog, which is used to specify a highlighting color for forces shown in analysis results.
Limits	Opens a color selection dialog, which is used to specify a highlighting color for limits in analysis results.
Results	Opens a color selection dialog, which is used to specify a highlighting color for results.
Node Dimension	Opens a color selection dialog, which is used to specify a highlighting color for node-to-node dimensions.
Current Structure Color Scheme	The following options are available for applying the selected color changes: Structure Only Force Limits Design Results

Tool Tip Options dialog

Used to display any customized input or output information about a node, beam, plate or solid element by simply placing the mouse over the structural entity.

The structural tool tips include the maximum and minimum nodal displacements and beam end forces out of all the load cases (envelope of displacements or forces). You can now place the mouse over a node or beam and have a tool tip report these values. Tool tips are similar to the information box displayed when the mouse hovers over a toolbar icon.

Opens when **Structural Tool Tip Options** is selected on the **Settings** page of the Backstage view.

Note: The tool tips automatically display the results for the active load case. All values are reported in the current display units.

Show Tool Tip	Turns the structural tool tips on or off.
Tip Delay	signifies the amount of time it takes from when the mouse hovers over an entity to when the tool tip actually pops up. This number is expressed in milliseconds (i.e., 1000 = 1 second).
Tool	Select the Node or Beam item from the left hand side On the right hand side, there will be new options to enable the display of the maximum and minimum nodal displacements or beam end forces out of all the load cases (i.e., an envelope).

Options

The options (items that can be displayed) for each entity are shown here. A check mark signifies that the particular data item will be displayed in the tool tip.

Note: An option with a “+” next to it signifies that further options can be enabled or disabled.

Note: A red “X” indicates the data will not be shown in the tool tip. Simply click on the check box to turn an option on or off.

The resulting tool tip for nodes with the Node Number, Displacement, and Support options set.

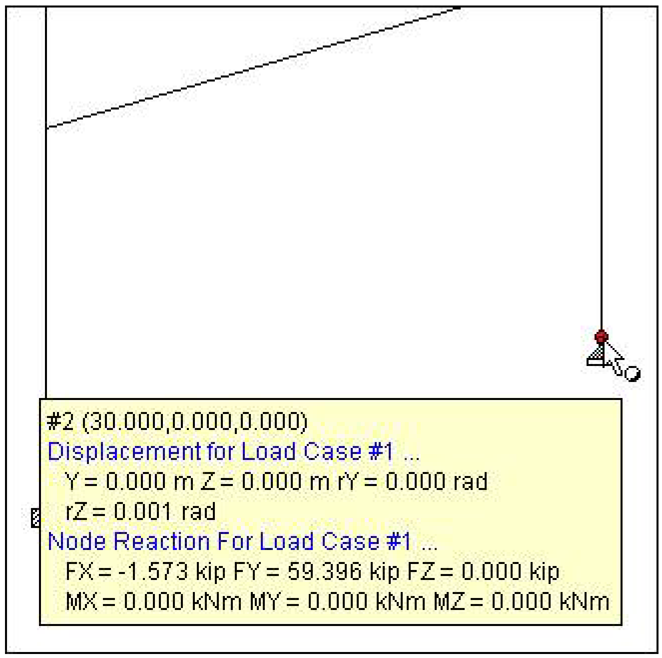


Figure 339: Example tool tip at a support

Select tab

Contains tools used for selecting objects in your analytical model.

Table 302: Cursors group

Tool name	Description
 **Nodes Cursor**	Used to graphically select nodes.

Tool name	Description
 **Beam Cursor**	Used to graphically select beams.
 **Plate Cursor**	Used to graphically select plates.
 **Solid Cursor**	Used to graphically select solids.
 **Geometry Cursor**	Used to graphically select nodes, members and elements of the structure simultaneously. To select nodes, members, or elements using the Geometry Cursor, simply click on the desired structural components. To select multiple nodes, members and elements, hold **<Ctrl>** while selecting. We may also select the structural components graphically by creating a window on screen with the cursor around these components.
 **Members Cursor**	Used in Steel Design or Concrete Design to graphically select all those beams defined as a same member in the member set up of member design, simultaneously. Members may be user defined or may be generated automatically using the Auto Form Member tool. To select all the beams defined as a same member using the Member Cursor, just click on one beam. The other beams having the same member name as the selected one will automatically be selected. To select multiple physical members, hold down **<Ctrl>** while selecting. You may also select the physical members graphically by dragging a fence area around these physical members using the cursor.
 **Plates & Solids Cursor**	Used to graphically select both plate and solid elements.

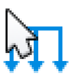
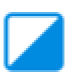
Tool name	Description
 **(Select) Text**	Used to add comments and titles to pictures and result diagrams. The added text can be plotted, too. The inserted text can be deleted, moved, and modified using the text cursor. Refer to the **Insert Text** on the **Utilities** ribbon for a detailed description on inserting text and modifying it using the text cursor.
 **Previous**	Selects the last objects selected if they have been deselected or if the selection set has changed.
 **Load**	Used to modify any load already applied on the model by double clicking it. When selected, the mouse pointer changes to the Load Edit Cursor.

Table 303: Geometry group

Tool name	Description
 **All**	Selects all nodes, members, elements, and solids in the model.
 **Inverse**	Used to select all objects but the ones which are currently selected.
 **List**	Opens the **Select Geometry** dialog, which is used to select one or more model entities (other than nodes) from a list of all entities in the model.

Tool name	Description
Parallel	Used to select all beams, elements, and surfaces which are parallel to a specified global axis. Select the desired global axis from the sub-menu. • **To XY** • **To YZ** • **To XZ**
Connected	Used to select all the entities that are connected to (i.e., have a common node with) any particular node, beam, plate, or solid. A dialog opens prompting you to select the node, beam, plate, or solid number. You can select to apply immediately or click **OK** to see all the model entities that are connected to the selection. • **To Node** • **To Beam** • **To Plate** • **To Solid** **Tip:** Options for geometry objects not contained in the current mode are disabled.
Highlight	Opens the **Visual Check** dialog, which is used to sequentially highlight a specific group of entities (beams, plates, or solids) in numerical order. Controls on the speed of the selection are also included.

Table 304: Nodes group

Tool name	Description
All	Selects all nodes in the model.
Inverse	All nodes in the selection set are deselected and any previously unselected nodes are added to the selection set.

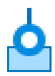
Tool name	Description
 **List**	Opens the **Select Nodes** dialog, which is used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.
 **Supports**	Selects all supported nodes in the model.

Table 305: Beams group

Tool name	Description
 **All**	Used to select all beams.
 **Inverse**	All beams in the selection set are deselected and any previously unselected beams are added to the selection set.
 **List**	Opens the **Select Beams** dialog, which is used to select one or more beams from a list of all beams in the model.
 **Parallel**	Select beams that are parallel to the selected global axis. • **To XY** • **To YZ** • **To XZ**

Tool name	Description
 **Connected**	Select beams that connect to the selected object type. • **To Node** • **To Beam** • **To Plate** • **To Solid** **Tip:** Options for geometry objects not contained in the current mode are disabled.

Table 306: Plates group

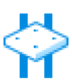
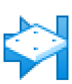
Tool name	Description
 **All**	Selects all plates in the model.
 **Inverse**	All plates in the selection set are deselected and any previously unselected plates are added to the selection set.
 **List**	Opens the **Select Plates** dialog, which is used to select one or more plates from a list of all plates in the model.
 **Parallel**	Select plates that are parallel to the selected global axes plane. • **To XY** • **To YZ** • **To XZ**
 **Connected**	Select plates that connect to the selected object type. • **To Node** • **To Beam** • **To Plate** • **To Solid** **Tip:** Options for geometry objects not contained in the current mode are disabled.

Table 307: Solids group

Tool name	Description
 **All**	Selects all solids in the model.
 **Inverse**	All solids in the selection set are deselected and any previously unselected solids are added to the selection set.
 **List**	Opens the **Select Solids** dialog, which is used to select one or more solids from a list of all solids in the model.
 **Connected**	Select solids that connect to the selected object type. • **To Node** • **To Beam** • **To Plate** • **To Solid** **Tip:** Options for geometry objects not contained in the current mode are disabled.

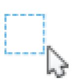
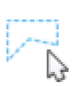
Table 308: Filter group

Tool name	Description
 **Filter**	Used to select multiple types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in one pass. This will reduce the time required to create new views and help quickly identify the location of certain entities on your structure. **Note:** Before this cursor can be used, the actual filter parameters must be defined in advance in the **Selection Filters** dialog.

Table 309: Attributes group

Tool name	Description
 **Group**	Opens the **Select Groups** dialog, which is used to select objects in a named group.
 **Property Name**	Used to select nodes and members based on specifications associated with them. A number of specification types are included in the sub-menu list.
 **Missing Property**	Used to select beams, plates, or solids that lack critical input data (Property, Density, Elasticity, Poisson's ratio, and Alpha) in the active view.

Table 310: Modes group

Tool name	Description
 **Drag Box**	Click to activate the drag box selection mode.
 **Drag Line**	Click to activate the drag line selection mode.
 **Region**	Click to activate the region selection mode.

Visual Check dialog

Used to easily identify the location of entities in a certain section of the model. Many times, commands like assigning properties or releases are used for a certain set or range of entities which may be difficult to graphically observe because of their close proximity to each other. This feature may help alleviate difficulties in checking the properties of a concentrated group of entities.

Commands in the dialog to control the speed of the automatic selection process are described below.

Opens when the **Geometry** > **Highlight Entities Sequentially** tool is selected in the Selection group on the Geometry ribbon tab.

Start With list
Select the entity type to highlight.

Currently Selected
Displays the currently selected entity

>
Automatically selects the type of entity chosen in the Start With list in a sequential order (by entity number).

||
Pauses the sequential selection operation. Inactive when the automatic selection is not active.

>>
Increases the speed at which the automatic selection occurs. The selection process becomes faster with each press of this button.

<<
Decreases the speed at which the automatic selection occurs. Each time this button is pressed, the selection process becomes slower.

|<
Increments the selection manually one entity at a time. If this is not highlighted, the Pause button must be pressed to stop the automatic selection process.

>|
Decrements the selection process manually one entity at a time. If this is not highlighted, the Pause button must be pressed to stop the automatic selection process.

Stop
Stops the selection process.

Query
Opens the entity query dialog for the currently selected entity.

Exit
Closes the Visual Check dialog.

Select Nodes dialog

Used to select one or more nodes from a list of all nodes in the model. A free list of node numbers may also be specified.

list of nodes
Select the desired nodes by clicking on their number in the displayed list. To select sequential nodes, either click and hold the mouse button while dragging over the numbers or hold the SHIFT To select non-sequential nodes, hold down **<Ctrl >**while clicking.

Selection Type
To select nodes by numbers in the list, select the **Select from list by cursor** option. To select nodes with a typed list of node numbers, click the **Select using typed list** option then click **Select Listed Entities** .

Select Listed Entities
Click to select nodes listed in the **Enter list** box.

Enter list
Specify node numbers for selection. You may use `to` to represent a range of sequential node numbers. Separate values by either spaces or commas.

> **Tip:** Clicking in this box selects the Select using types list option for Selection Type automatically.

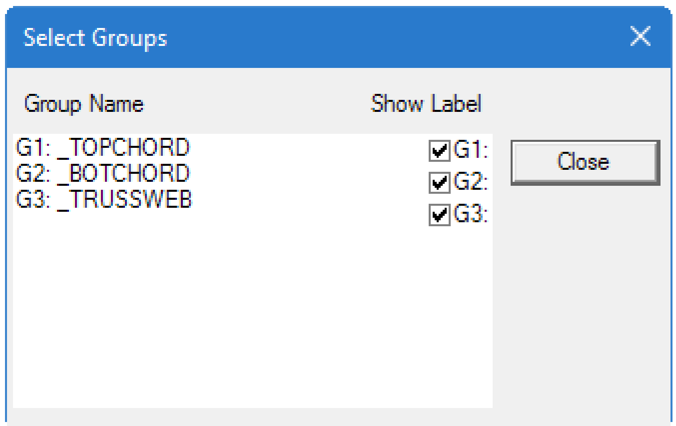
> **Note:** Similar features exist for selecting by lists of beams, plates, solids, surfaces, and geometry.

Opens when the **Nodes** > **Node List** tool is selected in the **Selection** group on the **Geometry** ribbon tab.

Select Groups dialog

Used to select members in a named group.

Opens when the **Group Selection** tool is selected in the **Selection** group on the **Selection** ribbon tab.



Group List Select a single group to highlight the component model elements in the View window. Multiple groups may be selected by pressing **<Ctrl>**.

Show Label If labels pertaining to certain entities are switched on (e.g., beam labels), the labels will be displayed only for members with a **Show Label** check box set. Clear the check box for any group for which you do not want labels displayed.

Close Closes the dialog.

Specification tab

Table 311: Beam Profiles group

Tool name	Description
 **Database**	Click the drop-down arrow to display a gallery list of all database section tables. Click a country name or material to open the corresponding section table dialog.

Tool name	Description
Prismatic	Opens the **Property** dialog, which is used to assign Circular, Rectangular, Tee, Trapezoidal, General(arbitrary), etc. cross sections to the members.
Tapered	Opens the Tapered I Property dialog, which is used to specify an I-section having a varying depth over the length of the member.
User Table > **User Table Manager**	Opens the **User Table Manager** dialog, which is used for adding and managing previously created user table sections to the structure.
User Table > **Assign from User Table**	Opens the User Property Table dialog, which is used to assign previously created user table sections to the structure. **Note:** If you have not created any user-provided tables, then you will asked if you would like to create one first.
Assign	Opens the **Assign Profile** dialog, which is used to select a section profile and optional material to a member for member selection during design.

Table 312: Plate Profiles group

Tool name	Description
Plate Thickness	Opens the **Plate Element/Surface Property** dialog, which is used to provide plate element properties (thickness) with or without the material specification. Plates can have a different thickness at each node. Surfaces have a constant thickness.

Table 313: Material Constants group

Tool name		Description
Constants >	**Young's Modulus**	Opens the **Material Constant - Elasticity** dialog, which is used to assign the modulus of elasticity, E, based on a predefined material value or a custom value.
	Shear Modulus	Opens the **Shear Modulus - G** dialog, which is used to assign the shear modulus, G, based on a predefined material value or a custom value.
	Poisson's Ratio	Opens the **Material Constant - Poisson's Ratio** dialog, which is used to assign Poisson's ratio, ν, based on a predefined material value or a custom value.
	Density	Opens the **Material Constant - Density** dialog, which is used to assign the material density, γ, based on a predefined material value or a custom value.
	Thermal Coefficient	Opens the **Material Constant - Alpha** dialog, which is used to assign the thermal coefficient, α, based on a predefined material value or a custom value.
	Damping Ratio	Opens the **Material Constant - Damping Ratio** dialog, which is used to assign the damping ratio, c, based on a predefined material value or a custom value.

Table 314: Specifications group

Tool name		Description
Node >	**Add Control/ Dependent Specification**	Opens the **Node Specification** dialog to the **Control/Dependent** tab, which is used to select a control node and the directions which other dependent nodes follow actions.
	Delete Control/ Dependent Specification	Used to delete a control/dependent specification.
Beam >	**Beta Angle**	Opens the **Beta Angle** dialog, which is used to specify the beta angle for members.
	Beam Reference Point	Opens the **Reference Point** dialog, which is used to specify a reference point for the members from which the program calculates the beta angle.
	Cable	Opens the **Member Specification** dialog to the **Cable** tab, which is used to assign standard cable and nonlinear cable specifications.
	Compression Only	Opens the **Member Specification** dialog to the **Compression** tab, which is used to define compression-only members. These members are capable of carrying compressive forces only.

Tool name		Description
	Tension Only	Opens the **Member Specification** dialog to the **Tension** tab, which is used to define tension-only members. These members are capable of carrying tensile forces only.
	Truss	Opens the **Member Specification** dialog to the **Truss** tab, which is used to specify truss members. These members are capable of carrying axial forces only.
	Imperfection	Opens the **Member Specification** dialog to the **Imperfection** tab, which is used to assign camber or drift values to members.
	Cracked Property	Opens the **Member Specification** dialog to the **Property Reduction Factors** tab, which is used to assign reduced (i.e., “cracked”) section property factors to members.
	Release	Opens the **Member Specification** dialog to the **Release** tab, which is used to assign releases, partial moment releases, or spring values to member ends.
	Offset	Opens the **Member Specification** dialog to the **Offset** tab, which is used to assign member end offset values to a member end.
	Inactive	Opens the **Member Specification** dialog to the **Inactive** tab, which is used to instruct the analysis and design engine to ignore a member for analysis.
	Fireproofing	Opens the **Member Specification** dialog to the **Fire Proofing** tab, which is used to assign block or contour fire proofing material to a member for weight calculations.
 **Plate >**	**Plate Reference Point**	Opens the **Plate Reference Point** dialog, which is used to specify options for determining the general direction of the local Z axis of elements,
	Release	Opens the **Plate Specs** dialog to the **Release** tab, which is used to assign released degrees of freedom at a plate node.
	Ignore Inplane Rotation	Opens the **Plate Specs** dialog to the **Ignore Inplane Rotation** tab, which is used to override the value that STAAD determines for the stiffness associated with the local axis Mz degree of freedom for plate elements
	Plane Stress	Opens the **Plate Specs** dialog to the **Plane Stress** tab, which is used to model plate elements as Plane Stress elements.
	Ignore Stiffness	Opens the **Plate Specs** dialog to the **Ignore Stiffness** tab, which is used to instruct the program to neglect the stiffness of the selected plates while assembling the stiffness matrix. Hence, the stiffness contribution of the selected plates is not considered. This is similar to the Inactive command for the members.

Table 315: Supports group

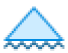
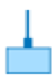
Tool name		Description
 **Fixed**		Opens the **Create Support** dialog **Fixed** tab, which is used to create a fixed support tag and (optionally) to assign that to selected nodes. A Fixed support is restrained in all 6 degrees of freedom.
 **Pinned**		Opens the **Create Support** dialog **Pinned** tab, which is used to create a pinned support tag and (optionally) to assign it to selected nodes. A Pinned support is restrained in all three translational degrees of freedom and free in the 3 rotational degrees of freedom.
 **Custom**		Opens the **Create Support** dialog **Fixed But** tab, which is used to create various types of roller, hinge and spring supports with specified restrained degrees of freedom and to assign them to selected nodes.
 **Foundation**		Opens the **Create Support** dialog **Foundations** tab, which is used to create spring supports for independent footings and mat foundations and to assign them to selected nodes.
 **One-way Spring**		Opens the **Create Support** dialog **Tension/Compression Only Spring** tab, which is used to designate certain support springs as Tension-only or Compression-only springs.
Other Supports >	**Inclined**	Opens the **Create Support** dialog **Inclined** tab, which is used to create supports that offer restraints in an axis system that is inclined with respect to the global axis system.
	Mulit-Linear Spring	Opens the **Create Support** dialog **Multilinear Spring** tab, which is used to enter displacement and spring stiffness data pairs for a multilinear spring definition, which is then used for a Fixed But support type.
	Enforced	Opens the **Create Support** dialog **Enforced** tab, which is used to create a fixed-condition support defined in terms of being stiff springs.
	Custom Enforced	Opens the **Create Support** dialog **Enforced But** tab, which is used to create a fixed-condition support defined in terms of being stiff springs with the option to release selected degrees of freedom.

Table 316: Tools group

Tool name	Description
 **Section Wizard**	Launches the **Section Wizard** application.
 **Section Database**	Launches the **Section Database Manager** application.

Specifications - Whole Structure dialog

Used for defining specifications and assigning them to nodes, members and elements.

Opens when either:

- the **Specifications** page is selected, or
- the **Specifications Layout** tool in the **Specifications** group on the **Specifications** ribbon tab is selected

Specification list
All Specifications that have been defined for the model are listed here.

Note: Physical Member specifications are designated with either PMEMBER or Physical tags.

Highlight Assigned Geometry
When this box is checked and you click on a specification in the **Specifications** dialog box, members that have been assigned the specification will appear highlighted in the structure window. For example in the previous figure, if you click on the MEMBER TRUSS specification, all truss members will be highlighted.

Edit/Delete
Used to modify a previously assigned attribute, or delete it altogether.

Node...
Opens the Node Specification dialog (on page 2784), which is used to specify rigid links or specialized linkages in the structure.

Beam...
Opens the Member Specification dialog (on page 2786), which is used for specifying frame member specifications.

Plate...
Opens the Plate Specs dialog (on page 2791), which is used to specify finite element specifications.

Toggle Specification
This is a switch setting which turns on what is called the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens.

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.

- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Assignment Method

The options under the **Assignment Method** are used to assign specifications to joints, members and elements. The assignment method can be one of **Assign To Selected Beams, Assign To View, Use Cursor To Assign** or **Assign To Edit List**.

- Assign To Selected Beams **-** To assign a specification to selected nodes, members or elements, first select the specification from the **Specifications** dialog box. The specification selected is highlighted. Then select the nodes, members or elements to which this specification is to be assigned. This is done by going to the **Select** menu, then choosing the appropriate cursor option. Select the desired joints, members or elements using the cursor. When all desired nodes, members or elements are selected, click the **Assign To Selected Beams** radio button, then click the **Assign** button. Note that the label for this button changes depending on whether nodes, members or elements are selected.
- Assign To View **-** To assign a specification to all nodes, members or elements in a view, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Assign To View** radio button, then click the **Assign** button. All nodes, members or elements in the structure are assigned this specification.
- Use Cursor To Assign **-** To assign a specification to nodes, members or elements using the cursor, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Use Cursor To Assign** radio button, then click the **Assign** button. The button will appear depressed and the label will change to **Assigning**. Use the appropriate cursor (from the Select menu) to assign the selected specification to the individual nodes, members or elements. Click on the **Assign** button again to finish.
- Assign To Edit List **-** To assign a specification using a typed list of node, member or element numbers, first select the specification from the **Specifications** dialog box. The selected specification is highlighted. Select the **Assign To Edit List** radio button, then type the list of node, member or element numbers and click the **Assign** button.

Related Links

- *M. To add a member specification* (on page 802)

Node Specification dialog

Used to specify rigid links or specialized linkages in the structure. This facility can be used to model special structural elements such as ties or a floor diaphragm which makes the floor rigid for in-plane movements. The process is to select a node to act as the control and then degrees of freedom for which the dependent nodes are linked to the control.

Note: Refer to TR.28 Rigid Diaphragm Modeling (on page 2327) for additional details.

Opens when **Node** is clicked on the **Specifications - Whole Structure** dialog.

Note: STAAD.Pro is also capable of defining a rigid floor diaphragm without the need to specify a “control” node. This feature is only available by manually editing the STAAD input file. Refer to TR.28.2 Floor Diaphragm (on page 2328) for additional details

Control/Dependent tab

Note: Control / Dependent was formerly referred to as master / slave.

Control Node
It is called as such because the displacements of the dependent node will be based on the displacement of this “control” node. From the drop-down list, select the node which should serve as the control node.

Dependent Directions

- **Rigid -** This is a means of defining that the link which connects the control with the dependent has infinite stiffness for all the six degrees of freedom. In other words, if one were to consider an imaginary object that has infinite stiffness axially, for shear, torsion and bending, that would be the type of link which connects the control and the dependent.

  Note: By default, STAAD.Pro keeps this box checked. Un-check this box to activate the options described below.

- **XY**, **YZ**, and **ZX -** This is a way of specifying a type of connection where the rigidity is limited to in-plane directions.

  The plane of this link is defined in terms of the global planes XY, YZ, or ZX. For example, if you have a concrete slab in a building, and the slab can be assumed to be nearly inflexible for translations along global X and global Z, and the rotation about global Y, that type of link would be called a “diaphragm” in global XZ.

  XY is equivalent to a Control-Dependent relationship for the degrees of FX, FY and MZ. Similarly, YZ is equivalent to linking FY, FZ and MX, and, ZX is equivalent to linking FX, FZ and MY.
- **FX**, **FY**, **FZ**, **MX**, **MY**, and **MZ -** This provides a way for linking only the specific degrees of freedom you want.

Floor Diaphragm tab

Diaphragm group

No Select diaphragm number.

Type Select the diaphragm type from the drop-down list:

Rigid - a diaphragm that translates & rotates as a rigid body.

Floor Level group
Select one of the options as the method to define the floor level of the diaphragm.

Height Type the global coordinate value (elevation), in Y direction, to specify floor level.

YRange Select this option to define the floor level by upper and lower bounds of Y coordinates.

Note: A single diaphragm may be defined at a given level in STAAD.Pro.

YRange group
Used when the **YRange** option is selected.

	Minimum	The lower bound of Y coordinates for the Y range.
	Maximum	The upper bound of Y coordinates for the Y range.
Control Node Information group	Select the method used to define the Control node of the diaphragm.	
	Calculate	The diaphragm center of mass is determined and a node is created at that location to use as the Control node.
	Select Control Node	Select the existing node to use from the drop-down list.
Define Floor Range	Set this option to define the floor range as a XZ range of coordinates or by manually selecting coordinates.	

Related Links

- *M. To assign nodes to a floor diaphragm* (on page 818)
- *G.14 Rigid Diaphragms* (on page 2128)
- *M. To assign a rigid link between nodes* (on page 817)

Member Specification dialog

Used to set and modify member (beam) specifications.

Opens when **Beam** is clicked on the **Specifications - Whole Structure** dialog.

This dialog contains multiple tabs, each of which is used to add a different member specification.

Release tab

Used to specify the end conditions of members by releasing specified degrees of freedom. Unless a release is specified, all members are rigidly connected the nodes (i.e., all degrees of freedom are restrained) except if the end is a cantilever end or supported.

Note: Refer to TR.22.1 Member Release Specification (on page 2286) for additional information.

Location	Specify either **Start** or **End** as the released joint **Member Incidence** of the member.
Release Type	Select either the **Partial Moment Release** or the **Release** option.
Partial Moment Release	Type a release **Factor** to define a partial release condition. The MP option is a means of specifying the same partial release for all the 3 moment degrees of freedom (MX, MY and MZ). Alternately, use the MPX, MPY or MPZ options to apply different release factors in each direction.
Release	Click the **Release** radio button and check the boxes for the **FX**, **FY**, **FZ**, **MX**, **MY** or **MZ** directions to define the member release condition. You may also specify a Spring Constant along the six degrees of freedom by using the **KFX, KFY, ... KMZ** edit boxes. For example, to define a Spring Constant in the local X direction, enter the Spring Constant in the **KFX** edit box.

Tension tab

Used to define tension-only members. These members are capable of carrying tensile forces only.

There are no additional parameters for this member specification.

Property Reduction Factors tab

Used to specify a set of reduction factors to be applied on the calculated section properties such as area, moments of inertia, and torsional constant.

Note: Refer to TR.20.10 Member Property Reduction Factors (on page 2282) for additional information.

Select the method of how to specify member property reduction factors:

- **Global** – specify the **Reduction Factors** to be used below, which will apply to any member regardless of material.
- **Code Specific** – select the building code to use reduction factors specific to load cases per that code which are only applied to concrete members:

  IS1893 2016

Reduction Factors

Provide a set of reduction factors between which will be used in the stiffness analysis. These factors are directly multiplied to the following section properties:

- Reduction Factor for Cross sectional Area (RAX)

  Note: For IS1893 2016 reduction factors, the **RAX** input field is inactive as this property reduction is not mandated by that code for concrete members.
- Reduction Factor for Torsion Constant (RIX)
- Reduction Factor for Moment of Inertia, major axis (RIY)
- Reduction Factor for Moment of Inertia, minor axis (RIZ)

Notes:

a. Reduction factors are considered for analysis only but not for design.
b. Results using the reduced section properties are not available when using the member query feature.

Note: Automated stiffness reduction requires a STAAD.Pro Advanced license.

Cable tab

Used to define cable members. Select either the Tension or Length option to specify a member tension. Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them. Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

Note: Refer to TR.23.2 Member Cable Specification (on page 2290) for more information.

Initial TENSION	Use this option to specify the initial tension in the cable as a force. **Average** The average tension in the cable is used (considered in advanced cable analysis only). **Start node** Tension is measured from the start node of the member (considered in advanced cable analysis only). **End node** Tension is measured from the end node of the member (considered in advanced cable analysis only).
Unstressed LENGTH	Use this option to specify the initial tension in the cable as a length for nonlinear cable analysis.
Factor in global X Fwx	Multiplying factors on self weight components applied in the global X direction.
Factor in global Y Fwy	Multiplying factors on self weight components applied in the global Y direction. **Tip:** When Y is up, use a negative value to act as gravity.
Factor in global Z Fwz	Multiplying factors on self weight components applied in the global Z direction. **Tip:** When Z is up, use a negative value to act as gravity.

Truss tab

Used to specify truss members. These members are capable of carrying axial forces only.

There are no additional parameters for this member specification.

Note: To specify tension in a truss member for a nonlinear cable analysis, you must edit the STAAD input file directly. Refer to TR.23.1 Member Truss Specification (on page 2289) for additional details.

Compression tab

Used to define compression-only members. These members are capable of carrying compressive forces only.

There are no additional parameters for this member specification.

Offset tab

Used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

The actual beams and columns of a physical structure are represented by lines in the mathematical (computer) model. In the actual structure, a beam spans a distance which in the clear span between the faces of columns. But in the model, the line for the beam spans between the centerlines of the column. The half-depth portion of either column is considerably stiffer than the beam itself from the standpoint of bending. To take advantage of this additional stiffness, you may specify that the start and end faces of the beam are offset from the node by a distance equal to the half-column-depths.

Member offsets can be specified in other situations too. For example, consider:

- when a bracing member does not meet the node which is defined in its incidence list
- a girder and top slab in a bridge where the centerline of the girder is several inches below the centerline of the slab.

This facility is useful when you want to design the structural parameters of a member by considering the clear distance of the member between the supports (e.g., in the case shear forces or bending moments).

Refer to TR.25.1 Member Offset Specification (on page 2296) for additional information on member offsets.

Location	Specify either **Start** or **End** as the joint at which to specify an offset.
Direction	Select either the **Local** or **Global** axis system for assigning offset distances.
Offsets	Enter the offset distance from the joint in the three axis directions.

Inactive tab

Used to temporarily inactivate members for a specific analysis cycle. Inactivated members may be re-activated later for further processing.

Information on using this is available in EX. US-4 Inactive Members in a Braced Frame (on page 6334) or EX. UK-4 Inactive Members in a Braced Frame (on page 6618). This option is also explained in TR.18 Inactive/ Delete Specification (on page 2240).

There are no additional parameters for this member specification.

Fire Proofing tab

Used to automatically consider the weight of fire proofing material applied to structural steel.

Refer to TR.20.9 Applying Fireproofing on members (on page 2279) for additional information on fireproofing.

Fire Proofing Type	Two types of fireproofing configurations are currently supported: • **BFP (Block Fireproofing):** The fire-protection material forms a rectangular block around the steel section. The thickness specified is the minimum thickness which defines the outer block dimensions.

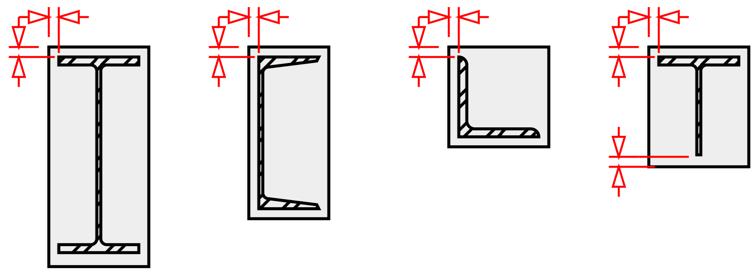


Figure 340: BFP: Block Fireproofing

- **CFP (Contour Fireproofing):**

  The fire-protection material forms a coating around the steel section. The thickness specified is a constant thickness around the section profile.

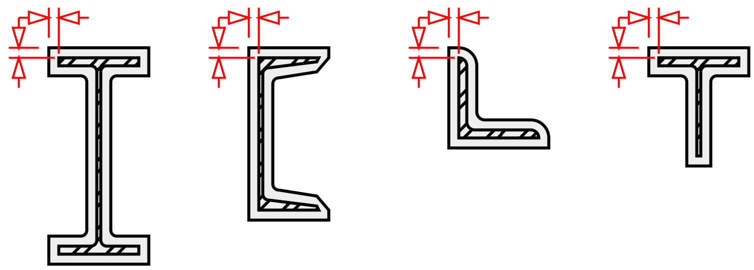


Figure 341: CFP Contour Fireproofing

Thickness	Thickness (dimension in the figures above) in length units
Density	Density of fireproofing material in (force / length^3) units

Imperfection tab

Used to define camber and drift specifications for selected members.

Note: Refer to TR.26.6 Member Imperfection Information (on page 2313) for additional details.

Imperfection type	Select if whether the imperfection is either **Camber** or **Drift**. Drift is usually for columns and camber for beams.
Local Direction	Specify if the camber or drift acts in either the member local **Y** or **Z** direction.
Value	Specify the value for the camber or drift, defined as a ratio of member length/offset. Default ratio value is 300.
Respect	(Camber option only) A dimensionless constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms, called ε, is calculated. If ε is less than the respect value given, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

$$\varepsilon_y = \text{Length} \times \sqrt{\frac{\left| \text{axial load} \right|}{EI_z}}$$

$$\varepsilon_z = \text{Length} \times \sqrt{\frac{\left| \text{axial load} \right|}{EI_y}}$$

Related Links

- *M. To assign cracked section properties to a member* (on page 807)
- *M. To add a member specification* (on page 802)
- *M. To assign nonlinear cable members* (on page 806)
- *G.8.1 Truss and Tension- or Compression-Only Members* (on page 2121)
- *M. To assign axial action members* (on page 803)
- *G.7 Member and Element Release* (on page 2120)
- *M. To assign member end release* (on page 804)
- *G.11 Member and Plate Offsets* (on page 2125)
- *M. To assign member end offsets* (on page 805)
- *M. To assign member imperfection for members* (on page 806)
- *M. To assign cracked section properties to a member* (on page 807)
- *M. To assign member fire proofing* (on page 808)
- *EX. US-4 Inactive Members in a Braced Frame* (on page 6334)
- *EX. UK-4 Inactive Members in a Braced Frame* (on page 6618)

Plate Specs dialog

Used to specify finite element specifications.

Opens when either:

- **Plate** is clicked on the **Specifications - Whole Structure** dialog, or
- the **Plate** tool in the **Specifications** group on the **Specifications** ribbon tab is selected

This dialog contains multiple tabs, each of which is used to add a different member specification.

Release tab

Used to release one or more degrees of freedom at the corner nodes of the elements. Unless a release is specified, elements are rigidly connected at the nodes (all six degrees of freedom are restrained) unless the node is a cantilever node or is supported.

Note: Refer to TR.22.2 Element Release Specification (on page 2287) for additional information on plate element releases.

Node — Select the node to be released. The node number depends on the Element Incidence for the element.

Release — Check the boxes for the **FX**, **FY**, **FZ**, **MX**, **MY**, or **MZ** directions to define the element release condition. These are based on the local coordinate system for the element.

Offset tab

Used to add rigid link offsets from element incident joints to the element corners.

Refer to TR.25.2 Element Offset Specification (on page 2298) for additional details.

Direction	Select one of the following direction options for assigning offsets: • **Local -** the nodal offsets at a specified joint are given in the element local axes. • **Global -** the nodal offsets at a specified joint are given in the global axes. • **Z Offset -** the offset is specified along the local z axis of the element (i.e., parallel offset to the plane of the element)
Local of Global Offsets	Select the incident node number of the element and type the offsets in the given units.
Z Offsets	Type the offset distance at each node. **Note:** By default, typing a distance for **Node 1** will populate the same value for the other nodes, thus indicating a parallel offset. You may override these values with different offsets but the offset element must remain coplanar.

Ignore Inplane Rotation tab

Used to override the value that STAAD determines for the stiffness associated with the local axis Mz degree of freedom for plate elements.

Refer to TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2295) for additional information.

There are no additional parameters for this plate element specification.

Rigid Inplane Rotation tab

Used to program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly

Refer to TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2295) for additional information.

There are no additional parameters for this plate element specification.

Plane Stress tab

Used to model plate elements as Plane Stress elements.

Refer to TR.24 Element Plane Stress and Ignore Inplane Rotation Specification (on page 2295) for additional information.

There are no additional parameters for this plate element specification.

Ignore Stiffness tab

Used to instruct the program to neglect the stiffness of the selected plates while assembling the stiffness matrix. Hence, the stiffness contribution of the selected plates is not considered. This is similar to the Inactive command for the members.

When modeling plate elements, often we come across situations where we want the plates to carry loads, but at the same time do not want the stiffness of the plates to be considered in the analysis. For example, structural units such as glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these elements are designed to carry the loads and transmit the same to the other parts of the structure, they are not designed to provide any additional stiffness to the structure.

Note: Refer to TR.22.3 Element Ignore Stiffness (on page 2288) for additional information.

There are no additional parameters for this plate element specification.

Related Links

- *G.11 Member and Plate Offsets* (on page 2125)
- *To assign plate offsets* (on page 813)
- *G.7 Member and Element Release* (on page 2120)
- *M. To assign plate corner release* (on page 814)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To assign plates as plane stress* (on page 815)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To assign inplane rotation behavior to plates* (on page 815)
- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To ignore plate stiffness* (on page 816)

Properties - Whole Strucutre dialog

Used to define and assign member, plate element and material properties.

Section tab

Used for defining member section properties and element thickness and assigning them to members and elements. Properties can be defined and assigned to members and elements using the options available on the **Section** tab. As property values are assigned, the Section Tab is updated with their tag number and property description. The items on the Section tab are described below.

Beta Angle tab

Used to define beta angles (rotation of the member about its local x-axis) and assign them to members.

Related Links

- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *M. To add a new table section property* (on page 715)

Section Profile Tables dialog

Used to assign catalog sections for steel, timber, and aluminum members. Sections or organized by material, country, and then profile.

Opens when **Section Database** is clicked on the **Properties - Whole Structure** dialog.

Tables list	Select the type of section by clicking on the appropriate tab and then select the specific section from the list box. Please note that depending on the type of section selected, additional properties may be specified.
View Table	This option displays all member properties for the aluminum table.
Material	Select this option to select the material from the drop down list if the new member property tag should include the material constants.
Type Specification	Refer to TR.20.1 Assigning Properties from Steel Tables (on page 2257) for details on type specifications for different table profiles.

Steel tab

Used to select hot-rolled and joint sections. Displays the contents of the various country steel tables in a series of tabs on the left hand side of the dialog.

Coldformed Steel tab

Used to select cold-formed steel sections.

Timber tab

Used to select standard species and size timbers as well as glue-laminated sections for American and Canadian standards.

Naming Convention	Click and hold to display a pop-up of the nomenclature conventions used for the currently selected catalog.

Aluminum tab

Used to assign aluminum sections to members from the built-in American aluminum table.

Related Links

TR.31.2.16 IBC 2015 Seismic Load Definition

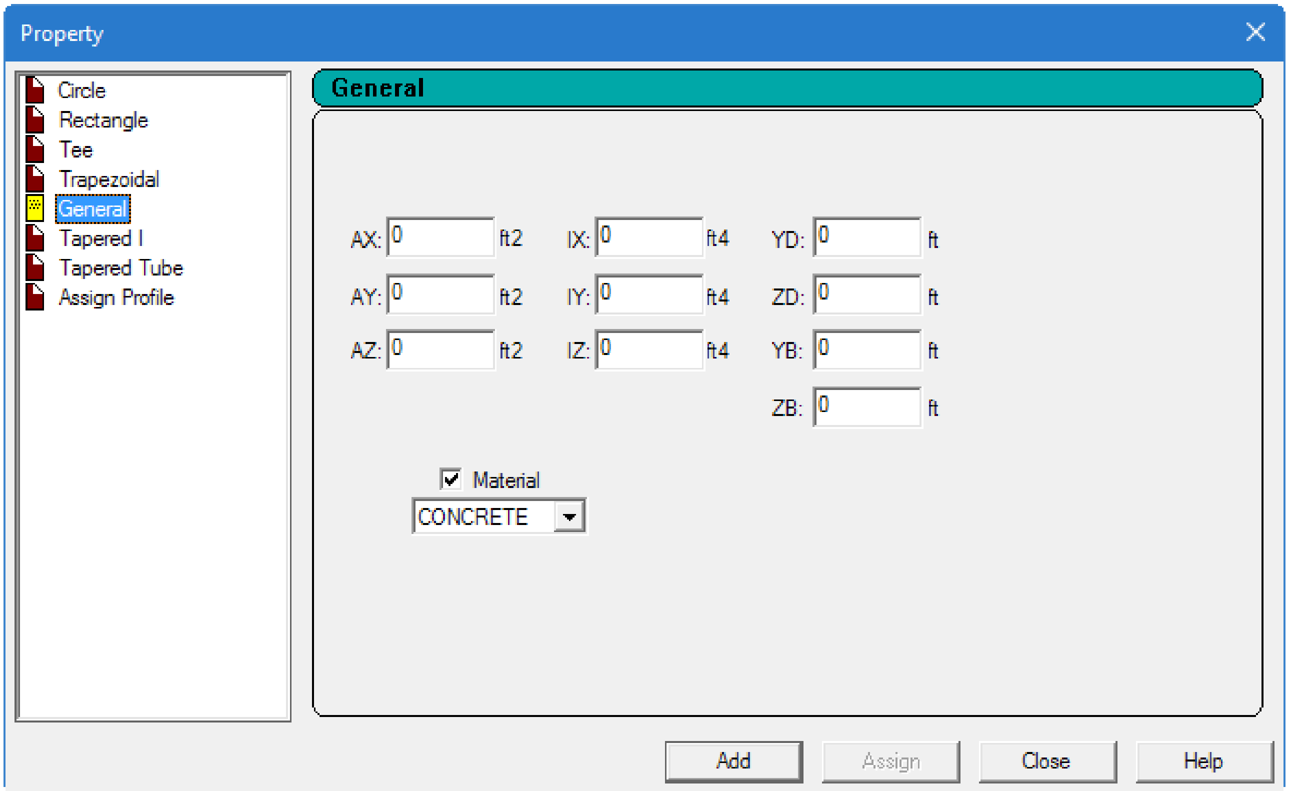
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *G.6.2 Built-In Steel Section Libraries* (on page 2116)
- *M. To add a new table section property* (on page 715)
- *M. To add an American steel joist section* (on page 716)

Property dialog

Used to assign prismatic cross sections to members.

Opens when the **Beam Profiles > Prismatic Profiles** tool is selected in the **Specifications** group on the **Specification** ribbon tab.

Note: The **Properties - Whole Structure** dialog also opens simultaneously so that some of the other options available from that dialog may be utilized.



Circle tab

YD Section diameter.

Material Check this box and select the material from the drop down list if the new member property tag should include the material constants.

Rectangle tab

YD Depth of the member in local y direction.

ZD Width of the member in the local z direction.

Tee tab

YD Depth of the member in local y direction.

ZD Width of the flange.

YB Depth of the stem (web).

ZB Width of the stem (web).

Trapezoidal tab

YD Depth of the member in local y direction.

ZD Top width.

ZB Bottom width.

General tab

AX Cross sectional area of the member.

AY Effective shear area in local y-axis.

AZ Effective shear area in local z-axis.

IX Torsional constant.

IY Moment of inertia about local y-axis.

IZ Moment of inertia about local z-axis (usually major).

YD Depth of the member in local y direction. Used as the diameter of section for circular members.

ZD Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.

YB Depth of stem for T-section.

ZB Width of stem for Tee section or bottom width for trapezoidal section.

Tapered I tab

F1 Depth of section at start node.

F2 Thickness of web.

F3 Depth of section at end node.

> **Note:** F3 should be *less than* F1 (i.e., the section should decrease in depth from start to end). You must provide the member incidences accordingly.

F4 Width of top flange.

F5 Thickness of top flange.

F6 Width of bottom flange. Defaults to F4 is zero.

F7 Thickness of bottom flange. Defaults to F5 if zero.

Tapered Tube tab

Type of Section Select one of the profile shapes:

Round

Hexdecagonal – 16-sided
Dodecagonal – 12-sided
Octagonal – 8-sided
Hexagonal – 6-sided
Square – 4-sided

d1	Depth of section at start of member.
d2	Depth of section at end of member.
th	Thickness of section (constant throughout the member length).

Notes:

1. Section properties are calculated using the rules applicable for thin-walled sections.
2. Shear deformation is not considered for tapered I-Beams and tapered poles. This means that the `SET SHEAR` command has no effect on the deformation computed for members with these cross sections.

Assign Profile tab

Used to instruct the program to select a suitable steel section based on a profile classification, such as beam, column, double-angle, etc.

Select Profile Specification

Specify a profile by selecting one of the options:

Angle
Double Angle
Beam
Column
Channel

The program will then assign a section based on that profile from the relevant built-in Steel table.

Related Links

- *G.6.1 Prismatic Properties* (on page 2114)
- *M. To assign a prismatic section* (on page 728)

Property: Tapered I dialog

Used to specify an I-section having a varying depth over the length of the member.

Opens when the **Beam Profiles > Tapered Profile** tool is selected in the **Specifications** group on the **Specification** ribbon tab.

Note: This dialog is identical to the **Tapered I** tab of the **Property** dialog.

F1 Depth of section at start node.

F2 Thickness of web.

F3 Depth of section at end node.

> **Note:** F3 should be *less than* F1 (i.e., the section should decrease in depth from start to end). You must provide the member incidences accordingly.

F4 Width of top flange.

F5 Thickness of top flange.

F6 Width of bottom flange. Defaults to F4 is zero.

F7 Thickness of bottom flange. Defaults to F5 if zero.

Related Links

- *G.6.4 Tapered Sections* (on page 2117)
- *M. To assign a tapered I section* (on page 729)

User Provided Table dialog

Used for adding and assigning previously created user table sections to the structure.

> **Note:** The Properties - Whole Strucutre dialog (on page 2793) also opens simultaneously so that some of the other options available from that dialog box may be utilized.

Opens when the **Beam Specifications** > **User Table Profile** tool is selected in the **Specifications** group on the **Specification** ribbon tab.

Select Existing Table	Select a previously created User Table number from the drop down list. User Tables are created using the User Table Manager dialog (on page 2798).
Section List	Select the specific section from the list.
Material	Check this box and select the material from the drop down list if the new member property tag should include the material constants.

Add	Click to add this property to the structure.
Assign	Click to assign the property to selected members as well as add this property to the structure.
Close	Closes the dialog without adding any properties.
Help	Opens the STAAD.Pro Help window.

Related Links

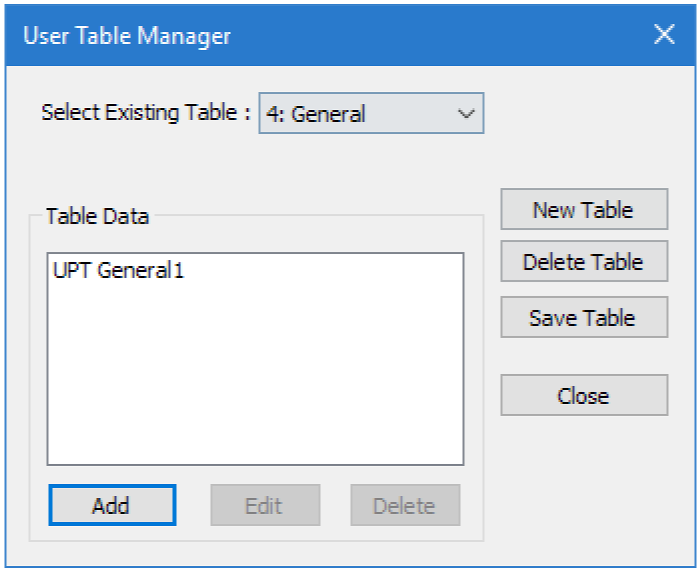
- *G.6.3 User-Provided Steel Table* (on page 2117)
- *M. To create a general section* (on page 743)
- *M.To create a wide-flange user table section* (on page 731)

User Table Manager dialog

Used to add new or edit existing user provided section properties tables.

Opens when either:

- you select a section type and click **OK** in the **Select Section Type** dialog, or
- or when you select **Beam Profiles > User Table Profile** tool in the **Specifications** group on the **Specification** tab, but no user provided tables are present.



Select Existing Table If you have already created a user provided table, you may select it from this drop-down list box for further editing.

> **Note:** If you select an external table (indicated by "(EXT)"), then the file path and file name will be displayed below.

New Table Opens the **New User Table** dialog, which is used to select the section type for a user provided table and to optional specify an external table file.

Add Click this to specify a custom UPT section after creating a new table or selecting an existing table. A dialog opens for inputting property values applicable to the section type chosen for the current table.

Once the section is defined, it may be assigned from the **Properties - Whole Structure** dialog.

> **Note:** You may not add entries to external tables from this interface.

The following values are required for the various shapes. Additional values may be entered or you may click **Calculate** for some section types to have the program calculate derived section properties.

- Wide Flange Sections - D, TF, WF, TW are required (and optionally TW1 and WF1 if the flanges are of different sizes).

  Ax, Iz, Ix, Iy, Ay, and Az can be calculated
- Channel Sections - D, TF, WF, and TW are required

  Ax, Iz, Ix, Iy, Cz, Ay, and Az can be calculated
- Angle Sections - D, WF, and TF are required

R, At, and Az can be calculated

- Double Angle Sections - D, WF, TF, and SP are required

  Iz, Ix, Iy, Cy, Ay, and Az can be calculated (does not calculate RVV)
- Tube Sections - D, WF, and TF are required

  Ax, Iz, Ix, Iy, Ay, and Az can be calculated
- General Sections - The general polygon profile is required

  Ax, D, TD, B, TB, Iz, Ix, Iy, Sz, Sy, Ay, Az, Pz, Py, HSS, and DEE can be calculated

Other section types do not have the option to calculate section properties. Therefore, any value not to be assumed as zero is required.

Edit / View	Click to open the details of a selected table data entry for viewing or editing.
Delete	Click to delete the currently selected table data entry from the table.
Delete Table	Deletes the current user provided table.
Save Table	Saves changes made to the user provided table.
Close	Closes the dialog.
Section Wizard	(Displayed only for External tables) Click to open the Section Wizard interface.

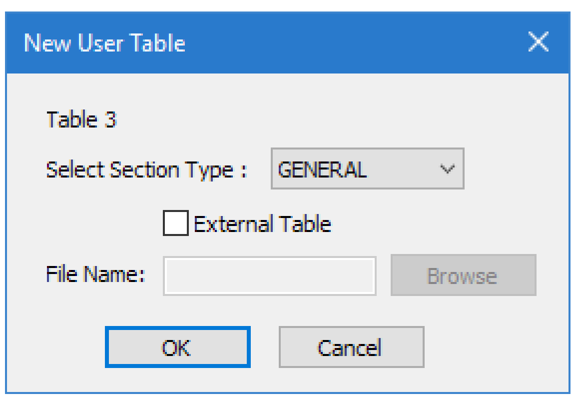
Related Links

- *G.6.3 User-Provided Steel Table* (on page 2117)
- *M. To create a general section* (on page 743)
- *M.To create a wide-flange user table section* (on page 731)

New User Table dialog

Used to select the section type for a User Provided Table and to optional specify an external table file.

Opens when **New Table** is selected in the **User Table Manager** dialog.



External Table	Set this option to load section data from an external file (e.g., a `.upt` file exported from Section Wizard). The File Name must be provided.
File Name	Type a name for the external table file. Click **Browse** to search for a location from which to load the table file.

Select Section Type	Select the type of cross section contain in the table. All sections within a table must be of the same type. • Wide Flange • Channel • Angle • Double Angle • Tee • Pipe • Tube • General • I Section • Prismatic **Note:** Refer to TR.19 User Steel Table Specification (on page 2241) for details on the property value requirements for various section types.
OK	Creates a new User Provided Table of the selected type.
Cancel	Closes the dialog without creating a new table.

Wide Flange dialog

Used to specify the section properties for a wide flange user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Wide Flange section table is selected.

Note: Refer to TR.19.1 Wide Flange (on page 2244) for detailed descriptions on properties of Wide Flange sections in User Steel tables.

Tip: To generate a tapered wide flange section, use an I Section dialog (on page 2808) instead.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of the section
TF	Thickness of top flange (or both flanges when `WF1` is not specified)
WF	Width of the top flange (or both flanges when `WF1` is not specified)
TW	Thickness of web
TF1	Thickness of bottom flange
WF1	Width of the bottom flange
Cross Section Area (Ax)	Cross section area. Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local z (Iz)	Moment of inertia about local z-axis (usually strong axis). Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local y (Iy)	Moment of inertia about local y-axis. Can be automatically calculated by clicking the **Calculate** button.

Torsional Constant (Ix)	Moment of inertia about local x-axis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Z (Az)	Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button.
Additional Composite Flange	Set this option to add a composite concrete slab to the section. **B(left)** Width of the composite slab to the left of the web center line **B(right)** Width of the composite slab to the right of the web center line **Thickness** Thickness of the composite slab **Modular Ratio** The ratio of the modulus of elasticity of steel to concrete
Additional Bottom Steel Plate	Set this option to add a bottom cover plate to the wide flange section. This may only be included if the **Additional Composite Flange** option is selected. **B(left)** Width of the additional bottom flange plate to the left of the web center line **B(right)** Width of the additional bottom flange plate to the right of the web center line **Thickness** Thickness of the additional bottom flange plate

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.
Calculate	Click to calculate the Ax, Iz, Iy, Ix, Ay, Az values from the provided section dimensions.

Related Links

- *M.To create a wide-flange user table section* (on page 731)

Channel dialog

Used to specify the section properties for a channel user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Channel section table is selected.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of the section
TW	Thickness of web
WF	Width of flange
TF	Thickness of flange
Cross Section Area (Ax)	Cross section area. Can be automatically calculated by clicking the **Calculate** button.

Inertia About Local z (Iz)	Moment of inertia about local z-axis (usually strong axis). Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local y (Iy)	Moment of inertia about local y-axis. Can be automatically calculated by clicking the **Calculate** button.
Torsional Constant (Ix)	Moment of inertia about local x-axis. Can be automatically calculated by clicking the **Calculate** button.
C.G. Along local Z (Cz)	Centroid of section along local z axis, measured from heal of channel. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Z (Az)	Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.
Calculate	Click to calculate the Ax, Iz, Iy, Ix, Cz, Ay, and Az values from the provided section dimensions.

Related Links

- *M.To create a channel user table section* (on page 733)

Angle dialog

Used to specify the section properties for a single angle user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Angle section table is selected.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of angle (i.e., length of the leg along the local y-axis)
WF	Width of angle (i.e., length of the leg along the local z-axis)
TF	Thickness of angle leg
Rad. of Gyration about Principal Axis (R)	Radius of gyration about principal axis, shown as “r(Z-Z)” in the AISC manual (this must not be zero). Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Z (Az)	Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.
Calculate	Click to calculate the R, Ay, and Az values from the provided section dimensions.

Related Links

- *M.To create an angle user table section* (on page 735)

Double Angle dialog

Used to specify the section properties for a double angle user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Double Angle section table is selected.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of angle (i.e., length of the leg along the local y-axis)
WF	Width of angle (i.e., length of the leg along the local z-axis)
TF	Thickness of angle leg
SP	Space between angles.
Cross Section Area (Ax)	Cross section area. Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local z (Iz)	Moment of inertia about local z-axis (usually strong axis). Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local y (Iy)	Moment of inertia about local y-axis. Can be automatically calculated by clicking the **Calculate** button.
Torsional Constant (Ix)	Moment of inertia about local x-axis. Can be automatically calculated by clicking the **Calculate** button.
C.G. along local Y (Cy)	Distance from the face of angles to the center of gravity (C.G.) along the local y-axis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Z (Az)	Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button.
Single Angle (rvv)	The radius of gyration about the minor principal axis for single angles - z-z axis for `TA ST` angles, and y-y axis for `TA RA` angles.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.
Calculate	Click to calculate the Iz, Iy, Ix, Cy, Ay, and Az values from the provided section dimensions.

Related Links

- *M.To create a double angle user table section* (on page 737)

Tee dialog

Used to specify the section properties for a tee user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Tee section table is selected.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of the section

TF	Thickness of flange
WF	Width of flange
TW	Thickness of web
Cross Section Area (Ax)	Cross section area.
Inertia about local z (Iz)	Moment of inertia about local z-axis (usually strong axis)
Inertia about local y (Iy)	Moment of inertia about local y-axis
IX	Torsional constant
C.G. along local Y (Cy)	Distance from back of web to center of gravity (C.G.) of the shape along the local y-axis.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
Shear Area in Z (Az)	Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.

Pipe dialog

Used to specify the section properties for a pipe user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Pipe section table is selected.

Section Name	Type a section name used to identify the shape in the table.
OD	Outer diameter
ID	Inner diameter
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
Shear Area in Z (Az)	Shear area in local z-axis. If zero, shear deformation is ignored in the analysis.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.

Tube dialog

Used to specify the section properties for a tube user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Tube section table is selected.

Section Name	Type a section name used to identify the shape in the table.
D	Depth of the section
WF	Width of flange
TF	Thickness of flange
Cross Section Area (Ax)	Cross section area. Can be automatically calculated by clicking the **Calculate** button.

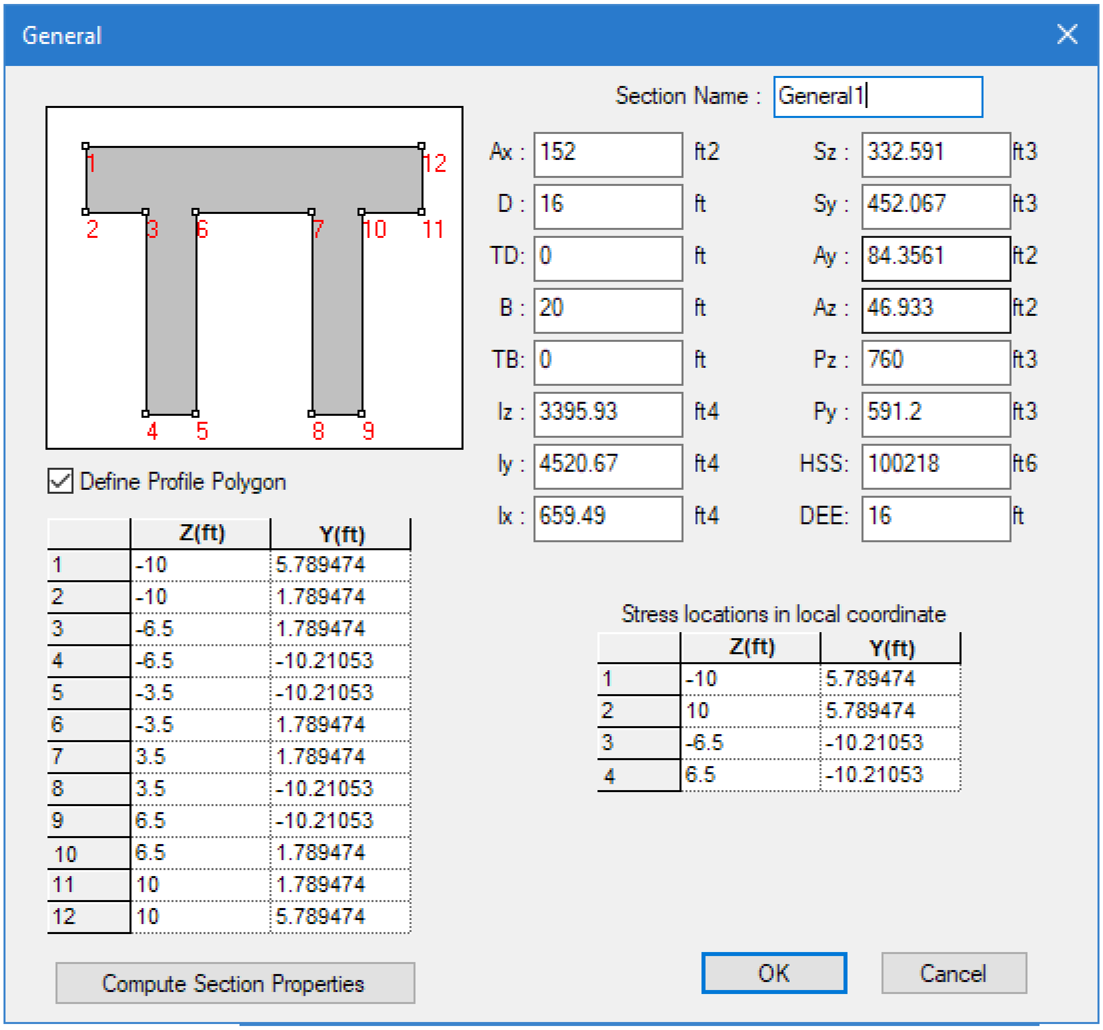
Inertia About Local z (Iz)	Moment of inertia about local z-axis (usually strong axis). Can be automatically calculated by clicking the **Calculate** button.
Inertia About Local y (Iy)	Moment of inertia about local y-axis. Can be automatically calculated by clicking the **Calculate** button.
Torsional Constant (Ix)	Moment of inertia about local x-axis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Y (Ay)	Shear area in local y-axis. If zero, shear deformation is ignored in the analysis. Can be automatically calculated by clicking the **Calculate** button.
Shear Area in Z (Az)	Same as above except in local z-axis. Can be automatically calculated by clicking the **Calculate** button.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.
Calculate	Click to calculate the AX, IZ, IY, IX, AY, and AZ values from the provided section dimensions.

General dialog

Used to specify the section dimensions and properties for a general polygonal user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a General section table is selected.



Section Name	Type a section name used to identify the shape in the table.
Define Profile Polygon	Check this option to define polygon vertices by Z and X coordinates in the following table.
	You also have the option compute the section properties for the shape automatically.
AX	Cross section area
D	Depth of the section
TD	Thickness associated with section element parallel to depth (usually web). To be used to check depth/thickness ratio
B	Width of the section
TB	Thickness associated with section element parallel to flange. To be used to check width/ thickness ratio
IZ	Moment of inertia about local z-axis (usually strong axis)
IY	Moment of inertia about local y-axis

IX	Torsional constant
SZ	Section modulus about local z-axis
SY	Section modulus about local y-axis
AY	Shear area in local y-axis.
AZ	Shear area in local z-axis.
PZ	Plastic modulus about local z-axis
PY	Plastic modulus about local y-axis
HSS	Warping constant for lateral torsional buckling calculations
DEE	Depth of web. For rolled sections, distance between fillets should be provided

Note: Properties PZ, PY, HSS, and DEE must be provided for code checking or member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

Compute Section Properties	Click to calculate the AX, D, B, IZ, IY, IX, SZ, SY, AY, AZ, PZ, PY, HSS, and DEE values from the provided polygon points.
OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.

I Section dialog

Used to specify the section properties for a I Section user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when an I Section table is selected.

This section can be of constant depth (effectively a TR.19.9 I Section (on page 2252)) or a tapered member.

DWW	Depth of section at start node
TWW	Thickness of web
TFF	Thickness of top flange
BFF	Width of top flange
TFF1	Thickness of bottom flange
BFF1	Width of bottom flange
Depth at Start Node	Depth of section at end node. This must be the larger depth of the two member nodes. You must provide member incidences accordingly.
Depth at End Node	Depth of section at end node. Must be less than the depth at the start node.
Torsional Constant (Ix)	Torsional constant (IX or J) (see note #2)
Shear Area in Y (Ay)	Shear area for shear parallel to Y-axis (see note #2)
Shear Area in Z (Az)	Shear area for shear parallel to Z-axis (see note #2)
Additional Composite Flange	Set this option to add a composite concrete slab to the section.

B(left)	Width of the composite slab to the left of the web center line
B(right)	Width of the composite slab to the right of the web center line
Thickness	Thickness of the composite slab
Modular Ratio	The ratio of the modulus of elasticity of steel to concrete

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.

Related Links

- *M.To create an I shape user table section* (on page 739)

Prismatic dialog

Used to specify the section dimensions and properties for a general, prismatic user defined table section.

Opens when you click **Add** in the **User Table Manager** dialog when a Prismatic section table is selected.

Note: This is a *non*-parametric section. That is, you must specify the all cross-sectional properties rather than dimensional parameters.

Section Name	Type a section name used to identify the shape in the table.
YD	Depth of the section in the direction of the local y-axis.
ZD	Depth of the section in the direction of the local z-axis
Cross Section Area (Ax)	Cross-section area
Inertia about local z (Iz)	Moment of inertia about the local z-axis
Inertia about local y (Iy)	Moment of inertia about the local y-axis
Torsional Constant (Ix)	Torsional constant
Shear Area in Y (Ay)	Shear area for shear parallel to local y-axis.
Shear Area in Z (Az)	Shear area for shear parallel to local z-axis.

OK	Saves the UPT section and closes the dialog.
Cancel	Closes the dialog without creating a new section.

Related Links

- *M.To create an prismatic user table section* (on page 741)

Material Constant dialog

Used to specify individual material constants of the materials of which members and elements are comprised.

Opens when a constant name is selected from the **Constants** tool in the **Materials** group on the **Specifications** ribbon tab. A separate dialog is used to define each constant value.

The types of constants available to define are:

- **Young's Modulus** – the modulus of elasticity, E
- **Poisson's Ratio**, ν
- **Shear Modulus**, G
- **Density** , γ
- **Thermal Coefficient** , α
- **Damping Ratio** ,c

Material Constant Select the pre-defined materials Aluminum, Concrete, or Steel to use a built-in value. See below for the built-in material values

Alternately, select the **Enter Value** option to type in a custom constant value in the current units.

Assign Select the scope of members which the material constant is to be assigned.

- **To View** assigns the material constant to all members within the current view.
- **To Selection** assigns the material constant to only those members selected.

Built-In Material Constants

For E, G, POISSON, DENSITY, ALPHA, and CDAMP, built-in material names can be entered instead of a value for *f1*. The built-in names are `STEEL`, `CONCRETE`, & `ALUMINUM`. Appropriate values will be automatically assigned for the built-in names.

Table 317: Constants (in Kip, inch, Fahrenheit units)

Constant	Material			Units
	Steel	Concrete	Aluminum	
E (US)	29,000	3,150	10,000	Kip/in^2
Poisson's	0.30	.17	.33	(ratio)
Density	.000283	.0000868	.000098	Kip/in^3
Alpha	6.5E-6	5.5E-6	12.8E-6	L/L/° F
CDAMP	.03	.05	.03	(ratio)
E (nonUS)	29,732.736			Kip/in^2

Table 318: Constants (in MKS, Celsius units)

Constant	Material			Units
	Steel	Concrete	Aluminum	
E (US)	199, 947,960	21,718,455	68,947,573	kN/m^2
Poisson's	0.30	.17	.33	(ratio)

Constant	Material			Units
	Steel	Concrete	Aluminum	
Density	76.819 541	23.561612	26.601820	kN/m^3
Alpha	12.0E-6	10.0E-6	23.0E-6	L/L/° C
CDAMP	.03	.05	.03	(ratio)
E (nonUS)	205, 000,000			kN/m^2

Note: E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

Related Links

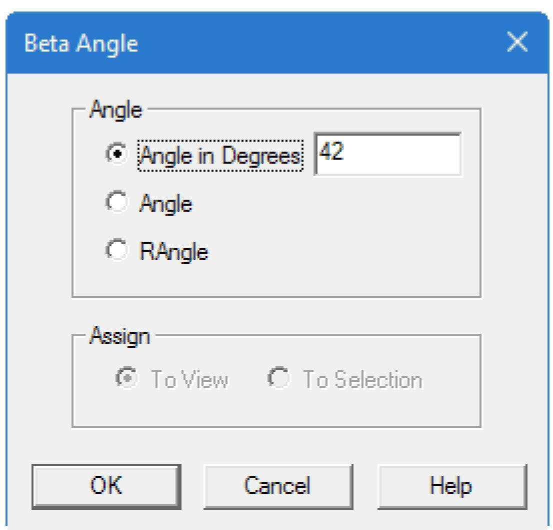
- *M. To assign material constants* (on page 798)
- *G.17.3.3.1 Composite Damping* (on page 2160)
- *M. To assign a composite damping ratio* (on page 895)

Beta Angle dialog

Used to specify the Beta angle for members.

Opens when:

- the **Beam >Beta Angle** tool is selected in the **Specifications** group on the **Specification** ribbon tab, or
- **Create Beta Angle** is clicked on the **Properties - Whole Structure** dialog **Beta Angle** tab



Angle Type the rotation about the longitudinal member axis (i.e., local x axis) in the **Angle in Degrees** field.

The **Angle** and **RAngle** options are for use only with single angle sections. These options are further described in G.4.3 Relationship Between Global and Local Coordinates (on page 2092).

Assign Select the scope of members which the geometric constant is to be assigned.

- **To View** assigns the geometric constant to all members within the current view.
- **To Selection** assigns the geometric constant to only those members selected.

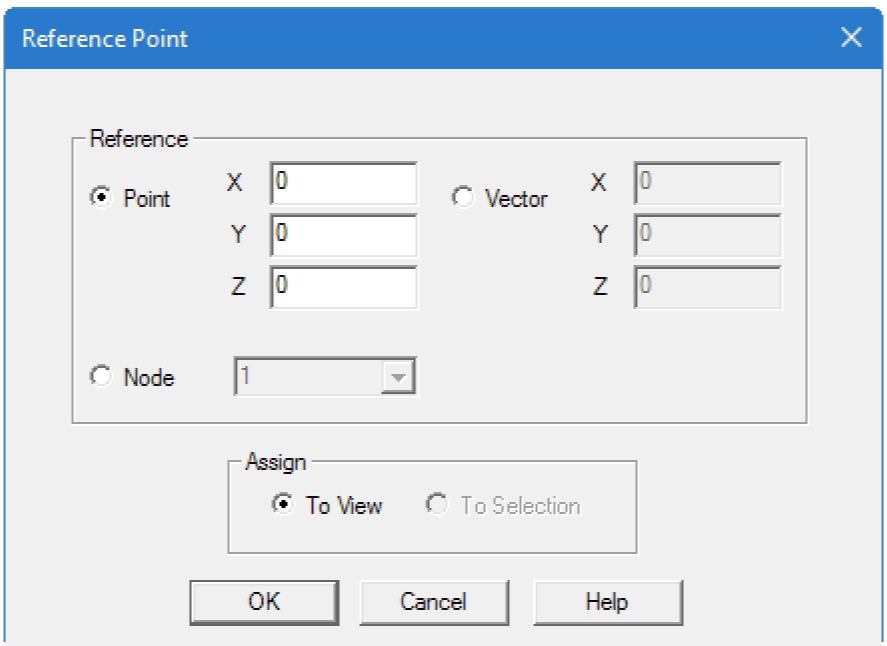
Related Links

- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *M. To assign a member rotation angle* (on page 799)
- *M. To align a single angle to its flanges* (on page 800)

Reference Point dialog

Used to specify a reference point for the members from which the program calculates the beta angle.

Opens when the **Beam Specifications > Beam Reference Point** tool is selected from the **Specifications** group on the **Specification** ribbon tab.



Point Provide the global coordinates of the point towards which the minor axis (local y axis) of the member should be directed. The point should not lie on the longitudinal axis (local x axis) of the member.

Vector A vector can be assigned with respect to the local coordinate system of the member. The local Y axis for the member is going to oriented along this vector.

This can be used to orient a member without the need to calculate a beta angle. For example, vector values of X = 0, Y = 2, and Z = 1 will create a vector at a slope of 1:2 from the member's axis. The resulting beta angle = $tan^{-1}(1/2) = 26.5651°$.

Refer to AD.2006.1.6 Specification of Member Orientation Using Reference Vector (on page 344) section for additional information on this feature.

> **Note:** The **Member Query** dialog **Geometry** tab will report the actual beta angle used.

Node An existing node in the structure can be set as the reference point for one or more members. This is the alternative to defining the coordinates of the reference point. When this button is turned on, the list of all nodes in the structure will appear in the drop-down list. Select the desired node number.

Assign Select the scope of members which the geometric constant is to be assigned.

- **To View** assigns the geometric constant to all members within the current view.
- **To Selection** assigns the geometric constant to only those members selected.

Related Links

- *G.4.3 Relationship Between Global and Local Coordinates* (on page 2092)
- *M. To align a member to a reference point* (on page 801)

Plate ElementProperty dialog

Used to provide plate element properties (thickness) with or without the material specification. Plates can have a different thickness at each node.

Opens when:

- the **Plate Thickness** tool is selected from the **Plate Profiles** group on the **Specification** ribbon tab, or

- **Thickness** is clicked on the **Properties - Whole Structure** dialog

Node 1, Node 2, etc. Enter the thickness at each of the nodes. For plates of uniform thickness, entering the thickness at Node 1 only will suffice. The other three values will default to that of Node 1 if no input is provided for them.

Material Check this box and select the material from the drop down list if the new element property tag should include the material constants. The specification of the standard materials (that is, Concrete, Steel, or Aluminum) can be added in this dialog box or can be done separately with other material specifications.

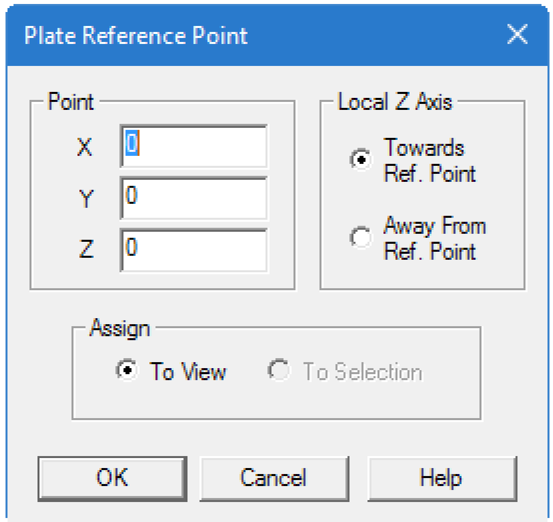
Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To specify plate thickness* (on page 811)

Plate Reference Point dialog

Used to specify options for determining the general direction of the local Z axis of elements.

Opens when the **Plate Specifications > Plate Reference Point** tool is selected from the **Specifications** group on the **Specification** ribbon tab.



Point Type the global coordinates of a point to use for orienting the local z axis of the plate elements.

> **Note:** The local coordinate system of the plate must still obey that the xy plane lie in the plane of the element itself. So the z axis won't necessarily point directly to this coordinate. Rather, the z axis will point away from the element to the side that the reference point is located.

Local Z Axis Select an option to define the direction of the local z axis of the plate element with reference to the **Point** provided.

Assign Select the scope of plates which the geometric constant is to be assigned:

To View – assigns the plate reference point to all plate elements within the current view.
To Selection – assigns the plate reference point to only those plate elements selected.

Related Links

- *G.5.1 Plate and Shell Elements* (on page 2099)
- *M. To align a plate to a reference point* (on page 811)

Supports - Whole Structure dialog

Used to define supports and assign them to nodes. All supports that have been defined for the model are listed here.

Edit Click to edit the parameters of the one of the following support types after it is created.

- Fixed But
- Enforced But
- Inclined
- Foundation

> **Note:** You cannot edit support types which are defined, such as fixed, pinned, or enforced.

Create Opens the **Create Support** dialog opens, which is used to create various support types.

Delete	Click to delete a previously assigned support.
Assignment Method	The options under the Assignment Method offer different choices for assigning supports to the structure.

Assign To Selected Nodes	To assign a support to selected nodes, first select the support from the Supports dialog box. The support selected is highlighted. Then select the nodes to which this support is to be assigned. When all desired nodes are selected, click the Assign To Selected Nodes radio button, then click the Assign button.
Assign To View	To assign a support to all free nodes in a view, first select the support from the Supports dialog box. The selected support is highlighted. Select the Assign To View radio button, then click the Assign button. All free nodes in the structure are assigned this support.
Use Cursor To Assign	To assign a support to nodes using the cursor, first select the support from the Supports dialog box. The selected support is highlighted. Select the Use Cursor To Assign radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Make sure that the Nodes Cursor is selected so that we can select the nodes. Using the cursor, click on the nodes to which this support is to be assigned. Click on the Assign button again to finish.
Assign To Edit List	To assign a support using a typed list of node numbers, first select the support from the Supports dialog. The selected support is highlighted. Select the Assign To Edit List radio button, then type the list of node numbers and click the Assign button.

Related Links

- *M. To assign a fixed or pinned support* (on page 819)
- *G.13 Supports* (on page 2127)

Create Support dialog

Used to define support (boundary) conditions of the structure to nodes. The dialog consists of a series of tabs, each of which are used to assign a different support condition. Each tab includes options for the specified support condition.

Note: The **Supports - Whole Structure** dialog also opens simultaneously so that some of the other options available from that dialog box may be utilized.

If the support is not assigned as it is created, it can be assigned later from the **Support** dialog. However, note that if it is not assigned to at least one spring support when is closed, then the definition will not be saved in the STD file.

Fixed tab

Used to create a fixed support tag and (optionally) to assign that to selected nodes. A Fixed support is restrained in all six degrees of freedom.

Refer to TR.27.1 Global Support Specification (on page 2316) for additional information on global supports.

Opens when **Add Support > Fixed** is selected from the **Specification** ribbon tab **Supports** group.

There are no settings for assigning a Fixed support.

Pinned tab

Used to create a pinned support tag and (optionally) to assign it to selected nodes. A Pinned support is restrained in all three translational degrees of freedom and free in the 3 rotational degrees of freedom.

Refer to TR.27.1 Global Support Specification (on page 2316) for additional information on global supports.

Opens when **Add Support > Pinned** is selected from the **Specification** ribbon tab **Supports** group.

There are no settings for assigning a Pinned support.

Fixed But tab

Used to create various types of roller, hinge and spring supports with specified restrained degrees of freedom and to assign them to selected nodes.

Refer to TR.27.1 Global Support Specification (on page 2316) for additional information on global supports.

Opens when **Add Support** > **Fixed But/Spring** is selected from the **Specification** ribbon tab **Supports** group.

Release	Select the degrees of freedom you want to release.
Define Spring	To specify a spring support, the spring constant in the appropriate edit box.

Enforced tab

Used to create a fixed-condition support definition. The Enforced support is the same as a Fixed support except that the restrained degrees of freedom are defined in terms of being stiff springs. The real advantage of using the Enforced type lies in the fact that it enables STAAD.Pro to accept loads such as support displacement loads in the case of plates and solids. Support displacement loads are not permitted for plates and solids if the FIXED support type is used. So, for structures without these characteristics, the Fixed type of support offers the same level of functionality as the Enforced support type.

Refer to TR.27.1 Global Support Specification (on page 2316) for additional information on global supports.

There are no settings for assigning an Enforced support.

Enforced But tab

Used to create a fixed-condition support defined in terms of being stiff springs with the option to release selected degrees of freedom. The Enforced But support type is the same as the Enforced support except that we have a choice on the degrees of freedom we wish to restrain. For example, you can select Enforced But and restrain just the FX, FY, and FZ degrees of freedom, and let the remaining three free to deform.

Refer to TR.27.1 Global Support Specification (on page 2316) for additional information on global supports.

Release	Select the degrees of freedom you want to release.

Inclined tab

Used to create supports that offer restraints in an axis system that is inclined with respect to the global axis system.

Refer to TR.27.2 Inclined Support Specification (on page 2318) for additional details on specifying inclined supports.

Incline Reference Point	Specify a point in space to define the incline.
Support Type	Select to use one of the standard STAAD.Pro support types: Pinned, Fixed, Fixed But, Enforced, or Enforced But.
Release	(Fixed But or Enforced But options only) Select the forces and moments to release by checking the respective box.
Spring	(Fixed But option only) To specify a spring support, the spring constant in the appropriate edit box.

Foundation tab

Used to create spring supports for independent footings and mat foundations and to assign them to selected nodes.

Refer to TR.27.3 Automatic Spring Support Generator for Foundations (on page 2319) for additional details on Foundation supports.

Foundation

- Footing - Select this option to define a spring support for an isolated footing. Provide the Length (**L**) and Width (**W**) dimensions of the footing in current units.

  In generating spring supports for mat foundations, there are two methods available in STAAD.Pro. Both those options require the program to calculate the influence area of the nodes which define the surface, and then multiply that area by the subgrade modulus of the medium. The difference between these two options lies in the way the influence area is calculated.
- Elastic Mat - In this method, the area is calculated using a Delaunay triangle principle. Hence, the candidates for this option are the nodes which define the mat. To achieve best results, one needs to ensure that the contour formed by the nodes form a convex hull.
- Plate Mat - If the foundation slab is modeled using plate elements, the spring supports can be generated using an influence area calculated using the principles used in determining the tributary area of nodes from the finite element modeling standpoint. Hence, the candidates for this option are the plates which define the mat. When the mat is modeled using plates, this produces superior results than the ELASTIC MAT type.

Direction

The X,Y, Z, XONLY, YONLY, and ZONLY indicate the direction of resistance of the spring supports. If X ,Y or Z is selected, then a spring support is generated in that direction only whereas the associated rotational degree of freedom and the other two translational d.o.f receive a fixed support. For example, if Y is selected, then FY is supported by a spring support, where as MY, FX and FZ are fixed supports; and MX and MZ are free. If XONLY,

	YONLY, or ZONLY is selected, then a spring support in that direction alone is generated, and every other d.o.f is set to be free to deform.
Subgrade	Type a value for the subgrade modulus of the soil.
Print Influence Area of Each Joint	(For Mat footing options only) Select this option to have the influence area of each support included in the output. The are used in the calculation of the spring stiffness of each joint used when defining a Plate Mat or Elastic Mat command will be included.
Generate Compression Only/Multi-linear Spring	(For Mat footing options only) When the Compression Only option is set, then if after any of the cycles of analysis, the force at a node included in the command range (in the elastic mat range or used to define a plate in the plate mat range) is found to be tensile (i.e. negative reaction), then the load case is marked for a re-analysis with that support removed.

Multilinear Spring tab

Used to enter displacement and spring stiffness data pairs for a multilinear spring definition, which is then used for a Fixed But support type.

The Multi Linear support type allows the user to model the support type for which the resistance offered to external loads varies with the extent of deformation of the support node. For example, when its behavior in tension differs from its behavior in compression, as in the case of soil springs, this facility can be utilized. Another example is a partial roller support where translation can occur without any resistance for a certain amount of displacement, after which it becomes fully restrained. The problem is solved iteratively using cycles of analysis and convergence checks. Hence, only on static load case can be specified per analysis command. It cannot be used with dynamic load cases.

Refer to TR.27.4 Multilinear Spring Support Specification (on page 2322) manual for additional information.

Multilinear Spring table	Values are entered in order from least to greatest displacement (including negative displacements, which are then ordered first) and their corresponding spring stiffness values.

Tension/Compression Only Spring tab

Used for support types which are capable of unidirectional action only, such as soil under a foundation slab.

Refer to TR.27.5 Spring Tension/Compression Specification (on page 2324) for additional details on Tension/ Compression Only Spring supports.

Reaction Type

Select whether the support degree(s) of freedom (d.o.f) are tension only or compression only. Indicates that if, after any of the cycles of analysis, the direction of the force in the spring is of the wrong 'type', then the support will be removed from that direction and a new analysis performed.

Reaction Type	Support will be active if the reaction is…	Support will be made inactive and a new analysis will be performed if the reaction is…
Tension Only	- ve	+ ve

Reaction Type	Support will be active if the reaction is…	Support will be made inactive and a new analysis will be performed if the reaction is…
Compression Only	+ ve	- ve

Spring Direction Select the degree(s) of freedom to set as unidirectional. More than one d.o.f may be set at a time.

Related Links

- *M. To assign a fixed or pinned support* (on page 819)
- *G.13 Supports* (on page 2127)
- *M. To assign a foundation support* (on page 824)
- *G.13 Supports* (on page 2127)
- *EX. US-23 Spring Support Generation for a Slab on Grade* (on page 6484)
- *EX. UK-23 Spring Support Generation for a Slab on Grade* (on page 6771)
- *M. To assign an enforced support* (on page 820)
- *G.13 Supports* (on page 2127)
- *EX. US-24 Analysis of a Concrete Block Using Solid Elements* (on page 6495)
- *EX. UK-24 Analysis of a Concrete Block Using Solid Elements* (on page 6782)
- *M. To assign an inclined support* (on page 823)
- *G.13 Supports* (on page 2127)
- *EX. US-19 Inclined Supports* (on page 6458)
- *EX. UK-19 Inclined Supports* (on page 6744)
- *M. To assign a spring support* (on page 821)
- *G.13 Supports* (on page 2127)
- *EX. US-3 Soil Springs for Portal Frame* (on page 6327)
- *EX. UK-3 Soil Springs for Portal Frame* (on page 6611)
- *M. To assign support springs as a one-way* (on page 823)

Material - Whole Structure dialog box

Used to define, manage, and assign materials used in the STAAD.Pro input file. The **Isotropic** tab is used for defining isotropic materials and assigning them to members and elements. The **Orthotropic 2D** tab is used for defining materials with different properties in two dimensions and assigning them to members and elements.

Material list All materials that have been defined for the model are listed here. Select the Isotropic or Orthotropic 2D tab to display materials which exhibit those directional properties.

We may specify Materials in two ways. We may first create a material tag and then select the members or elements to which this tag is applied. Alternatively, we may first select the members or elements and then specify a material to be assigned to the selected items. In the second case, a new material specification is created along with a material tag.Please note that the Assign button becomes active if you have already selected the members or elements to which the material tag is to be applied.

Highlight Assigned Geometry When this box is checked and we click on a material tag in the Material dialog box, members that have been assigned the material tag will appear highlighted in the structure window. For example in the previous figure, if we click on the STEEL material tag, all steel members will be highlighted.

Create When the Isotropic tab is selected, clicking **Create** opens the Isotropic Material dialog (on page 2821), which is used to add a new isotropic material and properties to the STAAD.Pro input file. When the Orthotropic2D tab is selected, this button opens the 2D Orthotropic Material Property dialog (on page 2822), which is used.

Edit Opens the Isotropic Material dialog (on page 2821) or the 2D Orthotropic Material Property dialog (on page 2822), which can be used to edit the properties for the currently selected material (the name field is inactive when editing).

Delete Deletes the currently selected material from the list.

Tip: If you accidentally delete one of the predefined materials, they can still be added using the Create button and selecting the predefined material name from the drop-down list.

Assignment Method The options under the Assignment Method are used to assign material tags to members and elements.First select a material tag from the Materials dialog box.Use one of the following assignment methods to apply this material: Assign To Selected Beams, Assign To View, Use Cursor To Assign or Assign To Edit List.

Option	Description
Assign To Selected Beams	To assign a material to selected members or elements, first select the material tag from the **Materials** dialog box. The selected material is highlighted.Then select the members or elements to which this material is to be assigned.This is done by going to the **Select** menu, then choosing the appropriate cursor option.Select the desired members or elements using the cursor.When all desired geometry is selected, click the **Assign To Selected Beams** radio button, then click the **Assign** button. Note that the label for this button changes depending on whether members or elements are selected.
Assign To View	To assign a material to all members or elements in a view, first select the material tag from the **Materials** dialog box.The selected material is highlighted.Select the **Assign To View** radio button, then click the **Assign** button.All members or elements in the structure are assigned this material.

Option	Description
Use Cursor To Assign	To assign a material to members or elements using the cursor, first select the material tag from the **Materials** dialog box.The selected material is highlighted.Select the **Use Cursor To Assign** radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Use the appropriate cursor (from the Select menu) to assign the selected material to the individual members or elements. Click on the Assign button again to finish.
Assign To Edit List	To assign a material using a typed list of member or element numbers, first select the material tag from the **Materials** dialog box.The selected material is highlighted.Select the **Assign To Edit List** radio button, then type the list of member or element numbers and click the **Assign** button.

Assign Assigns the material per the selected assignment method.

Close Closes the dialog.

Isotropic Material dialog

Used to add a new isotropic material and properties to the STAAD.Pro input file.

Note: Steel, Concrete, Stainless Steel, and Aluminum are default materials in the program and are always included.

Title Enter a unique name to use for a new material. The drop-down list provides the predefined materials available in STAAD.Pro.

Material Properties Define each of the material properties:

- Young's Modulus (E)
- Poisson's Raio (nu)
- Density
- Thermal Coeff (a)
- Critical Damping
- Shear Modulus (G)

Type of Material Select a predefined material type for associating Design Properties with the material. If one of the predefined materials is not appropriate, select Not Specified and no Design Properties will be assigned.

Yield Stress (Fy)	(Steel only)
Tensile Strength (Fu)	(Steel only)
Yield Strength Ratio (Ry)	(Steel only)
Tensile Strength Ratio (Rt)	(Steel only)
Compressive strength (Fcu)	(Concrete only)

Related Links

- *M. To create a material definition* (on page 792)

2D Orthotropic Material Property dialog

Used to add a new orthotropic material and properties to the STAAD.Pro input file.

Refer to TR.26.1 Define Material (on page 2303) for additional information.

Title	Enter a unique name to use for a new material. The drop-down list provides the predefined materials available in STAAD.Pro.
Material Properties	Define each of the material properties for the local X and Y directions: • **Young's Modulus** (E) – modulus of elasticity • **Thermal Coefficient** (a) The general properties are not orthotropic: • **Density** • **Critical Damping** • **Poisson's Ratio** (ν) The **Shear Modulus** (G) property is defined as follows: • **Gxy** – in-plane shear • **Gyz** – shear transverse to the local yz plane • **Gxz** – shear transverse to the local xz plane

Related Links

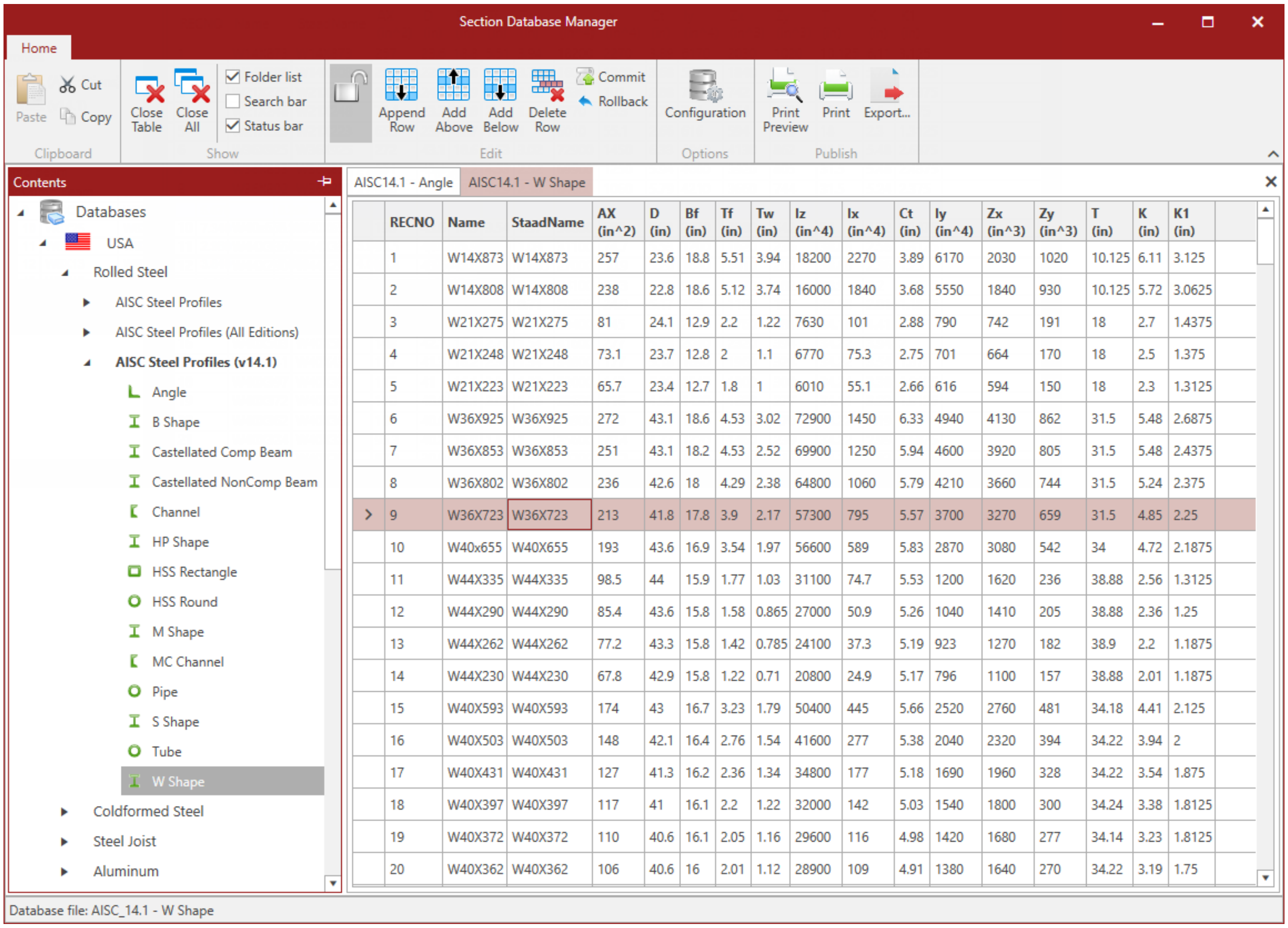
- *M. To create an orthotropic material* (on page 795)

Section Database Manager window

Used to view or edit the existing steel databases by country code and section type. On selection of the section type, the adjacent list box displays the available sections defined in the database. Selection of section in the list box enables the buttons for operating on the database.

In the Analytical Modeling workflow, this window opens when the **Section** > **Section Database** tool is selected in the **Specifications** group on the **Specification** ribbon tab.

In the Physical Modeling workflow, this window opens when the **Section Database** tool is selected in **Sections And Materials** group on the **Catalog** ribbon tab.



Double-clicking on a section opens a new dialog where there are buttons for operating on the database.

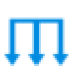
Table 319: SectionDBMAnager Home ribbon tab

Tool	What it Does
Close Table	Closes the currently selected table.
Close All	Closes all open tables.
Print	Click to open the currently displayed table in a Print Preview window, where you can print the contents to a Windows printer.
Export	Click to export the currently displayed table as a Microsoft® Office Excel® spreadsheet (file extension `.xslx`).

Tool	What it Does
Lock / Unlock	Click to unlock the table for editing. Editing-related tools (e.g., **Append Row**) are made active when unlocked. Click again to lock the table against further editing.
Append Row	Adds a new section to the database under the selected country and section type. It displays the following dialog box for the input of section properties.
Add Above	Click to add a new record befor the currently selected row.
Add Below	Click to a new record after to the currently selected row.
Delete Row	Deletes the selected section in the list box from the database under the selected country and section type.
Commit	Saves changes made to the current table.
Rollback	Undoes the any changes made to the current table.
Font controls	Use to change the display font of tables.
Search	Use to filter the table for the typed string. Click the "X" to clear the filter and display the entire table again.

Loading tab

Table 320: Loading Specifications group

Tool name	Description
 **Primary Load Cases**	Opens the **Create Primary Load Case** dialog, which is used to create new primary load cases.
 **Combination Load Case**	Opens the **Define Load Combinations** dialog, which is used to specify a load case that combines results of analysis performed for different primary load cases. **Note:** That this option combines the results of the analysis in the specified manner. It does *not* analyze the structure for the combined loading.

Tool name	Description
Reference Load Case	Opens the **Create Primary Load Case** dialog, which is used to create new reference load definition.
Load Items	Opens the **Create New Load Items** dialog, which is used to

Table 321: Loading group

Tool name		Description
Vehicle Load Generator		Opens the **Create New Load Generation** dialog, which is used to create a primary load case using the data of a pre-defined vehicle.
Wind Load Generator >	**IS 875 (Part 3) : 2015**	Opens the **Wind Load Generation - IS 875 (Part 3) : 2015** dialog, which is used to specify wind load definition data as well as generate wind load cases for the Indian IS-875 (Part 3): 2015 code.
Mass Model Generator		Opens the **Create Mass Model** dialog, which is used to generate a mass model for dynamic load cases based on a selected set of load cases.
Primary Load Type		Opens the **Define Load Type** dialog, which is used to define load types for multiple primary load cases in a single dialog.
	Add Auto Load Combintions	Opens the **Auto Load Combination** dialog, which is used to automatically create load combinations based on code-specified combination rules and primary load type selections.

Tool name		Description
Automatic Combinations >	**Edit Load Combination Rules**	Opens the **Edit Load Rules for Auto Load Combination Generator** dialog, which is used to edit load combination rules for the automatic generation of load combination cases.

Table 322: Define Load Systems group

Tool name	Description
Wind	Opens the **Create New Wind Type Definition** dialog, which is used to add a new wind load definition.
Direct Analysis	Opens the **Add New Direction Analysis Definition** dialog, which is used to define parameters used in the direct analysis method described in AISC 360-05 Appendix 7.
Snow	Opens the **Add New Snow Definition** dialog, which is used to define parameters used for generating snow loading on a structure per the ASCE 7-02 code.
Vehicle	Opens the **Add New Vehicle Definitions** dialog, which is used to define different types of moving loads. Several load cases can be generated by applying these types of loads.
Seismic > <country list>	Opens the **Add New Seismic Definitions** dialog, which is used to define the parameters for performing a dynamic analysis using the static equivalent approach as outlined in the various seismic codes supported by STAAD.Pro.
Pushover	Opens the **Add New: Pushover** dialog, which is used to define structural parameters for a pushover analysis.

Table 323: Dynamic Specifications group

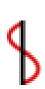
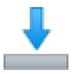
Tool name		Description
 **Time History >**	**Forcing Function**	Opens the **Add New Time History Definitions** dialog, which is used to define the Forcing Function of a time varying load.
	Parameters	Opens the **Define (Time History) Parameters** dialog, which is used to define time step, damping, and arrival times for time history loads.
 **Modal Damping**		Opens the **Modal Damping** dialog, which is used to define unique damping ratios for the individual modes used in a dynamic analysis.

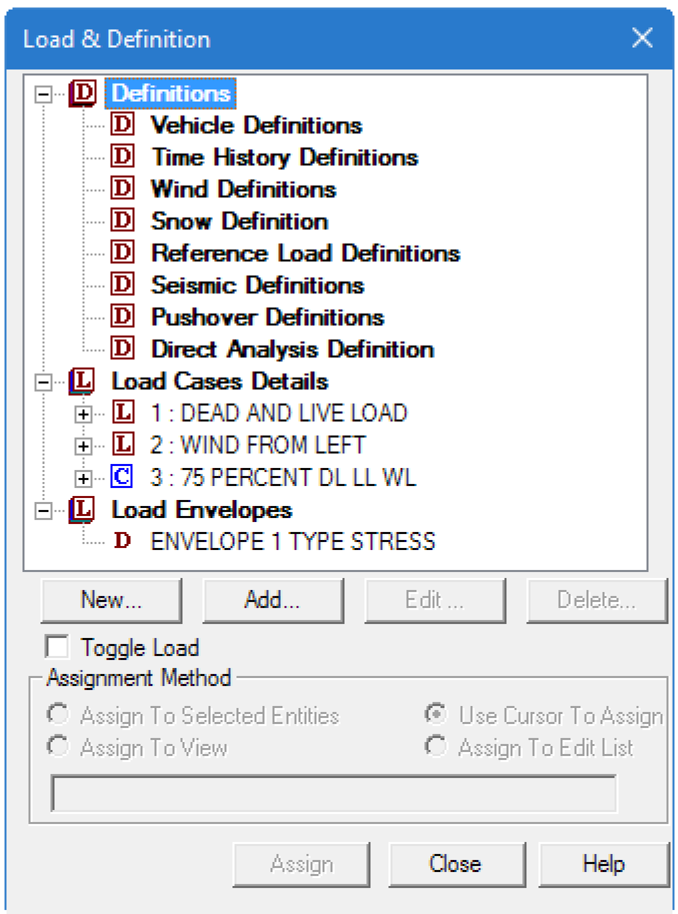
Table 324: Display group

Tool name	Description
Load	Select the active load case, load combination, or load envelope from the pop-up dialog.
 **View Loading Diagram**	Click to toggle the display of the current load case on the structure.

Load & Definition dialog

Used to create and assign load types to the structure.

Opens when the **Loading** page is selected in the **Analtyical Modeling** workflow.



Load & Definitions list	• **Definitions -** contains the options through which you create the `DEFINE` block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases. • **Load Cases Details -** contains the dialog boxes through which load cases can be created and loading data added to such cases. • **Load Envelopes -** Lists all load envelopes contained in the model. Selecting the Load Envelopes and click the Add button to open the Add New Load Envelopes dialog.
New	Opens the Create New Definitions/Load Cases/Load Items dialog, which is used to define new Primary Load Cases.
Add	Click to add the newly created load under the current load case in the Load dialog box.
Edit	Used to modify a previously assigned load component. When clicked, the corresponding load dialog is opened with additional controls for editing the load and structural objects to which the load is applied.
Delete	Removes a previously assigned load case or load component.
Toggle Load	Select this option to enable what is called the toggle mode. In this mode, when an attribute is selected and assigned using the **Use Cursor to assign** method, the following happens. • Click on the member or element once - the attribute gets assigned. • Click on the same member or element a second time - the attribute gets de-assigned. • Click on the same member or element again, - the attribute gets re-assigned.
Assignment Method	These options are used to assign predefined loads to nodes, members, and elements. First select a predefined load specification from the Loads & Definitions list. Use one of the following assignment methods to apply this load: Assign To Selected Beams, Assign To View, Use Cursor To Assign or Assign To Edit List. • **Assign To Selected Beams -** To assign a load to selected nodes, members or elements, first select the load from the Loads dialog box. The selected load is highlighted. Then select the nodes, members or elements to which this load is to be assigned. This is done by going to the Select menu, then choosing the appropriate cursor option. Select the desired nodes, members or elements using the cursor. When all desired geometry is selected, click the Assign To Selected Beams radio button, then click the Assign button. Note that the label for this button changes depending on whether nodes, members or elements are selected. • **Assign To View -** To assign a load to all nodes, members or elements in a view, first select the load from the Loads dialog box. The selected load is highlighted. Select the Assign To View radio button, then click the Assign button. All nodes, members or elements in the structure are assigned this load. • **Use Cursor To Assign -** To assign a load to nodes, members or elements using the cursor, first select the load from the Loads dialog box. The selected load is highlighted. Select the Use Cursor To Assign radio button, then click the Assign button. The button will appear depressed and the label will change to Assigning. Use the appropriate cursor (from the Select menu) to assign the selected load to the individual nodes, members or elements. Click on the Assign button again to finish. • **Assign To Edit List -** To assign a load using a typed list of node, member or element numbers, first select the load from the Loads dialog box. The selected load is highlighted. Select the Assign To Edit List radio button, then type the list of node, member or element numbers and click the Assign button.

Related Links

- *M. To edit a previously assigned load* (on page 901)

Create New Definitions / Load Cases / Load Items dialog box

Used to define new Load Definitions, Primary Load Cases, individual Load Items, and Load Envelopes. The Create New Definitions / Load Cases / Load Items dialog box acts as a container for many load- related dialogs.

Opens when **New...** is clicked in the **Load & Definitions** dialog.

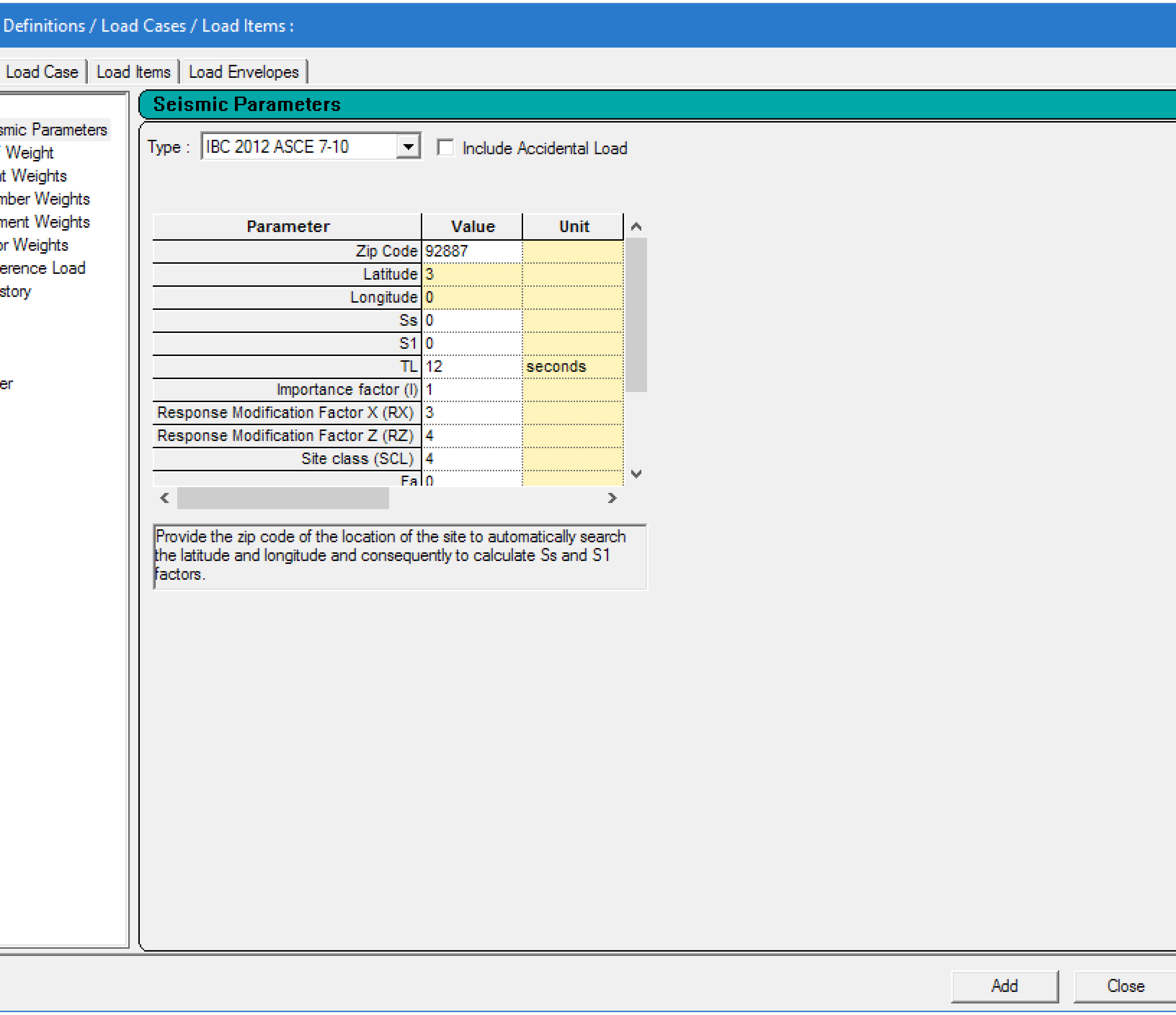


Figure 342: Create New Definitions dialog Seismic Parameters tab

This dialog contains the following tabs:

Definitions tab

This contains the options through which one creates the “Define” block of data required to create wind load cases, seismic load cases like IBC and UBC, moving load cases and time history load cases. The command syntax for such cases is explained in TR.31 Definition of Load Systems (on page 2345) .

Tab name		Dialog Contained
Seismic	Seismic Parameters	Add New Seismic Definitions dialog (on page 2895)
	Self Weight	
	Joint Weights	
	Member Weights	
	Element Weights	
	Floor Weights	
	Reference Load	
Time History	Define Time History	Add New Time History Definitions dialog (on page 2907)
	Define Parameters	Define (Time History) Parameters dialog (on page 2909)
Moving	Define Load	Add New Vehicle Definitions dialog (on page 2899)
	AASHTO Spec	
	File Input	
Wind		Create Wind Type Definition dialog (on page 2880)
Snow		Add New Snow Definition dialog (on page 2894)
Pushover	Define Input	Add New : Pushover dialog (on page 2900)
	Define Loading Pattern	
	Define Spectrum Details	
	Define Hinge Property	
	Define Solution Control	

Load Case tab

Used to generate new load cases (primary and load combination) as well as moving load generations.

Tab name		Dialog Contained
Primary		Create Primary Load Case dialog (on page 2835)
Moving	Load Generation	Load Generation dialog (on page 2866)
	Load Generation Type	(Moving) Load Generation Type dialog (on page 2832)
Combination	Define Combination	Define Load Combinations dialog (on page 2836)
	Auto Load Combination	Auto Load Combination dialog (on page 2872)

Load Items tab

Load Items contains the dialog boxes through which loading data can be added to load cases.

This tab contains the Add New Load Items dialog (on page 2839).

Load Envelopes tab

This tab contains the Add New Load Envelopes dialog (on page 2833).

Add Adds the specified load data to the model.

Close Closes the dialog without adding any load data.

Help Opens the STAAD.Pro Help window.

(Moving) Load Generation Type dialog

Used to specify direction on plane on the structure to simulate the movement of a predefined vehicle for a moving load definition.

Note: If a moving load generation command has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Load Generation dialog (on page 2866) to learn more about defining a moving load.

Opens when **Add** is clicked from the **Load & Definition** dialog when **Vehicle Definitions** is selected.

Note: Refer to TR.31.1 Definition of Moving Load System (on page 2346) for additional information.

Type Reference a previously defined moving load (e.g., vehicle definition) in the input file. Vehicles definitions are created using the Add New Vehicle Definitions dialog (on page 2899).

Range (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is added and subtracted to the reference vertical coordinate (y1 or z1)in the global vertical direction to form a range. The moving load will be externally distributed among

	all members within the vertical range thus generated. r always should be a positive number. In other words, the program always looks for members lying in the range Y 1 and Y1+ABS(r) or Z1 and Z1+ABS(r). The default r value is very small, so entering r is recommended.
Initial Position of Load	Global X, Y, and Z coordinates of the initial position of the reference load.
Load Increment	Incremental values of position of load system coordinates (in global axes) to be used for generation of subsequent load cases. **Note:** The defined vertical direction (e.g., either Y-up or Z-up) is not available to incremental placement of the moving load.

Related Links

- *M. Moving Loads* (on page 873)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 6398)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 6684)
- *M. To add vehicles to the load generation* (on page 874)

Add New Define Starting Mass Load dialog

Used to explicitly define the starting load vector for use with Ritz vector analysis.

The direction selected will extract a single set of Ritz vectors. This is thus best suited when the structural response is predominant in one translation direction. If responses in multiple translational degrees of freedom are predominant, then it is recommended that the program generated starting load vector is used (i.e., do not define a starting mass load with the load dependent Ritz vector eigen solution method for this case).

For each direction selected, enter the number of vectors to generate in that direction.

Tip: The number of vectors should typically be between 10 and 35, inclusive. If this number of vectors fails to achieve 90% mass participation in the direction of interest, then it is recommended to use full Ritz vector or Eigen solution instead.

X Direction Only
Y Direction Only
Z Direction Only
X and Y Directions
Y and Z Directions
Z and X Directions

Related Links

- *G.17.3.1 Solution of the Eigenproblem* (on page 2155)
- *M. To use starting vectors with load-dependent Ritz vectors* (on page 885)

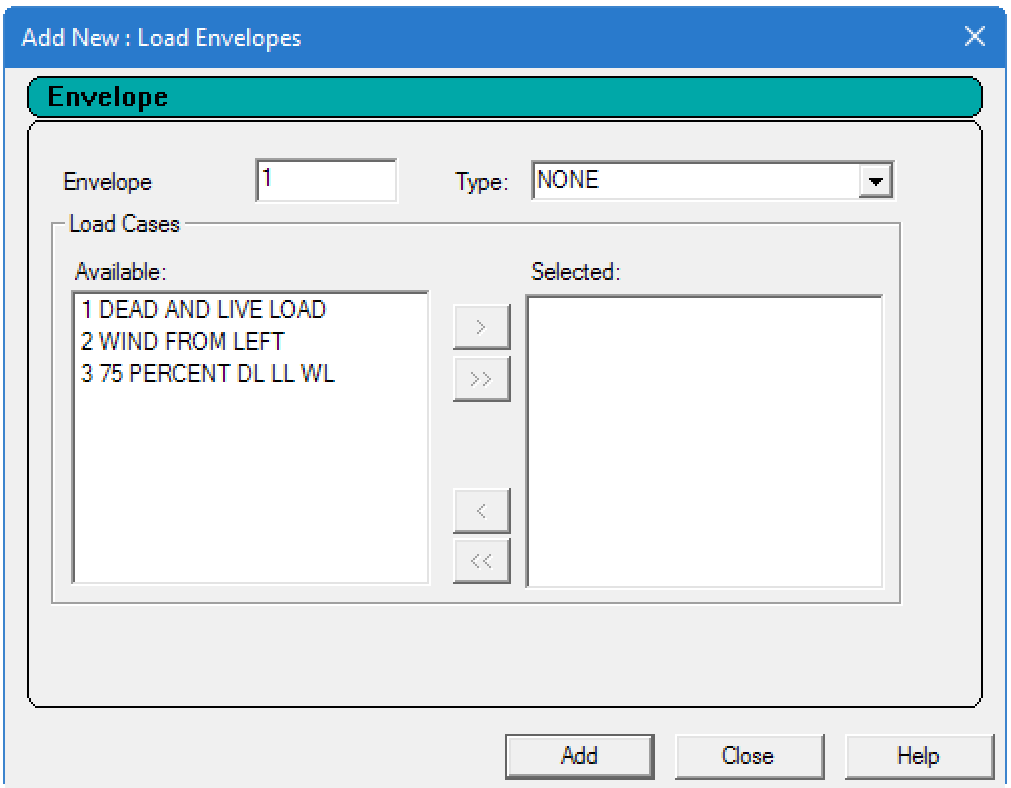
Add New Load Envelopes dialog

Used to create load envelopes, which are used to group results set under a single id and can later be used in post processing. If one or more tasks have to be performed for a set of load cases, such as, serviceability checks under steel design for one set of load cases, strength checks under steel design for another set of cases, etc., this feature is convenient.

Note: Load Envelopes are an alternative to using Load Lists. These commands are treated the same by the STAAD engine.

Once load envelopes have been defined, you can select a Load Envelope for reporting results or design in the **Result Setup dialog**or the appropriate design setup dialog.

Opens when **Add** is clicked on the **Load & Definition dialog** with the **Load Envelopes** entry selected.



Envelope Specify a unique ID number for the load envelope.

Type The envelope can be tagged with optional key words to specify qualitative nature of the load or load combination cases included in the envelop definition. Based on the nature of the load cases in the envelope, you can define appropriate design parameters for each envelope.

For example, for design under wind load condition, most of the design codes allow increase of allowable stresses. Design routine can increase the allowable stress used in interaction equation, when it does the design for the envelope. Another application of this feature can be to specify separate load groups for serviceability check, working stress and limit state checks.

Available list All primary load and load combinations cases are listed here.

List Operators

Click this button…	to…
>	Add the selected load case to the Selected list

Click this button…	to…
>>	Add all load cases to the Selected list.
<<	Remove all entries in the Selected list, load combination is placed back in the Available list.
<	Remove the selected entry from the Selected list.

Selected list Loads to be considered in this Load Envelope.

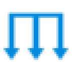
Related Links

- *M. To create a load envelope* (on page 901)

Create Primary Load Case dialog

Used to create new primary load cases.

Opens when the **Primary Load Case** tool is selected from the **Loading Specifications** group on the **Loading** ribbon tab.



Number

Specify a the load case Number the program recommends, or specify one of our own.

Note: If the load case number has been previously used in this input file, a warning message will be displayed.

Loading Type

Used to associate the load case we are creating with any of the ACI, AISC, or IBC definitions of Dead, Live, Ice, etc. This type of association needs to be done in order to make use the program's facility for automatically generating load combinations in accordance with those codes.

The available load types are the following:

Dead	Soil	Ice
Live	Rain Water/ Ice	Wind on Ice
Roof Live	Ponding	Crane Hook
Wind	Dust	Mass (Notes (on page 2461))
Seismic-H (horizontal, Notes (on page 2461))	Traffic	Gravity

Seismic-V (vertical, Notes (on page 2461))	Temperature	Push
Snow	Accidental	None
Fluids	Flood	

Reducible per UBC/IBC (for Loading Type = Live only) Select this option if the load will be a reducible load per building code.

The UBC 1997, IBC 2000, and IBC 2003 codes permit reduction of floor live loads under certain situations. Please refer to TR.32.4 Area, One-way, and Floor Load Specifications (on page 2475) for details of this feature.

Title Type an optional title which is used to describe the load for identification in modeling and in reports.

Related Links

- *M. To create a new primary load case* (on page 830)
- *M. To create a new primary load case* (on page 830)

Define Load Combinations dialog

Used to specify a load case that combines results of analysis performed for different primary load cases.

Opens when the **Combination Load Case** tool is selected from the **Loading Specifications** group on the **Loading** ribbon tab.

Tip: To automatically generate multiple combinations, use the **Edit Load Rules for Auto Load Combination Generator** dialog.

For more information about Load Combinations, refer to TR.35 Load Combination Specification (on page 2616).

Load No. Specify an integer value to define a new load case combination. This number is automatically incremented.

Name Specify a *Name* for the load case combination.

Type There are three types of combinations possible:

- **Normal -** A pure algebraic load combination in which the results of all constituent cases will be algebraically added. To accomplish this, select each case individually from the left pane, specify the value by wish we see its results factored, and move it over to the right pane by clicking on the > arrow. Do not check any of the boxes Combine Algebraically, SRSS or ABS.
- **SRSS -** The square root of the sum of the squares method of combination.

This enables one to achieve a combination such as L1 * f1 + r1 * [√(f2*L2^2 + f3*L3^2)]. The sequence of operations will be,

1. Select the **SRSS** check box. The *Combine Algebraically* check box will become active.
2. Select *L1*. In the box above Combine Algebraically, specify a *Factor*, (term f1 in the equation above), place a check against the *Combine Algebraically* check box, click on the > arrow.
3. Select *L2*. In the box above Combine Algebraically, specify a *Factor*, (term f2 in the equation above), un-check the *Combine Algebraically* check box, click on the > arrow.
4. Select *L3*. In the box above Combine Algebraically, specify a *Factor*, (term f3 in the equation above), keep the *Combine Algebraically* box un-checked, click on the > arrow.
5. In the box called *Factor* which appears to the right of the expression SRSS, specify the value for *r1*.

- **ABS -** Algebraic combination of absolute values.

  This enables one to achieve a combination such as f1*ABS(L1) + f2*ABS(L2). The sequence of operations will be,

  1. Switch on the *ABS* check box.
  2. Select *L1*. In the box above Combine Algebraically, specify a *Factor*, (term f1 in the equation above), click on the > arrow.
  3. Select *L2*. In the box above Combine Algebraically, specify a *Factor*, (term f2 in the equation above), click on the > arrow.

Factor The load factor coefficient, b, used for the square root value of the sum of the squares.

Default Indicates the default value of the load factor, a (or c, in the case of a SRSS combination), for any load case moved from the Available Load Cases to the Load Combination Definition.

Load factors may be changed from the current default by selecting the entry in the Load Combination Definition.

SRSS Component (Active only when the **SRSS** type is selected) The SRSS load combination type allows a mixture of algebraic combination with an SRSS combination. Select this option to for the load to include it as a SRSS in lieu algebraic addition.

Generate Combination (Active only when either the **SRSS** or **ABS** type is selected) Se this option to generate 1 - 64 load combinations for each combination of degrees of freedom at member ends to create load envelopes for positive and negative analysis results for SRSS and Absolute load combinations.

Available Load Cases list Lists all of the previously created Primary Load Cases or Load Combinations in the current input file.

Tip: Double clicking any load case or combination will add it to the Load Combination Definition list.

Load Combination Definition list Entries here represent the Primary Load Case, load factors, and SRSS component (if any) which will make up the Load Combination.

List Operators

Click this button…	to…
>	Add the selected load case or combination to the Load Combination Definition list
>>	Add all load cases and combinations to the Load Combination Definition list.
<<	Remove all entries in the Load Combination Definition list.
<	Remove the selected entry from the Load Combination Definition list.

Example

Thus, for example, if you want to create a combination case called 101, which combines the absolute values of the results of cases 7 and 15, factored by 1.5 and 1.64 respectively, the command would be

```
LOAD COMBINATION ABS 101
7 1.5 15 1.64
```

From the Create New Load Combination dialog:

1. Select the **ABS** option.
2. Select load case 7 in the Available Load Case list.
3. Specify a **Default** factor of 1.5 and move it to the right pane by clicking on the **>** button.
4. Repeat this process for case 15 with a factor of 1.64.
5. Click **Add**.

The table shown below illustrates the Load Combination logic for Algebraic and SRSS types assuming that the Primary Load Cases are L1, L2, and L3.

Primary Load Case	Factor	Combine Algebraically?	SRSS?	SRSS Factor	Resulting Combination Formula
L1	0.75	Yes	No	n/a	$0.75(L1) + 0.75(L2)+1.33(L3)$
L2	0.73	Yes	No	n/a	
L3	1.33	Yes	No	n/a	
L1	1.0	No	Yes	1.2	$1.2\sqrt{1(L1)^2 + 0.4(L2)^2 + 0.4(L3)^2}$
L2	0.4	No	Yes	1.2	

Primary Load Case	Factor	Combine Algebraically?	SRSS?	SRSS Factor	Resulting Combination Formula
L3	0.4	No	Yes	1.2	

Related Links

- *M. To define a new load combination* (on page 886)

Add New Reference Load Definitions dialog

Used to add a reference load case to the input file.

Opens when the **Reference Load Case** tool is selected in the **Loading Specifications** group on the **Loading** ribbon tab.

Note: See GS. Load Types in STAAD.Pro (on page 51) for additional information on when different load types are used in STAAD.Pro.

The parameters in this dialog are analogous to the Create Primary Load Case dialog (on page 2835).

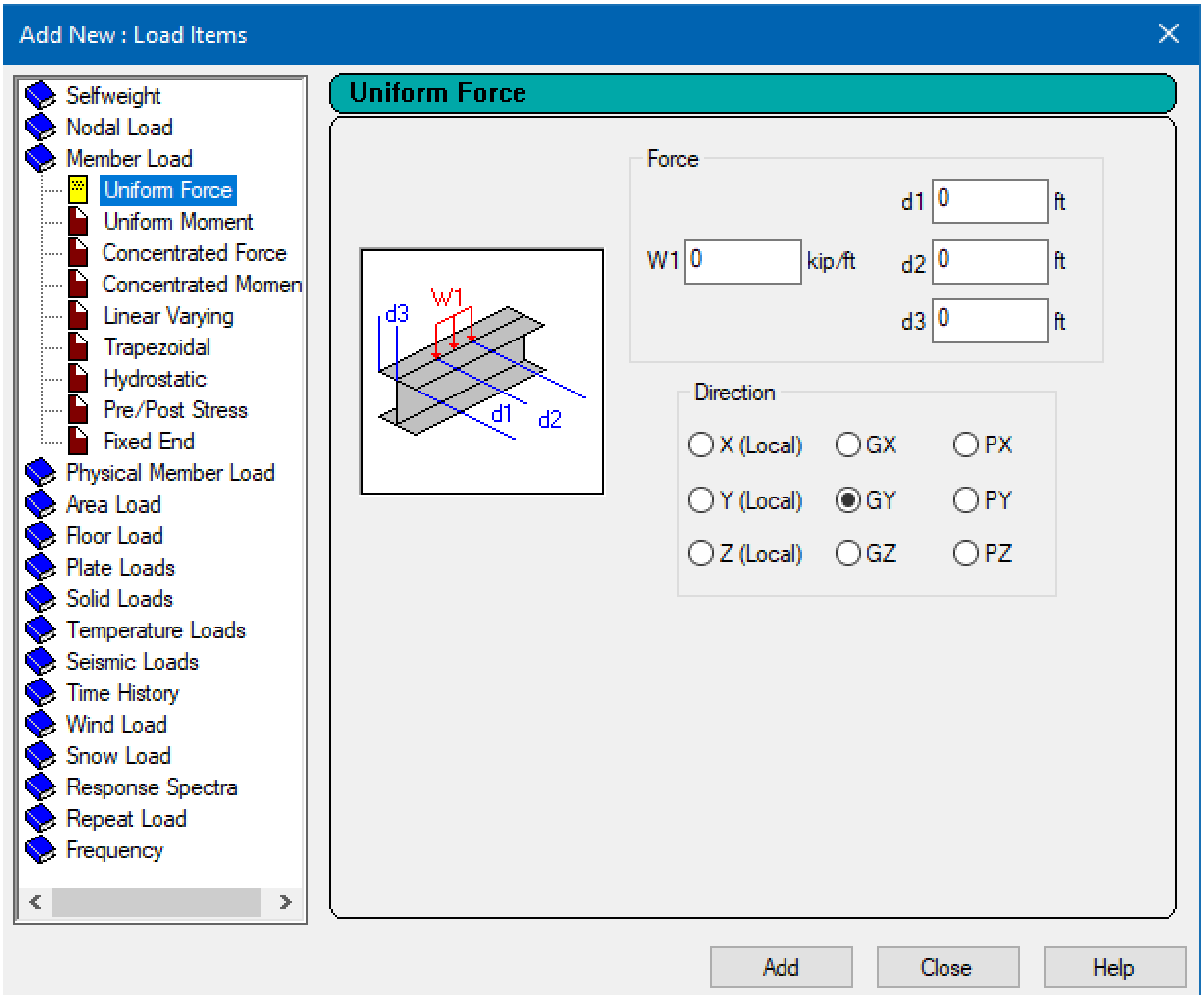
Note: In STAAD.Pro V8i (SELECTseries 3) and higher, the Loading Type Mass can be used to generate a single mass model for re-use in dynamic loads. Refer to TR.31.6 Defining Reference Load Types (on page 2453) for additional details.

Add New Load Items dialog

Used to define and assign loads to the structure.

Opens when the **Load Items** tool is selected in the **Loading Specifications** group on the **Loading** ribbon tab.

Load Item types are grouped into tabs on the right side of the dialog, with parameters for each type in the main dialog.



Note: If the current active load case is not a Primary Load, then many of the tabs will display an error message and the controls will be inactive.

Load Specifications list	Lists the categories into which load specifications are grouped. Selecting a load specification type causes the appropriate dialog controls to be displayed to the right of the list.
Add	Adds the load case on the current tab to the input file. The new load appears in the **Load & Definition** dialog under the currently selected load case.
Close	Closes the dialog. Any load data entered without clicking the Add button will be lost.
Help	Opens the STAAD.Pro help window.

Related Links

- *M. To edit a previously assigned load* (on page 901)

Selfweight tab

Used to apply the selfweight of the structure as a load. Selfweight of all active components of the structure are calculated and applied as a uniformly distributed load.

Note: The material (or the density parameter) of the members must be defined before this command is used.

Direction Specify the direction in which the selfweight load is to be applied by clicking on the X, Y, or Z buttons.

Factor Specify the factor with which the calculated selfweights are to be multiplied. A negative value indicates that the load is applied along the negative direction of the chosen axis.

Related Links

- *M. To add selfweight load* (on page 831)

Nodal Load tab

Used to apply nodal loads.

The following options are available under this tab:

Node

Fx/Fx', Fy/Fy', Fz/Fz' Type values of the forces in corresponding global directions or inclined axis (') directions when the **Inclined Load?** option is checked. A negative sign (-) may be used.

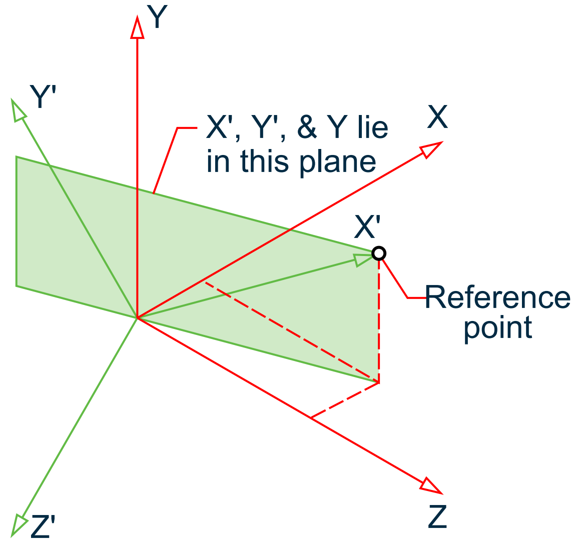
Mx/Mx', My/My', Mz/Mz' Type values of the moments about the corresponding global directions or inclined axis (') directions when the **Inclined Load?** option is checked. A negative sign (-) may be used.

Inclined Load? Check this option to specify the load along an inclined coordinate system. An inclined coordinate system is defined by an X' axis that pass from the loaded joint through a reference point. That reference point can be defined either a reference node, a set of absolute (global) coordinates, or a set of relative distances from the loaded joint.

Note: When this option is checked, the load vector labels are updated to reflect the inclined (') vector.

Inclination Select the option used to define the inclined coordinate system.

- **Reference Node -** select an existing node number from the model. The global coordinates of that node are displayed as read-only.
- **Absolute -** type the global coordinates of an arbitrary point to use as the reference.
- **Relative -** type relative distances from the loaded joint to the reference point. These distances are measured along the global axes.



The reference point –whether it is specified using absolute coordinates, relative distances, or a reference joint– defines the direction of the x' axes from the loaded node to the reference point. The direction of the Y' axes is then defined as the direction perpendicular to x' that lies in the plane of X' and Y. In the special case of when X' is in the direction of Y, Y' is taken to be the direction of Z. The direction of Z' is then defined as the direction perpendicular to the plane X' and Y' and follows the right-hand rule as for all other axes systems used in STAAD.Pro.

Support Displacement

Used to specify a support displacement. Multiple support displacements may be used for a single load case, but each support value and direction must be entered independently.

Displacement	Specify the value of the displacement. A negative sign (-) may be used.
Direction	Select the direction of the displacement as **Fx**, **Fy**, or **Fz** (translational), or **Mx**, **My**, or **Mz** (rotational).

Related Links
TR.31.2.16 IBC 2015 Seismic Load Definition

- *M. To add a nodal load* (on page 832)
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *G.15.1 Joint Loads* (on page 2128)
- *M. To add a nodal load* (on page 832)
- *G.15.7 Support Displacement Loads* (on page 2133)
- *M. To add a support displacement* (on page 833)

Member Load tab

Used to apply loads on the span of frame members or physical members (select the appropriate parent tab).

These following options are available:

Uniform Force / Moment

Used to specify a Uniformly Distributed Force or Uniformly Distributed Moment along a member.

W1 Value of the load in currently selected units.

d1, d2 Distance of starting and ending points of the load from the starting node (Node A) of the member. If these values are zero, then the load is applied over the entire length of the member.

d3 Perpendicular distance from the member's shear center to the plane of loading.

Direction Click on the appropriate radio button to specify the direction of the load. **X**, **Y**, **Z** indicate the direction in local coordinates; **GX**, **GY**, **GZ** indicate the loads in global coordinates; **PX**, **PY**, **PZ** indicate the loads along the projected length of the member in the corresponding global direction. However, d1, d2 and d3 are still measured along the length of the member and not along the projected length.

The projected load direction, PY, uses the component length in the XZ plane for the load effect.

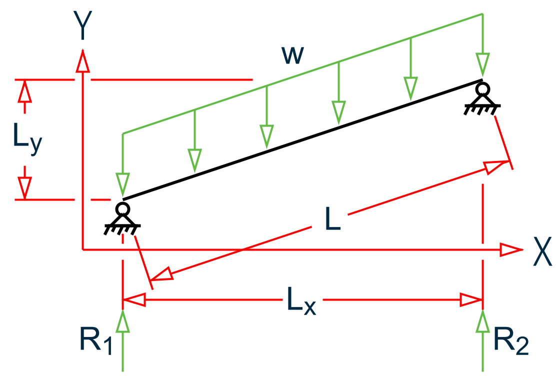


Figure 343: Example inclined beam parallel to the XY plane

Concentrated Force / Moment

Used to specify a concentrated force or moment load on a member.

P Value of the load in currently selected units.

d1 Distance of the load from the starting node (Node A) of the member. If this value is not specified, then the load is applied at the midpoint of the member.

d2 Perpendicular distance from the member's shear center to the plane of loading.

Direction Click on the appropriate radio button to specify the direction of the load. **X**, **Y**, **Z** indicate the direction in local coordinates; **GX**, **GY**, **GZ** indicate the loads in global coordinates; **PX**, **PY**, **PZ** indicate the loads along the projected length of the member in the corresponding global direction. However, d1 and d2 are still measured along the length of the member and not along the projected length.

Linear Varying

Used to specify a linearly varying load on a member. The load is applied over the entire length of the member.

W1, W2	For a linearly increasing or decreasing load, enter the values of force **W1** at the start of the beam and **W2** at the end of the beam in currently selected units.
W3	For a triangular load distribution, enter the value **W3** of the force in the middle of the beam.
Direction	Select the direction of the force in local coordinates from the radio buttons **X**, **Y** or **Z**.

Trapezoidal

Used to specify a trapezoidal load on a member.

W1, W2	Starting and ending load values in currently selected units.
d1, d2	Distance of starting and ending points of the load from the starting node (Node A) of the member. If these values are zero, the load is applied over the entire length of the member.
Direction	Click on the appropriate radio button to specify the direction of the load. **X**, **Y**, **Z** indicate the direction in local coordinates; **GX**, **GY**, **GZ** indicate the loads in global coordinates; **PX**, **PY**, **PZ** indicate the loads along the projected length of the member in the corresponding global direction. However, d1 and d2 are still measured along the length of the member and not along the projected length.

Hydrostatic

Used to specify loads due to hydrostatic pressure on one or more adjacent beams. The Hydrostatic load is converted to Trapezoidal loads on the beams. The load is applied over the entire length of the members.

W1, W2	Type the value of the load at the minimum and maximum global axis in currently selected units. For example, to model a retaining wall with soil pressure, **W1** is the force at the bottom of the wall and **W2** is the force at the top of the wall.
Direction	Select the direction in which the force is applied by clicking on the appropriate radio button. **X**, **Y**, **Z** indicate the direction in local coordinates; **GX**, **GY**, **GZ** indicate the loads in global coordinates; **PX**, **PY**, **PZ** indicate the loads along the projected length of the member in the corresponding global direction.
Interpolate along Global Axis	Specify the global axis along which the load would vary from **W1** to **W2**. For example, the load would vary along the Y axis on a vertical retaining wall.
Select Member	Opens a **Selected Item(s)** dialog, which is used to generate a list of selected members. The Create Loads dialog is dismissed until all loads are added using the Beams Cursor tool in the View Window. Once the **Done** button is clicked, the Create Loads dialog re-opens with the member table populated. Unlike other load definition options, we must select members for this option to become active.

Pre/Post Stress

Used to apply Poststress and Prestress loads on members.

Type	Select either the **Prestress** or **Poststress** radio button.
Load	Type the **Force** as a positive value in current units.
Eccentricity Distances	If applicable, specify the Eccentricity of the cable with respect to the center of gravity of the cross section at the **Start**, **Middle** and **End** of the member.

Note: Please refer to TR.32.5 Prestress Load Specification (on page 2489) for more detailed information on these options.

Fixed End

If a load acts at an intermediate point along the span of a frame member, there are two ways of assigning the load. a) It could be specified using the member load option. In this case, the magnitude and position along span are specified. b) It could be converted to fixed end actions and defined as a FIXED END load as shown below.

Start / End Node	Type values for FX, FY, FZ, MX, MY and MZ (axial, shear-y, shear-z, torsion, moment-y, and moment-z) in the local coordinates for the Start node and /or the End node.

Related Links

- *G.15.2 Member Load* (on page 2129)
- *M. To add a concentrated force or moment on members* (on page 834)
- *M. To add a uniform load to members* (on page 834)
- *M. To add a linear varying load to members* (on page 835)
- *M. To add a hydrostatic load to objects* (on page 842)
- *G.15.5 Prestress and Poststress Member Load* (on page 2131)
- *EX. US-6 Prestress and Poststress Loading* (on page 6351)
- *EX. UK-6 Prestress and Poststress Loading* (on page 6635)
- *M. To add a prestress or post-tension load to members* (on page 836)
- *G.15.4 Fixed End Member Load* (on page 2131)
- *M. To add fixed end member loads* (on page 837)

Area Load tab

Used to apply area (panel) load which will be distributed on surrounding beams based on a one way distribution.

Pressure	This load is a one-way distributed pressure load on members that circumscribe a panel. Enter the value of the area load (force per unit area) in the current units. This load always acts along the positive local y-axis on the two longest members in each panel.
Direction	Select the direction in which the force is applied. **Local Z** indicates the direction in local coordinates, perpendicular to the panel; **GX**, **GY**, **GZ** indicate the loads in global coordinates.

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *M. To add an area load* (on page 839)

Floor Load tab

Used to apply a panel load which will be distributed on surrounding beams based on a two way distribution.

This load is a two-way distributed pressure load on members that circumscribe a panel.

Load	Floor load value (force per unit area) in the current units. This load will act parallel to the global vertical axis.

XRANGE / ZRANGE / YRANGE options
Select one of these options to specify a floor load by range of global coordinates

Group option
Select this option to apply a floor load on a previously added member group or composite deck definition.

Pressure
Specify the pressure of the floor load in the current force per square length units.

Direction
The floor load may be considered as acting perpendicular to the plane of the panel on which it is defined. This is the normal static load condition. However, for the purposes of generating the mass matrix for dynamic analysis, it may also be considered as weights lumped at the nodes of its panels and acting (vibrating) parallel to the plane of the panel. Hence, the user has the choice of specifying it in one of the 3 global directions.

Note: Refer TR.32.4.3 Floor Load Specification (on page 2483) for more information.

Define X Range / Y Range / Define Z Range
Specify the location of the floor using the *Define X Range* option. The load will be calculated for all members lying between this range.

Corner points of the area on which the floor load acts, in global X and Z directions. If omitted, the floor load is assumed to be acting for all members located within the Y range specified.

One Way Distribution
Select this option to use one way distribution to get a one way type distribution of the pressure. In such cases the program finds out the shorter side of the panel. It then divides the load in between the long direction beams . No load is generated by this option if the panel is square in shape.

To specify a floor load with the Z-axis vertical, select the *Floor With Z Range* menu option. The data items are the same as the Floor load with Y axis vertical except that the Y and Z axes are interchanged.

Towards
(One Way Distribution) Used to define the direction in which the load is spanning by selecting a member in the loaded zone onto which the load is to be directed. The default is for the load to resolve toward the longer span of supporting beams framing the Floor load. However, you can select a beam number to direct the load to that beam.

Panel Information Output group

Print to output file
Includes the floor panel information in the output file (`.ANL`).

Print Options (output file):

- **Print -** The total number of panels identified, total area of all panels and total load generated will be printed in the output file.
- **Panel with member list -** This option is used to print panel member numbers on which loads are generated in addition to the **Print** option output.
- **Panel with member load -** This option is used to print panel loading information in addition to the **Print with member list** option output..

Print to external text file
This option will output the panel information in an external text file named `(filename) _FLD.TXT`.

Print Options (external text file):

- **Panel with member list -** This option is used to print panel member numbers on which loads are generated. Additionally, the total number of panels identified, total area of all panels and total load generated will be printed in the text file.
- **Panel with member load -** This option is used to print panel loading information in addition to the **Print with member list** option output..

Related Links

- *G.15.3 Area, One-way, and Floor Loads* (on page 2130)
- *M. To add a floor load or one-way load* (on page 840)

Plate Loads tab

Used to apply loads to elements.

Pressure on Full Plate

Used to define a pressure load that acts on the full surface of an element. (To define a pressure load that acts on a small part of an element, see the Partial Plate Pressure tab that is described in the coming pages).

Load W1 is the pressure magnitude, in the current units.

Direction The load may be applied along:

- the local X, Y, or Z axis,

> **Tip:** Loads in the **Local X** or **Local Y** directions can be used to represent in-plane friction loads.

- one of the global X, Y or Z axes (**GX**, **GY**, or **GZ**)
- projected area of the plate (**PX**, **PY**, or **PZ**)

Concentrated Load

Use this option to define a concentrated load that acts at a specific point within the boundary of an element. If a load acts at a node point of an element, it is advisable to apply it using the Nodal Load option described in earlier pages.

Load **Force** is the magnitude of the load.

Direction The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (GX, GY, GZ).

Partial Plate Pressure Load

Used to specify a uniform pressure load on the entire element or on a user specified portion of the element.

Load The element pressure (force per unit area) or concentrated load (force unit). For concentrated load, the values of **X2** and **Y2** must be omitted, while **X1** and **Y1** must be specified.

X1, Y1, X2, Y2 For element pressure (force per unit area), these values represent the coordinates of the rectangular boundary on which the pressure is applied. If **X1**, **Y1**, **X2**, and **Y2** are all zero,the pressure is applied over the entire element. If **X1** and **Y1** are specified but **X2** and **Y2** are omitted, then **W1** is treated as a concentrated load.

Direction	**GX**, **GY**, and **GZ** represent the global X, Y, and Z directions along which the pressure may be applied. **Local Z** indicates that the pressure is applied normal to the element in the local Z direction.

Note: For more information, please refer to TR.32.3 Element Load Specifications (on page 2468) .

Trapezoidal

Used to specify a trapezoidally varying pressure load on a plate. The load is applied over the entire element in the local Z direction, varying along the positive local X or Y direction.

Direction of Pressure	**GX**, **GY**, and **GZ** represent the global X, Y, and Z directions along which the pressure may be applied. **Local Z** indicates that the pressure is applied normal to the element in the local Z direction.
Variation along element	Define the direction in which the pressure varies as either the local **X** or **Y** direction or choose the joint option which is discussed next. The Joint option is used to apply different values of pressure at different nodes of the plate element. When checked, the dialog updates to allow a different pressure value for each of the four plate corner nodes.

Hydrostatic

Used to model loads due to hydrostatic pressure on one or more adjacent elements. The Hydrostatic load is converted to Trapezoidal loads on the elements. The load is applied over the entire area of the element.

Force	Enter value of the load at the minimum and maximum global axis in current units. For example, to model a retaining wall with soil pressure, **W1** is the force at the bottom of the wall and **W2** is the force at the top of the wall.
Interpolate along Global Axis	Specify the global axis (**X**, **Y**, or **Z**) along which the load should vary from **W1** to **W2**. For example, the load would vary along the Y axis on a vertical retaining wall.
Direction of pressure	Specify the direction of design pressure as **Local Z** axis or global axes (**GX, GY** or **GZ**) and click on **Add.** This will assign the linearly varying hydrostatic load on all the selected elements.
Select Plate(s)	Opens a **Selected Item(s)** dialog, which is used to generate a list of selected plates. The **Create Load Items** dialog is dismissed until all loads are added using the Plates Cursor tool in the View Window. Once **Done** is clicked, the **Create Load Items** dialog re-opens with the Plate table populated. Unlike other load definition options, we must select plates for this option to become active.

Element Joint Load

To specify a varying pressure at each joint on a plate, select the **Element Joint Load** option. This load item can be applied on joints directly whether they are plate corner nodes or not.

Joint Load Data	Choose Three Noded Facet / Four Noded Facet, depending on whether the plate element is 3 noded or 4 noded.

Pressure Table	Specify the corner node number and corresponding pressure load, in the current units (displayed).
Direction	The load may be applied along the local Z axis, or along one of the global X, Y or Z axes (GX, GY, GZ).

Related Links

- *M. To add pressure load on a plate* (on page 837)
- *G.15.8 Loading on Elements* (on page 2133)
- *M. To add pressure load on a plate* (on page 837)
- *M. To add a concentrated load on a plate* (on page 838)

Solid Loads tab

Used to apply uniform pressure and uniformly varying pressure on the faces of the solid element.

Face Number	Select the face on which we want to apply the load.
Node 1.... Node 4	Specify the pressure at each node on the selected face of the solid element. For example, if the pressure is uniform, the same value ought to be entered in all four columns.
Direction	Specify the direction of design pressure as *Local Z* axis or global axes (*GX, GY* or *GZ*) and click on *Add.* By doing this, the solid pressure load is added to that particular load case.

Temperature Loads tab

Used to apply Strain and Temperature loads on members.

Temperature

Used to specify a temperature load

Temperature Change for Axial Elongation	The change in temperature that will cause axial elongation in members or uniform volume expansion in elements.
Temperature Differential from Top to Bottom	Temperature differential of the member or element (T top surface – T bottom surface). **Note:** The top surface is in the positive local Y direction for members, and the positive local Z direction for elements. If the **Temperature Differential from Top to Bottom** is omitted, bending is not considered.
Temperature Differential from side to side (Local Z)	Accounted for when the temperature difference is not the same across the sides of the member from left to right.

Strain

Used to specify a strain load.

Initial Axial Elongation or Shrinkage	Enter the initial elongation or shrinkage in the member in current units. A positive value indicates elongation, a negative value indicates shrinkage.

Seismic Loads tab

Used to apply previously defined UBC loads on the structure.

Note: If a seismic load definition has not yet been added to the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Add New Seismic Definitions dialog (on page 2895) to learn more about defining a seismic load.

Direction group	Specify the global direction in which the seismic load is to be generated by selecting the **X Direction**, **Y Direction**, or **Z Direction** radio button. **Factor** Specify a multiplying factor, if applicable. The default factor is 1.0.
Multiplying factor for Accidental Torsion Moment	Set this option to include the accidental torsion load per the UBC, IBC, 1893, etc. code. Type the **Factor** in the associated field (may be a negative value).
Multiplying factor for Natural Torsion Moment	Set this option to include the torsion arising due to static eccentricity which is the difference between center of mass and center of rigidity of a rigid floor diaphragm. Type the **Factor** in the associated field (must be greater than or equal to zero).

Time History tab

Used to apply previously defined time history loads on the structure.

Note: If a time history load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Add New Time History Definitions dialog (on page 2907) or Define (Time History) Parameters dialog (on page 2909) to learn more about defining a time history load.

Loading Type	Select **Time Load** to apply the Time History load to joints in the structure or **Ground Motion** to apply the load at the structure's base.
Arrival Time	Select a previously defined Arrival Time to define the time at which the load begins to act.
Direction	Select the global direction in which to apply the Time History load.
Defined Types	Select a previously defined Type number.
Force Amplitude Factor	Specify a factor to multiply the values of force or acceleration which were input while defining the Time History loading.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *M. To add a time history load* (on page 880)

Wind Load tab

Used to apply previously created wind load types on the structure through the means of a load case.

Note: If a wind load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Create Wind Type Definition dialog (on page 2880) to learn more about defining a wind load.

Wind Load tab

Used to add static wind forces per ASCE 7 and other codes.

Select Type
Choose a previously defined wind load type from the drop down list.

Exposed Surface and Direction
Specify the global direction in which the wind load is to be generated by clicking the **X**, **Z**, **-X**, or **-Z** option. When wind is generated in X direction ,the wind load is applied on the near side and when -X is chosen the load is applied on the far side as explained in the following figures. Generation in Z or -Z is similar.

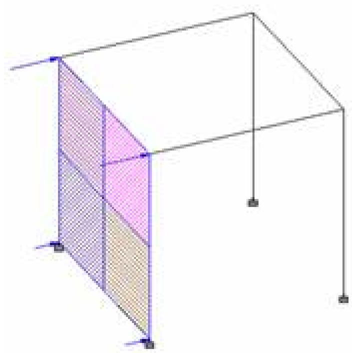
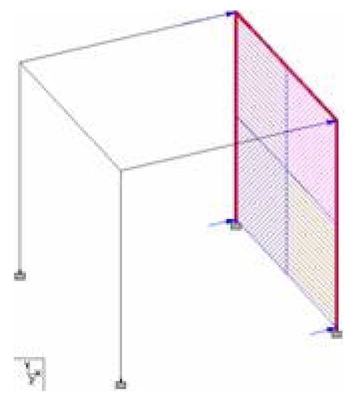
	
Wind in X	Wind in -X

Figure 344: Wind in positive and negative directions

Table 325: Wind direction explanation

If the direction is set to…	and f is…	Then the load is applied to surfaces facing…	and the load will be directed toward…	This can be though of as…
X or Z	Positive	Negative X or Z	Positive X or Z	Windward load for wind in the positive X or Z direction
	Negative	Positive X or Z	Negative X or Z	Windward load for wind in the negative X or Z direction

If the direction is set to…	and f is…	Then the load is applied to surfaces facing…	and the load will be directed toward…	This can be though of as…
-X or -Z	Positive	Positive X or Z	Positive X or Z	Leeward load for wind in the positive X or Z direction
	Negative	Negative X or Z	Negative X or Z	Leeward load for wind in the negative X or Z direction

Factor Specify the factor by which the calculated wind loads will be multiplied.

Open Structure Select this option to generate loads on open structures like highway signs or transmission towers. By default, the load generation is based on the assumption that the region between members is covered by panels (i.e., cladding). Setting this option will apply load to only the members (i.e., no cladding).

Tip: The term used here is not the same as the ASCE 7 definition of "open structure." Please use the ASCE 7 Wind Load dialog box (on page 2882) to generate describe the enclosure classification per this code.

SNiP Parameters The following parameters are used for wind loads generated per the Russian code (SNiP).

Apply Wind Load at the Corner Select this option for wind loads to be applied at 45°.

Note: This parameter is only applicable for rectangular buildings.

Select Configuration The building configuration as defined in SNiP 2.01.07–85 "Loads and Actions":

0. Prismatic building structure - Rectangular building- Outstanding architectural details on the left façade.
1. Prismatic building structure - Rectangular building- Both side façades are smooth
2. Prismatic building structure - Rectangular building- Outstanding architectural details on the right façade
3. Prismatic building structure - Rectangular building- Outstanding architectural details on both side façades
4. Prismatic building structure - Rectangular building- Triangular building
5. Prismatic building structure - Rectangular building- Rhombic building

6. Prismatic building structure - Rectangular building- Number of vertices of polygonal building, not more than 12
7. Prismatic building structure - Less than 3 — rectangular building
8. Prismatic building structure - 3 — triangular
9. Prismatic building structure - 4 — rhombic
10. Prismatic building structure - More than 4 — polygonal
11. Framed RC structure
12. Lattice steel structure

Reynolds Number (NU)	Specify the wind pressure correlation coefficient. If parameter is omitted or is exactly 1, a computed value is used instead. For rectangular buildings, the correlation coefficient is always calculated automatically, thus any specified value will be ignored.

Note: The first load case which has a Russian Wind Load command added to it will consider all other loads defined in it as the masses to be considered for calculating the dynamic effect which is required by this command.

Wind Load - Dynamic tab

Used to specify dynamic wind forces per SP 20.13330.2016.

Wind Definition	Choose an SP 20.13330.2016 wind load definition from the drop down list.
Width of Building along wind dir	Effective length of the structure parallel to wind direction.
Wid of Building across wind dir	Effective projection of the structure facing the wind direction.
Ground Level	Select if the ground level is not considered (i.e., global Y = 0 is the ground level) or if it is to be specified at different Y coordinate.
Factor	Correction factor along specified direction.
Internal	Set this option to indicate an internal, or suction, wind force on the surfaces (i.e., the load direction is reversed).
Allow all Mode Shapes	Check this option to compute dynamic wind load vector for all mode shapes extracted from Modal Analysis provided it is within code stipulated frequency limit. When this check box is clear, the calculation takes only the first mode shape.
Include Orthogonal Load	Check this option to include orthogonal directions in the load case.
Static Wind Case	Select a previously defined primary load cases in the model.
Wind Direction	Select the plan direction to apply the wind force.

Related Links

- *M. To add a SNiP wind load definition* (on page 847)
- *M. To apply a dynamic wind load per SP 20.13330.2016* (on page 856)
- *M. To add an ASCE 7 wind load definition* (on page 845)
- *M. To add an ASCE 7 wind load definition* (on page 845)

- *M. To add a SNiP wind load definition* (on page 847)

Snow Load tab

Used to generate snow loading on a structure in accordance with the provisions of the ASCE-7-02 code.

The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

Note: If a snow load has not yet been defined for the model, a warning message is displayed in the dialog and the controls are inactive. Refer to the Add New Snow Definition dialog (on page 2894) to learn more about defining a snow load.

Floor Group	Select the floor group on which the snow load is to be applied.
Condition	Specify whether the load is “Balanced” or “Unbalanced.” These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.
Define Snow Type	Select the snow load type number. This is the number specified against the *Type No.* field while creating the snow load definition in Part I described earlier in this section.
Roof Type	Specify the roof type from the available choices: • Default (if the roof type is not Mono, Hipped or Gable, it is referred to as Default) • Mono (see mono-sloped roof shown in figure 6-6 of the code) • Hipped (see figures 6-3 and 6-6 of the code) • Gable (see figures 6-3 and 6-6 of the code)
Roof Obstruction	Specify whether the roof is “obstructed” or “unobstructed.” This also is a term described in section Table 7-2 of ASCE 7-02.
Roof Slope Factor	For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

Related Links

- *G.16.4 Snow Load* (on page 2138)
- *M. To add an ASCE 7-02 snow load* (on page 870)

Response Spectra tab

Used to apply response spectrum loads on the structure.

Note: The dialog updates dynamically to reflect the **Code** selection.

Common Parameters for all **Code** options

Combination Method	The various methods available in STAAD.Pro for combining the contribution from the individual modes is listed under this heading. The details of these methods are explained in TR.32.10.1 Response Spectrum Analysis (on page 2498). **Note:** Not all methods are available for all code options.

- **SRSS** is the square root of summation of squares method.
- **ABS** is the absolute sum method. This method is very conservative and represents a worst case combination.
- **CQC** is the complete quadratic combination method. This method is recommended for closely spaced modes instead of SRSS.
- **ASCE** is the NRC Regulatory Guide Rev. 2 (2006) Gupta method for modal combinations and Rigid/Periodic parts of modes are used. The ASCE4-98 definitions are used where there is no conflict. ASCE4-98 Eq. 3.2-21 (modified Rosenblueth) is used for close mode interaction of the damped periodic portion of the modes. This method should only be used for a general response spectrum.
- **TEN** is the Ten Percent Method of combining closely spaced modes as per the NRC guideline 1.92 (1976).
- **CSM** is the closely spaced modes method. The peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined by Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method.
- **GRP** is the closely spaced modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined using the absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method. NRC Reg. Guide 1.92 (Rev. 1.2.1, 1976).

Save Select this option to generate a file (with `.acc`. extension) containing the joint accelerations in g's and radians/sec_2

Period vs. Acceleration table Provide the values of period (seconds) and corresponding acceleration (current length units/sec^2) or displacement (current length unit). Spectrum pairs should be provided in ascending value of period. As we provide the curve points, the program displays the curve at the bottom of the dialog box.

Spectrum Type Choose whether the response spectrum curve will be input as Period vs. Acceleration or Period vs. Displacement (Custom and IS 1394 only).

Interpolation Type From the spectrum data that are provided under the *Define Spectrum Pairs* tab of the dialog box shown above, STAAD fetches the spectral value for the actual modes of the structure using one of two interpolation methods – **Linear** and **Logarithmic**. Linear interpolation is the default method.

Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.

Damping Type

- **Damping -** This is to be used for specifying a single modal damping ratio which will be applied to all modes. The default value is 0.05.
- **CDAMP -** Select this option to use Composite Modal Damping. This evaluates the damping from that defined in the material or constant definitions. A Damping ratio is defined by the material definition or in the **Material Constants - Damping Ratios** dialog. If there is no damping information entered in the material or constant definitions, the behavior is the same as MDAMP.

- **MDAMP -** Selection this option to use Modal Damping, which is used for individual damping ratios for each mode. Individual mode damping ratios are defined using the Modal Damping dialog (on page 2910).

Scale	Linear scale factor by which the spectra data will be multiplied. Usually to factor g's to length/sec^2 units. This input is the appropriate value of acceleration due to gravity in the current unit system You may choose to provide the spectral acceleration or displacement data as a set of un-normalized values or as a set of normalized values. For normalized values, the normalization factor is specified through the means of the Scale factor. For example, if the curve is input in terms of "g" - the acceleration due to gravity - and the current length unit is feet, the *Scale* would be 32.2. For un-normalized values, the scale factor is provided as 1.0, which also happens to be the default. The spectra data will be multiplied by the scale factor during the analysis.
Missing Mass	Select this option to apply the Missing mass correction. The static effect of the masses not represented in the modes is included. If this option is selected on any spectrum case it will be used for all spectrum cases.
ZPA	Zero Period Acceleration: It is used only with the missing mass option. If no value is entered or a zero value is entered, the default considered by the program is 33 Hz. If an acceleration is entered corresponding to the Missing mass mode, then the ZPA value is ignored. If no acceleration value is entered for the missing mass mode, then spectral acceleration corresponding to the ZPA frequency is used.
Direction	Specify the global direction(s) in which the spectrum is to be applied. The response spectrum may be applied in one or more directions simultaneously. Directions not provided will default to zero.
Use Torsion	(IS 1893, IBC 2012, IBC 2018 and NRC only) **Dynamic Eccentricity (DEC)**: Factor to be multiplied with static eccentricity (i.e., eccentricity between center of mass and center of rigidity). If not specified (or zero), a value of 1.0 is assumed. **Accidental Eccentricity (ECC)**: Factor for accidental eccentricity. Positive values indicate clockwise torsion and negative values indicate counterclockwise torsion. If not specified (or zero), a value of 0.05 (5% of floor plan dimension perpendicular to force at a given level) is used.
Signed Response Spectrum Results Options	Two method are available for added mathematical signs to the spectrum response output: • **Dominant Mode No. -** Select this option and (optionally) specify a mode number to define as a dominant mode. The sign (sense) of this mode will be applied to other modes. • **Signed -** Select this option to create signed values for all results by comparing the sum of the squares values for positive and negative values to determine the governing sign.
Individual Modal Response Load Case Generation	Select this option to have the program automatically generate primary load cases from the mode shape scaled to the magnitude that the mode has in this spectrum analysis case before it is combined with other modes. A load case is generated for each of first number of modes specified, starting with the specified load case number.

> **Note:** The Individual Modal Response case generation is not available for SNiP II code response spectra.

Custom

Generate Spectrum Opens the Spectrum Parameters dialog (on page 2862), which is used to generate a response spectrum curve per the International Building Code.

> **Note:** This is the response spectrum type that is explained in Section TR.32.10.1.1 Response Spectrum Specification - Custom (on page 2499) l.

IS-1893 2002

Subsoil Class Select the soil type for the site (hard , medium, soft, or custom). Depending on the type of soil & time period, average response acceleration coefficient Sa/g is calculated.

> **Note:** This is the response spectrum type that is explained in Section TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002 (on page 2534).

IS-1893 2016

Subsoil Class Select the soil type for the site (hard , medium, soft, or custom). Depending on the type of soil & time period, average response acceleration coefficient Sa/g is calculated.

> **Note:** This is the response spectrum type that is explained in Section TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016 (on page 2544).

IS-1893(Part 4):2015

RSMIN Select a factor, given in percent, to multiply by the total seismic weight of the structure to obtain a minimum base shear value. This percentage is taken from Table 2 of IS:1839(Part 4)-2015. The default value is taken for a structure category 1 and zone 2.

If the minimum base shear value exceed the calculated response spectrum base shear, then all analysis results of the response spectrum load case are scaled by the ratio of the minimum base shear to the calculated response spectrum base shear. This multiplying factor will be noted in the output.

EURO (EC8) -1994

Load Type Select either a Elastic or Design response spectrum for the loading type.

Design Ground Acceleration Specify a design ground acceleration expressed in terms of acceleration due to gravity(g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.

Behaviour Factor Specify the value used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Subsoil Class

Used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds

- Type **A**: for Rock or stiff deposits of sand
- Type **B**: for deep deposits of medium dense sand,gravel or medium stiff clays.
- Type **C**: Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil.

Please refer section 3.2 of Eurocode8 for detailed guidelines regarding the choice of soil type.

Note: This is the response spectrum type that is explained in TR.32.10.1.4 Response Spectrum Specification per Eurocode 8 1994 (on page 2520) .

EURO (EC8) - 2004

Load Type

Select either a Elastic or Design response spectrum for the loading type.

Two types of response spectra curve can be generated based on either RS1 (for response spectra type 1 curve) or RS2 (for response spectra type 2 curve) f.

Design Ground Acceleration

Specify a design ground acceleration expressed in terms of acceleration due to gravity(g). For most of the application of Eurocode 8, the hazard is described in terms of a single parameter (i.e., the value of effective peak ground acceleration in rock or firm soil). This acceleration is termed as the design ground acceleration.

Behaviour Factor

Specify the value used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Subsoil Class

Used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds

- Type **A** : rock or other rock-like geographical formation.
- Type **B**: very dense sand, gravel or very stiff clay.
- Type **C**: Deep deposits of dense or medium dense sand, gravel or stiff clay.
- Type **D**: Deposits of loose-to-medium cohesionless soil or of predominantly soft to firm cohesive soil.
- Type **E**: Surface alluvium layer

Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

Note: This is the response spectrum type that is explained in TR.32.10.1.5 Response Spectrum Specification per Eurocode 8 2004 (on page 2525) .

IBC 2006

Zip	The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.
Latitude / Longitude	The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for **Zip**.
S1 / SS	Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.
Long Period (TL)	Long-Period transition period in seconds.
Fa / Fv	Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected **Site Class (SCL)** is **F**.
Site Class (SCL)	Select A through F for the Site Class as defined in the IBC code.

Note: This is the response spectrum type that is explained in TR.32.10.1.10 Response Spectrum Specification per IBC 2006 (on page 2561) .

IBC 2012

Zip	The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.
Latitude / Longitude	The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for **Zip**.
S1 / SS	Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.
Long Period (TL)	Long-Period transition period in seconds.
Fa / Fv	Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected **Site Class (SCL)** is **F**.
Site Class (SCL)	Select A through F for the Site Class as defined in the IBC code.

Note: This is the response spectrum type that is explained in TR.32.10.1.10 Response Spectrum Specification per IBC 2006 (on page 2561).

IBC 2015

Zip	The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.
Latitude / Longitude	The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for **Zip**.
S1 / SS	Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.
Long Period (TL)	Long-Period transition period in seconds.

Fa / Fv Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected **Site Class (SCL)** is **F**.

Site Class (SCL) Select A through F for the Site Class as defined in the IBC code.

Note: TR.32.10.1.12 Response Spectrum Specification per IBC 2015 (on page 2573).

IBC 2018

Zip The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors.

Latitude / Longitude The geographic coordinates of the site used to determine the Ss and S1 factors. This option may be used if no value is entered for **Zip**.

S1 / SS Mapped MCE for 0.2s spectral response acceleration and spectral acceleration for a 1-second period, respectively. These values may be entered if not geographic coordinate or postal code is provided.

Long Period (TL) Long-Period transition period in seconds.

Fa / Fv Optional Short-Period site coefficient at 0.2s and Long-Period site coefficient at 1.0s, respectively. Values must be provided if the selected **Site Class (SCL)** is **F**.

Site Class (SCL) Select A through F for the Site Class as defined in the IBC code.

Note: TR.32.10.1.13 Response Spectrum Specification per IBC 2018 (on page 2578).

SNiP II-7-81

Note: Only the SRSS and ABS combination methods are valid for the SNiP code.

Zoning Factor Specify the zoning factor per SNiP II-7-81.

Subsoil Class Defines the subsoil conditions on which the response spectrum will be generated.

1. Non-weathered rock and rock-like geological formation or permafrost subsoil.
2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

Direction The SNiP code allows an alternate method of specifying directional factors. You may input individual parameters such as KWX, KX1 the product of which is used as the factor along that direction.

Note: This is the response spectrum type that is explained in TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81 (on page 2584) .

SP 14.13330.2011

Subsoil Class Defines the subsoil conditions on which the response spectrum will be generated.

1. Non-weathered rock and rock-like geological formation or permafrost subsoil.

2. Weathered rock or deep deposits of medium dense sand, gravel or medium stiff clays.
3. Loose cohesion less soil deposits or deposits with soft to medium stiff cohesive soil.

Direction The SNiP code allows an alternate method of specifying directional factors. You may input individual parameters such as KWX, KX1 the product of which is used as the factor along that direction.

Note: This is the response spectrum type that is explained in TR.32.10.1.14 Response Spectrum Specification per SNiP II-7-81 (on page 2584).

NRC 2005

The NRC 2005 response spectrum has no code-specific parameters.

Note: This is the response spectrum type that is explained in TR.32.10.1.2 Response Spectrum Specification per NRC 2005 (on page 2506) .

NRC 2010

The NRC 2010 response spectrum has no code-specific parameters.

Note: This is the response spectrum type that is explained in TR.32.10.1.3 Response Spectrum Specification per NRC 2010 (on page 2512).

Chinese GB 50012 - 2010

Note: The parameters **Save** and **Scale** do not apply to the GB 50012-2010 response spectra. Further, only the CQC and SRSS Combination Methods are available for this response spectra.

Fortification Intensity Fortification Intensity (ref. table 5.1.4-1). Selectable values are 6, 7, 7(0.15g), 8, 8(0.30g), or 9.

Seismic Frequency Frequency of seismic action (ref. table 5.1.4-1 and clause 3.10.3), as specified by:

Frequent
Fortified
Rare

Seismic Group Design Seismic Group (ref. table 5.1.4-2). Selectable values are 1st, 2nd, or 3rd

Site Class (SCL) Site class (ref. table 5.1.4-2). Selectable values are I0, I1, II, III, or IV

Max. Horizontal Influence Factor Max horizontal seismic influence factor (ref. table 5.1.4-1 and clause 3.10.3).

Period (Tg) Characteristic period of the structure (Tg) (ref. table 5.1.4-2).

Note: This is the response spectrum type that is explained in TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010 (on page 2531).

Related Links

- *M. To add a GB 50011-2010 response spectrum* (on page 869)

- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *M. To add a generic response spectrum* (on page 861)
- *V. NRC 2005 Response Spectrum* (on page 3614)
- *V. NRC 2010 Response Spectrum* (on page 3604)

Generated Spectrum dialog

Displays the generated IBC response spectra data in tabular and graphical format.

Note: No data may be modified in this dialog.

Opens when **Generate Spectrum** is clicked on the Spectrum Parameters dialog (on page 2862) .

Spectra Table	Displays period values with the associated Acceleration, Velocity, or Displacement values for the generated spectra. The table updates based on the Graph Type selection.
Graph Paper Type	Select to display the response spectra period on a Linear or Logarithmic scale.
Draw Points	Select this option to display point markers at each period, displacement/velocity/ acceleration coordinate on the curve.
Show Peak Value	Displays the peak displacement/velocity/acceleration in the generated spectrum curve.
Graph Type	Select to display the period versus displacement, velocity, or acceleration on the graph and in the Spectra Table.
Close	Closes the dialog and returns focus to the **Create New Load Item** dialog.

Related Links

- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *G.17.3.4 Response Spectrum* (on page 2163)
- *M. To add an IBC 2000 response spectrum* (on page 862)

Spectrum Parameters dialog

Used to generate a response spectrum curve per generates the spectrum based on the inputted data and as per section 11.4.5 of the ASCE-7-05 code (which is referenced by the International Building Code).

Opens when the **Generate Spectrum** button is clicked in the **Create New Load Item** dialog from the **Response Spectra** tab (with **Custom** selected for the code).

Select Zip	Type or select the Zip code (US Postal code) to look up the geographic location.
Find Lat/Long	Click to update the Latitude and Longitude values if the postal code value changes.
Latitude / Longitude	Type the geographic coordinates of the site here, if know. These fields will be populated if the Select Zip or Find Lat/Long options are used.
Calculate S1 / SS	Click this button to recalculated the seismic acceleration coefficients if the Latitude and/or Longitude values were manually entered.
S1	This is the mapped MCE spectral response acceleration at a period of 1 second, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the **Calculate S1/SS** button. Alternatively, you may specify this value directly.

SS	This is the mapped MCE spectral response acceleration at short period, determined in accordance with section 11.4.1 of ASCE-7-05. It is calculated by setting the Zip code or by clicking the **Calculate S1/SS** button. Alternatively, you may specify this value directly.
Site Class	This is the classification (A to F) assigned to a site based on the soil type as defined in chapter 20 of ASCE-7-05. Values of Fa and Fv from the IBC table are displayed for reference. If the classification is set to F, then you must specify the parameters Fa and Fv.
Fa	Short period site coefficient determined in accordance with table 11.4 -1 of ASCE-7-05. This is determined by the Site Class, but must be specified if the Site Class is set to F.
Fv	Long period (1 sec) site coefficient determined in accordance with table 11.4 -2 of ASCE-7-05. This is determined by the Site Class, but must specified if the Site Class is set to F.
Start / End	Specify the start and end times (T) to define the response time range.
Interval	Specify a time step interval for generating the curve.
Generate Spectrum	Closes the dialog and generates response spectrum curve data. The Generated Spectrum dialog (on page 2862) opens to review the IBC response spectrum data.
Cancel	Closes the dialog without generating response spectrum data.

Related Links

- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *M. To add a GB 50011-2010 response spectrum* (on page 869)
- *G.17.3.4 Response Spectrum* (on page 2163)
- *M. To add an IBC 2000 response spectrum* (on page 862)

Repeat Load tab

Used to add load combinations in STAAD.Pro which will be used directly by the analysis engine.

Refer to GS. Load Types in STAAD.Pro (on page 51) for additional information.

Repeat Load

Used to create a primary load case using combinations of previously defined primary load cases. A Repeat Load is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. Load Combinations, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary loadings evaluated independently.

Available Load Cases All primary load cases that are defined for the structure are listed here.

Repeat Load Definition Loads included in this list make up the Repeat Load.

The *Factor* cell is used to specify a factor with which the selected primary load case is to be multiplied. The resulting values are utilized in the repeated load definition

Click this button…	to…
>	Include the selected Primary Load Case in the Repeat Load Definition list.

Click this button...	to...
>>	Include all Primary Load Cases in the Repeat Load Definition list.
<<	Remove all Primary Load Cases from the Repeat Load Definition list.
<	Remove the selected Primary Load Case from the Repeat Load Definition list.

Reference Load

Used to include Reference Loads in a Primary Load case here.

A reference load is defined using the Add New Reference Load Definitions dialog (on page 2839). Load specifications are then added to the Reference Load entry in the **Load & Definition** dialog for later reference.

Available Load Cases
All reference load cases that are defined for the structure are listed here.

Referenced Load list
Loads included in this list make up the Reference Load.

The *Factor*cell is used to specify a factor with which the selected reference load case is to be multiplied. The resulting values are utilized in the primary load definition

Click this button...	to...
>	Include the selected Reference Load Case in the Primary Load Definition list.
>>	Include all Reference Load Cases in the Primary Load Definition list.
<<	Remove all Reference Load Cases from the Primary Load Definition list.
<	Remove the selected Reference Load Case from the Primary Load Definition list.

Notional Load

Used to define a Notional Load from Primary Load Cases and / or Reference Load Cases. These are used for direct analysis.

Primary Load Cases
All primary load cases that are defined for the structure are listed here.

Reference Load Cases
All reference load cases that are defined for the structure are listed here.

Notional Load Definition

Loads included in this list make up the Notional Load.

The *Factor* cell is used to specify the factor with which the selected primary load case or reference load case is to be multiplied. The resulting values are utilized in the notional load definition

The **Direction** cell is used to select a global direction in which the (lateral) notional loads act for this case.

Click this button...	to...
>	Include the selected Primary Load Case or Reference Load Case, respectively, in the Notional Load Definition list.
>>	Include all Primary Load Cases or Reference Load Cases, respectively,in the Repeat Load Definition list.
<<	Remove all Primary Load Cases or Reference Load Case, respectively, from the Repeat Load Definition list.
<	Remove the selected Primary Load Case from the Repeat Load Definition list.

Related Links

- *M. To add a repeat load case* (on page 892)
- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *M. To add a notional load case* (on page 872)
- *M. To create a reference load* (on page 893)

Frequency tab

Used to specify commands to calculate the frequency using the Rayleigh method or by solving an eigenvalue problem.

Rayleigh Frequency

This option has no additional parameters

Modal Calculation

Consider Missing Mas Mode

Check this option to include the missing mass procedure in the analysis for Steady State/Harmonic analysis.

Related Links

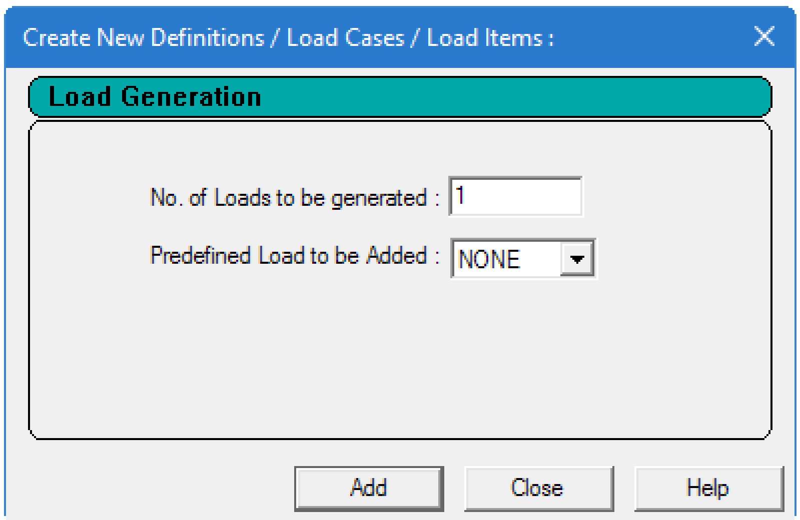
- *M. To calculate the structure frequency* (on page 843)

Load Generation dialog

Used to create a primary load case using the data of a pre-defined vehicle.

Opens when the **Load Generator** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.

Vehicles definitions are created using the **Create New Moving Load Definitions** dialog.



Note: Once a load definition has been created, the **(Moving) Load Generation Type** dialog is used to specify the direction and path for the moving loads.

Refer to TR.32.12.1 Generation of Moving Loads (on page 2596) for additional details.

No. of loads to be generated

The moving load is discretized into a number of distinct positions of the vehicle along the direction of movement. Each position represents a distinct load case whose loads are derived from the corresponding position of the vehicle on the structure. The number of such positions hence has to be communicated in the form of the number of load cases to be generated.

For example, if a bridge is 200 ft long, the first axle to last axle distance of your design vehicle is 15 ft, and you want enough cases to be generated to account for the first axle entering the bridge up to and including the last axle leaving the bridge, that would account for a total distance of 230 ft. If you assume 2 ft increments, then the number of cases would be 230/2 + 1 = 116.

Predefined load to be added

With each generated load case, the program Used to include within that load case, other loads from a previous defined primary load case. Such a primary load case might be for example, the dead load case, which the user may defined as load case 1. So, if the user wishes to have each generated load case to consist of a) the selfweight and other dead loads of the structure b) loads resulting from the position occupied on the structure by the vehicle, then, he/she could specify load case 1 as the predefined load case to be added to each of the generated moving load cases.

Related Links

- *G.16.1 Moving Load Generator* (on page 2134)
- *M. To generate moving load cases* (on page 874)

Wind Load Generation - IS 875 (Part 3): 2015 dialog

Used to automatically generate both wind load definition and wind load cases per the IS 875 (Part 3) : 2015 code.

Opens when the **Wind Load Generator** > **IS 875 (Part 3) : 2015** tool is selected from the **Load Generation** group on the **Loading** ribbon tab.

Notes: The following limitations pertain to generation of IS 875 (Part 3) : 2015 wind loads:

- The wind profile for the loading is calculated based on Y ordinate set to 0 rather than lowest point of the building. Thus, the topographical factor (k3) may not be relevant and should be applied with caution.
- Only Y-up models may be used (that is, the `SET Z UP` command may not be used).
- The resulting wind load intensity vs. height values generated for IS 875 (Part 3) : 2015 are in metric, regardless of the current input units.
- The dynamic effect of wind load is not considered.
- Cl. 6.3.2.4 and Annex B for changes in terrain categories (Table 3 Fetch and Developed Height Relationships) are not considered.
- For rectangular buildings, the frictional drag per Cl. 7.4.1 is not considered.
- For free standing walls and hoarding type structures, the check for oblique wind loads per Cl. 7.4.2.3 is not considered.
- For lattice tower structures, the corner wind effect on square lattice towers with flat-sided members per Cl. 7.4.3.5(b) is not considered.

Wind Profile tab

The calculated wind intensity values (speeds and pressures) are tabulated and graphically plotted as the input parameters are updated. The calculated values are per Cl. 7.2 of IS 875 (Part 3) and the minimum pressure, $pd \geq 0.7 \times pz$, is also considered.

General Information	**Year of Publication**	Select the edition of the IS-875 code from the drop-down list. • 2015
	Structure Type	Select the type of structure from the drop-down list: • Rectangular Clad Building • Unclad Building

	Maximum Height	By default, this dimension is taken as the maximum Y coordinate of the structure. You may also type a **Height** value.
	Ground Level Elevation	(Read-only) Always assumed as zero.
	Height Interval for Intensity	The vertical interval (along the global Y axis) for calculating the wind intensity.
Wind Parameters	**Basic Wind Speed (Vb)**	Either select the name of the location from the drop-down list to use the mapped wind speed per Annex A of IS 875 (Part 3) : 2015 *or* check the **Custom** to type the wind speed directly in m/s.
	Risk Coefficient Factor (k1)	Select class of structure from drop-down option and corresponding k1 value will be automatically selected as per Table 1 of IS 875 (Part 3) : 2015. Alternatively, select **Custom** from the drop-down list to directly type the k1 value for use.
	Topography Factor (k3)	Type the topographic factor to use. The value should be between 1 to 1.36 as per clause 6.3.3.1 of IS 875 (Part3) : 2015.
	Importance Factor (k4)	Select the importance category of the structure from drop-down option and corresponding Importance Factor (k4) value will be selected as per Clause 6.3.4 of IS 875 (Part -3) : 2015. Alternatively, select **Custom** from the drop-down list to directly type the k4 value for use.
	Terrain Category	Select terrain category from drop-down option and corresponding height factor (k2) will be automatically calculated as per Table 2 of IS 875 (Part 3) : 2015 and corresponding Aerodynamic Roughness Height (z0i) value will be automatically selected as per clause 6.3.2.1 of IS 875 (Part 3) : 2015. The Aerodynamic Roughness Height is used for calculating hourly mean wind speed factor (k2,i) and determined mean hourly wind speed at different height as per clause 6.4 of IS 875 (Part 3) : 2015. Alternatively, select **Custom** from the drop-down list to directly type the k2,i and z0i values for use.
	Wind Directionality Factor (Kd)	Defaults to 1.0
	Area Averaging Factor (Ka)	Defaults to 1.0
	Combination Factor (Kc)	Defaults to 1.0

Wind Load Data tab - Rectangular Clad Building

Building Dimensions By default, the dimensions are calculated based on the minimum and maximum global coordinates of the structure along the X, Z, and Y global axes, respectively. You may also type the **Length Along X (Lx)**, **Length Along Z (Lz)**, and **Height (H)** dimension values.

Coefficients Select the option for evaluating the coefficient factor. The associated table with the selection is made active.

- **Pressure Coefficient -** select this option to specify parameters to calculate the pressure coefficients on building face per Cl. 7.3.2 and 7.3.3.
- **Force Coefficient -** select this option to directly specify the **Force Coefficient Factor (Cf)** on the building per Cl. 7.4.2.

External Pressure Coefficient (Cpe)

table	The calculated *Cpe* values for each direction and face are displayed in the table. Double-click to enter a custom value in any table cell. Custom values are highlighted. A **Reset** option is made active if any custom values are used in the table. **Tip:** A tool-tip will also display the calculated value for any cell.
Internal Pressure Coefficient (Cpi)	Check this option to include the Internal Pressure Coefficient (Cpi).
Building Permeability	Select the range of openings to overall building exterior from the drop-down list. This is used to select the Cpi and Cpnet values per Clause 7.3.2.1 of IS-875. Alternatively, select **Custom** from the drop-down list to type a value to use for Cpi
Use (-) Cpi	Check this option to use negative Internal Pressure Coefficient (Cpi) value to calculate Net Pressure Coefficient (Cpnet)

Force Coefficient (Cf)

table	Type a *Cf* value to use for each face (default is 1.0).

Assignments There are four method of assignments for the generated load cases:

- **Automatic -** The program will select faces A, B, C, and D. The list of members in this selection are displayed in the read-only **Member List** cell.
- **Group -** An existing group name is used to define the members which make up the building wind faces.
- **Member List -** Each face is defined by typing a list of members.
- **Range -** Type the minimum and maximum values for the global coordinates (in the units shown) for each face.

Check the **Exclude** option for an row to exclude assigning wind loads to this building face.

Wind Load Data - Unclad Frame Building

Number of Frames Select the number of frames in the structure along the wind direction.

Wind Direction Select the wind direction for this load definition and load case.

Frame Information Select the method for specifying the frame spacing:

- **Dimension -** the actual distance between each frame in the units shown is entered in the **Space Between Frames** cell.
- **Ratio -** the frame spacing ratio as indicated in Table 32 (Clause 7.4.3.4) of IS-875 is entered in the **Spacing Ratio** cell.

Member Profile Cross Section Select if the cross section profile of the frame(s) is **Flat Sided** or **Rounded.** For rounded members, you must also specify the **Diameter** of the members used at each height interval in the frame information table.

Width / Height Type the overall dimensions of the frame. The default values are used based on the detected dimensions of the structure.

Force Coefficient A separate table is listed for each frame in the structure. Select the frame number to display that table's values.

The intermediate values along with the calculated values are displayed in the table.

You must specify the **Solidity Ratio** (Φ) at each height interval of the structure in each frame table. For multiple frames, you must also specify the **Effective Solidity Ratio** (Φ_e) at each height interval for frames No. 2 and higher.

If the **Custom** option is selected, then you must specify the Force Coefficient (**Cf**) values at each height interval. For multiple frames in the Custom option, you must also specify the **Shielding Factor** at each height interval for frames No. 2 and higher.

Assignments There are three method of assignments for the generated load cases:

- **Group -** An existing group name is used to define the members which make up the frames.
- **Member List -** Each frame is defined by typing a list of members.
- **Range -** Type the minimum and maximum values for the global coordinates (in the units shown) for each frame.

Check the **Exclude** option for an row to exclude assigning wind loads to this frame.

Related Links

- *M. To add an IS-875 (Part 3): 2015 wind load* (on page 850)

Create Mass Model dialog

Used to generate a mass model for dynamic load cases based on a selected set of load cases.

Opens the **Mass Model Generator** tool is selected from the **Load Generation** group on the **Loading** ribbon tab.

Add or remove available primary and reference load cases to the selected load cases list using the **>** and **>>** buttons. These will be included with the current **Load Factor** value.

You can remove unwanted load cases from the selected list by using the **<** and **<<** buttons.

Generate Mass as
Select the load case in which the mass model will be generated:

- **New Reference Load Case**
- **Add into 1st Dynamic Load Case**

Load Factor
Type a load factor to use for the load cases to include. This will be applied to the load cases added but can be changed for subsequent load cases.

Available Primary Load Cases
A list of all primary load cases in the model. Only those of the appropriate load types for mass modeling are active for use.

Available Reference Load Cases
A list of all reference load cases in the model. Only those of the appropriate load types for mass modeling are active for use.

Selected Load Cases
This is the list of load cases that will be included in the generated mass model, along with their load type and load factor.

Related Links

- *Rules Used for Generation of Mass Models* (on page 897)
- *M. To generate a mass model* (on page 899)
- *G.17.3.2 Mass Modeling* (on page 2157)
- *M. To generate a mass model* (on page 899)

Define Load Type dialog

Used to define load types for multiple primary load cases in a single dialog. The load type definition is required if you want the program to automatically generate load combinations from the defined primary loads. Load type and reducible options may be set or editing for primary load cases individually when they are created, as well.

Opens when the **Primary Load Type** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.

Load
The primary load case number.

Title
The name of the primary load case.

Type
Select the appropriate load type from the drop-down list in this cell. If the chosen load type is Live or Roof Live, the *Reducible* option is active.

Dead	Soil	Ice
Live	Rain Water/ Ice	Wind on Ice

Roof Live	Ponding	Crane Hook
Wind	Dust	Mass
Seismic-H (horizontal)	Traffic	Gravity
Seismic-V (vertical)	Temperature	Push
Snow	Accidental	None (default)
Fluids	Flood	

Note: If the load type was defined at the time of defining the primary load, the type of the defined load (dead, live, wind etc.) is shown here.

Reducible-IBC 2003 — Check this option if you want the live load to be reduced as per the provisions in IBC 2003.

Related Links

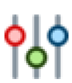
- *M. To define primary load type* (on page 886)

Auto Load Combination dialog

Used to automatically create load combinations based on code-specified combination rules and primary load type selections.

In order to use this feature, one or more primary load cases must be created with a load type defined. A load type can be assigned to a primary load case either at the time when the load is being created or later.

Opens when the **Automatic Combinations** > **Automatic Load Combinations** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.



Note: If no load cases have yet been defined for the model, a warning message is displayed over the dialog and the parameters are inactive. Primary load cases are defined using the **Create Primary Load Case** dialog.

Select Load Combination Code — The load types available include the following:

ACI:318-2002
AISC 9th Ed
ASCE 7-10
BS:5950
BS:8110
GB:50009-2012
GB:55002-2021

IBC-2012
IBC-1997
NBCC-2005
NBCC-1995
IS:456 / IS:800
SNiP 2.01.07-85
UBC-1997
ASCE 7-16

Note: Any Load Rule sets you have added or edited in the **Edit Load Rules for Auto Load Combination Generator** dialog are also available here.

Select Load Combination Category Select the load combination category as specified in the selected load combination code. Refer to the selected code for details.

Select Starting Combination No Type an integer for the initial load combination number. This number should not be a current load case number.

Create Repeat Load Cases Select this option to create primary load cases for all the load case combinations in the Selected Load Combinations list using the Repeat Load Command, rather than combined results using Load Combinations.

See GS. Load Types in STAAD.Pro (on page 51) for additional information on using Repeat loads versus Load Combinations.

Include Notional Load Select this option to include Notional Load cases in the generated combinations when the **Create Repeat Load Cases** option is selected, if they are present in the input file.

Factor Type the notional load factor to use

Use Floor Count Optionally, you may select to use the floor count to determine the notional load factor by checking this option and typing the floor count value. This method is adapted from the Chinese GB50017-2017 code and uses the following formula modified from Cl. 5.2.1-2:

$$\text{factor} = \frac{A}{250}$$

where:

$$2/3 \leq A = \sqrt{0.2 + \frac{1}{n_s}} \leq 1.0$$

n_s = the number of stories

Direction Check the directions for which to generate notional loads: **X**, **-X**, **Z**, and **-Z**.

Generate Loads Click to create all the load combinations. The generated load combinations, with their associated load factors, are shown in the Selected Load Combinations list on the right-hand side.

Discarded Load Combinations Any load combinations which are generated can be removed from the final load combination. These will be listed here.

list operators

Click this button…	to…
>	Add the selected primary load case to the Selected Load Combinations list
>>	Add all primary load cases to the Selected Load Combinations list.
<<	Remove all entries in the Selected Load Combinations list, load combination is placed in the Discarded Load Combinations list.
<	Remove the selected entry from the Selected Load Combinations list.

Selected Load Combinations list All load combinations which will be added to the input file are listed here.

Automatically Generated Load Combinations in the STAAD input file

In the STAAD.Pro Input File, the generated load combinations look like the following:

```
UNIT FEET KIP
LOAD 1 DEAD + LIVE
MEMBER LOAD
2 UNI GY -2.5
LOAD 2 WIND FROM LEFT
JOINT LOAD
2 FX 10
LOAD COMB 3 Generated AISC 1
1 1.6
LOAD COMB 4 Generated AISC 2
1 1.4
LOAD COMB 5 Generated AISC 3
1 1.7 2 0.8
LOAD COMB 6 Manual
1 1.7 2 -1.2
PERFORM ANALYSIS
PRINT MEMBER FORCES
```

The syntax `Generated xxx n`, where *xxx* is the design code name (AISC, BOCA, etc.) and *n* is the index in the series of load combinations generated for a specific code, are interpreted by STAAD.Pro. Deleting any of the aforementioned keywords will break the program's ability to read back in the generated load combinations.

The following are some important caveats to keep in mind when using the automatic load combination generator:

- If the automatically generated load combinations are edited (or deleted) from the STAAD input file, they will be reflected in the graphical interface as well.
- The load combination numbers (i.e. `LOAD COMB` *n* `GENERATED AISC 1`) can also be edited from the input file.
- Deleting the keyword `GENERATED` code index will remove that particular load combination from the automatically generated list. It will not remove the load combination.
- The factors assigned to the primary load cases in a generated load combination *cannot* be changed in the graphical interface using the **Combine** button from the **Loads** dialog box.

Related Links

- *M. To automatically generate load combinations* (on page 888)
- *M. To automatically generate load combinations* (on page 888)

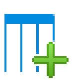
Edit Load Rules for Auto Load Combination Generator dialog

STAAD.Pro includes a facility for the automatic generation of load combination cases. Using the rules described in ACI, AISC, UBC and IBC codes for combining standard load types, you can instruct the program to automatically generate load combination cases from primary load cases which have been assigned load types (i.e., DEAD, LIVE, FLOOD, SNOW, RAIN WATER/ICE, etc.).

You can also use the Edit Loading Rules facility to alter the definitions and factors which come included with the program for these codes or to add new codes containing their own definitions and factors.

Opens when the **Automatic Combinations** > **Edit Auto Combination Rules** tool is selected in the **Load Generation** group on the **Loading** ribbon tab.

Table 326: Automatic Load Combination Manager Home ribbon tab

Tool name	What it Does
 **Code**	Add a new code to the Codes and tables section of the Contents panel
 **Table**	Add a new table to the currently selected code
 **Unlock / Lock Table Updating**	Toggles the editable state of the current code table or configuration page

Tool name	What it Does
' **Append Row**	Adds a new row (load combination) at the end of the current table
Add Above	Adds a new row (load combination) before the currently selected row in the current table
Add Below	Adds a new row (load combination) after the currently selected row in the current table
Commit	Saves changes made to the current table or configuration page
Rollback	Undoes the any changes made to the current table or configuration page
Rename	Click to rename the currently selected table
Copy	Copies the current selection.

Tool name	What it Does
 **Paste**	Pastes the clipboard contents to the current selection.
 **Cut**	Copies and the deletes the current selection
 **Delete**	Deletes the current selection
 **Close Table**	Closes the active table or configuration page.
 **Close All**	Closes all open tables and configuration pages
 **Reset In-built Codes**	Used to reset all codes and tables to the default settings
 **Reset to Default**	Used to reset the currently selected code or table to the default settings

Tables

Displays the rules associated with the selected code and category.

Rows in the table typically represent a single load combination rule. Columns in the table represent load types.

Configurations

Contains instructions for how each load type for tables in this Code should be handled.

Notional Load Select this option for each load type to include Notional Load cases in the generated combinations, if they are present in the input file. These loads are used for Perform Direct Analysis tab (on page 2920).

Combination Type As some load types can result in different actions, a single rule may result in multiple load combinations (i.e., wind or seismic acting in positive or negative directions). The combination rule directs the program how to handle such cases:

- **Aggregate -** Combine all cases together: For each rule, a single combination will be created which will include all the load cases of that load category multiplied by the factor in the table.
- **Separate -** Separate combination for each case: For each rule, multiple combinations will be created, each will include one of the load cases of that load category multiplied by the factor in the table.
- **Matrix -** All possible combinations: For each rule, multiple combinations will be created which will include each of the load cases of that load category on their own and with each and every other load case of that category multiplied by the factor in the table.

Example 1: Combine All Load Cases

Consider both load categories are set with the option **Aggregate.** This will result in a single load combination:

[(LC1 + LC2 + LC3 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]

The STAAD input command generated in the Analytical Workflow is then:

```
LOAD COMBINATION 7 Generated Code 1
1 1.4 2 1.4 3 1.4 4 1.4 5 1.6 6 1.6
```

Example 2: Separate Combinations for Each Case

Consider the DEAD category now is set to **Aggregate** and the LIVE category is set to **Separate.** For this single rule, this will result in two load combinations due to the two LIVE load cases which are to be considered separately, thus:

[(LC1 + LC2 + LC3 + LC4) × 1.4] + [(LC5) × 1.6]
[(LC1 + LC2 + LC3 + LC4) × 1.4] + [(LC6) × 1.6]

The STAAD input command generated in the Analytical Workflow is then:

```
LOAD COMBINATION 7 Generated Code 1
1 1.4 2 1.4 3 1.4 4 1.4 5 1.6
LOAD COMBINATION 8 Generated Code 2
1 1.4 2 1.4 3 1.4 4 1.4 6 1.6
```

Example 3: All Possible Combinations

Consider the DEAD category now is set to **Matrix** and the LIVE category is set to **Aggregate.** For this single rule, this will result in 15 load combinations due to the fourteen ways the four dead loads can be combined together and the two LIVE load cases which are to be considered together, thus:

[(LC1) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC2) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC2 + LC3) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC2 + LC3 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC2 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC3) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC3 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC1 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC2) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC2 + LC3) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC2 + LC3 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC2 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC3) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC3 + LC4) × 1.4] + [(LC5 + LC6) × 1.6]
[(LC4) × 1.4] + [(LC5 + LC6) × 1.6]

The STAAD input command generated in the Analytical Workflow is then:

```
LOAD COMB 7 Generated Code1
1 1.4 5 1.6 6 1.6
LOAD COMB 8 Generated Code2
2 1.4 5 1.6 6 1.6
LOAD COMB 9 Generated Code3
1 1.4 2 1.4 5 1.6 6 1.6
LOAD COMB 10 Generated Code4
3 1.4 5 1.6 6 1.6
LOAD COMB 11 Generated Code5
1 1.4 3 1.4 5 1.6 6 1.6
LOAD COMB 12 Generated Code6
2 1.4 3 1.4 5 1.6 6 1.6
LOAD COMB 13 Generated Code7
1 1.4 2 1.4 3 1.4 5 1.6 6 1.6
LOAD COMB 14 Generated Code8
4 1.4 5 1.6 6 1.6
LOAD COMB 15 Generated Code9
1 1.4 4 1.4 5 1.6 6 1.6
LOAD COMB 16 Generated Code10
2 1.4 4 1.4 5 1.6 6 1.6
LOAD COMB 17 Generated Code11
1 1.4 2 1.4 4 1.4 5 1.6 6 1.6
LOAD COMB 18 Generated Code12
3 1.4 4 1.4 5 1.6 6 1.6
LOAD COMB 19 Generated Code13
1 1.4 3 1.4 4 1.4 5 1.6 6 1.6
LOAD COMB 20 Generated Code14
2 1.4 3 1.4 4 1.4 5 1.6 6 1.6
```

```
LOAD COMB 21 Generated Code15
1 1.4 2 1.4 3 1.4 4 1.4 5 1.6 6 1.6
```

Related Links

- *M. To define automatic load combination rules* (on page 887)
- *M. To define automatic load combination rules* (on page 887)

Create Wind Type Definition dialog

Used to add a new wind load definition.

Opens when the **Wind** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

Type No.	Denotes a number by which the wind load type will be identified. Multiple wind types can be created in the same model.
Comments	An optional description used to help identify the load.
Code	Select the method or building code you want to use for wind load definitions: • Custom - enter a list of height vs wind pressure value pairs • ASCE 7: 1995 • ASCE 7: 2002 • ASCE 7: 2010 • ASCE 7: 2016 • GB 50009-2012 • IS 875 (Part 3): 2015 • SNiP 2.01.07-85 • SP 20.133330.2016

Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *M. To add a wind load definition* (on page 844)
- *M. To add a wind load definition* (on page 844)
- *M. To add an ASCE 7 wind load definition* (on page 845)
- *M. To add a SNiP wind load definition* (on page 847)
- *M. To add a GB50009 wind load definition* (on page 848)
- *M. To add an IS-875 (Part 3): 2015 wind load* (on page 850)

Add New Wind Definitions (data) dialog

Used to add wind pressure data or code-specific wind parameters.

Intensity tab

Select Type	Choose the method of specifying wind pressure values from the drop-down: • **ASCE 7**, **IS 875**, **GB 50009**, or **Custom** – Code generated or user-entered intensity values:

Intensity vs. Height table

Enter pairs of data for wind pressure at a given height (in the current units).

Int cells – wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.

Height cells – corresponding heights in global vertical direction, measured in terms of actual Y (or Z for Z UP) coordinates up to which the corresponding intensities occur.

> **Note:** These values may be negative in the case of GB 50009-2012 wind loads. This indicates a ground level below the base of the building.
>
> In this case, the first row of the Intensity vs. Height is used to simply indicate the ground level.

Generate

Opens the code-specific wind load dialog, which is used to generate a wind load per the ASCE 7, IS 875, or GB 50009 specifications (depending on code selection). Input data on the type of structure, surrounding terrain, and wind.

- **SNiP 2.01.07-85** or **SP 20.13330.2016** – Russian wind load parameters:

Pressure the characteristic value of wind pressure, always positive

Terrain terrain roughness category:

A. Coastal Zone.
B. Urban Zone.
C. Large City.

Region Wind region as per clause 11.5 of SNiP 2.01.07-85* 2016. This is used to determine the wind pressure in determine the dynamic wind component.

Delta logarithmic decrement of oscillations, Delta (see table 11.5 - Section 11.1.8 for the definition). Typically values are:

0.15 for steel towers, masts, lined chimneys, column means including the ones on the reinforced concrete pedestals
0.3 for reinforced concrete and stone structures as well as for buildings with steel framework if there are walling structures

Exposures tab

Factor exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors.

> **Note:** Note any joint does not have an exposure assigned, the exposure will be taken as 1.0.

Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *M. To add a wind load definition* (on page 844)
- *M. To add an ASCE 7 wind load definition* (on page 845)

- *M. To add a SNiP wind load definition* (on page 847)
- *M. To add a GB50009 wind load definition* (on page 848)

ASCE 7 Wind Load dialog box

Used to generate a wind load per the ASCE 7 specification. Input data on the type of structure, surrounding terrain, and wind.

Opens when **Generate** is clicked on the Wind Load tab (on page 2850) when an **ASCE 7** code edition is selected as the code.

Using this feature, you can define different intensity vs. height data for the windward side, leeward side and side walls for a structure. Each can be defined as a different load type and then applied in the relevant direction with appropriate direction factor. The defined wind loading can then be applied to the structure following the usual method.

Note: For details on generation of wind load and command syntax, refer to TR.32.12.3 Generation of Wind Loads (on page 2603) .

Common tab

ASCE -7

Displays the version of the ASCE 7 (ASCE 7 - *Minimum Design Loads for Buildings and Other Structures*) code selected in the **Add New: Wind Definition** dialog: **1995**, **2002**, **2010**, or **2016**.

Building classification category

Select the building classification category, based on the nature of occupancy, as obtained from Table 1-1 in ASCE 7-95 / 02 or from Table 1.5-1 in ASCE 7-10 / 16.

Category can be I, II, III or IV.

Basic Wind Speed

Specify a basic wind speed as described in Figure 6-1 in ASCE 7-95, section 6.5.4 of ASCE 7-02, or section 26.5.1 of ASCE 7-10 / 16, along with units.

Wind speed is assumed to be the plotted or tabulated 3-second gust wind speed.

Exposure Category

Select an exposure category as per section 6.5.3 in ASCE 7-95, section 6.5.6.3 of ASCE 7-02, or section 26.7.3 of ASCE 7-10 / 16.

Structure Type

Select the type or structure that best fits the model from the following:

- Building structures
- Chimneys, Tanks, and similar structures
- Solid Signs
- Open Signs
- Latticed Framework
- Trussed Tower

Note: The data input tab changes to the selected structure type once the **Apply** button is clicked.

Height Above Sea Level

(ASCE 7-16 only) Type the height above sea level for the base of the structure, z_g in Table 26.9-1, in the selected units. This is used to calculate the groud elevation factor, K_e.

Consider Wind speed-up over hill or escarpment	Select **Yes** to consider wind speed-up over a hill or an escarpment and **No** to ignore it. If there are isolated hills and escarpments that constitute abrupt changes in the general topography, the increase in speed can be considered as per clause 6.5.5 in ASCE 7-95, per section 6.5.7 in ASCE 7-02, or per section 26.8.1 of ASCE 7-10 / 16.

Type of Hill or Escarpment	Select the type of hill, ridge or escarpment on which the structure is located, based on Figure 6-2 of ASCE-7-95, Figure 6-4 of ASCE 7-02, or Figure 26.8-1 of ASCE 7-10 / 16. The options available are: • **2-D Ridge** • **2-D Escarpment** • **3-D Axisymmetrical Hill**
Height of Hill or Escarpment (H)	Specify the height of the hill or escarpment relative to the upwind terrain (H in Figure 6-2 of ASCE 7-95, Figure 6-4 ASCE 7-02, Figure 26.8-1 of ASCE 7-10 / 16).
Distance upwind of crest (L_h)	Specify the distance upwind of crest to where the difference in general elevation is half the height of the hill or escarpment (L_h in Figure 6-2 of ASCE 7-95, Figure 6-4 ASCE 7-02, Figure 26.8-1 of ASCE 7-10 / 16).
Distance from the crest to the building (x)	Specify the distance from the crest to the building site. A negative value signifies the distance is in the downwind direction (x in Figure 6-2 of ASCE 7-95, Figure 6-4 ASCE 7-02, or Figure 26.8-1 of ASCE 7-10 / 16).

<structure type> tab

When the structure type is changed on the Common tab, the list of parameters on the Main Building tab changes. The full list of parameters explained here is for the structure type of **Building Structures**. Those parameters that are different are explained below for each type of structure.

Structure Type: Building Structures

Building Height	Specify the height above ground to the highest point on the roof surface, along with units.
Building Length along the direction of Wind (L)	Length of building measured along the direction of wind.
Building Length Normal to the direction of Wind (B)	Length of building measured normal to the direction of wind.
Building Natural Frequency	Specify the natural frequency of the building to calculate Gust Effect Factor.
Building Damping Ratio	Specify the damping ratio to calculate Gust Effect Factor.

Enclosure Classification	Select the building classification as open, partially enclosed, partially open (ASCE 7-16 only), or enclosed per the provisions of section 6.2 of ASCE 7-02, section 26.10 of ASCE-7-10, or section 26.12 of ASCE 7-16. **Note:** Not applicable for ASCE 7-95.
Kz	Velocity Pressure exposure coefficient calculated as per Table 6-3 of ASCE 7-95, Table 6-3 of ASCE 7-02, or Table 26.10-1 of ASCE 7-16).
Use Kzt	Select this option to specify a topographic factor value for wind speedup. If not selected, the topographic factor is calculated and displayed. When wind speedup is considered, Kzt is calculated as per Eq 6-2 of ASCE 7-95, Eq 6-4 of ASCE 7-02, or section 26.8.2 of ASCE 7-16.
Use I	Select this option to specify an importance factor value. If not selected, the importance factor is calculated and displayed. The importance factor used is as per Table 6-2 of ASCE 7-95 or as per section 6.2 of ASCE 7-02.
Use Kd	Select this option to specify a wind directionality factor. If not selected, the wind directionality factor is calculated and displayed. The wind directionality factor is calculated as per Table 6-4 of ASCE 7-02 or Table 26.6-1 of ASCE 7-10 / 16. **Note:** Not applicable for ASCE 7-95.
Use Ke	(ASCE 7-16 only) Select this option to specify a ground elevation factor. If not selected, the ground elevation factor is calculated and displayed. The ground elevation factor is calculated as per Table 26.9-1 of ASCE 7-16.

Structure type : Chimney, Tank and similar Structures

Height(H)	Height of the structure as defined by the term “h” in Figure 6-19 of ASCE 7-02, Figure 29.5-1 of ASCE 7-10, or Figure 29.4-1 of ASCE 7-16.
Least Horizontal Dimension (W)	Smaller of the plan dimensions. In case the cross section of the structure in plan is circular, the diameter needs to be specified.
Horizontal Cross-Section Type	This is the cross section of the structure in plan as defined in Figure 6-19 of ASCE 7-02, Figure 29.5-1 of ASCE 7-10, or Figure 29.4-1 of ASCE 7-16. The available options include square with wind being normal to face or acting along the diagonal, hexagonal, octagonal and round. For ASCE 7-16, additional options are octagonal (non-axisymmetric), octagonal (axisymmetric), round (non-axisymmetric), and round (axisymmetric).
Depth of protruding elements such as ribs and spoilers (D’)	For round type cross sections, depth of protruding elements need to be defined which is a measure of the surface roughness as indicated in Figure 6-19 of ASCE 7-02, Figure 29.5-1 of ASCE 7-10, or Figure 29.4-1 of ASCE 7-16.

Structure type: Solid Signs

Height (H)	Height of the structure which is used for calculating the height to width ratio as defined by the term "v" in Figure 6-20 of ASCE 7-02 or the term "h" in Figure 29.4-1 of ASCE 7-10 or Figure 29.3-1 of ASCE 7-16.
Horizontal Dimension of Sign (M)	Horizontal dimension of the solid sign as defined by the term "B" in Figure 29.4-1 of ASCE 7-10 or Figure 29.3-1 of ASCE 7-16.
Vertical Dimension of Sign (N)	Vertical dimension of the solid sign as defined by the term "s" in Figure 29.4-1 of ASCE 7-10 or Figure 29.3-1 of ASCE 7-16. **Note:** If the sign is at the ground level, the height (H) and vertical dimension (N) should both be specified the same value.

Structure type: Open Signs / Lattice Frame Work

Ratio of Solid Area to Gross Area	Ratio of solid area to gross area as indicated by the term "$\in$" in Figure 6-21 of ASCE 7-02, Figure 29.5-2 of ASCE 7-10, or Figure 29.4-2 of ASCE 7-16.
Orientation of the members exposed to wind	The type of member surfaces which are exposed to wind. Select flat-sided members or rounded members in Figure 6-21 of ASCE 7-02, Figure 29.5-2 of ASCE 7-10, or Figure 29.4-2 of ASCE 7-16.
Diameter of typical round member	Diameter for round members as defined by the term "D" in Figure 6-21 of ASCE 7-02, Figure 29.5-2 of ASCE 7-10, or Figure 29.4-2 of ASCE 7-16.

Structure type : Trussed Tower

Horizontal Cross Section	The type of cross section of the tower in plan as defined in Figure 6-22 of ASCE 7-02, Figure 29.5-3 of ASCE 7-10, or Figure 29.4-3 of ASCE 7-16. The available options include square and triangle.

Building Design Pressure tab

Building Wall to generate Wind Load on:	Select the side of the structure, with respect to wind direction, for which you wish to generate a wind load. The pressure will be calculated as per Table 6-1 of ASCE 7-95 , Equation 6-23 of ASCE 7-02, Figure 27.4-1 of ASCE 7-2010, or Figure 27.3-1 of ASCE 7-16, based on the code selection made on the **Common** tab. The relevant equation will also be displayed below. • **Windward -** Used to generate the design wind pressure for the windward side. • **Leeward -** Used to generate the design wind pressure for the side wall. • **Side Wall -** Used to generate the design wind pressure for the side wall. **Note:** The pressure profile for each of the building walls has to be individually determined under a unique load type number. Thus, generating the profile for the three sides of the building constitutes three separate steps and thus, three separate types. Each type can then be applied with one load case or separate load cases and then applied in the relevant direction with the appropriate direction factor. Examples illustrating wind load generation can be found in the examples manual.
Use G	Select this option to specify a gust effect factor value.

If not selected, the Gust effect factor is calculated and displayed. The gust effect factor is calculated as per Table 6-1 of ASCE 7-95 or as per section 6.5.8 of ASCE 7-02.

For rigid structures (natural frequency ≥ 1Hz), the gust effect factor is calculated as per equation 26.9-6 for ASCE 7-10 or equation 26.11-9 for ASCE 7-16 . For flexible structures (natural frequency < 1Hz), the gust effect factor is calculated section 26.9-10 for ASCE 7-10 or section 26.11-10 for ASCE 7-16.

Use C_p Select this option to specify a wall pressure coefficient value.

If not selected, the wall pressure coefficient is calculated and displayed. The wall pressure coefficient is calculated as per as per Fig 6-3 of ASCE 7-95 or as per Figures 6-11 through 6-17 of ASCE 7-02. For ASCE 7-10, the wall pressure coefficient is calculated as per Figure 27.4-1, 27.4-2 and 27.4-3. For ASCE 7-16, the wall pressure coefficient is calculated as per Figure 27.3-1.

Use GC_{pi} Select this option to specify an internal pressure coefficient value.

If not selected, the internal pressure coefficient is calculated and displayed. The internal pressure coefficient is calculated as per Table 6-4 of ASCE 7-95 or as per Figure 6-5 of ASCE 7-02. For ASCE 7-10, the internal pressure coefficient is calculated as per Figure 27.4-1, 27.4-2 and 27.4-3. For ASCE 7-16, the internal pressure coefficient is calculated as per Figure 26.13-1.

Use C_f Select this option to specify a force coefficient value used for structures other than buildings. If not selected, the force coefficient is calculated and displayed as follows:

- ASCE 7-02: If not selected, the force coefficient is calculated as per Figures 6-19, 6-20, 6-21, or 6-22, depending on the non-building structure type selected.
- ASCE 7-10: If not selected, the force coefficient is calculated as per Figures 29.4-1, 29.5-1, 29.5-2 or 29.4-3, depending on the non-building structure type selected.
- ASCE 7-16: If not selected, the force coefficient is calculate as per sections C29.3.1., Figure 29.4-1, Figure 29.4-2, or Figure 29.4-3, depending on the non-building structure type selected.

The force coefficient is used for the calculation of the design pressure.

Height vs. Intensity table Displays the generated wind profile based on the current input in the dialog.

OK Accepts the current Height vs. Intensity table values displayed on the Building Design Pressure tab and applies them to the **Add New: Wind Definitions** dialog.

Apply Applies changes made in one of the dialog tabs to other tabs for use (e.g., the code selection was changed).

Cancel Closes the dialog without adding an ASCE wind definition.

Help Opens the STAAD.Pro Help window.

Related Links

- *M. To add an ASCE 7 wind load definition* (on page 845)
- *M. To add an ASCE 7 wind load definition* (on page 845)
- *M. To add an ASCE 7 wind load definition* (on page 845)

Generate Wind Definition and Wind Load Case for Chinese GB 50009 dialog

Used to enter code parameters for generating wind intensity values based on the GB 50009 2012 code

Opens when **Generate** is clicked in the **Add New Wind Definitions** dialog with the **GB 50009-2012** type selected.

Wind Definition tab

Calculate Wind Height Factor (mu_z)

The following parameters are input used calculate the height variation factor for the wind pressure, μ_z:

Specify Ground Line

Check this option to specify the **Mapped Ordinate in Model** value. This is the elevation of the bottom of the current calculation part of the current building. If unchecked, the program assumes a ground level of Y = 0.

> **Note:** STAAD.Pro supports negative values for the Y coordinate used for wind load generation for the GB 50009-2012 code only.

Focused Building Height (H)

This is the total height (H) of the current building. Click the **Geometry Figure** button to open a diagram with further explanation of the building height vs. ground level values.

Height (z) & Wind Intensity Curve

Select the method to sub-divide the height above ground into discreet points of the wind intensity curve:

- **Segment Count to Divide Height (H) Equally -** This is the segment count to divide equally the total height (H) of the current building. This is used to calculate the individual height (z) where the wind pressure needs to be calculated.
- **Equal Segment Length Along Height (H) -** This is the segment length, and the length is equal. The length is along the total height (H) of the current building. This is used to calculate the individual height (z) where the wind pressure needs to be calculated.
- **Provide Special Height (z) -** This is the special height (z) list. This is the individual height (z) where the wind pressure needs to be calculated. The delimiter for the individual height (z) can be comma (,) or space, such as `3,6,9`.

Roughness Type

This is the roughness type. The value can be A, B, C or D.

Modification Factor

This is the modification factor. This is to modify the height variation factor for the wind pressure.

Calculate Wind Shape Factor (mu_s)

The following parameters are input used to calculate the wind shape factor, μ_s:

Shape Item ID

The list of the shape item ID and the shape item name, these shape items are from Chinese load code GB 50009.

- 30 : Closed Polygon Building
- 37 : Circular Section Structure (Chimney)

Secondary Shape Type	The secondary shape types of the shape item 30 (Closed Polygon Building) are shown as the following: • Rectangle • L Shape • C Shape

Set Shape Factor ...	This is used to set the shape factor of the structure. STAAD.Pro will set the default values of the shape factor of the current secondary shape type of the current shape item from the config database on the **Set Shape Factor** dialog. You can modify the shape factor.
	Positive value is for the windward face.
	Negative value is for the leeward face.
	Every face name is shown in the first row, every direction of wind from is shown in the first column, on the **Set Shape Factor** dialog.
	Tip: Refer to the diagrams in the dialog for the which sides of the building each structure face label refers to.
	For circular structures, the shape factor is calculated based on several parameters:
	• Bulge Height on Face (delta) -
	• Section Diameter - select if there is any change in diameter along the height and if that change is linear or nonlinear:
	  • **No Change** (i.e., straight) - type the **Section Diameter** of the entire structure
	  • **Linear Change** (i.e., tapered) - type the **Section Diameter** (at the base) and the **Top Section Diameter**.
	  • **Nonlienar Change** (i.e., stepped) - type the height and diameter of each section along the structure in the table

Interference Factor from Other Building	This is the mutual interference factor from other building.

Calculate Reference Wind Pressure (w0)

The following parameters are used to calculate the reference wind pressure, $w0$:

Reference Wind Pressure (w0)	This is the reference wind pressure (w0).
Modification Factor	This is the modification factor. This is to modify the reference wind pressure (w0).

Calculate Along-wind Vibration Factor (beta_z)

The following parameters are input used to calculate the wind-induced vibration factor, β_z:

Consider Along-wind Vibration Factor	Consider the wind-induced vibration factor, β_z, along-wind or not.

Damping Ratio	This is the damping ratio of the current building. It should be greater than 0.
Basic Natural Vibration Period (T1)	This is the basic natural vibration period (T1) of the current building.
Structure Type	Select either: • **High-rise Building -** The structure type of the current building is high-rise building. The high-rise building is the modern building which is very tall and has many levels or floors. The width of the windward face of the high-rise building is larger than that of the high-tower structure. • **High-tower Structure -** The structure type of the current structure is high-tower structure. The width of the windward face of the high-tower structure is far less than its height, and is also less than that of the high-rise building.
Provide Width of Windward Face ...	This is used to provide the width of the windward face of the current structure. **Wind From**: This is the direction of the wind from. **Width**: This is the width of the windward face of the current structure. This is just for the high-rise building. **Tip:** It is recommended to use the top width of the windward face is equal to the bottom width for the high-rise building. **Bottom**: This is the bottom width of the windward face of the current structure. This is just for the high-tower structure. **Top**: This is the top width of the windward face of the current structure. This is just for the high-tower structure.

Set Influence Factor for Wind Direction: Click this to open a dialog where independent influence factors can be specified for each wind direction: Left (+X), Right (-X), Front (-Z), and Rear (+Z). These are all taken as unity (1.0) by default.

Wind Load Case tab

Generate Wind Load Case: The wind load case associated with the wind definition will be generated when this parameter is checked, otherwise, the wind load case will be not generated.

Define Range of Wind Load ...: This is used to define the range of the wind load action for every face of the structure. Every face is shown in the first column on the dialog.

X Min	The min value of X axis for the current face of the structure.
X Max	The max value of X axis for the current face of the structure.
Y Min	The min value of Y axis for the current face of the structure.

Y Max	The max value of Y axis for the current face of the structure.
Z Min	The min value of Z axis for the current face of the structure.
Z Max	The max value of Z axis for the current face of the structure.
Open	The structure is open or not.

Related Links

- *M. To add a GB50009 wind load definition* (on page 848)
- *M. To add a GB50009 wind load definition* (on page 848)

IS-875 (Part 3): Wind Load dialog

Used to enter code parameters for generating wind intensity values based on the IS-875 (Part 3): Wind Load code

Opens when **Generate** is clicked in the **Add New Wind Definitions** dialog with the **IS-875 (Part 3): Wind Load** type selected.

Notes: The following limitations pertain to generation of IS 875 (Part 3) : 2015 wind loads:

- The wind profile for the loading is calculated based on Y ordinate set to 0 rather than lowest point of the building. Thus, the topographical factor (k3) may not be relevant and should be applied with caution.
- Only Y-up models may be used (that is, the `SET Z UP` command may not be used).
- The resulting wind load intensity vs. height values generated for IS 875 (Part 3) : 2015 are in metric, regardless of the current input units.
- The dynamic effect of wind load is not considered.
- Cl. 6.3.2.4 and Annex B for changes in terrain categories (Table 3 Fetch and Developed Height Relationships) are not considered.
- For rectangular buildings, the frictional drag per Cl. 7.4.1 is not considered.
- For free standing walls and hoarding type structures, the check for oblique wind loads per Cl. 7.4.2.3 is not considered.
- For lattice tower structures, the corner wind effect on square lattice towers with flat-sided members per Cl. 7.4.3.5(b) is not considered.

Common tab

Year of Publication	Select the edition of the IS-875 code from the drop-down list. • 2015
Structure Type	Select the type of structure from the drop-down list: • Rectangular Clad Building • Unclad Building • Free Wall and Hoardings • Lattice Tower

Maximum Height	By default, this dimension is taken as the maximum Y coordinate of the structure. You may also type a **Height** value.	
Ground Level Elevation	(Read-only) Always assumed as zero.	
Height Interval for Intensity	The vertical interval (along the global Y axis) for calculating the wind intensity.	
Basic Wind Speed (Vb)	**Cities**	Select the name of the location from the drop-down list to use the mapped wind speed per Annex A of IS 875 (Part 3) : 2015.
	Use Custom	Check this option to type the wind speed directly in m/s.
Wind Parameters	Based on the selected or typed values, the Design Wind Speed is automatically calculated using the formula ($Vb \times k1 \times k2 \times k3 \times k4$) per clause 6.3 of IS 875 (Part 3) : 2015 at given height interval. A Height vs. Velocity graph is plotted for the value of height and design wind speed (Vz).	
	Class of Structure	Select class of structure from drop-down option and corresponding **Risk Coefficient Factor (k1)** value will be automatically selected as per Table 1 of IS 875 (Part 3) : 2015. Alternatively, select **Custom** from the drop-down list to directly type the k1 value for use.
	Terrain Category	Select terrain category from drop-down option and corresponding height factor (k2) will be automatically calculated as per Table 2 of IS 875 (Part 3) : 2015 and corresponding Aerodynamic Roughness Height (z0i) value will be automatically selected as per clause 6.3.2.1 of IS 875 (Part 3) : 2015. The Aerodynamic Roughness Height is used for calculating hourly mean wind speed factor (k2,i) and determined mean hourly wind speed at different height as per clause 6.4 of IS 875 (Part 3) : 2015. Alternatively, select **Custom** from the drop-down list to directly type the k2,i and z0i values for use.
	Importance of Structure	Select the importance category of the structure from drop-down option and corresponding Importance Factor (k4) value will be selected as per Clause 6.3.4 of IS 875 (Part -3) : 2015.
	Topography Factor (k3)	Type the topographic factor to use. The value should be between 1 to 1.36 as per clause 6.3.3.1 of IS 875 (Part3) : 2015.

<structure type> tab

Note: The title and parameters on this tab vary based on the **Structure Type** selection made on the **Common** tab.

Rectangular Clad Buildings

Wind Direction	Select one of the global directions for the wind: +X, -X, +Z, or -Z.

Building Dimensions	By default, the dimensions are calculated based on the minimum and maximum global coordinates of the structure along the X, Z, and Y global axes, respectively. You may also type the **Length Along X (Lx)**, **Length Along Z (Lz)**, and **Height (H)** dimension values.
Use Coefficient Factor	Select the option for evaluating the coefficient factor: • **None -** no coefficient factor is multiplied with wind intensity value. • **Pressure Coefficient -** select this option to specify parameters to calculate the pressure coefficients on building face per Cl. 7.3.2 and 7.3.3. • **Force Coefficient -** select this option to directly specify the **Force Coefficient Factor (Cf)** on the building per Cl. 7.4.2.
Pressure Coefficient on Face	Use these options when the **Use Coefficient Factor** option selected is **Pressure Coefficient**:

Wind on Face	Select the face of the structure (as labeled in the diagram in the dialog) to which this wind definition is applicable.
External Pressure Coefficient (Cpe)	Select to use either the **Code Calculated** or **Custom** (manually typed) value of Cpe. If the maximum range of l/w from Tabe 5 of IS 875 (Part 3): 2015 is exceeded, then the upper values of Cpe for $3/2 < l/w < 4$ are still used. You may also enter custom Cpe values.
Internal Pressure Coefficient (Cpi)	Check this option to include the Internal Pressure Coefficient (Cpi).
Building Permeability	Select the range of openings to overall building exterior from the drop-down list. This is used to select the Cpi and Cpnet values per Clause 7.3.2.1 of IS-875. Alternatively, select **Custom** from the drop-down list to type a value to use for Cpi
Use (-) Cpi	Check this option to use negative Internal Pressure Coefficient (Cpi) value to calculate Net Pressure Coefficient (Cpnet)

Force Coefficient on Building	Type the Force Coefficient (Cf) value as per clause 7.4.2 of IS-875 when the **Use Coefficient Factor** option selected is **Force Coefficient**.

Unclad Frame Building

Frame Type	Select if the structure contains a **Single** frame or **Multiple** frames spaced along the wind direction. For either frame type, you must specify the **Solidity Ratio** (Φ) at each height interval of the structure in the frame information table. For multiple frames, you must also specify the **Effective Solidity Ratio** (Φ_e) at each height interval. Providing the **Solidity Ratio** (Φ) for a single frame at the given height interval, the force coefficient (Cf) will be selected according to the user provided solidity ratio values from Table 31 of IS 875 (Part 3) : 2015 by interpolation. Providing the **Effective Solidity Ratio** (Φ_e) for multiple frames at the given height interval, the shielding factor (H) will selected according to the user provided effective solidity ratio values from Table 32 of IS 875 (Part 3) : 2015 by interpolation.
Member Profile Cross Section	Select if the cross section profile of the frame(s) is **Flat Sided** or **Rounded.** For rounded members, you must also specify the **Diameter** of the members used at each height interval in the frame information table.

Frame Spacing Ratio For **Multiple** frames, type the frame spacing ratio as indicated in Table 32 (Clause 7.4.3.4) of IS-875.

Force Coefficient The intermediate values along with the calculated values are displayed in the table.

If the **Custom** option is selected, then you must specify the Force Coefficient (**Cf**) values at each height interval. For multiple frames in the Custom option, you must also specify the **Shielding Factor** at each height interval in the table.

Free Wall and Hoardings

Walls or Hoarding Parameters

Horizontal Dimension (b)	Type the horizontal dimension of the wall or hoarding component.
Vertical Dimension (h)	Type the vertical dimension of the wall or hoarding component.
Structure Height (H)	This is calculated as the difference between the maximum and minimum Y coordinates in the model. You may also type a value here.

Height Above Ground Select either the **Automatic** (calculated) or **Custom** option. For the automatic option, the height above ground (h') value used is taken as the structure height, H, minus the vertical dimension, h. The custom option lets you type a different value.

Force Coefficient The Force coefficients factor (Cf) will be calculated automatically based on the width to height ratio (horizontal dimension, b, divided by vertical dimension, h) and whether the free-standing wall or hoarding are above ground or on ground as given in Table 26 of IS 875 Part (3) : 2015. No interpolation of Table 26 is assumed. So if a width to heigh ratio, b/h, is between to tabulated values, then the next greater value of Cf is used.

If the **Custom** option is selected, then you must specify the Force Coefficient (**Cf**) value.

Lattice Tower

Tower Information

Structure Cross Section	Select if the tower structure cross section is **Square** or **Equilateral Triangle**.
Member Profile Cross Section	Select if the cross section profile of the tower is **Flat Sided** or **Rounded**. For rounded members, you must also specify the **Diameter** of the members used at each height interval in the solidity ratio table.

Solidity Ratio For either tower type, you must specify the **Solidity Ratio** of the front face (Φ) at each height interval of the structure in the solidity ratio table.

Force Coefficient The Force coefficients factor (Cf) will be calculated at each height interval automatically based solidity ratio and the tower type as per the following table (with linear interpolation between tabulated values). The force coefficient table also displays the intermediate calculated values.

Tower Type	Member Cross Section	Code Reference
Square	Flat Sided	Table 33 / Clause 7.4.3.5(a)

Tower Type	Member Cross Section	Code Reference
	Rounded	Table 34 / Clause 7.4.3.5(d)
Equilateral Triangle	Flat Sided	Table 33 / Clause 7.4.3.5(a)
	Rounded	Table 35 / Clause 7.4.3.5(e)

If the **Custom** option is selected, then you must specify the Force Coefficient (**Cf**) value.

Design Wind Pressure tab

This tab displays the resulting height vs. design wind pressure (pd) chart and graph per Cl. 7.2 of IS 875 (Part 3). The program includes a check for the minimum design wind pressure of $0.7 \times pz$. The intensity values generated are the net pressure coefficient *or* the force coefficient times the design wind pressure (pd).

Wind Directionality Factor (Kd)	Defaults to 1.0.
Area Averaging Factor (Ka)	Defaults to 1.0.
Combination Factor (Kc)	Defaults to 1.0.

Related Links

- *M. To add an IS-875 (Part 3): 2015 wind load* (on page 850)

Add New Snow Definition dialog

Used to define parameters used for generating snow loading on a structure per the ASCE 7-02 code.

Opens when the **Snow** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

As a result, the members for which the snow load is to be generated must be clustered together into Floor Groups. For details on creation of groups, refer to TR.16.1 Listing of Entities by Specifying Groups (on page 2235) .

Type No:	Specify an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model.
Ground Snow Load	The pressure or, weight per unit area, to be used for the calculation of the design snow load, in current units. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of the SEI/ASCE 7-02 code.
Exposure Factor	Specify the exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.
Thermal Factor	Specify the thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

Importance Factor	Specify the snow importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Related Links

- *M. To add an ASCE 7-02 snow load* (on page 870)

Add New Seismic Definitions dialog

Used to define the parameters for performing a dynamic analysis using the static equivalent approach as outlined in the various seismic codes supported by STAAD.Pro.

Opens when the **Seismic** > ***<code name>*** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

Weights are added to a seismic definition through the other tabs of the **Add New: Seismic Definition** dialog. Seismic load items using this definition are applied using the **Add New : Load Items** dialog.

Seismic Parameters tab

Type

Select the code you wish to use for defining seismic parameters.

- TR.31.2.22 UBC 1994 or 1985 Load Definition (on page 2429)
- TR.31.2.23 UBC 1997 Load Definition (on page 2432)
- TR.31.2.13 IBC 2000/2003 Load Definition (on page 2401)
- TR.31.2.14 IBC 2006/2009 Seismic Load Definition (on page 2405)
- TR.31.2.15 IBC 2012 Seismic Load Definition (on page 2409)
- TR.31.2.16 IBC 2015 Seismic Load Definition (on page 2413)
- TR.31.2.17 IBC 2018 Seismic Load Definition (on page 2416)
- TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387)
- TR.31.2.9 IS:1893 - 1984 Code - Lateral Seismic Load (on page 2386)
- TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load (on page 2393)
- TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load (on page 2398)
- TR.31.2.2 Canadian Seismic Code (NRC) - 1995 (on page 2355)
- TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1 (on page 2358)
- TR.31.2.4 Canadian Seismic Code (NRC) - 2010 (on page 2362)
- TR.31.2.19 CFE (Comisión Federal De Electricidad) Seismic Load (on page 2420)
- TR.31.2.20 NTC (Normas Técnicas Complementarias) Seismic Load (on page 2423)
- TR.31.2.18 Japanese Seismic Load (on page 2418)
- TR.31.2.7 Colombian NSR-98 Seismic Load (on page 2382)
- TR.31.2.21 Turkish Seismic Code (on page 2426)
- TR.31.2.1 RPA (Algerian) Seismic Load (on page 2353)
- TR.31.2.5 Chinese Static Seismic per GB50011-2001 (on page 2369)
- TR.31.2.6 Chinese Static Seismic per GB50011-2010 (on page 2375)
- TR.31.2.7 Colombian NSR-98 Seismic Load (on page 2382)
- TR.31.2.8 Colombian NSR-10 Seismic Load (on page 2384)

Include Accidental Load	Select this option to calculate the accidental torsion component described in the appropriate code.
Include IS 1893 Part 4	(IS 1893 - 2002/2005 only) Set this option to specify loads per IS 1893 (Part 4) 2005 for industrial or stack-like structures. When set, the a value for ST is required (1 or 5). When set and ST of 5 is used, CV and DV are also required. **Notes:** If this check box is clear, then IS 1893 (Part 1) 2002 is used. To use IS 1893 (Part 4) 2015, select that code separately.
Generate	(IS 1893 - 2002/2005 only) Opens the **IS:1893 Seismic Parameters** dialog box, which is used to populate the parameter fields with appropriate values for the IS 1893 - 2002 code.
Parameters table	A table of the code-relevant seismic parameters is listed. Refer to TR.31.2 Definitions for Static Force Procedures for Seismic Analysis (on page 2349) for technical details on each code parameter. **Note:** Required parameters are marked with an "*".

Wall Area tab

Used to add shear wall areas for the first story above ground necessary to compute static seismic loads for IS1893 2016 only. Specify seismic resisting walls for each global direction (X and Z) only. The width is the thickness of the wall and length is the length along the global direction (i.e., plan dimensions of the walls).

Note: Specifying wall data for any seismic code *other* than IS1893 2016 has no effect.

Refer to Wall Area Definitions (on page 2352) for details.

Self Weight tab

Used to add the selfweight of the structure.

SelfWeight Factor	The factor to be used to multiply the selfweight.

Joint Weights tab

Used to add the concentrated weights acting at one or more joints.

Joint Weight	Specify the magnitude of the joint weight.

Member Weights tab

Used to add distributed and concentrated weights acting on member spans.

Loading Type	Select either a **Concentrated** or **Uniform** load type.
Weight	Specify the intensity of the distributed weight or magnitude of the concentrated weight.

Starting / Ending Distance	Specify the location along the member where the weight is applied, in current units.

Element Weights tab

Used to add pressure loads on slabs if the structural model consists of plate elements representing entities like floor slabs.

Pressure	Specify the magnitude of the uniform pressure, in the current units. Since it is a weight, it is a quantity without a sign.

Reference Load tab

Used to add reference loads to the seismic definition.

Available Load Cases	A list of all previously defined reference loads included in the STAAD.Pro input file. Reference loads are created using the Add New Reference Load Definitions dialog (on page 2839).
Referenced Load	A list of included reference load cases for to be used in the seismic definition are included here. You may specify a different **Factor** value by which the reference load case is multiplied when used for the seismic load.
Along	Select a global direction along which the reference load case acts as a weight.

Floor Weights tab

Used to add floor loads when no slab is present or defined in the structural model.

For additional information on the Floor Load command, refer to TR.32.4.3 Floor Load Specification (on page 2483) . The parameters for Floor Weight are analogous to those for the Floor Load command.

Tip: If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better. See illustrative example at the end of this section

Range	Select this option to specify a Floor load or One-way load by specifying a range.
Group	Select this option to specify a Floor load or One-way load to act on a previously defined Floor group. Groups are created using the Create Group dialog (on page 2950).
Pressure	Specify the magnitude of the uniform pressure, in the current units. The pressure value is provided as a quantity without sign because it is contributing to the overall weight - a numerically positive term.
Define Y/X/Z Range	Specify **Minimum** and **Maximum** values to define a range in the global directions when the Range option is selected.
One Way Distribution	Set this option to use a one-way load distribution. Refer to TR.32.4.2 One-way Load Specification (on page 2476) for additional information on using a one-way load.

Towards	(One-way Distribution only) Select an existing member onto which the loading is directed and defines the span direction for the one way loading. Otherwise, the load will be distributed to the longer member.
Member Group	Select a previously defined floor group from the list.
Inclined	(Group option only) This option is required to be set when a group of members that form a panel are inclined to the global XY, YZ, or ZX planes.

Related Links

- *M. To add a seismic load definition* (on page 857)
- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *M. To add wall data area to an IS1893 2016 seismic definition* (on page 858)

IS:1893 Seismic Parameters dialog box

Used to populate the parameter fields with appropriate values for the IS 1893 - 2002 code.

Opens when **Generate** is clicked in the Add New Seismic Definitions dialog (on page 2895) with the **IS 1893 - 2002** code selected.

Refer to TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load (on page 2387) for additional information on all parameters.

Zone Factor Choice	In the first drop-down list, select either City or Zone. The second drop-down list is then populated with a list of major Indian cities or seismic zones. The corresponding zone factor, Z, is displayed for the current selection.
Response Reduction	Select the type of lateral load resisting system, as defined by the IS 1893 code, used in the structure for the direction of loading.
Importance Factor	Select the importance classification, as defined by the IS 1893 code.
Rock/Soil Type	Select the soil classification for the structure site.
Structure Type	Select the material used for the structure.
Damping Ratio	Type a damping ratio, in percent.
Foundation Depth	Set this option to display a field in which to type the depth of foundation below ground level, in the current units of length.
Period in X (sec)	Set this option to display a field in which the period of structure (in sec) in X direction.
Period in Z (sec)	Set this option to display a field in which the period of structure (in sec) in Z direction.
Generate	Closes the dialog box and populates the values in the Seismic Parameters dialog box.
Cancel	Closes the dialog box without generating the values for the Seismic Parameters dialog box.

Related Links

- *TR.31.2 Definitions for Static Force Procedures for Seismic Analysis* (on page 2349)
- *M. To add wall data area to an IS1893 2016 seismic definition* (on page 858)

Add New Direct Analysis Definition dialog

Used to define parameters used in the direct analysis method described in AISC 360 Appendix 7.

Opens then the **Direct Analysis** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

Note: This feature is used in conjunction with the **Perform Direct Analysis** command.

Refer to TR.31.7 Definition of Direct Analysis Members (on page 2454) for additional details.

Each tab in the dialog contains a different parameter used for direct analysis. The parameters are:

FLEX Identification of members whose flexural stiffness is considered to contribute to the lateral stability of the structure, along with the initial value of τb that should be used. Members listed with `FLEX` will have their EI factored by 0.80 times τ_b while performing the global solution. The final member forces and code check will be with 100% of the flexural stiffness.

AXIAL Identification of members whose axial stiffness is considered to contribute to the lateral stability of the structure. These members will have their EA factored by 0.80 while performing the global solution. The final member forces and code check will be with 100% of the axial stiffness.

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *M. To define direct analysis parameters* (on page 871)

Add New Vehicle Definitions dialog

Used to define different types of moving loads. Several load cases can be generated by applying these types of loads.

Opens when the **Vehcile** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

In order to generate a set of static loads, due to the movement of the vehicle or load on the structure, there are two steps involved. The first step is to define the vehicle. The second step involves the Load Generation dialog (on page 2866).

Define Load tab

Used to define a moving load system in the input file.

Vehicle Type Ref	This is the reference number.
Width	Specify a wheel spacing, if applicable. The **Width** is the spacing between loads perpendicular to the direction of movement. If omitted, one directional loading is assumed.
Load, Dist table	Enter the value of the concentrated loads and the distance between them. The first load is used as the Reference load for vehicle placement in the **(Moving) Load Generation Type** dialog.

AASTHO Spec tab

Used to specify a standard AASHTO load in the input file.

Vehicle Type Ref	This is the reference number.
AASHTO Specification	Choose from the following standard AASHTO loading: HS20, HS15, H20 and H15.
Factor	Specify a multiplying factor, if applicable. The default factor is 1.0.
Variable Spacing	(For use with HS loads only) Specify a value for a variable wheel spacing between the second and third axles, per Fig. 3.7.7A of AASHTO Standard Specification or Cl. 3.6.1.2.2 of the AASHTO LRFD. Valid range from 14 feet to 30 feet (inclusive), with the default being 14 feet if not specified.

File Input tab

Used to specify a moving load defined in an external file.

Vehicle Type Ref	This is the reference number.
File Name	Name of the external file containing moving load data. File name should be limited to 16 characters, reside in the same directory as the current input and have no file extension. For more information on creating an external moving load file, see TR.31.1 Definition of Moving Load System (on page 2346).
Load Name	Name assigned to the moving load system in the external file.
Factor	Specify a multiplying factor, if applicable.

Related Links

- *G.16.1 Moving Load Generator* (on page 2134)
- *M. To define a vehicle for loading* (on page 873)

Add New : Pushover dialog

Used to define structural parameters for a pushover analysis.

Opens when the **Pushover** tool is selected in the **Define Load Systems** group on the **Loading** ribbon tab.

Common Dialog Controls

Add	Adds the parameters set on the current tab to the input file.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Define Input tab

Used to define general definition and member specific parameters for a pushover analysis.

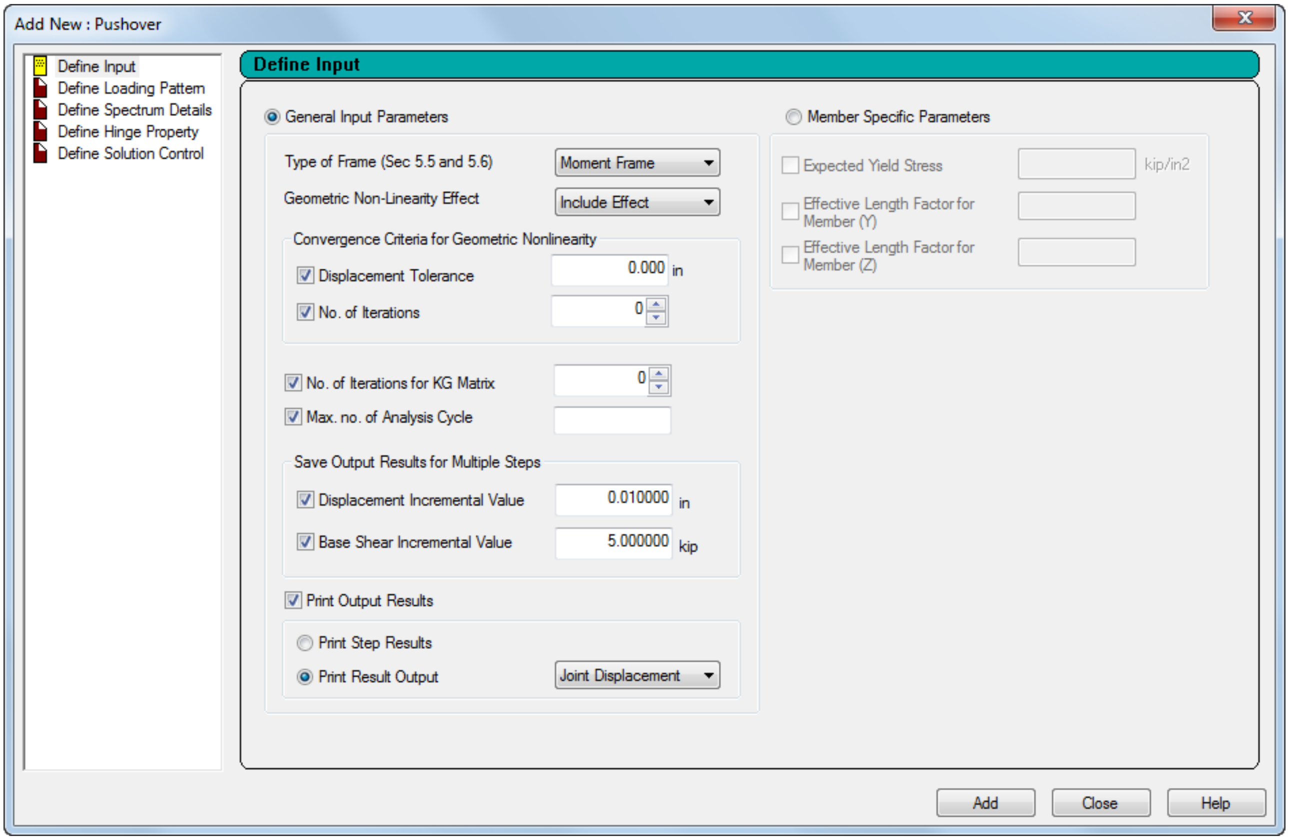


Figure 345: Add New : Pushover dialog Define Input tab

General Input Parameters

Select this control to add general parameters for the entire structure.

Type of Frame
Select either a fully restrained **Moment Frame** (FR) or a (Concentric) **Braced Frame** (CBF) to defined the type of steel frame.

Geometric Non-Linearity Effect
Select either **Ignore Effect**, if nonlinear effects are not to be considered, or **Include Effect**, if nonlinear effects are to be considered.

Displacement Tolerance
(For when Geometric Non-Linearity Effect is considered only) Set this option to specify a displacement tolerance for convergence, in current input units.

No. of Iterations
(For when Geometric Non-Linearity Effect is considered only) Set this option to specify a target number of iterations for geometric nonlinearity convergence.

No. of Iterations for KG Matrix
Set this option to specify the number of iterations to be performed for geometric stiffness, KG, matrix. The value is a integer, with a maximum value of 50.

Max. no. of Analysis Cycles
Set this option to specify a maximum number of cycles to be considered during the strain hardening stage for the load increment stage.

Displacement Incremental Value
Set this option to include intermediate displacement increments, in current units of length, at which the program will record analysis results for post-processing.

Base Shear Incremental Value	Set this option to include intermediate base shear increments, in current units of force, at which the program will record analysis results for post-processing.
Print Output Results	Set this option to print analysis results for the final state for a nonlinear static analysis. • Select the **Print Step Results** option to include the analysis results of the final load step in the output. • Select the **Print Results Output** option and then a results type to limit the output.

Member Specific Parameters

Select this control to add parameters for members which will be assigned later.

Expected Yield Stress	Expected yield strength of the material based on Section 5.3.2.3 and Tables 5-1 and 5-2 of FEMA 356.
Effective Length Factor for Member (Y/Z)	The effective length factor, k, defined for any member in either the local Y or local Z direction.

Related Links

- *M. To define general pushover data* (on page 881)
- *M. To define member specific pushover data* (on page 882)
- *M. To define member specific pushover data* (on page 882)
- *G.17.4.2.1.1 Define Steel Moment and Braced Frames* (on page 2178)
- *M. To define general pushover data* (on page 881)
- *M. To define general pushover data* (on page 881)
- *M. To define general pushover data* (on page 881)
- *M. To define general pushover data* (on page 881)
- *M. To define general pushover data* (on page 881)
- *M. To define general pushover data* (on page 881)

Define Loading Pattern tab

Used to define general definition and member specific parameters for a pushover analysis.

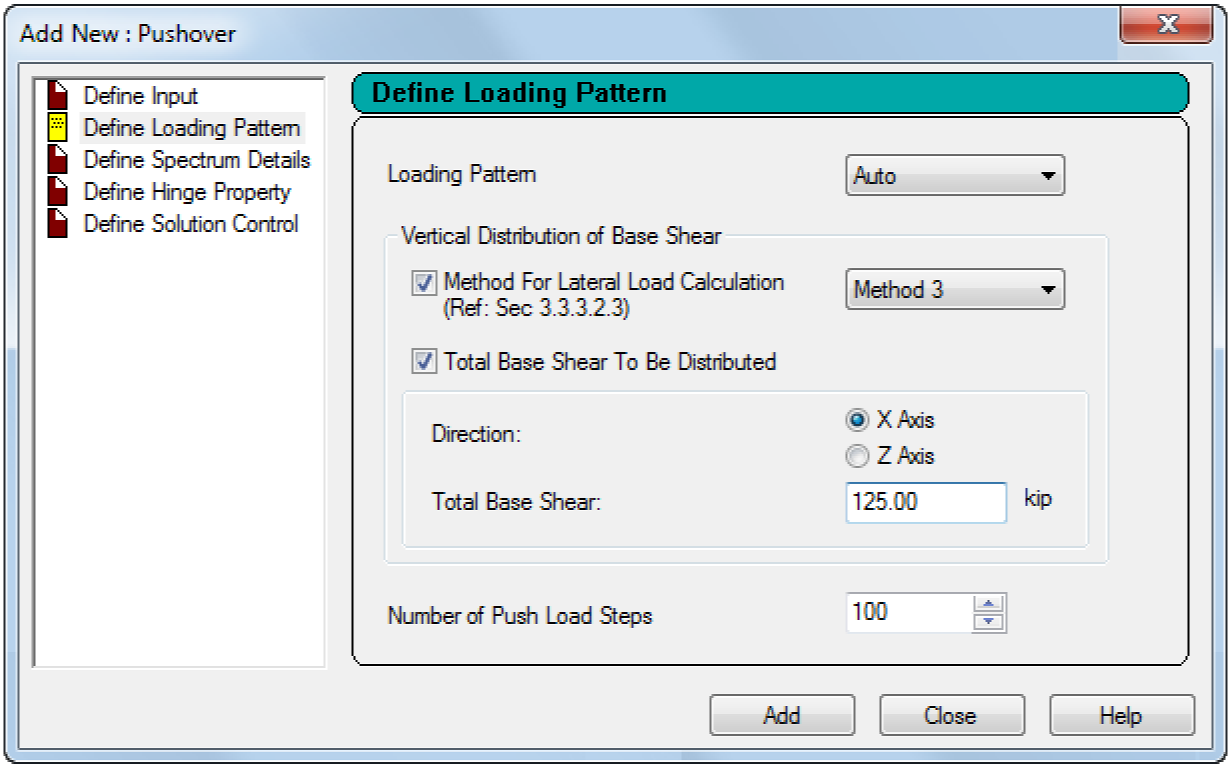


Figure 346: Add New : Pushover dialog Define Loading Pattern tab

Loading Pattern	Select either **Auto** or **User Defined** method for specifying loading patterns.
Method for Lateral Load Calculation	(Auto loading pattern only) Set this option to specify the G.17.4.1.8 Lateral Load Distribution (on page 2177) for defining the load pattern.
Total Base Shear To Be Distributed	(Auto loading pattern only) Set this option to define base shear that will be distributed vertically along the height of the structure at each floor level. If no value is specified, the program distributes 10 percent of gravity loading as lateral load.
Direction	(Total base shear distributed only) Select the global direction in which the pushover load acts.
Total Base Shear	(Total base shear distributed only) Base shear value in current units of force used for loading.
Number of Push Load Steps	The number of load steps. The lateral load at each floor divided by the number of load steps gives the push load increment at that floor level.

Related Links

- *M. To add a pushover loading* (on page 884)
- *M. To add a pushover loading* (on page 884)
- *M. To add a pushover loading* (on page 884)
- *G.17.4.2.1.3 Define Lateral (Push) Loading* (on page 2179)
- *G.17.4.1.8 Lateral Load Distribution* (on page 2177)
- *M. To add a pushover loading* (on page 884)

Define Spectrum Details tab

Used to define the response spectrum according to FEMA 356.

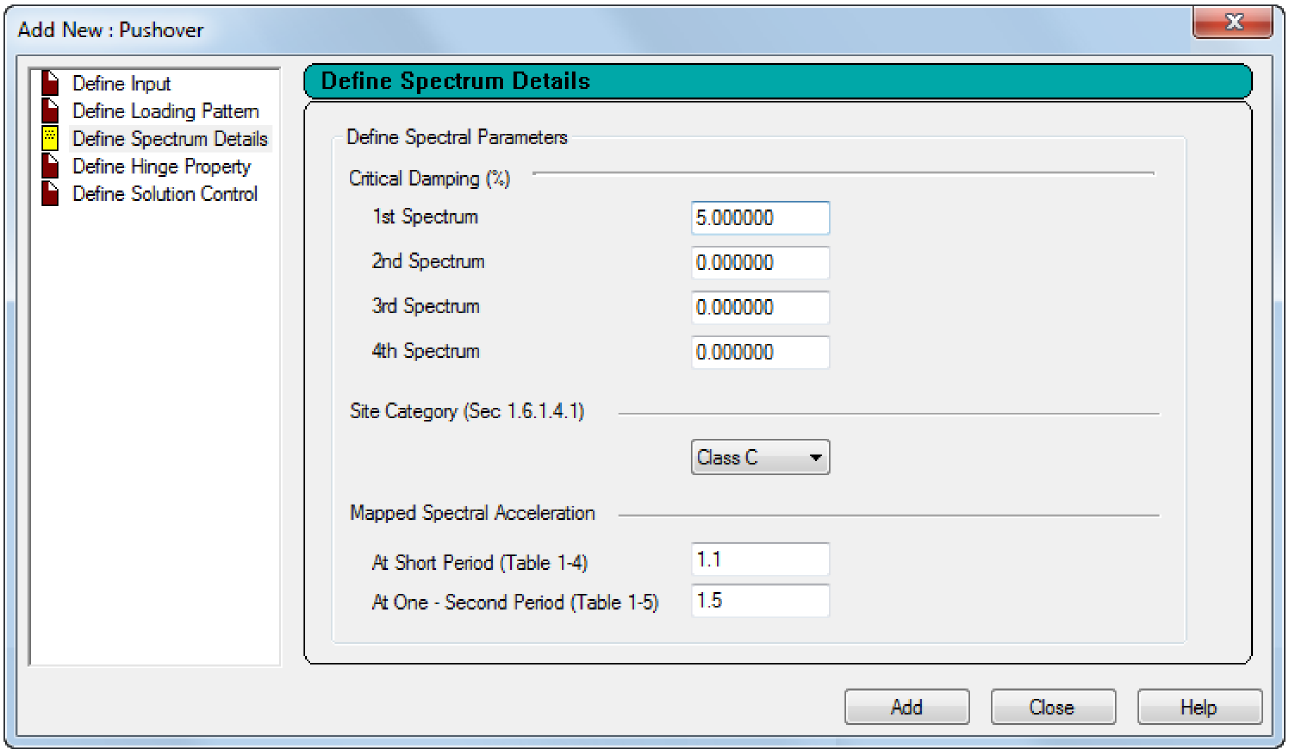


Figure 347: Add New : Pushover dialog Define Spectrum Details tab

Critical Damping (%) 1st | 2nd | 3rd | 4th Spectrum	Specify critical damping ratios to be used for the first (required, 0.05 is the default) through the fourth spectra.
Site Category	Specify the site category which describes the soil conditions.
At Short Period	Specify the mapped spectral acceleration at short period, S_s.
At One-Second Period	Specify the mapped spectral acceleration at one-second period, S_1.

Related Links

- *G.17.4.2.1.7 Define Input for Demand Spectrum* (on page 2182)
- *M. To define pushover spectral data* (on page 883)

Define Hinge Property tab

Used to define the hinge type used in the pushover analysis and create custom hinge definitions.

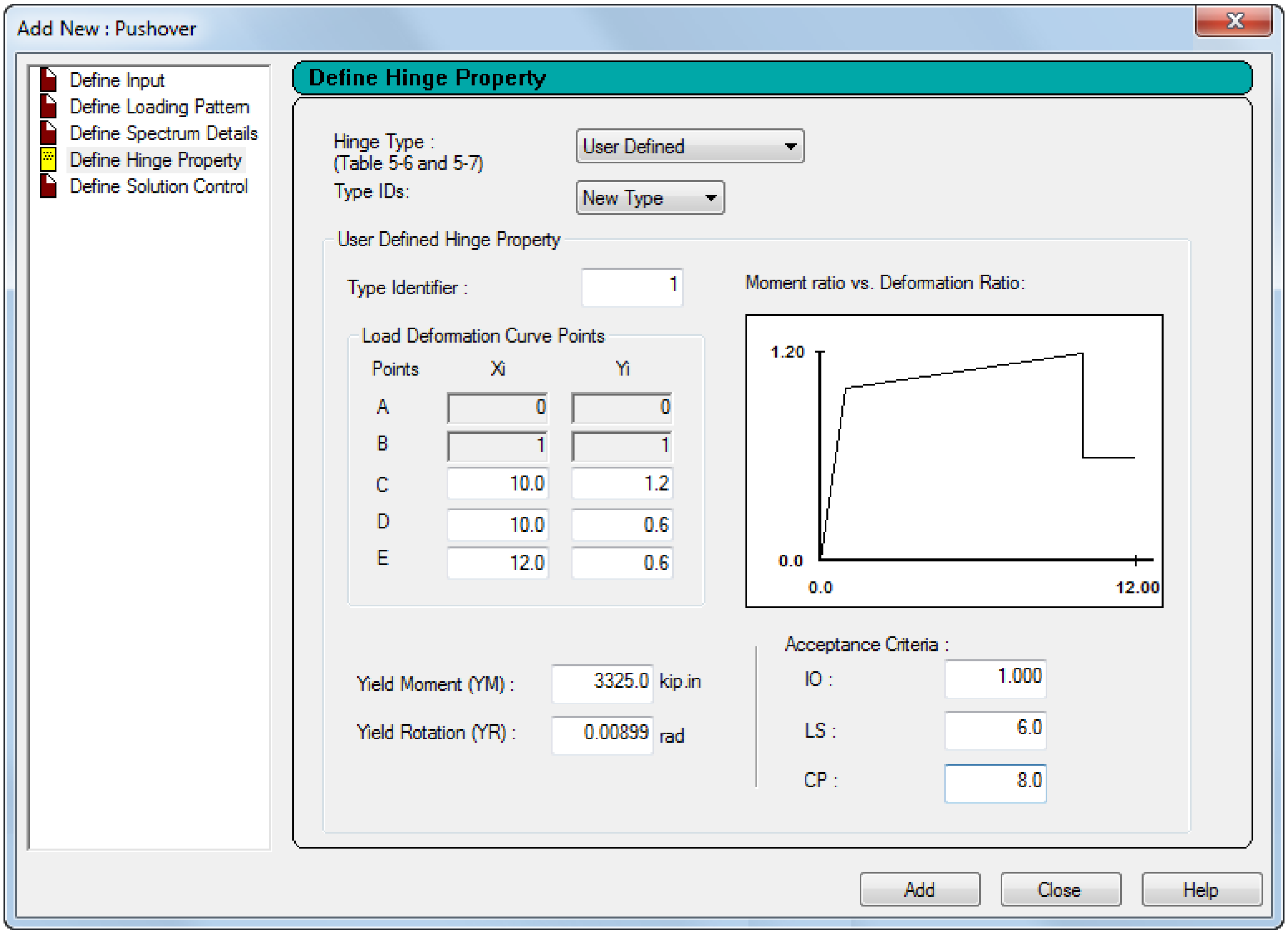


Figure 348: Add New : Pushover dialog Define Hinge Property tab

Hinge Type

Select either

- **FEMA** — instructs the program to use the built-in hinge definitions.
- **Ignore** — instructs the program to ignore hinge formation in some members.
- **User Defined** — used to define a custom hinge by **Load Deformation Curve Points**, **Yield Moment**, **Yield Rotation,** and **Acceptance Criteria**.

Type IDs

(User defined type only) Select an existing hinge type ID number of select **New Type** to define a new hinge type..

Type Identifier

(User defined type only) Specify an unique ID number for the hinge type (hinge property number).

Load Deformation Cure Points

Points on the load deformation curve at points A, B, C, D and E as specified in FEMA. The Xi and Yi coordinates of the load deformation curve represent the Deformation Ratio and Moment Ratio, respectively. The coordinates of A and B are preset at 0,0 and 1,1, respectively, to normalize the ratio curve.

Yield Moment (YM)

Yield moment in current force units.

Yield Rotation (YR)	Yield rotation in radians.
IO	Acceptance criteria limit (deformation ratio) for Immediate Occupancy level.
LS	Acceptance criteria limit (deformation ratio) for Life Safety level.
CP	Acceptance criteria limit for (deformation ratio) Collapse Prevention level.

Related Links

- *M. To manually define and assign hinges* (on page 882)
- *G.17.4.2.1.5 Define Pushover Hinges Properties and Acceptance Criteria* (on page 2180)
- *G.17.4.1.4 Types of Nonlinearity* (on page 2173)
- *G.17.4.1.6 Frame element hinge properties* (on page 2174)
- *M. To manually define and assign hinges* (on page 882)

Define Solution Control tab

Used to define the limit of the pushover analysis by either limiting base shear or displacement at a control joint.

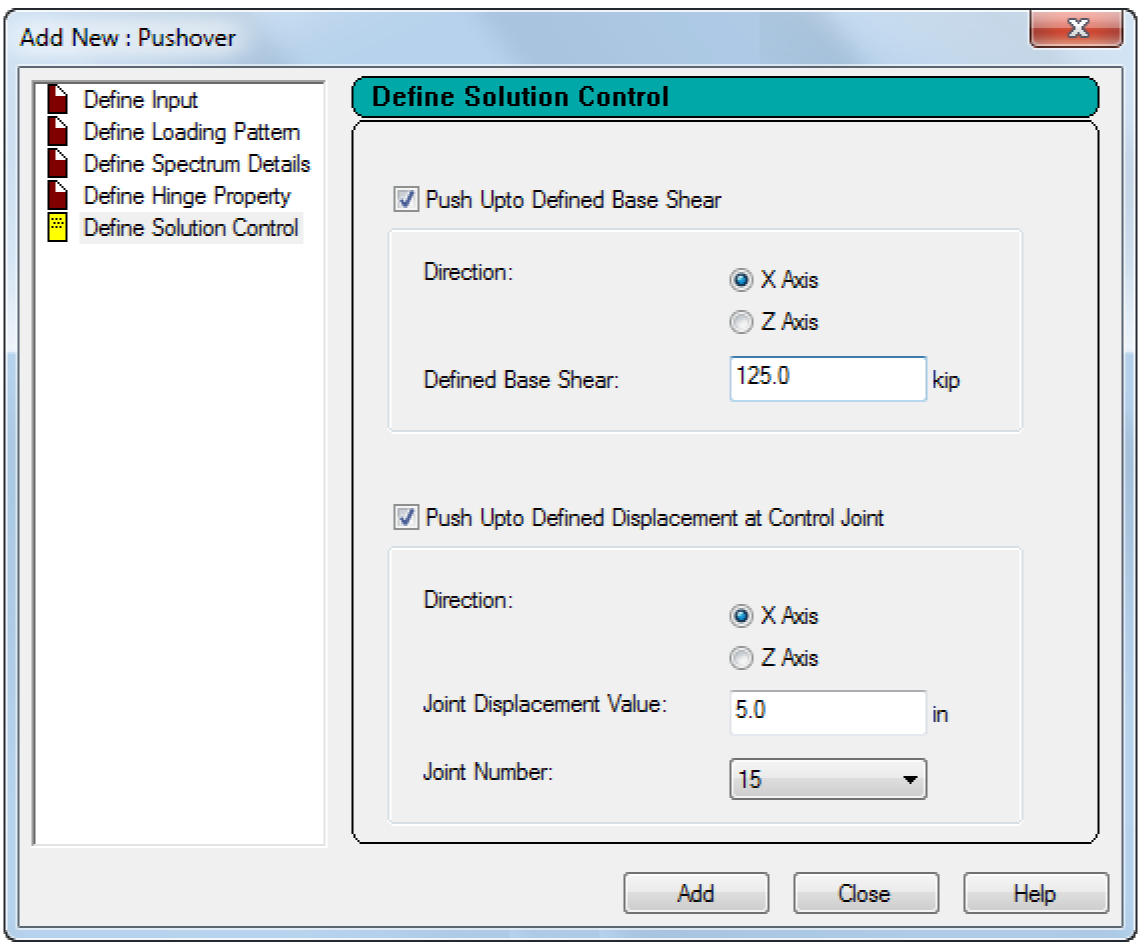


Figure 349: Add New : Pushover dialog Define Solution Control tab

Push Up to Defined Base Shear	Set this option to limit the analysis by force. The pushover analysis will continue until the cumulative base shear is less than or equal to the base shear specified by this command or the structure has additional strength.
Direction	Select the global direction of base shear for this solution control.
Defined Base Shear	Specify a target base shear value, in current units of force.
Push Up to Defined Displacement at Control Joint	Set this option to limit the analysis by displacement. The pushover analysis will continue until the displacement at the specified joint at the specified direction exceeds specified displacement.
Direction	Select the global direction of displacement for this solution control.
Joint Displacement Value	Specify a target displacement value, in current units of length.
Joint Number	Select the control joint from the list of nodes.

Related Links

- *G.17.4.2.1.6 Define Pushover Analysis Solution Control* (on page 2182)
- *M. To define solution control* (on page 884)

Add New Time History Definitions dialog

Used to define the Forcing Function of a time varying load.

Opens when the **Time History** > **Forcing Fucntions** tool is selected in the **Dynamic Specifications** group on the **Loading** ribbon tab.

Note: Refer to TR.31.4 Definition of Time History Load (on page 2441) for additional information.

Note: After defining a Time History Load, use the **Create New Load Items** dialog **Time History** tab to apply the load to the structure.

Integration Time Step	Solution time step used in the step-by-step integration of the uncoupled equations.
Consider Missing Mass Mode	Check this option to include the missing mass procedure in the time history analysis.
Type	This refers to the number of the type of functions.
Loading Type	Select the **Acceleration**, **Force**, or **Moment** option to define the type of function being input.
Save	Select this option to create an external file (input file name with a `.tmh` extension) containing the history of the displacements of every node of the structure at every time step.

The following function type options are available:

Select the *Define Time vs. Acceleration, Harmonic Function* or *From External File* option and provide function data.

Define Time vs. <Loading Type>

Used to specify a time history forcing function, where the loading type is that selected above (either Moment, Force, or Acceleration).

Specify the values Time and corresponding *Force* or *Acceleration*.

Note: The time history forcing function is plotted on the graph at the bottom of the dialog as data pairs are entered.

From External File

File Name	If the **From External File** option is selected, the **File Name** edit box becomes active. Type the name of the external file containing time history data. File name should be no more than 8 characters, reside in the same directory as the current input and have no file extension. The data must be provided as one time-force or time-acceleration pair per line. **Note:** For more information on creating an external time history file, see TR.31.4 Definition of Time History Load (on page 2441).

Harmonic Function

To specify a time history harmonic function, check the *Harmonic Function* button.

Curve Shape	Specify if the harmonic function is a **SINE** or **COSINE** curve.
Frequency or RPM	Choose *Frequency* and enter a circular frequency in cycles per second or *RPM* and enter revolutions per minute.
Amplitude	Max. Amplitude of the forcing function in current units.
Phase	Phase Angle in degrees.
Cycles	No. of cycles of loading.
Step or SubDiv	Choose the Step option to time step of loading or SubDiv to subdivide a ¼ cycle into this many integer time steps.

Spectrum

Select this Function Option to provide Spectrum parameters for your time history loading.

Tmax	The maximum time (in seconds) in the generated time history. Must be greater than **T3**.
DeltaT	Difference (delta) value in subsequent time steps in the generated time history.
Damp	Damping ratio (5% is entered as 0.05) associated with the input spectrum.
T1 / T2 / T3	Ending time of the acceleration rise, stead acceleration, and acceleration decay, respectively (in seconds).

> **Note:** The value of **T1** must be greater than zero. Further, **T3** > **T2** > **T1** and **Tmax** must be greater than **T3**.

Random Seed Enter a positive integer (1 to 2,147,483,647) to be used as a unique random number generation "seed." A unique time history will be produced for each seed value. Change this value when you want to produce a "different (from the time history generated with the prior seed value)" but statistically equivalent time history.

No of digitized Freq. The input shock spectrum will be re-digitized at this number of equally spaced frequencies by interpolation. The default is taken as the greater of 35 or the number of points in the input spectrum.

No. of Iteration The number of iterations which will be used to perfect the computed time history.

Related Links

- *G.17.3.5 Response Time History* (on page 2165)
- *M. To define a time history type from tabular data* (on page 875)
- *M. To define a time history type from a function* (on page 876)
- *M. To define a time history type by spectrum* (on page 877)
- *M. To define a time history type by external file* (on page 879)

Define (Time History) Parameters dialog

Used to define time step, damping, and arrival times for time history loads.

Opens when the **Time History** > **Parameters** tool is selected in the **Dynamic Specifications** group on the **Loading** ribbon tab.

Time Step Specify a solution time step to be used in the step-by-step integration of the uncoupled equations. The default value of *Time Step* is determined as explained in TR.31.4 Definition of Time History Load (on page 2441) .

Damping The following options are available for specifying damping:

- **Damping** – This is to be used for specifying a single modal damping ratio which will be applied to all modes. The default value is 0.05.
- **CDAMP -** If a damping ratio has already been specified under `CONSTANTS` based on the type of material in the structure, that value may be used directly in the time history analysis. Check this option for that purpose.
- **MDAMP -** In we wish to utilize individual damping ratios for individual modes, that is achieved through the means of the MDAMP option. The first step to doing this is the specification of those individual damping ratios, as explained under TR.26.4 Modal Damping Information (on page 2312) , and is done graphically by using the **Modal Damping** tool on the **Loading** ribbon tab. If this first step has been completed, the instruction to utilize `MDAMP` is done by selecting this option shown above.

Arrival Time
Specify values of possible arrival times of the various dynamic load types. The arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load may have different arrival times for different joints and hence all these values must be specified here. The arrival times and time-force pairs for the load types are used to create the load vector needed for each time step of the analysis.

Add
Adds the specified load details to the currently selected load case in the Load dialog and closes the dialog.

Close
Closes the dialog without adding any load data.

Help
Opens the STAAD.Pro Help window.

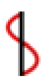
Related Links

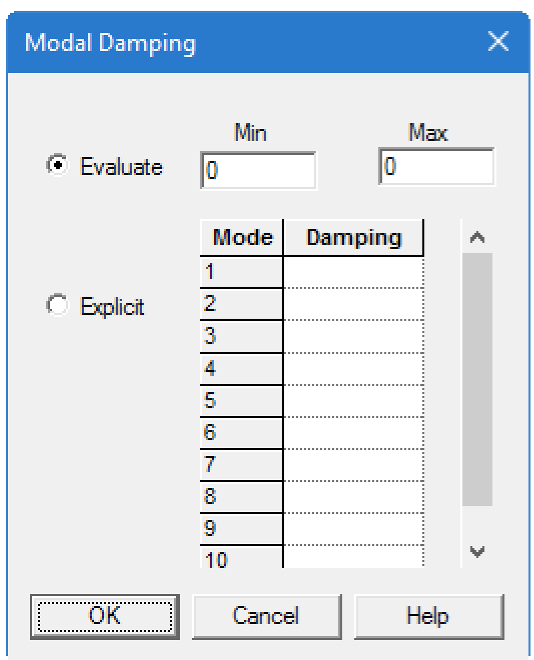
- *G.17.3.5 Response Time History* (on page 2165)
- *M. To define time history parameters* (on page 880)

Modal Damping dialog

Used to define unique damping ratios for the individual modes used in a dynamic analysis. It is used in response spectrum and time history analysis.

Opens when the **Modal Damping** tool is selected in the **Dynamic Specifications** group on the **Loading** ribbon tab.





Note: STAAD.Pro is also capable of calculating each mode's damping based on the frequency of the mode, the mass-proportional damping coefficient (α), and the stiffness-proportional damping coefficient (β). This input must be provided using the STAAD.Pro Editor. Refer to Calculate Damping (on page 2160) for additional details.

Evaluate This option is to instruct STAAD.Pro to calculate the damping ratio for each mode based on the frequency of the mode and the **Min**imum and **Max**imum values entered in the respective fields.

The formula used to evaluate the damping ratio is explained in Evaluate Damping (on page 2162).

Explicit Provide values of damping ratio for the corresponding mode number.

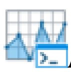
Refer to Explicit Damping (on page 2160) for additional details.

Related Links

- *G.17.3.3.2 Modal Damping* (on page 2160)
- *M. To explicitly define damping values for modes* (on page 895)
- *M. To evaluate damping for modes* (on page 896)

Analysis and Design tab

Table 327: Analysis Data group

Tool name		Description
 **Analysis Commands**		Opens the **Analysis/Print Commands** dialog for analysis commands, which is used to define the analysis commands to be included in the Input Command File.
 **Pre Analysis Commands**		Opens the **Analysis/Print Commands** dialog for pre-analysis print commands, which is used to define the pre-analysis print commands to be included in the Input Command File.
 **Post Analysis Commands**		Opens the **Analysis/Print Commands** dialog for post-analysis print commands, which is used define the post-analysis print commands to be included in the Input Command File.
 **Miscellaneous**	**Input Width**	Opens the **Input Width** dialog, which is used to set the maximum allowable number of characters per line in the input command file for processing. Choose 72 or 79 columns and then click **OK**. Refer to TR.4 Input/Output Width Specification (on page 2205) for additional information.

Tool name		Description
Commands >	**Output Width**	Opens the **Output Width** dialog, which is used to set the maximum allowable number of characters per line in the output file. Choose 72 or 118 columns and then click **OK**. **Note:** If you choose 118 Columns, your printer must have the capability of supporting that wide of an output. Refer to TR.4 Input/Output Width Specification (on page 2205) for additional information.
	Floor Diaphragm Options	Used for analysis of a seismic requirements for IS 1893 (Indian) or ASCE 7 (US) codes. Refer to TR.28.2.1 Soft Story Checking (on page 2331) for additional information.
	Set NL	Opens the **Set NL** dialog, which is used to specify the maximum number of primary load cases for processing. It is required if multiple cycles of analysis is performed. Type the **Maximum Number of Primary Loads** to provide in the input file and then click **OK.** Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set Echo	Opens the **Set Echo** dialog, which is used to activate or deactivate the echoing of input commands to the output (`.ANL`) file. By default, Echo is On (i.e., input commands are echoed to the output file). This option is used if you want to process any structure in which a node may be associated with more than 16 other joints. Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set Z Up	By default, STAAD.Pro assumes that the global X and Z axes are horizontal and the Y axis is vertical. This command is used to orient the global Z axis as the vertical axis and sets X and Y to be the horizontal axes. Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set Displacement	Opens the **Maximum allowable displacement tolerance** dialog, which is used to specify limiting values for nodal displacements for `PDELTA` and `NONLINEAR` analyses. **Tip:** Make sure it is in the current length unit (see the bottom right hand corner of the STAAD.Pro window for the current input units). Refer to TR.5 Set Command Specification (on page 2206) for additional information.

Tool name		Description
	Set Floor Load Tolerance	Opens the **Tolerance used in Floor load calculation for length** dialog, which is used to specify the tolerance for out of plane nodes in a floor load. The value is given as a length, in current units.
	Set Floor Angle Tolerance	Opens the **Tolerance used in Floor load calculation for angle** dialog, which is used to specify the he tolerance for out of plane nodes in a floor load. The value is given in degrees.
	Set SDAMP	Opens the **Damping ratio to be used for all springs** dialog, which allows damping property of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set Warp	Opens the **Warping restraint ratio to compute torsional rigidity** dialog, which allows end warping restraint to be considered for I-shaped members. Refer to TR.5 Set Command Specification (on page 2206) for additional information. Assign a value which in the range 0 to 1. 0 indicates no restraint. 1 indicates full restraint. A fraction indicates a partial restraint.
	Set Shear	This command is used for omitting the additional pure shear distortion stiffness terms in forming beam member stiffnesses. With this command you can exactly match simple textbook beam theory results. A message dialog opens to confirm you wish to omit the additional shear distortion terms. Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set ITERLIM	Opens the Maximum number of tension/compression iterations dialog, which is used to override default limits for the maximum number of iterations of analysis performed for cases involving tension-only and compression-only components. Refer to TR.5 Set Command Specification (on page 2206) for additional information.
	Set NoWarning	This command will toggle off all warning messages in the output file. A message dialog opens to confirm you wish to toggle off all warning messages. Refer to TR.5 Set Command Specification (on page 2206) for additional information.

<table>
<tr><th colspan="2">Tool name</th><th>Description</th></tr>
<tr><td></td><td>Set Eigen Method</td><td>Opens the Set Eigen Method dialog, which is used to specify the method used for eigen vectors or to use load-dependent Ritz vectors instead.

Refer to TR.5 Set Command Specification (on page 2206) for additional information.

Refer to G.17.3.1 Solution of the Eigenproblem (on page 2155) for details on load-dependent Ritz vectors.</td></tr>
<tr><td></td><td>Cut Off Mode Shape</td><td>Opens the Cut Off Mode Shape dialog, which is used to instruct the program to extract a higher or lower number of modes. By default, STAAD.Pro calculates only a specific number of modes during modal calculation, response spectrum analysis, and time history analysis. This defaults to the first six modes.

<table><tr><th>Option</th><th>Description</th></tr><tr><td>Allow Automatic Shifting</td><td>When the Arnoldi/Lanczos method is selected in the Set Eigen Method dialog, this option can be set to reduce memory demand. This allows for efficient solution of large models with a large number of degrees of freedom.</td></tr><tr><td>Modes Per Shift</td><td>If the Allow Automatic Shifting option is set, then you must specify a targeted number of eigen modes to be searched in each shift.</td></tr><tr><td>Initial Shift Frequency</td><td>When the Arnoldi/Lanczos method is selected in the Set Eigen Method dialog, you may specify an initial frequency shift value.</td></tr></table>
Provide the maximum number of mode shapes we want the program to extract, and click OK.

Refer to TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) for additional information.</td></tr>
<tr><td></td><td>Cut Off Frequency</td><td>Opens the Cut Off Frequency dialog, which is used to over-ride the built in cut off limit. During modal calculation, response spectrum analysis, and time history analysis, STAAD.Pro will extract only those modes whose frequency is below a built-in cut-off level.

Refer to TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) for additional information.</td></tr>
<tr><td></td><td>Cut Off Time</td><td>Opens the Cut Off Time dialog, which is used to compute the results for a longer period of time than that at which the loads stop acting, specify the value using the CUT OFF TIME command. During time history analysis, by default, STAAD.Pro calculates displacements, forces, reactions and stresses for only up to a time which is equal to the duration of the longest acting forcing function or ground motion.

Refer to TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) for additional information.</td></tr>
</table>

Tool name		Description
	Clear Above Commands	Opens the **Delete Miscellaneous Commands** dialog, which is to the delete previously specified Miscellaneous commands such as Input Width, Set CO, etc. Check the boxes associated with the commands you want to delete and then click **Delete.**
Load List		Opens the **Load List** dialog, which is used to specify a list of existing load cases and load combination cases to be used for subsequent processes, like design, print, plot etc.

Table 328: Analysis group

Tool name	Description	Shortcut
Run Analysis	Performs the STAAD analysis as directed by input commands. The **STAAD Analysis and Design** window opens to display the progress of the design engine. When completed, you can select a next process and then click **Done.**	**<Ctrl+F5>**

Table 329: Cloud Analysis Services group

Tool name		Description
Run Cloud Analysis		Opens the **Run Cloud Analysis** dialog, which is used to select a solution, name your scenario, and submit for analysis via Bentley's Scenario Services. **Note:** Requires you to be logged into your valid CONNECT account.
Download Results	**Download & Load Results**	Select to download and open the results in the typical postprocessing workflow.
	Download Results To	Select to save the output from a cloud analysis to your local computer.

Table 330: Design Commands group

Tool name	Description
Steel Design	Opens the **Steel Design - Whole Structure** dialog, which is used to select a steel design code, select parameters to use for design, and assign parameters to the model.

Tool name	Description
Concrete Design	Opens the **Concrete Design - Whole Structure** dialog, which is used to select a concrete design code, select parameters to use for design, and assign parameters to the model.
Aluminium Design	Opens the **Aluminum Design - Whole Structure** dialog, which is used to select a aluminum design code, select parameters to use for design, and assign parameters to the model.
Timber Design	Opens the **Timber Design - Whole Structure** dialog, which is used to select a timber design code, select parameters to use for design, and assign parameters to the model.

Analysis - Whole Structure dialog

Used to view, add, and assign analysis commands to the STAAD input file.

Opens when the **Analysis** page is selected.

Command Tree The dialog box shows a tree view (called Command Tree) of the commands in the Input Command File. The Command Tree is a pictorial view of the Input Command File with the commands displayed as "Nodes" of the tree in the same sequence in which they appear in the Input Command File. Click on a command Node to expand the branch of the command and the command parameters appear as the "leaves" under that command node. The following paragraph describes the conventions followed in the Command Tree.

- Green Check ( ) marks: Commands which can be re-specified or modified.
- Gray Check ( ) marks: Commands which may not be changed.
- Question ( ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Define Commands... Opens the Analysis/Print Commands (on page 2916), which is used to define the analysis commands to be included in the Input Command File.

Close Closes the dialog.

Help Opens the STAAD.Pro help window.

Analysis/Print Commands

Used to define the analysis commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when the **Analysis Commands** tool is selected from the **Analysis Data** group on the **Analysis and Design** ribbon tab.

After current Select this option to add the analysis statement being defined in the Command Tree after the currently selected (highlighted) command.

Add Assigns the selected print command to the input command file.

Assign	Used to assign the command to the current selection set.
Cancel	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Perform Analysis tab

Used to direct STAAD.Pro to perform a linear elastic analysis.

Refer to TR.37.1 Linear Elastic Analysis (on page 2620) for additional information.

The dialog contains several print options, of which one may be selected.

No Print	No analysis results will be printed to the output file.
Load Data	Prints all the load data.
Statics Check	Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
Statics Load	Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output. **Note:** Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.
Mode Shapes	Prints mode shape values at the joints for all calculated mode shapes.
Both	This option is equivalent to the **Load Data** plus **Statics Check** options.
All	This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *A. To specify a linear elastic analysis* (on page 952)

P Delta Analysis tab

Used to direct STAAD.Pro to perform a second order analysis.

Refer to TR.37.2 P-Delta Analysis Options (on page 2621) for additional information.

Number of Iterations	Specify the number of iterations to be performed. If specified, the program will perform the specified number of iterations whether or not the solution converges.
Use Geometric Stiffness (Kg)	P-Delta analysis including stress stiffening effect of the KG matrix. With this option the global stiffness is adjusted with every iteration. A regular STAAD.Pro P-Delta Analysis performs a first order linear analysis and obtains a set of joint forces from member/plates based on the large P-Delta effect. These forces are added to the original load vector. A second analysis is then performed on this updated load vector (5 to 10 iterations will usually be sufficient). In the new P-Delta KG Analysis, that is, with the Kg option selected, the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is

then performed using the original load vector. Large & small P-Delta effects are always included (1 or 2 iterations will usually be sufficient).

Small Delta Select this option to include both P-small and large Delta effects (recommended), sometimes referred to as P-δ & P-Δ. Leave this box uncheck to only include the "PlargeDelta" effects (P-Δ only).

Without the **Small Delta** option (i.e., a default STAAD.Pro P-Delta analysis), STAAD.Pro performs a first order linear analysis and obtains a set of joint forces, from members/plates based on the large P-Delta effect, which are then added to the original load vector. A second analysis is then performed on this updated load vector.

With the **Small Delta** option selected, both the large & small P-Delta effects are included in calculating the end forces, (5 to 10 iterations will usually be sufficient).

Print Option Standard STAAD.Pro analysis print options:

No Print No analysis results will be printed to the output file.

Load Data Prints all the load data.

Statics Check Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

Statics Load Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

Note: Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

Mode Shapes Prints mode shape values at the joints for all calculated mode shapes.

Both This option is equivalent to the **Load Data** plus **Statics Check** options.

All This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *G.17.2.1 P-Delta Analysis* (on page 2141)
- *A. To specify a P-Delta analysis* (on page 952)

Perform Cable Analysis tab

Used to assign the commands required to perform a non-linear cable analysis on the model. This requires the presence of non-linear cables in the structure.

Note:

Refer to TR.37.3 Nonlinear Cable Analysis (on page 2624) for additional information.

Refer to G.8.2 Cable Members (on page 2122) for an explanation of non-linear cable input options.

Advanced Cable Analysis group

The options in this group are used to specify the advanced cable analysis feature and options specific to that analysis type.

Advanced Cable Analysis
Set this option to specify an advanced cable analysis. Refer to G.17.2.8 Advanced Nonlinear Cable Analysis (on page 2150) for details on the Advanced Cable analysis.

> **Note:** Use of the Advanced Cable analysis feature requires the Advanced Analysis License. This option is the default if you have the license.

Use Modified Newton-Raphson
Set this option to use the modified Newton-Raphson method, which can reduce computation effort in compared to the full Newton-Raphson method (at the expense of more iterations).

Use Geometric Matrix (Kg)
Set this option to specify the use of geometric stiffness.

Cable Analysis Parameters group

Steps
Specify the number of load steps. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. Should be in the range 5 to 145, with 145 as the recommended number.

Eq-Iterations
Maximum number of iterations permitted in each load step. Should be in the range of 10 to 500, with 300 as the recommended number.

Eq-tolerance
The convergence tolerance for the above iterations.

Sag Minimum
Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction). As soon as **Sag Minimum** becomes less than 0.95 the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The **Eq-Iterations** may need to be 300 or more. The **Eq-tolerance** may need to be greater or smaller.

Stability Stiffness
Specify a stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first number of **Load Steps**. The amount added linearly decreases with each of the Load Steps. If a Stability Stiffness value is specified, it should be within the range of 0.0 to 1000.0. This parameter alters the stiffness of the structure.

Load Steps
The number of load steps over which the **Stability Stiffness** is gradually applied. Default is one step.

KSMALL
A stiffness matrix value that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. The range for KSMALL is between 0.0 and 1.0, with a default of 0.0. This parameter alters the stiffness of the structure.

Print Option

Standard STAAD.Pro analysis print options:

No Print
No analysis results will be printed to the output file.

Load Data
Prints all the load data.

Statics Check	Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
Statics Load	Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output. **Note:** Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.
Mode Shapes	Prints mode shape values at the joints for all calculated mode shapes.
Both	This option is equivalent to the **Load Data** plus **Statics Check** options.
All	This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *A. To specify a nonlinear cable analysis* (on page 955)
- *A. To specify a nonlinear cable analysis* (on page 955)

Change tab

Used to add the Change command to the input command file.

Related Links

- *G.19 Multiple Analyses* (on page 2194)
- *A. To add a change command* (on page 958)

Perform Direct Analysis tab

Used to instruct STAAD.Pro to perform a direct analysis per AISC 360 Appendix 7.This method accounts for the second order effects resulting from deformation in the structure due to applied loading, imperfections and reduced bending stiffness of members due to the presence of axial load.

This is a non-linear iterative analysis as the stiffness of the members is dependent upon the forces generated by the load. The analysis will iterate, in each step changing the member characteristics until the maximum change in any τ_b is less than the **Tau** tolerance. If the maximum change in any τ_b is less than 100x the **Tau** tolerance *and* the maximum change in any displacement degree of freedom is less than the **Displacement** tolerance; then the solution has converged for this case.

This analysis method is to be used in conjunction with Direct Analysis definition command. The Define Direct command should be added prior to instructing the program to perform this type of analysis.

Additionally, a direct analysis typically requires the definition of Notional Load (on page 2864).

Note: Refer to TR.37.5 Direct Analysis (on page 2630) and AD.2007-03.2.2 Direct Analysis (on page 263) for additional information.

Option	Select if the design methodology is Load Resistance and Factor (LRFD) or Allowable Stress (ASD).

Tolerances	**Tau**	The limit to the change in the τ_b value in each iteration. If this value is exceeded by any member, then an additional iteration will be required unless the maximum number of iterations has been achieved
	Displacement	The limit to the change in the maximum displacement in each iteration. If this has been exceeded, then an additional iteration will be required unless the maximum number of iterations has been achieved.

No. of Iterations	Specify the maximum number of iterations.
PDelta Iterations	Select the number of iterations to be used for a PDelta with Small Delta analysis. Typically this should be from 5 to 25. The default number of 15 is recommended.
Reduced EI	Check this option to use the reduced EI (Tau-b × 0.8 ×EI) for member section moment & section displacement. Clear the check box to use the full EI value.
Perform Tau-b Iteration	Check this box to instruct the program to iterate Tau-b.
Print Option	Standard STAAD.Pro analysis print options:

No Print	No analysis results will be printed to the output file.
Load Data	Prints all the load data.
Statics Check	Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
Statics Load	Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

Note: Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

Mode Shapes	Prints mode shape values at the joints for all calculated mode shapes.
Both	This option is equivalent to the **Load Data** plus **Statics Check** options.
All	This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)
- *A. To specify a direct analysis* (on page 953)

Generate Floor Spectrum tab

Used to specify the calculation of floor and/or joint response spectrum when subjected to a time history loading. This information can be used in conjunction with equipment that will be supported by these floors and is often required by the equipment manufacturers.

Note: This feature requires STAAD.Pro V8i (release 20.07.04) or higher and an Advanced Analysis Package license.

The Floor Response Spectrum command must immediately follow an analysis command. That analysis can only contain a single time history load case. The Create Group dialog (on page 2950) used to define a floor must be defined prior to adding this command. Results from this analysis will be displayed in the **Floor Spectrum** table in the Postprocessing workflow.

Note: Refer to TR.37.10 Floor Spectrum Command (on page 2657) for additional information.

Floor Groups table	Each row constitutes one floor which can have one or more floor groups assigned. • **GX / GY / GZ -** Select the global direction options for which acceleration vs frequency spectrums will be generated for this floor. • **Title -** Provide an optional title for this floor. • **Node Groups -** Click the Selected Nodes to open a drop-down list of all previously defined node groups. The resulting response spectra will be based on the collective responses of all the nodes in the selected groups.
Lowest Frequency	Lowest frequency to be in the calculated spectrum. This value should be at least 0.01 Hz.
Highest Frequency	Highest frequency to be in the calculated spectrum.
Frequency Interval	The change in frequency values used for each interval calculation. The spectrum will be calculated at this interval from the **Lowest Frequency** to **Highest Frequency** specified.
Damping Ratios table	Up to ten damping values may be entered. One spectrum will be generated for each damping value for each global direction requested for each floor defined. The spectrum will be based on these modal damping ratios. Damping is specified as a fraction of one (that is, 3% damping should be entered as 0.03). The default is 0.05.
Relative Acceleration?	Select this option if there is ground motion defined and you want the spectrums to be based on the relative acceleration of the floor to the ground acceleration. Otherwise, absolute values are considered.
Print Time History Acceleration used?	Select this option to print the time history acceleration being used in each spectrum calculation.
Print Calculated Spectrum?	Select this option to print the calculated spectrum.

Related Links

- *A. To generate a floor spectrum* (on page 959)

Nonlinear Analysis tab

Used to direct STAAD.Pro to perform a nonlinear analysis which can account for nonlinear effects of moderate displacement and small strain. This solution holds for where the element distortion is small and small rotations are assumed.

Note: This requires a STAAD.Pro Advanced license.

Refer to TR.37.8 Geometric Nonlinear Analysis (on page 2654) for additional information.

The first analysis step must be stable, otherwise use ARC control to prevent instability. The procedure does not use follower loads. Loads are evaluated at the joints before the first step; then those loads translate with the joint but do not rotate with the joint. Equilibrium is computed in the displaced position.

Nonlinear entities such as tension/compression members, multilinear springs, gaps, etc. are not supported when using a nonlinear analysis. Temperature loads are *not* supported for geometric nonlinear analysis. Additionally, nonlinear analysis does not account for post-buckling stiffness of members.

Note: The older `NONLINEAR nn ANALYSIS` command will adopt the new procedure unless a `SET` command is used. If a `SET` command is entered this will invoke the old procedure for backward compatibility.

The values for the Displacement Limit are used to limit analysis cycle to a specified displacement value and degree of freedom for a monitored node. When two or more **Load Steps** are specified, the calculated displacement or rotation is compared to the **Target Value** along or about the **DOF** for the specified **Node** each step. If the calculated displacement or rotation meets or exceeds the target value, then the analysis is stopped.

ARC
Specify a displacement control, which is the value of the absolute displacement limit for the first analysis step. If the maximum displacement is greater than this limit, ARC will calculate a new step size for the first step and a new value for **Load Step**. Value should be in current length units.

Note: ARC = 0 indicates no displacement control.

Iterations
Max. Number of iterations to achieve equilibrium in the deformed position to the tolerance specified.

Tolerance
For convergence, two successive iteration results must have all displacements the same within this tolerance. Value entered is in current units.

Load Steps
Specify the number of load steps to be used. Load is applied in stages if entered. One means that all of the load is applied in the first step.

Rebuild Steps
Frequency of rebuilds of the tangent K matrix per load step & iteration.

0 = once per load step
1 = every load step & iteration

Use KG
Select this option to add the geometric stiffness, KG, to the stiffness matrix, K.

Node
Enter a node number or click **[...]**. to enter the selected node to monitor during the geometric nonlinear analysis.

DOF
Select the degree of freedom to monitor during the analysis for specified **Node**.

Target Value
Specify a target displacement value along or about the selected **DOF** for the monitored **Node**.

Print Option
Standard STAAD.Pro analysis print options:

No Print
No analysis results will be printed to the output file.

Load Data
Prints all the load data.

Statics Check
Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

Statics Load	Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output. **Note:** Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.
Mode Shapes	Prints mode shape values at the joints for all calculated mode shapes.
Both	This option is equivalent to the **Load Data** plus **Statics Check** options.
All	This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *G.17.2.3 Static Geometrically Nonlinear Analysis* (on page 2147)
- *A. To specify a nonlinear analysis* (on page 954)
- *A. To specify a nonlinear analysis* (on page 954)

Perform Imperfection Analysis tab

Used to instruct STAAD.Pro to perform an imperfection analysis, which will reflect the secondary effects only if the camber and/or drift is specified in a using the Define Imperfection command. For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

Refer to sections TR.26.6 Member Imperfection Information (on page 2313) and TR.37.9 Imperfection Analysis (on page 2657) for additional details.

There are no other parameters associated with this command.

Print Option Standard STAAD.Pro analysis print options:

No Print	No analysis results will be printed to the output file.
Load Data	Prints all the load data.
Statics Check	Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
Statics Load	Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output. **Note:** Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.
Mode Shapes	Prints mode shape values at the joints for all calculated mode shapes.
Both	This option is equivalent to the **Load Data** plus **Statics Check** options.
All	This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *G.17.2.4 Imperfection Analysis* (on page 2148)
- *A. To specify an imperfection analysis* (on page 956)
- *M. To assign member imperfection for members* (on page 806)

Perform Buckling Analysis tab

Used to instruct STAAD.Pro to perform a buckling analysis.

Refer to TR.37.4 Buckling Analysis (on page 2628) for additional information.

Iterative method: When this option is selected, the program will perform a P-Delta analysis including Kg Stiffening (geometric stiffness of members & plates) due to large & small P-Delta effects. If a non-singular stiffness matrix can be created, then buckling has not occurred. Then the load is increased from the last increment repeatedly until buckling does occur. Then the load is decreased halfway back to the prior increment. This bounds the buckling factor between the last 2 increments. Then STAAD.Pro proceeds to halve the interval until either the change between increments is 0.1% of each other, or the specified number of increments has been exceeded. The resulting factor is reported in the output file. The buckling deformed shape is simply the deformed shape from a static analysis with the near buckling load applied. This could appear more like a crushing, small displacement shape rather than a buckling mode shape. At least 15 iterations are recommended. Buckling will be applied to all primary cases.

- **Number of Iterations**: Maximum number of iterations. Fifteen is recommended.

  Note: This parameter is only used for the iterative method. It is ignored for the eigen method.

Eigen method: This buckling method may only be used if an Advanced Analysis license is available. When using the Advanced Solver, the corresponding “buckling modes” are included in the output file.

Print Option: Standard STAAD.Pro analysis print options:

- **No Print**: No analysis results will be printed to the output file.
- **Load Data**: Prints all the load data.
- **Statics Check**: Provides a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.
- **Statics Load**: Prints everything that **Statics Check** does, plus a summation of all internal and external forces at each joint. This option potentially generates a large volume of output.

  Note: Since `PRINT STATIC LOAD` generates voluminous output, the printing of summation of internal and external forces at each joint is done for structures which have less than 1,000 joints. If the structure has 1,000 joints or more, this printing will be skipped.

- **Mode Shapes**: Prints mode shape values at the joints for all calculated mode shapes.
- **Both**: This option is equivalent to the **Load Data** plus **Statics Check** options.
- **All**: This option is equivalent to the **Load Data** plus **Statics Check**.

Related Links

- *G.17.2.2 Buckling Analysis* (on page 2145)
- *A. To specify buckling analysis* (on page 957)

Perform Pushover Analysis tab

Used to instruct the program to perform a pushover analysis. This command has no other parameters.

Related Links

- *TR.37.7 Pushover Analysis* (on page 2643)
- *G.17.4 Pushover Analysis* (on page 2168)
- *M. Pushover Loads* (on page 881)
- *A. To specify a pushover analysis* (on page 957)

Analysis/Print Commands dialog (Pre Print)

Used to define the pre-analysis print commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when:

- The **Pre Analysis Commands** tool is selected from the **Analysis Data** group on the **Analysis and Design** ribbon tab, or
- **Define Commands** is clicked on the **Pre Analysis Print** dialog

Problem Statistics tab

This command was previously used to include structure information such as total number of joints, members, supports, disk space requirements, the estimated band-width of the stiffness matrix, etc.

Note: This command has been deprecated and will be ignored by the engine during analysis.

Joint Coordinates tab

Used to add the joint coordinate values.

Member Information tab

Used to add member length, member incidences, beta angles, and member specifications such as truss member, and the member release conditions at start and end of the member (i.e., 1 = released, 0 = not released).

Material Properties tab

Used to add material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members.

Note: This command is available for members only.

Support Information tab

Used to add information regarding restraints, releases and spring constant values of the supports and the joint numbers where they are located.

All tab

Used to print joint coordinates, member information, member properties, material properties, and support information, in that order.

Element Information tab

Used to add element incidences, element thickness, and Poisson ratios for plate elements.

Solid Information tab

Used to add element incidences and Poisson ratios for solid elements.

Member Properties tab

Used to add member properties including cross sectional area, moments of inertia, and section moduli in both axes.

After current	Select this option to add the pre analysis print statement being defined in the Command Tree after the currently selected (highlighted) command.

Add	Assigns the selected print command to the input command file.
Assign	Used to assign the command to the current selection set.
Cancel	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *A. To specify pre-analysis commands* (on page 960)

Analysis/Print Commands dialog (Post Print)

Used to define the post-analysis print commands to be included in the Input Command File. Commands are indicated by their respective tab.

Opens when:

Load List tab

Used to select the Load Cases to be included in the output.

Load Cases list	All load cases and combinations in the input file are listed here.
Selection tools	
Load List	Contains all the selected load cases which will be included in the output.

Section tab

Used to specify up to five sections along the length of frame members where forces and moment results are desired.

Joint Displacement tab

Used to print joint displacements in a tabulated form. The displacements for all six degrees of freedom will be printed for all specified joints for all specified load cases.

Member Forces tab

Used to print member forces (i.e. axial force (AXIAL), shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), torsional moment (TORSION), moments about local Y and Z axes (MOM-Y and MOM-Z) in a tabulated form for all specified members for all specified load cases.

Global Selection this option to display the member forces in terms of the Global axis. Otherwise, member forces are presented in local member axis.

Support Reactions tab

Used to print support reactions in a tabulated form for all specified supports for all specified load cases.

Story Drift tab

Used to obtain a print-out of the average lateral displacement of all joints at each vertical level of the structure.

Cg tab

Prints the coordinates of the center of gravity of the structure.

Surface Forces tab

Physical Member Forces tab

Used to print produce member forces for all the analytical members in the physical member (PMEMBER) group

Buckling Shapes tab

Used to print the buckling shape for a Buckling Analysis.

Dia CR tab

Used to print the center of rigidity and center of mass for each rigid floor diaphragm in the model, which are used to determine the natural torsion moment for seismic loads which use that option.

Cable Sag

Used to include the sagging deflected shape of cable members for an Advanced Cable Analysis.

The output of a successful advanced cable analysis will report the cable sag in local XYZ coordinates. Post-analysis print will calculate the actual, nonlinear cable displacements along the length of cable.

Mode Shapes tab

Used to print mode shape values at every joint for all calculated modes.

Element Stress Solid tab

Used to print stresses at the center of solid elements for all specified solids for all specified load cases.

Print Element Stresses	Select this option to print center stresses.
Print Element Joint Stresses	Select this option to print stresses at the corner nodes.

Section Displacement tab

Used to calculate and print section displacements at sections (intermediate points) of frame members.

Number of Sections	Number of intermediate sections at which results are to be reported.
Options	The *Maximum* option prints only the maximum local displacements from among all load cases. The *Save* option saves the section displacements to a file for future reference. The *No Print* option does not print the section displacements to the .ANL file and should be used in conjunction with the *Save* option.
Assign	Select if the command is to be assigned to all members in the current view or to a selection of members. **Note:** Selection must be made prior to selecting the menu command item.

Force tab

Used to print force/moment envelopes for frame members for the specified *Number of Sections*. It prints the maximum and minimum values for every section for each specified load case.

Number of Sections	Specify the number of equally-spaced sections to be considered.

Max Force tab

Used to print force/moment envelopes for frame members for the specified *Number of Sections*. It prints the maximum and minimum values of all sections from among the specified load cases.

Number of Sections	Specify the number of equally-spaced sections to be considered.

Analysis Results tab

Used to print Joint Displacements, Support Reactions, and Member Forces for all specified joints/members for all specified load cases. A message dialog will open to confirm you wish to add the Analysis Results command.

Member Section Forces tab

Member Stresses tab

Used to print member stresses at the start and end joints and at all specified intermediate sections of selected members. These stresses include axial, (axial force over area), bending-y (moment-y over section modulus in local y-axis), bending-z (moment-z over section modulus in local z-axis), shear stress in both local y and z directions and combined (absolute combination of axial, bending-y and bending-z) stresses.

Element Forces/Stresses tab

Used to print stresses (FX, FY, FXY, QX, QY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) at the centroid of plate elements. The Von Mises stresses (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.

Selecting the Print Nodal Point Forces & Moments checkbox will also print stresses and moments at the corner nodes. The Print Force at Point option can be used to print element forces at any point within the element.

Note: For further information, please refer to G.5.1 Plate and Shell Elements (on page 2099) and TR.41 Section Specification (on page 2665) .

After current	Select this option to add the pre analysis print statement being defined in the Command Tree after the currently selected (highlighted) command.
Add	Assigns the selected print command to the input command file.
Assign	Used to assign the command to the current selection set.
Cancel	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *A. To specify post-analysis print commands* (on page 963)

Floor Diaphragm Options dialog

Used to select a code by which the seismic code checks should be performed.

Opens when the **Miscellaneous Commands** > **Floor Diaphragm Options** tool is selected in the **Analysis Data** group on the **Analysis and Design** ribbon tab.

Design Code	Select one of the following codes: • **IS1893 2002** • **ASCE7 05/10/16** • **IS1893 2016**
Check Soft Story	Check this option to check for soft story (vertical irregularities) between two adjacent diaphragms.
Check Irregularities	(inactive for IS1893 2002) Check this option to check for plan as described in TR.28.2.2 Check Irregularities (on page 2333).

- For IS1893 2016, this will check for torsion irregularities, reentrant corner irregularities, mass irregularities, and irregular modes of oscillation (when zone 4 or 5 have been selected in the static seismic definition).
- For ASCE7 05/10/16, this will check for torsion irregularities, reentrant corners, and mass irregularities.

Base Level If the base level of the structure is not at the minimum global Y value of the entire model, then check this option and type the value in the current length units.

No of Diaphragms Displays the number of diaphragms in the model. This is field is for information only and cannot be edited from this dialog.

Related Links

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

TR.31.2.16 IBC 2015 Seismic Load Definition

- *A. To check for soft stories and seismic code irregularities* (on page 962)
- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)
- *A. To check for soft stories and seismic code irregularities* (on page 962)

Load List dialog

Used to specify a list of existing load cases and load combination cases to be used for subsequent processes, like design, print, plot etc.

Opens when the **Load List** tool is selected from the **Analysis Data** group on the **Analysis and Design** ribbon tab.

Load Cases list All existing load cases including combination load cases are listed in the left hand side list box. These load cases can be selected and transferred to the right hand side **Load list**.

List Operators

Click this button…	to…
>	Add the selected primary load case to the Load list
>>	Add all primary load cases to the Load list.
<<	Remove all entries in the Selected Load Combinations list, load combination is placed in the Load Cases list.
<	Remove the selected entry from the Selected Load Combinations list.

Load list This list box contains the list of selected load cases to be included in the load group.

Related Links

- *A. To create a load list* (on page 961)

STAAD Analysis and Design dialog

This dialog box displays the status of the analysis process.

If an error occurs during the analysis, the above dialog box displays the error message.

> For example, if you are running a demonstration version, or the hardware lock is not found, you may encounter an error message like the following:
>
> ```
> DEMO VERSION: Analysis Limited to 6 members.
> ```

The following options allow you to control what happens when you click **Done**.

- **View Output File** – opens the STAAD Output Viewer with the analysis results presented in a textual format.
- **Go to Post Processing Mode** – opens the STAAD.Pro Postprocessor workflow where you can graphically view results.
- **Stay in Modeling Mode** – closes the dialog while staying in the Analytical Modeling workflow.

Related Links

- *STAAD Analysis and Design dialog* (on page 2932)
- *A. To perform an analysis in STAAD.Pro* (on page 966)
- *STAAD Analysis and Design dialog* (on page 2932)
- *A. To perform an analysis in STAAD.Pro* (on page 966)

Steel Design - Whole Structure dialog

Used to select a steel design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design** > **Steel Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

Note: For additional information on steel design codes available in STAAD.Pro, refer to D. Available Steel Design Codes (on page 969). The D. Design Codes (on page 1086) section contains detailed descriptions of the design parameters used for each code.

Current Code	Select the code to use for design from the drop-down list.
Command Tree	Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters. • Green Check ( ) marks: Commands which can be re-specified or modified. • Gray Check ( ) marks: Commands which may not be changed. • Question ( ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Highlight Assigned Geometry	Select this option to have model objects highlighted in the View Window as commands are selected in the **Command Tree**
Toggle Assign	Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens. • Click on the member or element once - the attribute gets assigned. • Click on the same member or element a second time - the attribute gets de-assigned. • Click on the same member or element again, - the attribute gets re-assigned. Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.
Select Parameters...	Opens the Parameters Selection dialog (on page 2934), which is used to select the parameters included in the STAAD.Pro input file for design
Define Parameters...	Opens the Design Parameters dialog (on page 2934) for the selected code, which is used define design parameters and add them to the input file
Commands...	Opens the Design Commands dialog (on page 2935), which is used to add design commands for the selected design code to the input file.
Assignment Method	Select the method for command/parameter assignment. • Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the **Command Tree**. Note that the label for this button changes depending on whether nodes, members or elements are selected. • Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams). • Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode. • Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button. • Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group / Deck (on page 2936), which is used to select the group for assignment. Once an assignment method has been selected, the
Assign	Used to assign the selected parameter to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *D.Steel Design Overview* (on page 975)
- *D. To specify steel design code and parameters* (on page 976)
- *TR.48 Steel and Aluminum Design Specifications* (on page 2675)

- *D. To design steel members in groups* (on page 977)
- *D. To specify steel design commands* (on page 977)

Parameters Selection dialog

Used to select the parameters included in the **Design Parameters** dialog.

Opens when **Select Parameters** is clicked on a **<material> Design - Whole Structure** dialog.

Available Parameters — Lists all available parameters for the current design code.

Selected Parameters — Parameters added to this list will be made available for use for the current design code in the Define Parameters... dialog.

> **Note:** By default, all parameters are added to the list.

Click this button...	to...
>	Include the selected design parameter in the Selected Parameters list.
>>	Include all design parameters in the Selected Parameters list.
<<	Remove all design parameters from the Selected Parameters list.
<	Remove the selected design parameter from the Selected Parameters list.

OK — Updates the Parameters list in the Design Parameters dialog (on page 2934) and closes the Parameter Selection dialog.

Cancel — Closes the dialog without updating changing the selected parameters.

Help — Opens the STAAD.Pro help window.

Design Parameters dialog

Used to define design parameters for the current design code and add them to the input file.

Opens when **Define Parameters** is clicked on a **<material> Design - Whole Structure** dialog.

Parameter list — All parameters available for the current design code are displayed here.

> **Note:** For details on each design code parameters, refer to D. Design Codes (on page 1086).

After Current — Select this option to have the new design parameter inserted after the currently selected in the Design dialog explorer list.

Add	Adds the selected design parameter to the current code in the input file. The design parameter must then be assigned to model objects.
Assign	Active if appropriate model objects were selected prior to clicking **Define Parameter**. Adds the design parameter to the input file and assigns the design parameter to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *D.Steel Design Overview* (on page 975)
- *D. To specify steel design code and parameters* (on page 976)
- *D. To specify concrete design code and parameters* (on page 1069)
- *D. To specify aluminum design code and parameters* (on page 1083)
- *TR.52 Timber Design Specifications* (on page 2682)
- *D. To specify timber design code and parameters* (on page 1085)

Design Commands dialog

Used to add design commands for the selected design code to the input file.

Opens when **Commands** is clicked on a **<material> Design - Whole Structure** dialog.

Command list	All design commands available for the current design code are displayed here. **Note:** For details on the available design commands for each material and code, refer to D. Design Codes (on page 1086).
After Current	Select this option to have the new design command inserted after the currently selected command in the Design dialog explorer list.
Add	Adds the selected design command to the current code in the input file. The design command must then be assigned to model objects.
Assign	Active if appropriate model objects were selected prior to clicking **Commands**. Adds the design command to the input file and assigns the design command to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *TR.48 Steel and Aluminum Design Specifications* (on page 2675)
- *D. To design steel members in groups* (on page 977)
- *D. To specify steel design commands* (on page 977)
- *D. To specify concrete beam design command* (on page 1070)
- *D. To generate concrete take off* (on page 1071)
- *D. To specify aluminum design commands* (on page 1083)
- *D. To generate aluminum take off* (on page 1084)
- *D. To generate steel take off* (on page 978)
- *D. To specify timber design commands* (on page 1086)

Select Group / Deck

Used to select the group for assignment of commands.

Opens when **Select Group/Deck** is clicked in a **<material> Design - Whole Structure** dialog.

List	Lists all previously defined groups and composite decks in the input file, organized by type.
OK	Assigns the parameter selected in the Design dialog list to the selected group or deck.
Cancel	Closes the dialog without making any assignments.

Concrete Design - Whole Structure dialog

Used to select a concrete design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design** > **Concrete Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

Note: For additional information on concrete design codes available in STAAD.Pro, refer to D. Available Concrete Design Codes (on page 1067). The D. Design Codes (on page 1086) section contains detailed descriptions of the design parameters used for each code.

Current Code

Select the code to use for design from the drop-down list.

Command Tree

Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters.

- Green Check ( ) marks: Commands which can be re-specified or modified.
- Gray Check ( ) marks: Commands which may not be changed.
- Question ( ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Highlight Assigned Geometry

Select this option to have model objects highlighted in the View Window as commands are selected in the **Command Tree**

Toggle Assign

Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens.

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.
- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Select Parameters...

Opens the Parameters Selection dialog (on page 2934), which is used to select the parameters included in the STAAD.Pro input file for design

Define Parameters...	Opens the Design Parameters dialog (on page 2934) for the selected code, which is used define design parameters and add them to the input file
Commands...	Opens the Design Commands dialog (on page 2935), which is used to add design commands for the selected design code to the input file.
Assignment Method	Select the method for command/parameter assignment. • Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the **Command Tree**. Note that the label for this button changes depending on whether nodes, members or elements are selected. • Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams). • Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode. • Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button. • Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group / Deck (on page 2936), which is used to select the group for assignment. Once an assignment method has been selected, the
Assign	Used to assign the selected parameter to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *D. To specify concrete design code and parameters* (on page 1069)
- *D. To specify concrete beam design command* (on page 1070)
- *D. To generate concrete take off* (on page 1071)

Aluminum Design - Whole Structure dialog

Used to select an aluminum design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design** > **Aluminum Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

Note: For additional information on aluminum design codes available in STAAD.Pro, refer to D. Available Aluminum Design Codes (on page 1081). The D. Design Codes (on page 1086) section contains detailed descriptions of the design parameters used for each code.

Current Code	Select the code to use for design from the drop-down list.

Command Tree

Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters.

- Green Check ( ) marks: Commands which can be re-specified or modified.
- Gray Check ( ) marks: Commands which may not be changed.
- Question ( ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Highlight Assigned Geometry

Select this option to have model objects highlighted in the View Window as commands are selected in the **Command Tree**

Toggle Assign

Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens.

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.
- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Select Parameters...

Opens the Parameters Selection dialog (on page 2934), which is used to select the parameters included in the STAAD.Pro input file for design

Define Parameters...

Opens the Design Parameters dialog (on page 2934) for the selected code, which is used define design parameters and add them to the input file

Commands...

Opens the Design Commands dialog (on page 2935), which is used to add design commands for the selected design code to the input file.

Assignment Method

Select the method for command/parameter assignment.

- Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the **Command Tree**. Note that the label for this button changes depending on whether nodes, members or elements are selected.
- Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams).
- Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode.
- Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button.
- Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group / Deck (on page 2936), which is used to select the group for assignment.

Once an assignment method has been selected, the

Assign	Used to assign the selected parameter to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *D. To specify aluminum design code and parameters* (on page 1083)
- *D. To specify aluminum design commands* (on page 1083)

Timber Design - Whole Structure dialog

Used to select a timber design code, select parameters to use for design, and assign parameters to the model.

Opens when the **Design** > **Timber Design** tool is selected from the **Design** group on the **Analysis and Design** ribbon tab.

Note: For additional information on timber design codes available in STAAD.Pro, refer to D. Available Timber Design Codes (on page 1084). The D. Design Codes (on page 1086) section contains detailed descriptions of the design parameters used for each code.

Current Code

Select the code to use for design from the drop-down list.

Command Tree

Displays a summary of all input file commands. Each command is represented with that commands parameters as child elements. Click the [+] icon to expand the command to display parameters.

- Green Check ( ) marks: Commands which can be re-specified or modified.
- Gray Check ( ) marks: Commands which may not be changed.
- Question ( ) marks: Commands which have been added but not assigned to any structural element (joint/member/element).

Highlight Assigned Geometry

Select this option to have model objects highlighted in the View Window as commands are selected in the **Command Tree**

Toggle Assign

Select this option to use the toggle mode. In this mode, when an attribute is selected and assigned using the "Use Cursor to assign" method, the following happens.

- Click on the member or element once - the attribute gets assigned.
- Click on the same member or element a second time - the attribute gets de-assigned.
- Click on the same member or element again, - the attribute gets re-assigned.

Thus each click will result in an assign if the attribute was un-assigned, and a de-assign if the attribute was assigned.

Select Parameters...

Opens the Parameters Selection dialog (on page 2934), which is used to select the parameters included in the STAAD.Pro input file for design

Define Parameters...

Opens the Design Parameters dialog (on page 2934) for the selected code, which is used define design parameters and add them to the input file

Commands...	Opens the Design Commands dialog (on page 2935), which is used to add design commands for the selected design code to the input file.
Assignment Method	Select the method for command/parameter assignment. • Assign to Selected <objects> - Available once objects are selected in the View window. Used to assign a Design option to selected joints, members or elements, first select the command from the **Command Tree**. Note that the label for this button changes depending on whether nodes, members or elements are selected. • Assign to View - Assigns the selected parameter all objects in the active View window. Only objects to which the parameter is relevant will receive the parameter (e.g., steel member design parameters are only assigned to beams). • Use Cursor To Assign - Use the mouse pointer to assign the parameter to model objects. Once selected, click the Assign button to activate this mode and click again to deactivate. The **Toggle Assign** option can also be used to aid in this mode. • Assign To Edit List - Used to assign with a typed list of object numbers, first select the material tag from the Materials dialog box. The selected material is highlighted. Select the Assign To Edit List radio button, then type the list of member or element numbers and click the Assign button. • Select Group/Deck - (Steel Design only) Active if either a group or deck has been defined in the input file and an appropriate parameter is selected. Opens the Select Group / Deck (on page 2936), which is used to select the group for assignment. Once an assignment method has been selected, the
Assign	Used to assign the selected parameter to the selected objects.
Close	Closes the dialog.
Help	Opens the STAAD.Pro help window.

Related Links

- *TR.52 Timber Design Specifications* (on page 2682)
- *D. To specify timber design code and parameters* (on page 1085)
- *D. To specify timber design commands* (on page 1086)

Utilities tab

Table 331: Geometry Tools group

Tool Name		Description
 **Structure Tools >**	**Multiple Structures**	Opens the **List of Structures** dialog, which is used to determine if the current model consists of more than one unconnected structure. Select a Structure in the list to see each structure highlighted.

Tool Name		Description
	Beam Plate Connectivity	Used to check for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.
	Merge Properties	Used to merge the properties of two or more similar objects. When a STAAD input file has a number of references of the same property, this tool can be used to consolidate all these properties into a single command. When a STAAD file has a number of references of the same property, there is now a tool to consolidate all these properties into a single command. Clicking the Yes button, all instances of a given section property will be collated into a single property reference. **Note:** Properties references with differing additional parameters will not be collated. Properties references with differing assigned material properties will not be collated.
	Cut Section	Opens the **Section** dialog, which is used to cut a section through the structure along a specified global plane at a desired location of the 3rd axis.
Node Tools >	**Duplicate Nodes**	Opens the **Remove Duplicate Nodes** dialog, which is used to set the tolerance distance between two nodes the program should consider as duplicate. Used for detecting the presence of two or more instances of the same node.
	Orphan Nodes	Highlights all nodes in the structure which are not connected to any member, element, or solid.
	Remove Orphan Nodes	Used to remove all detected orphan nodes.
	Node to Node Distance	Display the distance between nodes.
	Remove Node to Node Distance	Used to remove the display of all node to node dimensions from the current view.
Beam Tools >	**Duplicate Beams**	Opens the **Remove Duplicate Beams** dialog, which is used to set the tolerance distance between two beams the program should consider as duplicate. Used for detecting the presence of two or more instances of the same beam.

Tool Name		Description
	Zero Length	Opens the **Zero Length Tolerance** dialog, which is used to set the tolerance for zero length members. A member connected between duplicate nodes, which have the same (X,Y,Z) coordinates, will have a length of zero. This tool detects such members.
	Overlapping Collinear Members	When two members are collinear, and further, at least one of the nodes of one of those members happens to lie within the span of the other, but the two members are not connected at that node, those two members are considered as overlapping collinear members. This tool detects such members. The usefulness of this tool comes from the fact that it enables you to detect modeling errors which are not easily visible. Two lines overlapping on the drawing area become indistinguishable on large models, and this is one of the tools that can spot such errors.
	Beam Incidence	Used to reverse the incidence of selected beams so that the global coordinates of the end node are farther from the origin than that of the start node.
	Dimension Beams	Display the dimension of the members in the structure.
 **Plate Tools >**	**Duplicate Plates**	Opens the **Remove Duplicate Plates** dialog, which is used to set the tolerance distance between two plates the program should consider as duplicate. Used for detecting the presence of two or more instances of the same plate.
	Warped Plates	A warped plate is defined as a four-noded plate whose nodes do not lie on the same plane. This tool detects such plates.
	Plate Connectivity	Checks the model for plates that overlap or intersect each other at their boundaries and, if found, opens the Overlapping Collinear Plates dialog. Typically, these overlaps cause improper load transfers or instabilities in the model.
 **Solid Tools >**	**Negative Volume**	Used to verify that the solid elements in their model have the proper sequence (order) of node numbering to prevent warnings in the output file of solids containing negative volumes.
	Warped Solids	Used to check if a solid element is warped. Warped solids cannot be analyzed and will produce errors in the output file.

Tool Name	Description
Physical Member Restraints	Used to automatically generate top and bottom flange restraint conditions for the selected physical members. **Note:** The PBRACE command generated by this menu item is valid only for D2.B.12 Physical Member Design (on page 1372) per AS 4100-1998 (Australian) steel design. When selected, the program will search through all the nodes along the Physical Member and determine if any other beam is connected besides the beams in the current physical member definition. If another beam is present, a full restraint is placed on both the flanges. Otherwise, an unrestrained condition is imposed on both the flanges.
Groups	Opens the **Create Group** dialog, which is used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name. **Note:** If no groups exist, you will be prompted to create a group using the **Define Group Name** dialog.

Table 332: Physical Model group

Tool name	Description
Drop Physical Model	For models that have a linked physical model, this will open a confirmation dialog asking if you want to remove that link. Removing this link will enable the model to be edited using all analytical modeling tools as well as remove the disabled sections of the input file in the STAAD.Pro Editor.

Table 333: Query group

Tool Name		Description
Query >	**Beams**	Used to view detailed information for one beam at a time in a dialog.
	Physical Members	Used to view detailed information for one physical member at a time in a dialog.
	Plates	Used to view detailed information for one plate at a time in a dialog.
	Solids	Used to view detailed information for one solid at a time in a dialog.

Tool Name		Description
	Nodes	Used to view detailed information for one node at a time in a dialog.

Table 334: Display group

Tool name	Description
 **Insert Text**	Opens the **Write Text to Insert** dialog, which is used to add custom text to the View window (and, subsequently to pictures you take of the window).

Table 335: Edit group

Tool Name	Description
 **Command File**	Opens the current input command file (file extension `.std`.) in the STAAD.Pro Editor. If any change has been made in the structure that has not been saved, you are prompted to save the structure first. **Note:** Refer to I. STAAD.Pro Editor (on page 2043) for additional assistance.

Table 336: View group

Tool Name	Description
 **Analysis Output**	Opens the results of a successful Analysis and Design run in the STAAD.Pro Editor window.
 **Analysis Log**	Opens the analysis and design results file (file extension `.anl`) in the STAAD.Pro Editor.

Tool Name	Description
Error Log	If analysis errors occurred, this tool is active. Click to open the error log in the STAAD.Pro Editor.

Table 337: Tools group

Tool Name		Description
Connection Tags >	**Load Connection Tag File**	
	Edit Connection Tag File	
	View Connection Tags	Opens the **Assign Connection Tags** dialog.
	Check Connection Tags	(Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.
Calculator		Opens the **STAAD.Pro Calculator** window, which is capable of performing mathematical operations.
Unit Converter		Opens the **STAAD.Pro Converter** window, which is used to convert data from one unit system to another.
Take Picture		Used to take a snapshot image of current view. The picture is automatically added to a picture album. **Tip:** The picture is stored in a STAAD-native format and can be subsequently included in custom reports during Report Setup. The Copy Picture and Export View tools may be used to copy a model image or to save it to a common file format.

Tool Name	Description
Copy Picture	Used to copy the current picture to the clipboard for pasting in other Windows applications such as image editors, spreadsheet applications, or word processors,.
 **AVI File**	Opens the **Create AVI File** dialog, which is used to create a video file recording of for animated deflection, section displacement, mode shape, and plate stress contour diagrams.

Table 338: Developer group

Tool name	Description
 **Macro Editor**	Opens the **STAAD.Pro Script Editor** window, which is used to write macros.
 **Macros**	Opens the Macro dialog, which is used to create, manage, and run Visual Basic Script macros in STAAD.Pro.

Table 339: User Tools group

Tool Name		Description
 **Configure**		Opens the Customize User Defined Tools dialog (on page 2957), which is used for running a macro utilizing OpenSTAAD functions. For additional information, refer to the OpenSTAAD (on page 7000).
 **User Tools**	**Export Model to AutoPIPE**	Opens the **Export STAAD Model to AutoPipe** dialog. Refer to M. To export STAAD.Pro model data into AutoPIPE (on page 907) for information on using this macro.
	AIJ Route 2 Utility	Opens the **AIJ Route 2 Design Tool** which is custom tool for use with the Japanese design code.

Tool Name		Description
	List Object GUIDs	Generates a custom table with tabs containing the unique IDs for each node, member, and physical member in the model.
	CIS/2 Object Update Report Tool	Opens the **Report Option** dialog, which is used to select the CIS/2 object report type to generate.
	Export Model to SACS	Opens the **Export STAAD Model to SACS** dialog. Refer to I. To export to a SACS input file (on page 2081) for information on using this macro.
	Euro Code Load Combination Generator	Opens the **Eurocode Combination Generator** dialog. Refer to M. To generate load combinations per Eurocode (on page 890) for information on using this macro.
	Add Material	Opens the **Add Materials** dialog, which is used to add standard material definitions to your model. Refer to M. To add a predefined material (on page 795) for details on using this macro.
	Additional user tools	Any user tools you have added will also appear in this list.

Improper Connectivity dialog

Used to the model for plates that are improperly connected to beams. Beams must be connected to plates at their nodes in order to ensure proper coupling and load transfer. This tool will inform the user if any improper connections are present in the model.

If there are improper connections between beams and plates (not connected by their nodes), a dialog box displaying a list of the plates and the beams intersecting them (but not connected at the node) will be shown.

Opens when the **Structure Tools** > **Check Beam Plate Connectivity** tool is selected in the **Geometry Tools** group on the **Utilities** ribbon tab.

Overlapped Entities list	A list overlapping entities is displayed. Select one to highlight in the active view window by the selected option.

Highlight options	The first option (Highlight-Both) will highlight both of selected entities listed while the last two options will highlight either beam .
Close	Closes the dialog.

Example

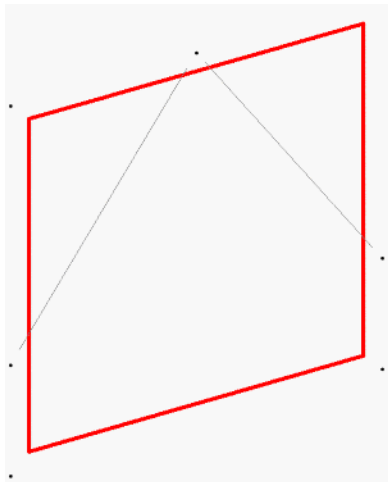


Figure 350: An example of improper beam to plate connectivity

List of Duplicate Nodes / Beams / Plates dialog

Used to detect the presence of two or more instances of the same beam or plate, depending on the ribbon tool selected.

Opens when either:

- **Node Tools** > **Duplicate Nodes** is selected from the Geometry Tools group on the **Utilities ribbon** tab,
- **Beam Tools** > **Duplicate Beams** is selected from the Geometry Tools group on the **Utilities ribbon** tab,
- **Plate Tools** > **Duplicate Plates** is selected from the Geometry Tools group on the **Utilities ribbon** tab.

If any duplicates of the selected object type are detected (that is, beams that share common end nodes or plates that share all common vertices), then a list of the duplicate beam or plate pairs is displayed.

Retain	When a pair is selected, click **Retain.** You are then prompted to select which member or plate number to retain.
Remove All Duplicates	Click to remove all duplicate items in the model. You are prompted to confirm you want to remove the second and additional items listed from the model. Click **OK.**
Show Node Coordinates	(Nodes only) Click to display a list of nodal coordinates for the duplicate nodes.

Related Links

- *M. To check for and remove duplicate entities* (on page 914)

Overlapping Plates dialog

Used to determine if two or more plates have intersecting boundaries not attached at nodes or otherwise overlap in the same plane.

An overlapping plate is defined as a plate whose nodes intersect other plates at points other than the defined nodes. This entails plates whose boundaries with adjacent plates are not attached at the nodes or plates within other plates (in the same plane). However, if two elements are such that the plane of one element is at an inclination to the plane of the other, and none of the nodes of one element lie on the plane of the other, those cannot be detected.

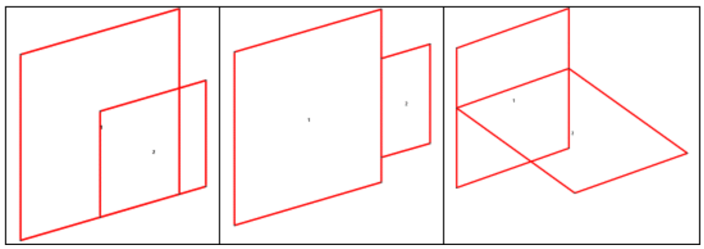


Figure 351: Examples of overlapping plates

Tip: Typically, a more refined meshing of the overlapping elements is required to eliminate the above problem.

Overlapped Entities list	A list overlapping plate pairs is displayed. Select one to highlight in the active view window by the selected option.
Highlight options	The first option (Highlight-Both) will highlight both of the plates listed while the last two options will highlight each individual plate separately.

Display/Remove Dimensions dialog

Used to display dimensions of beams in the view.

Opens when the **Beam Tools** > **Dimension Beams** tool is selected in the **Geomtry Tools** group on the **Utilities** ribbon tab.

Dimensions can be displayed or removed from the entire structure, selected beams, or a list of beam numbers.

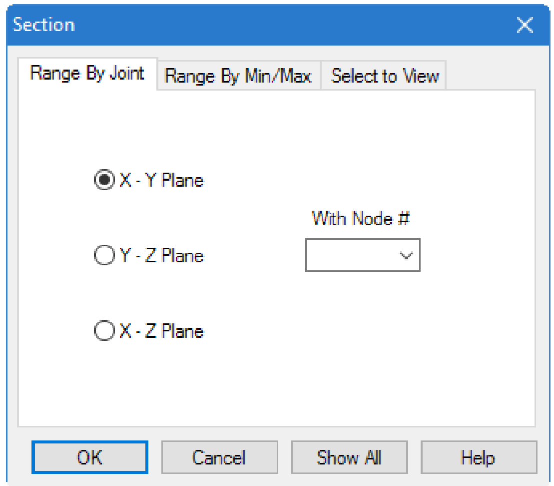
Related Links

- *M. To display beam lengths* (on page 918)

Section dialog

Used to cut a section through the structure along a specified global plane at a desired location of the third axis. Sectional views parallel to global XY, YZ, and XZ planes may be created.

Opens when the **Structure Tools > Cut Section** tool is selected from the **Geometry Tools** group on the **Utilities** ribbon tab.



Range By Joint tab

Select the plane of the section by selecting either the **X-Y Plane**, **Y-Z Plane**, or **X-Z Plane** option. In the **With Node #** drop down list, select the joint which lies on the sectional plane.

> For example, if the section is to be drawn parallel to XY plane at Z=5, then, select the **X-Y Plane** option and select a joint number with a Z coordinate of **5**.

Range By Min/Max tab

Select the plane of the section by selecting either the **X-Y Plane**, **Y-Z Plane**, or **X-Z Plane** option. The **Minimum** and **Maximum** fields represent the boundary distances along the axis perpendicular to the sectional plane. Every object lying between these two distances would be displayed.

> For example, if the section is to be drawn parallel to XY plane at Z=5, select the **X-Y Plane** option and select a **Minimum** value of 4 and a **Maximum** value of 6.

Select to View tab

Used to specify the portion of the structure to view by using selected objects.

Window/Rubber Band allows you to select the portion of the structure to view by specifying a rubber-band window around it.

View Highlighted Only displays only the selected (highlighted) objects on screen. You must select the members and elements to view before choosing this option.

Select To View allows you to view only Beams, Plates, and/or Solids, depending on the selection of the corresponding options.

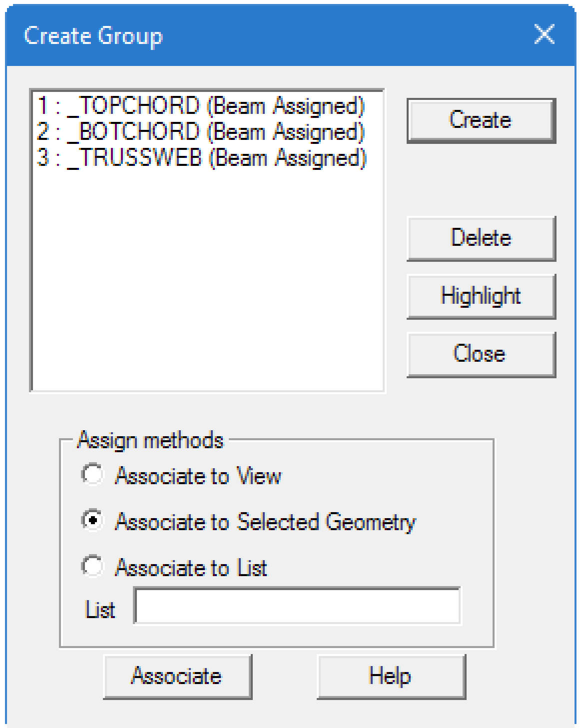
Create Group dialog

Used to cluster a set of joints, beams, plates or solids into a single entity identified by a distinct name.

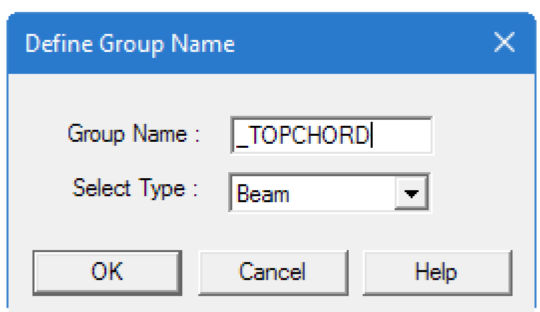
Opens when the **Groups** tool is selected in the **Geometry Tools** group on the **Utilities** ribbon tab.

The group name may be subsequently used instead of a joint, member, or element list to specify properties, supports, loads, etc. This prevents the need for you to repeat the same list of objects within your model. Additionally, group names can be used to highlight geometry using the **Group Selection** tool.

For example, the top chord members of an truss can be grouped into a name called "_TOPCHORD".



Create Opens the **Define Group Name** dialog, which is used to assign a group name and specify Node, Beam, Plate, Geometry, or Floor from the drop down list, as shown below.



Caution: Group names must begin with the underscore "_" character.

Delete This button allows us to delete any group that has already been created. The names of all available groups are displayed along with a reference number in a list box. Select the one we wish to remove and click on the Delete button. That group will be removed from the list.

Highlight	This button allows us to highlight the structural components which comprise a given group. From the list box where the available group names are displayed with a reference number, select the group we wish to highlight.
Close	Click group specification is complete.
Assign Methods	Options: • **Associate to View** – assigns all geometry shown in the current view to the group • **Associate to Selected Geometry** – assigns selected (highlighted) geometry to the group • **Associate to List** – assigns a typed list of joint, member or element numbers to the group
Associate	Click to add the joints, members, or elements to the group. **Note:** Only the objects in the current view, selection set, or list will be added to the selected Group name. Any objects previously in that group that are not in the association method will no longer be contained in that group.

Related Links

- *M. To create a group from a selection* (on page 689)

Check Connection Tags dialog

Used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

Opens when **Connection Tags** > **Check Connection Tags** is selected from the right-click pop-up menu when a member is selected.

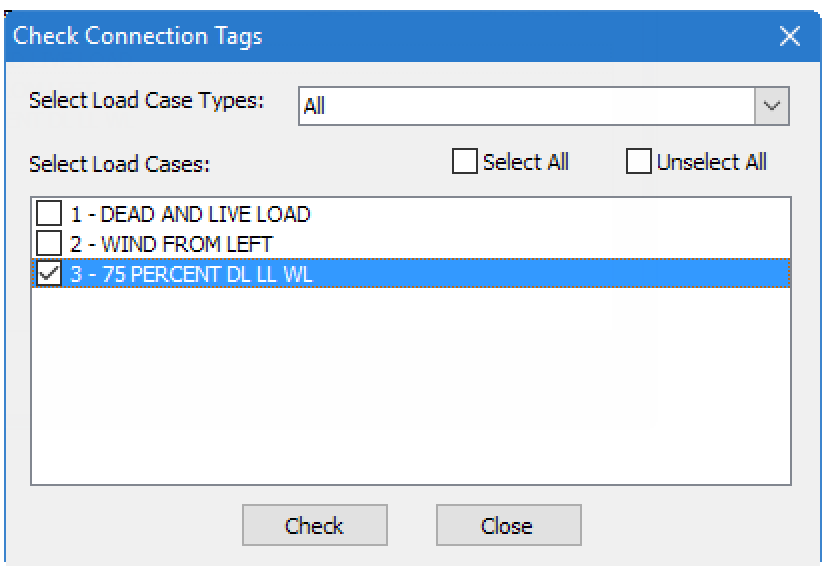


*Figure 352: The **Check Connection Tags** dialog*

Select Load Case Types	Select All load cases, only Primary Load Cases, or only Load Combinations to display in the list.

Select Load Cases	Select one or more load cases from the list by setting the individual check boxes. You can set the **Select All** check box to select all displayed load cases (those filtered by the **Select Load Case Types** selection). To clear all load cases selections, set the **Unselect All** check box.
Check	Checks analysis results at each member end with a connection tag assigned from the selected set of beam members against the connection capacities defined in the Connection Tag XML file. A table with the results of the check opens.
Close	Closes the dialog.

Related Links

- *D. Connection Tags Capacity Checks* (on page 1051)
- *D. To check connection tags* (on page 1050)

STAAD.Pro Calculator utility

STAAD.Pro includes an on-screen calculator which is displayed in a separate window.

Opens when the **Calculator** tool is selected from the **Tools** group on the **Utilities** ribbon tab.

The calculator functions similar to a handheld calculator.

Normal, Engineering, and Integer mode are available.

Create AVI File dialog

Used to create a video file recording of for animated deflection, section displacement, mode shape, and plate stress contour diagrams.

Opens when the **Create AVI File** tool is selected in the **Tools** group on the **Utilities** ribbon tab.

Total No. of Frames	Sets the number of frames used for the entire animation in the AVI file. This value divided by the Frame Rate /sec is the total length of the animation, in seconds.
Frame Rate /sec	The number of frames displayed per second of animation. In other words, this is the speed at which the animation is displayed.
Animation Type	Select the analysis results to be animated from the available options: • **Deflection** — animates the deflected shape of the currently selected load case or combination. • **Sectional Displacement** — animates the local displacement of members. • **Mode Shape** — animates the deflection of the selected mode. Only active if the Frequency tab (on page 2865) in the current STAAD input file. • **Stress Contour** — animates the selected **Plate Stress Type** or **Solid Stress Type.** Only active if Plates or Solid elements are included in the current STAADinput file.
OK	Opens a Save As dialog, which is used to specify a location and file name for the AVI file. Click OK and
Cancel	Closes the dialog without creating an AVI file.

Video Compression dialog

Used to select the video compression codec and set associated compression parameters for an AVI file. AVI files can be quite large, and compression is a technique by which the size of these files may be reduced.

Opens when **OK** is clicked in the Create AVI File dialog (on page 2953).

Compressor

Select one of the included means of video compression:

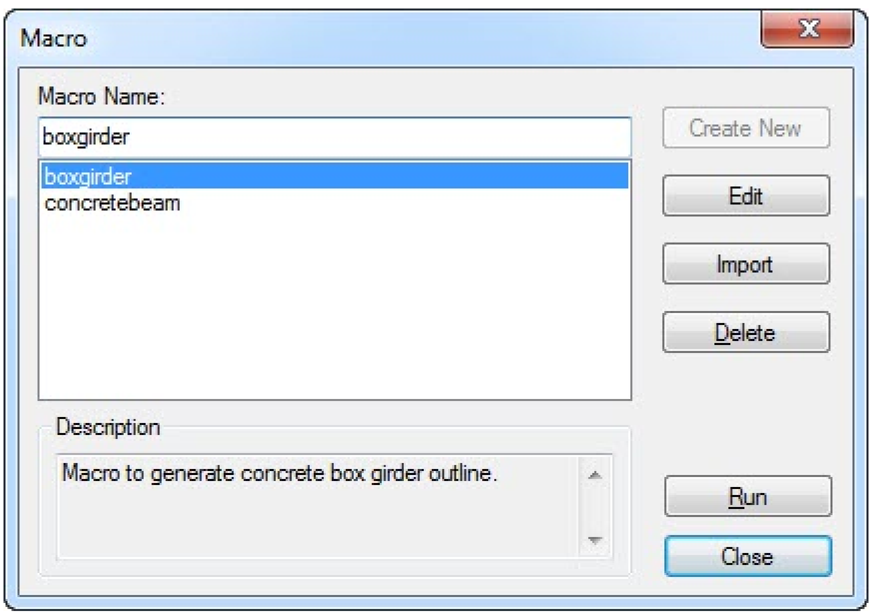
Compressor	Description	Settings
Full Frames (Uncompressed)	No video compression is used. This results in the largest file size but does not require any additional settings or video codecs installed.	none
Microsoft RLE	The Microsoft Run Length Encoding codec is supported in Windows Media Player 9 and higher as well Microsoft Windows XP SP2 and newer.	Compression Quality
Microsoft Video 1	Uses the Microsoft Video 1 Compressor.	Compression Quality
Intel IYUV codec	An uncompressed video codec. The compression quality is set to 75%.	none
Cinepak Codec by Radius	The Cinepak video codec is supported in Windows Media Player 9 and higher as well Microsoft Windows XP SP2 and newer.	Video can be compressed to color or black & white. Compression Quality
VMnc v2	The VMware lossless (uncompressed) codec.	none
TechSmith Screen Capture Codec	Uses the TechSmith Screen Capture Codec, which provides high compression rates with little video degradation for screen recordings.	Compression Control

Compression Quality	Use the slider control to set the compression percentage. The lower the number, the higher the compression amount but with reduced video quality. The default setting for each Compressor is recommended. A value of 100 means no compression is used.
OK	Accepts the settings and produces the AVI video. A message dialog displays the status of the video creation and the animation opens in the AVI Player window.
Cancel	Closes the dialog without creating the AVI video.
Configure	(For select codecs only) Opens a dialog with additional video compression settings, as described in the table above.
About	(For select codecs only) Opens a dialog displaying additional license and contact information about he codec supplier.

Macro dialog

Used to manage VB macros in STAAD.Pro.

Opens when the Run VB Macros tool is selected.



Macro name and list	The first box displays the name of the currently selected macro file, if one is selected. The list box displays a list of all linked macro files.
Description	Displays a brief description of the macro, if available.
Create New	Opens the Select New Macro File Name dialog, which is used to specify a file name, file type, and description for an empty VB macro file. The Macro editor window then opens, which used to record or edit macro commands. Active only when no macro is selected.
Edit	Opens the Macro editor window, which is used to record or edit macro commands.

Import Opens the Add an existing Macro dialog, which is used to select macro for linking to STAAD.Pro.

Delete Removes the macro from the linked macros list.

Note: The macro file is not deleted from the disk, only the link association in STAAD.Pro.

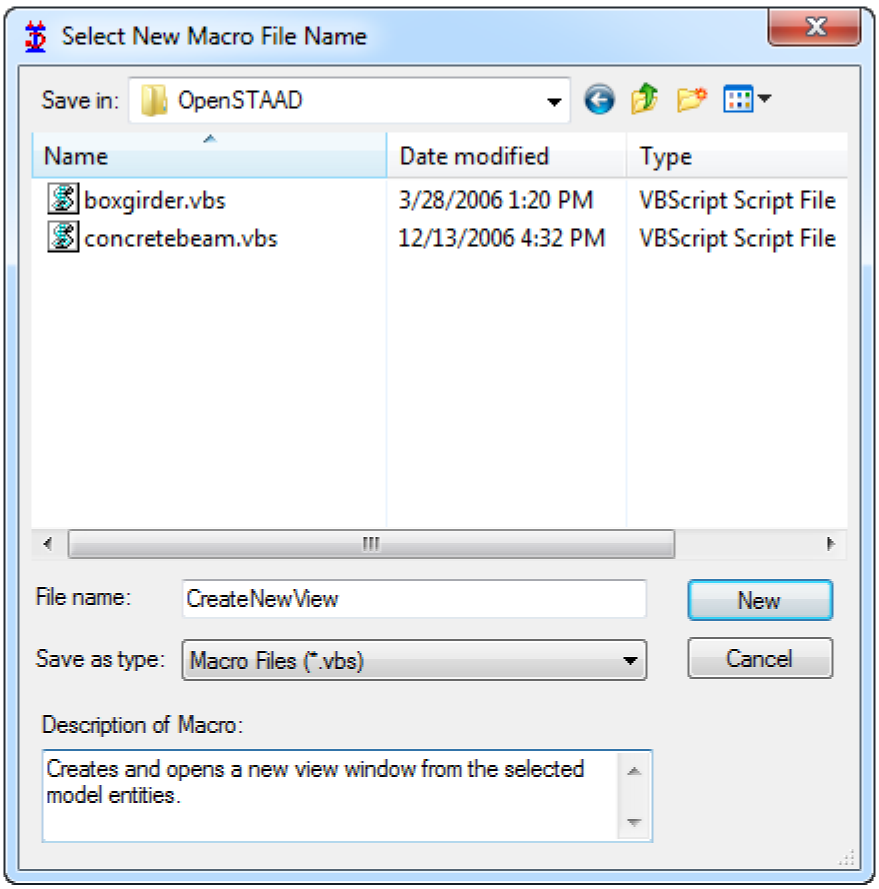
Run Runs the selected macro.

Close Closes the dialog.

Select New Macro File Name dialog

Used to create a new VB macro file for use in STAAD.Pro.

Opens when either **Edit > Create New VB Macro...** is selected or **Create New** is clicked in the Macro dialog (on page 2955).



Save in Select the drive and folder where the new macro file will be saved. Several common Windows folder navigation tools are provided.

File list A list of existing macro files is displayed here.

File name Specify the file name in this text box.

Save as type VB macros can be saved in one of two file types, depending on the level of file protection you want.

- A VBS macro file is a standard macro file, the contents (code) of which can be viewed by other users in any standard text editor.
- A VBZ macro file is a protected macro file, the contents of which cannot be viewed even in an external editor like *Notepad*. This is useful when you want to sell the macro or simply protect its contents from unintentional editing as part of a quality control program.

Description Used to add a brief description of the macros function. This is helpful in identifying similarly named macros

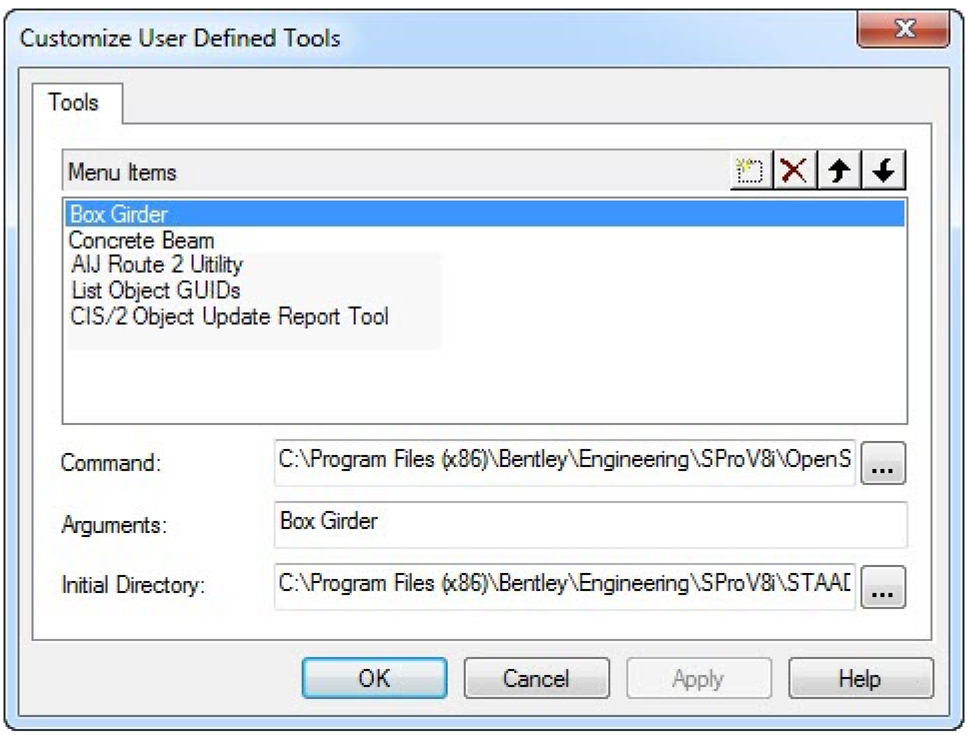
Create New Creates the file with the specified information and opens the file in the STAAD.Pro Macro Editor window.

Cancel Closes the dialog without creating a new macro file

Customize User Defined Tools dialog

Used to customize the tools which appear in the User Tools drop-down menu.

Opens when **Tools** > **Configure User Tools** is selected.



Toolbar

Tool	What it Does
New tool	Adds an empty entry to the Menu Items list. Type a name for a new menu item.

Tool	What it Does
Delete	Removes the current tool from the User Tools list
Move Up	Moves the selected tool up in the Menu Items list.
Move Down	Moves the selected tool down in the Menu Items list.

Command Type the path and filename of the linked macro for the menu item. Click [...] to open a file Select File dialog, which is used to locate and select files.

Arguments Specify any additional parameters associated with the macro, if required.

Initial Directory

OK Saves the changes and closes the dialog.

Cancel Closes the dialog without saving any changes since the Apply was clicked.

Apply Saves any changes made in the dialog.

Help Opens the STAAD.Pro Help contents.

Export STAAD Model to SACS

Used to export the current STAAD.Pro model to a SACS model, which can be opened in the SACS system Interactive Modeling program.

Opens when the **User Tools** > **STAAD to SACS Export** tool is selected in the **User Tools** group on the **Utilities** ribbon tab.

Staad Input File Displays the path and filename for the current STAAD input file.

SACS Model File The file path and file name for the SACS model file (file extension `.inp`) to which you want to export STAAD.Pro data.

[...] Opens a Windows **Save As...** dialog box, which is used to navigate your computer's storage and provide a file name.

Title Specify a model title for use in the SACS file.

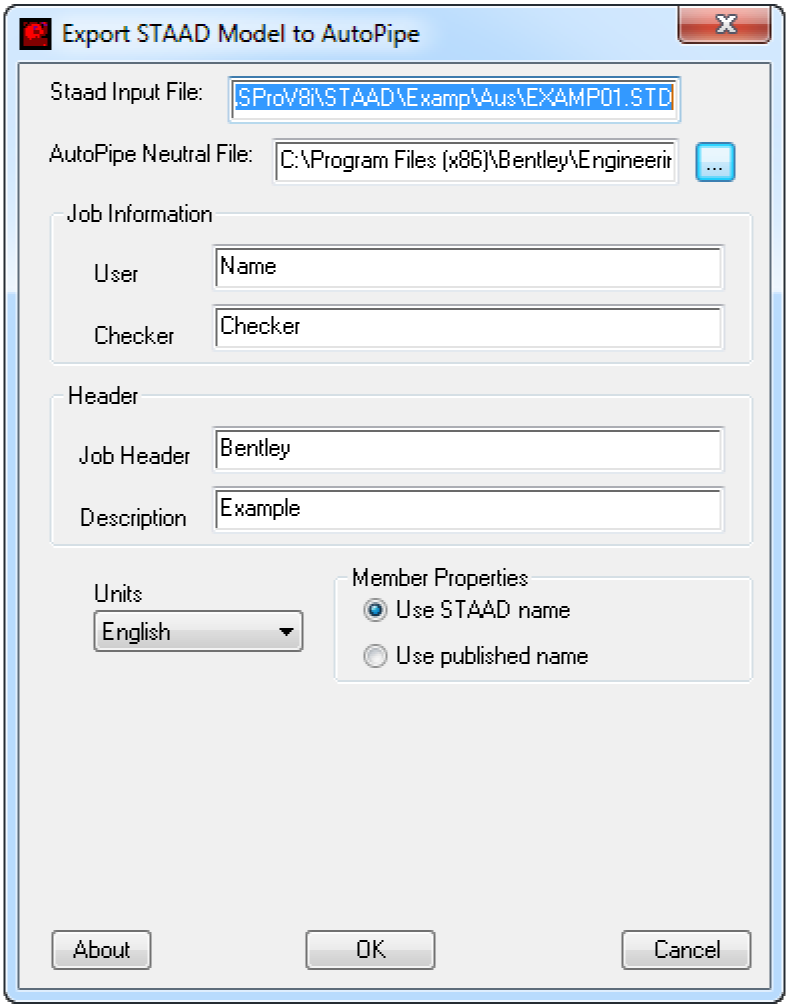
About Displays version information about the macro behind this export facility.

OK Closes the dialog and generates the SACS file.

Cancel Closes the dialog without generating a SACS file.

Export STAAD Model to AutoPIPE

This feature is not yet documented.



Related Links

- *M. To export STAAD.Pro model data into AutoPIPE* (on page 907)

Piping tab

Table 340: Models group

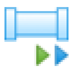
Tool Name		Description
 **Interop >**	**Import**	Opens a **Open** dialog with the **Files of Type** filter set to PipeLink files (file extension `.pipelink`.), which are piping data files created by the PipeLink utility to transfer data between Bentley AutoPIPE (V8i SELECTseries 2 and greater) and STAAD.Pro (V8i SELECTseries 2 and greater).
	Export	Opens the Export Revised Model dialog (on page 2961), which is used to specify the user name and optional comment associated with the updated data set being sent from STAAD.Pro.
	Reload	Reloads the current PipeLink database and initiates the **Support Connection Wizard**. Any changes made to the active model since the last save action will be discarded. Only the current active model is reloaded. **Note:** A reload is always performed when a model is re-imported.
	Remove	Used to remove all imported pipe data and transferred loads from the STAAD.Pro project. A warning message will appear to confirm you wish to proceed with removing the pipe model data. **Caution:** This action cannot be undone.
	Save	Saves the current state of the active pipe model. **Note:** Any changes to the currently active model are saved to the local database whenever the pipe mode is left, the model is deactivated due to another model becoming active, an export operation is initiated, or the **Save** tool is selected on the **Piping** ribbon tab.
 **Pipe Models**		Opens the **Pipe Model** dialog, which is used to select the active pipe model for the Piping workflow. Job and import information for the active model is also displayed here.

Table 341: Connection group

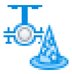
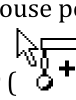
Tool name	Description
 **Support Connection Wizard**	Opens the Support Connection Wizard (on page 2964), which is used to establish logical data links between pipe stress model data imported from a PipeLink file and the current STAAD.Pro model.
 **Connect Support**	Used to specify a connection between a piping model support and the STAAD.Pro structural model. Once this item is selected, the mouse pointer changes to the support connection cursor (), which is used to draw a support association between piping model supports and structural model nodes.
 **Pipe Supports**	Opens the **Pipe Supports** table, which identifies supported pipe nodes and details each support.

Table 342: Loading group

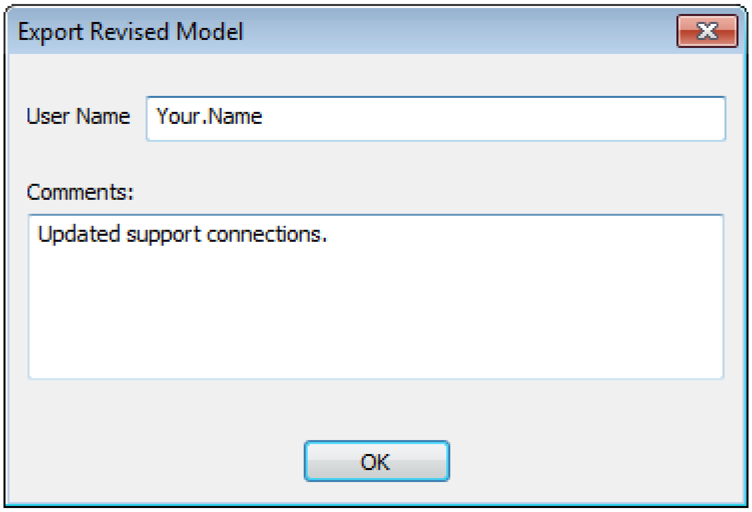
Tool name	Description
 **Transfer Loading**	Opens the **Transfer Pipe Reactions to Structure Model** dialog, which is used to specify STAAD.Pro load data for importing pipe support load data.

Export Revised Model dialog

A database may be created for importing into AutoPIPE. This database includes the pipe information, the structural model (including loads and analysis results), and the connection and load mapping information. This database is not constructed by the STAAD.Pro but rather the PipeLink application.

Note: The STAAD model should be analyzed (if loaded at all) prior to the export. Otherwise, a warning dialog will appear instructing you to do so.

Opens when **Piping** > **Export** is selected.



User Name Typically, enter the name of the engineer responsible for the structural analysis. The current Windows user name is specified here by default.

Comments (Optional) Enter any comments regarding actions performed for reference purposes.

OK Opens the PipeLink for STAAD.Pro V8i application, which is used to select the AutoPIPE data file for exchanging database information.

> **Note:** Refer to the PipeLink manual for assistance in using this utility application.

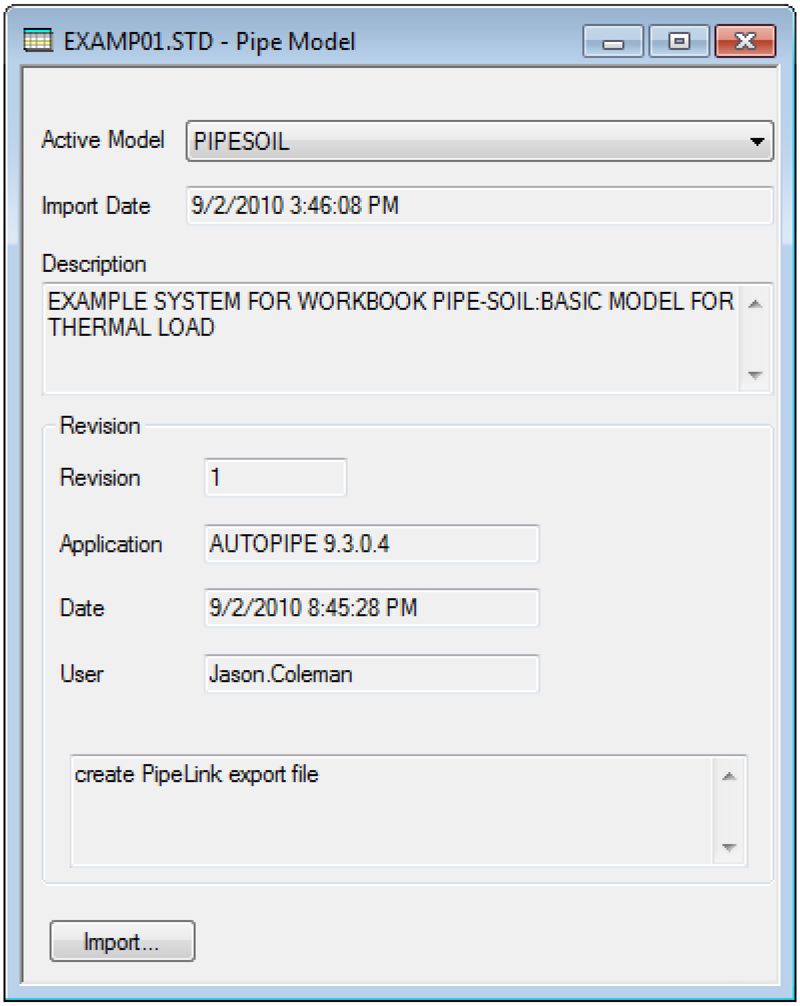
Related Links

- *M. To export model data for use in AutoPIPE* (on page 906)

Pipe Model dialog

Used to select the active pipe model for the Piping workflow. Job and import information for the active model is also displayed here.

Opens when the **Piping** page is selected in the **Piping** workflow.



Active Model

Used to select the current active model. Any actions taken in the Piping mode are done only on the active model.

Note: The view window displays all models, with the inactive models displayed in the color set in the Diagrams dialog Pipe tab (on page 2972). Supports and labels are not displayed for inactive models for clarity.

Import Data, Description, and Revision

The most recent revision is detailed and the date/time when the model was imported into the local database is also displayed. The revising application version number is shown along with the application name.

Import...

Opens a **Open** dialog with the **Files of Type** filter set to PipeLink files (file extension `.pipelink`.), which are piping data files created by the PipeLink utility to transfer data between Bentley AutoPIPE (V8i SELECTseries 2 and greater) and STAAD.Pro (V8i SELECTseries 2 and greater).

Related Links

- *M. To import a piping model* (on page 903)

Support Connection Wizard

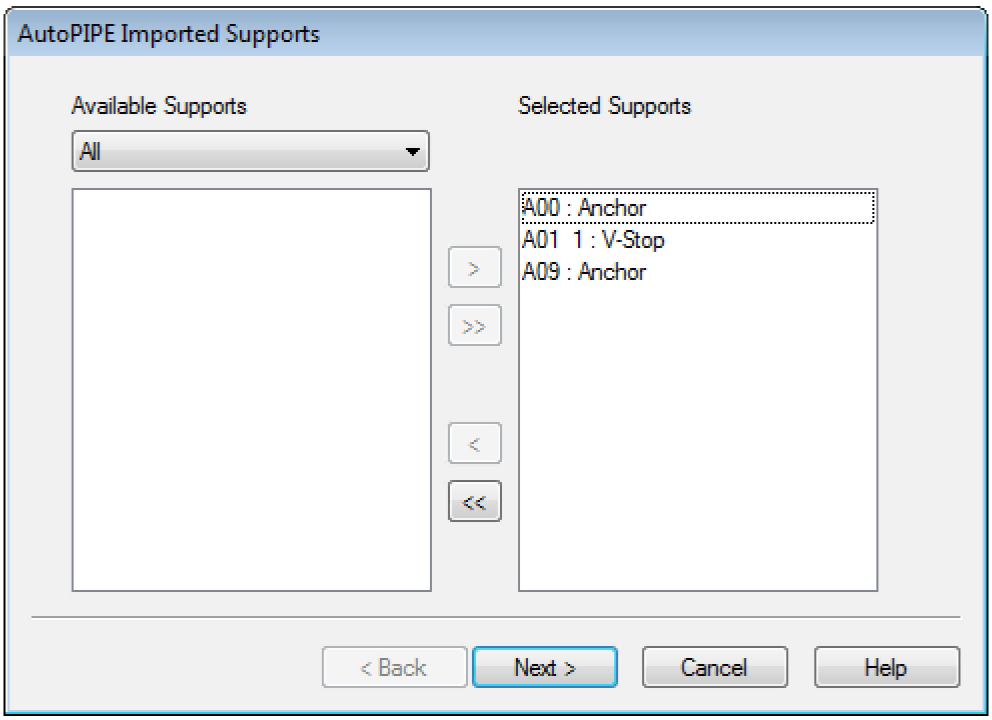
Used to establish logical data links between pipe stress model data imported from a PipeLink file and the current STAAD.Pro model.

Opens when:

- A PipeLink database is successfully imported or reloaded, or
- the **Support Connection Wizard** tool is selected in the **Connection** group on the **Piping** ribbon tab.

AutoPIPE Imported Supports page

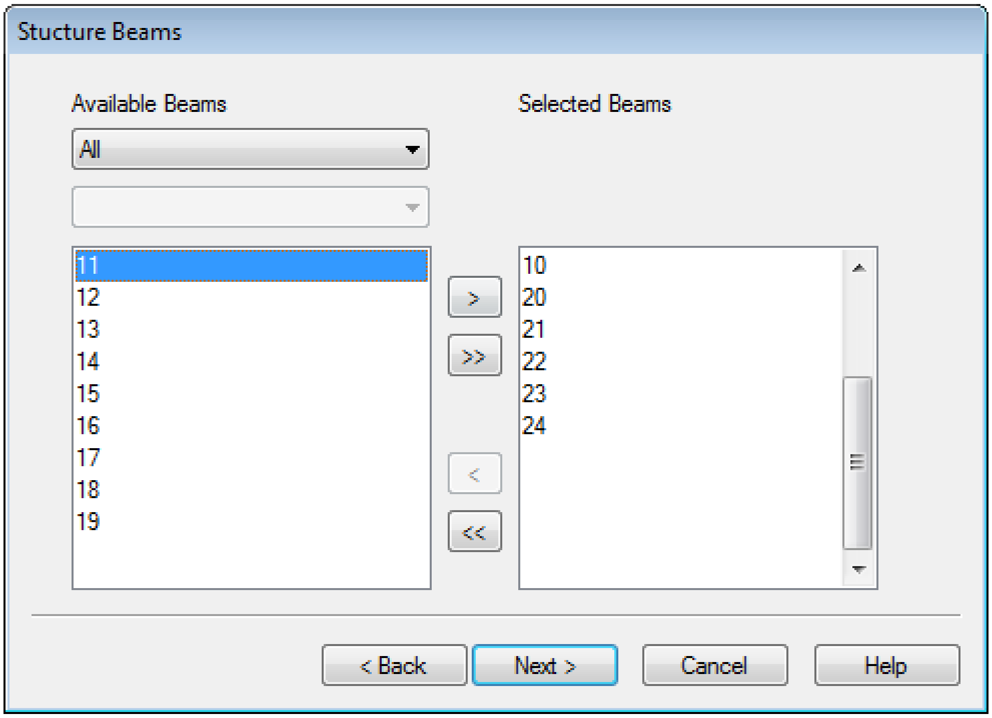
Used to select pipe supports which were defined in the AutoPIPE model.



Available Supports list	Lists all available supports which were exported from the AutoPIPE model. This list can be filtered using the drop-down list. The filtering options are All, Connected, Unconnected, V-Stop, or Anchor (relating to whether a connection to the STAAD.Pro model has been defined). When support type information is available the filter will be expanded to include these as well.
Selected Supports list	All supports added to this list will be imported from the PipeLink data file into the STAAD.Pro Piping mode.

Structure Beams page

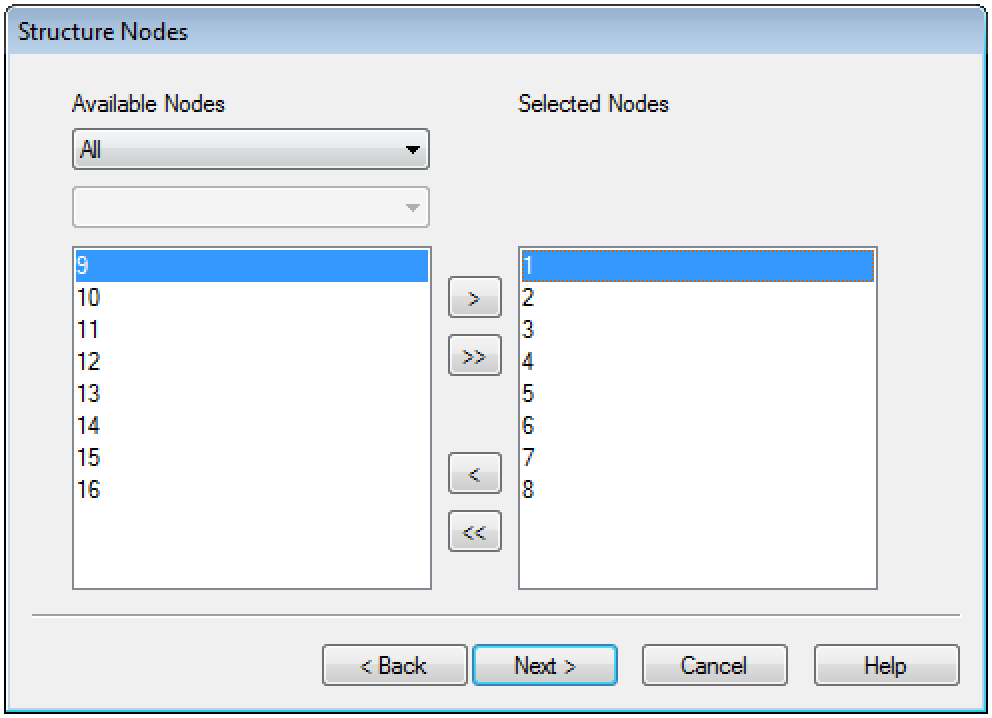
Used to select which structure elements will be used.



Available Beams list	The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group, View, and Property.
Selected Beams list	All members added to this list will be used in the STAAD.Pro Piping mode.

Structure Nodes page

Used to select which structure nodes will be used.

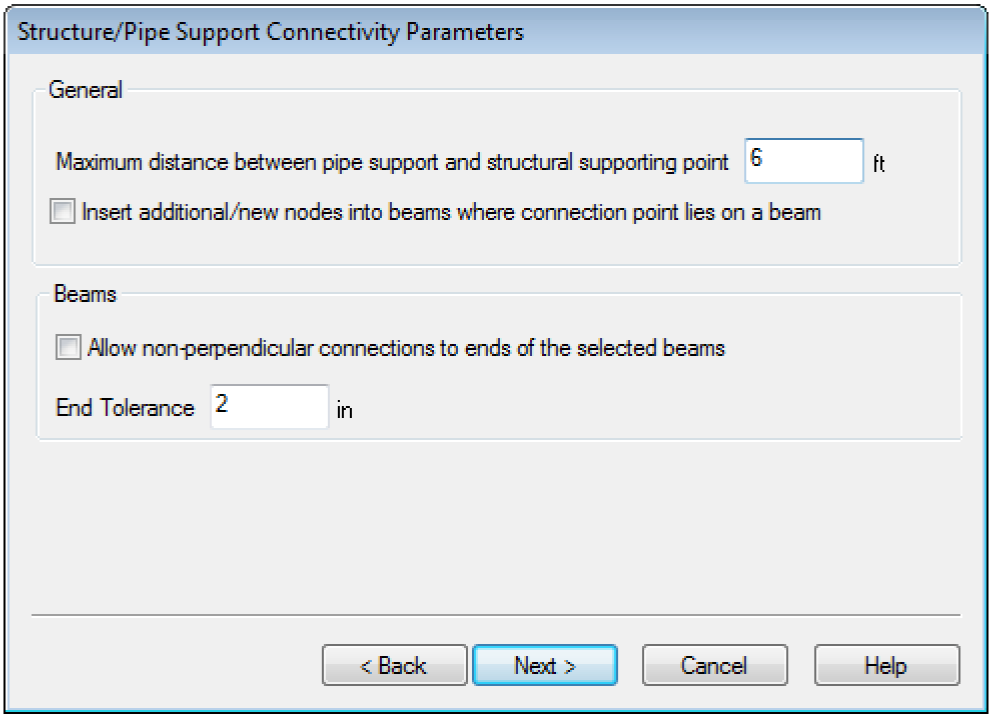


Available Nodes list The filtering options are implemented with two combo boxes, one for the category and one to identify the subset within that category. Available filtering options are All, Group and View.

Selected Nodes list

Structure/Pipe Support Connectivity Parameters page

Used to set parameters for the pipe wizard to establish connections. These parameters provide a means of controlling the results returned by the wizard in attempting to find suitable support points on the structure. The default values of distance-based parameters will depend on the base unit of STAAD.Pro.



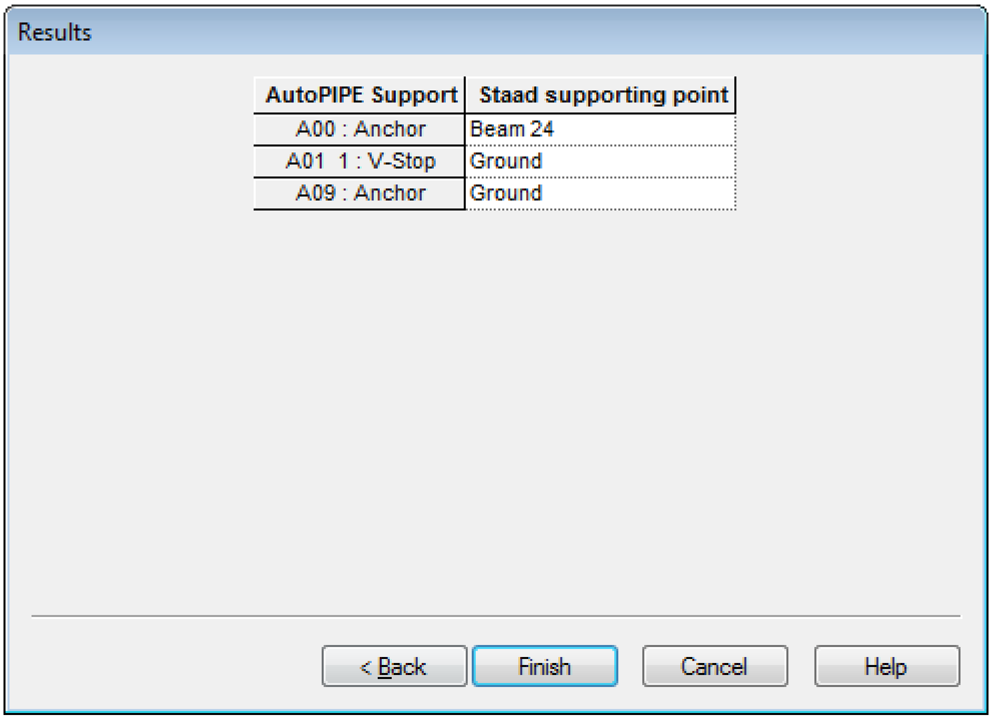
Maximum Distance between pipe support and structural supporting point	This is the sensitivity of the connection wizard when attempting to locate appropriate points on the structure to which to connect piping support nodes. The default value is 2m / 6 feet. Any potential connections beyond Max. Range will be discarded by the wizard.
Insert additional/new nodes into beams where connection points lies on a beam	This parameter effects the connection of the structures rather the point finding algorithm. If set true a new node will be created at each intermediate beam point and the connection made to that rather than to the beam itself. If the algorithm finds new node points within "End Tolerance" of each other then only one new node will be added. This option is not selected by default.
Allow Non-Perpendicular Connection at End Nodes	Select this option to include the beam end nodes in the node search. Otherwise, the end nodes of a given beam will only be considered for connections perpendicular to the beam, unless they have been explicitly added to the node subset. This is only relevant if the node subset does not explicitly include the nodes at the end of members in the beam subset.
End Tolerance	To allow for differences in precision and to avoid very short beam breaks this parameter will determine at what distance from the node the perpendicular will be considered to be at the node itself. The default value is 5 cm / 2 inches.

Results page

Used to review the results of the pipe data import. If changes are necessary, you may go back to previous steps in the wizard to make changes. The connection finding routine runs in two parts. First looking at beams (length of perpendicular from beam) and then looking at nodes (straight line length). The five closest found connectable points are saved to be presented on the results page.

The results will be presented in a table with columns for pipe node id and for the structural item it is to be connected to. This second column will provide a drop list allowing you to choose either **No connection** or one of up to 5 closest items. The items will be listed with the closest at the top. The initial state will be the closest item or **Ground**, if no good matches were found.

Note: Pipe-structure links are not part of the undo system. Nodes created at the end of the wizard will be removed by an undo but the links are not changed.



< Back Return to the previous import wizard page.

Next > Advance to the next page in the import wizard.

Finish (Results Page only) Accepts the import settings and closes the wizard. The pipe data is displayed in the active view window.

Cancel Closes the wizard without importing any pipe data.

Help Opens the STAAD.Pro help window.

Related Links

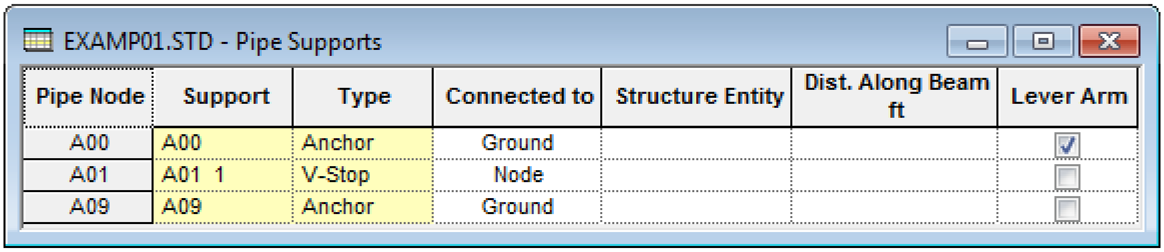
- *M. To import a piping model* (on page 903)
- *M. To use the Support Connection Wizard* (on page 904)

Pipe Supports table

Identifies supported pipe nodes and details each support.

Note: The rows are initially sorted by Pipe Node. Clicking in the headers for Support or Type will sort on that field. As the Pipe Node button is used to select the entire contents it is not possible to return to Pipe Node sorting without using the pop up menu.

Opens when the **Supports** page is selected in the **Piping** workflow.



Tip: Multiple Pipe Nodes (rows) can be selected by holding down the <CTRL> key while making selections.

Pipe Node / Support / Type
Each imported **Pipe Node** is listed in the table, along with the associated **Support** ID and support **Type**. These fields are imported from the pipe stress model and may not be edited.

Connected to
The type of model entity to which the **Pipe Node** is connected. If set to **Ground** (default when the wizard can not establish a connection), then the Pipe Node is assumed as not supported by the structure. You may manually connect the Pipe Node here by selecting either **Beam** or **Node**.

Note: This will be updated automatically if the Connect Supports tool is used.

Structure Entity
The Beam or Node number (as specified in the **Connected to** column) to which this **Pipe Node** is connected.

Dist. Along Beam
(Connected to Beam only) The distance from the start node of the beam where the pipe node is connected, in the current length units.

Lever Arm
Select this option to indicate that the Pipe Node connection is a **Lever Arm** and thus capable of transferring the pipe loads not only as force but also as a moment. If this option is not selected, the loads transferred to STAAD.Pro will not consider the connection rigid for moment transfer (default). This is analogous to setting the connection as “fixed” versus “pinned” in STAAD.Pro for the generated forces applied to the structural model.

For example, this option would be selected for the case of an intermediate pipe anchor where the connection is rigidly attached at both pipe and structure ends or the for case when the connection at the structure end has a moment resisting design.

Right-click Pop-up Menu

Contains tools used for controlling connections and sorting the Pipe Supports table.

Menu Item	Description	Save Effect As...
Set All as Lever	Sets the option for all Pipe Nodes as Lever Arms in the active model.	
Set All as Not Lever	Removes the Lever Arm option for all Pipe Nodes in the active model.	
Set Selection as Lever	Sets the option for the selected Pipe Node(s) as Lever Arms.	

Menu Item	Description	Save Effect As...
Set Selection as Not Lever	Removes the Lever Arm option for the selected Pipe Node(s).	
Disconnect All	Removes the connection for all Pipe Nodes in the active model	
Disconnect Selection	Removes the connection for the currently select Pipe Node(s).	
Sort by Pipe Node	Sorts the table by the Pipe Node name (default)	
Sort by Pipe Support	Sorts the table by the name in the Support column.	Double-clicking the Support column heading
Sort by Support Type	Sorts the table by the name in the Type column.	Double-clicking the Type column heading

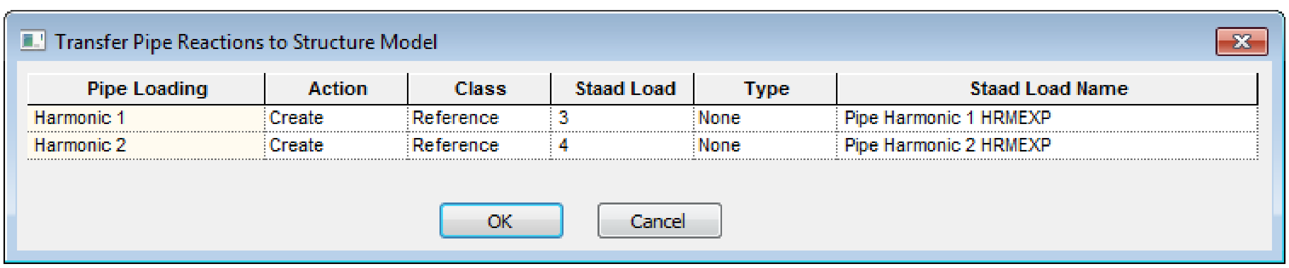
Related Links

- *M. To draw connections between piping supports and the structure* (on page 905)
- *M. To manually specify connections between piping supports and the structure* (on page 905)

Transfer Pipe Reactions to Structure Model dialog

Used to specify STAAD.Pro load data for importing pipe support load data.

Opens when **Piping** > **Transfer Loadings** is selected.



Load data table

The table contains the following columns:

- Pipe Loading - Each pipe support is included as a separate row, with the nature of the load and load number described here.
- Action - Set to Create (for generating a new load) or No Action (which does not result in a new load).

  If the loads are being updated, additional options to Update or Remove existing loads are provided.
- Class - Specify if the load is a Primary load case or Reference load case type.

> **Tip:** You can quickly set all new loads (Action = Create) to either Primary or Reference class by right-clicking anywhere along the column heading and selecting the appropriate option from the pop-up menu.

- STAAD Load - The load case number which will be assigned to the load.
- Type - You can classify the load type, which is used for automatic load combination generation.
- STAAD Load Name - A description load name is generated here. You may enter a different title if needed.

OK Closes the dialog and proceeds with the load data transfer. A message dialog opens to provide you with a summary of the transfer process.

> **Note:** Warnings are displayed in orange and errors are displayed in red.

Cancel Closes the dialog without transferring any load data.

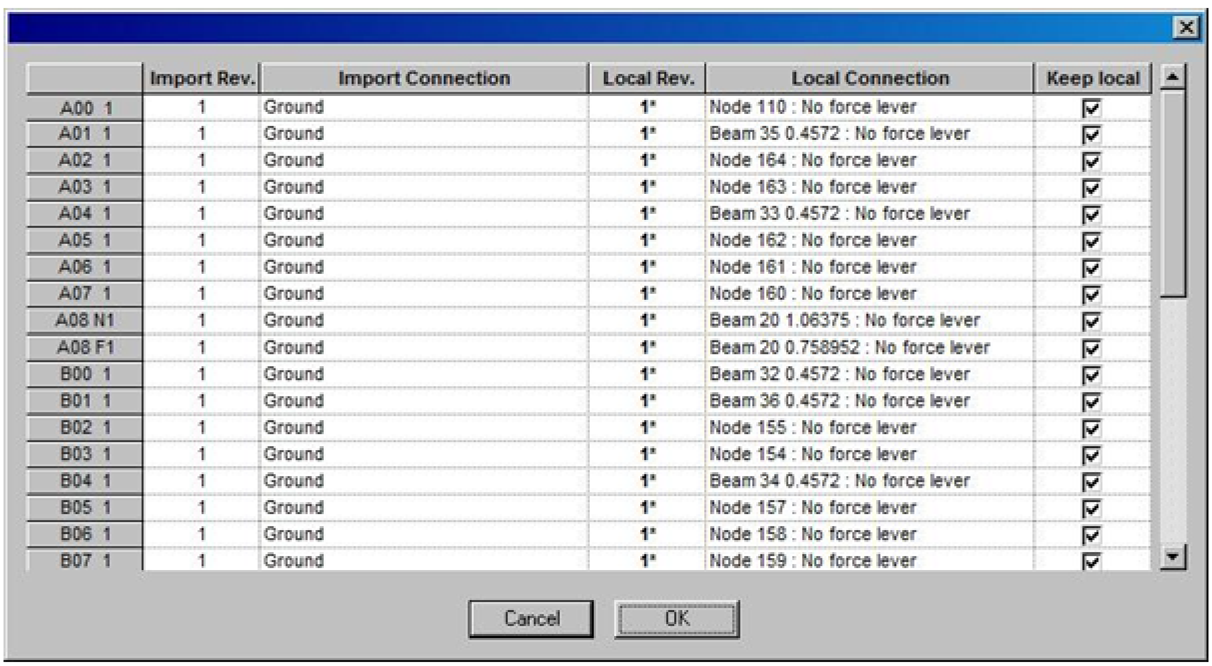
Related Links

- *M. To transfer load data for structural analysis* (on page 906)

Merging Support Connection dialog

Used to determine which of the conflicting connection definitions to use. All data contained within the table is read-only except for the **Keep Local** options for each pipe node.

Opens when clashes are detected during the import or re-import process.



	Import Rev.	Import Connection	Local Rev.	Local Connection	Keep local
A00 1	1	Ground	1*	Node 110 : No force lever	☑
A01 1	1	Ground	1*	Beam 35 0.4572 : No force lever	☑
A02 1	1	Ground	1*	Node 164 : No force lever	☑
A03 1	1	Ground	1*	Node 163 : No force lever	☑
A04 1	1	Ground	1*	Beam 33 0.4572 : No force lever	☑
A05 1	1	Ground	1*	Node 162 : No force lever	☑
A06 1	1	Ground	1*	Node 161 : No force lever	☑
A07 1	1	Ground	1*	Node 160 : No force lever	☑
A08 N1	1	Ground	1*	Beam 20 1.06375 : No force lever	☑
A08 F1	1	Ground	1*	Beam 20 0.758952 : No force lever	☑
B00 1	1	Ground	1*	Beam 32 0.4572 : No force lever	☑
B01 1	1	Ground	1*	Beam 36 0.4572 : No force lever	☑
B02 1	1	Ground	1*	Node 155 : No force lever	☑
B03 1	1	Ground	1*	Node 154 : No force lever	☑
B04 1	1	Ground	1*	Beam 34 0.4572 : No force lever	☑
B05 1	1	Ground	1*	Node 157 : No force lever	☑
B06 1	1	Ground	1*	Node 158 : No force lever	☑
B07 1	1	Ground	1*	Node 159 : No force lever	☑

Pipe Supports data table Select the **Keep Local** option for any pipe node support which you do not wish to be overwritten during the import process.

Cancel Closes the dialog and cancels the import process.

OK	Accepts the clash decisions made in the **Keep Local** choices selected and proceeds with the data import.

Related Links

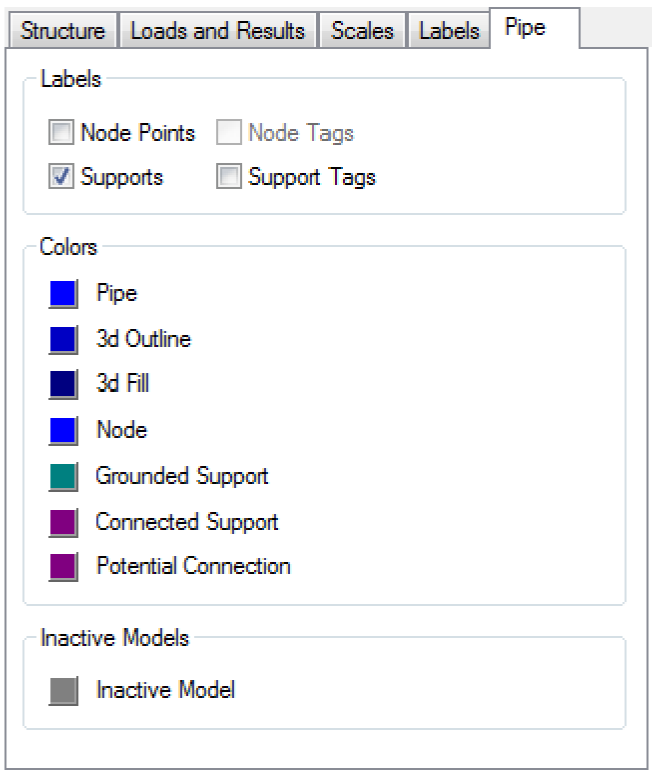
- *M. To import a piping model* (on page 903)

Diagrams dialog Pipe tab

An additional set of view controls is available to you in the Piping mode. This tab is only active when STAAD.Pro is in the Piping mode.

Note: The Structure tab, Loads and Results tab, Scales tab, and Labels tab from the Modeling mode are available in the Piping mode.

Opens when **View** > **Structural Diagrams** is selected (Piping mode only).



Tip: 3D Rendering of Piping models can be viewed using the 3D Rendering features of STAAD.Pro by either toggling on the Full Sections view in the Diagrams dialog or by using the 3D Rendering view.

Labels	Toggles the display of pipe support and node labels.
Colors	Used to select colors for pipe model elements.
Inactive Models	Controls the color for inactive models. No support or node data is displayed for inactive models.

Bridge Deck tab

Table 343: Deck group

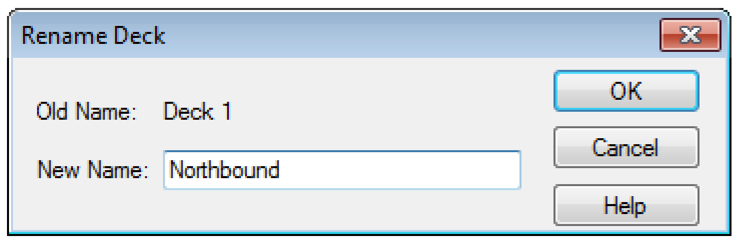
Tool name		Description
 **Create Deck**		Used to create a deck definition from the selected structure geometry. **Note:** When a deck is defined from beam elements, the program will generate a surface of fictitious, triangular plate elements which are used in determining the load distribution. These elements have no inherent stiffness and are not transferred to STAAD.Pro for analysis of the structure. These plate elements may be reviewed by toggling on their display in the Deck tab (on page 2988).
Create Deck >	**Rename**	Opens the **Rename Deck** dialog. 
	Delete	Used to delete the selected deck definition. **Note:** Structure geometry objects (i.e., plates or beams) used in the deck definition are not deleted.
 **Select Deck**		Opens the Select Plates in Deck dialog (on page 2975), which is used to select the members and plates associated with a previously created deck definition.
 **Define Roadway**		Opens the Roadways dialog (on page 2974), which is used to create and manage roadway definitions.

Table 344: Loading group

Tool name		Description
 **Loading >**	**Influence Surface Generator**	Initiates the STAAD Analysis and Design engine to generate influence surfaces for the structure. **Note:** When selected, the **STAAD Analysis and Design** dialog opens to display the progress of the analysis used in influence surface generation. This dialog automatically closes upon completion.
	Influence Diagram	Opens the Influence tab (on page 2987), which is used to control the display of influence diagrams on the defined bridge deck.
	Run Load Generator	Opens the Load Generator Parameters dialog (on page 2981), which is used to select design codes, specify code relevant data, and select points of interest for placing loads.
	View Results file	Opens the results file (file extension `.bva`) in a text editor for your review and for use in post-processing.
	Browse Load Generator Results	
	Create Loading in Staad Model	Used to create a primary load cases in the current STAAD input file for each action requested in the Load Generator Parameters dialog (on page 2981). These load cases can then be used in the same way any other static load case would be used for analysis and design.

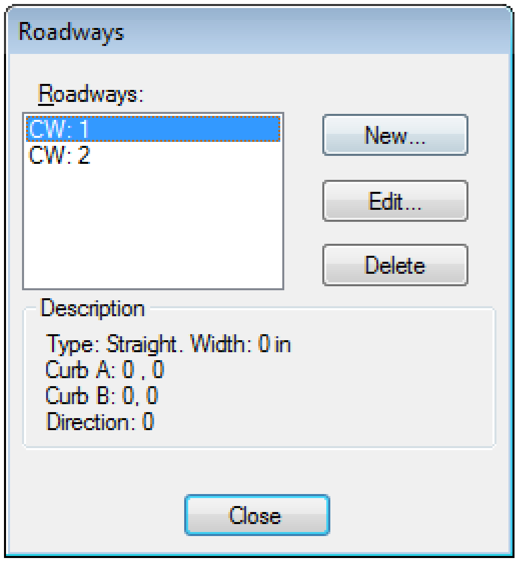
Table 345: Vehicle group

Tool name	Description
 **Database**	Opens the Vehicle Database dialog (on page 2989), which is used to display code specified vehicle load data and to create user-defined loads.

Roadways dialog

Used to create and manage roadway definitions. This should be the clear roadway width between curbs or other barriers. The program will determine the number of traffic and design lanes available in this width.

Opens when **Deck > Define Roadways** is selected.



Roadways list A list of each roadway definition for the selected deck is contained here. Selecting a Roadway definition in the list will display the characteristic parameters in the Description block below.

New... Opens the Roadways dialog (on page 2974), which is used to parametrically define straight, curved, or custom roadways.

Edit... Opens the Roadways dialog (on page 2974), which is used to edit the selected roadway definition.

Delete Deletes the selected Roadway definition.

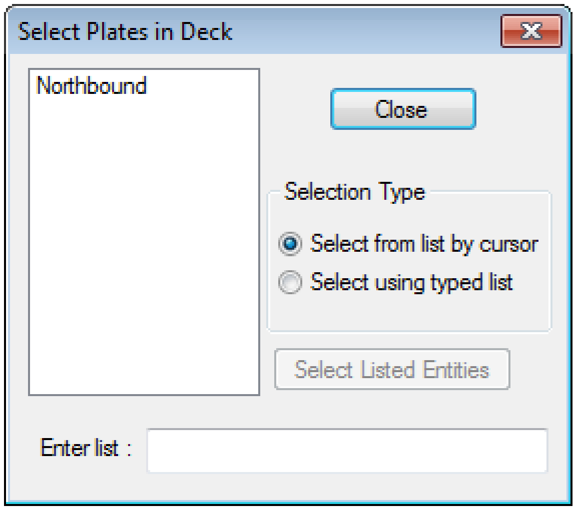
> **Caution:** No confirmation is required to delete a Roadway definition. This action cannot be undone.

Close Closes the Roadways dialog.

Select Plates in Deck dialog

Used to select the members and plates associated with a previously created deck definition.

Opens when **Deck** > **Select Deck** is selected.

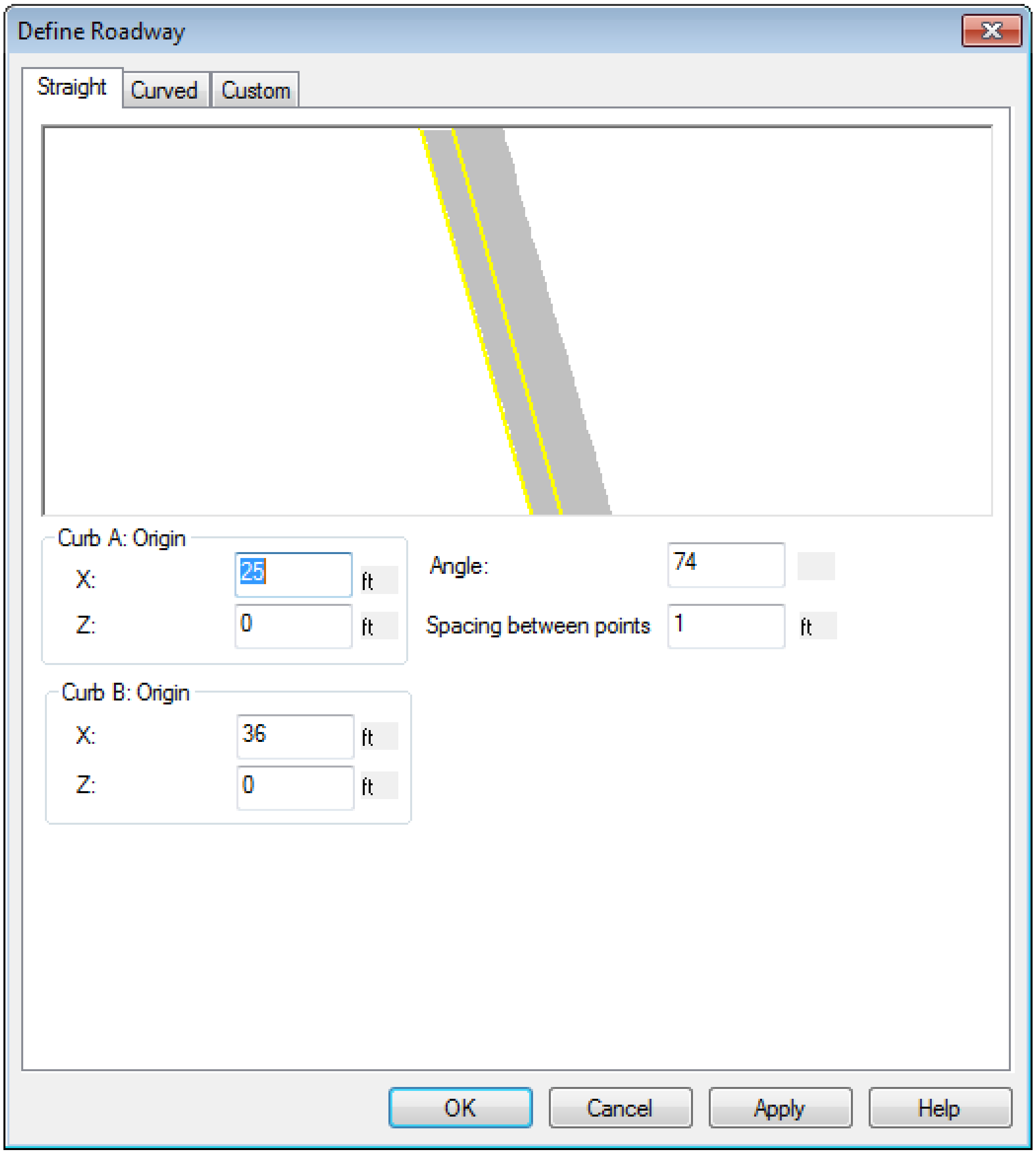


decks list	Lists all deck definitions in the current STAAD project.
Close	Closes the dialog.
Selection Type	Used to either **Select from list by cursor**, which allows you to select the deck by name from the decks list, or to **Select using type list**, which allows you to type a list of deck names in the **Enter list**.
Select Listed Entities	Used with the **Select using type list** option to select the entities specified in the **Enter list**.
Enter list	Specify the names of decks here.

Define Roadway dialog

Used to define roadways for vehicle loading on a selected deck. Either Straight, Curved, or Custom roadways may be defined.

Opens when the **New** or **Edit** buttons are clicked in the Define Roadway dialog (on page 2976).



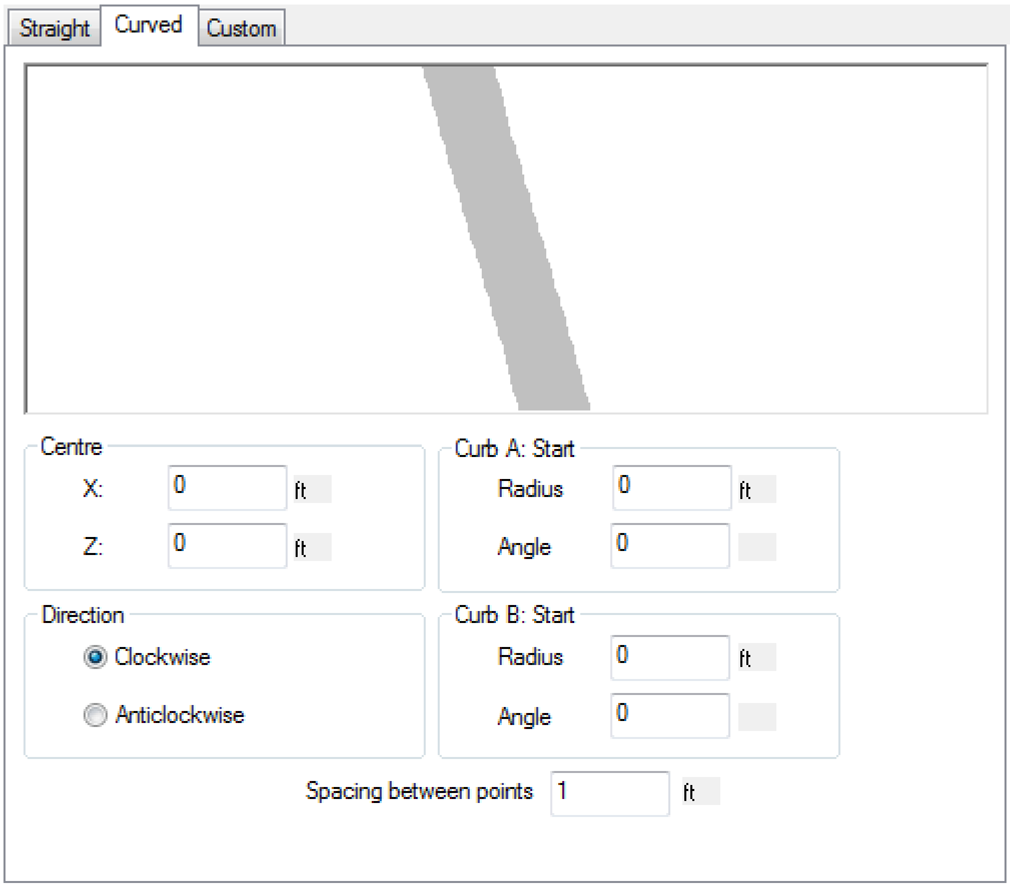
Straight tab

Used to parametrically define a straight roadway.

Curb A / B Origin	Curb A is the closes to the origin, with Curb B being the other side of the roadway. Dimensions are all taken from the origin (i.e., the distance to Curb B is not taken from Curb A).
Angle	The angle of the roadway taken clockwise from the positive X axis.
Spacing between points	Spacing ... in current length units.

Curved tab

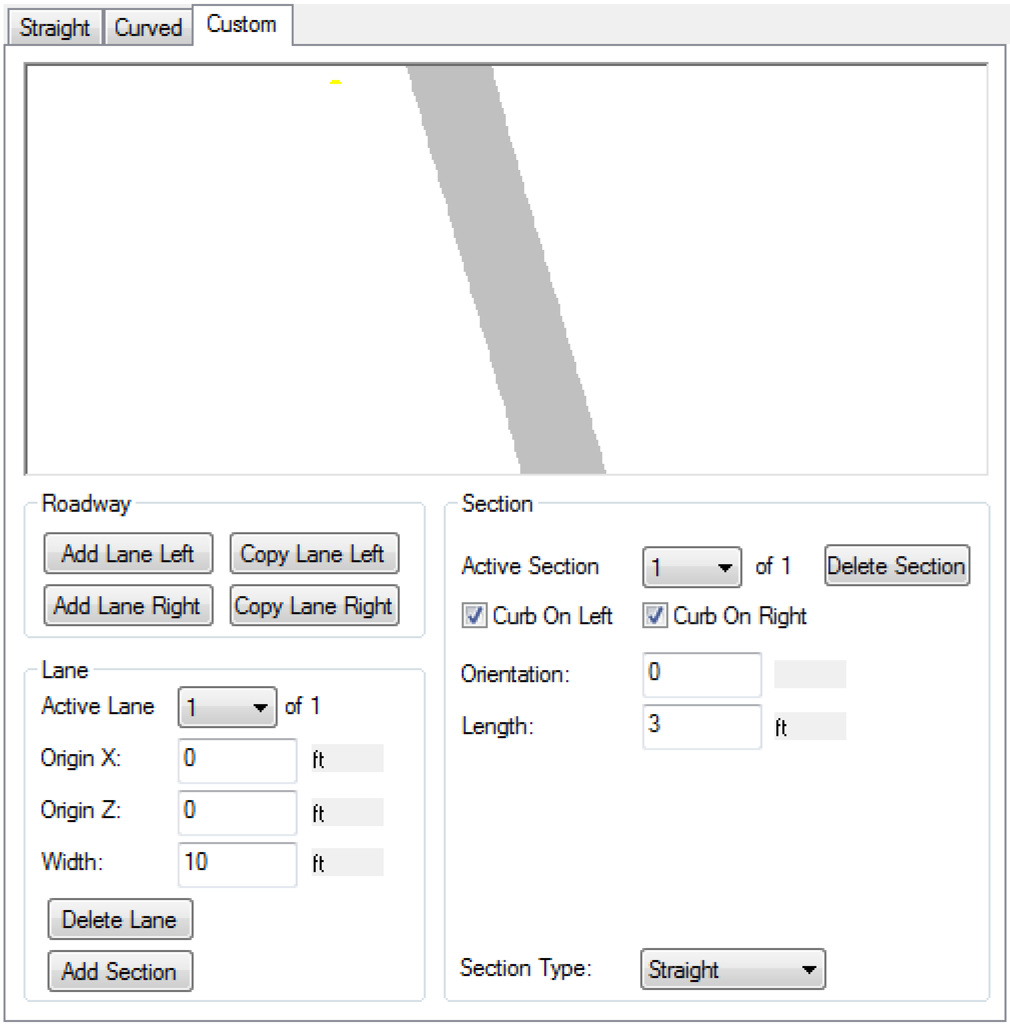
Used to parametrically define a curved roadway.



Center	Define the center of a circular arc.
Direction	Select if the arc sweep is defined as Clockwise or Anticlockwise (Counter-clockwise) from ...
Radius	
Angle	
Spacing between points	Spacing ... in current length units.

Custom tab

Used to define a generic roadway section.



Add Lane Left

Copy Lane Left

Add Lane Right

Copy Lane Right

Active Lane Select the lane you wish to create or edit from the drop down list. The total number of lanes present is also displayed.

Origin X / Z

Width

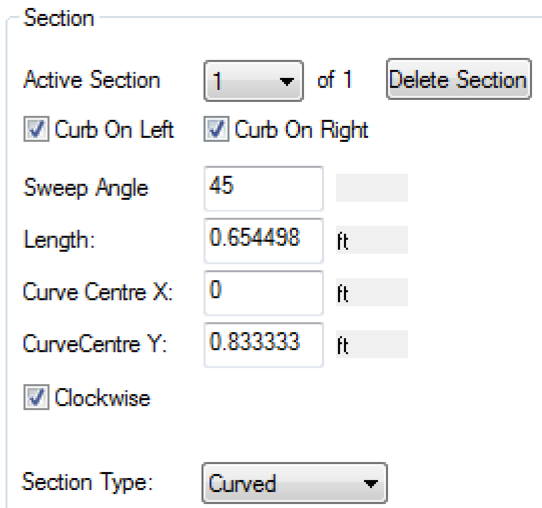
Delete Lane

Add Section

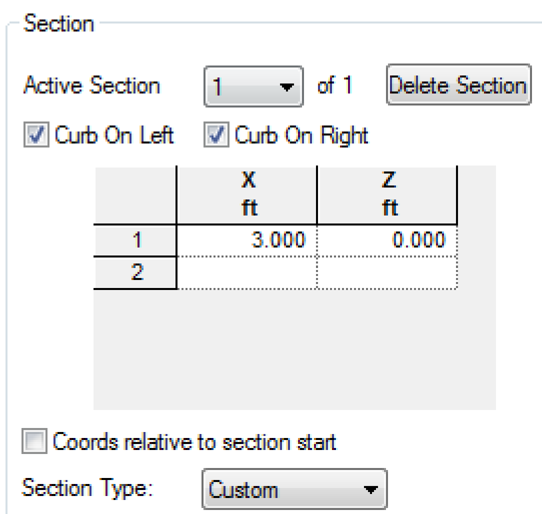
Section Type Select if the roadway segment is **Straight**, **Curved**, or **Custom**. The section panel updates to allow entering parameters for this selection.

- Straight section

    Active Section
    Delete Section
    Curb on Left / Right
    Select either or both of these options to ...
    Orientation
    Length
- Curved section



- Custom section



OK Closes the dialog and creates a new roadway or updates changes made to an existing roadway.

Cancel Closes the dialog without saving any changes or creating a roadway definition.

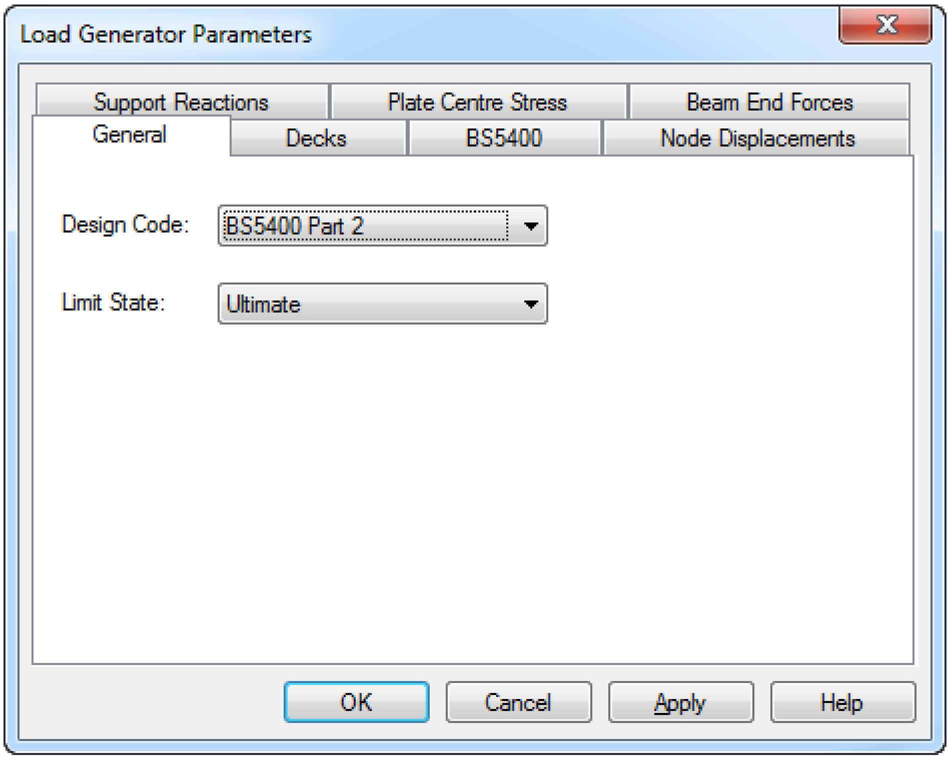
Apply Updates the current roadway definition with any changes made in the dialog.

Help Opens the STAAD.Pro help window.

Load Generator Parameters dialog

Used to select design codes, specify code relevant data, and select points of interest for placing loads.

Opens when **Loading** > **Run Load Generator** is selected.



General tab

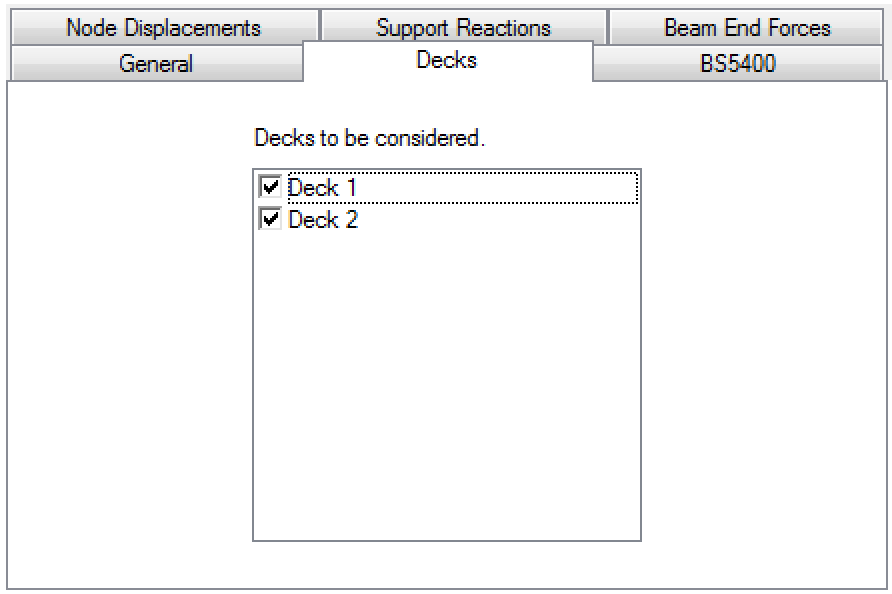
Design Code Select the specification from which the loadings are to be taken.

> **Note:** The ***<code>*** tab updates accordingly with this choice.

Limit State Specify if the vehicle loads are to be used with the Ultimate or Serviceability limit state.

Decks tab

Used to select the previously defined decks to be considered for loading. A check is placed in the box associated each deck name to be considered.



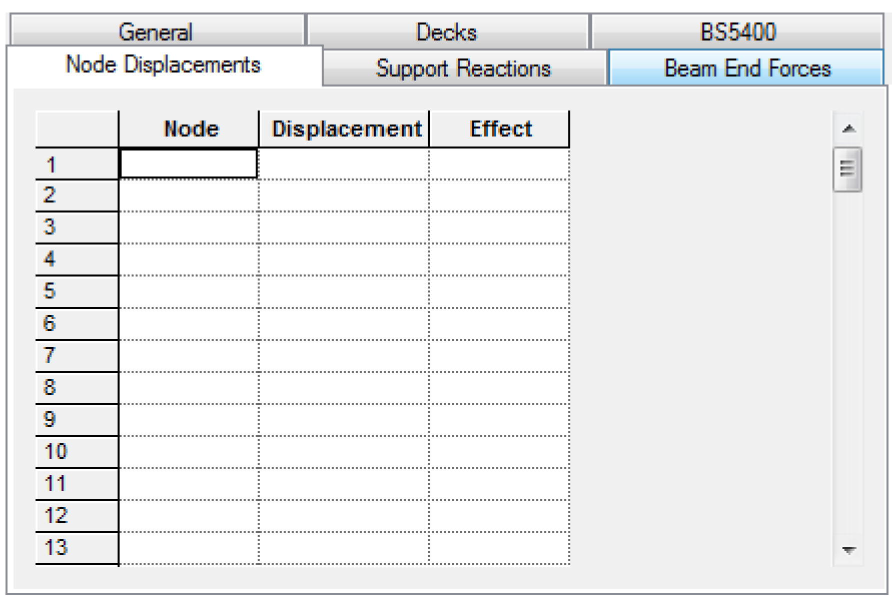
<code> tab

Used to input code-specific parameters. This tab dynamically updates based on the Design code selection made on the General tab.

Note: The following pages provide details on these code-specific parameters.

Node Displacements / Support Reactions / Beam End Forces/ Plate Center Stress tabs

Used to specify a structural object, action, and effect as a point of interest for influence surfaces which determine the placement of the design loads on the structure to achieve maximum (or minimum) action.



Node / Beam / Plate Specify the number of the structural object to act as a point of interest. The point of interest is the location on the structure the resulting action at which will be used to determine the placement of the loads.

Displacement / Support Reaction / Stress Force The action of interest. This is usually the direction of the displacement, force, moment, or stress.

In the case of beam end forces, you must select the member end as well as the direction of force or moment.

In the case of plate stresses, additional options for the maximum combined effect (von Mises or Principle stresses) are also available.

Effect Select either the maximum (+ve) or minimum (-ve) effect at the point of interest due to the selected action.

OK Accepts the specified parameters and closes the dialog.

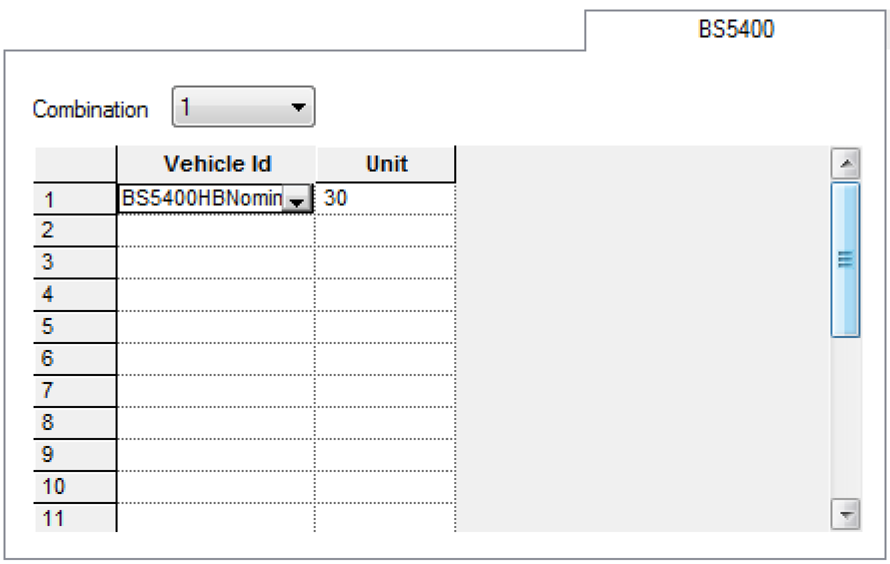
Cancel Closes the dialog without adding load generation parameters.

Apply Applies the parameters from the current tab.

Help Opens the STAAD.Pro help window.

BS 5400 Specific Parameters

Used to input parameters for loading per Cl.6.3.1 of British Standard 5400 “Steel, concrete and composite bridges, Part 2. Specification for loads”.



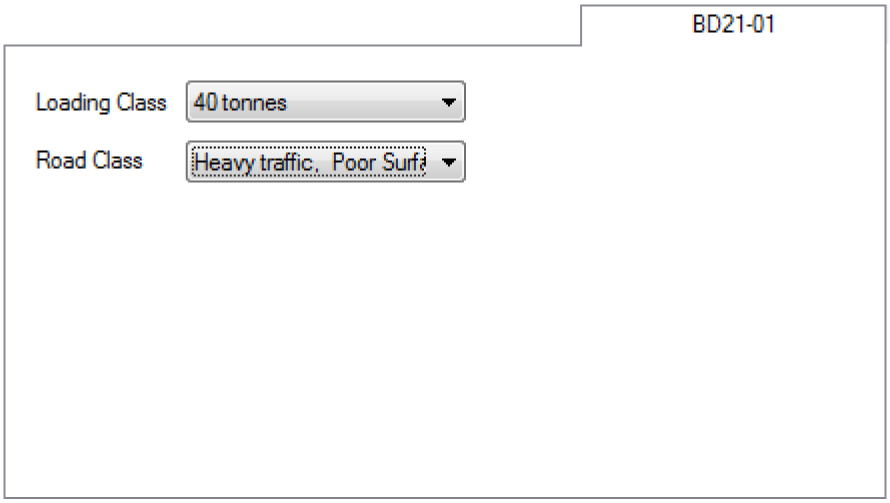
Combination Select the load combination number to be considered which include highway bridge live loading (BS 5400:Part 2 table 1). Refer to Cl.6.2.7 and 6.3.4 for the load factors used in the program for the load combinations.

Vehicle table Select the design vehicle to use. The default is the HB vehicle per Cl.6.3.1. User defined vehicles may also be selected.

Specify the number of units of 10 kN load per axle to be used for type HB to be used per Cl.6.3.

BD21/01 Specific Parameters

Used to input parameters for loading per BD21/01 “The Assessment of Highway Bridges and Structures”.

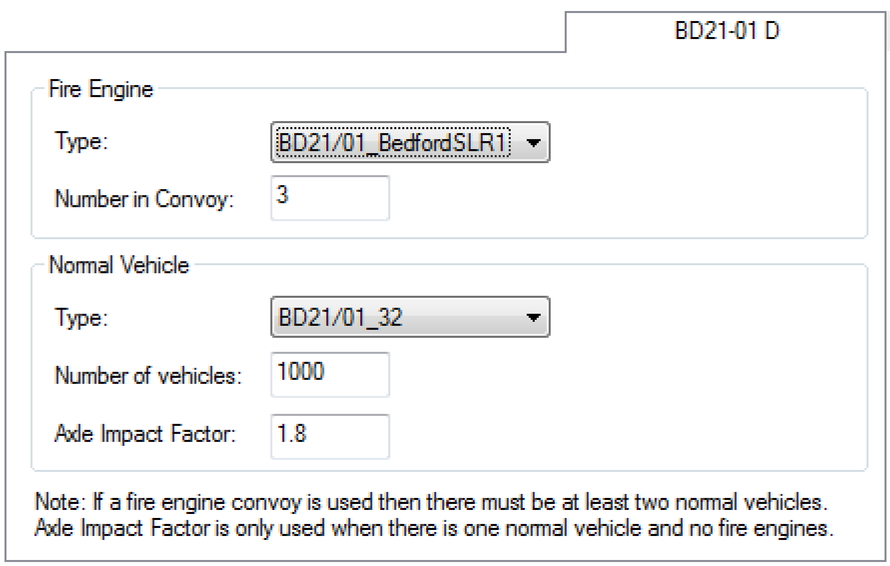


Loading Class Select the Assessment Loading vehicle from this list per Cl.5.12 through 5.17 of BD21/01

Road Class Select the bridge situation in terms of road surface characteristics and traffic level per Cl.5.22 of BD21/01.

BD21/01 Annex D Specific Parameters

Used to input parameters for loading per Annex D: Loading From Vehicles of BD21/01 Annex D. This code is used for traffic on cross girders and transversely spanning slabs.



Fire Engine
Select the make of fire engine from Table E1 of BD21/01 Annex E as the **Type**. Specify the **Number in Convey** to indicate the number of fire engines (all same type) with which to simultaneously load the bridge.

Nominal Vehicle
Select the nominal vehicle Type from Tables D1 and D2 of BD21/01 Annex D.

Vehicle nomenclature:

```
BD21/01_WT[_#][dir]
```

Where:

WT Either the vehicle gross weight value (Table D1 vehciles) or the vehicle reference (Table D2 vehicles).

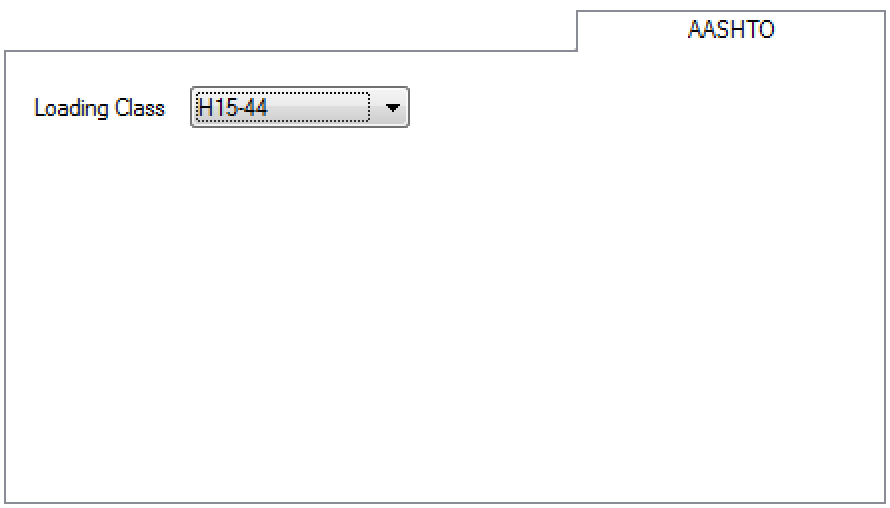
Separate id number for vehicles which are listed more than once in the table.

div a or b signifies a vehicle in which the W2 and W3 loads have exchanged position.

Specify the **Number of vehicles** to indicate the number of nominal vehicles (all same type) with which to simultaneously load the bridge. Specify an Axle Impact Factor for use with a single vehicle per Cl.D.3(a).

AASHTO Specific Parameters

Used to input parameters for loading per “Standard Specification for Highway Bridges”, AASTHTO ASD / LFD.

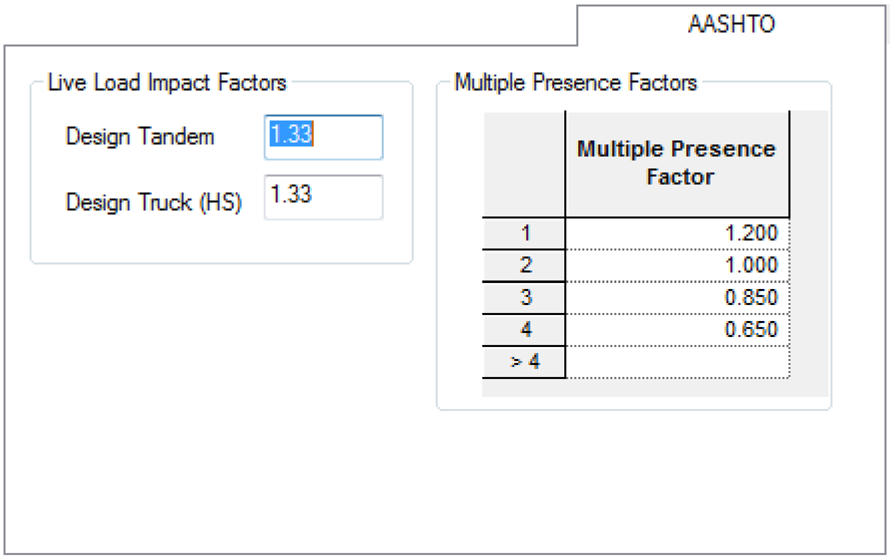


Loading Class
Select the nominal vehicle load to use per Cl.3.7. All vehicles in the Vehicle Database dialog are listed here, with the following being provided in the AASHTO ASD/LFD code:

- H15-44
- H20-44
- HS15-44
- HS20-44

AASHTO LRFD Specific Parameters

Used to input parameters for loading per the AASTHTO LRFD Bridge Design Specifications.



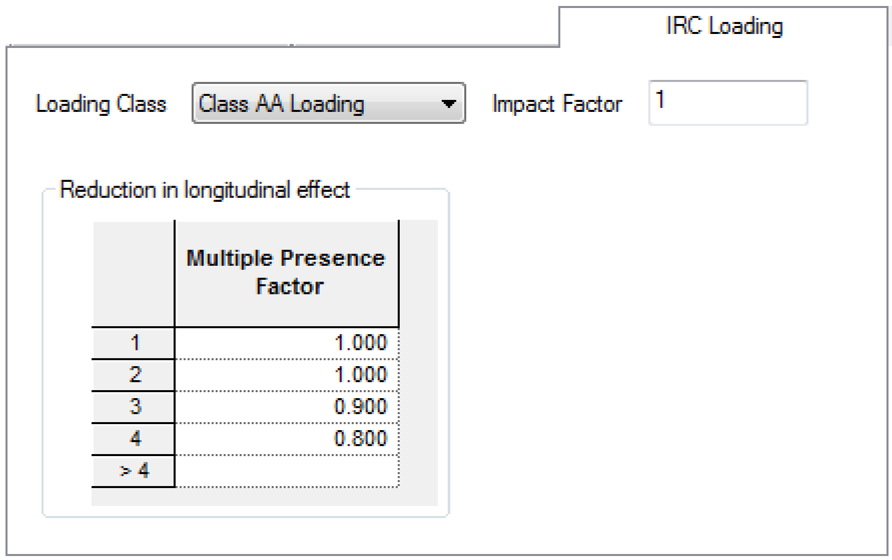
Live Load Impact Factors — Multiplier accounting for imperfection in road surface resulting in additional dynamic loading. These factors vary depending upon the bridge component being designed. The default value of 1.33 is specified for bridge components (other than deck joints) for limit states other than fatigue or fracture (AASHTO LRFD table 3.6.2.1-1). Separate values may be specified for the **Design Tandem** vehicle and the **Design Truck (HS)**.

Multiple Presence Factors — Multipliers for design vehicles based on the number of design lanes in the transverse section of a roadway. The multiple presence factors, m, account for the statistical improbability of three or more adjacent lanes being loaded simultaneously with the design vehicle. The default values are taken from AASHTO LRFD table 3.6.1.1.2-1.

IRC Specific Parameters

Used to input parameters for loading per IRC Chapter 6.

The IRC rules have been implemented such that the program will select the appropriate live loads, number of design lanes, and generate all the possible load combinations as stated for a specified roadway width.



Loading Class	Select the vehicles to be used for this load generation.
Impact Factor	Specify a value to account for dynamic effects resulting for road surface imperfections.
Reduction in longitudinal effect	Contains a table of Multiple Presence Factor for the number of lanes for a specified roadway.

Related Links

- *AD.2007-07.3.5 Transverse IRC Loading in STAAD.Beava* (on page 210)

Diagrams dialog

The diagrams dialog contains additional tabs in the Bridge Deck mode used to control the display of decks and influence diagrams.

Opens when **View > Structural Diagrams** is selected.

Influence tab

Used to control the display of influence diagrams on the defined bridge deck as displayed on the **Deck | Deck Definition** page.

Opens when:

- **View** > **Structure Diagrams** is selected, or
- **Loading > Influence Diagrams** is selected.

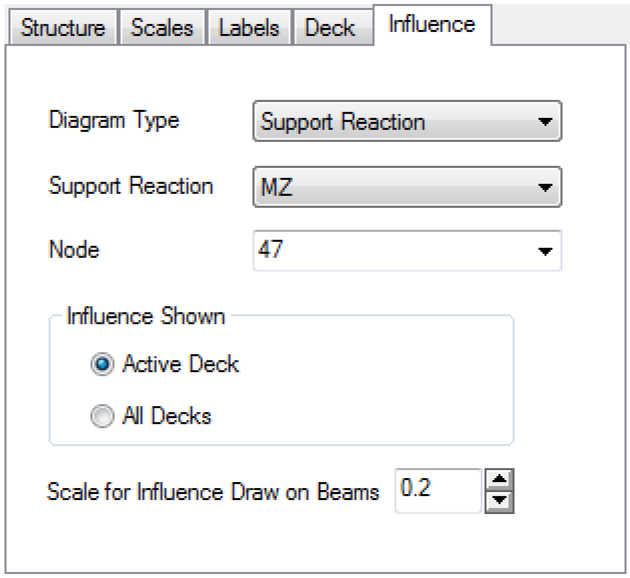


Diagram Type Select the type of action you wish to display in a contour plot the deck. The <action type> and <structural object> fields update with the selected type.

<action type> Select the action and direction for the action.

<structural object> Select the node, beam, or plate for which the action will be displayed.

Influence Show Select if the influence is drawn for **All Decks** defined in the Bridge Deck mode or only the **Active Deck**.

Tip: The Active Deck is the selected deck in the View toolbar.

Scale for Influence Draw on Beams Adjusts the scale for influence curves drawn onto beam objects.

Deck tab

Used to control the display of bridge decks and loads on the **Deck | Deck Definition** page.

Opens when:

- **View** > **Structure Diagrams** is selected, or
- **Loading > Influence Diagrams** is selected.

Roadways This option will display the defined roadways and the design lanes, as determined by the program. Different colors are available for the Active (currently selected) and Inactive elements.

Results These options are used to display the Loads and Vehicles on the structure. Different colors are available for the Active (currently selected) and Inactive loads.

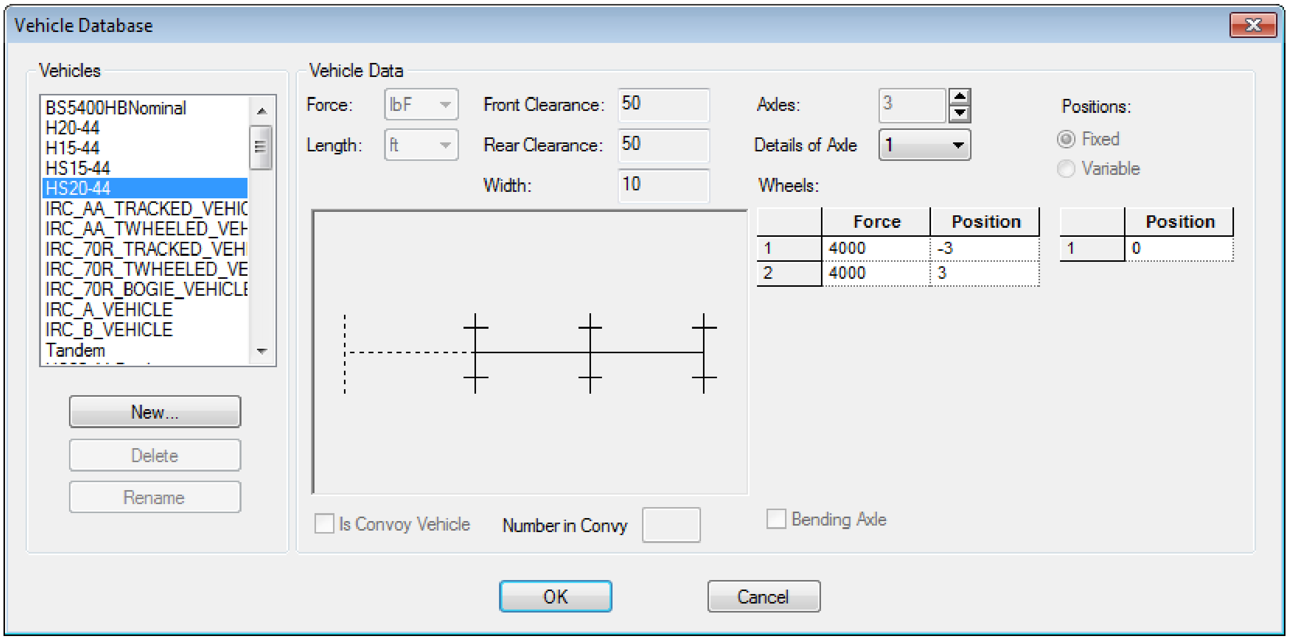
Triangulation Used to display the triangular, pseudo-plate elements created when a deck is defined from beam members only.

Vehicle Database dialog

Used to display code specified vehicle load data and to create user-defined loads.

Note: Vehicles included in Bridge Deck may not be edited or deleted.

Opens when **Vehicle > Database** is selected.



Vehicles list — All vehicle definitions in Bridge Deck are listed here. User-defined vehicles are added at the bottom of this list. Select a user-defined vehicle to make any changes.

New ... — Opens the **Add New Vehicle** dialog, which is used to specify a name for the vehicle load to be added.

Delete — (User-defined loads only) Deletes the currently selected vehicle definition.

Caution: There is no confirmation of this action and this action cannot be undone.

Rename — (User-defined loads only) Opens a Rename Vehicle dialog, which is used to specify a new name for the selected vehicle definition.

Vehicle Data — The dialog contains a set of simple parameters which can be used to define virtually any vehicle load.

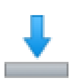
- Force / Length - Select the units for use when specifying **Force** (lbF or kN) and **Length** (m, mm, or ft) values.
- Front / Rear Clearance - Specify the clear distance between the first and last axle, respectively, to another vehicle or load within the same design lane.
- Width - Specify the total width (half on either side of the vehicle center line) for placement within a design lane.

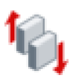
- A plan diagram displays the configuration of the vehicle load. Axle 1 is the right-most solid line, with subsequent axles to the left. Variable and multiple Fixed axle Positions are shown in dashed lines.
- Is Convoy Vehicle - Select this option to signify to the program that this vehicle is to applied repeatedly along the design lane, using the Front and Rear Clearances specified.
- Number in Convoy - Specify the number of vehicles present in the convoy.
- Axles - Use the combo box to select the number of axles (fixed or variable) on the vehicle.
- Details of Axle - Select the axle (up to the number of Axles specified) for which the current load and position data is entered.
- Wheels - Use the table to enter the Force (wheel weight) and Position (distance from vehicle center line), in the selected units, for the currently selected axle.
- Bending Axle - Select this option ...
- Positions - Select if the current axle is a **Fixed** distance (or a number of different fixed distances) from the previous axle or if the distance is **Variable**. For **Fixed** positions, the table below allows for one or more set distances between the current and previous axle. For **Variable** positions, a **Min**imum and **Max**imum distance must be specified, along with an **Increment** value for generating a finite number of positions.

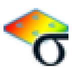
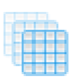
OK	Accepts changes made to the currently selected vehicle (user-defined only) and closes the dialog.
Cancel	Closes the dialog without saving any changes.

Results tab

Table 346: View Results group

Tool name	Description
Load	Select the active load case, load combination, or load envelope from the pop-up dialog.
 **View Loading Diagram**	Click to toggle the display of the current load case on the structure.
 **Deflection**	Displays the deflected shape on the structure using nodal displacements.

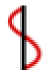
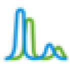
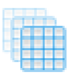
Tool name	Description
 **Displacement**	Displays the nodal and beam sectional displacements. **Tip:** This will show a more accurate representation of the deflected shape compared to the **Deflection** tool, but may take longer to render for large structures.
 **Utilization Ratio**	Displays the design code based utilization ratio, for code checks and member selection operations, in annotated form on the structure diagram. **Note:** The colors representing the pass or fail status of a member can be controlled using the **Color Manager** dialog and the **Diagrams** dialog Design Results tab.
 **FX**	Displays the axial force diagram on the structure.
 **FY**	Displays the shear force Y diagram on the structure.
 **FX**	Displays the shear force Z diagram on the structure.
 **MX**	Displays the torsion diagram on the structure.
 **MY**	Displays the bending moment Y diagram on the structure.

Tool name		Description
 **MZ**		Displays the bending moment Z diagram on the structure.
 **Beam Stress**		Displays the beam stress diagrams on the structure.
 **Plate Stress**		Displays plate stress contours on the structure.
 **Solid Stress**		Displays solid stress contours on the structure.
 **Layouts >**	**Displacement**	Displays the nodal and beam sectional displacements, along with the P. Node Displacements table (on page 2002) and P. Beam Relative Displacement Detail table (on page 2003). This has the same effect as selecting the **Displacements** page in the Postprocessing workflow.
	Reaction	Displays the support reactions, along with the P. Support Reactions table (on page 2003) and P. Statics Check Results table (on page 2004). This has the same effect as selecting the **Reactions** page in the Postprocessing workflow.
	Instability	Displays unstable joints if any were detected in the analysis along with the P. Unstable Joints table (on page 2004). **Note:** This tool is only active if instabilities are detected during the analysis.
	Base Pressure	Displays the soil pressure at support node locations for slabs consisting of plate elements generated using either the elastic mat or plate mat commands along with the P. Base Pressure table (on page 2004). **Note:** This tool is only active if `ELASTIC MAT` or `PLATE MAT` commands are present in the input file.

Tool name		Description
	Beam Forces	Displays the force diagram on the structure for the selected load case along with the P. Beam End Forces table (on page 2005) and **Beam Force Detail** table. This has the same effect as selecting the **Beam Results** page in the Postprocessing workflow.
	Beam Stress	Displays the member stress diagram on the structure (same as the **Beam Stress** tool), along with the **3D Beam Stress Contour** view and P. Beam Combined Axial and Bending Stress table (on page 2007).
	Utilization	Displays the design code based utilization ratio, for code checks and member selection operations, in annotated form on the structure diagram (same as the **Utilization** tool) along with the P. Design Results table (on page 2008).
	Graphs	Displays the current loading on the structure in the main view along with a set of three Beam Graph windows which display force or bending diagrams for the currently selected member.
	Plate Stress	Displays the graphical plate stress contour diagram superimposed on the elements along with the P. Plate Corner Stress table (on page 2009) and P. Plate Center Stress table (on page 2009). This item will also open the Diagrams dialog to the Plate Stress Contour tab.
	Results Along Line	Opens the **Results Along Line** dialog along with the line graph and **Result along selected line** table.
	Solid Stress	Displays the graphical Solid Stress Contour diagram superimposed on the elements along with the P. Solid Corner Stress table (on page 2010) and the P. Solid Center Stress table (on page 2011). This has the same effect as selecting the **Solid Results** page in the Postprocessing workflow. This item will also open the Diagrams dialog to the Solid Stress Contour tab.
	Node Displacement	Displays the deflected shape for the selected load case and load step, along with the **Node Displacements Curve** graph and **Node Displacement** table.
	Beam Forces	Opens the Load Level vs Section Forces diagram and **Beam Force Detail** table.

Table 347: Dynamics group

Tool name	Description
Mode	Select the mode number from the drop-down list.

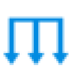
<table>
<tr><th colspan="2">Tool name</th><th>Description</th></tr>
<tr><td colspan="2">
Mode Shape</td><td>Displays the mode shape diagrams for individual modes.</td></tr>
<tr><td colspan="2">
Relative Response</td><td></td></tr>
<tr><td colspan="2">
Time Steps</td><td>Opens a drop-down control which allows you to move through time history steps using a slider control or by typing the time directly.</td></tr>
<tr><td rowspan="7">
Layouts ></td><td>Dynamic - Mode Shape</td><td>Displays the mode shape diagram on the structure, along with a table of mode shapes. This has the same effect as selecting the Dynamics page in the Postprocessing workflow.</td></tr>
<tr><td>Dynamic - Displacement</td><td>Displays the time-displacement graphs for a selected node or node group in the structure.</td></tr>
<tr><td>Dynamic - Velocity</td><td>Displays the time-velocity graphs for a selected node or node group in the structure.</td></tr>
<tr><td>Dynamic - Acceleration</td><td>Displays the time-acceleration graphs for a selected node or node group in the structure.</td></tr>
<tr><td>Dynamic - Floor Spectrum</td><td>Displays the Response Spectrum graph and Response Spectrum table.</td></tr>
<tr><td>Pushover - Loads</td><td>Opens the (Pushover) Load Values table (on page 2012), which displays the nodal load values and total base shear for the selected load step, along with the Capacity Curve table.
The magnitude and direction of the loads in the current load step are displayed in the view window.</td></tr>
<tr><td>Pushover - Graphs</td><td>Opens the Capacity Curve graph (on page 2013), which displays displacement vs. force for the pushover analysis.
The Capacity Curve table contains the roof or control joint displacement and base shear values recorded for each load step.</td></tr>
</table>

Tool name		Description
	Pushover - Node Results	Displays nodal displacements and rotations in the (Pushover) Node Displacements table (on page 2014) and the support reactions in the (Pushover) Support Reactions table (on page 2015) for the selected load step. The deflected shape is overlaid on the structure in the view window.
	Pushover - Beam Results	Opens the (Pushover) Beam Hinge Results table (on page 2016) which displays the status of the beam and hinge conditions along each beam's length. The (Pushover) Beam Force Details table (on page 2017) displays the cross-sectional forces and moments at sections along each beam's length are displayed for the selected load step. The status of beams and hinges are displayed in the view window.
	Steady State - Displacement	Displays the frequency-displacement graphs in the global directions for a steady state analysis.
	Steady State - Velocity	Displays the frequency-velocity graphs in the global directions for a steady state analysis.
	Steady State - Acceleration	Displays the frequency-acceleration graphs in the global directions for a steady state analysis.
	Buckling	Opens the **Buckling Factors** table, which displays the calculated buckling factors from a buckling analysis, along with the **Buckling Modes** table.

Table 348: Animation group

Tool name	Description
 **Animation**	Opens the **Diagrams** dialog **Animation** tab, which is used to control what effect is being animated on the Animation page in the **Postprocessing** workflow.

Table 349: Configuration group

Tool name	Description
 **Select Load Case**	Opens the **Results Setup** dialog, which is used to select the load cases, mode shapes, and structural elements (beams, nodes, elements, etc.), which are included in the Postprocessing operations

Tool name	Description
 **Structure**	Opens the **Diagrams** dialog, which is used to customize the view of the structure by setting different view-related parameters.
 **Scale**	Opens the **Diagrams** dialog to the **Scales** tab, which is used to specify the scales to which the different diagrams for viewing input and results should be plotted. **Tip:** Select the **Enable Automatic Scaling** option on the **Results View Options** tab of the **Results Setup** dialog to automatically scale displays in the View window.
 **Annotate**	Opens the **Annotation** dialog, which is used to annotate the display of results annotations in the View window with numerical values.

Table 350: Properties group

Tool name	Description
 **Update Properties**	Updates the section properties assigned to members in the input file to those which are the result of a design member selection or group member command. If member selection is carried out for steel, timber or aluminum sections in the model, or if a `GROUP MEMBER` command is specified (see TR.50 Group Specification (on page 2680)), the properties of the structure at the end of these processes will no longer be the same as what we assigned initially or started out with. A warning dialog opens to confirm you wish to proceed before making any changes. **Note:** By performing this operation, the input file gets modified. Consequently, the analysis results are no longer compatible with the modified structure. Therefore, the **Analytical Modeling** workflow is automatically selected so the structure can be re-analyzed. Before doing so, it is advisable to go back and replace the `SELECT` and `GROUP`ing commands with `CHECK CODE` commands.

Table 351: Reports group

Tool name		Description
 **Reports >**	**Node Displacement**	When a node selection is made, this tool opens Node Displacement dialog, which is used to generate

Tool name		Description
	Support Reaction	Display tabular results for support reaction.
	Beam Property	Opens the **Beam Property** dialog, which is used to generate a sorted table of the selected beams for inclusion in the report.
	Beam End Forces	Display tabular results for beam end forces.
	Section Displacement	Display tabular results for beam section displacements.
	Section Forces	Display tabular results for beam stresses.
	Beam Stresses	Display tabular results for beam stresses.
	Column Transfer Force	Opens the **Transfer Forces for Selected Members** dialog, which is used to report transfer forces across beam connections on either side of columns for a selected set of members.
	Plate Stresses	Display tabular results for plate element stresses.
	Principal Stresses	Display tabular results for plate element stresses.
	Floor Vibration Report	Opens the **Floor Vibration Output** dialog, which is used to review the floor vibration analysis per AISC Design Guide No. 11 for a specific floor system and load combination. To utilize this feature, the floor system must be defined as a composite deck in the modeling mode.

Annotation dialog

Used to annotate the display of results annotations in the View window with numerical values.

For example, when a bending moment diagram is displayed in the Main Window area, we may use this option to display the values of the maximum bending moment and/or the end-moment values of selected members.

Opens when the **Annotate** tool is selected in the **Configuration** group on the **Results** ribbon tab.

Ranges tab

Used to select the members and nodes for which we want to display the values.

None	Does not annotate any Member, Element, or Node.
All	Annotates all Members, Elements, and Nodes in the current view.

View Annotates all Members, Elements and Nodes in a previously saved view. Select the desired view from list of available views using the drop down list.

Group Annotates all Members, Elements and Nodes in a previously saved group of structural elements. Select the *Group* option and then select the desired group from the view from list of available groups using the drop down list.

Property Select this radio button to select the members by assigned Property Reference Number and description.

Ranges Specify Node, Member and/or Element numbers. You may specify individual values separated by comma, by a list, or specify a range such as 5-10.

Beam Results tab

Used to select the values, which will be displayed on the diagram.

Bending This group of check boxes determines the Bending Moment values which will be displayed on the Bending Moment Diagram. *Ends* option will display the end-moments of the members. *Maximum* option will display the maximum moment value. *Mid point* option will display the Bending Moment value at the mid point of the member.

Shear This group of check boxes determines the Shear Force values, which will be displayed on the Member Force diagram. *Ends* option will display the values at the two end nodes of the member. *Maximum* option will display the maximum force value. *Mid point* option will display the force value at the mid point of the member.

Axial Switch this on to annotate the axial tension or compression values at the start and end of selected members.

Max Resultant Displays the maximum resultant section displacement value for each member.

Combined Bending and Axial Stress This group of check boxes determines the stress values, which will be displayed on the Stress diagram. *Ends* option will display the values at the two end nodes of the member. *Maximum* option will display the maximum value. *Mid point* option will display the value at the mid point of the member

Node tab

Nodal point displacements can be annotated using these options. More than one may be used simultaneously.

Global X / Y / Z Display the point displacements of a node in any of the global directions.

Resultant Displays the SRSS resultant displacement. That is, the displacement as calculated by:

$$\sqrt{(\text{Global-X}^2 + \text{Global-Y}^2 + \text{Global-Z}^2)}$$

Reactions tab

Used to select the Annotation type for the Support Reactions diagram.

Direct/Bending Check one or more check boxes from Global X/Global rX, Global Y/Global rY, Global Z/Global rZ to select the degrees of freedom to Annotate.

Show Lines Check this box to display line or arc with arrowhead to show the direction of the Reactions.

Scale	Use these spin boxes to specify the Scale factors for the Reaction force and Reaction moment in the diagram.

Annotate	Applies the selected annotation(s) to the View window.
Remove All	Removes all annotations from the View window.
Close	Closes the dialog box.

Related Links

- *P. To graphically display reactions at each support* (on page 1981)
- *P. To graphically display reactions at each support* (on page 1981)

Beam Property dialog

Used to generate a sorted table of the selected beams for inclusion in the report.

Opens when the **Report** > **Beam Property** tool is selected in the **Reports** group on the **Results** ribbon tab.

Sorting tab

Sort By Properties	Select the criteria by which the selected beams will be sorted in the generated table.
Absolute Values	Select this option to sort by absolute value of the selected property.
Set Sorting Order	Select the order by which the selected property is to sorted.

Report tab

Title	Enter a title for the generated table.
Save Report	Select this option to save the properties of the generated table

OK	Closes the dialog and generates a beam table for inclusion in a report.
Cancel	Closes the dialog without generating a beam report.
Help	Opens the STAAD.Pro help window.

Results Setup dialog

Used to select the load cases, mode shapes, and structural elements (beams, nodes, elements, etc.), which are included in the Postprocessing operations.

Opens when:

- the Postprocessing workflow is first selected, or
- when the **Select Load Case** tool is selected in the **Configuration** group on the **Results** ribbon tab.

Loads tab

Used to select the Load Cases to be included for Post Processing.

The **Available** list on the left displays all available load cases.

Click this button…	to…
>	Add the selected load case to the Selected list
>>	Add all load cases to the Selected list.
<<	Remove all entries in the Selected list, load combination is placed back in the Available list.
<	Remove the selected entry from the Selected list.

The **Selected** list displays load cases that have been selected.

Modes tab

For dynamic analyses including response spectrum and time history analysis and/or modal extraction commands, the *Modes* tab becomes activated. This tab lists all available mode shape numbers.

Range tab

Offers selection conditions based on the attributes of Members, Nodes, Groups, Properties, etc. These options determine whether the result tables are to be displayed for all Members/Nodes/Elements, or for the selected ones.

All Selects all structural elements.

View Selects the structural elements stored in a previously saved view. The drop down list displays all previously saved views for selection.

Group Selects the structural elements stored in a previously saved group. The drop down list displays all previously saved groups for selection.

Property Selects the structural elements having a specified property tag. The drop down list displays all existing property tags.

Ranges These edit boxes allow you to select *Nodes* and *Beams* by specifying a range. Specify Node, Member and/or Element numbers. You may specify individual values separated by comma, by a list, or specify a range such as 5-10.

Detail Tables The **Increments** option specifies the number of segments into which a member would be broken up for printing section forces, displacements, etc. These are used in some output tables, such as:

P. Beam Relative Displacement Detail table (on page 2003)
P. Statics Check Results table (on page 2004)

Results View Options tab

STAAD.Pro provides you with two choices in the size to which results such as displacements and moment diagrams are plotted :

a. Instruct STAAD.Pro to plot the result diagrams using a scale that will best fit the diagram within the window bounds. The drawback of this approach is that the variation in result values from one load case to the next will be hard to perceive if all load cases are plotted to the same maximum size dictated by the bounds of the drawing window.

b. Instruct STAAD.Pro to use the same basic scale for all load cases. The drawback of this approach is that while some load cases will have their results fit within the window bounds, some others may be too hard to perceive due to their small values, and others may go out of the bounds of the screen. The user is then required to modify the scale of each plot individually to better see each plot.

Choosing between these two settings is done using this facility.

Enable Automatic Scaling Use this to switch on the type of scaling described in (a). If we wish to use (b) instead, leave this box unchecked.

Displacement/Deflection/ Mode Shape/Beam Results These are the check boxes for the individual result types for which we want Auto-Scaling enabled. Any box left unchecked will be subject to the rules of method (b).

Node Displacement dialog

Used to create a Joint Displacement report for the selected node(s).

Opens when the **Report** > **Joint Displacement** tool is selected in the **Reports** group on the **Results** ribbon tab.

Sorting tab

Sort By Properties Select the criteria by which the selected beams will be sorted in the generated table.

Absolute Values Select this option to sort by absolute value of the selected property.

Set Sorting Order Select the order by which the selected property is to sorted.

Loading tab

Used to select the load cases to be considered in the sorted report.

Available List List of available load cases in the model which will not be included in the generated table.

List Operators

Click this button…	to…
>	Add the selected load case to the Selected list
>>	Add all load cases to the Selected list.
<<	Remove all entries in the Selected list, load combination is placed in the Discarded Load Combinations list.
<	Remove the selected entry from the Selected list.

Selected list List of loads for which the nodal displacement results will be included in the generated report.

Report tab

Title	Enter a title for the generated table.
Save Report	Select this option to save the properties of the generated table

Related Links

- *P. To generate a node displacement report* (on page 1995)

Transfer Forces for Selected Members dialog

Used to report transfer forces across beam connections on either side of columns for a selected set of members.

STAAD.Pro calculates the "Transfer force" (i.e., "pass through force"), which can be used for connection design. This feature is based on a paper on the subject by Dr. William. A. Thornton.

Transfer force is simply the maximum net horizontal force that gets transferred from the one side of the column to the other through the connection. So STAAD.Pro checks the forces in the members framing into each side of the column and finds out the resultant horizontal force for either side. Typically the resultant forces on the two sides would not be equal as some amount of force will be taken up by the column in shear. The greater of the two resultants is reported by STAAD.Pro as the transfer force. The option to determine transfer force automatically, will save engineers considerable time and effort as in most cases, they have to report the transfer forces in the design drawings. The same concept can be applied to floor bracing in horizontal plane.

Opens when the **Report** > **Column Transfer Forces** tool is selected in the **Reports** group on the **Results** ribbon tab.

Loads	Displays the load cases that has been considered to calculate the transfer force. By default, all load cases are considered but one can exclude a few of them by simply clicking on the load case within the Loads box.
Left / Right Beams	List all the members on the respective sides of the column along with their incidences. The boxes Left TF and Right TF shows the resultant horizontal force from either side. Max TF reports the transfer force.
Insert to Table	Generates the Transfer Force Report table which contains all the transfer force information and can be included in reports.
Close	Closes the dialog.

Related Links

- *P. To generate transfer forces report* (on page 1996)

Floor Vibration Output dialog

Used to review the floor vibration analysis per AISC Design Guide No. 11 for a specific floor system and load combination. To utilize this feature, the floor system must be defined as a composite deck in the modeling mode.

Opens when the **Report** > **Floor Vibration Report** tool is selected from the **Reports** group on the **Results** ribbon tab.

The adequacy of a floor system from the standpoint of its vibration serviceability due to human activity, specifically walking excitation, can be assessed using STAAD.Pro. The procedures of Chapters 3 and 4 of the AISC Steel Design Guide Series No. 11 - *Floor Vibrations due to Human Activity* are used by the program.

The vibration calculation is done for:

a. beam or the joist mode
b. girder mode

The two modes are then combined to obtain the system frequency and other results of the combined mode using the Dunkerley relationship described in chapter 3 of the AISC Design Guide. Results for the 2 basic modes and the combined mode are provided in a tabular form.

The output for the combined mode consists of:

a. the peak acceleration for walking excitation
b. allowable acceleration (known in the code as the acceleration limit)

The design criterion as stated in the code in the third paragraph in Chapter 4 is that a floor system is satisfactory if the peak acceleration does not exceed the acceleration limit.

Select Select a previously defined Composite Deck.

> **Note:** Composite Decks are defined use the **Composite Deck Layout** tool.

Load Case Select the load case for analysis.

Check Performs the floor vibration checks per AISC Design Guide 11.

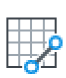
Print Opens a Print dialog, which is used to create a hard copy of the currently displayed floor vibration report.

Related Links

- *P. Floor Vibrations Engineering Theory* (on page 1998)
- *P. To generate a floor vibration report* (on page 1996)

Steel AutoDrafter tab

Table 352: Edit group

Tool name	Description
 **Grids**	Opens the **Grid Manager** dialog, which is used to select which plan grids and elevation marks are displayed on the structure as well as in generated drawings.

Tool name	Description
 **Drawing Groups**	Opens the Drawing Groups tab in the **Drawing Lists & Groups** panel.
 **Drawing Style Manager**	Opens the **Drawing Style Manager** dialog, which is used to create drawing styles for use in Steel AutoDrafter.

Table 353: View group

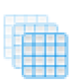
Tool name	Description
 **Node Number**	Controls the display of node numbers in the view area.
 **Member Number**	Controls the display of member numbers in the view area.
 **Member Property**	Controls the display of member material names in the view area.
 **Grids**	Controls the display of grids in the view area.

Table 354: Drawings group

Tool name	Description
 **View Drawings List**	Opens the **Drawing List & Groups** panel in the view area. This panel is used to generate drawings individual, all drawings in the list, and to manage groups.

Table 355: Material Take Off group

Tool name	Description
 **Drawing**	Opens the material take-off for the entire structure in a drawing.
 **Text**	Opens the material take-off for the entire structure in an HTML text table.

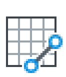
Related Links

- *P. To open the Steel AutoDrafter workflow* (on page 2019)

Grid Manager dialog

Used to select which plan grids and elevation marks are displayed on the structure as well as in generated drawings.

Opens when the **Grids** tool is selected from the **Edit** group on the **Steel AutoDrafter** ribbon tab.



The program will generate a grid line for nodal X and Z coordinate in the model. An elevation mark will be generated for each nodal Y coordinate.

Check the option for each grid or elevation mark to display it n the view area and in generated drawings.

Change grid or elevation mark labels by typing in the associated field for each.

Dialog controls

Import	Used to import grids from an existing `adfx` file.
OK	Updates the grid display and closes the dialog.
Cancel	Closes the dialog without saving any changes.

Related Links

- *P. To edit grid labels* (on page 2028)

Drawing Style Manager dialog

Used to create drawing styles for Steel AutoDrafter. This facilitates production of drawings as per selected style. You can also create standard styles with the required variations.

Opens when the **Drawing Style Manager** tool is selected from the **Edit** group on the **Steel AutoDrafter** ribbon tab.

All aspects of the drawing; Layer Descriptions, Layer colors, Connection / Member Descriptions, Dimension styles, Text Styles and sizes, and gird styles can be changed to create new text styles.

The Drawing Style can be imported, exported, and restored from the default file.

Drawing Style can be imported from another `adfx` file using the import grid option provided. The `adfx` file will be available along with the relevant STAAD.Pro file in the same folder.

Layers tab

This table lists all the layers that are used in the drawings generated by Steel AD. The layer descriptions and the colors can be changed.

Member Labels tab

Member / Connection type table lists all the various symbols that are used in the drawings. These symbols can be changed as desired.

In case you do not wish to have any symbols in the drawing, keep the symbols column blank.

The Section Referencing box, determines how the various sections will be labeled in the drawing, by using a reference number or the entire description of the section.

Appearance tab

This tab lists all the appearance related data. Fonts, Text Heights, Dimensions Styles and Grid Styles can be set in this tab.

Related Links

- *P. To configure units in Steel AutoDrafter* (on page 2019)

Member Labels

The Default labels to be presented are as mentioned below. You can change them as required using the settings provided in the **Drawing Style Manager** dialog.

Member Labels

Member Type	Description
GM	Beam - Column Moment Connection
GS	Beam - Column Shear Connection
BS	Beam to beam Shear Connection
C	Column
V	Bracing in Elevation
HV	Bracing in Plan
PST	Post
PTL	Portal
TC	Top Chord of Truss
BC	Bottom Chord of Truss
PUR	Purlin

Member Labeling System

A	B	C	D
Memb Type	**Section Ref No**	**Built Up Reference**	**Labels**
GS	14	F - Channel F/F	GS14-F1
GM	14	B -Channel B/B	GS14-B2
GM	10	TB- 2 Cover Plates on I/H sxn	GM10-TB1
GM	10	TC- 1 Cover Plates on I/H sxn	GM10-TC2
GM	10	BC- 1 Cover Plates on I/H sxn	GM10-BC3

A	B	C	D
Memb Type	**Section Ref No**	**Built Up Reference**	**Labels**
V	33	L - Long Legs of Angle	V33-L1
V	33	- Equal Angle	2V33-10
V	34	S - Short Legs for angle	V34-S2
GM	35	SA - Star of angles	GM35-SA1
V	10	SI - Star Of I/H Sxn	V10-SI2
GM	0 - BU	BU - Built Up Girder out of plates	GM0-BU1
GM	15	2 I/H Sections	2GM15-10

Member Grouping System in STAAD.Pro

A	B	C	Final Label	Remarks
Cantilever Label	**Group Type**	**Grp Number**	**STAAD Group Names**	
NC	GS	1001	_NC-GS-1001	
EC	GM	1002	_EC-GS-1002	
SC	BS	1003	_SC-BS-1003	
WC	GM	21	_WC-GM-21	
	C	51	_C-51	
	V	101	_V-101	Optional For Single Members
	HV	2005	_HV-2005	Optional For Single Members
	VB (Vert Bracing)	100	_VB-100	All Members can be put in one group
	PST (Post)	100	_PST-100	
	PTL (Portal)	1	PTL-1-ISMC150-150	Each half of the portal Asc and Desc to be grouped seperately

A	B	C	Final Label	Remarks
Cantilever Label	**Group Type**	**Grp Number**	**STAAD Group Names**	
	TC (Top Chord)	1	TC-1-ISMC150-3001	Each Top Chord ascending and descending to be grouped separately
	BC (Bot Chord)	100	_BC-100	
	TV (Truss Bracing)	100	_TV-100	All members Vertical and inclined can be put in one group
	PUR (Purlin)	100	_PUR-100	Compulsory to group if modeled

Related Links

- *P. To configure units in Steel AutoDrafter* (on page 2019)

Chinese Steel Design tab

Table 356: Settings group

Tool name	Description
 **Secondary Member**	Opens the **Assign Secondary Member** dialog, which is used to identify a member as a secondary member. The members they frame into will not be considered as restrained when determining automatic effective length calculations.
 **Brace Angle**	Opens the **Brace Angle** dialog, which is used to set the angle between the brace and the beam in both directions.
 **Material**	Opens the **Material Parameter** dialog, which is used to add or modify a custom steel material for use in the Chinese Steel Design workflow.
General Section	Opens the **General Section** dialog, which is used to specify built-up section profiles for general shapes used in analysis.

Table 357: Design group

Tool name	Description
 **Design**	Opens the **Chinese Steel Design** dialog, which is used to select the design options and checks to be performed as well as to initiate the design process.

Table 358: Solution group

Tool name	Description
 **Configuration**	Opens the **Configure Solultion** dialog, which is used to select parameters to include within a solution set. This allows you to run different solution scenarios within the same model.
Solution Selection	Select the current solution set name from the drop-down list to use in the design process.

Table 359: Select group

Tool name	Description
 **Failed**	Selects all members that have failed one or more design checks.
 **Passed**	Selects all members that passed all the design checks.
 **Minimal**	Selects all members that have a maximum utilization of less than 30% of the design requirements.

Tool name	Description
Not Designed	Selects all members that have not been included in a design.
Inverse	All beams in the selection set are deselected and any previously unselected beams are added to the selection set.
Parallel	Select beams that are parallel to the selected global axis. • **To XY** • **To YZ** • **To XZ**
Group	Opens the **Select Groups** dialog, which is used to select objects in a named group.
Property Name	Used to select nodes and members based on specifications associated with them. A number of specification types are included in the sub-menu list.
Missing Parameter	Select members which do not have parameter set assigned.

Table 360: Labels group

Tool name	Description
Parameter	Toggles the display of assigned parameter name for members.

Tool name	Description
 **Ratio**	Toggles the display of governing code check ratio for members.

Table 361: Results group

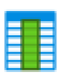
Tool name	Description
 **Apply Optimize Results**	Click to apply any update section sizes selected in the Results detail dialog. You will be prompted to re-analyze the structure.
 **Table Settings**	Opens the **Define Results Column** dialog, which is used to select the table columns included in the results.

Table 362: Output group

Tool name	Description
 **Member Report**	Opens the detailed design results for a single member. Use the tools to select which items are included in the report as well as to export it to different outputs.
 **Export Report**	Opens the **Export Report** dialog, which is used to select portions of the results to include in a report.

Assign Secondary Member dialog

Used to identify a member as a secondary member. The members they frame into will not be considered as restrained when determining automatic effective length calculations.

Opens when the **Secondary Member** tool is selected on the **Chinese Steel Design** ribbon tab in the **Settings** group.

Related Links
- *D. To assign secondary members* (on page 981)

Brace Angle dialog

Used to set the angle between the brace and the beam in both directions.

Opens when the **Brace Angle** tool is selected on the **Chinese Steel Design** ribbon tab in the **Settings** group.

When the angle is exceeded, the brace will not work. You can also set the member extend angle, when the angle is exceeded, the extended member will not work.

Brace and member major axis angle (1)	Type a maximum value for considering a brace between the member's major axis and the brace. The default value is 60°.
Brace and member minor axis angle (2)	Type a maximum value for considering a brace between the member's minor axis and the brace. The default value is 60°.
Member extend angle (3)	Type the maximum angle between two adjacent members to be considered as coplanar. The default value is 15°.

Related Links
- *D. To specify brace angle threshold values* (on page 982)

Material Parameter dialog

Used to add or modify a custom steel material for use in the Chinese Steel Design workflow.

Opens when the **Material** tool is selected on the **Chinese Steel Design** ribbon tab in the **Settings** group.

Tip: A set of predefined steel materials from GB50017-2017 Table 4.4.1 are included in the **Materials - Whole Structure** dialog and can be used for the model.

Note: Custom materials added in the Chinese Steel Design workflow are saved to the current model only.

Click **Add** to add a new user-defined material which can then be edited by selecting it in the table below and then clicking **Modify**. Only user-defined materials can be edited or deleted.

Material Name	Type a unique ID used to identify the material in parameters and reports.

Max Thickness	Type the maximum thickness for which this material can be used (in mm). If a section element exceeds this thickness when the material is used, a warning will be generated in the design results.
Yield Strength	fy
Tensile Strength	f
Shear Strength	fv
Section Bearing Strength	fce
Ultimate Tensile Strength	The rupture tensile strength, fu.

Related Links

- *D. To add a custom material definition* (on page 982)

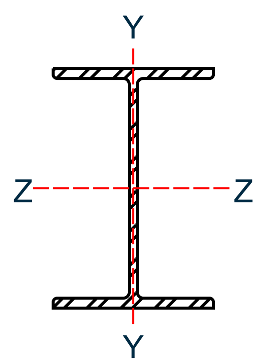
General Section dialog

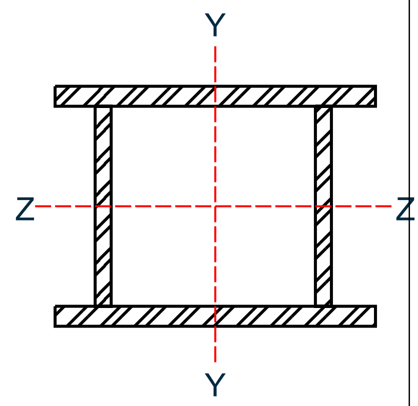
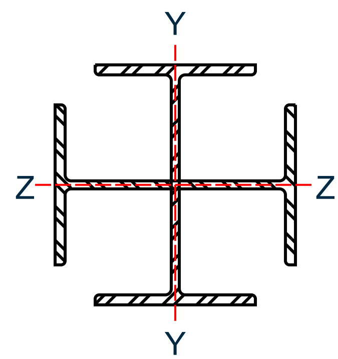
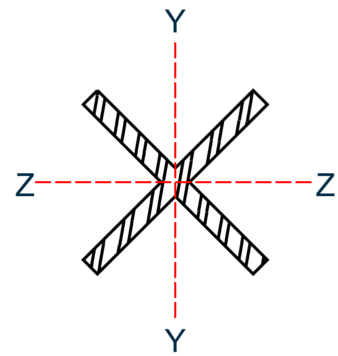
Used to specify built-up section profiles for general shapes used in analysis.

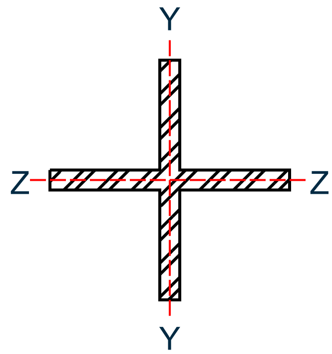
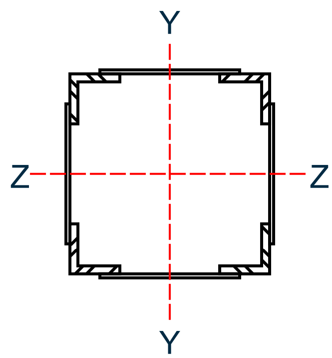
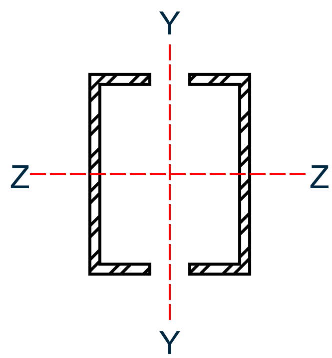
Opens when the **General Section** tool is selected on the **Chinese Steel Design** ribbon tab in the **Settings** group.

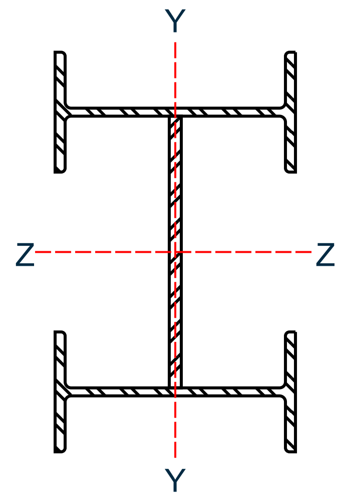
Type

Select the general section type from the drop-down list:

Type name	Cross Section
I Section	

Type name	Cross Section
Box	
CrossedH	
CrossedP1	

Type name	Cross Section
CrossedP2	
Lattice 1	
Lattice 2	

Type name	Cross Section
EnhancedH	

Outstanding Width of Flange (BF)

Thickness of Flange (TF)

Height of Web (BW)

Thickness of Web (TW)

Distance from neutral axis in Z direction to the bottom (y0)

Distance from neutral axis in Y direction to the left (y0)

Chinese Steel Design dialog

Used to specify the code check items to be performed and then perform a code check.

Opens when the **Design** tool is selected on the **Chinese Steel Design** ribbon tab in the **Design** group.

Click **OK** to perform the selected code checks. The **Results** window opens to display the results of the code check operation.

Structure tab

Setup Default Design Parameters	Used to specify the design settings for members which do not have a specific design parameter set assigned.
Seismic Option	Option to include the seismic provisions and to identify the precautionary intensity to use:

G6
G7
G8
G9

> **Note:** In order to include seismic design, the mode should include one or more seismic load cases and the “Seismic Measure Grade” should be specified in the member parameter sets.

Design Range

Select whether the design process should be performed on the entire model or only the currently selected members.

Optimize Option

The standard design process will check the structure using the data of the member profiles assigned to the model. However, it is possible to determine if a different profile would be better suited using an optimisation design. This action is applied to the members included in a parameter set that has the option **Optimise Design** set in the General Parameter page.

The optimization of any given member is determined by checking the applied forces on the given member against all the profiles in the section table from which the member was specified and selecting the profile that provides a maximum utilization that is as large as possible, but less than 1.0.

Note to view the profiles that are assigned to the members after an optimized design

When performing the optimization, there are three options that can be used to manage the optimization process.

Group Optimize

With this option a single profile is selected that provides a maximum utilization, but less than 1.0, when used on all the members in the parameter set that has the **Optimise Design** specified. This resulting in all members in the parameter set having the same Optimized Profile.

Fast Optimize

With this option specified, each member will be designed by sequentially stepping through the table of profiles from which the profile has been specified ordered by either

- None, i.e. defined by the table sequence, or
- the profile area AX, or
- the profile major moment of inertia IZ,

until a suitable profile has been identified. If not set, then every section profile in the table is checked.

Iterations

If a number of iterations is specified (i.e., a number >0), then after performing an optimization, the new profiles are assigned to each optimized member and a new analysis is performed. In doing so this may result in a new/different distribution of forces and stresses. The design is then re-performed. These steps are repeated for the number of iterations specified.

Design Checks tab

Select the code check you want the program to perform. There are buttons on the left to quickly select those that pertain to strength, stability, displacement, and detailing. There are also selection tools for all, none, or inverting the selection set.

Note: You can hold **<Ctrl>** to select multiple items.

Related Links

- *D. To perform steel design per the Chinese code* (on page 983)

Configure Solution dialog

Used to select parameters to include within a solution set. This allows you to run different solution scenarios within the same model.

Opens when the **Configuration** tool is selected on the **Chinese Steel Design** ribbon tab in the **Solution** group.

Select Solution	Select the solution set name from the drop-down list for management. **Note:** The type of analysis associated with the solution is indicated at the bottom-left corner of the dialog.
Parameter List	All parameter sets in the current solution set can be added for use in the solution set here. Use the **Used** (included) and **Unused** (excluded) lists and controls to configure the solution set.
Add	Click to add a new solution set.
Delete	Click to remove the current solution set from the model.
Rename	Click to rename the current solution set.

Related Links

- *RR 22.08.00-4.4 Multiple Parameters in Chinese Steel Design* (on page 100)
- *D. To add a new solution set* (on page 979)

Assign Design Parameter dialog

Used to modify and assign design parameters per the Chinese steel design code.

Opens when the **Chinese Steel Design** workflow is selected.

A list of the current parameter sets associated with the current STAAD model. is displayed in the top portion of the dialog.

Add	Opens the **Add new: Chinese Steel Design Parameters** dialog, which is used to add a parameter definition to the model for use in Chinese steel design.

Edit	Used to edit the currently selected parameter set in the **Edit: Chinese Steel Design Parameters** dialog.
Edit Parameters (Text)	Opens the parameter sets in a plain text editor (e.g., Notepad) for editing. **Note:** No data validation of any kind is performed in the plain text editor. Therefore, the accuracy and validity of these changes is dependent upon the user to verify.
Delete	Delete the selected parameter set from the model.
Load Template	Used to open a set of parameters in a plain text (file extensions `.gsp` or `.txt`) for use in the current STAAD model.
Save As Template	Save the current parameter sets as a template file which can be loaded for use in other STAAD input file for Chinese steel design.
Assignment Method	The options under the **Assignment Method** are used to assign parameter sets to steel members. The assignment method can be one of **Assign To Selected Beams, Assign To View, Use Cursor To Assign** or **Assign To Edit List**. • Assign To Selected Beams **-** To assign a specification to selected members, first select the parameter set from the table. Then select the members to which this parameter set is to be assigned. This is done by going to the **Select** menu, then choosing the appropriate cursor option. Select the desired members using the cursor. When all desired nodes, members or elements are selected, click the **Assign To Selected Beams** radio button, then click the **Assign** button. • Assign To View **-** To assign a parameter set to all members in a view, first select the parameter from the table. Select the **Assign To View** radio button, then click the **Assign** button. All members in the structure are assigned this parameter set. • Use Cursor To Assign **-** To assign a parameter set to members using the cursor, first select the parameter set from the table. Select the **Use Cursor To Assign** radio button, then click the **Assign** button. The button will appear active and the label will change to **Assigning**. Use the appropriate cursor (from the Select menu) to assign the selected parameter set to the individual members. Click on the **Assign** button again to finish. • Assign To Edit List **-** To assign a specification using a typed list of node, member or element numbers, first select the parameter set from the table. Select the **Assign To Edit List** radio button, then type the list of member numbers and click the **Assign** button.

Chinese Steel Design Parameters dialog

Used to add a parameter definition to the model for use in Chinese steel design.

Opens when the **Add** or **Edit** button is clicked on the **Chinese Steel Design Parameters - Whole Structure** dialog.

General Parameters tab

Parameter Name	Type a unique label used to identify the parameter set.
Steel No	Select the steel grade used (independent of any steel grade assigned as part of a material definition):

- Q235 (default)
- Q345
- Q355
- Q390
- Q420
- Q460

Refer to Steel Grade (on page 990) for additional details.

Member Type

Select the member type for design:

- Automatic (default) - the actual type of the member is determined by the program based on the **Principle** selection.
- Bending - a beam for which bending (MZ, MY) are considered but axial force (FX) is ignored for design.
- Truss - only axial force (FY) is considered for design.
- Column - both moment and axial force are considered for design.
- Defined - for this type of member, the provisions for code checks are all user-defined.

Refer to Member Type (on page 989) for additional details.

Principle

Select the option which the program should use to determine the member type for Automatic:

- **Geometry -** uses the geometric conditions of the member for selection as follows:

geometric condition	member type
member in a horizontal plane	bending
member with a truss specification	truss
any other condition	column

- **Stress Feature -** uses the stress conditions of the member for selection as follows:

stress condition	member type
no axial force	bending
no bending moment	truss
any other condition	column

Refer to Member Type (on page 989) for additional details.

Check as Cantilever Member

Specify any cantilever member checks:

- None (default)
- Both - both major and minor axes
- Major Axis
- Minor Axis

Truss Secondary Moment

Section Slenderness Grade	Select the slenderness grade to check the width-to-thickness ratio of beam and column members per Table 3.5.1: • S1 • S2 • S3 (default • S4 • S5
Use Axis forced member Post-Buckling	
Displacements Load List	Click **[...]** to open the **Select Load Case for Displacements** dialog, which is used to select the load cases and combinations which the program should use for displacement checks.
Fatigue Check Request	
Force Loads List	Click **[...]** to open the **Select Load Case for Design** dialog, which is used to select the load cases and combinations which the program should use for design checks.
Optimize Design	
Failure Ratio Above	
Safety Ratio Below	

Seismic Design tab

Seismic Adjusting Factor	Check this option to specify the seismic adjusting factor values for **Stability** and **Strength** checks from the following: • Auto (default) • 0.75 • 0.8 • 1.0
Seismic Measure Grade	Select the levels as per Clauses 8.3.1, 8.3.2, and 8.4.1 and from Table 8.3.2. Select from the following: • None (default) • Level 1 • Level 2 • Level 3 • Level 4
Reduction Factor of Bearing Capacity	

Allowable Slenderness tab

Allowable Compression Slenderness

Select the allowable value of slenderness ratio for members in compression:

- Auto (default) - the value is determined as per Table 7.4.6
- 150
- 200

Allowable Tension Slenderness

Select the allowable value of slenderness ratio for members in tension:

- Auto (default) - the value is determined as per Table 7.4.7
- 200
- 250
- 300
- 350
- 400

Seismic Brace, GB50011-2010 8.4.1

Seismic Multi-storey Plant, GB50011-2010 H.2.8

Seismic Column, GB50011-2010 8.3.1

Seismic Single-storey Plant, GB50011-2010 9.2.13

Plastic Development Factor tab

The plastic development coefficient is a coefficient related to the shape of the section. The coefficient can be according to article 6.1.2 8.1.1 of the specification. If you enter the design parameters γ_z, γ_y, γ_{sharp}, the program will use those values. If you select to use automatic calculation, the program will use values for γ_z, γ_y, γ_{sharp} as specified in Plastic Development Coefficients γx, γy, γsharp (on page 992).

Major Axis (Z Axis)

The γ_z value.

- Auto (default)
- 1.05
- 1.20

Minor Axis (Y Axis)

The γ_y value.

- Auto (default)
- 1.05
- 1.20

Sharp (Tee/Double Angle/Channel)

The γ_{sharp} value.

- Auto (default)
- 1.05
- 1.20

Equivalent Moment Factor tab

The equivalent moment coefficient is used to "correct" the bending moment when checking the stability of the solid web bending member with bending moment acting on the symmetry axis plane, so it is divided into in-plane equivalent moment coefficient and out of plane equivalent moment coefficient. These coefficients can be calculated automatically by program or manually input.

Equivalent Moment Factor in-Plane

- Auto (default)
- 1.00
- 0.85
- 0.65

Equivalent Moment Factor out-Plane

- Auto (default)
- 1.00
- 0.85
- 0.65

Lateral Load Along Local y

- Auto (default)
- 1.00
- 0.85
- 0.65

Lateral Load Along Local z

- Auto (default)
- 1.00
- 0.85
- 0.65

Deflection Parameters tab

Refer to Deformation Parameters (on page 991) for additional details.

Span/Deflection (DFF) Select the span-to-deflection ratio limit:

- 150
- 200
- 250
- 300
- 350
- 400 (default)
- 500
- 600
- 750
- 1,000

Start Joint (DJ1) Select the starting joint used to define the member for deflection checking:

- Auto - forms a continuous beam

- Member start joint - selects the current member start joint

End Joint (DJ2)	Select the ending joint used to define the member for deflection checking: • Auto - forms a continuous beam • Member end joint - selects the current member end joint
Check Horizontal Section Displacement	

Stability Factors tab

The stability coefficient is divided into overall stability coefficient of flexural member (beam) and stability coefficient of axial compression member (truss). The program can automatically calculate the coefficients according to the section type and force characteristics of the designed member as described in Stability Coefficient (on page 992).

Overall Stability Factor of Beam	**Major Axis (Z Axis)**	• Auto
	Minor Axis (Y Axis)	• Auto
	Stress Feature (GB50017-2017 Appendix Table C.1)	• Type1 • Type2 • Type3 • Type4 • Type5 • Type6 • Type7 • Type8 • Type9 • Type10
Stability Factor of Axial Compression	**Major Axis (Z Axis)**	• Auto
	Minor Axis (Y Axis)	• Auto
Axial Compression Section Type	**Major Axis (Z Axis)**	• Auto • Class a • Class b • Class c • Class d
	Minor Axis (Y Axis)	• Auto • Class a • Class b

- Class c
- Class d

Effective Length tab

The unsupported effective length and effective length coefficient are the calculation parameters needed to check the stability of members. The direction of the strong axis and the weak axis of the principal inertial axis should be considered respectively. For complex structures, it can difficult to select the effective length and effective length coefficient of members. Therefore, the program can automatically compute these values. When these coefficients are automatically calculated by the program, they are calculated as described in Unsupported Effective Length and Effective Length Coefficient (on page 993). The values may also be user-specified.

Unbraced Length: The unbraced length (in meters) of the members, in both the **Major Axis (Z Axis)** and **Minor Axis (Y Axis)**.

Effective Length: Select the member bracing condition in the **Brace Type** drop-down. Then, specify the effective length coefficients, μ_z (**Major Axis (Z Axis)**) and μ_y (**Minor Axis (Y Axis)**).

> **Note:** If the current solution set is set to use a P-Delta analysis type, then the effective length options are disabled. An effective length of 1.0 is used for all design parameter sets for this solution set type.

Gyration Radius Calculation of Single Angle: Select the option to use for the radius of gyration for single angles:

- Parallel Leg Calculation (Y-Z Axis) - the geometric axes
- Strong Weak Axis Calculation (U-V Axis) - the primary axes

Refer to Rigid Fixed Stepped Column at the Lower End of Frame of Single Story Workshop (on page 996) for details.

Detailing Checks tab

The structural requirement parameters specified here refer to the limits of flange width thickness ratio and web height thickness ratio, as well as the section type of axial compression members. Generally, this parameter is automatically calculated by the program. However, in order to make it easier for users to design the program more flexibly, you can also set the parameters required here. Refer to Construction Requirements (on page 1002) for details.

Flange Slenderness Limit (b/t)

- Auto

Web Slenderness Limit (h0/tw)

- Auto

Include check for web of H profiles in a Truss as per GB50017-2017, Table 8.5.2: Check this option to use the values for Table 8.5.2 instead.

Section Factor tab

The net section coefficient refers to the ratio of net section to gross section area and section modulus (temporarily replaced by net section modulus coefficients An, Wnz, and Wny). The default value is 1.0. Refer to Section Coefficient (on page 1003) for details.

Net Section Factor	**Net Area Factor (An/A)**	Type a value to use for the net section area to gross area ratio.
	Net Resistance Moment Factor (Wny/Wy)	Type a value to use for the net moment resistance to gross section moment resistance ratio about the y direction.
	Net Resistance Moment Factor (Wnz/Wz)	Type a value to use for the net moment resistance to gross section moment resistance ratio about the z direction.
Effective Section Factor of Axis Force (nA)	Type a value to use for the net section area to gross area ratio.	

Reduce Factor	**Reduce Strength of Angle Section (Ref. 7.6.1)**	Check this option to use a re
	Strength Reduction	• Auto • 0.70 • 0.85 • 0.90
	Stability Reduction	• Auto • 0.70 • 0.85 • 0.90
	Unequal Angle Connection	• Short leg • Long leg (default)

Optional Checks tab

Optional checks based on the selected member type. Any of the checks can be un-selected to prevent that check from being performed.

For bending members:

- Beam Bending Strength
- Beam Deflection
- Flange Slenderness
- Overall Stability
- Equivalent Stress
- Web Slenderness
- Shear Strength

For truss members:

- Compression Flange
- Compression Shear
- Compression Stability

- Compression Web Slenderness
- Compression Slenderness
- Truss Strength
- Tension Slenderness

For column members:

- Column Strength
- Compression Flange Slenderness
- Compression Slenderness
- Compression Web Slenderness
- Shear Strength
- Stability in-plane
- Stability out-plane
- Tension Slenderness

For defined members:

Strength	Select the strength check: • Beam Bending Strength • Truss Strength • Column Strength
Stability	• Overall Stability • Compression Stability • Stability in-plane
Flange Slenderness	• Flange Slenderness • Compression Flange Slenderness
Web Slenderness	• Web Slenderness • Compression Web Slenderness
Other	• Shear Strength • Compression Shear • Equivalent Stress • Stability out-plane • Beam Deflection • Compression Slenderness • Tension Slenderness

Related Links

- *D. To add a parameter set* (on page 980)

Connection Design tab

Table 363: Assign Connections group

Tool name		Description
 **Joint Cursor**		Used to select joints for connection design.
Select Joints >	*`<joint types>`*	Tools for selecting joints in the model by member type and orientation.
	Select Special Joints	Opens the **Special selection of joints** dialog, which is used to select joint types by category as well as by filtering for member depth and end release.
Select Connections >	**Select all connections**	This will select all the connection in the model
	Select identical connections	This will select all the identical connection in the view **Tip:** This effectively allows you to “group” connections together to edit as one.
	Select connections of the same type	This will select the connection of same type. **Note:** To use this feature, you must select at least one connection before selecting this tool.
	Select connections with same template	This will select the connection of same template. **Note:** To use this feature, you must select at least one connection before selecting this tool.
	Select connections of the same family	This will select the connection of same family name, as used in RAM Connection. **Note:** To use this feature, you must select at least one connection before selecting this tool.
	Select connections with same tag names	This will select the connection of same tag name, as used in RAM Connection. **Note:** To use this feature, you must select at least one connection before selecting this tool.

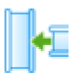
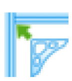
Tool name		Description
	Select connections with same template and sections	This will select the connection of same template and where the section of the members which participate in the connection are also same. **Note:** To use this feature, you must select at least one connection before selecting this tool.
	Select connection nodes and members	This will select all the nodes and members which participate in the connection. **Note:** To use this feature, you must select at least one connection before selecting this tool.
 **Assign Basic Connection**		Opens the **Basic Connections** dialog, which is used to assign basic connection templates to selected joints.
 **Assign Smart Connection**		Opens the **Smart Connections** dialog, which is used to assign smart connection templates to selected joints.
 **Assign Gusset Connection**		Opens the **Gusset Connections** dialog, which is used to assign gusset connection templates to selected joints.

Table 364: Frames group

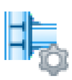
Tool name	Description
 **Identify Beam and Grider**	Opens the **Beam-Girder Identification** dialog, which is used to switch the beam and girder assignments for connections. **Note:** If no Beam-Girder (BG) connections have been assigned, a warning dialog opens to notify you no matching connections could be found .

Tool name	Description
 **Create Seismic Frame**	Opens the **Seismic Frames** dialog, which is used to specify the type of seismic frame used for the selected members creating a seismic frame definition.

Table 365: Reports group

Tool name	Description
 **Connection Report**	Opens the **RAM Report Export** dialog, which is used to manage connections and other report details to include in an report.

Table 366: Configure group

Tool name	Description
 **Connection Database**	Opens the connection database dialog, which is used to manage and create connection templates.

Special Selection of Joints dialog

Used to select joint types by category as well as by filtering for member depth and end release.

Opens when the **Select Joints** > **Select Special Joints** tool is selected in the **Assign Connections** group on the **Connection Design** ribbon tab.

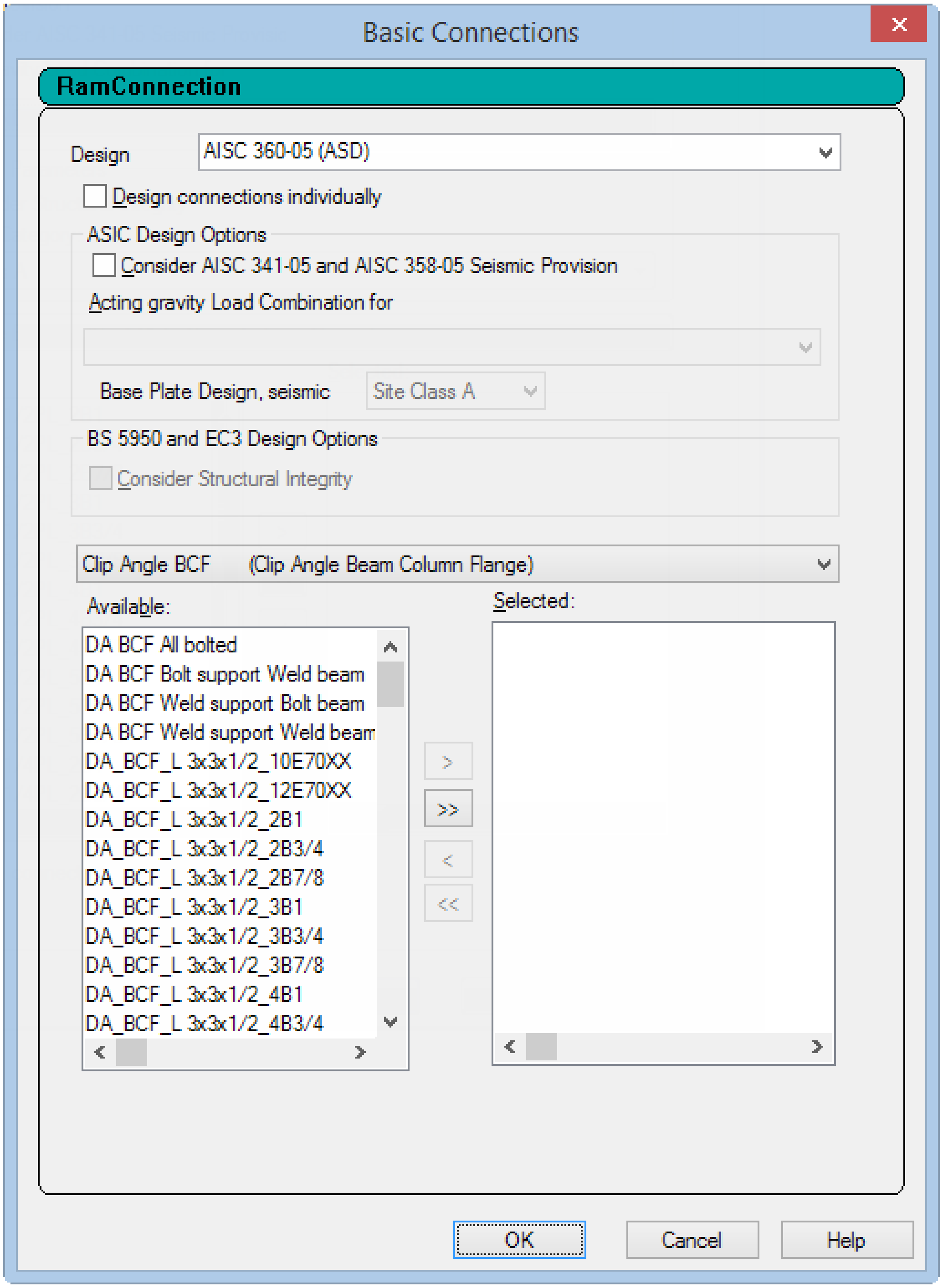
Select from	Choose an option by which to limit the selecting range if you had previously selected members prior to this dialog opening.
Family	Select one connection family type. These are the same options as found on the **Select Joints** tool n the **Assign Connections** group on the **Connection Design** ribbon tab.
Limit selection by beam depth	Check this option and provide minimum and maximum beam depth values (in the units displayed) in order to further limit the matching connections for selection.
Verify Releases	Check this option and then check one or both of the end release types (pinned or fixed).

Basic Connections dialog

Used to assign Basic connection templates to selected joints. A “Basic” connection template contains all the information about the connection (such as the plate sizes and bolt locations etc) which is applied to a joint and then checked for code compliance.

A set of joints that is to use Basic connections can contain multiple connections (i.e., each with a different plate size and bolt diameter). If the first connection does not achieve compliance, then the next is checked until either a suitable connection is defined or all the selected connections have been checked. Therefore, when assigning basic connections, you will typically select multiple connections to be checked.

Opens when **Connection Design** > **Assign Basic Connections** is selected.



Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the **Connection Design** menu. See the RAM Connection manual for more information on editing the connection database.

Design	Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.

Seismic Provisions

(AISC codes only)

Select the Consider AISC 341-05 Seismic Provisions option to instruction the program to design the connections per this specification.

A gravity load combination must be selected when this option is used.

Consider Structural Integrity

(BS5950 only) Select this option to include the requirements for structural integrity in the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for these verifications. The appendices A to D (BS5950 1:2000) give the required information on the behavior and methodology to resist tying forces in the design.

Seismic Category

(Base Plate only) Specify the site class which best describes the soil conditions supporting the foundation and base plate.

Connection types

Select the class of connection to be used from this list. The connection types available depend on the code selected.

Available list

A list of all available connection templates for the selected connection type are listed here.

List Operators

Click this button…	to…
>	Add the selected connection template to the Selected list.
>>	Add all connection templates to the Selected list.
<<	Remove all entries in the Selected list.
<	Remove the selected entry from the Selected list.

Selected list

The connection templates selected for design will be added to the list.

Design Connections Individually

Select this option to instruct the program to optimize connections at each joint separately, rather than grouping for the worst case among common connections.

Button	Description
OK	Accepts the design criteria and performs a design for the selected joints using the selected connection templates.
Cancel	Closes the dialog without designing any connections.
Help	Opens the STAAD.Pro help window.

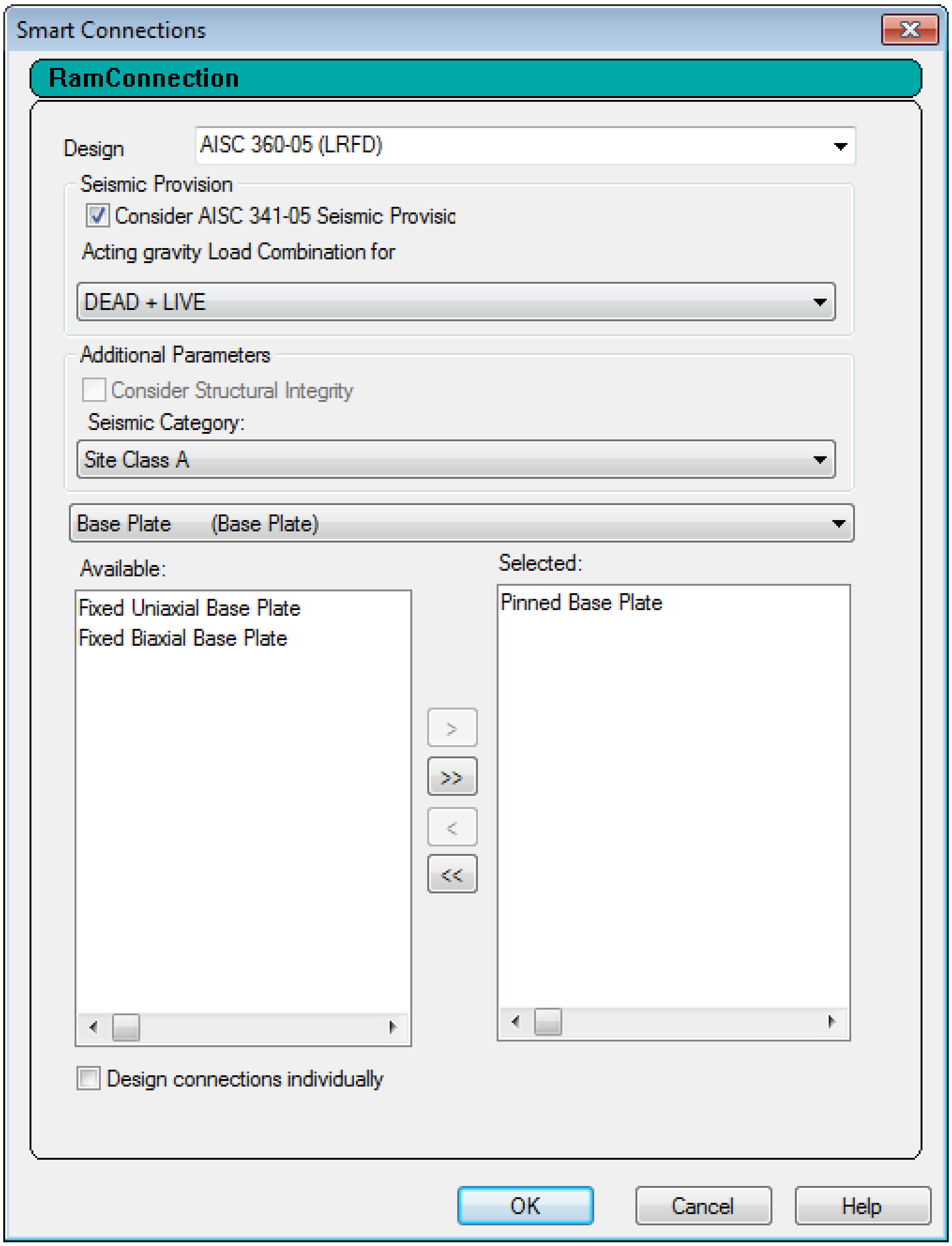
Related Links

- *D. Design Connections Individually* (on page 1037)
- *D. To select a basic connection template* (on page 1033)

Smart Connections

Used to assign smart connection templates to selected joints. A “smart” connection template contains parametric rules defined in a macro which allows characteristics of the connection to be modified (within limits) in order to achieve code compliance. A joint selection set that is to use a Smart connection can only specify a single smart connection definition.

Opens when **Connection Design** > **Assign Smart Connections** is selected.



Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the **Connection Design** menu. See the RAM Connection manual for more information on editing the connection database.

Design
Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.

Seismic Provisions
(AISC codes only)

Select the Consider AISC 341-05 Seismic Provisions option to instruction the program to design the connections per this specification.

A gravity load combination must be selected when this option is used.

Consider Structural Integrity
(BS5950 only) Select this option to include the requirements for structural integrity in the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for these verifications. The appendices A to D (BS5950 1:2000) give the required information on the behavior and methodology to resist tying forces in the design.

Seismic Category
(Base Plate only) Specify the site class which best describes the soil conditions supporting the foundation and base plate.

Connection types
Select the class of connection to be used from this list. The connection types available depend on the code selected.

Available list
A list of all available connection templates for the selected connection type are listed here.

List Operators

Click this button…	to…
>	Add the selected connection template to the Selected list.
>>	Add all connection templates to the Selected list.
<<	Remove all entries in the Selected list.
<	Remove the selected entry from the Selected list.

Selected list
The connection templates selected for design will be added to the list.

Design Connections Individually
Select this option to instruct the program to optimize connections at each joint separately, rather than grouping for the worst case among common connections.

Button	Description
OK	Accepts the design criteria and performs a design for the selected joints using the selected connection templates.
Cancel	Closes the dialog without designing any connections.
Help	Opens the STAAD.Pro help window.

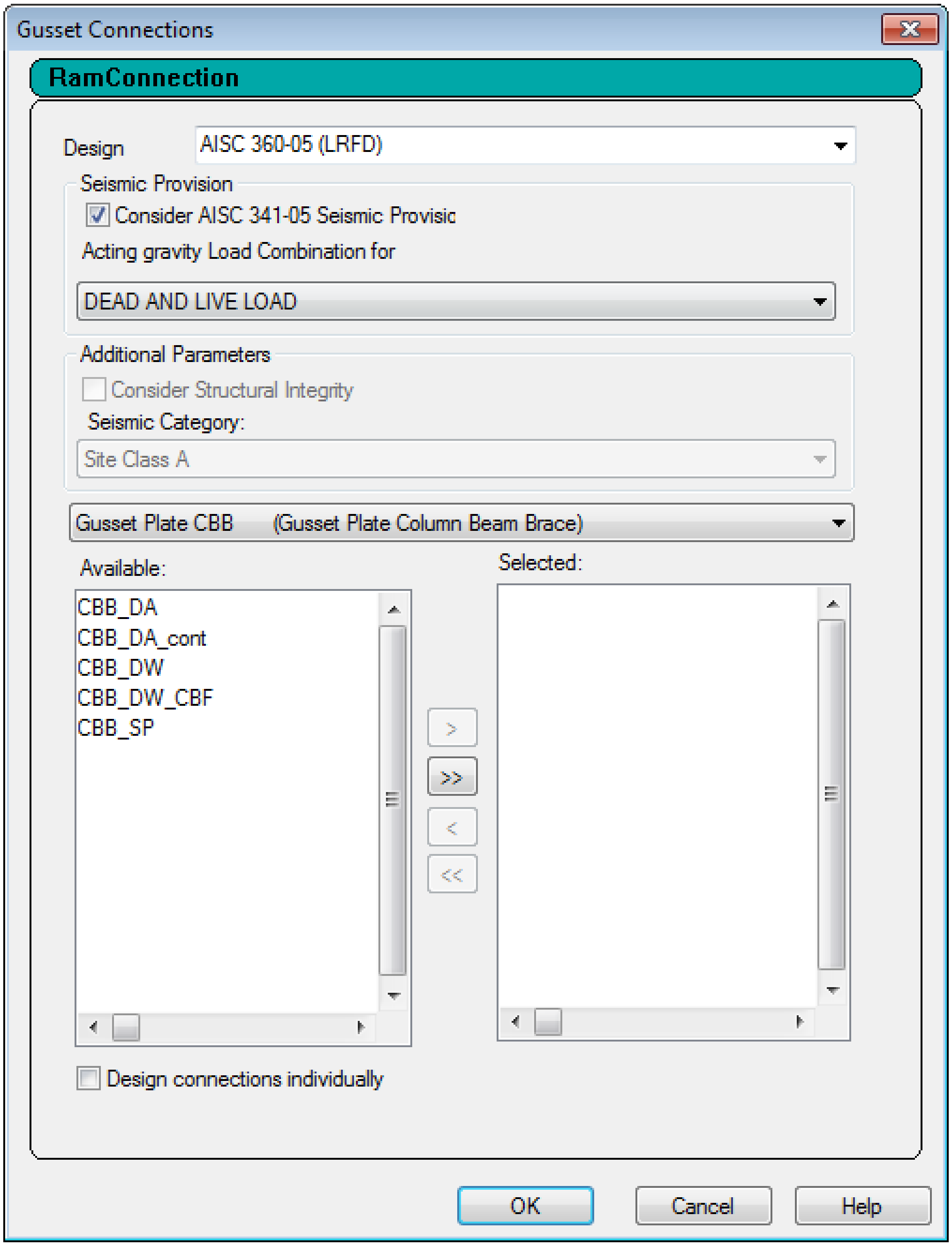
Related Links

- *D. Design Connections Individually* (on page 1037)
- *D. To Select a Smart Connection Template* (on page 1034)

Gusset Connections dialog

Used to assign gusset connection templates to selected joints. A Gusset connection template contains parametric rules similar to those in a Smart connection template but that are limited to connections using gusset plates.

Opens when **Connection Design** > **Assign Gusset Connections** is selected.



Note: The definition of each connection template and the order in which they occur, are defined in a database which can be displayed and/or modified from the **Connection Design** menu. See the RAM Connection manual for more information on editing the connection database.

Design	Select the design code to be used. The field is populated with the default code selected in the RAM Settings dialog.
Seismic Provisions	(AISC codes only)

Select the Consider AISC 341-05 Seismic Provisions option to instruction the program to design the connections per this specification.

A gravity load combination must be selected when this option is used.

Consider Structural Integrity
(BS5950 only) Select this option to include the requirements for structural integrity in the design (Cl 2.4.5.2 BS5950 1:2000). A tie force not less than 75 kN is considered for these verifications. The appendices A to D (BS5950 1:2000) give the required information on the behavior and methodology to resist tying forces in the design.

Seismic Category
(Base Plate only) Specify the site class which best describes the soil conditions supporting the foundation and base plate.

Connection types
Select the class of connection to be used from this list. The connection types available depend on the code selected.

Available list
A list of all available connection templates for the selected connection type are listed here.

List Operators

Click this button...	to...
>	Add the selected connection template to the Selected list.
>>	Add all connection templates to the Selected list.
<<	Remove all entries in the Selected list.
<	Remove the selected entry from the Selected list.

Selected list
The connection templates selected for design will be added to the list.

Design Connections Individually
Select this option to instruct the program to optimize connections at each joint separately, rather than grouping for the worst case among common connections.

Button	Description
OK	Accepts the design criteria and performs a design for the selected joints using the selected connection templates.
Cancel	Closes the dialog without designing any connections.
Help	Opens the STAAD.Pro help window.

Related Links

- *D. Design Connections Individually* (on page 1037)
- *D. To Select a Gusset Connection Template* (on page 1034)

Beam-Girder Identification dialog

Used to switch beam and girder assignments in a Beam-Girder connection. STAAD.Pro will automatically assign the Beam and Girder designations. This dialog allows you to switch the assignment between the two members in the connection.

Opens when **Connections** > **Identify Beam and Girder** is selected.

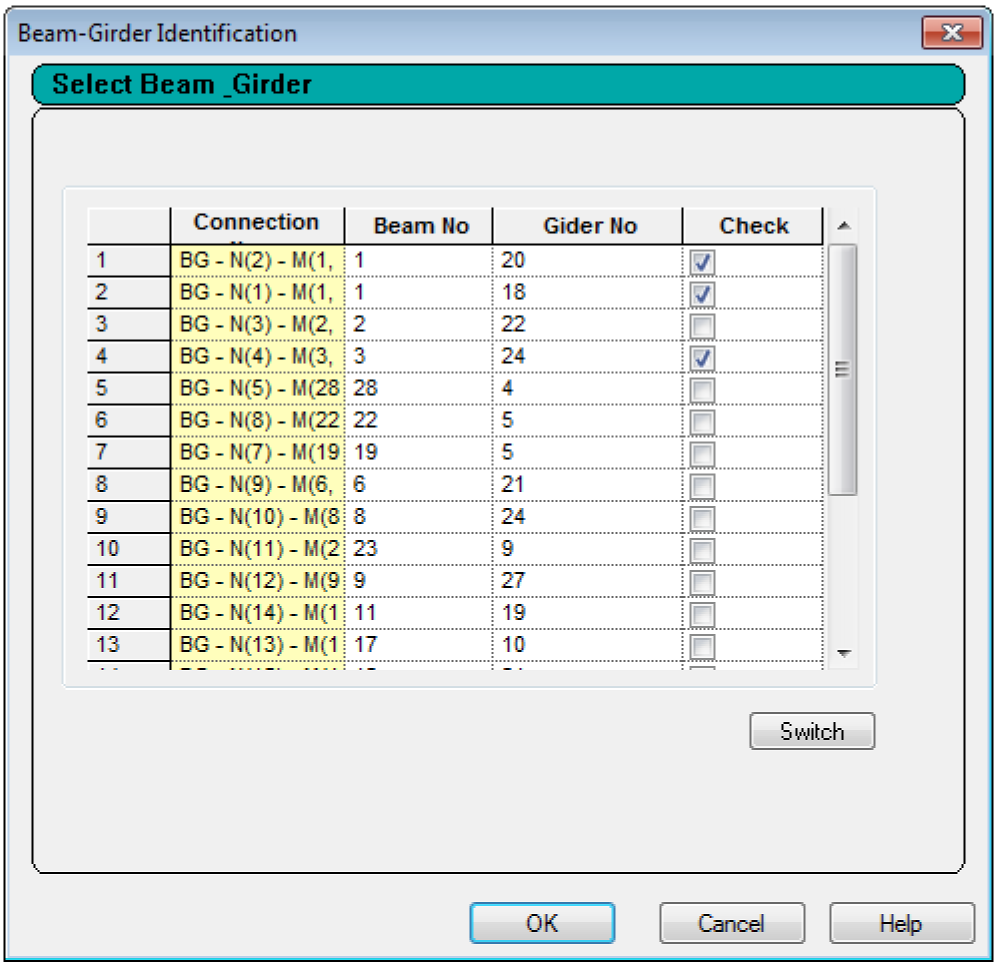


Table of BG Connections	All Beam-Girder (BG) connections in the model are listed here along with the members in the connection which are have been automatically assigned as Beam or Girder. Select the **Check** option for each connection for which you wish to switch member assignments.
Switch	Click this button to swap the **Beam No** and **Girder No** member assignments of members selected in the **Check** column.
OK	Accepts the settings changes and closes the dialog.
Cancel	Closes the dialog without saving changes.

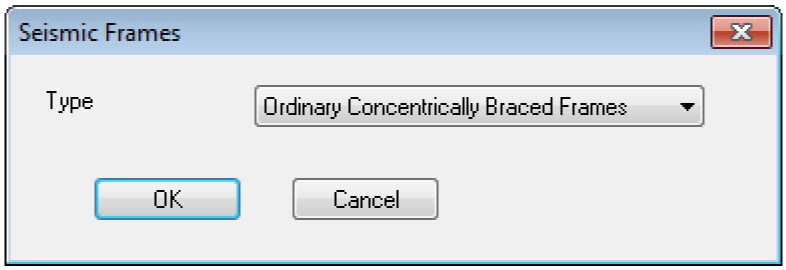
Help Opens the Help window.

Seismic Frames

Used to specify the type of seismic frame used for the selected members creating a seismic frame definition.

Note: This dialog is accessed from the D. Seismic Frames page (on page 1044).

Opens when **Connection** > **Create Seismic Frame** is selected.



Type Select the predefined method of lateral seismic force resisting system the frame represents.

- None
- Ordinary Moment Frames (OMF)
- Intermediate Moment Frames (IMF)
- Special Moment Frames (SMF)
- Ordinary Concentrically Braced Frames (OCBF)
- Special Concentrically Braced Frames (SCBF)

OK Accept the seismic frame assignment and closes the dialog.

Cancel Closes the dialog without making a seismic frame assignment.

RAM Report Export dialog

Used to manage connections and other report details to include in an report.

Note: The report is saved to a Microsoft Office Word document in the same folder as the STAAD.Pro input file, as `inputfile_Report.doc`, where *inputfile* is the STAAD.Pro Input filename.

Connection list Contains a list of all the designed connections. Use the **Up** and **Down** buttons to reorder the connections in the report. Select the connections you want to include in the report.

Data Report Check this option to include the input values in the report.

Result Report Set this option to include the code check output for the connections in the report.

formula Check this option to include the formulas used for code checks in the report.

Individual / Merged Report Select to generate either individual reports for each connection or a single report with all connections merged into a single file.

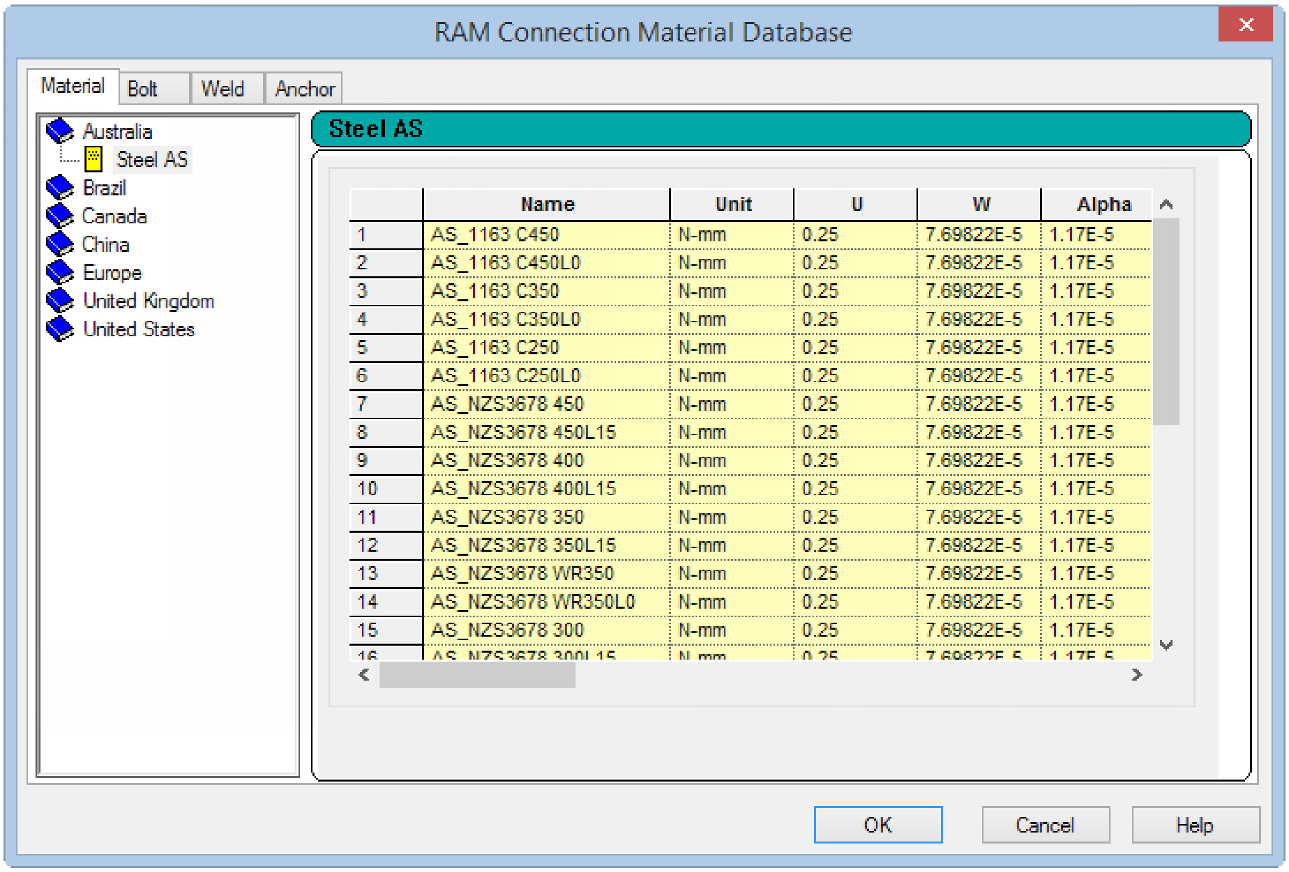
Related Links

- *D. To Export Connection Designs to a Report* (on page 1037)

RAM Connection Material Database dialog

Used to review additional material data required by Connection Design for the design of steel connections. Material strengths are displayed here to allow for the assignment of material grades.

Opens when **Connection Design** > **Show RAM Database** is selected.



Additional material data is required by RAM Connection beyond that which is kept in STAAD.Pro's databases. This information can be viewed and edited from the RAM Material dialog. In the case of the same material being defined in both STAAD and RAM Connection, the local STAAD value is used.

The STAAD materials are read from the STAAD.Pro `.ini` file. Some generic materials do not have any strength materials defined and these should be entered here.

Materials listed in the country tabs (i.e., United States or United Kingdom) are read from RAM Connection. These values are displayed in yellow fields and may not be edited in this dialog.

Materials table The following explains the columns used in the RAM Materials table:

Material Property	Description
Fy	Yield stress
Ry	Yield strength ratio
Rt	Ratio of the expected tensile strength to the specified minimum tensile strength, Fu
Fu	Specified minimum tensile strength
FCu	Ultimate strength of concrete

OK Accepts the material changes entered in the table and closes the dialog.

Cancel Closes the dialog without saving changes.

Help Opens the Help window.

Bolts tab

Database tables lists bolt geometry and material properties. The following tables are included:

- AISC - American Institute of Steel Construction bolts, in imperial units.
  - Db: Bolt Diameter
  - Fv: Nominal shear stress
  - Ft: Nominal tensile stress
  - Tm : Minimum Bolt pretension
  - Mu: Minimum Slip Coefficient
- BS - British Standard bolts, in metric.
  - Db: Bolt Diameter
  - Pt: Tensile Strength
  - Ps: Shear Strength
  - P0: Minimum Shank Tension
  - Mu: Slip Factor

Weld tab

Database table lists weld material properties. The following tables are included:

- AISC - American Institute of Steel Construction welds, in imperial units.
  - Fexx: Weld electrode Strength

Advanced Slab Design tab

Contains tools used for selecting elements of a defined slab as well as for transmitting data to RAM Concept for slab design.

Table 367: Highlight group

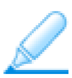
Tool name	Description
 **Highlight Plates**	Selects all plate elements associated with the selected slab definition in the Slabs table.
 **Highlight Beams**	Selects all beam members associated with the selected slab definition in the Slabs table.
 **Highlight Columns**	Selects all column members associated with the selected slab definition in the Slabs table.
 **Highlight Geometry**	Selects all structural elements associated with the selected slab definition in the Slabs table.

Table 368: RAM Concept group

Tool name	Description
 **Export to RAM Concept**	Creates the required data files for RAM Concept to analyze and design the selected slab definition in the Slabs table. **Note:** This command does not open the external slab design program nor does it execute the analysis and design functions in that program.
 **Open in RAM Concept**	Creates the required data files for RAM Concept to analyze and design the selected slab definition in the Slabs table. RAM Concept is launched. **Note:** If RAM Concept is not installed on your computer or if the version installed is not compatible with Advanced Slab Design link, a warning dialog opens to inform you that the data could not be sent to RAM Concept.

Node Tools tab

Note: The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.

Table 369: Model group

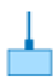
Tool name	Description
 **Move Joint**	Opens the **Move Entities** dialog, which is used to specify the translational offset for moving a selection of nodes.
 **Renumber Nodes**	Opens the **Renumber** dialog, which is used to renumber selected nodes starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.
 **Assign Supports**	Opens the **Supports - Whole Structure** dialog, which is used to define supports and assign them to nodes.

Table 370: View group

Tool name	Description
 **New View**	Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.

Tool name	Description
 **3D Rendering**	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Node dialog

Displays information for the selected node.

Opens when a node is double clicked with the Nodes Cursor.

Node	Displays the node number and coordinates for the currently selected node.
Nodes	Opens the Nodes table (on page 714).
Supports	Opens the Node Supports (on page 715).
Loads	Opens the **Load Values** table.
Reactions	Opens the P. Support Reactions table (on page 2003).
Displacements	Opens the P. Node Displacements table (on page 2002). If no results have been setup, the Results Setup dialog (on page 2999) will be opened first to specify the results you wish to view.
Close	Closes the dialog.

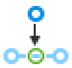
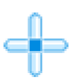
Related Links

- *GS. Object Properties Inspection* (on page 53)

Beam Tools tab

Note: The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.

Table 371: Model group

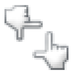
Tool name	Description
 **Move Beams**	Opens the **Move Selected Beams** dialog, which is used to specify the translational offset for moving a selection of beams.
 **Insert Node**	Opens the **Insert Node into Beam #** dialog, which is used to insert one or more nodes at specified distances along selected members.
 **Form Member**	Used to manually form a physical structural member from a selection of one or more connected analytical beam segments.
 **Stretch Beam**	Opens the **Stretch Member(s)** dialog, which is used to increase the length of a member in various ways.
 **Merge Beams**	Opens the **Merge Selected Beams** dialog, which is used to join two collinear beams and replace them with one beam.
 **Intersect Beams**	Splits members at detected intersection points and creates an addition node at this intersection point such that the members are now connected.
 **Renumber Beams**	Opens the **Renumber** dialog, which is used to renumber selected beams starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.

Tool name	Description
 **Define Section Profile**	Opens the **Define Section Profile** dialog, which is used to specify beam stress locations for general user-provided table sections for use in post-processing. **Note:** This tool is only active when a member using a general UPT section is selected in the Postprocessing workflow.
 **Properties**	Opens the **Beam** dialog which displays properties for the selected member. Additional tabs become available after analysis and design is performed.

Table 372: Connection Tags group

Tool name	Description
 **Assign**	Opens the **Assign Connection Tags** dialog and **New Connection tag** dialog, which are used to assign existing connection tags to member ends in the model and to create new connection tags, respectively.
 **Remove**	Opens the **Assign Connection Tags** dialog and initiates the remove connection tag tool for the connections on the selected member. A confirmation dialog opens to confirm the remove action.
 **View**	Opens the **Assign Connection Tags** dialog.
 **Check**	(Active only after a successful analysis has been performed) Opens the **Assign Connection Tags** dialog and **Check Connection Tags** dialog, the latter of which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

Table 373: View group

Tool name	Description
 **New View**	Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.
 **3D Rendering**	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Plugins

Tool name	Description
Add Attribute	Opens the **Member Attribute** dialog which is used to select and apply a member attribute to selected members. Refer to TR.29 Definition of Member Attributes (on page 2340) for additional information on the member attributes assigned using this dialog.
List/Delete Attribute	Opens the **Member Attribute** dialog to display the currently used Member Attribute for a single selected member

Define Section Profile dialog

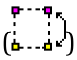
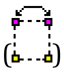
Used to specify beam stress locations for general user-provided table sections for use in post-processing.

Opens when either:

- the **Define Section Profile** tool is selected in the **Model** group on the **Beam** ribbon tab
- **Define Section Profile** is selected from the right-click pop-up menu

Note: This tool is only active when a member using a general UPT section is selected in the Postprocessing workflow.

Enter the local Y and Z coordinates of the section extreme fibers to specify stress points.

You can also "swap" the top and bottom sides () or left and right sides () in order to facilitate entering stress points by clicking the corresponding tools.

Tip: Additional profile name, material, and section properties are provided to assist in identifying the section.

Related Links

- *P. To view stress contour of a General UPT beam* (on page 1986)

Beam dialog

The member query box includes the ability to change many of the member attributes from the dialog box itself including the property definition, specifications (releases, truss, cable, etc.) and beta angle.

A list of the load items assigned to the member are displayed here. New load items can now be added to existing load cases, and, existing load items edited or deleted using the Member Query window.

Select Load Case	Choose the load case to you want to display or edit. This list will be empty if no primary load cases have been defined in the model.
Add new Load Item	Opens the Create New Load Items dialog, which is used to add new load items to the selected load case and assign them to the currently selected member. The list of load items displayed is limited to those which can be assigned to members.
Remove Selected Load Item.	Click to remove an existing load item selected in the list.
Edit Selected Load item	Click to make changes to the currently selected load item in the list. The load item dialog for the load type opens. Make the desired changes and then click **Change**.

Opens when a member is double-clicked with the Beams cursor.

Tip: While this dialog box is open, you may select any other member to Query by double-clicking it. This updates the dialog box contents automatically for the newly selected member. Also, the units can be changed dynamically while this dialog box is open. Changing the units using the *View* | *Options* or *Tools | set Current Display Units* menu affects the contents of this dialog box automatically.

To change the property, member specifications or the beta angle on the member, simply click on one of the buttons. If the member contains output result tabs (Displacement, Steel Design, etc.) in the query box, changing the member attributes will cause these tabs to disappear. This is due to the fact that the current output no longer reflects the new input.

Note: Changing the member attributes for one member will subsequently change the attributes of all other members belonging to the same attribute list. For example, if the current member's property is also assigned to other members, changing the property on the current member will change the property on all the members.

Geometry

Shows a diagram of the member property and lists the length, connecting nodes and their coordinates, and whether the member has a beta angle, any releases or other member specifications.One can assign/change the beta angle of the member by clicking on the **Change Beta** button.

Change Beta	Click to assign or change the beta angle for the selected member.
Change Releases at Start / End	Click to assign or change the start or end release attributes, member offsets, cable specification, truss specification, tension-only specification, compression-only specification, ignore stiffness, or inactive specifications.

Property

Shows a diagram of the member property and lists the length, cross sectional and material properties. One can assign/ change the property or member specifications of the member by clicking on the respective buttons.

Assign/Change Property	Click to assign or change the section property of the member. The choice of properties will only from the same family as the current cross-section. For example, if the current property is an America W12x45, only W, M, S, HP, and B shapes and sizes will be available.
Assign Material	Click to update the material definition with one selected from the drop-down list.

Shear Bending

This tab appears only after running the analysis. It also contains facilities for viewing values for shears and moments, selecting the load cases for which those results are presented, a slider bar for looking at the values at specific points along the member length, and a Print option for printing the items on display.

The slider bar in the member query box can be used to display the shear, bending and deflection values at any point along the beam. A corresponding edit box is also provided to manually type in the distance for which the aforementioned values will be calculated and displayed.

To obtain the values for the shear, bending or deflection at any point along the beam for a specific load case, grab the slider bar (the draggable arrow) and drag it to the desired location on the beam. The location of the slider bar is reported in the *Dist* edit box in the current display units. This edit box can also be manually changed which conversely, changes the location of the slider bar. The values for the results in question will automatically be updated every time the slider bar moves.

Deflection

This tab appears only after running the analysis. Shows deflection diagram in the global or local directions for the selected load case. Also lists tabulated deflection values. Points of inflection are displayed on the diagram. The points of inflection or contraflexure are the distances along the member where the sign of the force, deflection or moment changes (the point where these values are zero).

Refer to the next figure to see where the points of inflection are labeled. The distances are always reported from the start of the member (in the figure, the left-hand side of the member).

Composite Property

The dialog box displays various properties for the composite beam including effective width, whether shoring during construction has been considered or not, grade of concrete, unit weight of concrete, thickness of concrete above flutes, rib properties, stud properties etc. These data will enable the user to get a clear picture of the geometric properties for the composite beam.

Design Property

If the STAAD input file contains instructions for steel design - Member Selection (refer to TR.49.1 Member Selection Specification (on page 2678), and US or UK Example 1 in the Application Examples) which may result in a change of section property from the one originally assigned by the user - that new section name, and associated property values will be displayed under this tab. One can assign/change the material attributes for the beam by clicking on the Assign Material button.

Steel Design

This tab appears only after running the analysis if a steel design operation has been performed on the selected member. Shows a diagram of the member property and lists the length, design stresses, critical load, code being used, ratio, result, critical condition and slenderness value.

fSB	Actual Steel Stress before Concrete Hardens
FSB	Allowable Steel Stress before Concrete Hardens
fCB	Actual Concrete Stress before Hardening
FCB	Allowable Concrete Stress before Hardening
fSA	Actual Steel Stress after Concrete Hardens
FSA	Allowable Steel Stress after Concrete Hardens
fCA	Actual Concrete Stress after Hardening
FCA	Allowable Concrete Stress after Hardening
Critical load	The table displays information like critical load case, critical force values, and location of the critical section along the member length
Code	The design code followed for designing the composite beam
Result	Displays whether the beam has PASSED or FAILED the check
Ratio	Ratio of actual to allowable stresses
Critical	Reports the critical condition. In the previous dialog box CONC-STR-AH is displayed which indicates that Concrete Stress after hardening is the critical condition.
KLR	Reports the slenderness ratio (kL/r for compression or L/r for tension). The slender ratio value displayed is based on the critical axial force and is marked “T” for tension or “C” for compression.
STUDS	Display the number of shear connectors required and diameter of shear connectors.

Castellated Beam Design

Concrete Design

Related Links

- *GS. Object Properties Inspection* (on page 53)

Physical Member dialog

Used to display information about a selected physical member, similar to the member query dialog.

Opens when physical member is double-clicked with the Physical Member cursor.

Related Links

- *GS. Object Properties Inspection* (on page 53)

Assign Connection Tags dialog

Used to assign existing connection tags to member ends in the model.

Opens when **Connection Tags** > ***any* Connection Tags** is selected from the right-click pop-up menu when a member is selected.

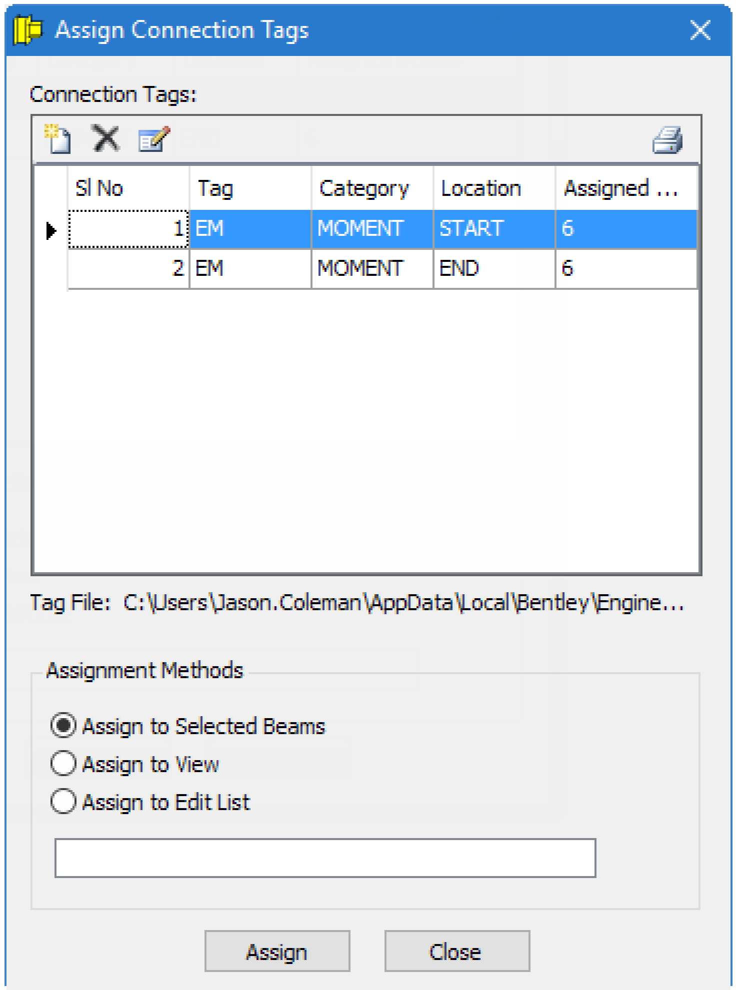


*Figure 353: The **Assign Connection Tags** dialog*

Table 374: Dialog toolbar controls

Toolbar icon	What It Does
New	Opens the New Connection Tag dialog, which is used to define new connection tags and optionally assign member end restraints associated with a connection tag.
Delete	Used to remove the selected connection tag entry from the table. A confirmation dialog opens to confirm you want to remove the entry.

Toolbar icon	What It Does
Check	(Disabled if no current analysis results are available for the current STAAD.Pro model) Opens the Check Connection Tags dialog, which is used to check the load case results from the analysis against the defined connection capacities in the Connection Tags XML data file.

table Each row in the table is a separate connection tag added to the model from the Connection Tags XML file.

Assignment Methods Select on of the following methods for assigning connection tags:

- **Assign to Selected Beams —** Uses the current members selected in the main view window
- **Assign to View —** Uses all members visible in the ain view window
- **Assign to Edit List —** Uses the member numbers listed in the associated text field

> **Tip:** You can type a dash between two numbers to indicate all member numbers, inclusive.

Assign Assigns the currently selected connection tags in the table using the selected Assignment Method. The members assigned area added to the Assigned Beams cell in the row.

Close Closes the dialog.

Related Links

- *D. To create a connection tag* (on page 1047)

New Connection Tag dialog

Used to define new connection tags and optionally assign member end restraints associated with a connection tag.

Opens when either:

- **Connection Tags** > **Remove Connection Tags** is selected from the right-click pop-up menu when a member is selected, or
- the **New** tool is selected in the Assign Connection Tags dialog

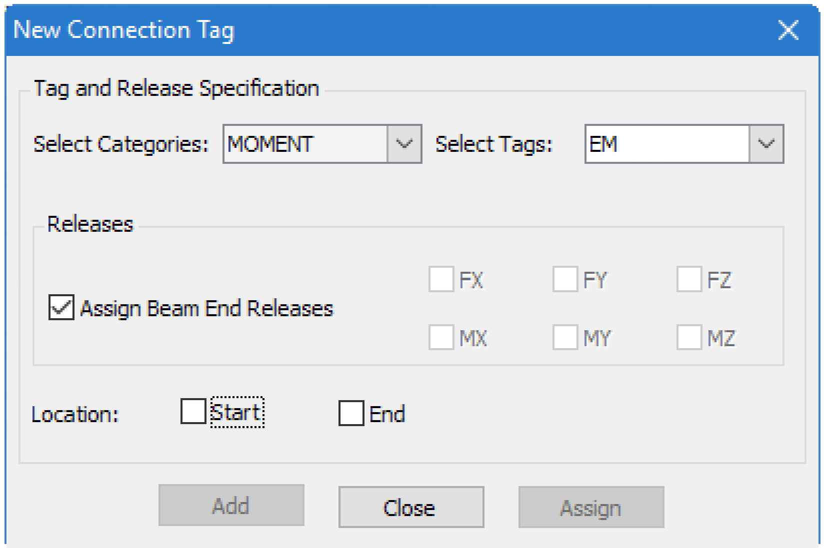


*Figure 354: The **New Connection Tag** dialog*

Select Categories Use the drop down list to select one of the connection tag categories defined in the Connection Tag XML file. Additional category descriptions are displayed.

Select Tags Use the drop down list to select one of the connection tags defined in the selected category. When selected, the member end releases defined for this connection tag are indicated in the inactive degree of freedom check boxes in the Releases group.

Assign Beam End Releases Set this check box option on to assign the indicated member end releases for the connection tag in the STAAD.Pro model when the connection tag is assigned.

> **Tip:** This is recommended so the releases match those assumed by the connection tag.

Location Select either the **Start** or **End** member end (or both) for adding this connection tag to the Assign Connection Tags dialog table or assigning to currently selected members.

> **Note:** At least one Location must be selected in order to add or assign the connection tag.

Start – The start node of the member.
End – The end node of the member.

Add Adds the selected connection tags at the selected location to the Assign Connection Tags dialog table.

Close Closes the dialog.

Assign Assigns the current connection tags to the current member selection in the STAAD.Pro model. These connection tags will also be added to the Assign Connection Tags dialog.

Related Links

- *D. To create a connection tag* (on page 1047)

Remove Connection Tags dialog

Used to select which member end from which the connection tag assignment will be removed.

Opens when **Connection Tags** > **Remove Connection Tags** is selected from the right-click pop-up menu when a member is selected.

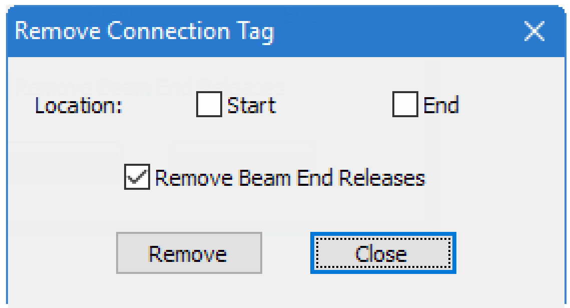


*Figure 355: The **Remove Connection Tag** dialog*

Location	Select one or both of the **Start** or **End** member ends for removing a connection tag assignment.
Remove Beam End Releases	Set this option to have the associated member end releases returned to default (fixed) in the STAAD.Pro input file.
Remove	Performs the assignment removal. A confirmation dialog opens
Close	Closes the dialog without removing any connection tag assignments.

Related Links

- *D. To remove connection tag assignments* (on page 1049)

Member Attribute dialog

Used to assign StructLink and member type attributes to selected members.

Note: These member attributes are separate attribute definitions from those which can be associated with adding new members (e.g., beta angle, section, etc.).

Opens when either:

- the **Add Attribute** tool from the **Plugins** group is selected on the **Beam Tools** ribbon tab, or
- **Add Member Attribute** is selected from the right-click pop-up menu

Attribute Name	Select the attribute you want to define for the member selection: • **STRUCTLINK -** the StructLink attribute • **MEMBTYPE -** member functionality used in RAM Connection design **Tip:** There are separate tools available on the **Beam Tools** ribbon tab and on the right-click pop-up menu for assigning Connection Tags, which are another member attribute type.

Attribute Values	For the selected **Attribute Name**, a list of predefined attribute values can be selected.

Related Links

- *D. To assign member type attributes* (on page 1031)

Plate Tools tab

This ribbon tab is available when one or more plate objects are selected.

Note: The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.

Table 375: Model group

Tool name	Description
 **Move Plate**	Opens the **Move Selected Plates** dialog, which is used to specify the translational offset for moving a selection of plates.
 **Renumber Plates**	Opens the **Renumber** dialog, which is used to renumber selected plates starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.
 **Generate Plate Mesh**	Used to generate a finite element mesh for an existing plate element. The polygon can be meshed into quadrilateral or triangular elements. You have control over parameters like number of divisions along each side of the polygon. Polygonal holes can also be defined within the surface during the meshing process for polygonal meshing.
 **Properties**	Opens the **Plate** dialog which displays properties for the selected element. Additional tabs become available after analysis and design is performed.

Table 376: View group

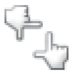
Tool name	Description
 **New View**	Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.
 **3D Rendering**	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Plate dialog

Used to view a summary of the input and output data for a plate element, including geometry, properties, and output results.

If the structure has been successfully analyzed, there will be three tabs on the plate query box: Geometry, Property Constants and Stresses. If there are no analysis results, the last tab (Stresses) will not be shown.

Opens when a quad or triangular plate element is double-clicked with the Plates cursor.

Geometry tab

Displays the plate incidences, the plate corner coordinates and the lengths of each of the edges.

Plate Spec There are certain attributes which can be assigned to plates in order to simulate characteristics like the type of connections that are present at their corners. These are - Releases of individual degrees of freedom at its nodes, defining them as Plane Stress elements, ignoring their stiffness for the purpose of analysis, etc. If such attributes are assigned, those will be listed against the Plate Spec: field.

Property Constants tab

displays the thickness of each of the sides of the plate and the material properties. One may change/assign the properties of the plate by clicking on the Assign/Change Property button. One can also assign/change the material attributes by clicking on the Assign Material button.

Center Stresses tab

Displays the 8 center stresses along with the top and bottom Principal, Von Mises and Tresca stresses for a particular load case. The load case can be changed using the Load List dropdown box.

Corner Stresses tab

Used to display the stresses at the corner node points of the selected plate. The node numbers of the selected plate and the stresses at those nodes are displayed.

Load List	Select any one of the predefined load cases for displaying the corner stresses of the selected plate due to that particular type of loading.
Plate Corner Stresses	Different types of stresses at the corner nodal points of the selected plate under the specified load case are listed here.

Princ Stress and Disp tab

Used to display the nodal displacements at corner points of the selected plate.

Load List	Select any one of the predefined load cases for displaying the displacements at the corner node points and also the principal stresses of the selected plate due to that particular type of loading.
Plate Corner Displacement	After selecting a particular type of load case in the Load List, displacements in the X, Y and Z direction at the corner node points of the selected plate will be listed here.
Plate Principal Stresses	Top and bottom principal stresses of the selected plate under the specified load case are displayed here.

Related Links

- *GS. Object Properties Inspection* (on page 53)

Surface Query dialog

Used to view detailed information for one surface entity at a time.

Opens when a surface element is double-clicked with the Surface cursor.

While this dialog box is open, you can query any other surface element by double-clicking on it. The dialog updates automatically for the newly selected element. Similarly, the display units can be changed dynamically while this dialog box is open using the Change Graphical Display dialog.

Geometry tab

Shows a diagram of the surface element and a table listing all nodes on the perimeter of the element, as well as node to node distances along the perimeter.

Property tab

Lists surface element’s thickness and properties of the element’s material: Elasticity, Poisson’s ratio, Density, and thermal expansion coefficient (Alpha).

Surface Forces tab

(Available after a successful shearwall analysis and design) Reports internal forces (Fx, Fy, Fxy, Mx, My, Mz, Qx, and Qy) for five levels at which the shear wall design is carried out: 1/8h, 1/4h, 1/2h, 3/4h, and 7/8h. The forces may be viewed for all load cases / combinations included in the design.

Concrete Design (Shear wall) tab

(Available after a successful shearwall analysis and design) This page of the dialog box contains a full summary of the design, including shear wall geometry, concrete and steel strengths, min. reinforcing ratios, and reinforcing details for all of the five design levels. In addition, a diagram of the horizontal cross-section of the wall is shown for each level.

Related Links

- *GS. Object Properties Inspection* (on page 53)

Solid Tools tab

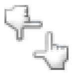
This ribbon tab is available when one or more solid objects are selected.

Note: The Clipboard group contains **Cut**, **Copy**, **Paste**, and **Delete** tools found on the **Geometry** ribbon tab.

Table 377: Model group

Tool name	Description
 **Move Solid**	Opens the **Move Selected Solids** dialog, which is used to specify the translational offset for moving a selection of solids.
 **Renumber Solids**	Opens the **Renumber** dialog, which is used to renumber selected solids starting with a specified number. The numbering sequence can be in an ascending or descending order and the order can be sorted by some criteria if needed.

Table 378: View group

Tool name	Description
 **New View**	Opens the **New View** dialog, which is used to create a new view window for displaying the selected structural elements. You are prompted to indicate whether the selected view would be opened in a new (child) window or whether it would replace the current (parent) view. Any number of “child” view windows in this way. **Note:** This option becomes active only after you select one or more structural elements on screen.
 **3D Rendering**	Used to render the model using true lighting, reflection and shading in a separate window. It enables walk-through, dynamic zoom and panning capabilities in the 3D rendered view. Once the 3D Rendering option is chosen, a separate window opens displaying the rendered view. The structure can be dynamically rotated about all three axes by simply holding the left mouse button down and dragging the structure in the intended direction. Right-clicking the mouse button will display a myriad of viewing options. Depending on the material used (steel, concrete, etc.), an appropriate texture will be applied to the structure. A property or material must be assigned to the entities of the model before this feature can be used. This is for visual and presentation purposes only.

Solid dialog

Similar to the beam and plate query dialog box, this will enable the users to obtain information regarding only the geometry of the solid element. As in the case of beams or plates, the user can simply double click on the solid element to see the element node numbers and the global X, Y and Z coordinates of the boundary nodes without having to scan through the Nodes table.

Opens when a solid element is double-clicked with the Solids cursor.

The table in the query box (which is not editable) displays the solid node numbers and the corresponding global X, Y and Z coordinates for the nodes.

Related Links

- *GS. Object Properties Inspection* (on page 53)

B
Verification Examples

The examples within this section are used to validate the accuracy of the STAAD.Pro analysis and design engine.

V. Notes on Comparisons

In each verification problem, a table is used to summarize the numerical outputs from STAAD.Pro with a reference or hand calculations. A difference column presents the percent deviation from the reference, when a significant difference exists.

Where a non-significant difference (typically taken to be less than 0.5%) is present, the difference is given is "negligible." Otherwise, the percent difference in results is listed. Where a difference of greater than 2% exists, a comment is provided regarding the difference.

This level of significance is intended to reflect that small differences in rounding or calculation methods do not present a realistic difference in the calculated result.

V.01 Beams

V. Axially Loaded Column

To find support reactions due to an axial load applied at two locations on a column with both ends pinned.

Reference

Hand calculation using the following reference:

Timoshenko, S., *Strength of Materials*, Part I, D. Van Nostrand Co., Inc., 3rd Edition, 1956. Page 26, Problem 10.

Problem

Find the support reactions at the end joints 1 and 4.

$E = 30\times(10)^6$ psi

$F_1 = 1,000$ lbs

$F_2 = 500$ lbs

$L = 3$ in.

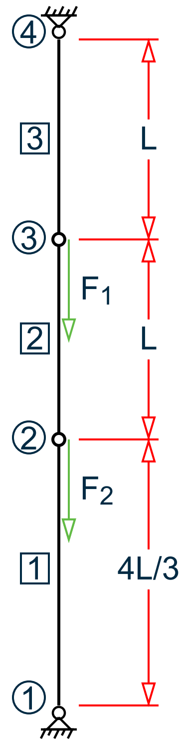


Figure 356: Pinned-pinned column with axial loads

Comparison

Table 379: Comparison of results

Result	Theory	STAAD.Pro	Difference
Reaction at Node 1 (lb)	600	600	none
Reaction at Node 4 (lb)	900	900	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Axially Loaded Column.STD`is typically installed with the program.

```
STAAD PLANE :A PINNED-PINNED COLUMN
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: Timoshenko, Strength of Materials, Part 1
*            Problem 10, Page 26
*
INPUT WIDTH 79
```

```
UNIT INCHES POUND
JOINT COORDINATES
1 48 72 0; 2 48 76 0; 3 48 79 0; 4 48 82 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1 IZ 0.0001
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 PINNED
LOAD 1 AXIAL LOAD
JOINT LOAD
3 FY -1000
2 FY -500
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT   LOAD   FORCE-X   FORCE-Y   FORCE-Z    MOM-X     MOM-Y     MOM Z
      1    1       0.00    600.00      0.00      0.00      0.00      0.00
      4    1       0.00    900.00      0.00      0.00      0.00      0.00
```

V. Beam on Elastic Foundation

To find deflection and stress at the center due to a uniform, static load on a simply supported beam on elastic foundation.

Reference

Hand calculation using the following reference:

Peterson, F.E., *Elastic Analysis for Structural Engineering (EASE2), Example Problem Manual*, Engineering Analysis Corporation, Berkeley, CA, 1981.

Problem

Find the vertical deflection and bending stress at the center of the beam.

beam width, $b = 1.0$ *in.*
beam depth, $h = 7.114$ *in.*

$$E = 30 \times (10)^6 \text{ psi}$$

L = 240 in.

w_u = 43.3 lb/in.

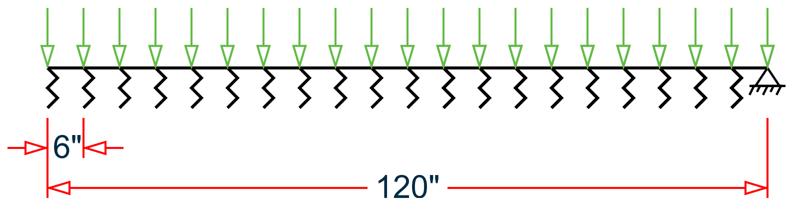


Figure 357: One-half beam for mathematical model

Comparison

Table 380: Comparison of results

Result	Theory	STAAD.Pro	Difference
Bending stress, σ (psi)	18,052	18,053.29	none
Vertical deflection (in.)	1.0453	1.04549	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Beam on Elastic Foundation.STD` is typically installed with the program.

```
STAAD SPACE :A SIMPLY SUPPORTED BEAM ON ELASTIC FOUNDATION
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: PETERSON, EASE2, EXAMPLE PROBLEM MANUAL
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0 7.114 0; 3 0 0 6; 4 0 7.114 6; 5 0 0 12; 6 0 7.114 12;
7 0 0 18; 8 0 7.114 18; 9 0 0 24; 10 0 7.114 24; 11 0 0 30;
12 0 7.114 30; 13 0 0 36; 14 0 7.114 36; 15 0 0 42; 16 0 7.114 42;
17 0 0 48; 18 0 7.114 48; 19 0 0 54; 20 0 7.114 54; 21 0 0 60;
22 0 7.114 60; 23 0 0 66; 24 0 7.114 66; 25 0 0 72; 26 0 7.114 72;
27 0 0 78; 28 0 7.114 78; 29 0 0 84; 30 0 7.114 84; 31 0 0 90;
32 0 7.114 90; 33 0 0 96; 34 0 7.114 96; 35 0 0 102; 36 0 7.114 102;
37 0 0 108; 38 0 7.114 108; 39 0 0 114; 40 0 7.114 114; 41 0 0 120;
42 0 7.114 120;
ELEMENT INCIDENCES SHELL
1 1 2 4 3; 2 3 4 6 5; 3 5 6 8 7; 4 7 8 10 9; 5 9 10 12 11;
6 11 12 14 13; 7 13 14 16 15; 8 15 16 18 17; 9 17 18 20 19;
10 19 20 22 21; 11 21 22 24 23; 12 23 24 26 25; 13 25 26 28 27;
14 27 28 30 29; 15 29 30 32 31; 16 31 32 34 33; 17 33 34 36 35;
18 35 36 38 37; 19 37 38 40 39; 20 39 40 42 41;
```

```
ELEMENT PROPERTY
1 TO 20 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED BUT KFY 78.125
41 FIXED BUT FZ MX
2 FIXED BUT FY
3 5 7 9 11 13 15 17 19 21 23 25 27 29 31 33 35 37 -
39 FIXED BUT FZ MX KFY 156.25
LOAD 1 UNIFORM LOAD OF 43.4 LBS/IN
JOINT LOAD
4 6 8 10 12 14 16 18 20 22 24 26 28 30 32 34 36 38 40 FY 260.4
2 42 FY 130.2
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1 2
PRINT ELEMENT JOINT STRESSES LIST 1 2
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   1.04548   0.00000   0.00000   0.00000   0.00000
      2    1    0.00000   1.04549   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
    47. PRINT ELEMENT JOINT STRESSES LIST 1 2
  ELEMENT  JOINT    STRESSES LIST
      :A SIMPLY SUPPORTED BEAM ON ELASTIC FOUNDATION           -- PAGE NO.
4
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
      1      1         0.00        0.00        0.00        0.00        0.00
                      37.23       37.23       35.30       -0.00       -6.82
                      37.85       37.85
       TOP : SMAX=      36.57 SMIN=      -1.27 TMAX=      18.92 ANGLE=-10.6
       BOTT: SMAX=      36.57 SMIN=      -1.27 TMAX=      18.92 ANGLE=-10.6
         JOINT         0.00        0.00        0.00        0.00        0.00
           1      18071.09    18071.09       35.32   -18053.40       -6.80
       TOP : SMAX=      35.32 SMIN=  -18053.41 TMAX=    9044.36 ANGLE= -0.0
       BOTT: SMAX=      35.32 SMIN=  -18053.41 TMAX=    9044.36 ANGLE= -0.0
         JOINT         0.00        0.00        0.00        0.00        0.00
           2      18035.75    18035.75       35.32    18053.38       -6.84
       TOP : SMAX=   18053.38 SMIN=      35.31 TMAX=    9009.03 ANGLE=-90.0
       BOTT: SMAX=   18053.38 SMIN=      35.31 TMAX=    9009.03 ANGLE=-90.0
         JOINT         0.00        0.00        0.00        0.00        0.00
```

```
           4      18035.79    18035.79       35.29    18053.40       -6.84
       TOP : SMAX=   18053.41 SMIN=      35.28 TMAX=    9009.06 ANGLE=-90.0
       BOTT: SMAX=   18053.41 SMIN=      35.28 TMAX=    9009.06 ANGLE=-90.0
         JOINT         0.00        0.00        0.00        0.00        0.00
           3      18071.05    18071.05       35.29   -18053.38       -6.80
       TOP : SMAX=      35.29 SMIN=  -18053.38 TMAX=    9044.33 ANGLE= -0.0
       BOTT: SMAX=      35.29 SMIN=  -18053.38 TMAX=    9044.33 ANGLE= -0.0
      2      1         0.00        0.00        0.00        0.00        0.00
                      50.05       50.05       35.22       -0.00      -20.53
                      54.10       54.10
       TOP : SMAX=      44.66 SMIN=      -9.44 TMAX=      27.05 ANGLE=-24.7
       BOTT: SMAX=      44.66 SMIN=      -9.44 TMAX=      27.05 ANGLE=-24.7
         JOINT         0.00        0.00        0.00        0.00        0.00
           3      18001.93    18001.93       35.27   -17984.23      -20.48
       TOP : SMAX=      35.29 SMIN=  -17984.25 TMAX=    9009.77 ANGLE= -0.1
       BOTT: SMAX=      35.29 SMIN=  -17984.25 TMAX=    9009.77 ANGLE= -0.1
         JOINT         0.00        0.00        0.00        0.00        0.00
           4      17966.57    17966.57       35.27    17984.15      -20.58
       TOP : SMAX=   17984.17 SMIN=      35.25 TMAX=    8974.46 ANGLE=-89.9
       BOTT: SMAX=   17984.17 SMIN=      35.25 TMAX=    8974.46 ANGLE=-89.9
         JOINT         0.00        0.00        0.00        0.00        0.00
           6      17966.71    17966.71       35.17    17984.23      -20.58
       TOP : SMAX=   17984.25 SMIN=      35.15 TMAX=    8974.55 ANGLE=-89.9
       BOTT: SMAX=   17984.25 SMIN=      35.15 TMAX=    8974.55 ANGLE=-89.9
         JOINT         0.00        0.00        0.00        0.00        0.00
           5      18001.79    18001.79       35.17   -17984.15      -20.48
       TOP : SMAX=      35.20 SMIN=  -17984.17 TMAX=    9009.68 ANGLE= -0.1
       BOTT: SMAX=      35.20 SMIN=  -17984.17 TMAX=    9009.68 ANGLE= -0.1
      :A SIMPLY SUPPORTED BEAM ON ELASTIC FOUNDATION           -- PAGE NO.
5
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          1.805341E+04 -1.805341E+04  9.044362E+03  5.004878E+01  5.409544E+01
 PLATE NO.       1             1             1             2             2
 CASE  NO.       1             1             1             1             1
```

V. Bent Beam Thermal Loading

To find member forces and moments due to a temperature load on a Zee shaped plane bent.

Reference

Seeley, F.B., and Smith, J.O., *Advanced Mechanics of Materials*, 2nd Edition, John Wiley and Sons, 1955, Pages 494-497.

Problem

Calculate reactions and maximum moments in the structure due to a temperature increase of 430 ºF. Do not consider shear deformation.

$E = 26{,}400$ ksi

$\alpha = 7.26744 \times 10^{-6}$ in/in/ºF

OD = 12 in

ID = 10.255 in

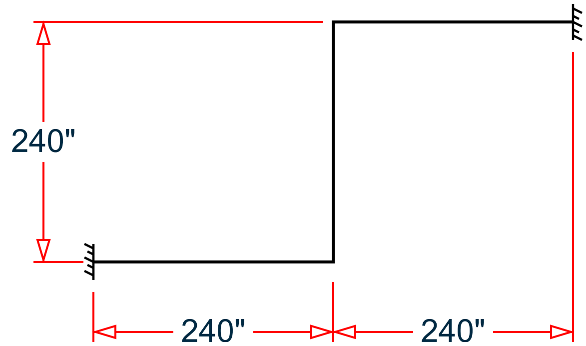


Figure 358: Frame subject to temperature change

Comparison

Table 381: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
Horizontal reaction (lbs)	8,980	8,949.43	negligible	
Vertical reaction (lbs)	7,756	7,729.42	negligible	
Moment at supports (in·lb)	783,750	781,127.75	negligible	
Moment at node 2 (in·lb)	1,077,656	1,073,932.12	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Bent Beam Thermal Loading.STD` is typically installed with the program.

```
STAAD PLANE :A FIXED-FIXED ZEE SHAPED PLANE BENT BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: Seeley, F.B., and Smith, J.O., Advanced Mechanics of Materials,
*             2nd Edition, John Wiley and Sons, 1955, Pages 494-497
*
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
```

```
1 0 0 0; 2 240 0 0; 3 240 240 0; 4 480 240 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST PIPE OD 12 ID 10.255
PRINT MEMBER PROPERTIES
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.64e+07
POISSON 0.3
ALPHA 7.26744e-06
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
* DONOT PROVIDE POISSON'S RATIO, NO SHEAR DEFORMATION CONSIDERED
SUPPORTS
1 4 FIXED
LOAD 1
TEMPERATURE LOAD
1 TO 3 TEMP 430
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 1
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1   8949.43   7729.42      0.00      0.00      0.00 781127.94
     4    1  -8949.43  -7729.42      0.00      0.00      0.00 781127.94
   ************** END OF LATEST ANALYSIS RESULT **************
    34. PRINT MEMBER FORCES LIST 1
  MEMBER   FORCES   LIST     1
     :A FIXED-FIXED ZEE SHAPED PLANE BENT BEAM               -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1   8949.43   7729.42     0.00      0.00      0.00  781127.94
                2  -8949.43  -7729.42     0.00      0.00      0.00 1073932.12
```

V. Bent Cantilever Deflection

Find deflection due to load at the free end of a cantilever plane bent.

Reference

Kinney, J. S., *Indeterminate Structural Analysis*, Addison - Wesley Publishing Co., 1957, Page 13, Problem 4 - 38.

Problem

Find the vertical, horizontal and rotational deflection components of point A.

E = 30,000 ksi

I = 200 in^4

A = 10 in^2

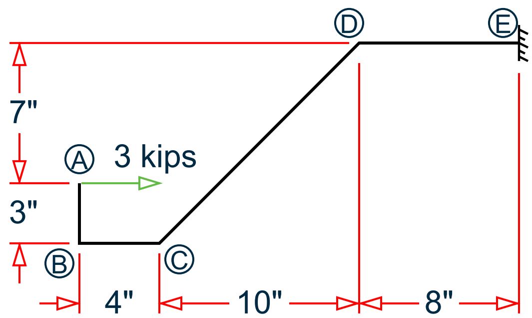


Figure 359: Bent plate frame

Comparison

Table 382: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Deflection right, δ_x (in)	0.53	0.53056	none
Deflection down, δ_y (in)	1.16	-1.17109	<1%
Rotation, θ (rad)	0.0049	0.00488	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Bent Cantilever Deflection.STD` is typically installed with the program.

```
STAAD PLANE :A CANTILEVER PLANE BENT
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: INDETERMINATE STRUCTURAL ANALYSIS, KINNEY, 1957,
*    ADISON-WESLEY PUBLISHING CO., PAGE 113, PROBLEM 4-38
*
UNIT FEET KIP
JOINT COORDINATES
1 0 3 0; 2 0 0 0; 3 4 0 0; 4 14 10 0; 5 22 10 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
```

```
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 10 IZ 200
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
5 FIXED
LOAD 1 HORIZONTAL JOINT LOAD
JOINT LOAD
1 FX 3
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.53056  -1.17109   0.00000   0.00000   0.00000   0.00488
```

V. Curved Beam

To find the out-of-plane deflection and stress in a Circular cantilever member with a concentrated load at the free end.

Reference

Hand calculation using the following reference:

Timoshenko, S., *Strength of Materials*, Part I, D. Van Nostrand, 3rd Edition., 1955.

Problem

Calculate the displacement at the free end and the bending stress at the fixed end due to a concentrated load producing out–of–plane bending.

$E = 30\times(10)^6$ psi

$P = 50$ lb

$r = 100$ in.

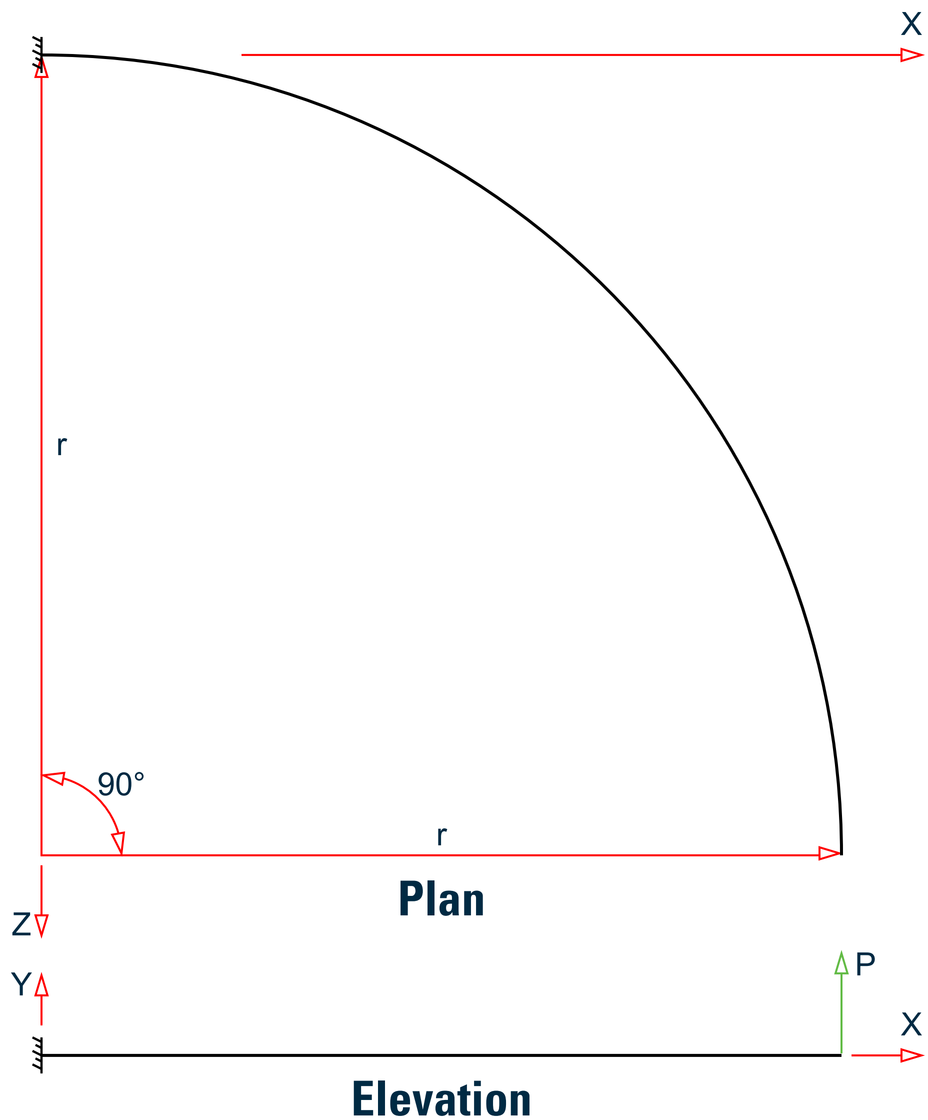


Figure 360: Curved beam

Comparison

Table 383: Comparison of results

Result	Theory	STAAD.Pro	Difference	Comments
Maximum deflection at free end (in.)	2.648	2.64658	negligible	

Result	Theory	STAAD.Pro	Difference	Comments
Principle stress (psi)	6,366.0	6,638.1	4.3%	The result from a classical theory of a beam curved in plan is compared with the result from a piecewise linear set of beams, which closely resembles the behavior of a curved beam, but not exactly. Hence the difference in results. The difference may be further reduced to a certain level by discretizing the curved beam in more smaller subdivisions.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Curved Beam.STD`is typically installed with the program.

```
STAAD SPACE :A CURVED BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  REFERENCE: TIMOSHENKO, S., "STENGTH OF MATERIALS, PART I, ELEMENTARY
*             THEORY AND PROBLEMS", 3RD EDITION, D. VAN NOSTRAND CO.,
*             INC., NEW YORK, 1955.
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 100 0 0; 2 99.619 0 -8.716; 3 98.481 0 -17.365; 4 96.593 0 -25.882;
5 93.969 0 -34.202; 6 90.631 0 -42.262; 7 86.603 0 -50;
8 81.915 0 -57.358; 9 76.604 0 -64.279; 10 70.711 0 -70.711;
11 64.279 0 -76.604; 12 57.358 0 -81.915; 13 50 0 -86.603;
14 42.262 0 -90.631; 15 34.202 0 -93.969; 16 25.882 0 -96.593;
17 17.365 0 -98.481; 18 8.716 0 -99.619; 19 0 0 -100;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19;
MEMBER PROPERTY AMERICAN
1 TO 18 PRIS AX 3.14 IX 1.57 IY 0.7854 IZ 0.7854 YD 2 ZD 2
DEFINE MATERIAL START
```

```
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
19 FIXED
LOAD 1 POINT LOAD APPLIED AT THE FREE END
JOINT LOAD
1 FY 50
PERFORM ANALYSIS
PRINT MEMBER STRESSES LIST 18
PRINT JOINT DISPLACEMENTS LIST 1
FINISH
```

STAAD Output

```
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
     18    1   .0      0.0        0.0    6082.7    6082.7     23.9      0.0
              1.0      0.0 C      0.0    6638.1    6638.1     23.9      0.0
   ************** END OF LATEST ANALYSIS RESULT **************
    39. PRINT JOINT DISPLACEMENTS LIST 1
  JOINT    DISPLACE LIST     1
      :A CURVED BEAM                                         -- PAGE NO.
4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   2.64658   0.00000  -0.02438   0.00000   0.01076
```

V. Deflection and Reactions in a Beam

To find the deflection and support reactions due to a trapezoidally varying load applied on part of the span of a pinned-fixed beam.

Reference

Hand calculation using the following reference:

Roark's Formulas for Stress and Strain, Warren C. Young, 6th edition, McGraw-Hill, Table 3, Case (2c), p.103

Problem

The beam in the following geometric, load, and section properties: $a = 3\ m$, $b = 4.5\ m$, $w_a = 4\ KN/m$, $w_l = 7\ KN/m$, $I_Z = 5{,}000\ cm^4$, $E = 200\ KN/mm^2$.

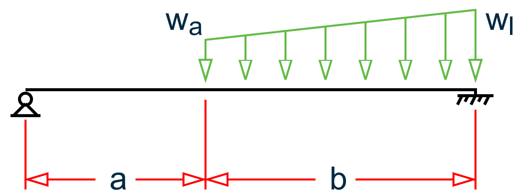


Figure 361: Beam with partial, trapezoidal load

Theoretical Solution

$$R_A = \frac{W_a}{8l^3}(l-a)^3\left(3l+a\right) + \frac{W_l - W_a}{40l^3}(l-a)^3\left(4l+a\right)$$

$$O_A = \frac{-W_a}{48EIl}(l-a)^3\left(l+3a\right) - \frac{W_l - W_a}{240EIl}(l-a)^3\left(2l+3a\right)$$

$$R_B = \frac{W_a - W_l}{2}\left(l-a\right) - R_A$$

$$M_B = R_A l - \frac{W_a}{2}(l-a)^2 - \frac{W_l - W_a}{6}(l-a)^2$$

where

l = a + b

Comparison

Table 384: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Rotation at A, O_A (radians)	0.0020	0.0020	none
Vertical reaction at A, R_A (kN)	3.2886	3.2886	none
Vertical reaction at B, R_B (kN)	21.461	21.461	none
Moment at B, M_B (kN·m)	25.9605	25.9605	none

Note: In the STAAD model, two load cases are used. In case 1, the load is applied using the `MEMBER LOAD - TRAP` option. In case 2, the load is applied using a combination of `MEMBER LOAD - UNI` and `MEMBER LOAD - LIN` options. Both cases yield identical results.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Deflection and Reactions in a Beam.STD` is typically installed with the program.

```
STAAD PLANE REACTIONS AND DISPLACEMENTS OF A PINNED-FIXED BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE : ROARK'S FORMULAS FOR STRESS &amp; STRAIN
* WARREN C. YOUNG, 6TH EDITION, MCGRAW-HILL
*
* TABLE 3, CASE (2C), LOAD ON PARTIAL SPAN
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 7.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT CM KN
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 50 IZ 5000
UNIT METER NEWTON
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.3
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
3 FIXED
LOAD 1
MEMBER LOAD
2 TRAP GY -4 -7
LOAD 2
MEMBER LOAD
2 UNI GY -4
2 LIN GY 0 -3
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
UNIT METER NEWTON
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
```

```
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0020
           2     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0020
      2    1     0.0000   -0.4626    0.0000    0.0000    0.0000   -0.0006
           2     0.0000   -0.4626    0.0000    0.0000    0.0000   -0.0006
      3    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
   ************** END OF LATEST ANALYSIS RESULT **************
    40. UNIT METER NEWTON
    41. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      REACTIONS AND DISPLACEMENTS OF A PINNED-FIXED BEAM       -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT NEWT METE    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00   3288.60      0.00      0.00      0.00      0.00
           2      0.00   3288.60      0.00      0.00      0.00      0.00
      3    1      0.00  21461.40      0.00      0.00      0.00 -25960.51
           2      0.00  21461.40      0.00      0.00      0.00 -25960.51
```

V. End Moments in a Non Uniform Beam

To find end moments due to a uniform load on a beam with nonuniform sections, fixed at both ends.

Reference

Hand calculation using the following reference:

McCormack, J.C., *Structural Analysis*, Intext Educational Publishers, 3rd Edition, 1975.

Problem

Find the moment at the supports. Assume for input a unit width for the beam. Depths are as shown.

$E = 30\times(10)^6$ psi

$w = 4$ k/ft

$d_1 = 10$ in., $d_2 = 20$ in.

$L_1 = 12$ ft, $L_2 = 8$ ft

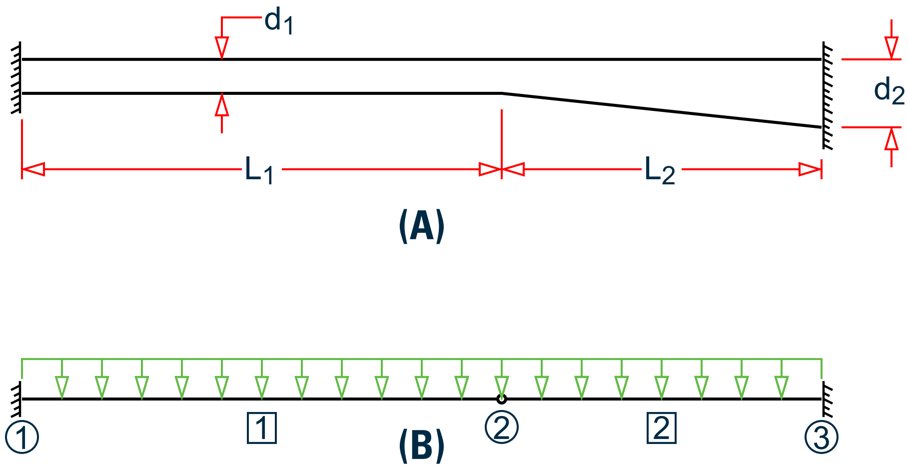


Figure 362: Beam A) problem sketch and B) mathematical model

Comparison

Table 385: Comparison of results

Result	Theory	STAAD.Pro	Difference
Moment at Node 1 (in·lb)	-98.2	-96.93	1.3%
Moment at Node 2 (in·lb)	-217.2	-220.42	1.5%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\01 Beams\End Moments in a Non Uniform Beam.STD` is typically installed with the program.

```
STAAD PLANE :A FIXED-FIXED BEAM OF UNIFORM AND TAPERED SECTIONS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MCCORMAC, J.C., STRUCTURAL ANALYSIS, INTEXT
*             EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975
*
INPUT WIDTH 72
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 144 0 0; 3 240 0 0;
MEMBER INCIDENCES
1 1 2; 2 3 2;
MEMBER PROPERTY AMERICAN
1 PRIS YD 10 ZD 1
2 TAPERED 20 1 10 1 1 1 1
PRINT MEMBER PROPERTIES
UNIT FEET KIP
```

```
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 3 FIXED
LOAD 1 UNIFORM LOAD OVER ENTIRE BEAM
UNIT FEET KIP
MEMBER LOAD
1 2 UNI GY -4
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00     33.83      0.00      0.00      0.00     96.93
      3    1      0.00     46.17      0.00      0.00      0.00   -220.42
```

V. Forces on a Propped Cantilever 1

To find the deflection and member forces due to an applied load on a propped cantilever beam with a compression only support.

Reference

Hand calculation using the following reference:

Manual of Steel Construction, Load and Resistance Factor Design, Second Edition, American Institute of Steel Construction, 1998, pp. 4-194, 4-197.

Problem

A propped cantilever beam with an propped-end support capable of resisting only a compressive force is analyzed for two concentrated loads at 0.6xL:

1. P = +0.5 lbs (up), and
2. P = -0.5 lbs (down)

$E = 10(10)^6$
width = 0.6 in
depth = 0.3 in
L = 4 ft

Note: A "dummy" member is used to represent a compression only support. The dimensions of this member are arbitrary; just that the member is specified as compression-only.

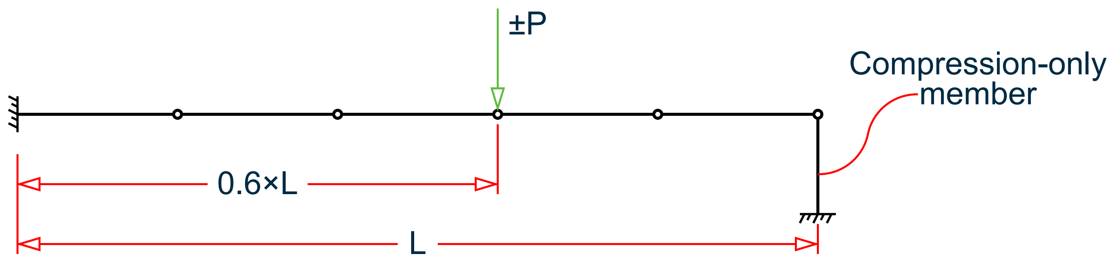


Figure 363: Propped cantilever member with a compression only support

Theoretical Solution

Load Case 1

General solution equations found on p.2-121 of the reference. In this case, the "propped" end carries no load as the support cannot carry tension. Thus, the member acts like a pure cantilever.

$$M(x < b) = -P(b - x) = -0.5lb(28.8 - x)$$

At the rigid support, $M = 14.4\ in \cdot lb$.

By inspection, fixed end shear is equal to load value = 0.50 lb.

$$\Delta\left(x < b\right) = \frac{Px^2}{6EI}\left(3x - b\right)$$

$$= \frac{0.5lb(28.8\text{in.})^2}{6\left(10,\ 000,\ 000psi\right)\left(0.00135\text{in.}^4\right)}\left[3\left(28.8\text{in.}\right) - 28.8\text{in.}\right] = 0.295\text{in.}$$

Load Case 2

General solution equations found on p.2-118 of the reference. In this case, the "propped" is capable of carrying load and thus the beam acts like a "propped" cantilever case.

$$R_1 = \frac{Pb^2\left(a + 2\ \ell\ \right)}{2\ell^3} = \frac{0.5lb(48\text{in.})^2}{2(48\text{in.})^3}\left[19.2\text{in} + 2\left(48\text{in}\right)\right] = 0.216lb.$$

$$\mathrm{R_2 = P - R_1 = 0.50lb - 0.216lb = 0.284lb.}$$

Moment at rigid support:

$$M(x < b) = R_1(\ \ell\ - x) - P(\ \ell\ - x - a)$$

$$= 0.216lb(48\text{in} - 0) - 0.5lb(48\text{in} - 0 - 19.2\text{in}) = -4.032\text{in}\cdot\text{lb}$$

Deflection at point of load:

$$\Delta\left(x < b\right) = \frac{-Pax^2}{12EI\ell^3}\left(3\ell^3\ - 3\ell^2\ x - 3a^2\ \ell\ + a^2x\right)$$

$$= \frac{-0.5\left(19.2\right)(28.8)^3}{12\left(10,\ 000,\ 000\right)\left(0.00135\right)(48)^3}\left[3(48)^3 - 3(48)^2(28.8) - 3(19.2)^2(48) + (19.2)^2(28.8)\right] = -0.040\text{ in.}$$

Comparison

Table 386: Comparison of results

Load Case	Result Type	Theory	STAAD.Pro	Difference
LC 1	Moment at fixed end (in·lb)	-14.4	-14.4	none
	Shear at fixed end (lb)	0.50	0.50	none
	Deflection at load (.in)	0.295	0.295	none
LC 2	Moment at fixed end (in·lb)	4.032	4.03	none
	Shear at fixed end (lb)	0.284	0.28	1.4%
	Deflection at load (.in)	-0.040	-.0401	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Forces on a Propped Cantilever 1.STD`is typically installed with the program.

```
STAAD PLANE :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*********************************************
* The end support is to be defined as a     *
* compression only support.  A dummy member,*
* #7, is set as compression only to model   *
* this.                                     *
*********************************************
SET NL 2
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 9.6 0 0; 3 19.2 0 0; 4 28.8 0 0; 5 38.4 0 0; 6 48 0 0;
7 48 -4 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7;
MEMBER PROPERTY AMERICAN
1 TO 5 PRIS YD 0.3 ZD 0.6
6 PRIS YD 40
MEMBER COMPRESSION
6
```

```
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.33
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
7 FIXED
*7 PINNED
LOAD 1 UPWARD
JOINT LOAD
4 FY 0.5
PERFORM ANALYSIS
PRINT MEMBER FORCES LIST 1 TO 5
PRINT JOINT DISPLACEMENTS LIST 1 TO 6
PRINT SUPPORT REACTION
CHANGE
LOAD 2 DOWNWARD
JOINT LOAD
4 FY -0.5
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 1 TO 5
PRINT JOINT DISPLACEMENTS LIST 1 TO 6
FINISH
```

STAAD Output

```
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00     -0.50     0.00      0.00      0.00     -14.40
                 2      0.00      0.50     0.00      0.00      0.00       9.60
      2    1     2      0.00     -0.50     0.00      0.00      0.00      -9.60
                 3      0.00      0.50     0.00      0.00      0.00       4.80
      3    1     3      0.00     -0.50     0.00      0.00      0.00      -4.80
                 4      0.00      0.50     0.00      0.00      0.00       0.00
      4    1     4      0.00     -0.00     0.00      0.00      0.00      -0.00
                 5      0.00      0.00     0.00      0.00      0.00       0.00
      5    1     5      0.00      0.00     0.00      0.00      0.00       0.00
                 6      0.00     -0.00     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    39. PRINT JOINT DISPLACEMENTS LIST 1 TO 6
  JOINT    DISPLACE LIST     1
      :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT      -- PAGE NO.
4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00000   0.04370   0.00000   0.00000   0.00000   0.00853
      3    1    0.00000   0.15293   0.00000   0.00000   0.00000   0.01365
      4    1    0.00000   0.29494   0.00000   0.00000   0.00000   0.01536
      5    1    0.00000   0.44239   0.00000   0.00000   0.00000   0.01536
      6    1    0.00000   0.58985   0.00000   0.00000   0.00000   0.01536
```

```
  ************** END OF LATEST ANALYSIS RESULT **************
    40. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT      -- PAGE NO.
5
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00     -0.50      0.00      0.00      0.00    -14.40
      7    1      0.00      0.00      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    41. CHANGE
    42. LOAD 2 DOWNWARD
    43. JOINT LOAD
    44. 4 FY -0.5
    45. PERFORM ANALYSIS
  **NOTE-Tension/Compression converged after  1 iterations, Case=    2
    46. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT      -- PAGE NO.
6
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    2      0.00      0.28      0.00      0.00      0.00      4.03
      7    2      0.00      0.22      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    47. PRINT MEMBER FORCES LIST 1 TO 5
  MEMBER   FORCES   LIST     1
      :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT      -- PAGE NO.
7
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    2     1      0.00      0.28     0.00      0.00      0.00       4.03
                 2      0.00     -0.28     0.00      0.00      0.00      -1.31
      2    2     2      0.00      0.28     0.00      0.00      0.00       1.31
                 3      0.00     -0.28     0.00      0.00      0.00       1.42
      3    2     3      0.00      0.28     0.00      0.00      0.00      -1.42
                 4      0.00     -0.28     0.00      0.00      0.00       4.15
      4    2     4      0.00     -0.22     0.00      0.00      0.00      -4.15
                 5      0.00      0.22     0.00      0.00      0.00       2.07
      5    2     5      0.00     -0.22     0.00      0.00      0.00      -2.07
                 6      0.00      0.22     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    48. PRINT JOINT DISPLACEMENTS LIST 1 TO 6
  JOINT    DISPLACE LIST     1
      :A PROPPED CANTILEVER WITH COMPRESSION ONLY SUPPORT      -- PAGE NO.
8
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    2    0.00000  -0.01066   0.00000   0.00000   0.00000  -0.00190
      3    2    0.00000  -0.03024   0.00000   0.00000   0.00000  -0.00186
      4    2    0.00000  -0.04012   0.00000   0.00000   0.00000   0.00012
```

```
     5     2     0.00000  -0.02714   0.00000   0.00000   0.00000   0.00233
     6     2     0.00000   0.00000   0.00000   0.00000   0.00000   0.00307
```

V. Forces on a Propped Cantilever 2

To find deflections, stress and support reactions due to a uniform load on a beam with one end fixed and the other end supported by a roller.

Reference

Hand calculation using the following reference:

Roark, R.J., and Young, W.C., *Formulas for Stress and Strain*, 5th Edition, Page 109, Problem 23.

Problem

A horizontal beam of length = 100 in, area = 4 in^2, height = 2 in, and moment of inertia = 1.3333 in^4 is simply supported at one end and fixed at the other end (a "propped" cantilever). The beam is subjected to an arbitrary uniform loading. Determine the deflection, δ, at $x = 42.15$ *in.*, the slope, θ, at end A, the maximum bending stress, σ_{bend}, in the beam, and the support reactions.

$E = 30\times(10)^6$ psi

Density = 0.2821 lbs/in^3

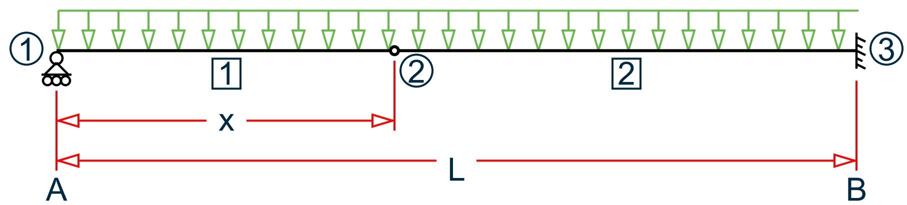


Figure 364: Propped cantilever beam model

Comparison

Table 387: Comparison of results

Result	Theory	STAAD.Pro	Difference
Reaction at Node 1 (lb)	42.31	42.31	none
Reaction at Node 3 (lb)	70.52	70.52	none
Moment at Node 3 (in·lb)	1,410.4	1410.4	none
Bending Stress, σ_{bend} at Node 2 (psi)	585.9	585.9	none
Rotation at Node 1 (rad.)	-.000588	-.00059	none

Result	Theory	STAAD.Pro	Difference
Deflection at Node 2 (in.)	-0.01528	-0.01528	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Forces on a Propped Cantilever 2.STD` is typically installed with the program.

```
STAAD PLANE :A FIXED-ROLLER BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: ROARK AND YOUNG, PAGE 109, NO. 23.
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 42.15 0 0; 3 100 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 4 IZ 1.3333 YD 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
DENSITY 0.282072
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
3 FIXED
1 PINNED
LOAD 1 SELF WEIGHT
SELFWEIGHT Y -1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PRINT MEMBER STRESSES ALL
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00059
      2    1    0.00000  -0.01528   0.00000   0.00000   0.00000  -0.00000
      3    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      :A FIXED-ROLLER BEAM                                        -- PAGE NO.
4
```

```
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      3    1      0.00     70.52      0.00      0.00      0.00  -1410.36
      1    1      0.00     42.31      0.00      0.00      0.00      0.00
      :A FIXED-ROLLER BEAM                                     -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00     42.31     0.00      0.00      0.00       0.00
                 2      0.00      5.25     0.00      0.00      0.00     781.13
      2    1     2      0.00     -5.25     0.00      0.00      0.00    -781.13
                 3      0.00     70.52     0.00      0.00      0.00   -1410.36
   ************** END OF LATEST ANALYSIS RESULT **************
    31. PRINT MEMBER STRESSES ALL
  MEMBER   STRESSES ALL
      :A FIXED-ROLLER BEAM                                     -- PAGE NO.
6
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0      0.0       0.0        0.0       0.0     15.9      0.0
              1.0      0.0 C     0.0      585.9     585.9      2.0      0.0
      2    1   .0      0.0       0.0      585.9     585.9      2.0      0.0
              1.0      0.0 C     0.0     1057.8    1057.8     26.4      0.0
```

V. Hanging Bar Axial Stress

Two vertical bars are supported by a rigid bar, which is pinned-supported on one end. Find stresses in vertical bars due to a load at the free end of the rigid bar.

Reference

Higdon, Ohlsen, Stiles, Weese and Riley, *Mechanics of Materials*, 3rd Edition, John Wiley & Sons, Page 135, Problem 3-37.

Problem

TBars A and B are connected by rigid links to a fixed support at the top and to a rigid bar at the bottom. Determine the axial stresses in bars A and B when the load P is 177.92888 kN applied as shown.

$P = 177.93$ kN

$A_A = 1{,}290.3\ mm^2$

$A_B = 1{,}612.9\ mm^2$

$E_A = 68.95$ GPa

$E_B = 206.84$ GPa

Assume the moment of inertia of member CD to be very large.

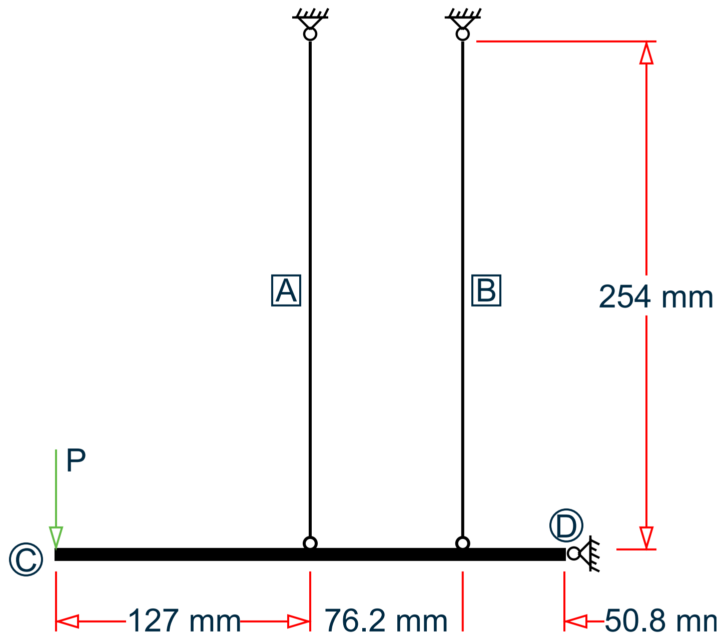


Figure 365: Rigid bar hanging from a pair of rods

Comparison

Table 388: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Stress in A (GPa)	0.17237	0.1723889	negligible
Stress in B (GPa)	0.20684	0.2068032	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Hanging Bar Axial Stress.STD` is typically installed with the program.

```
STAAD PLANE :TWO VERTICAL MEMBERS SUPPORT A RIGID BAR
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MECHANICS OF MATERIALS, HIGDON, OHLSEN, STILES, WEESE
*            AND RILEY, 3RD EDITION, JOHN WILEY &amp; SONS,
*            PAGE 135, PROBLEM 3-37
*
UNIT METER NEWTON
JOINT COORDINATES
1 0 0 0; 2 0.127 0 0; 3 0.2032 0 0; 4 0.254 0 0; 5 0.127 0.254 0;
6 0.2032 0.254 0;
```

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 0.032258 IZ 0.00208226
4 PRIS AX 0.00129032
5 PRIS AX 0.0016129
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.06843e+08
POISSON 0.290909
ISOTROPIC MATERIAL2
E 6.89477e+07
POISSON 0.329996
END DEFINE MATERIAL
UNIT METER NEWTON
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 TO 3 5
MATERIAL MATERIAL2 MEMB 4
MEMBER TRUSS
4 5
SUPPORTS
4 PINNED
5 6 FIXED
LOAD 1 VERTICAL JOINT LOAD
JOINT LOAD
1 FY -177929
PERFORM ANALYSIS
UNIT METER KN
PRINT MEMBER STRESSES LIST 4 5
FINISH
```

STAAD Output

```
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE KN  /SQ METE
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      4    1   .0 172388.9 T      0.0       0.0  172388.9      0.0      0.0
              1.0 172388.9 T      0.0       0.0  172388.9      0.0      0.0
      5    1   .0 206803.2 T      0.0       0.0  206803.2      0.0      0.0
              1.0 206803.2 T      0.0       0.0  206803.2      0.0      0.0
```

V. Stresses in a Cable due to Thermal Loading

A rigid bar is suspended by two copper wires and one steel wire. Find the stresses in the wires due to a rise in temperature.

Reference

Timoshenko, S., *Strength of Materials,* Part 1, D. Van Nostrand Co., 3rd edition, 1956, page 30, problem 9.

Problem

Assuming the horizontal member to be very rigid, determine the stresses in the copper and steel wires if the temperature rise is 10° F. Members 1 and 3 are copper and member 2 is steel.

$E_{steel} = 30\times(10)^6$ psi, $E_{copper} = 16\times(10)^6$ psi

$\alpha_{steel} = 70E\text{-}7$ in/in/°F, $\alpha_{copper} = 92E\text{-}7$ in/in/°F

$A_X = 0.1$ in^2

w = 400 lbf/in.

L = 20 in.

d= 5 in.

Tip: When modeling, assume a large moment of inertia for the horizontal rigid member and distribute of the concentrated load as uniform.

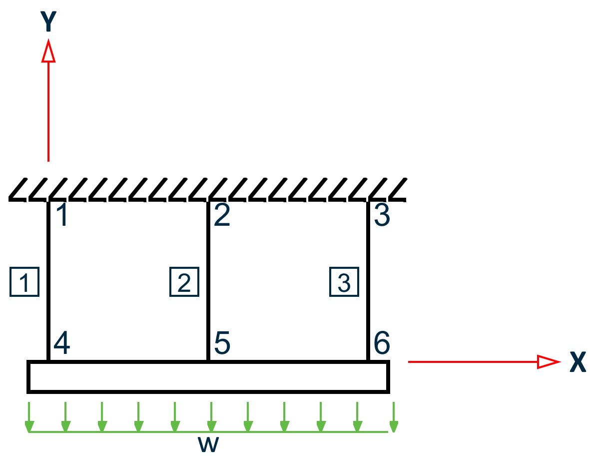


Figure 366: Model of a rigid wire suspended by wires

Comparison

Table 389: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
σ_{Steel} (psi)	19,695	19,698.3	negligible
σ_{Copper} (psi)	10,152	10,150.8	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Stresses in a Cable due to Thermal Loading.STD` is typically installed with the program.

```
STAAD PLANE :A RIGID BAR SUSPENDED BY TWO COPPER WIRES AND A STEEL WIRE
START JOB INFORMATION
ENGINEER DATE 09-Oct-17
END JOB INFORMATION
*
*  REFERENCE: 'STRENGTH OF MATERIALS', TIMOSHENKO (PART 1),
*              PAGE 30, PROB 9.
*              THE ANSWERS ARE 19700 PSI AND 10200 PSI.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 20 0; 2 5 20 0; 3 10 20 0; 4 0 0 0; 5 5 0 0; 6 10 0 0;
MEMBER INCIDENCES
1 1 4; 2 2 5; 3 3 6; 4 4 5; 5 5 6;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 0.1 IZ 0.0001
4 5 PRIS AX 1 IZ 100
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1.6e+007
POISSON 0.230769
ALPHA 9.2e-006
ISOTROPIC MATERIAL2
E 3e+007
POISSON 0.290909
ALPHA 7e-006
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 3
MATERIAL MATERIAL2 MEMB 2 4 5
SUPPORTS
1 2 3 Fixed
MEMBER RELEASE
1 TO 3 START MPZ 0.99
1 TO 3 END MZ
LOAD 1 VERT LOAD + TEMP LOAD
MEMBER LOAD
4 5 UNI Y -400
TEMPERATURE LOAD
1 TO 3 TEMP 10
PERFORM ANALYSIS
PRINT MEMBER STRESSES LIST 1 TO 3
FINISH
```

STAAD Output

```
  MEMBER STRESSES
  ---------------
```

```
 ALL UNITS ARE POUN/SQ INCH
MEMB    LD  SECT     AXIAL     BEND-Y      BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
     1     1   .0  10150.8 T      0.0         0.0   10150.8      0.0      0.0
               1.0  10150.8 T      0.0         0.0   10150.8      0.0      0.0
     2     1   .0  19698.3 T      0.0         0.0   19698.3      0.0      0.0
               1.0  19698.3 T      0.0         0.0   19698.3      0.0      0.0
     3     1   .0  10150.8 T      0.0         0.0   10150.8      0.0      0.0
```

V. Stresses in a Circular Beam

Find deflections and stress at the center of a locomotive axle.

Reference

Timoshenko, S., *Strength of Materials*, Part- 1, D. Van Nostrand Co., 3rd edition, 1956.

p. 94, problems 1, 2.

Problem

Determine the maximum stress in a locomotive axle (as shown in the figure) as well as the deflection at the middle of the axle.

Diameter = 10 in.

P = 26,000 lbf

E = $30\times(10)^6$ psi

L1 = 13.5 in., L2 = 59 in.

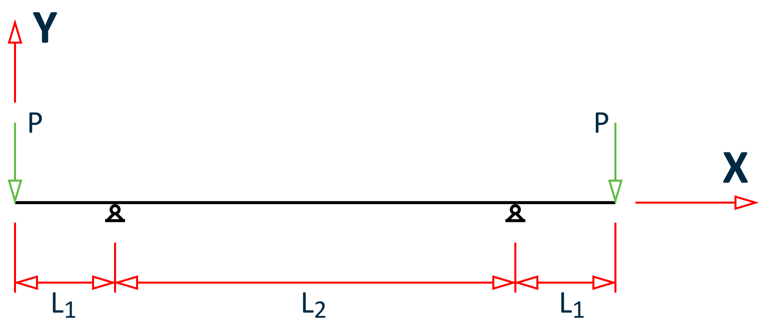


Figure 367: Locomotive axle model

Comparison

Table 390: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
σ (psi), Node 3	3,575.*	3,575.3	negligible
δ (in.), Node 3	0.01040	0.01037	negligible

* The value is recalculated.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Stresses in a Circular Beam.STD` is typically installed with the program.

```
STAAD PLANE : STRESSES IN A CIRCULAR BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  REFERENCE 'STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
*  PAGE 97 PROBLEM NO. 1 AND 2. ANSWERS ARE 3580 FOR MAX. STRESS
*  AND 0.104 INCH FOR MAX. DEFLECTION.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 13.5 0 0; 3 43 0 0; 4 72.5 0 0; 5 86 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 TO 4 TABLE ST PIPE OD 10 ID 0
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
2 4 PINNED
LOAD 1
JOINT LOAD
1 5 FY -26000
PERFORM ANALYSIS
PRINT MEMBER STRESSES
PRINT JOINT DISPLACEMENTS
FINISH
```

STAAD Output

```
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0      0.0         0.0        0.0        0.0    441.4      0.0
              1.0      0.0 C       0.0     3575.3     3575.3    441.4      0.0
      2    1   .0      0.0         0.0     3575.3     3575.3      0.0      0.0
              1.0      0.0 C       0.0     3575.3     3575.3      0.0      0.0
      3    1   .0      0.0         0.0     3575.3     3575.3      0.0      0.0
              1.0      0.0 C       0.0     3575.3     3575.3      0.0      0.0
      4    1   .0      0.0         0.0     3575.3     3575.3    441.4      0.0
```

```
              1.0      0.0 C      0.0       0.0       0.0    441.4      0.0
   ************** END OF LATEST ANALYSIS RESULT **************
    31. PRINT JOINT DISPLACEMENTS
  JOINT    DISPLACE
      : STRESSES IN A CIRCULAR BEAM                            -- PAGE NO.
4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000  -0.01138   0.00000   0.00000   0.00000   0.00086
      2    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00070
      3    1    0.00000   0.01037   0.00000   0.00000   0.00000   0.00000
      4    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00070
      5    1    0.00000  -0.01138   0.00000   0.00000   0.00000  -0.00086
```

V. Stresses in a Tapered Cantilever

To find the maximum deflection and principal stress due to a load on the free end of a cantilever beam with tapered section.

Reference

Hand calculation using the following reference:

Harris, C.O., *Introduction to Stress Analysis*, The Macmillan Co., 1959. Page 114, Problem 61.

Problem

Find the maximum deflection, δ, and the principal normal stress, σ, in the beam.

$E = 30\times(10)^6$ psi

Load at free end, P = 10 lb

Width at support, b = 3 in., Depth, d = 0.5 in.

Length, L = 20 in.

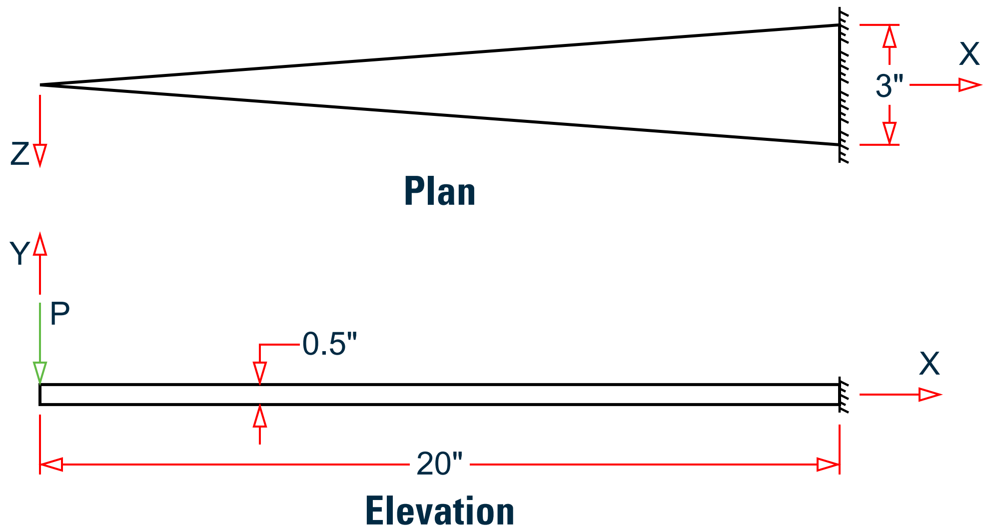


Figure 368: Beam with a tapering cross section

Note: The STAAD.Pro model is created using 10 beam segments, each with a tapered section. The tapered section is modeled using the tapered web and (effectively) no flanges, rotated 90°.

Comparison

Table 391: Comparison of results

Result	Theory	STAAD.Pro	Difference
Maximum deflection at free end (in.)	-0.04267	-0.04265	none
Principle stress (psi)	1,600	1,600	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\01 Beams\Stresses in a Tapered Cantilever.STD` is typically installed with the program.

```
STAAD PLANE :A CANTILEVER BEAM OF TAPERED SECTION
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: HARRIS, C.O., INTRODUCTION TO STRESS ANALYSIS,
*            THE MACMILLAN CO., NEW YORK, 1956
*
* USING A TAPERED BEAM ELEMENT
*
INPUT WIDTH 72
```

```
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 2 0 0; 3 4 0 0; 4 6 0 0; 5 8 0 0; 6 10 0 0; 7 12 0 0;
8 14 0 0; 9 16 0 0; 10 18 0 0; 11 20 0 0;
MEMBER INCIDENCES
1 11 10; 2 10 9; 3 9 8; 4 8 7; 5 7 6; 6 6 5; 7 5 4; 8 4 3; 9 3 2;
10 2 1;
MEMBER PROPERTY AMERICAN
1 TAPERED 3 0.5 2.7 0.5 0.01
2 TAPERED 2.7 0.5 2.4 0.5 0.01
3 TAPERED 2.4 0.5 2.1 0.5 0.01
4 TAPERED 2.1 0.5 1.8 0.5 0.01
5 TAPERED 1.8 0.5 1.5 0.5 0.01
6 TAPERED 1.5 0.5 1.2 0.5 0.01
7 TAPERED 1.2 0.5 0.9 0.5 0.01
8 TAPERED 0.9 0.5 0.6 0.5 0.01
9 TAPERED 0.6 0.5 0.3 0.5 0.01
10 TAPERED 0.3 0.5 0.03 0.5 0.01
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
BETA 90 ALL
MATERIAL MATERIAL1 ALL
SUPPORTS
11 FIXED
LOAD 1 POINT LOAD AT TIP
JOINT LOAD
1 FY -10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
SECTION 0.5 ALL
PRINT MEMBER SECTION FORCES ALL
PRINT JOINT DISPLACEMENTS LIST 1
PRINT MEMBER STRESSES ALL
FINISH
```

STAAD Output

```
   MEMBER PROPERTIES. UNIT - INCH
   -----------------
   MEMB   PROFILE              AX/          IZ/          IY/        IX/
                               AY           AZ           SZ         SY
     1      TAPERED            1.42         0.97         0.03        0.12
                               1.42         0.01         0.68        0.12
     2      TAPERED            1.28         0.70         0.03        0.11
                               1.28         0.01         0.54        0.11
     3      TAPERED            1.12         0.48         0.02        0.09
                               1.12         0.01         0.42        0.09
     4      TAPERED            0.97         0.31         0.02        0.08
                               0.97         0.01         0.32        0.08
     5      TAPERED            0.82         0.19         0.02        0.07
```

```
                              0.82         0.01         0.23         0.07
     6      TAPERED           0.68         0.11         0.01         0.06
                              0.68         0.01         0.15         0.06
     7      TAPERED           0.52         0.05         0.01         0.04
                              0.52         0.01         0.09         0.04
     8      TAPERED           0.38         0.02         0.01         0.03
                              0.38         0.01         0.05         0.03
     9      TAPERED           0.23         0.01         0.00         0.02
                              0.23         0.01         0.02         0.02
    10      TAPERED           0.08         0.00         0.00         0.01
                              0.08         0.01         0.00         0.01
   ************ END OF DATA FROM INTERNAL STORAGE ************
    45. PRINT MEMBER FORCES
  MEMBER   FORCES
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
4
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1    11      0.00      0.00   -10.00      0.00    200.00       0.00
                10      0.00      0.00    10.00      0.00   -180.00       0.00
      2    1    10      0.00      0.00   -10.00      0.00    180.00       0.00
                 9      0.00      0.00    10.00      0.00   -160.00       0.00
      3    1     9      0.00      0.00   -10.00      0.00    160.00       0.00
                 8      0.00      0.00    10.00      0.00   -140.00       0.00
      4    1     8      0.00      0.00   -10.00      0.00    140.00       0.00
                 7      0.00      0.00    10.00      0.00   -120.00       0.00
      5    1     7      0.00      0.00   -10.00      0.00    120.00       0.00
                 6      0.00      0.00    10.00      0.00   -100.00       0.00
      6    1     6      0.00      0.00   -10.00      0.00    100.00       0.00
                 5      0.00      0.00    10.00      0.00    -80.00       0.00
      7    1     5      0.00      0.00   -10.00      0.00     80.00       0.00
                 4      0.00      0.00    10.00      0.00    -60.00       0.00
      8    1     4      0.00      0.00   -10.00      0.00     60.00       0.00
                 3      0.00      0.00    10.00      0.00    -40.00       0.00
      9    1     3      0.00      0.00   -10.00      0.00     40.00       0.00
                 2      0.00      0.00    10.00      0.00    -20.00       0.00
     10    1     2      0.00      0.00   -10.00      0.00     20.00       0.00
                 1      0.00      0.00    10.00      0.00     -0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    46. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
5
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     11    1      0.00     10.00      0.00      0.00      0.00   -200.00
   ************** END OF LATEST ANALYSIS RESULT **************
    47. SECTION 0.5 ALL
    48. PRINT MEMBER SECTION FORCES ALL
  MEMBER   SECTION  FORCES   ALL
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
6
   MEMBER FORCES AT INTERMEDIATE SECTIONS
   -------------------------------------
   ALL UNITS ARE -- POUN INCH
```

```
 MEMB  LOAD  SEC       AXIAL   SHEAR-Y   SHEAR-Z       MOM-X       MOM-
Y       MOM-Z
     1    1  0.50       0.00      0.00    -10.00        0.00
190.00        0.00
     2    1  0.50       0.00      0.00    -10.00        0.00
170.00        0.00
     3    1  0.50       0.00      0.00    -10.00        0.00
150.00        0.00
     4    1  0.50       0.00      0.00    -10.00        0.00
130.00        0.00
     5    1  0.50       0.00      0.00    -10.00        0.00
110.00        0.00
     6    1  0.50       0.00      0.00    -10.00        0.00
90.00       0.00
     7    1  0.50       0.00      0.00    -10.00        0.00
70.00       0.00
     8    1  0.50       0.00      0.00    -10.00        0.00
50.00       0.00
     9    1  0.50       0.00      0.00    -10.00        0.00
30.00       0.00
    10    1  0.50       0.00      0.00    -10.00        0.00
10.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    49. PRINT JOINT DISPLACEMENTS LIST 1
  JOINT    DISPLACE LIST     1
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
7
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000  -0.04265   0.00000   0.00000   0.00000   0.00419
   ************** END OF LATEST ANALYSIS RESULT **************
    50. PRINT MEMBER STRESSES ALL
  MEMBER   STRESSES ALL
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
8
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      2    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      3    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      4    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      5    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      6    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
              0.50     0.0     1600.0       0.0    1600.0      0.0   1500.0
              1.0      0.0 C   1600.0       0.0    1600.0      0.0   1500.0
```

```
      7    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
                0.50    0.0     1600.0       0.0    1600.0      0.0   1500.0
                1.0     0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      8    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
                0.50    0.0     1600.0       0.0    1600.0      0.0   1500.0
                1.0     0.0 C   1600.0       0.0    1600.0      0.0   1500.0
      9    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
                0.50    0.0     1600.0       0.0    1600.0      0.0   1500.0
                1.0     0.0 C   1600.0       0.0    1600.0      0.0   1500.0
     10    1   .0      0.0     1600.0       0.0    1600.0      0.0   1500.0
      :A CANTILEVER BEAM OF TAPERED SECTION                    -- PAGE NO.
9
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
                0.50    0.0     1454.5       0.0    1454.5      0.0   1500.0
                1.0     0.0 C      0.0       0.0       0.0      0.0   1500.0
```

V. Tee Shaped Cantilever

To find the stress due to an applied moment at the free end of a cantilever beam with inverted tee section.

Reference

Hand calculation using the following reference:

Crandall, S.H., and Dahl, N.C., *An Introduction to the Mechanics of Solids*, McGraw-Hill, Inc., 1959, Page 294, Problem 7.2..

Problem

Find the maximum bending stress in the beam of length L with due to moment, M, and the free end.

$E = 30\times(10)^6$ psi

$L = 10$ in.

$M = 1{,}000{,}000$ in.·lb.

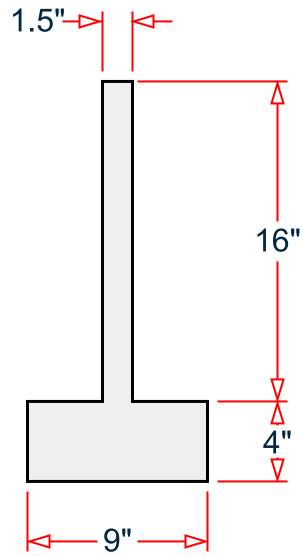


Figure 369: Cross section of cantilever beam

Comparison

Table 392: Comparison of results

Result	Theory	STAAD.Pro	Difference
Bending stress, σ (psi)	700	700	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Tee Shaped Cantilever.STD`is typically installed with the program.

```
STAAD PLANE :A CANTILEVERED BEAM OF INVERTED TEE SECTION
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: CRANDALL &amp; DAHL, AN INTRODUCTION TO THE MECHANICS
*            OF SOLIDS, PAGE294, EX. 7.2
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 PRIS YD 20 ZD 9 YB 16 ZB 1.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
```

```
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
BETA 180 ALL
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1 CONSTANT MOMENT
JOINT LOAD
2 MZ 100000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT MEMBER STRESSES ALL
FINISH
```

STAAD Output

```
  ALL UNITS ARE POUN/SQ INCH
 MEMB    LD  SECT     AXIAL     BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0       0.0        0.0      700.0      700.0      0.0      0.0
                1.0      0.0 C      0.0      700.0      700.0      0.0      0.0
```

V. Thermal Loading on a Beam

To find the support reactions due to a temperature loads applied on a fixed-fixed beam.

Reference

Hand calculation using the following reference:

Matrix Analysis of Framed Structures, 3rd edition, W.Weaver Jr. & J.M.Gere, Van Nostrand Reinhold, Table B2, Appendix B, p.500

Problem

The beam in the following geometric, load, and section properties: $L = 7.5\ m$, $\alpha = 11.7(10)^{-6} / °F, I_Z = 5{,}000\ cm^4$, $E = 200\ KN/mm^2$, $A = 50\ cm^2$.

Case 1: $T = 40°\ F$ (uniform across section)
Case 2: $T_1 = 0°\ F$ (below beam), $T_2 = 50°\ F$ (above beam)

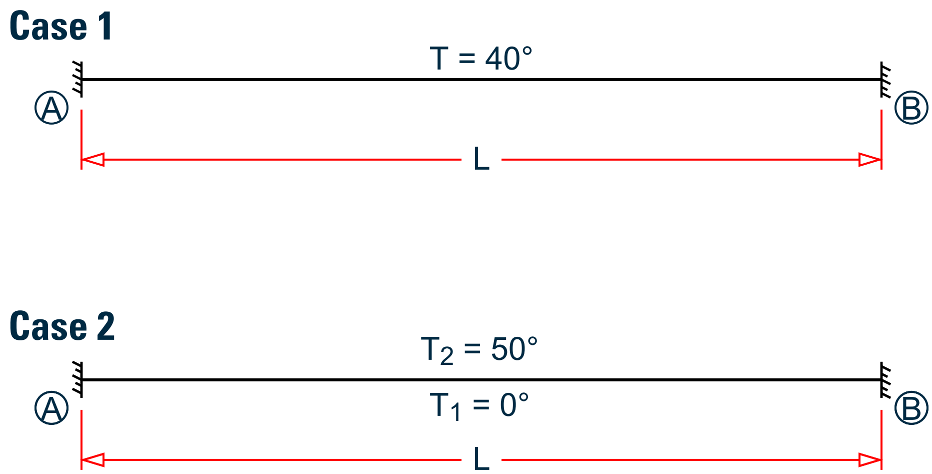


Figure 370: Fixed support beam with two different temperature load cases

Theoretical Solution

Horizontal reactions due to case 1 loads:

$$R_A = - R_B = EA\alpha\Delta T = [200(10)^6] \cdot [50(10)^{-4}] \cdot [11.7(10)^{-6}] \cdot (40) = 468 \text{ kN}$$

Moment reactions due to case 2 loads:

$$M_A = -M_B = \frac{\alpha EI\Delta T}{d} = \frac{\left[11.7(10)^{-6}\right] \cdot \left[200(10)^{-6}\right] \cdot \left[5,\ 000(10)^{-8}\right](50)}{0.3} = 19.5kN \cdot m$$

Comparison

Table 393: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Horizontal Reaction at Node A (kN)	468	468	none
Moment at Node A (kN·m)	19.5	19.5	none

Note: In the STAAD.Pro model, two load cases are used. In case 1, the uniform expansion is applied. In case 2, the temperature change between top and bottom flanges is applied.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Thermal Loading on a Beam.STD` is typically installed with the program.

```
STAAD PLANE TEMPERATURE LOAD ON A FIXED-FIXED BEAM
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
```

```
END JOB INFORMATION
*
* REFERENCE : MATRIX ANALYSIS OF FRAMED STRUCTURES
* GERE &amp; WEAVER, 3RD EDITION, VAN NOSTRAND REINHOLD
*
* TABLE B-2, APPENDIX B, PAGE 500
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 7.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT CM KN
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 50 IZ 5000 YD 30
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+08
POISSON 0.3
ALPHA 1.17e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
3 FIXED
LOAD 1
TEMPERATURE LOAD
1 2 TEMP 40
LOAD 2
TEMPERATURE LOAD
1 2 TEMP 0 50
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    468.00      0.00      0.00      0.00      0.00      0.00
           2      0.00      0.00      0.00      0.00      0.00    -19.50
      3    1   -468.00      0.00      0.00      0.00      0.00      0.00
           2      0.00      0.00      0.00      0.00      0.00     19.50
```

V. Torsion on a Stepped Cantilever

To find end rotation due to torques on a stepped cantilever shaft.

Reference

Hand calculation using the following reference:

Gere J. M., and Timoshenko, S. P., *Mechanics of Materials,* 2nd Edition, PWS Engineering, Page 171, Problem 3.3 - 1.

Problem

A stepped shaft is subjected to torques as shown in the figure. The material has shear modulus of elasticity G = 80 Gpa. Determine the angle of twist θ_x in degrees at the free end.

$T_1 = 3{,}000\ N{\cdot}mm$
$T_2 = 2{,}000 N{\cdot}mm$
$T_3 = 800 N{\cdot}mm$

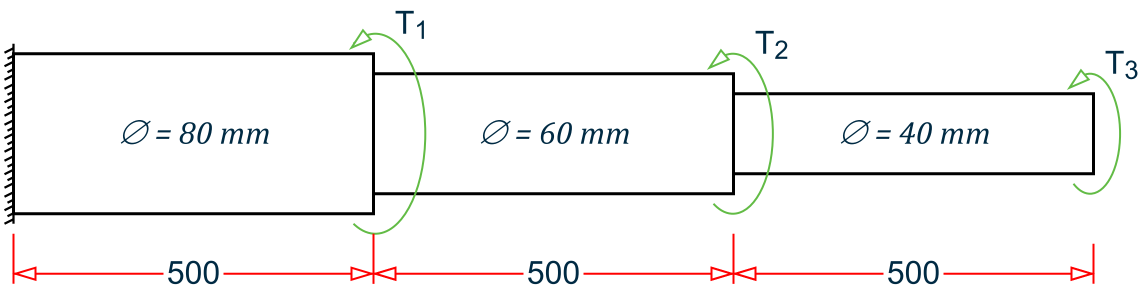


Figure 371: Cantilevered member subject to torsional loads

Theoretical Solution

Moment of Inertia:

$$I_{p1} = \frac{\pi(80mm/2)^4}{2} = 4.021(10)^6 mm^4$$

$$I_{p2} = \frac{\pi(60mm/2)^4}{2} = 1.272(10)^6 mm^4$$

$$I_{p2} = \frac{\pi(40mm/2)^4}{2} = 0.251(10)^6 mm^4$$

Angle of twist is given by:

$$\theta = \sum_i \frac{L_i T_i}{G I_p}$$

$$= \frac{5,\ 800\text{N-mm}(500mm)}{4.021(10)^6 mm^4 80GPa} + \frac{2,\ 800\text{N-mm}(500mm)}{1.272(10)^6 mm^4 80GPa} + \frac{,\ 800\text{N-mm}(500mm)}{0.251(10)^6 mm^4 80GPa}$$

$$= 0.0090 + 0.0138 + 0.0199 = 0.0427$$

which is equal to 2.446°.

Comparison

Table 394: Comparison of results

Result	Theory	STAAD.Pro	Difference
Angle of twist (rad.)	0.0427	0.0427	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Torsion on a Stepped Cantilever.STD`is typically installed with the program.

```
STAAD SPACE :A STEPPED CANTILEVER SHAFT
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MECHANICS OF MATERIALS, GERE AND TIMOSHENKO, 2ND EDITION
*            PROBLEM 3.3-1 PAGE 171
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0.5 0 0; 3 1 0 0; 4 1.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPE OD 80 ID 0
2 TABLE ST PIPE OD 60 ID 0
3 TABLE ST PIPE OD 40 ID 0
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 200
POISSON 0.25
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT MMS KN
LOAD 1 TORSIONAL MOMENT
JOINT LOAD
2 MX 3000
3 MX 2000
4 MX 800
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 4
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      4    1     0.0000    0.0000    0.0000    0.0427    0.0000    0.0000
```

V. Twist in a Tapered Tube

To find the twist at the free end of a hollow tapered shaft of uniform thickness.

Reference

Hand calculation using the following reference:

Mechanics of Materials, F.P.Beer & E.R.Johnston, 1981, McGraw-Hill Review Problem 3.120, Page 149.

Problem

The beam in the following figure which has the following geometric, load, and section properties: $L = 2\ m$, outside diameter at fixed end = 80 mm, outside diameter at free end = 40 mm, uniform wall thickness of 10 mm, , $I_Z = 5{,}000\ cm^4$, $E = 200\ KN/mm^2$, $T = 2.0\ KNm$, and Poisson's ratio = 0.3.

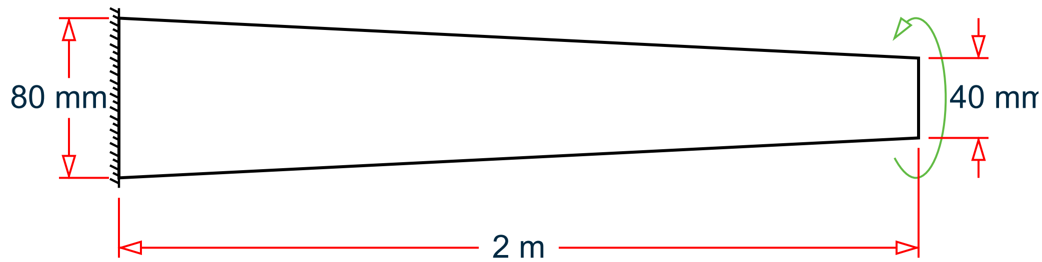


Figure 372: Cantilevered member with a tapered, hollow shaft cross section

Theoretical Solution

According to the reference, the twist at the free end is

$$\Phi_A = \left(\frac{TL}{4\pi G t}\right)\left(\frac{C_A + C_B}{C_A^2 C_B^2}\right)$$

where

C_A = centerline radius at free end = $40\ mm/2 - 10\ mm/2 = 15\ mm$

C_B = center line radius at fixed end = $80\ mm/2 - 10\ mm/2 = 35\ mm$

G = $\frac{E}{2(1+\nu)} = \frac{200kN/mm^2}{2(1+0.3)} = 76.9$

$$\Phi_A = \left(\frac{2,000kN \cdot m \times 2,000m}{4\pi \times 76.9kN/mm^2 \times 10mm}\right)\left[\frac{15mm + 35mm}{(15mm)^2(35mm)^2}\right] = 0.0751 \text{ rad.}$$

Comparison

Table 395: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Twist at free end (radians)	0.0751	0.0751	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\01 Beams\Twist in a Tapered Tube.STD`is typically installed with the program.

```
STAAD SPACE TORSION ON CONICAL SHAFT
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE : MECHANICS OF MATERIALS, F.P.BEER &amp; E.R.JOHNSTON
* 1981, MCGRAW-HILL
*
* REVIEW PROBLEM 3.120, PAGE 149. TWIST AT FREE END SHOULD BE
* ABOUT 0.0751 RADIAN
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 2 0;
MEMBER INCIDENCES
1 1 2;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
1 PRIS ROUND STA 80 END 40 THI 10
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+08
POISSON 0.3
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT METER KN
LOAD 1
JOINT LOAD
2 MY 2
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     2    1     0.0000    0.0000    0.0000    0.0000    0.0751    0.0000
```

V.02 Trusses

V. Axial Force in a 2D Plane Frame 1

To find member forces due to joint loads in a plane truss.

Reference

Norris C.H., Wilbur J. B., *Elementary Structural Analysis*, 2nd Edition, McGraw – Hill, Inc., Page 159, Problem 4.3. (Original data is in US Customary Units)

Problem

Compute the bar forces in the bars a, b, c, d, e of the truss due to the loads shown.

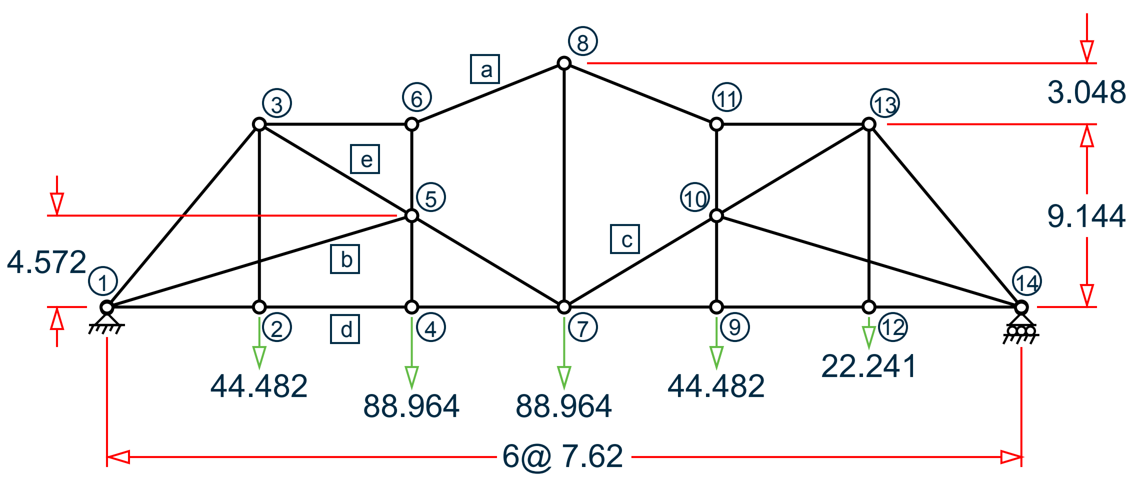


Figure 373: Plane truss

Comparison

Table 396: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
a	-202.13	-202.13	none
b	-185.76	-190.28	2.4% (negligible)
c	-22.42	-23.42	4.5%
d	266.23	269.52	1.2% (negligible)
e	119.61	117.08	2.1% (negligible)

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Force in a 2D Plane Frame 1.STD` is typically installed with the program.

```
STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: ELEMENTARY STRUCTURAL ANALYSIS, NORIS AND WILBUR,
*            2nd EDITION, McGRAW-HILL BOOK COMPANY, PAGE 159,
*            PROBLEM 4.3 (ORIGINAL DATA IN US CUSTOMARY UNITS)
*
UNIT METER NEWTON
JOINT COORDINATES
1 0 0 0; 2 7.62 0 0; 3 7.62 9.144 0; 4 15.24 0 0; 5 15.24 4.572 0;
6 15.24 9.144 0; 7 22.86 0 0; 8 22.86 12.192 0; 9 30.48 0 0;
10 30.48 4.572 0; 11 30.48 9.144 0; 12 38.1 0 0; 13 38.1 9.144 0;
14 45.72 0 0;
MEMBER INCIDENCES
1 1 2; 2 1 5; 3 1 3; 4 2 3; 5 2 4; 6 5 3; 7 3 6; 8 4 7; 9 7 5; 10 6 8;
11 7 9; 12 7 10; 13 11 8; 14 9 12; 15 10 13; 16 11 13; 17 12 14;
18 14 10; 19 12 13; 20 14 13; 21 4 5; 22 5 6; 23 7 8; 24 9 10; 25 10 11;
MEMBER PROPERTY AMERICAN
1 TO 25 PRIS AX 0.00129032
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.06843e+08
POISSON 0.290909
END DEFINE MATERIAL
UNIT METER NEWTON
CONSTANTS
MATERIAL MATERIAL1 ALL
MEMBER TRUSS
```

```
1 TO 25
SUPPORTS
1 PINNED
14 FIXED BUT FX
LOAD 1 VERTICAL JOINT LOADS
JOINT LOAD
2 9 FY -44482.2
4 7 FY -88964.4
12 FY -22241.1
PERFORM ANALYSIS
UNIT METER KN
PRINT MEMBER FORCES LIST 2 5 6 10 12
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = TRUSS
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      2     1     1    190.28     0.00     0.00      0.00      0.00       0.00
                  5   -190.28     0.00     0.00      0.00      0.00       0.00
      5     1     2   -269.52     0.00     0.00      0.00      0.00       0.00
                  4    269.52     0.00     0.00      0.00      0.00       0.00
      6     1     5   -117.08     0.00     0.00      0.00      0.00       0.00
                  3    117.08     0.00     0.00      0.00      0.00       0.00
     10     1     6    202.12     0.00     0.00      0.00      0.00       0.00
                  8   -202.12     0.00     0.00      0.00      0.00       0.00
     12     1     7     23.42     0.00     0.00      0.00      0.00       0.00
                 10    -23.42     0.00     0.00      0.00      0.00       0.00
```

V. Axial Force on a Cable

To find member force due to a member load in a plane articulate structure.

Reference

Kinney, J. S.,*Indeterminate Structural Analysis*, Addison - Wesley Publishing Co., 1957, p.275, Problem 6 - 19. (Original data is in US Customary Units)

Problem

Find the tensile stress in the cable. The cross - sectional area of the cable is 967.74 mm^2 with an E of 137.895 GPa. The timber beam 1-3 is 304.8 mm x 304.8 mm in section, with E = 11.03161 GPa. Each member of the steel cantilever truss has a cross - sectional area of 2,580.64 mm^2, and E of 206.8427 GPa.

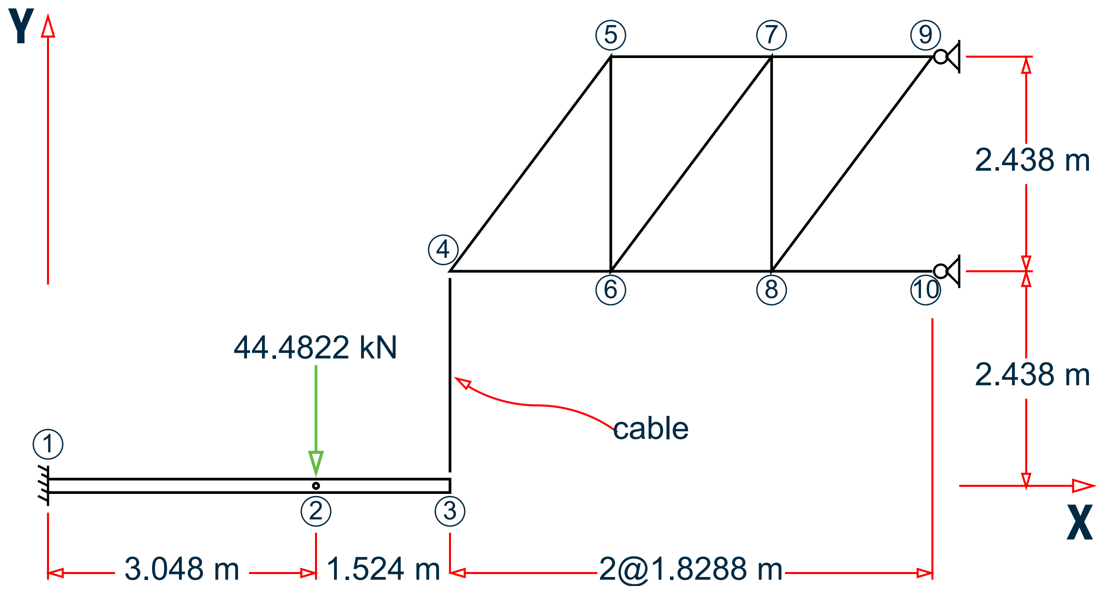


Figure 374: Plane articulate truss

Comparison

Table 397: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Cable, 3-4	22.7	22.58	0.6% (negligible)

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Force on a Cable.STD` is typically installed with the program.

```
STAAD PLANE :A PLANE ARTICULATE STRUCTURE
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: INDETERMINATE STRURAL ANALYSIS, KINNEY, 1957,
*            ADISON-WESLEY PUBLISHING COMPANY, PAGE 275, PROBLEM 6-19.
*
UNIT METER NEWTON
JOINT COORDINATES
1 0 0 0; 2 3.048 0 0; 3 4.572 0 0; 4 4.572 2.4384 0; 5 6.4008 4.8768 0;
6 6.4008 2.4384 0; 7 8.2296 4.8768 0; 8 8.2296 2.4384 0;
9 10.0586 4.8768 0; 10 10.0586 2.4384 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 6; 5 6 8; 6 8 10; 7 4 5; 8 6 5; 9 6 7; 10 8 7;
11 8 9; 12 5 7; 13 7 9;
MEMBER PROPERTY AMERICAN
```

```
1 2 PRIS YD 0.3048 ZD 0.3048
3 PRIS AX 0.00096774 IZ 4.16231e-07
4 TO 13 PRIS AX 0.00258064 IZ 4.16231e-07
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1.10316e+07
POISSON 0.22857
ISOTROPIC MATERIAL2
E 1.37895e+08
POISSON 0.261539
ISOTROPIC MATERIAL3
E 2.06843e+08
POISSON 0.290909
END DEFINE MATERIAL
UNIT METER NEWTON
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 2
MATERIAL MATERIAL2 MEMB 3
MATERIAL MATERIAL3 MEMB 4 TO 13
MEMBER RELEASE
3 TO 13 START MZ
SUPPORTS
1 9 10 FIXED
LOAD 1
JOINT LOAD
2 FY -44482.2
PERFORM ANALYSIS
UNIT METER KN
PRINT MEMBER FORCES LIST 3
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      3    1     3    -22.58     -0.00     0.00      0.00      0.00       0.00
                 4     22.58      0.00     0.00      0.00      0.00       0.00
```

V. Axial Forces in a Plane Frame 2

To find member forces due to a thermal load in a plane truss.

Reference

Gere J. M., and Timoshenko, S. P., *Mechanics of Materials*, 2nd Edition, PWS Engineering, p.21, Problem 2.6 - 23.

Problem

A symmetric, three-bar truss ABCD undergoes a temperature increase of 20°C in the two outer bars and 70°C in the middle bar. Calculate the forces F_1 and F_2 in the bars.

$E = 200$ GPa

$\alpha = 14(10)^{-6}$ / °C

$A = 900$ mm^2

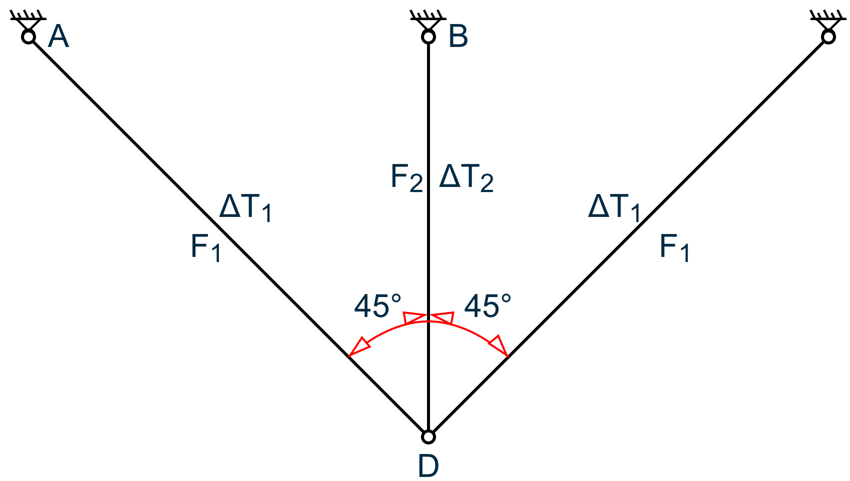


Figure 375: Plane truss subject to differential thermal loading

Comparison

Table 398: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
F_1	22,100	22,143	0.2% (negligible)
F_2	-31,300	-31,315	0.1% (negligible)

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Forces in a Plane Frame 2.STD` is typically installed with the program.

```
STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MECHANICS OF MATERIALS, GERE AND TIMOSHENKO, 2ND EDITION
*           PWS ENGINEERING, PAGE 121, PROBLEM 2.6-23
*
```

```
UNIT METER NEWTON
JOINT COORDINATES
1 -2.12132 2.12132 0; 2 0 2.12132 0; 3 2.12132 2.12132 0; 4 0 0 0;
MEMBER INCIDENCES
1 1 4; 2 2 4; 3 3 4;
UNIT MMS NEWTON
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 900
UNIT METER NEWTON
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.29542
ALPHA 1.4e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 3 PINNED
LOAD 1 TEMPERATURE LOAD
TEMPERATURE LOAD
1 3 TEMP 20 800
2 TEMP 70
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = TRUSS
   -----------------
   ALL UNITS ARE -- NEWT METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1 -22142.72      0.00     0.00      0.00      0.00       0.00
                 4  22142.72      0.00     0.00      0.00      0.00       0.00
      2    1     2  31314.54      0.00     0.00      0.00      0.00       0.00
                 4 -31314.54      0.00     0.00      0.00      0.00       0.00
      3    1     3 -22142.72      0.00     0.00      0.00      0.00       0.00
                 4  22142.72      0.00     0.00      0.00      0.00       0.00
```

V. Axial Forces on a 3D Space Model

To find support reactions and member forces due to a joint load in a space truss.

Reference

Beer F. P., and Johnston, E. R., *Vector Mechanics for Engineers - Statics*, 4th Edition, McGraw – Hill, Inc., p.216, Problem 6.20.

Problem

The space truss is supported by the six reactions shown. If a horizontal 2,700 N load is applied at A, determine the reactions and the force in each member.

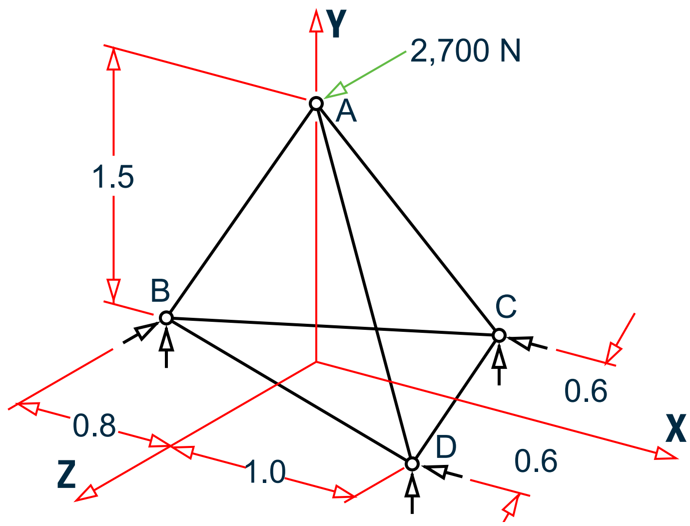


Figure 376: Space truss

Comparison

Table 399: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
B_y	0	0	none
B_z	2,700	2,700	none
C_x	1,800	1,800	none
C_y	3,375	3,375	none
D_x	1,800	1,800	none
D_y	3,375	3,375	none

Table 400: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
AB	0	0	none
AC	4,275	4,275	none

Result Type	Theory	STAAD.Pro	Difference
AD	4,275	4,275	none
BC	4,270	4,269	negligible
CD	0	0	none
BD	4,270	4,269	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Forces on a 3D Space Model.STD` is typically installed with the program.

```
STAAD TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: VECTOR MECHANICS FOR ENGINEERS - STATICS, BEER AND
*            JOHNSTON, 4th EDITION, McGRAW-HILL BOOK CO.,
*            PAGE 216, PROBLEM 6.20.
*
UNIT METER NEWTON
JOINT COORDINATES
1 1 0 0.6; 2 1 0 -0.6; 3 -0.8 0 0; 4 0 1.5 0;
MEMBER INCIDENCES
1 1 4; 2 2 4; 3 3 4; 4 1 2; 5 2 3; 6 3 1;
MEMBER PROPERTY AMERICAN
1 TO 6 PRIS AX 0.001
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+11
POISSON 0.29542
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
MEMBER TRUSS
1 TO 6
SUPPORTS
1 FIXED BUT FZ
2 FIXED BUT FZ
3 FIXED BUT FX
LOAD 1 HORIZONTAL LOAD
JOINT LOAD
4 FZ 2700
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
  SUPPORT  REACTION
      STAAD TRUSS                                                -- PAGE NO.
3
   SUPPORT REACTIONS -UNIT NEWT METE    STRUCTURE TYPE = TRUSS
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1   1800.00   3375.00      0.00      0.00      0.00      0.00
      2    1  -1800.00  -3375.00      0.00      0.00      0.00      0.00
      3    1      0.00      0.00  -2700.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    35. PRINT MEMBER FORCES
  MEMBER   FORCES
      STAAD TRUSS                                                -- PAGE NO.
4
   MEMBER END FORCES    STRUCTURE TYPE = TRUSS
   -----------------
   ALL UNITS ARE -- NEWT METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1   4275.00      0.00     0.00      0.00      0.00       0.00
                 4  -4275.00      0.00     0.00      0.00      0.00       0.00
      2    1     2  -4275.00      0.00     0.00      0.00      0.00       0.00
                 4   4275.00      0.00     0.00      0.00      0.00       0.00
      3    1     3     -0.00      0.00     0.00      0.00      0.00       0.00
                 4      0.00      0.00     0.00      0.00      0.00       0.00
      4    1     1      0.00      0.00     0.00      0.00      0.00       0.00
                 2      0.00      0.00     0.00      0.00      0.00       0.00
      5    1     2   4269.07      0.00     0.00      0.00      0.00       0.00
                 3  -4269.07      0.00     0.00      0.00      0.00       0.00
      6    1     3  -4269.07      0.00     0.00      0.00      0.00       0.00
                 1   4269.07      0.00     0.00      0.00      0.00       0.00
```

V. Axial Stress on a Truss Model

To find member stress due to a joint load in a space truss using static analysis.

Reference

Beer, F.P., and Johnston, Jr., E.R., *Vector Mechanics for Engineers, Statics and Dynamics*, McGraw - Hill, Inc., New York, 1962, p.47, Problem 2.70.

Problem

A 50 lb load is supported by three bars which are pinned to a ceiling as shown. Determine the stress, σ, in each bar.

Area of each bar = 1 in^2, E = 30 (10)^6 psi

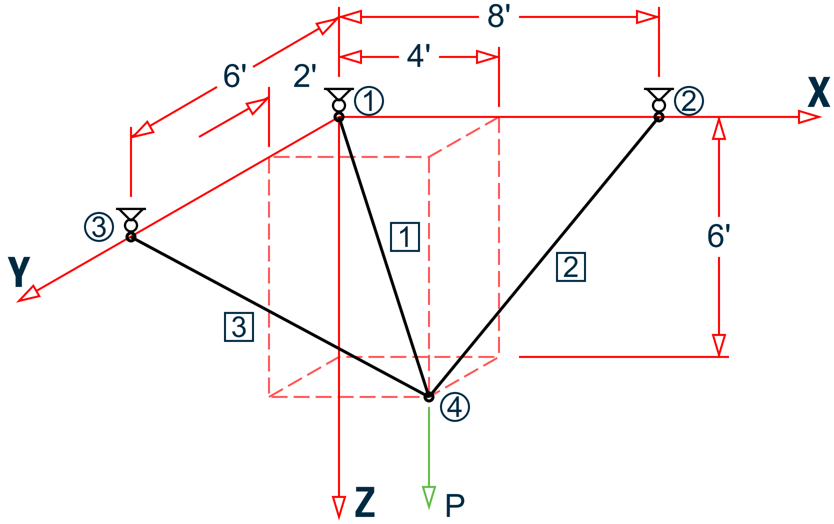


Figure 377: Space truss

P = 50 lbs

Comparison

Table 401: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
σ_{AD}	31.2	31.2	none
σ_{BD}	10.4	10.4	none
σ_{CD}	22.9	22.9	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Axial Stress on a Truss Model.STD` is typically installed with the program.

```
STAAD TRUSS :A SPACE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REF: "VECTOR MECHANICS FOR ENGINEERS, STATICS &amp; DYNAMICS",
*       MCGRAW HILL BOOK  CO., INC., NEW YORK, 1962
*       PROBLEM 2.70, PAGE 47.
*
UNIT FEET POUND
JOINT COORDINATES
```

```
1 0 0 0; 2 8 0 0; 3 0 6 0; 4 4 2 6;
MEMBER INCIDENCES
1 1 4; 2 2 4; 3 3 4;
UNIT INCHES POUND
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 3 PINNED
LOAD 1 WEIGHT
JOINT LOAD
4 FZ 50
PERFORM ANALYSIS
PRINT MEMBER STRESSES
FINISH
```

STAAD Output

```
  ALL UNITS ARE POUN/SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0     10.4 T      0.0       0.0      10.4      0.0      0.0
                1.0     10.4 T      0.0       0.0      10.4      0.0      0.0
      2    1   .0     31.2 T      0.0       0.0      31.2      0.0      0.0
                1.0     31.2 T      0.0       0.0      31.2      0.0      0.0
      3    1   .0     22.9 T      0.0       0.0      22.9      0.0      0.0
                1.0     22.9 T      0.0       0.0      22.9      0.0      0.0
   ************** END OF LATEST ANALYSIS RESULT **************
    32. FINISH
```

V. Deflections in a 2D Truss Model

Find the joint deflection due to joint loads in a plane truss.

Reference

McCormac, J. C., *Structural Analysis*, Intext Educational Publishers, 3rd edition, 1975, page 271, example 18 - 2.

Problem

Determine the vertical deflection at point 5 of plane truss structure shown in the figure.

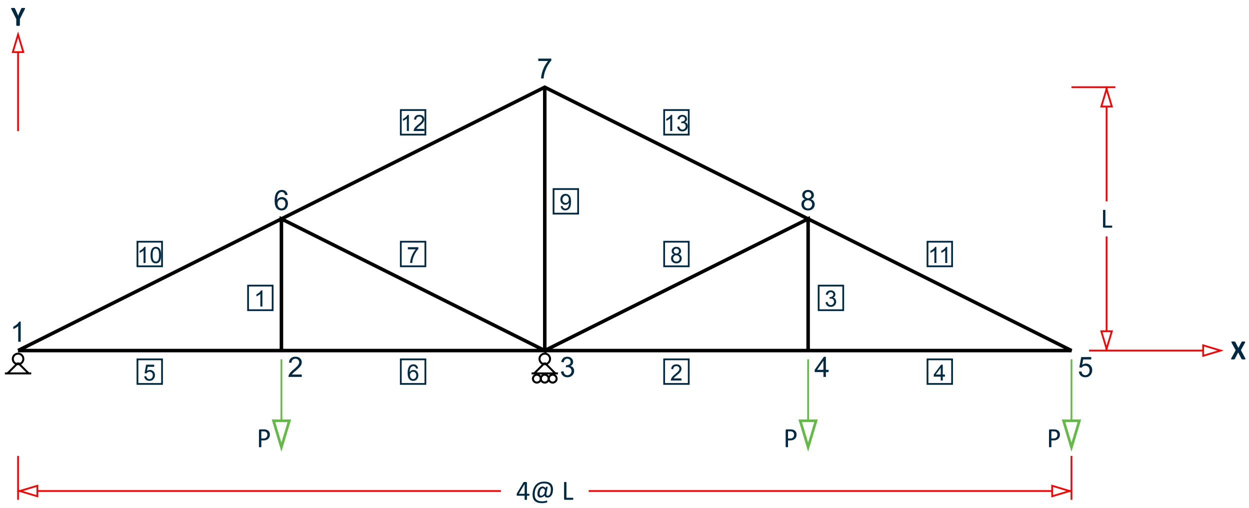


Figure 378: Plane truss model

P = 20 kip

L = 15 ft

Truss width = 4 spaces at 15 ft = 60 ft

Truss height = 15 ft

$A_{X\ 1\text{-}4} = 1\ in^2$, $A_{X\ 5\text{-}6} = 2\ in^2$, $A_{X\ 7\text{-}8} = 1.5\ in^2$,

$A_{X\ 9\text{-}11\ B} = 3\ in^{2\ P}$, $A_{X\ 12\text{-}13} = 4\ in^2$

E = 30E3 ksi

Comparison

Table 402: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
δ_5 (in.)	2.63	2.63033	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Deflections in a 2D Truss Model.STD` is typically installed with the program.

```
STAAD TRUSS : DEFLECTION IN TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
```

```
*  REFERENCE 'STRUCTURAL ANALYSIS' BY JACK McCORMACK, PAGE
*  271 EXAMPLE 18-2. ANSWER - Y-DISP AT JOINT 5 = 2.63 INCH
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 15 0 0; 3 30 0 0; 4 45 0 0; 5 60 0 0; 6 15 7.5 0; 7 30 15 0;
8 45 7.5 0;
MEMBER INCIDENCES
1 2 6; 2 3 4; 3 4 8; 4 4 5; 5 1 2; 6 2 3; 7 3 6; 8 3 8; 9 3 7; 10 1 6;
11 5 8; 12 6 7; 13 7 8;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 1
5 6 PRIS AX 2
7 8 PRIS AX 1.5
9 TO 11 PRIS AX 3
12 13 PRIS AX 4
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
3 FIXED BUT FX MZ
LOAD 1 VERTICAL LOAD
JOINT LOAD
2 4 5 FY -20
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = TRUSS
   ------------------
 JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1   -0.12000   0.18000   0.00000   0.00000   0.00000   0.00000
      3    1   -0.24000   0.00000   0.00000   0.00000   0.00000   0.00000
      4    1   -0.48000  -0.89516   0.00000   0.00000   0.00000   0.00000
      5    1   -0.72000  -2.63033   0.00000   0.00000   0.00000   0.00000
      6    1   -0.00820   0.24000   0.00000   0.00000   0.00000   0.00000
      7    1    0.29758  -0.12000   0.00000   0.00000   0.00000   0.00000
      8    1    0.06578  -0.83516   0.00000   0.00000   0.00000   0.00000
```

V. Reactions in a 2D Truss Model 1

To find support reactions due to joint loads in a plane truss.

Reference

McCormack, J.C. *Structural Analysis*, Intext Educational Publishers, 3rd Edition, 1975.

Problem

Find the vertical support reactions of the truss.

$$E = 30{,}000.0 \text{ ksi}$$

$$A = 100 \text{ in}^2$$

Loads as shown.

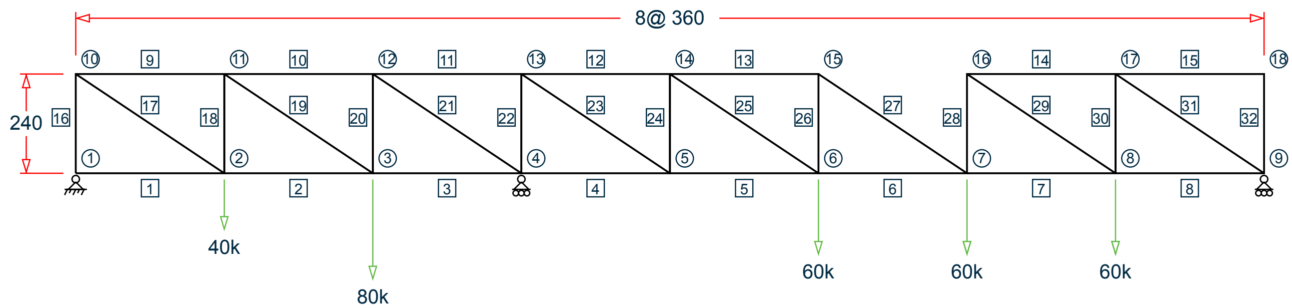


Figure 379: Plane truss

Comparison

Table 403: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
R_1 (kips)	-76.7	-76.67	none
R_4 (kips)	346.7	346.67	none
R_9 (kips)	30	30	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Reactions in a 2D Truss Model 1.STD` is typically installed with the program.

```
STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
```

```
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MCCORMAC, J.C.,"STRUCTURAL ANALYSIS", INTEXT
*             EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975.
*
INPUT WIDTH 72
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 60 0 0; 4 90 0 0; 5 120 0 0; 6 150 0 0; 7 180 0 0;
8 210 0 0; 9 240 0 0; 10 0 20 0; 11 30 20 0; 12 60 20 0; 13 90 20 0;
14 120 20 0; 15 150 20 0; 16 180 20 0; 17 210 20 0; 18 240 20 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 10 11;
10 11 12; 11 12 13; 12 13 14; 13 14 15; 14 16 17; 15 17 18; 16 1 10;
17 10 2; 18 2 11; 19 11 3; 20 3 12; 21 12 4; 22 4 13; 23 13 5; 24 5 14;
25 14 6; 26 6 15; 27 15 7; 28 7 16; 29 16 8; 30 8 17; 31 17 9; 32 9 18;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TO 32 PRIS AX 100
* MEMBER TRUSS
* 1 TO 32
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
4 9 FIXED BUT FX MZ
LOAD 1 POINT LOADS AT SPECIFIC JOINTS
JOINT LOAD
6 TO 8 FY -60
3 FY -80
2 FY -40
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = TRUSS
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00    -76.67      0.00      0.00      0.00      0.00
      4    1      0.00    346.67      0.00      0.00      0.00      0.00
      9    1      0.00     30.00      0.00      0.00      0.00      0.00
```

V. Reactions in a 2D Truss Model 2

To find support reactions due to joint loads in a plane truss.

Reference

McCormack, J.C., *Structural Analysis*, Intext Educational Publishers, 3rd Edition, 1975.

Problem

Find the vertical and horizontal reactions at the supports of the truss.

$E = 30{,}000.0$ ksi

$A = 100$ in^2

Loads as shown.

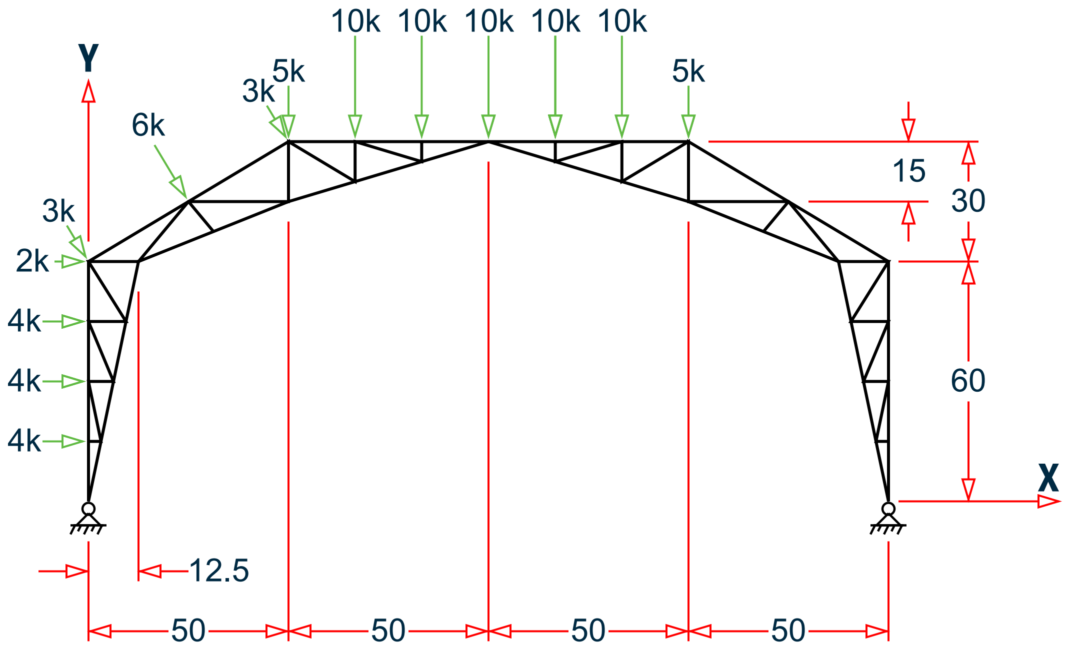


Figure 380: Plane truss

Comparison

Table 404: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Horizontal, R_1 (kips)	11.5	11.49	none
Vertical, R_1 (kips)	34.3	34.29	none
Horizontal, R_2 (kips)	-31.7	-31.67	none
Vertical, R_2 (kips)	36.0	36.0	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\02 Trusses\Reactions in a 2D Truss Model 2.STD` is typically installed with the program.

```
STAAD TRUSS :A PLANE TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
* REFERENCE: MCCORMAC, J.C.,"STRUCTURAL ANALYSIS", INTEXT
*            EDUCATIONAL PUBLISHERS, NEW YORK, 3RD EDITION, 1975.
*
INPUT WIDTH 72
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 12.5 60 0; 4 0 15 0; 5 0 30 0; 6 0 45 0;
7 9.375 45 0; 8 6.25 30 0; 9 3.125 15 0; 10 50 90 0; 11 100 90 0;
12 50 75 0; 13 25 75 0; 14 66.667 90 0; 15 83.333 90 0; 16 83.333 85 0;
17 66.667 80 0; 18 31.25 67.5 0; 19 200 0 0; 20 200 60 0; 21 187.5 60 0;
22 200 15 0; 23 200 30 0; 24 200 45 0; 25 190.625 45 0; 26 193.75 30 0;
27 196.875 15 0; 28 150 90 0; 29 150 75 0; 30 175 75 0; 31 133.333 90 0;
32 116.667 90 0; 33 116.667 85 0; 34 133.333 80 0; 35 168.75 67.5 0;
MEMBER INCIDENCES
1 1 4; 2 2 3; 3 3 7; 4 4 5; 5 5 6; 6 6 2; 7 7 8; 8 8 9; 9 9 1; 10 4 9;
11 9 5; 12 5 8; 13 8 6; 14 6 7; 15 7 2; 16 2 13; 17 10 14; 18 11 16;
19 12 18; 20 13 10; 21 14 15; 22 15 11; 23 16 17; 24 17 12; 25 18 3;
26 3 13; 27 13 18; 28 13 12; 29 12 10; 30 10 17; 31 17 14; 32 14 16;
33 16 15; 34 19 22; 35 20 21; 36 21 25; 37 22 23; 38 23 24; 39 24 20;
40 25 26; 41 26 27; 42 27 19; 43 22 27; 44 27 23; 45 23 26; 46 26 24;
47 24 25; 48 25 20; 49 20 30; 50 28 31; 51 11 33; 52 29 35; 53 30 28;
54 31 32; 55 32 11; 56 33 34; 57 34 29; 58 35 21; 59 21 30; 60 30 35;
61 30 29; 62 29 28; 63 28 34; 64 34 31; 65 31 33; 66 33 32;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TO 66 PRIS AX 100
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 19 PINNED
UNIT FEET KIP
LOAD 1 JOINT LOADS AT SPECIFIC NODES
JOINT LOAD
11 14 15 31 32 FY -10
10 28 FY -5
4 TO 6 FX 4
2 FX 2
2 10 FX 1.5435
13 FX 3.087
```

```
2 10 FY -2.5725
13 FY -5.145
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP   FEET     STRUCTURE TYPE = TRUSS
   -----------------
 JOINT   LOAD    FORCE-X    FORCE-Y    FORCE-Z     MOM-X      MOM-Y      MOM Z
      1     1      11.49      34.29       0.00      0.00       0.00       0.00
     19     1     -31.67      36.00       0.00      0.00       0.00       0.00
```

V. Reactions in a 2D Truss Model 3

Find the support reactions due to a joint load in a plane truss.

Reference

Timoshenko, S., *Strength of Materials*, Part 1, D. Van Nostrand Co., Inc., 3rd edition, 1956, p.346, problem 3.

Problem

Determine the horizontal reaction at support 4 of the system.

L = 50 in.

P = 10 kips

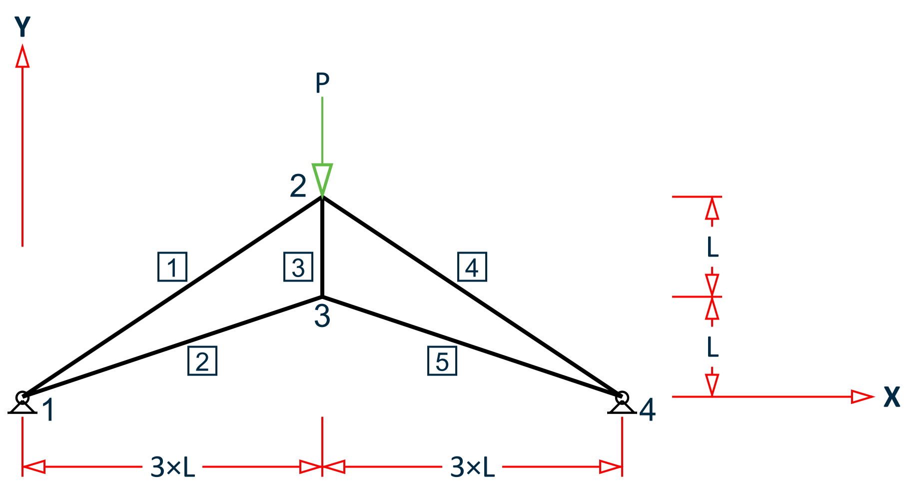


Figure 381: Plane truss

Comparison

Table 405: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
R_4 (kips)	8.77	8.77	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Reactions in a 2D Truss Model 3.STD` is typically installed with the program.

```
STAAD TRUSS : REACTION IN TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  REFERENCE `STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
*  PAGE 346 PROBLEM NO. 3. THE ANSWER IS REACTION = 0.877P.
*  THEREFORE IF P=10, REACTION = 8.77
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 150 100 0; 3 150 50 0; 4 300 0 0;
MEMBER INCIDENCES
1 1 2; 2 1 3; 3 2 3; 4 2 4; 5 3 4;
MEMBER PROPERTY AMERICAN
1 4 PRIS AX 5
2 5 PRIS AX 3
3 PRIS AX 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 PINNED
LOAD 1
JOINT LOAD
2 FY -10
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = TRUSS
   -----------------
 JOINT  LOAD    FORCE-X    FORCE-Y    FORCE-Z      MOM-X      MOM-Y      MOM Z
      1     1       8.77       5.00       0.00       0.00       0.00       0.00
      4     1      -8.77       5.00       0.00       0.00       0.00       0.00
```

V. Roof Truss Axial Forces

To find the forces in the members of a truss structure due to fabrication defect in the length of one bar which is 0.75 cm too short.

Reference

C.K. Wang, *Intermediate Structural Analysis*, International Student Edition, 1983, McGraw Hill, Section 10.13, p. 387

.

Problem

Structure to be solved as a truss. To achieve this in the STAAD model, instead of declaring them truss members, define them as frame members, and release MZ at both ends of all members. In the above figure, the area of cross section of the members is as follows.

1 TO 4 = 20 cm^2

5, 6 = 24 cm^2

7, 8 = 30 cm^2

9, 10 = 15 cm^2

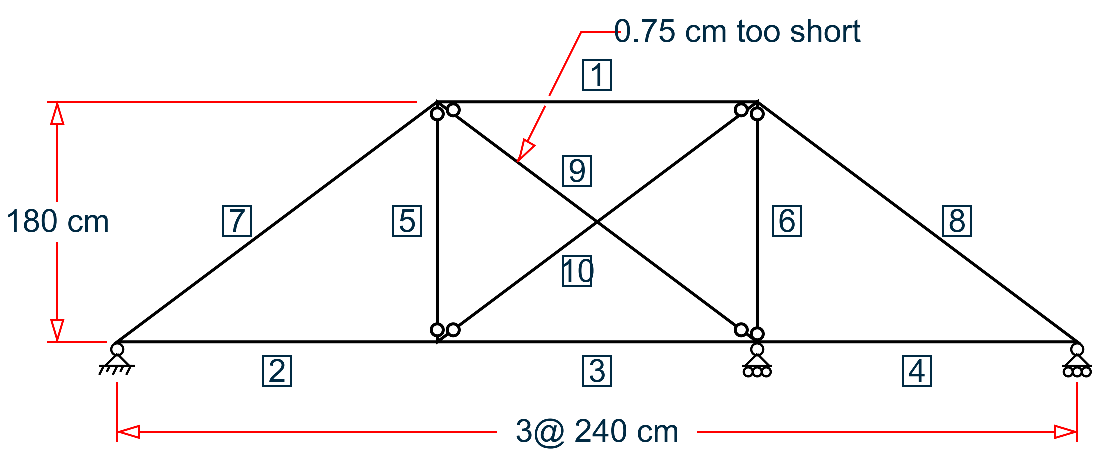


Figure 382: Frame subject to temperature change

Comparison

Table 406: Comparison of results

Member Number	Theory	STAAD.Pro	Difference
1	280.56 (C)	-280.57	none
2	50.86 (T)	50.86	none
3	127.99 (C)	-127.99	none
4	101.72 (T)	101.72	none
5	134.14 (C)	-134.14	none
6	57.85 (C)	-57.85	none
7	63.57 (C)	-63.57	none
8	127.14 (C)	-127.15	none
9	287.13 (T)	287.13	none
10	223.56 (T)	223.56	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\02 Trusses\Roof Truss Axial Forces.STD` is typically installed with the program.

```
STAAD SPACE EFFECT OF FABRICATION DEFECT - PRESTRAIN
START JOB INFORMATION
ENGINEER DATE 10-Oct-17
END JOB INFORMATION
*
* REFERENCE: INTERMEDIATE STRUCTURAL ANALYSIS, C.K.WANG
*     INTERNATIONAL STUDENT EDITION, 1983, MCGRAW HILL
*     SECTION 10.13, PAGE 387
*
UNIT METER KN
JOINT COORDINATES
1 2.4 1.8 0; 2 4.8 1.8 0; 3 0 0 0; 4 2.4 0 0; 5 4.8 0 0; 6 7.2 0 0;
MEMBER INCIDENCES
1 1 2; 2 3 4; 3 4 5; 4 5 6; 5 4 1; 6 5 2; 7 3 1; 8 2 6; 9 1 5; 10 4 2;
UNIT CM KN
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 20 IX 0.01 IY 0.01 IZ 0.01
5 6 PRIS AX 24 IX 0.01 IY 0.01 IZ 0.01
7 8 PRIS AX 30 IX 0.01 IY 0.01 IZ 0.01
9 10 PRIS AX 15 IX 0.01 IY 0.01 IZ 0.01
```

```
UNIT CM NEWTON
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2e+07
POISSON 0.3
END DEFINE MATERIAL
UNIT CM KN
CONSTANTS
MATERIAL MATERIAL1 ALL
* ALL MOMENTS RELEASED TO SIMULATE TRUSS ACTION
MEMBER RELEASE
5 6 9 10 START MZ
5 6 9 10 END MZ
*1 TO 10 START MZ
*1 TO 10 END MZ
SUPPORTS
3 PINNED
5 6 FIXED BUT FX MZ
LOAD 1
TEMPERATURE LOAD
9 STRAIN -0.75
PERFORM ANALYSIS
UNIT CM NEWTON
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- NEWT CM       (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1 280565.47     -0.03     0.00      0.00      0.00      -4.05
                 2-280565.47      0.03     0.00      0.00      0.00      -2.71
      2    1     3 -50858.06      0.02     0.00      0.00      0.00      -0.38
                 4  50858.06     -0.02     0.00      0.00      0.00       4.02
      3    1     4 127991.16     -0.03     0.00      0.00      0.00      -4.02
                 5-127991.16      0.03     0.00      0.00      0.00      -2.67
      4    1     5-101716.24      0.01     0.00      0.00      0.00       2.67
                 6 101716.24     -0.01     0.00      0.00      0.00      -0.23
      5    1     4 134136.97      0.00     0.00      0.00      0.00       0.00
                 1-134136.97      0.00     0.00      0.00      0.00       0.00
      6    1     5  57849.78      0.00     0.00      0.00      0.00       0.00
                 2 -57849.78      0.00     0.00      0.00      0.00       0.00
      7    1     3  63572.59      0.01     0.00      0.00      0.00       0.38
                 1 -63572.59     -0.01     0.00      0.00      0.00       4.05
      8    1     2 127145.30      0.01     0.00      0.00      0.00       2.71
                 6-127145.30     -0.01     0.00      0.00      0.00       0.23
      9    1     1-287134.25      0.00     0.00      0.00      0.00       0.00
                 5 287134.25      0.00     0.00      0.00      0.00       0.00
     10    1     4-223561.55      0.00     0.00      0.00      0.00       0.00
                 2 223561.55      0.00     0.00      0.00      0.00       0.00
```

V. Stress in a 2D Truss Model

Find the joint deflection and member stress due to a joint load in a plane truss.

Reference

Timoshenko, S., *Strength of Materials*, Part 1, D. Van Nostrand Co., Inc., 3rd edition, 1956, page 10, problem 2.

Problem

Determine the vertical deflection at point A and the member stresses

$A_X = 0.5\ in^2$

$E = 30E6$ psi

$P = 5000$ lbf

$L = 180$ in.

angle = 30°

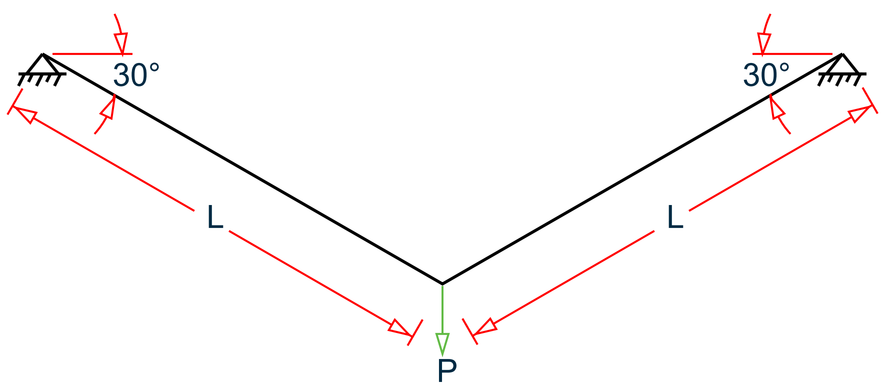


Figure 383: Model of two member truss

Comparison

Table 407: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
σ_A (psi)	10,000.	10,000.	none
δ_A (in)	0.12	0.12	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\02 Trusses\Stress in a 2D Truss Model.STD` is typically installed with the program.

```
STAAD TRUSS : STRESS IN TRUSS
START JOB INFORMATION
ENGINEER DATE 18-Sep-18
END JOB INFORMATION
*
*  THIS EXAMPLE IS TAKEN FROM 'STRENGTH OF MATERIALS'
*  (PART 1) BY TIMOSHENKO, PAGE 10 PROB 2.
*  THE ANSWER IN THE BOOK , DEFLECTION = 0.12 INCH
*  AND STRESS =10000 PSI
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 155.885 -90 0; 3 311.769 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 0.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.15
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 3 PINNED
LOAD 1 VERT LOAD
JOINT LOAD
2 FY -5000
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
PRINT MEMBER STRESSES
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = TRUSS
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00000  -0.12000   0.00000   0.00000   0.00000   0.00000
      3    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
    32. PRINT MEMBER STRESSES
  MEMBER   STRESSES
      : STRESS IN TRUSS                                        -- PAGE NO.
4
  MEMBER STRESSES
```

```
 ---------------
 ALL UNITS ARE POUN/SQ INCH
MEMB    LD   SECT     AXIAL     BEND-Y      BEND-Z    COMBINED   SHEAR-Y   SHEAR-Z
     1     1    .0  10000.0 T      0.0         0.0     10000.0       0.0       0.0
                1.0  10000.0 T      0.0         0.0     10000.0       0.0       0.0
     2     1    .0  10000.0 T      0.0         0.0     10000.0       0.0       0.0
                1.0  10000.0 T      0.0         0.0     10000.0       0.0       0.0
```

V.03 Frames

V. 1x2 Plane Frame Lateral Load

Find the maximum moment due to lateral joint loads in a 1x2 bay plane frame.

Reference

McCormac, J. C., *Structural Analysis*, Intext Educational Publishers, 3rd edition, 1975, page 388, example 22 - 7.

Problem

Determine the maximum moment in the frame.

L = 20 ft

H3 = 20 kip, H5 = 10 kip

E and I same for all members.

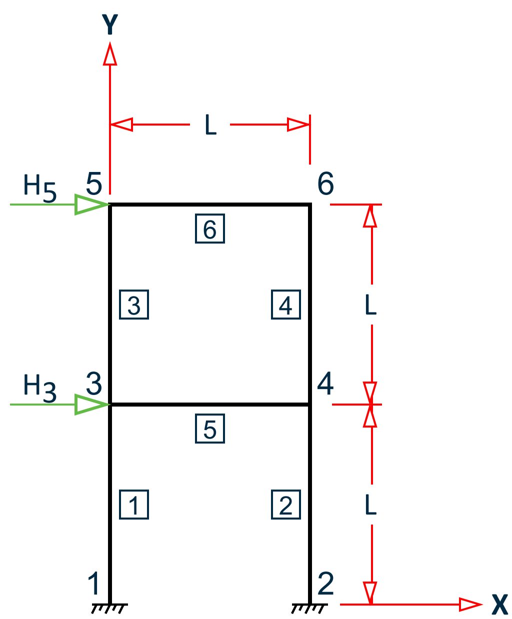


Figure 384: Two story frame model

Comparison

Table 408: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
M_{Max} (ft·kip)	176.40	178.01	0.9%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\1x2 Plane Frame Lateral Load.STD` is typically installed with the program.

```
STAAD PLANE  : 1X2 PLANE FRAME LATERAL LOAD
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* MULTIPLE LEVEL PLANE FRAME WITH HORIZONTAL LOAD.
* REFERENCE 'STRUCTURAL ANALYSIS' BY JACK McCORMACK,
* PAGE 388, PROB 22-7. ANSWER - MAX MOM IN MEMB 1 = 176.4 K-F
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0; 3 0 20 0; 4 20 20 0; 5 0 40 0; 6 20 40 0;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 6; 5 3 4; 6 5 6;
MEMBER PROPERTY AMERICAN
1 TO 6 PRIS AX 0.2 IZ 0.1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 FIXED
LOAD 1 HORIZONTAL LOAD
JOINT LOAD
3 FX 20
5 FX 10
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
```

```
 ALL UNITS ARE -- KIP  FEET      (LOCAL )
MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z    TORSION     MOM-Y      MOM-Z
    1     1     1    -22.26    15.06     0.00       0.00       0.00     178.01
                3     22.26   -15.06     0.00       0.00       0.00     123.16
    2     1     2     22.26    14.94     0.00       0.00       0.00     176.73
                4    -22.26   -14.94     0.00       0.00       0.00     122.10
    3     1     3     -6.51     4.97     0.00       0.00       0.00      34.49
                5      6.51    -4.97     0.00       0.00       0.00      64.93
    4     1     4      6.51     5.03     0.00       0.00       0.00      35.34
                6     -6.51    -5.03     0.00       0.00       0.00      65.24
    5     1     3      9.91   -15.75     0.00       0.00       0.00    -157.65
                4     -9.91    15.75     0.00       0.00       0.00    -157.44
    6     1     5      5.03    -6.51     0.00       0.00       0.00     -64.93
                6     -5.03     6.51     0.00       0.00       0.00     -65.24
```

V. 2 Bay Frame Moments and Shear

To find the member forces in a 1x2 bay plane frame with members of rectangular section.

Reference

Manual of Steel Construction – Allowable Stress Design, AISC, 9th Edition, 1989.

Problem

The frame supports a uniformly distributed load and concentrated loads. Calculate the bending moment and shear force at the mid point of the beam of the first bay.

$$E = 30{,}000 \text{ ksi}$$

Columns are 12"×24", beams are 12"×30"

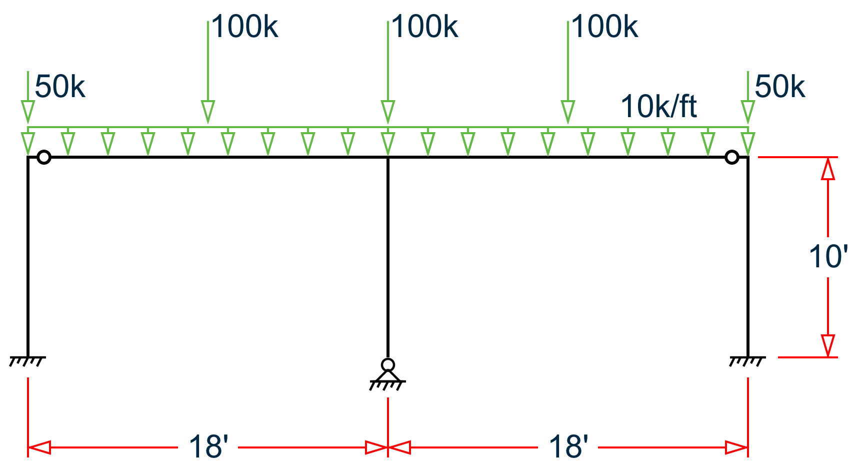


Figure 385: 2 bay frame

Comparison

Table 409: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Load Case 1	Moment , M (in·kips)	3,375	3,360.15	0.4%
	Shear, V (kips)	68.75	68.75	none
Load Case 2	Moment, M (in·kips)	2,430	2,425.12	negligible
	Shear, V (kips)	22.50	22.68	0.8%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2 Bay Frame Moments and Shear.STD` is typically installed with the program.

```
STAAD PLANE : 2 BAY FRAME MOMENTS AND SHEAR
START JOB INFORMATION
ENGINEER DATE 09-Oct-17
END JOB INFORMATION
*
*  REFERENCE: "MANUAL OF STEEL CONSTRUCTION-ALLOWABLE STRESS DESIGN",
*             AISC, CHICAGO, ILLINOIS, 1989.
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 18 0 0; 3 36 0 0; 4 0 10 0; 5 18 10 0; 6 36 10 0;
MEMBER INCIDENCES
1 1 4; 2 2 5; 3 3 6; 4 4 5; 5 5 6;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1e+07 IZ 13824
4 5 PRIS AX 1e+07 IZ 27000
MEMBER RELEASE
5 END MZ
4 START MZ
SUPPORTS
1 3 FIXED
2 PINNED
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
UNIT FEET KIP
LOAD 1 VERTICAL POINT LOADS
JOINT LOAD
```

```
4 6 FY -50
5 FY -100
MEMBER LOAD
4 CON GY -100 9
5 CON GY -100 9
LOAD 2 VERTICAL UNIFORM LOADS
MEMBER LOAD
4 UNI GY -10
5 UNI GY -10
PERFORM ANALYSIS
UNIT INCHES KIP
SECTION 0.501 MEMB 4
PRINT MEMBER SECTION FORCES LIST 4
FINISH
```

STAAD Output

```
   MEMBER FORCES AT INTERMEDIATE SECTIONS
   --------------------------------------
   ALL UNITS ARE -- KIP  INCH
 MEMB  LOAD  SEC       AXIAL   SHEAR-Y   SHEAR-Z       MOM-X       MOM-
Y       MOM-Z
     4    1  0.50      -0.00    -68.75      0.00        0.00        0.00
-3360.15
          2  0.50      -0.00    -22.68      0.00        0.00        0.00
-2425.12
```

V. 2D Portal Reactions 1

To find section properties, member forces and support reactions for a 1x1 bay plane frame with members of rectangular section.

Reference

Timoshenko, S., *Strength of Materials, Part I, Elementary Theory and Problems*, 2nd Edition, Van Nostrand Company, 1940, Pages 188-191,

Problem

The frame supports a concentrated load at middle of the horizontal member. Verify the internally calculated section properties, support reactions and bending moments at the ends of the horizontal member. Columns are square 2" x 2"; beams are rectangular with b = 2" and h = 4".

$P = 1{,}000$ lb

$h = 100$ in.

$l = 120$ in.

$E = 30\times(10)^6$ psi

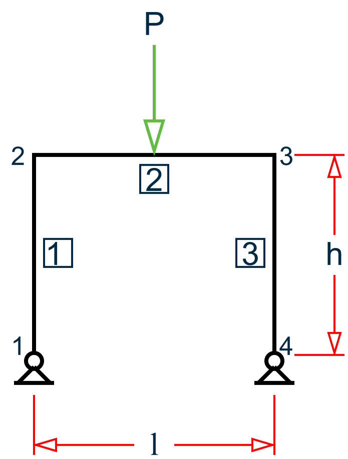


Figure 386: Symmetric portal frame

Calculations

From the reference:

$$M = \frac{Pl}{8}\frac{1}{1 + \frac{2}{3}\frac{h}{l}\frac{I_b}{I_c}} = \frac{1,\ 000(120)}{8}\frac{1}{1 + \frac{2}{3}\frac{100}{120}\frac{10.67}{1.333}} = 2,\ 754\text{in-lb}$$

Comparison

Table 410: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Column Cross Section	A_x (in.^{2})	4.0	4.0	none
	I_x (in.^{4})	2.25	2.25	none
	I_y (in.^{4})	1.333	1.33	none
	I_z (in.^{4})	1.333	1.33	none
Beam Cross Section	A_x (in.^{2})	8.0	8.0	none
	I_x (in.^{4})	7.324	7.32	none
	I_y (in.^{4})	2.667	2.67	none
	I_z (in.^{4})	10.667	10.67	none
R_y (lb)		500	500	none
R_x (lb)		27.55	27.54	none

Result Type		Theory	STAAD.Pro	Difference
M(in·lb)		2,754.97	2,754.35	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions 1.STD` is typically installed with the program.

```
STAAD PLANE :A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE: TIMOSHENKO, S., STRENGTH OF MATERIALS, PART 1,
*            2ND EDITION, D. VAN NOSTRAND CO., 1940,
*            PAGES 188 THRU 191.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0 100 0; 3 60 100 0; 4 120 100 0; 5 120 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 4 PRIS YD 2 ZD 2
2 3 PRIS YD 4 ZD 2
PRINT MEMBER PROPERTIES
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 5 PINNED
LOAD 1 MID SPAN
JOINT LOAD
3 FY 1000
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES LIST 2 3
FINISH
```

STAAD Output

```
      :A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION            -- PAGE NO.
2
   MEMBER PROPERTIES. UNIT - INCH
   -----------------
   MEMB   PROFILE               AX/          IZ/          IY/        IX/
                                AY           AZ           SZ         SY
```

```
     1  PRISMATIC              4.00        1.33         1.33       2.25
                               3.40        3.40         1.33       1.33
     2  PRISMATIC              8.00       10.67         2.67       7.32
                               6.80        6.80         5.33       2.67
     3  PRISMATIC              8.00       10.67         2.67       7.32
                               6.80        6.80         5.33       2.67
     4  PRISMATIC              4.00        1.33         1.33       2.25
                               3.40        3.40         1.33       1.33
  ************ END OF DATA FROM INTERNAL STORAGE ************
    19. DEFINE MATERIAL START
    20. ISOTROPIC MATERIAL1
    21. E 3E+07
    22. POISSON 0.290909
    23. END DEFINE MATERIAL
    24. CONSTANTS
    25. MATERIAL MATERIAL1 ALL
    26. SUPPORTS
    27. 1 5 PINNED
    28. LOAD 1 MID SPAN
    29. JOINT LOAD
    30. 3 FY 1000
    31. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          5  NUMBER OF MEMBERS       4
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      11
      :A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION            -- PAGE NO.
3
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    32. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      :A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION            -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -27.54   -500.00      0.00      0.00      0.00      0.00
      5    1     27.54   -500.00      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    33. PRINT MEMBER FORCES LIST 2 3
  MEMBER   FORCES   LIST     2
      :A 1X1 BAY PLANE FRAME OF RECTANGULAR SECTION            -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      2    1     2    -27.54   -500.00     0.00      0.00      0.00   -2754.35
                 3     27.54    500.00     0.00      0.00      0.00  -27245.65
      3    1     3    -27.54    500.00     0.00      0.00      0.00   27245.65
                 4     27.54   -500.00     0.00      0.00      0.00    2754.35
   ************** END OF LATEST ANALYSIS RESULT **************
```

V. 2D Portal Reactions 2

Find the maximum moment due to a uniform load on the horizontal member in a 1x1 bay plane frame.

Reference

McCormac, J. C., *Structural Analysis*, Intext Educational Publishers, 3rd edition, 1975, page 383, example 22 - 5.

Problem

Determine the maximum moment in the frame.

E and I same for all members.

L = 20 ft

w = 2 kips/ft

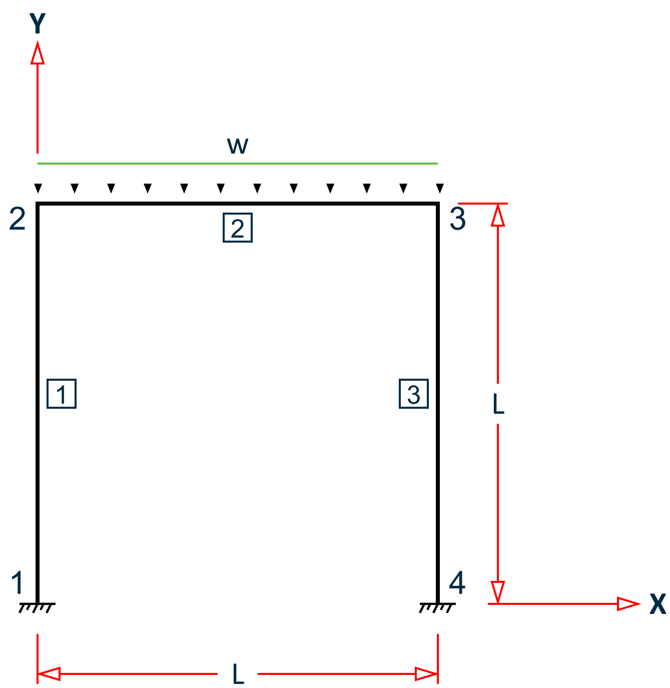


Figure 387: 1x1 bay plane frame

Comparison

Table 411: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
M_{Max} (kip·ft)	44.40	44.44	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions 2.STD` is typically installed with the program.

```
STAAD PLANE : 2D PORTAL REACTIONS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
*  REFERENCE 'STRUCTURAL ANALYSIS' BY JACK C. McCORMACK,
*  PAGE 383 EXAMPLE 22-5, PLANE FRAME WITH NO SIDESWAY
*  ANSWER - MAX BENDING = 44.4 FT-KIP
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 20 20 0; 4 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 1 IZ 0.05
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.132e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 FIXED
LOAD 1
MEMBER LOAD
2 UNI Y -2
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  FEET     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1     1     1     20.00     -3.33     0.00      0.00      0.00     -22.21
                  2    -20.00      3.33     0.00      0.00      0.00     -44.44
      2     1     2      3.33     20.00     0.00      0.00      0.00      44.44
                  3     -3.33     20.00     0.00      0.00      0.00     -44.44
      3     1     3     20.00      3.33     0.00      0.00      0.00      44.44
                  4    -20.00     -3.33     0.00      0.00      0.00      22.21
```

V. 2D Portal Reactions Sidesway 1

To find the displacements at the nodes of a frame due to movements of supports.

Reference

C.K. Wang, *Intermediate Structural Analysis*, International Student Edition, 1983, McGraw Hill, Section 2.11, p47.

Problem

Calculate the deflections at node B and support D.

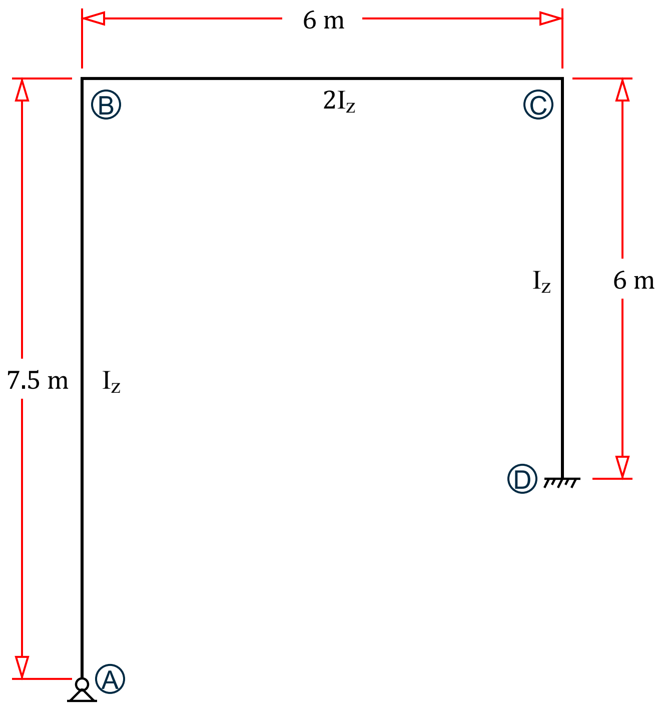


Figure 388: Frame subject to imposed displacements

Load Cases:

1. Vertical displacement of 1 cm at Node A
2. Vertical displacement of 1 cm at Node B

Comparison

Table 412: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Load Case 1	Horizontal displacement at node B (cm)	1.25	-1.25	none
	Rotation at node B	0.001667	0.0017	none
	Horizontal displacement at node D (cm)	0.4167	-0.4167	none
Load Case 2	Horizontal displacement at node B (cm)	1.25	1.25	none
	Rotation at node B	0.001667	0.0017	none
	Horizontal displacement at node D (cm)	0.4167	0.4167	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions Sidesway 1.STD` is typically installed with the program.

```
STAAD PLANE DEFLECTIONS DUE TO MOVEMENT OF SUPPORTS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE: INTERMEDIATE STRUCTURAL ANALYSIS, C.K.WANG
*     INTERNATIONAL STUDENT EDITION, 1983, MCGRAW HILL
*     SECTION 2.11, PAGE 47
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 7.5 0; 3 6 7.5 0; 4 6 2.5 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT CM KN
MEMBER PROPERTY AMERICAN
1 3 PRIS AX 2400 IZ 720000
2 PRIS AX 4800 IZ 1.44e+06
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
```

```
E 2171.84
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
4 FIXED BUT FX MZ
LOAD 1
SUPPORT DISPLACEMENT LOAD
1 FY -1
LOAD 2
SUPPORT DISPLACEMENT LOAD
4 FY -1
PERFORM ANALYSIS
LOAD LIST 1
PRINT JOINT DISPLACEMENTS
LOAD LIST 2
PRINT JOINT DISPLACEMENTS
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000   -1.0000    0.0000    0.0000    0.0000    0.0017
      2    1    -1.2500   -1.0000    0.0000    0.0000    0.0000    0.0017
      3    1    -1.2500    0.0000    0.0000    0.0000    0.0000    0.0017
      4    1    -0.4167    0.0000    0.0000    0.0000    0.0000    0.0017
   ************** END OF LATEST ANALYSIS RESULT **************
    38. LOAD LIST 2
    39. PRINT JOINT DISPLACEMENTS
  JOINT    DISPLACE
      DEFLECTIONS DUE TO MOVEMENT OF SUPPORTS                  -- PAGE NO.
4
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    2     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0017
      2    2     1.2500    0.0000    0.0000    0.0000    0.0000   -0.0017
      3    2     1.2500   -1.0000    0.0000    0.0000    0.0000   -0.0017
      4    2     0.4167   -1.0000    0.0000    0.0000    0.0000   -0.0017
```

V. 2D Portal Reactions Sidesway 2

Find the maximum moment due to a concentrated load on the horizontal member in a 1x1 bay plane frame.

Reference

McCormac, J. C., *Structural Analysis*, Intext Educational Publishers, 3rd edition, 1975, page 385, problem 22 - 6.

Problem

Determine the maximum moment in the structure.

$P = 30$ kip

$L_1 = 20$ ft, $L_2 = 30$ ft

E and I same for all members

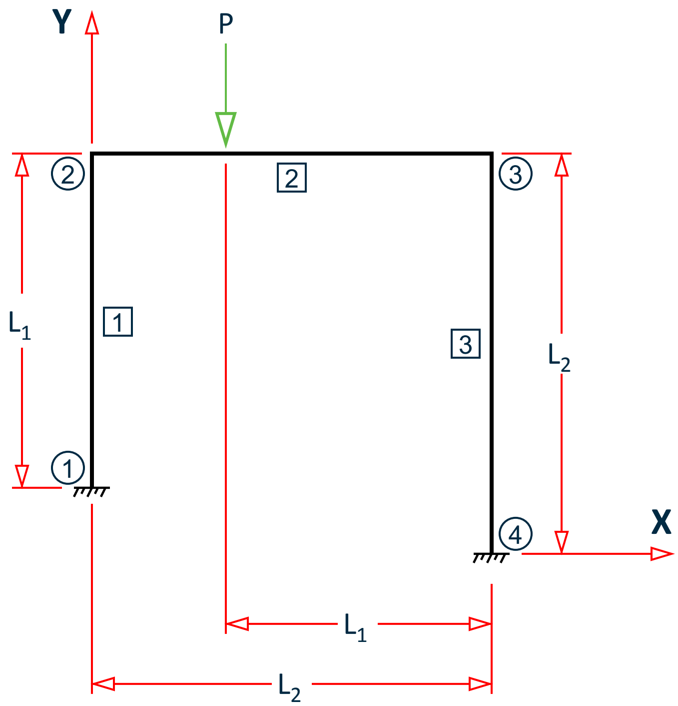


Figure 389: Unequal leg bay model

Comparison

Table 413: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
M_{Max} (ft·kip)	69.40	69.44	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\2D Portal Reactions Sidesway 2.STD` is typically installed with the program.

```
STAAD PLANE : 2D PORTAL REACTIONS SIDEWAYS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
```

```
END JOB INFORMATION
*
* PLANE FRAME WITH SIDESWAY. REFERENCE 'STRUCTURAL ANALYSIS'
* BY JACK McCORMACK. PAGE 385 PROB 22-6.
* ANSWER - MAX BENDING IN MEMB 1 = 69.4 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 10 0; 2 0 30 0; 3 30 30 0; 4 30 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST W12X26
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 FIXED
LOAD 1 VERTICAL LOAD
MEMBER LOAD
2 CON Y -30 10
PERFORM ANALYSIS
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  FEET     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y       MOM-Z
      1     1     1     20.09     -3.74     0.00      0.00      0.00       -5.33
                  2    -20.09      3.74     0.00      0.00      0.00      -69.44
      2     1     2      3.74     20.09     0.00      0.00      0.00       69.44
                  3     -3.74      9.91     0.00      0.00      0.00      -66.66
      3     1     3      9.91      3.74     0.00      0.00      0.00       66.66
                  4     -9.91     -3.74     0.00      0.00      0.00       45.50
```

V. 3D Frame Max Forces

Find the maximum axial force and moment due to load and moment applied at a joint in a Space frame.

Reference

Weaver Jr., W., *Computer Programs for Structural Analysis*, page 146, problem 8.

Problem

Determine the maximum axial force and moment in the space structure.

F = 2 kip, P = 1 kip, M = 120 in·kip

L = 120 in.

E = 30E3 ksi,

AX = 11 in^2

IX = 83 in^4

IY = 56 in^4

IZ = 56 in^4

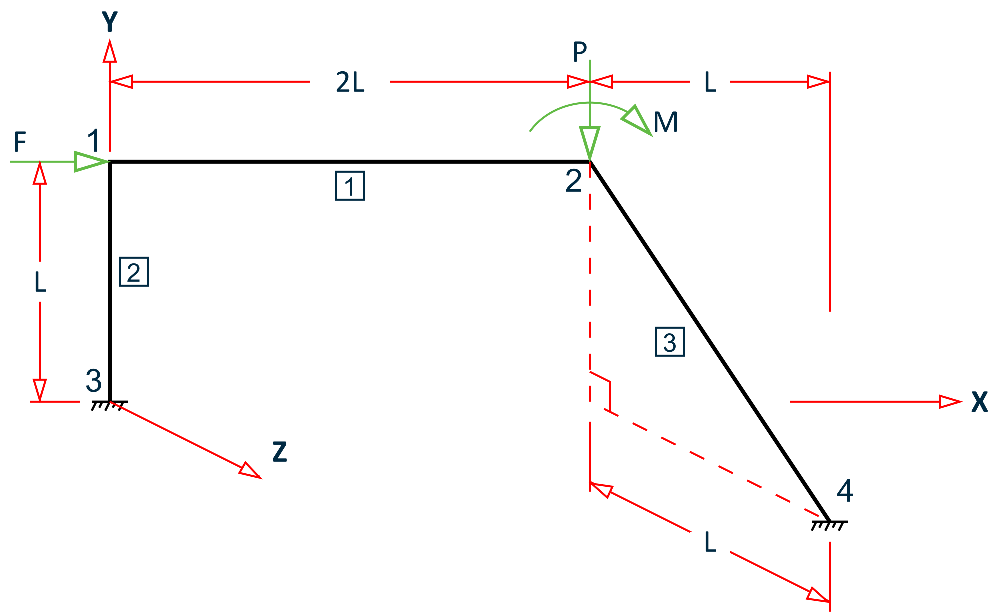


Figure 390: Space frame model

Comparison

Table 414: Comparison of results

Result Type	Theory	Advanced Analysis	Difference
F_{Max} (kips)	1.47	1.47	
$M_{Y,\ Max}$ (in·kip)	84.04	84.04	
$M_{Z,\ Max}$ (in·kip)	95.319	95.32	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\3D Frame Max Forces.STD` is typically installed with the program.

```
STAAD SPACE : 3D FRAME MAX FORCES
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE 'COMPUTER PROGRAMS FOR STRUCTURAL ANALYSIS'
* BY WILLIAM WEAVER JR. PAGE 146 STRUCTURE NO. 8.
* ANSWER - MAX AXIAL FORCE= 1.47 (MEMB 3)
* MAX BEND-Y= 84.04, MAX BEND-Z= 95.319 (BOTH MEMB 3)
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 120 0; 2 240 120 0; 3 0 0 0; 4 360 0 120;
MEMBER INCIDENCES
1 1 2; 2 3 1; 3 2 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 11 IX 83 IY 56 IZ 56
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 30000
POISSON 0.25
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
3 4 FIXED
LOAD 1 JOINT LOAD
JOINT LOAD
1 FX 2
2 FY -1
2 MZ -120
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.22267   0.00016  -0.17182  -0.00255   0.00217  -0.00213
      2    1    0.22202  -0.48119  -0.70161  -0.00802   0.00101  -0.00435
      3    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      4    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      : 3D FRAME MAX FORCES                                    -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
   -----------------
```

```
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      3    1     -1.10     -0.43      0.22     48.78    -17.97     96.12
      4    1     -0.90      1.43     -0.22    123.08     47.25    -11.72
      : 3D FRAME MAX FORCES                                    -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.90     -0.43     0.22     22.71    -17.97     -36.37
                 2     -0.90      0.43    -0.22    -22.71    -34.18     -67.36
      2    1     3     -0.43      1.10     0.22    -17.97    -48.78      96.12
                 1      0.43     -1.10    -0.22     17.97     22.71      36.37
      3    1     2      1.47     -0.71    -0.48    -37.02     15.69     -53.28
                 4     -1.47      0.71     0.48     37.02     84.04     -95.32
   ************** END OF LATEST ANALYSIS RESULT **************
    34. FINISH
```

V. 3x2 Plane Frame Moments

To find the bending moments due to lateral joint loads in a 3x2 bay plane frame.

Reference

Noris and Wilbur, *Elementary Structural Analysis*, 2nd Edition, McGraw – Hill, Inc., Page 304.

Problem

Determine the bending moments in the members of frame.

E = 30,000 ksi

$I_{AE} = I_{EI} = 240\ in^4$

$I_{BF} = I_{FJ} = 480\ in^4$

$I_{CG} = I_{GK} = 600\ in^4$

$I_{DH} = I_{HL} = 360\ in^4$

$I_{EF} = I_{IJ} = 600\ in^4$

$I_{FG} = I_{JK} = 1{,}200\ in^4$

$I_{GH} = I_{KL} = 1{,}800\ in^4$

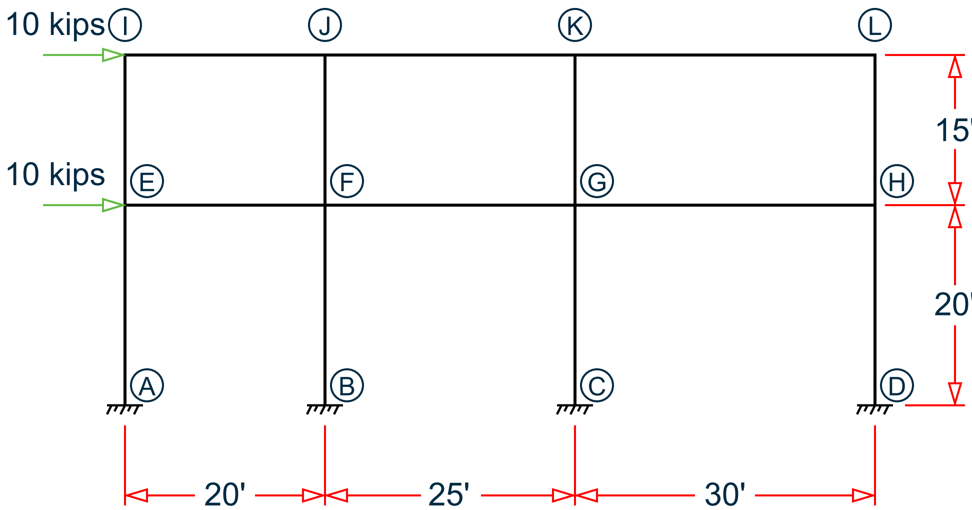


Figure 391: 3x2 bay plane frame

Comparison

Table 415: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Bending in member (ft·kips)	AE	29.6	29.80	<1%
	EA	25.1	25.21	none
	EI	6.5	6.44	<1%
	IE	10.6	10.48	1.1%
	BF	60.9	61.03	none
	FB	53.7	53.69	none
	FJ	18.3	18.50	1.1%
	JF	24.8	24.84	none
	CG	76.8	76.70	none
	GC	68.3	68.24	none
	GK	25.2	25.25	none
	KG	32.4	32.52	none
	DH	45.6	45.45	none
	HD	40.1	39.87	<1%

Result Type		Theory	STAAD.Pro	Difference
	HL	13.5	13.58	<1%
	LH	18.5	18.39	<1%
	EF	31.6	-31.65	none
	FE	29.5	-29.35	<1%
	IJ	10.6	-10.48	1.1%
	JI	10.0	-9.79	2.1%
	FG	42.5	-42.84	<1%
	GF	41.3	-41.71	<1%
	JK	14.8	-15.05	1.7%
	KJ	14.4	-14.71	2.2%
	GH	52.2	-51.78	<1%
	HG	53.6	-53.45	none
	KL	18.0	-17.81	1.1%
	LK	18.5	-18.39	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\3x2 Plane Frame Moments.STD` is typically installed with the program.

```
STAAD PLANE :A 3X2 BAY PLANE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
* REFERENCE: ELEMENTARY STRUCTURAL ANALYSIS, NORIS AND WILBUR, 2ND
*   EDITION. MCGRAW-HILL BOOK COMPANY, PAGE 304, WORKOUT PROBLEM
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 0 35 0; 4 20 0 0; 5 20 20 0; 6 20 35 0; 7 45 0 0;
8 45 20 0; 9 45 35 0; 10 75 0 0; 11 75 20 0; 12 75 35 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 5; 4 5 6; 5 7 8; 6 8 9; 7 10 11; 8 11 12; 9 2 5;
10 5 8; 11 8 11; 12 3 6; 13 6 9; 14 9 12;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
```

```
1 2 PRIS AX 10 IZ 240
3 4 PRIS AX 15 IZ 480
5 6 PRIS AX 20 IZ 600
7 8 PRIS AX 12 IZ 360
9 12 PRIS AX 20 IZ 600
10 13 PRIS AX 30 IZ 1200
11 14 PRIS AX 35 IZ 1800
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 7 10 FIXED
LOAD 1 HORIZONTAL JOINT LOAD
JOINT LOAD
2 3 FX 10
UNIT FEET KIP
PERFORM ANALYSIS
PRINT MEMBER FORCES ALL
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  FEET     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y       MOM-Z
      1     1     1     -4.06      2.75     0.00      0.00      0.00       29.80
                  2      4.06     -2.75     0.00      0.00      0.00       25.21
      2     1     2     -1.01      1.13     0.00      0.00      0.00        6.44
                  3      1.01     -1.13     0.00      0.00      0.00       10.48
      3     1     4     -0.51      5.74     0.00      0.00      0.00       61.03
                  5      0.51     -5.74     0.00      0.00      0.00       53.69
      4     1     5     -0.18      2.89     0.00      0.00      0.00       18.50
                  6      0.18     -2.89     0.00      0.00      0.00       24.84
      5     1     7     -0.14      7.25     0.00      0.00      0.00       76.70
                  8      0.14     -7.25     0.00      0.00      0.00       68.24
      6     1     8     -0.02      3.85     0.00      0.00      0.00       25.25
                  9      0.02     -3.85     0.00      0.00      0.00       32.52
      7     1    10      4.71      4.27     0.00      0.00      0.00       45.45
                 11     -4.71     -4.27     0.00      0.00      0.00       39.87
      8     1    11      1.21      2.13     0.00      0.00      0.00       13.58
                 12     -1.21     -2.13     0.00      0.00      0.00       18.39
      9     1     2      8.38     -3.05     0.00      0.00      0.00      -31.65
                  5     -8.38      3.05     0.00      0.00      0.00      -29.35
     10     1     5      5.53     -3.38     0.00      0.00      0.00      -42.84
                  8     -5.53      3.38     0.00      0.00      0.00      -41.71
     11     1     8      2.14     -3.51     0.00      0.00      0.00      -51.78
                 11     -2.14      3.51     0.00      0.00      0.00      -53.45
     12     1     3      8.87     -1.01     0.00      0.00      0.00      -10.48
                  6     -8.87      1.01     0.00      0.00      0.00       -9.79
```

```
     13     1       6        5.98       -1.19      0.00       0.00       0.00      -15.05
                    9       -5.98        1.19      0.00       0.00       0.00      -14.71
     14     1       9        2.13       -1.21      0.00       0.00       0.00      -17.81
                   12       -2.13        1.21      0.00       0.00       0.00      -18.39
      :A 3X2 BAY PLANE FRAME                                               -- PAGE NO.
4
```

V. Support Reactions for a Simple Frame

Find support reactions due to a load at the free end of a cantilever bent plate with an intermediate support.

Reference

Timoshenko, S., *Strength of Materials*, Part 1, D. Van Nostrand Co., Inc., 3rd edition, 1956, page 346, problem 2.

Problem

Determine the reaction of the system as shown in the figure.

P = 1 kip

L = 10 in

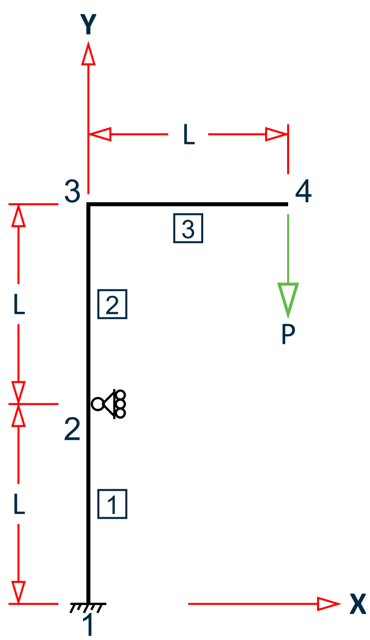


Figure 392: Cantilever model

Comparison

Table 416: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
R_x (kips)	1.5	1.5	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\03 Frames\Support Reactions for a Simple Frame.STD` is typically installed with the program.

```
STAAD PLANE : SUPPORT REACTIONS FOR A SIMPLE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
*
*  REFERENCE 'STRENGTH OF MATERIALS' PART-1 BY S. TIMOSHENKO
*  PAGE 346 PROBLEM NO. 2. THE ANSWER IN THE BOOK AFTER
*  RECALCULATION = 1.5
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 0 20 0; 4 10 20 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS AX 10 IZ 100
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3000
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
2 FIXED BUT FY MZ
LOAD 1
JOINT LOAD
4 FY -1
PERFORM ANALYSIS
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = PLANE
   -----------------
```

```
JOINT  LOAD   FORCE-X    FORCE-Y    FORCE-Z      MOM-X      MOM-Y      MOM Z
     1     1      1.50       1.00       0.00       0.00       0.00      -5.00
     2     1     -1.50       0.00       0.00       0.00       0.00       0.00
  ************** END OF LATEST ANALYSIS RESULT **************
```

V.04 Plate and Shell Elements

V. 2D Cantilever Beam End Deflection 1

To find the free end deflection due to a joint load on a Cantilever beam modeled using Plate/shell elements.

Reference

Hand calculation.

Problem

Using the finite element method calculate the deflection of the free end of the cantilever beam.

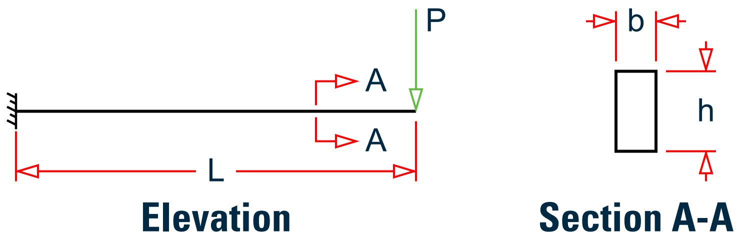


Figure 393: Cantilever beam and section

E = 4,278 ksi

h = 10 in

b = 5 in

P = 2 kips

L = 60 in

Theoretical Solution

Moment of Inertia:

$$I = (5 \text{ in})(10 \text{ in})^3/12 = 416.7 \text{ in}^4$$

Deflection at free end:

$$\delta = \frac{PL^3}{3EI} = \frac{2 \text{ kips}(60 \text{ in})^3}{3(4,278 \text{ ksi})(416.7 \text{ in}^4)} = 0.0808 \text{ in}$$

Comparison

Table 417: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
Deflection at node B (in)	0.0808	0.08245	2.1%	The expression for deflection for beam is used to compare the results of a model with Plate Elements with FE formulation, hence the difference in results.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Cantilever Beam End Deflection 1.STD` is typically installed with the program.

```
STAAD SPACE :A CANTILEVER BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*REFERENCE : HAND CALCULATION
*
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 2 0; 3 0 4 0; 4 0 6 0; 5 0 8 0; 6 0 10 0; 7 2.5 0 0;
8 2.5 2 0; 9 2.5 4 0; 10 2.5 6 0; 11 2.5 8 0; 12 2.5 10 0; 13 5 0 0;
14 5 2 0; 15 5 4 0; 16 5 6 0; 17 5 8 0; 18 5 10 0; 19 7.5 0 0;
20 7.5 2 0; 21 7.5 4 0; 22 7.5 6 0; 23 7.5 8 0; 24 7.5 10 0; 25 10 0 0;
26 10 2 0; 27 10 4 0; 28 10 6 0; 29 10 8 0; 30 10 10 0; 31 12.5 0 0;
32 12.5 2 0; 33 12.5 4 0; 34 12.5 6 0; 35 12.5 8 0; 36 12.5 10 0;
37 15 0 0; 38 15 2 0; 39 15 4 0; 40 15 6 0; 41 15 8 0; 42 15 10 0;
43 17.5 0 0; 44 17.5 2 0; 45 17.5 4 0; 46 17.5 6 0; 47 17.5 8 0;
48 17.5 10 0; 49 20 0 0; 50 20 2 0; 51 20 4 0; 52 20 6 0; 53 20 8 0;
54 20 10 0; 55 22.5 0 0; 56 22.5 2 0; 57 22.5 4 0; 58 22.5 6 0;
59 22.5 8 0; 60 22.5 10 0; 61 25 0 0; 62 25 2 0; 63 25 4 0; 64 25 6 0;
65 25 8 0; 66 25 10 0; 67 27.5 0 0; 68 27.5 2 0; 69 27.5 4 0;
70 27.5 6 0; 71 27.5 8 0; 72 27.5 10 0; 73 30 0 0; 74 30 2 0; 75 30 4 0;
76 30 6 0; 77 30 8 0; 78 30 10 0; 79 32.5 0 0; 80 32.5 2 0; 81 32.5 4 0;
82 32.5 6 0; 83 32.5 8 0; 84 32.5 10 0; 85 35 0 0; 86 35 2 0; 87 35 4 0;
88 35 6 0; 89 35 8 0; 90 35 10 0; 91 37.5 0 0; 92 37.5 2 0; 93 37.5 4 0;
94 37.5 6 0; 95 37.5 8 0; 96 37.5 10 0; 97 40 0 0; 98 40 2 0; 99 40 4 0;
100 40 6 0; 101 40 8 0; 102 40 10 0; 103 42.5 0 0; 104 42.5 2 0;
105 42.5 4 0; 106 42.5 6 0; 107 42.5 8 0; 108 42.5 10 0; 109 45 0 0;
110 45 2 0; 111 45 4 0; 112 45 6 0; 113 45 8 0; 114 45 10 0;
115 47.5 0 0; 116 47.5 2 0; 117 47.5 4 0; 118 47.5 6 0; 119 47.5 8 0;
120 47.5 10 0; 121 50 0 0; 122 50 2 0; 123 50 4 0; 124 50 6 0;
```

```
125 50 8 0; 126 50 10 0; 127 52.5 0 0; 128 52.5 2 0; 129 52.5 4 0;
130 52.5 6 0; 131 52.5 8 0; 132 52.5 10 0; 133 55 0 0; 134 55 2 0;
135 55 4 0; 136 55 6 0; 137 55 8 0; 138 55 10 0; 139 57.5 0 0;
140 57.5 2 0; 141 57.5 4 0; 142 57.5 6 0; 143 57.5 8 0; 144 57.5 10 0;
145 60 0 0; 146 60 2 0; 147 60 4 0; 148 60 6 0; 149 60 8 0; 150 60 10 0;
ELEMENT INCIDENCES SHELL
1 7 1 2 8; 2 8 2 3 9; 3 9 3 4 10; 4 10 4 5 11; 5 11 5 6 12; 6 13 7 8 14;
7 14 8 9 15; 8 15 9 10 16; 9 16 10 11 17; 10 17 11 12 18;
11 19 13 14 20; 12 20 14 15 21; 13 21 15 16 22; 14 22 16 17 23;
15 23 17 18 24; 16 25 19 20 26; 17 26 20 21 27; 18 27 21 22 28;
19 28 22 23 29; 20 29 23 24 30; 21 31 25 26 32; 22 32 26 27 33;
23 33 27 28 34; 24 34 28 29 35; 25 35 29 30 36; 26 37 31 32 38;
27 38 32 33 39; 28 39 33 34 40; 29 40 34 35 41; 30 41 35 36 42;
31 43 37 38 44; 32 44 38 39 45; 33 45 39 40 46; 34 46 40 41 47;
35 47 41 42 48; 36 49 43 44 50; 37 50 44 45 51; 38 51 45 46 52;
39 52 46 47 53; 40 53 47 48 54; 41 55 49 50 56; 42 56 50 51 57;
43 57 51 52 58; 44 58 52 53 59; 45 59 53 54 60; 46 61 55 56 62;
47 62 56 57 63; 48 63 57 58 64; 49 64 58 59 65; 50 65 59 60 66;
51 67 61 62 68; 52 68 62 63 69; 53 69 63 64 70; 54 70 64 65 71;
55 71 65 66 72; 56 73 67 68 74; 57 74 68 69 75; 58 75 69 70 76;
59 76 70 71 77; 60 77 71 72 78; 61 79 73 74 80; 62 80 74 75 81;
63 81 75 76 82; 64 82 76 77 83; 65 83 77 78 84; 66 85 79 80 86;
67 86 80 81 87; 68 87 81 82 88; 69 88 82 83 89; 70 89 83 84 90;
71 91 85 86 92; 72 92 86 87 93; 73 93 87 88 94; 74 94 88 89 95;
75 95 89 90 96; 76 97 91 92 98; 77 98 92 93 99; 78 99 93 94 100;
79 100 94 95 101; 80 101 95 96 102; 81 103 97 98 104; 82 104 98 99 105;
83 105 99 100 106; 84 106 100 101 107; 85 107 101 102 108;
86 109 103 104 110; 87 110 104 105 111; 88 111 105 106 112;
89 112 106 107 113; 90 113 107 108 114; 91 115 109 110 116;
92 116 110 111 117; 93 117 111 112 118; 94 118 112 113 119;
95 119 113 114 120; 96 121 115 116 122; 97 122 116 117 123;
98 123 117 118 124; 99 124 118 119 125; 100 125 119 120 126;
101 127 121 122 128; 102 128 122 123 129; 103 129 123 124 130;
104 130 124 125 131; 105 131 125 126 132; 106 133 127 128 134;
107 134 128 129 135; 108 135 129 130 136; 109 136 130 131 137;
110 137 131 132 138; 111 139 133 134 140; 112 140 134 135 141;
113 141 135 136 142; 114 142 136 137 143; 115 143 137 138 144;
116 145 139 140 146; 117 146 140 141 147; 118 147 141 142 148;
119 148 142 143 149; 120 149 143 144 150;
ELEMENT PROPERTY
1 TO 120 THICKNESS 5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4278
POISSON 0.1889
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 5 6 FIXED BUT FY
3 4 FIXED
LOAD 1 VERTICAL JOINT LOADS
JOINT LOAD
145 TO 150 FY -0.333333
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 145 TO 150
FINISH
```

STAAD Output

```
  JOINT    DISPLACE LIST     145
      :A CANTILEVER BEAM                                       -- PAGE NO.
4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    145    1   -0.01016  -0.08254   0.00000   0.00000   0.00000  -0.00189
    146    1   -0.00605  -0.08248   0.00000   0.00000   0.00000  -0.00177
    147    1   -0.00201  -0.08245   0.00000   0.00000   0.00000  -0.00175
    148    1    0.00201  -0.08245   0.00000   0.00000   0.00000  -0.00175
    149    1    0.00605  -0.08248   0.00000   0.00000   0.00000  -0.00177
    150    1    0.01016  -0.08254   0.00000   0.00000   0.00000  -0.00189
   ************** END OF LATEST ANALYSIS RESULT **************
```

V. 2D Cantilever Beam End Deflection 2

Find the deflection and moments for plate-bending finite element due to a pressure load.

Reference

Simple hand calculation by considering the entire structure as a cantilever beam.

Problem

A simple cantilever plate is divided into 12 4-noded finite elements. A uniform pressure load is applied and the maximum deflection at the tip of the cantilever and the maximum bending at the support are calculated.

Plate thickness = 25 mm

Uniform pressure= 5 N/mm^2

Plate length = 6 spaces at 50 mm = 300 mm

Plate width = 2 space at 50 mm = 100 mm

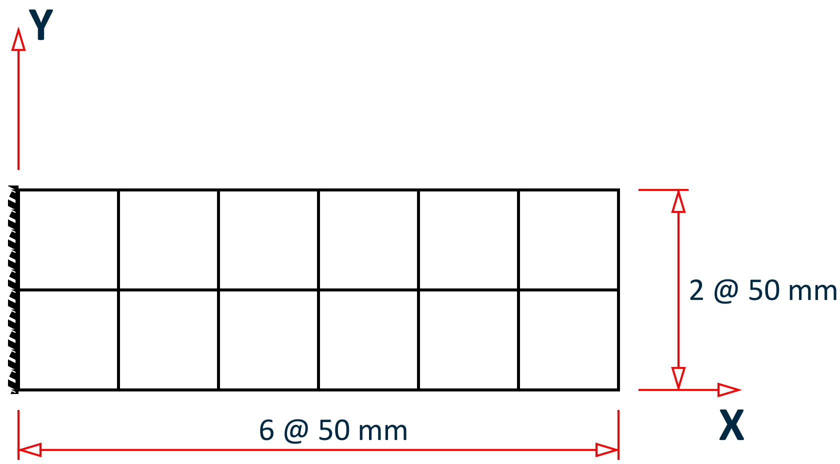


Figure 394: Finite element mesh of cantilevered plate.

Theoretical Solution

Maximum deflection is equal to $WL^3/8EI$, where:

$$\Delta_{max} = \frac{5(300)(100)(300)^3}{8(210 \cdot 10^3)\left(\frac{100 \cdot 25^3}{12}\right)} = \frac{4050(10)^9}{218.75(10)^9} = 18.51\text{mm}$$

Maximum Moment:

$$M_{max} = \frac{WL}{2} = \frac{5(300)(100)(300)}{2} = 22.5(10)^6 \text{N} \cdot \text{mm}$$

Comparison

Table 418: Comparison of results

Result Type	Hand Calculation	STAAD.Pro	Difference
δ_{max} (mm)	18.51	18.159	1.9%
M_{max} (kN·m)	22.50	22.50	none

Note: The maximum moment is taken as the sum of the moments at nodes 1, 8, and 15 (i.e., $5.47 + 11.56 + 5.47 = 22.5\ kN \cdot m$).

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Cantilever Beam End Deflection 2.STD` is typically installed with the program.

```
STAAD SPACE : DEFLECTION OF A CANTILEVER PLATE UNDER UNIFORM PRESSURE
START JOB INFORMATION
```

```
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* DEFLECTION OF A CANTILEVER PLATE UNDER UNIFORM PRESSURE.
* COMPARISON WITH ESTABLISHED FORMULA (WL^3/8EI)
*
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 50 0 0; 3 100 0 0; 4 150 0 0; 5 200 0 0; 6 250 0 0;
7 300 0 0; 8 0 50 0; 9 50 50 0; 10 100 50 0; 11 150 50 0; 12 200 50 0;
13 250 50 0; 14 300 50 0; 15 0 100 0; 16 50 100 0; 17 100 100 0;
18 150 100 0; 19 200 100 0; 20 250 100 0; 21 300 100 0;
ELEMENT INCIDENCES SHELL
1 1 2 9 8; 2 2 3 10 9; 3 3 4 11 10; 4 4 5 12 11; 5 5 6 13 12;
6 6 7 14 13; 7 8 9 16 15; 8 9 10 17 16; 9 10 11 18 17; 10 11 12 19 18;
11 12 13 20 19; 12 13 14 21 20;
ELEMENT PROPERTY
1 TO 12 THICKNESS 25
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.1e+08
POISSON 0.3
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 8 15 FIXED
UNIT MMS NEWTON
LOAD 1 5N/SQ.MM. UNIFORM LOAD
ELEMENT LOAD
1 TO 12 PR 5
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 14
UNIT METER KN
PRINT SUPPORT REACTION
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     14    1    0.0000    0.0000    1.8159    0.0000   -0.0813    0.0000
   ************** END OF LATEST ANALYSIS RESULT **************
    38. UNIT METER KN
    39. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      : DEFLECTION OF A CANTILEVER PLATE UNDER UNIFORM PRESSUR -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00      0.00    -18.91     -1.54      5.47      0.00
```

8	1	0.00	0.00	-112.19	0.00	11.56	0.00
15	1	0.00	0.00	-18.91	1.54	5.47	0.00

V. 2D Circular Plate In-Plane Stresses

A thick cylindrical plate supported along 2 radial edges. Find the radial displacement, radial stress, tangential stress and longitudinal stress at inner surface due to a unit pressure applied at the inner surface.

Reference

MacNeal, R.H. and Harder, R.C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*, 1985.

Problem

Loading is unit pressure at inner radius

$E = 1 \times (10)^6$ psi

Poisson's ratio = 0.3

Inner radius = 3.0 in

Outer radius = 9.0 in

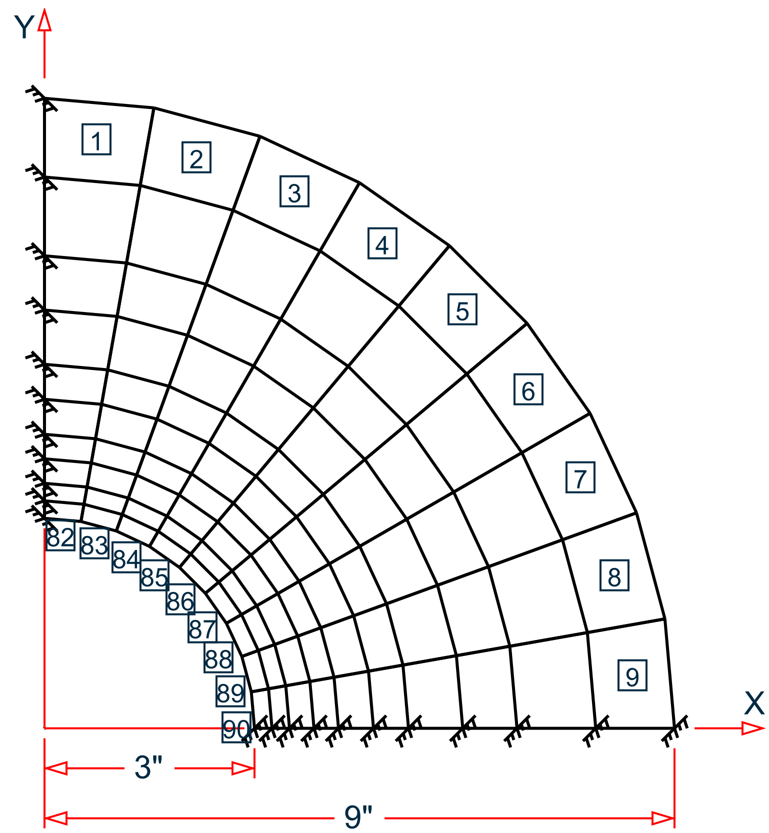


Figure 395: Semi-circular plate finite element model

Comparison

Table 419: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Radial deflection (10^{-3} in)	4.582	4.650[a]	1.5%
Radial stress (psi)	-1.00	-0.95[b]	5.0%
Tangential stress (psi)	1.25	1.26[b]	0.8%
Longitudinal stress (psi)	0.075	0.080	7.0%

a. Radial displacements are measured along FY at node 102 and FX at node 101.
b. On element 82, at node 102, SX is tangential stress, SY is radial stress.

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Circular Plate In-Plane Stresses.STD` is typically installed with the program.

```
STAAD SPACE :A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL EDGES
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
*            TO TEST FINITE ELEMENT ACCURACY,
*            FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
*            1985
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 9 0 0; 2 0 9 0; 3 1.563 8.863 0; 4 3.078 8.457 0; 5 4.5 7.794 0;
6 5.785 6.894 0; 7 6.894 5.785 0; 8 7.794 4.5 0; 9 8.457 3.078 0;
10 8.863 1.563 0; 11 7.875 0 0; 12 0 7.875 0; 13 1.367 7.755 0;
14 2.693 7.4 0; 15 3.938 6.82 0; 16 5.062 6.033 0; 17 6.033 5.062 0;
18 6.82 3.937 0; 19 7.4 2.693 0; 20 7.755 1.367 0; 21 6.75 0 0;
22 0 6.75 0; 23 1.172 6.647 0; 24 2.309 6.343 0; 25 3.375 5.846 0;
26 4.339 5.171 0; 27 5.171 4.339 0; 28 5.846 3.375 0; 29 6.343 2.309 0;
30 6.647 1.172 0; 31 5.975 0 0; 32 0 5.975 0; 33 1.038 5.884 0;
34 2.044 5.615 0; 35 2.988 5.174 0; 36 3.841 4.577 0; 37 4.577 3.841 0;
38 5.175 2.987 0; 39 5.615 2.044 0; 40 5.884 1.038 0; 41 5.2 0 0;
42 0 5.2 0; 43 0.903 5.121 0; 44 1.779 4.886 0; 45 2.6 4.503 0;
46 3.343 3.983 0; 47 3.983 3.342 0; 48 4.503 2.6 0; 49 4.886 1.778 0;
50 5.121 0.903 0; 51 4.7 0 0; 52 0 4.7 0; 53 0.816 4.629 0;
54 1.608 4.417 0; 55 2.35 4.07 0; 56 3.021 3.6 0; 57 3.6 3.021 0;
58 4.07 2.35 0; 59 4.417 1.607 0; 60 4.629 0.816 0; 61 4.2 0 0;
```

```
62 0 4.2 0; 63 0.729 4.136 0; 64 1.436 3.947 0; 65 2.1 3.637 0;
66 2.7 3.217 0; 67 3.217 2.7 0; 68 3.637 2.1 0; 69 3.947 1.436 0;
70 4.136 0.729 0; 71 3.85 0 0; 72 0 3.85 0; 73 0.669 3.792 0;
74 1.317 3.618 0; 75 1.925 3.334 0; 76 2.475 2.949 0; 77 2.949 2.475 0;
78 3.334 1.925 0; 79 3.618 1.317 0; 80 3.792 0.669 0; 81 3.5 0 0;
82 0 3.5 0; 83 0.608 3.447 0; 84 1.197 3.289 0; 85 1.75 3.031 0;
86 2.25 2.681 0; 87 2.681 2.25 0; 88 3.031 1.75 0; 89 3.289 1.197 0;
90 3.447 0.608 0; 91 3.25 0 0; 92 0 3.25 0; 93 0.564 3.201 0;
94 1.112 3.054 0; 95 1.625 2.815 0; 96 2.089 2.49 0; 97 2.49 2.089 0;
98 2.815 1.625 0; 99 3.054 1.112 0; 100 3.201 0.564 0; 101 3 0 0;
102 0 3 0; 103 0.521 2.954 0; 104 1.026 2.819 0; 105 1.5 2.598 0;
106 1.928 2.298 0; 107 2.298 1.928 0; 108 2.598 1.5 0;
109 2.819 1.026 0; 110 2.954 0.521 0;
ELEMENT INCIDENCES SHELL
1 2 3 13 12; 2 3 4 14 13; 3 4 5 15 14; 4 5 6 16 15; 5 6 7 17 16;
6 7 8 18 17; 7 8 9 19 18; 8 9 10 20 19; 9 10 1 11 20; 10 12 13 23 22;
11 13 14 24 23; 12 14 15 25 24; 13 15 16 26 25; 14 16 17 27 26;
15 17 18 28 27; 16 18 19 29 28; 17 19 20 30 29; 18 20 11 21 30;
19 22 23 33 32; 20 23 24 34 33; 21 24 25 35 34; 22 25 26 36 35;
23 26 27 37 36; 24 27 28 38 37; 25 28 29 39 38; 26 29 30 40 39;
27 30 21 31 40; 28 32 33 43 42; 29 33 34 44 43; 30 34 35 45 44;
31 35 36 46 45; 32 36 37 47 46; 33 37 38 48 47; 34 38 39 49 48;
35 39 40 50 49; 36 40 31 41 50; 37 42 43 53 52; 38 43 44 54 53;
39 44 45 55 54; 40 45 46 56 55; 41 46 47 57 56; 42 47 48 58 57;
43 48 49 59 58; 44 49 50 60 59; 45 50 41 51 60; 46 52 53 63 62;
47 53 54 64 63; 48 54 55 65 64; 49 55 56 66 65; 50 56 57 67 66;
51 57 58 68 67; 52 58 59 69 68; 53 59 60 70 69; 54 60 51 61 70;
55 62 63 73 72; 56 63 64 74 73; 57 64 65 75 74; 58 65 66 76 75;
59 66 67 77 76; 60 67 68 78 77; 61 68 69 79 78; 62 69 70 80 79;
63 70 61 71 80; 64 72 73 83 82; 65 73 74 84 83; 66 74 75 85 84;
67 75 76 86 85; 68 76 77 87 86; 69 77 78 88 87; 70 78 79 89 88;
71 79 80 90 89; 72 80 71 81 90; 73 82 83 93 92; 74 83 84 94 93;
75 84 85 95 94; 76 85 86 96 95; 77 86 87 97 96; 78 87 88 98 97;
79 88 89 99 98; 80 89 90 100 99; 81 90 81 91 100; 82 92 93 103 102;
83 93 94 104 103; 84 94 95 105 104; 85 95 96 106 105; 86 96 97 107 106;
87 97 98 108 107; 88 98 99 109 108; 89 99 100 110 109;
90 100 91 101 110;
ELEMENT PROPERTY
1 TO 90 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
2 12 22 32 42 52 62 72 82 92 102 FIXED BUT FY
1 11 21 31 41 51 61 71 81 91 101 FIXED BUT FX
LOAD 1 UNIT PRESSURE AT INNER RADIUS
JOINT LOAD
101 FX 0.2618
110 FX 0.5156 FY 0.0909
109 FX 0.492 FY 0.1791
108 FX 0.4534 FY 0.2618
107 FX 0.4011 FY 0.3366
106 FX 0.3366 FY 0.4011
105 FX 0.2618 FY 0.4534
```

```
104 FX 0.1791 FY 0.492
103 FX 0.0909 FY 0.5156
102 FY 0.2618
* CREATED LOAD 2 IN ORDER TO PRINT DISPLACEMENT VALUES
LOAD 2 MULTIPLY LOAD BY 1000
REPEAT LOAD
1 1000.0
PERFORM ANALYSIS
LOAD LIST 2
PRINT JOINT DISPLACEMENTS LIST 101 102 107
LOAD LIST 1
PRINT ELEMENT JOINT STRESSES LIST 82 83 89 90
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    101    2    0.00465   0.00000   0.00000   0.00000   0.00000   0.00000
    102    2    0.00000   0.00465   0.00000   0.00000   0.00000   0.00000
    107    2    0.00357   0.00299   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
    97. LOAD LIST 1
    98. PRINT ELEMENT JOINT STRESSES LIST 82 83 89 90
  ELEMENT  JOINT    STRESSES LIST
      :A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL  -- PAGE NO.
5
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY          MXY
                      VONT       VONB         SX           SY          SXY
                      TRESCAT    TRESCAB
     82      1         0.00        0.00        0.00        0.00        0.00
                       1.81        1.81        1.17       -0.91        0.00
                       2.08        2.08
       TOP : SMAX=       1.17 SMIN=      -0.91 TMAX=       1.04 ANGLE=  0.0
       BOTT: SMAX=       1.17 SMIN=      -0.91 TMAX=       1.04 ANGLE=  0.0
         JOINT         0.00        0.00        0.00        0.00        0.00
           92          1.69        1.69        1.07       -0.88       -0.08
       TOP : SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE= -2.3
       BOTT: SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE= -2.3
         JOINT         0.00        0.00        0.00        0.00        0.00
           93          1.69        1.69        1.07       -0.88        0.08
       TOP : SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE=  2.3
       BOTT: SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE=  2.3
         JOINT         0.00        0.00        0.00        0.00        0.00
          103          1.93        1.93        1.26       -0.95        0.08
       TOP : SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE=  2.1
       BOTT: SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE=  2.1
         JOINT         0.00        0.00        0.00        0.00        0.00
          102          1.93        1.93        1.26       -0.95       -0.08
       TOP : SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE= -2.0
       BOTT: SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE= -2.0
     83      1         0.00        0.00        0.00        0.00        0.00
```

```
                    1.81        1.81        1.17       -0.92       -0.00
                    2.08        2.08
       TOP : SMAX=    1.17 SMIN=     -0.92 TMAX=      1.04 ANGLE= -0.0
       BOTT: SMAX=    1.17 SMIN=     -0.92 TMAX=      1.04 ANGLE= -0.0
         JOINT      0.00        0.00        0.00        0.00        0.00
           93       1.69        1.69        1.07       -0.88       -0.08
       TOP : SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
       BOTT: SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
         JOINT      0.00        0.00        0.00        0.00        0.00
           94       1.69        1.69        1.06       -0.88        0.08
       TOP : SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE=  2.3
       BOTT: SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE=  2.3
         JOINT      0.00        0.00        0.00        0.00        0.00
          104       1.93        1.93        1.26       -0.95        0.08
       TOP : SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE=  2.1
       BOTT: SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE=  2.1
         JOINT      0.00        0.00        0.00        0.00        0.00
          103       1.93        1.93        1.26       -0.95       -0.08
       TOP : SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE= -2.1
       BOTT: SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE= -2.1
     89      1      0.00        0.00        0.00        0.00        0.00
                    1.81        1.81        1.17       -0.92        0.00
                    2.08        2.08
       TOP : SMAX=    1.17 SMIN=     -0.92 TMAX=      1.04 ANGLE=  0.0
       BOTT: SMAX=    1.17 SMIN=     -0.92 TMAX=      1.04 ANGLE=  0.0
      :A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL  -- PAGE NO.
6
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX          MY          MXY
                      VONT        VONB         SX          SY          SXY
                      TRESCAT     TRESCAB
         JOINT      0.00        0.00        0.00        0.00        0.00
           99       1.69        1.69        1.06       -0.88       -0.08
       TOP : SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
       BOTT: SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
         JOINT      0.00        0.00        0.00        0.00        0.00
          100       1.69        1.69        1.07       -0.88        0.08
       TOP : SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE=  2.3
       BOTT: SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE=  2.3
         JOINT      0.00        0.00        0.00        0.00        0.00
          110       1.93        1.93        1.26       -0.95        0.08
       TOP : SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE=  2.1
       BOTT: SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE=  2.1
         JOINT      0.00        0.00        0.00        0.00        0.00
          109       1.93        1.93        1.26       -0.95       -0.08
       TOP : SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE= -2.1
       BOTT: SMAX=    1.26 SMIN=     -0.96 TMAX=      1.11 ANGLE= -2.1
     90      1      0.00        0.00        0.00        0.00        0.00
                    1.81        1.81        1.17       -0.91       -0.00
                    2.08        2.08
       TOP : SMAX=    1.17 SMIN=     -0.91 TMAX=      1.04 ANGLE= -0.0
       BOTT: SMAX=    1.17 SMIN=     -0.91 TMAX=      1.04 ANGLE= -0.0
         JOINT      0.00        0.00        0.00        0.00        0.00
          100       1.69        1.69        1.07       -0.88       -0.08
       TOP : SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
       BOTT: SMAX=    1.07 SMIN=     -0.88 TMAX=      0.98 ANGLE= -2.3
```

```
          JOINT         0.00        0.00        0.00        0.00        0.00
            91          1.69        1.69        1.07       -0.88        0.08
        TOP : SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE=  2.3
        BOTT: SMAX=       1.07 SMIN=      -0.88 TMAX=       0.98 ANGLE=  2.3
          JOINT         0.00        0.00        0.00        0.00        0.00
           101          1.93        1.93        1.26       -0.95        0.08
        TOP : SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE=  2.0
        BOTT: SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE=  2.0
          JOINT         0.00        0.00        0.00        0.00        0.00
           110          1.93        1.93        1.26       -0.95       -0.08
        TOP : SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE= -2.1
        BOTT: SMAX=       1.26 SMIN=      -0.95 TMAX=       1.11 ANGLE= -2.1
       :A THICK WALLED CYLINDER PLATE SUPPORTED ALONG 2 RADIAL  -- PAGE NO.
7
           **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
              MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
             PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
              STRESS        STRESS        STRESS        STRESS        STRESS
           1.263807E+00 -9.557953E-01  1.109365E+00  1.807750E+00  2.082330E+00
 PLATE NO.      82            83            82            89            89
 CASE  NO.       1             1             1             1             1
```

V. 2D Circular Surface Displacements and Stresses

A circular plate is fixed along its perimeter. Using plate/shell elements, find the deflection at the center, maximum bending stress due to a uniformly distributed load, and a concentrated load at the center.

Reference

Timoshenko, S., *Strength of Materials*, Part II, 3rd Edition, Van Nostrand Co., 1956, pp.96-97, 103.

Problem

The circular plate shown below is subject to two load cases. Load 1 is a uniform pressure, w, and load 2 is a concentrated force, P, at the center. Determine:

- deflection at the center for both load cases
- bending stress at the support for both load cases
- moment at the center for load case 1

E = 30,000.0 ksi

Poisson's ratio = 0.3

r = 40 in.

t = 1 in.

w = 6 psi.

P = 7,539.82 lbs

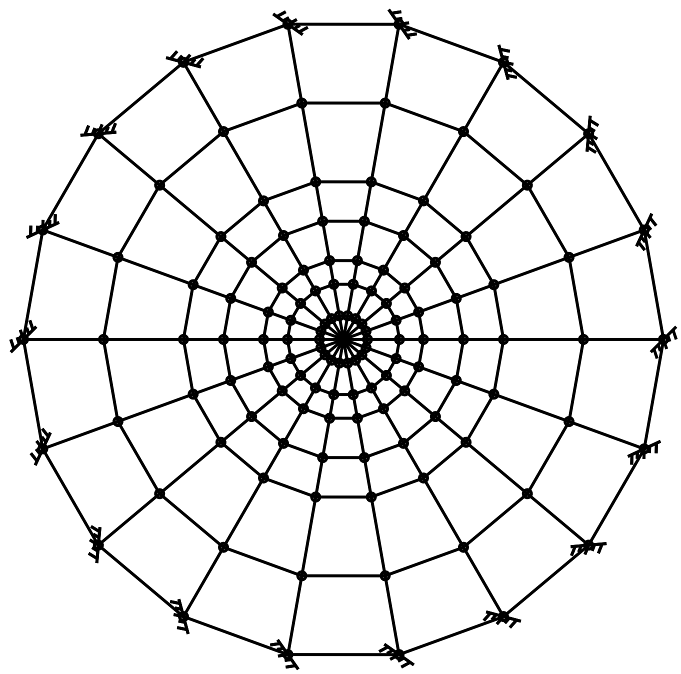


Figure 396: Finite element model of a circular plate

Comparison

Table 420: Comparison of results

Result Type		Theory	STAAD.Pro	Difference	Comments
Load Case 1	σ_{bend} (psi)	7,200	6,887[a]	4.3%	The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.
	δ_{max} (in) (Y translation at Node 127)	-0.0874	-0.08618	1.4%	
	Moment at center (in·lb/in)	780	806.1[b]	3.4%	
Load Case 2	σ_{bend} (psi)	3,600	3,700[c]	2.8%	
	δ_{max} (in) (Y translation at Node 127)	-0.0874	-0.08646	1.0%	

a. In element 9 at node 10, My = 1,147.91 in·lb/in. This yields σ_{bend} = 1,147.91(6) = 6,887 psi.
b. In element 109 at node 127, Mx = 811.88 in·lb/in and My = 800.33 in·lb/in, the average of which is 806.1 in·lb/in.
c. In element 9 at node 10, My = 616.69 in·lb/in. This yields σ_{bend} = 616.19(6) = 3,700 psi.

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\04 Plates Shells\2D Circular Surface Displacements and Stresses.STD` is typically installed with the program.

```
STAAD SPACE :A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: TIMOSHENKO, S., STRENGTH OF MATERIALS
*            PART II, ADVANCED THEORY AND PROBLEMS
*            PAGES 96, 97, AND 103.
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 40 0 0; 2 37.588 0 -13.681; 3 30.642 0 -25.712; 4 20 0 -34.641;
5 6.946 0 -39.392; 6 -6.946 0 -39.392; 7 -20 0 -34.641; 8 -30.642 0 -25.711;
9 -37.588 0 -13.681; 10 -40 0 0; 11 -37.588 0 13.681; 12 -30.642 0 25.712;
13 -20 0 34.641; 14 -6.946 0 39.392; 15 6.946 0 39.392; 16 20 0 34.641;
17 30.642 0 25.711; 18 37.588 0 13.681; 19 30 0 0; 20 28.191 0 -10.261;
21 22.981 0 -19.284; 22 15 0 -25.981; 23 5.209 0 -29.544; 24 -5.209 0 -29.544;
25 -15 0 -25.981; 26 -22.981 0 -19.284; 27 -28.191 0 -10.261; 28 -30 0 0;
29 -28.191 0 10.261; 30 -22.981 0 19.284; 31 -15 0 25.981; 32 -5.209 0 29.544;
33 5.21 0 29.544; 34 15 0 25.981; 35 22.981 0 19.284; 36 28.191 0 10.261;
37 20 0 0; 38 18.794 0 -6.84; 39 15.321 0 -12.856; 40 10 0 -17.321;
41 3.473 0 -19.696; 42 -3.473 0 -19.696; 43 -10 0 -17.32; 44 -15.321 0
-12.856;
45 -18.794 0 -6.84; 46 -20 0 0; 47 -18.794 0 6.84; 48 -15.321 0 12.856;
49 -10 0 17.321; 50 -3.473 0 19.696; 51 3.473 0 19.696; 52 10 0 17.32;
53 15.321 0 12.856; 54 18.794 0 6.84; 55 15 0 0; 56 14.095 0 -5.13;
57 11.491 0 -9.642; 58 7.5 0 -12.99; 59 2.605 0 -14.772; 60 -2.605 0 -14.772;
61 -7.5 0 -12.99; 62 -11.491 0 -9.642; 63 -14.095 0 -5.13; 64 -15 0 0;
65 -14.095 0 5.13; 66 -11.491 0 9.642; 67 -7.5 0 12.99; 68 -2.605 0 14.772;
69 2.605 0 14.772; 70 7.5 0 12.99; 71 11.491 0 9.642; 72 14.095 0 5.13;
73 10 0 0; 74 9.397 0 -3.42; 75 7.66 0 -6.428; 76 5 0 -8.66; 77 1.736 0
-9.848;
78 -1.736 0 -9.848; 79 -5 0 -8.66; 80 -7.66 0 -6.428; 81 -9.397 0 -3.42;
82 -10 0 0; 83 -9.397 0 3.42; 84 -7.66 0 6.428; 85 -5 0 8.66;
86 -1.736 0 9.848; 87 1.737 0 9.848; 88 5 0 8.66; 89 7.66 0 6.428;
90 9.397 0 3.42; 91 7 0 0; 92 6.578 0 -2.394; 93 5.362 0 -4.5; 94 3.5 0
-6.062;
95 1.216 0 -6.894; 96 -1.216 0 -6.894; 97 -3.5 0 -6.062; 98 -5.362 0 -4.5;
99 -6.578 0 -2.394; 100 -7 0 0; 101 -6.578 0 2.394; 102 -5.362 0 4.5;
103 -3.5 0 6.062; 104 -1.216 0 6.894; 105 1.216 0 6.894; 106 3.5 0 6.062;
107 5.362 0 4.5; 108 6.578 0 2.394; 109 3 0 0; 110 2.819 0 -1.026;
111 2.298 0 -1.928; 112 1.5 0 -2.598; 113 0.521 0 -2.954; 114 -0.521 0 -2.954;
115 -1.5 0 -2.598; 116 -2.298 0 -1.928; 117 -2.819 0 -1.026; 118 -3 0 0;
119 -2.819 0 1.026; 120 -2.298 0 1.928; 121 -1.5 0 2.598; 122 -0.521 0 2.954;
123 0.521 0 2.954; 124 1.5 0 2.598; 125 2.298 0 1.928; 126 2.819 0 1.026;
127 0 0 0;
ELEMENT INCIDENCES SHELL
```

```
1 1 2 20 19; 2 2 3 21 20; 3 3 4 22 21; 4 4 5 23 22; 5 5 6 24 23; 6 6 7 25 24;
7 7 8 26 25; 8 8 9 27 26; 9 9 10 28 27; 10 10 11 29 28; 11 11 12 30 29;
12 12 13 31 30; 13 13 14 32 31; 14 14 15 33 32; 15 15 16 34 33; 16 16 17 35
34;
17 17 18 36 35; 18 18 1 19 36; 19 19 20 38 37; 20 20 21 39 38; 21 21 22 40 39;
22 22 23 41 40; 23 23 24 42 41; 24 24 25 43 42; 25 25 26 44 43; 26 26 27 45
44;
27 27 28 46 45; 28 28 29 47 46; 29 29 30 48 47; 30 30 31 49 48; 31 31 32 50
49;
32 32 33 51 50; 33 33 34 52 51; 34 34 35 53 52; 35 35 36 54 53; 36 36 19 37
54;
37 37 38 56 55; 38 38 39 57 56; 39 39 40 58 57; 40 40 41 59 58; 41 41 42 60
59;
42 42 43 61 60; 43 43 44 62 61; 44 44 45 63 62; 45 45 46 64 63; 46 46 47 65
64;
47 47 48 66 65; 48 48 49 67 66; 49 49 50 68 67; 50 50 51 69 68; 51 51 52 70
69;
52 52 53 71 70; 53 53 54 72 71; 54 54 37 55 72; 55 55 56 74 73; 56 56 57 75
74;
57 57 58 76 75; 58 58 59 77 76; 59 59 60 78 77; 60 60 61 79 78; 61 61 62 80
79;
62 62 63 81 80; 63 63 64 82 81; 64 64 65 83 82; 65 65 66 84 83; 66 66 67 85
84;
67 67 68 86 85; 68 68 69 87 86; 69 69 70 88 87; 70 70 71 89 88; 71 71 72 90
89;
72 72 55 73 90; 73 73 74 92 91; 74 74 75 93 92; 75 75 76 94 93; 76 76 77 95
94;
77 77 78 96 95; 78 78 79 97 96; 79 79 80 98 97; 80 80 81 99 98;
81 81 82 100 99; 82 82 83 101 100; 83 83 84 102 101; 84 84 85 103 102;
85 85 86 104 103; 86 86 87 105 104; 87 87 88 106 105; 88 88 89 107 106;
89 89 90 108 107; 90 90 73 91 108; 91 91 92 110 109; 92 92 93 111 110;
93 93 94 112 111; 94 94 95 113 112; 95 95 96 114 113; 96 96 97 115 114;
97 97 98 116 115; 98 98 99 117 116; 99 99 100 118 117; 100 100 101 119 118;
101 101 102 120 119; 102 102 103 121 120; 103 103 104 122 121;
104 104 105 123 122; 105 105 106 124 123; 106 106 107 125 124;
107 107 108 126 125; 108 108 91 109 126; 109 127 109 110; 110 110 111 127;
111 127 111 112; 112 112 113 127; 113 127 113 114; 114 114 115 127;
115 127 115 116; 116 116 117 127; 117 127 117 118; 118 127 109 126;
119 126 125 127; 120 127 125 124; 121 124 123 127; 122 127 123 122;
123 122 121 127; 124 127 121 120; 125 120 119 127; 126 127 119 118;
ELEMENT PROPERTY
1 TO 126 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 18 FIXED
LOAD 1 UNIFORM LOAD
ELEMENT LOAD
1 TO 126 PR GY -6
LOAD 2 POINT LOAD
JOINT LOAD
127 FY -7539.82
PERFORM ANALYSIS
```

```
PRINT JOINT DISPLACEMENTS LIST 127
LOAD LIST 1
PRINT ELEMENT JOINT STRESSES LIST 9 109
LOAD LIST 2
PRINT ELEMENT JOINT STRESSES LIST 9
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    127    1    0.00000  -0.08618   0.00000  -0.00000   0.00000  -0.00000
           2    0.00000  -0.08646   0.00000   0.00000   0.00000  -0.00001
   ************** END OF LATEST ANALYSIS RESULT **************
    93. LOAD LIST 1
    94. PRINT ELEMENT JOINT STRESSES LIST 9 109
  ELEMENT  JOINT    STRESSES LIST
      :A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER              -- PAGE NO.
5
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
      9      1         0.00     -103.62      115.36      719.90        0.01
                    4018.27     4018.27        0.00        0.00        0.00
                    4319.39     4319.39
       TOP : SMAX=    4319.39 SMIN=     692.18 TMAX=    1813.61 ANGLE= 90.0
       BOTT: SMAX=    -692.18 SMIN=   -4319.39 TMAX=    1813.61 ANGLE= -0.0
         JOINT        14.39     -113.81      338.29     1147.77      123.92
           9        6263.05     6263.05        0.00        0.00        0.00
       TOP : SMAX=    6997.90 SMIN=    1918.50 TMAX=    2539.70 ANGLE= 81.5
       BOTT: SMAX=   -1918.50 SMIN=   -6997.90 TMAX=    2539.70 ANGLE= -8.5
         JOINT       -14.38     -113.83      338.35     1147.91     -123.91
          10        6263.75     6263.75        0.00        0.00        0.00
       TOP : SMAX=    6998.72 SMIN=    1918.87 TMAX=    2539.93 ANGLE=-81.5
       BOTT: SMAX=   -1918.87 SMIN=   -6998.72 TMAX=    2539.93 ANGLE=  8.5
         JOINT       -10.79      -93.43     -124.84      381.27     -175.70
          28        3292.73     3292.73        0.00        0.00        0.00
       TOP : SMAX=    2617.74 SMIN=   -1079.14 TMAX=    1848.44 ANGLE=-72.6
       BOTT: SMAX=    1079.14 SMIN=   -2617.74 TMAX=    1848.44 ANGLE= 17.4
         JOINT        10.79      -93.42     -124.84      381.32      175.75
          27        3293.22     3293.22        0.00        0.00        0.00
       TOP : SMAX=    2618.18 SMIN=   -1079.25 TMAX=    1848.71 ANGLE= 72.6
       BOTT: SMAX=    1079.25 SMIN=   -2618.18 TMAX=    1848.71 ANGLE=-17.4
    109      1         6.37        1.93     -799.52     -799.67       -0.21
                    4797.59     4797.59        0.00        0.00        0.00
                    4798.94     4798.94
       TOP : SMAX=   -4796.24 SMIN=   -4798.94 TMAX=       1.35 ANGLE=-35.3
       BOTT: SMAX=    4798.94 SMIN=    4796.24 TMAX=       1.35 ANGLE= 54.7
         JOINT         6.37        1.93     -811.88     -800.33       -0.21
          127       4837.01     4837.01        0.00        0.00        0.00
       TOP : SMAX=   -4801.97 SMIN=   -4871.30 TMAX=      34.67 ANGLE=-88.9
       BOTT: SMAX=    4871.30 SMIN=    4801.97 TMAX=      34.67 ANGLE=  1.1
```

```
        JOINT         6.37        1.93     -792.77     -800.33       -0.21
         109       4779.46     4779.46        0.00        0.00        0.00
      TOP : SMAX=   -4756.58 SMIN=   -4802.02 TMAX=      22.72 ANGLE= -1.6
      BOTT: SMAX=    4802.02 SMIN=    4756.58 TMAX=      22.72 ANGLE= 88.4
        JOINT         6.37        1.93     -793.92     -798.36       -0.21
         110       4776.89     4776.89        0.00        0.00        0.00
      TOP : SMAX=   -4763.47 SMIN=   -4790.19 TMAX=      13.36 ANGLE= -2.7
      BOTT: SMAX=    4790.19 SMIN=    4763.47 TMAX=      13.36 ANGLE= 87.3
     :A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER              -- PAGE NO.
6
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         6.998721E+03 -6.998721E+03  2.539927E+03  4.797589E+03  4.798940E+03
 PLATE NO.       9             9             9           109           109
 CASE  NO.       1             1             1             1             1
   *******************END OF ELEMENT FORCES*******************
    95. LOAD LIST 2
    96. PRINT ELEMENT JOINT STRESSES LIST 9
  ELEMENT  JOINT    STRESSES LIST
     :A CIRCULAR PLATE-FIXED ALONG ITS PERIMETER              -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX           MY           MXY
                      VONT        VONB         SX           SY           SXY
                      TRESCAT     TRESCAB
     9      2         0.00      -34.32       75.98      484.41        0.01
                    2707.43     2707.43        0.00        0.00        0.00
                    2906.43     2906.43
      TOP : SMAX=    2906.43 SMIN=     455.87 TMAX=    1225.28 ANGLE= 90.0
      BOTT: SMAX=    -455.87 SMIN=   -2906.43 TMAX=    1225.28 ANGLE= -0.0
        JOINT         4.34      -37.38      188.37      616.60       52.05
           9       3328.03     3328.03        0.00        0.00        0.00
      TOP : SMAX=    3737.03 SMIN=    1092.82 TMAX=    1322.10 ANGLE= 83.2
      BOTT: SMAX=   -1092.82 SMIN=   -3737.03 TMAX=    1322.10 ANGLE= -6.8
        JOINT        -4.33      -37.39      188.41      616.69      -52.04
          10       3328.46     3328.46        0.00        0.00        0.00
      TOP : SMAX=    3737.54 SMIN=    1093.08 TMAX=    1322.23 ANGLE=-83.2
      BOTT: SMAX=   -1093.08 SMIN=   -3737.54 TMAX=    1322.23 ANGLE=  6.8
        JOINT        -3.25      -31.25      -36.94      382.08      -70.33
          28       2519.30     2519.30        0.00        0.00        0.00
      TOP : SMAX=    2361.41 SMIN=    -290.59 TMAX=    1326.00 ANGLE=-80.7
      BOTT: SMAX=     290.59 SMIN=   -2361.41 TMAX=    1326.00 ANGLE=  9.3
        JOINT         3.25      -31.24      -36.95      382.09       70.36
          27       2519.50     2519.50        0.00        0.00        0.00
      TOP : SMAX=    2361.54 SMIN=    -290.69 TMAX=    1326.12 ANGLE= 80.7
      BOTT: SMAX=     290.69 SMIN=   -2361.54 TMAX=    1326.12 ANGLE= -9.3
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         3.737542E+03 -3.737542E+03  1.326119E+03  2.707434E+03  2.906431E+03
 PLATE NO.       9             9             9             9             9
 CASE  NO.       2             2             2             2             2
```

V. 2D Circular Surface Edge Stress

The objective of this example is to find the displacement at center, and bending stress at the center and at the perimeter of a circular plate fixed at its periphery.

Reference

Young, W. C., *Roark's Formulas for Stress and Strain*, McGraw-Hill Inc., 6th Edition, 1989, Page 429.

Problem

Find the normal stress on the edge of the circular hole for the plate shown, when an in-plane load causes tension. Use a one-quarter, doubly symmetric model.

E =10,000.0 ksi

Radius = 10 in

Thickness = 0.02 in

w = 0.1 psi

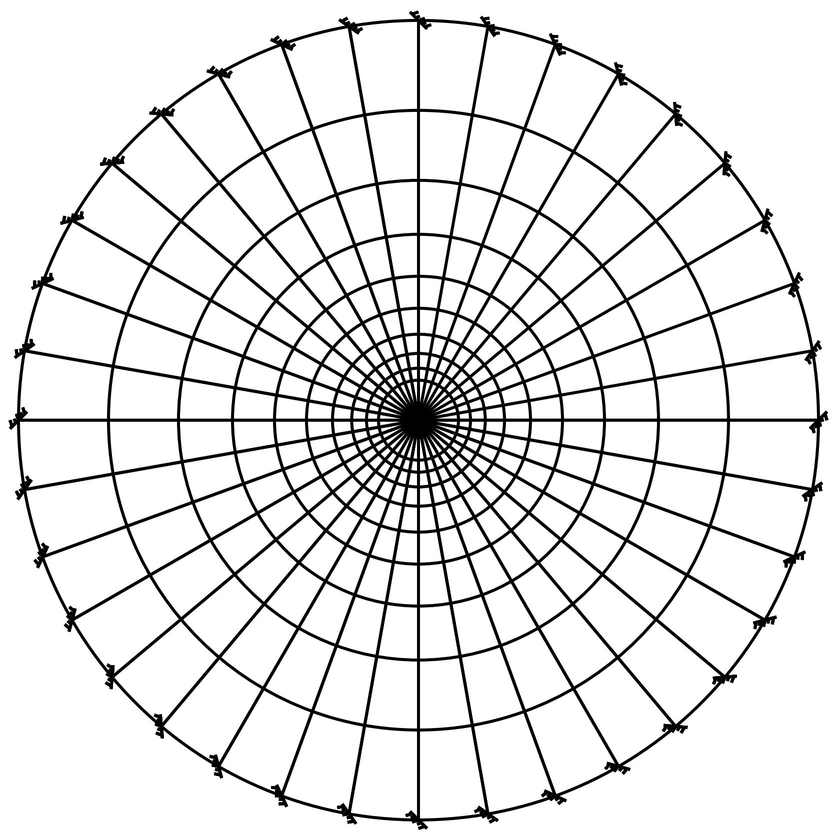


Figure 397: Model

Comparison

Table 421: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
δ at node 1 (in)	2.133	2.16005	1.3%	The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.
Moment at joint 1 (in·lbs)	0.813	0.830	2.6%	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Circular Surface Edge Stress.STD` is typically installed with the program.

```
STAAD SPACE :UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE:  "ROARK'S FORMULAS FOR STRESS AND STRAIN", WARREN C. YOUNG,
*             SIXTH EDITION, MCGRAW-HILL, PAGE 429.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 0.984808 0 -0.173648; 4 0.939693 0 -0.34202;
5 0.866025 0 -0.5; 6 0.766044 0 -0.642788; 7 0.642787 0 -0.766045;
8 0.5 0 -0.866026; 9 0.34202 0 -0.939693; 10 0.173648 0 -0.984808;
11 -6.39758e-07 0 -1; 12 -0.173649 0 -0.984808;
13 -0.342021 0 -0.939692; 14 -0.500001 0 -0.866025;
15 -0.642788 0 -0.766044; 16 -0.766045 0 -0.642787;
17 -0.866026 0 -0.499999; 18 -0.939693 0 -0.342019;
19 -0.984808 0 -0.173647; 20 -1 0 1.27952e-06; 21 -0.984807 0 0.17365;
22 -0.939692 0 0.342021; 23 -0.866025 0 0.500001;
24 -0.766043 0 0.642789; 25 -0.642786 0 0.766045;
26 -0.499998 0 0.866026; 27 -0.342018 0 0.939693;
28 -0.173646 0 0.984808; 29 1.91927e-06 0 1; 30 0.17365 0 0.984807;
31 0.342022 0 0.939692; 32 0.500002 0 0.866024; 33 0.642789 0 0.766043;
34 0.766046 0 0.642786; 35 0.866027 0 0.499998; 36 0.939694 0 0.342018;
37 0.984808 0 0.173646; 38 1.2916 0 0; 39 1.27198 0 -0.224284;
40 1.21371 0 -0.441753; 41 1.11856 0 -0.6458; 42 0.989423 0 -0.830225;
43 0.830224 0 -0.989423; 44 0.645799 0 -1.11856; 45 0.441753 0 -1.21371;
46 0.224283 0 -1.27198; 47 -8.26311e-07 0 -1.2916;
48 -0.224285 0 -1.27198; 49 -0.441754 0 -1.21371;
```

```
50 -0.645801 0 -1.11856; 51 -0.830225 0 -0.989422;
52 -0.989424 0 -0.830224; 53 -1.11856 0 -0.645799;
54 -1.21371 0 -0.441752; 55 -1.27198 0 -0.224282;
56 -1.2916 0 1.65262e-06; 57 -1.27198 0 0.224286;
58 -1.21371 0 0.441755; 59 -1.11856 0 0.645802; 60 -0.989422 0 0.830226;
61 -0.830223 0 0.989424; 62 -0.645798 0 1.11856; 63 -0.441751 0 1.21371;
64 -0.224282 0 1.27198; 65 2.47893e-06 0 1.2916; 66 0.224286 0 1.27198;
67 0.441756 0 1.21371; 68 0.645803 0 1.11856; 69 0.830227 0 0.989421;
70 0.989425 0 0.830222; 71 1.11856 0 0.645797; 72 1.21371 0 0.44175;
73 1.27198 0 0.224281; 74 1.6681 0 0; 75 1.64276 0 -0.289663;
76 1.5675 0 -0.570524; 77 1.44462 0 -0.83405; 78 1.27784 0 -1.07223;
79 1.07223 0 -1.27784; 80 0.834049 0 -1.44462; 81 0.570523 0 -1.5675;
82 0.289662 0 -1.64276; 83 -1.06718e-06 0 -1.6681;
84 -0.289664 0 -1.64276; 85 -0.570525 0 -1.5675;
86 -0.834051 0 -1.44462; 87 -1.07224 0 -1.27784; 88 -1.27784 0 -1.07223;
89 -1.44462 0 -0.834048; 90 -1.5675 0 -0.570522; 91 -1.64276 0 -0.28966;
92 -1.6681 0 2.13436e-06; 93 -1.64276 0 0.289665; 94 -1.5675 0 0.570526;
95 -1.44462 0 0.834052; 96 -1.27784 0 1.07224; 97 -1.07223 0 1.27784;
98 -0.834047 0 1.44462; 99 -0.570521 0 1.5675; 100 -0.289659 0 1.64276;
101 3.20154e-06 0 1.6681; 102 0.289666 0 1.64276; 103 0.570527 0 1.5675;
104 0.834053 0 1.44462; 105 1.07224 0 1.27784; 106 1.27784 0 1.07223;
107 1.44462 0 0.834046; 108 1.5675 0 0.57052; 109 1.64276 0 0.289658;
110 2.1544 0 0; 111 2.12167 0 -0.374108; 112 2.02447 0 -0.736849;
113 1.86576 0 -1.0772; 114 1.65037 0 -1.38482; 115 1.38482 0 -1.65037;
116 1.0772 0 -1.86577; 117 0.736847 0 -2.02447; 118 0.374106 0 -2.12167;
119 -1.37829e-06 0 -2.1544; 120 -0.374109 0 -2.12167;
121 -0.73685 0 -2.02447; 122 -1.0772 0 -1.86576;
123 -1.38482 0 -1.65036; 124 -1.65037 0 -1.38482;
125 -1.86577 0 -1.0772; 126 -2.02447 0 -0.736846;
127 -2.12167 0 -0.374105; 128 -2.1544 0 2.75659e-06;
129 -2.12167 0 0.374111; 130 -2.02447 0 0.736851; 131 -1.86576 0 1.0772;
132 -1.65036 0 1.38482; 133 -1.38482 0 1.65037; 134 -1.0772 0 1.86577;
135 -0.736844 0 2.02448; 136 -0.374104 0 2.12167;
137 4.13488e-06 0 2.1544; 138 0.374112 0 2.12167;
139 0.736853 0 2.02447; 140 1.0772 0 1.86576; 141 1.38483 0 1.65036;
142 1.65037 0 1.38482; 143 1.86577 0 1.0772; 144 2.02448 0 0.736843;
145 2.12167 0 0.374102; 146 2.7826 0 0; 147 2.74033 0 -0.483194;
148 2.61479 0 -0.951706; 149 2.4098 0 -1.3913; 150 2.13159 0 -1.78862;
151 1.78862 0 -2.1316; 152 1.3913 0 -2.4098; 153 0.951704 0 -2.61479;
154 0.483192 0 -2.74033; 155 -1.78019e-06 0 -2.7826;
156 -0.483195 0 -2.74033; 157 -0.951707 0 -2.61479;
158 -1.3913 0 -2.4098; 159 -1.78862 0 -2.13159; 160 -2.1316 0 -1.78862;
161 -2.4098 0 -1.3913; 162 -2.61479 0 -0.951702;
163 -2.74033 0 -0.48319; 164 -2.7826 0 3.56038e-06;
165 -2.74033 0 0.483197; 166 -2.61479 0 0.951709; 167 -2.4098 0 1.3913;
168 -2.13159 0 1.78862; 169 -1.78862 0 2.1316; 170 -1.3913 0 2.40981;
171 -0.9517 0 2.61479; 172 -0.483188 0 2.74033;
173 5.34057e-06 0 2.7826; 174 0.483199 0 2.74032;
175 0.951711 0 2.61479; 176 1.39131 0 2.4098; 177 1.78863 0 2.13159;
178 2.1316 0 1.78862; 179 2.40981 0 1.39129; 180 2.61479 0 0.951698;
181 2.74033 0 0.483186; 182 3.5938 0 0; 183 3.5392 0 -0.624057;
184 3.37707 0 -1.22915; 185 3.11232 0 -1.7969; 186 2.75301 0 -2.31005;
187 2.31005 0 -2.75301; 188 1.7969 0 -3.11232; 189 1.22915 0 -3.37707;
190 0.624055 0 -3.5392; 191 -2.29916e-06 0 -3.5938;
192 -0.624059 0 -3.5392; 193 -1.22915 0 -3.37707;
194 -1.7969 0 -3.11232; 195 -2.31005 0 -2.75301;
196 -2.75301 0 -2.31005; 197 -3.11232 0 -1.7969;
198 -3.37707 0 -1.22915; 199 -3.5392 0 -0.624052;
```

```
200 -3.5938 0 4.59832e-06; 201 -3.5392 0 0.624062;
202 -3.37707 0 1.22916; 203 -3.11232 0 1.79691; 204 -2.75301 0 2.31005;
205 -2.31005 0 2.75301; 206 -1.79689 0 3.11233; 207 -1.22915 0 3.37707;
208 -0.62405 0 3.5392; 209 6.89749e-06 0 3.5938; 210 0.624063 0 3.5392;
211 1.22916 0 3.37706; 212 1.79691 0 3.11232; 213 2.31006 0 2.75301;
214 2.75302 0 2.31004; 215 3.11233 0 1.79689; 216 3.37707 0 1.22914;
217 3.5392 0 0.624048; 218 4.6416 0 0; 219 4.57108 0 -0.806006;
220 4.36168 0 -1.58752; 221 4.01974 0 -2.3208; 222 3.55567 0 -2.98356;
223 2.98356 0 -3.55567; 224 2.3208 0 -4.01974; 225 1.58752 0 -4.36168;
226 0.806003 0 -4.57108; 227 -2.9695e-06 0 -4.6416;
228 -0.806009 0 -4.57108; 229 -1.58752 0 -4.36168;
230 -2.3208 0 -4.01974; 231 -2.98357 0 -3.55567;
232 -3.55567 0 -2.98356; 233 -4.01975 0 -2.3208;
234 -4.36168 0 -1.58752; 235 -4.57108 0 -0.805999;
236 -4.6416 0 5.939e-06; 237 -4.57108 0 0.806012;
238 -4.36168 0 1.58753; 239 -4.01974 0 2.32081; 240 -3.55567 0 2.98357;
241 -2.98356 0 3.55568; 242 -2.32079 0 4.01975; 243 -1.58751 0 4.36168;
244 -0.805996 0 4.57109; 245 8.9085e-06 0 4.6416;
246 0.806014 0 4.57108; 247 1.58753 0 4.36167; 248 2.32081 0 4.01974;
249 2.98357 0 3.55567; 250 3.55568 0 2.98356; 251 4.01975 0 2.32079;
252 4.36168 0 1.58751; 253 4.57109 0 0.805994; 254 5.9948 0 0;
255 5.90373 0 -1.04099; 256 5.63327 0 -2.05034; 257 5.19165 0 -2.9974;
258 4.59228 0 -3.85338; 259 3.85338 0 -4.59228; 260 2.9974 0 -5.19165;
261 2.05034 0 -5.63327; 262 1.04098 0 -5.90373;
263 -3.83522e-06 0 -5.9948; 264 -1.04099 0 -5.90372;
265 -2.05035 0 -5.63327; 266 -2.99741 0 -5.19165;
267 -3.85339 0 -4.59228; 268 -4.59229 0 -3.85338;
269 -5.19165 0 -2.99739; 270 -5.63327 0 -2.05034;
271 -5.90373 0 -1.04098; 272 -5.9948 0 7.67044e-06;
273 -5.90372 0 1.04099; 274 -5.63327 0 2.05035; 275 -5.19164 0 2.99741;
276 -4.59228 0 3.85339; 277 -3.85338 0 4.59229; 278 -2.99739 0 5.19166;
279 -2.05033 0 5.63327; 280 -1.04097 0 5.90373;
281 1.15057e-05 0 5.9948; 282 1.041 0 5.90372; 283 2.05036 0 5.63326;
284 2.99741 0 5.19164; 285 3.85339 0 4.59227; 286 4.59229 0 3.85337;
287 5.19166 0 2.99739; 288 5.63327 0 2.05033; 289 5.90373 0 1.04097;
290 7.7426 0 0; 291 7.62497 0 -1.34449; 292 7.27566 0 -2.64813;
293 6.70529 0 -3.8713; 294 5.93117 0 -4.97685; 295 4.97685 0 -5.93118;
296 3.8713 0 -6.70529; 297 2.64812 0 -7.27567; 298 1.34448 0 -7.62497;
299 -4.95339e-06 0 -7.7426; 300 -1.34449 0 -7.62497;
301 -2.64813 0 -7.27566; 302 -3.87131 0 -6.70528;
303 -4.97685 0 -5.93117; 304 -5.93118 0 -4.97684;
305 -6.70529 0 -3.87129; 306 -7.27567 0 -2.64812;
307 -7.62497 0 -1.34448; 308 -7.7426 0 9.90678e-06;
309 -7.62497 0 1.3445; 310 -7.27566 0 2.64814; 311 -6.70528 0 3.87131;
312 -5.93117 0 4.97686; 313 -4.97684 0 5.93118; 314 -3.87129 0 6.7053;
315 -2.64811 0 7.27567; 316 -1.34447 0 7.62498;
317 1.48602e-05 0 7.7426; 318 1.3445 0 7.62497; 319 2.64814 0 7.27566;
320 3.87132 0 6.70528; 321 4.97686 0 5.93116; 322 5.93119 0 4.97683;
323 6.7053 0 3.87128; 324 7.27567 0 2.64811; 325 7.62498 0 1.34447;
326 10 0 0; 327 9.84808 0 -1.73648; 328 9.39693 0 -3.4202;
329 8.66025 0 -5; 330 7.66044 0 -6.42788; 331 6.42787 0 -7.66045;
332 5 0 -8.66026; 333 3.4202 0 -9.39693; 334 1.73648 0 -9.84808;
335 -6.39758e-06 0 -10; 336 -1.73649 0 -9.84808;
337 -3.42021 0 -9.39692; 338 -5.00001 0 -8.66025;
339 -6.42788 0 -7.66044; 340 -7.66045 0 -6.42787;
341 -8.66026 0 -4.99999; 342 -9.39693 0 -3.42019;
343 -9.84808 0 -1.73647; 344 -10 0 1.27952e-05; 345 -9.84807 0 1.7365;
346 -9.39692 0 3.42021; 347 -8.66025 0 5.00001; 348 -7.66043 0 6.42789;
```

```
349 -6.42786 0 7.66045; 350 -4.99998 0 8.66026; 351 -3.42018 0 9.39693;
352 -1.73646 0 9.84808; 353 1.91927e-05 0 10; 354 1.7365 0 9.84807;
355 3.42022 0 9.39692; 356 5.00002 0 8.66024; 357 6.42789 0 7.66043;
358 7.66046 0 6.42786; 359 8.66027 0 4.99998; 360 9.39694 0 3.42018;
361 9.84808 0 1.73646;
ELEMENT INCIDENCES SHELL
1 1 2 3; 2 1 3 4; 3 1 4 5; 4 1 5 6; 5 1 6 7; 6 1 7 8; 7 1 8 9; 8 1 9 10;
9 1 10 11; 10 1 11 12; 11 1 12 13; 12 1 13 14; 13 1 14 15; 14 1 15 16;
15 1 16 17; 16 1 17 18; 17 1 18 19; 18 1 19 20; 19 1 20 21; 20 1 21 22;
21 1 22 23; 22 1 23 24; 23 1 24 25; 24 1 25 26; 25 1 26 27; 26 1 27 28;
27 1 28 29; 28 1 29 30; 29 1 30 31; 30 1 31 32; 31 1 32 33; 32 1 33 34;
33 1 34 35; 34 1 35 36; 35 1 36 37; 36 1 37 2; 37 2 38 39 3;
38 3 39 40 4; 39 4 40 41 5; 40 5 41 42 6; 41 6 42 43 7; 42 7 43 44 8;
43 8 44 45 9; 44 9 45 46 10; 45 10 46 47 11; 46 11 47 48 12;
47 12 48 49 13; 48 13 49 50 14; 49 14 50 51 15; 50 15 51 52 16;
51 16 52 53 17; 52 17 53 54 18; 53 18 54 55 19; 54 19 55 56 20;
55 20 56 57 21; 56 21 57 58 22; 57 22 58 59 23; 58 23 59 60 24;
59 24 60 61 25; 60 25 61 62 26; 61 26 62 63 27; 62 27 63 64 28;
63 28 64 65 29; 64 29 65 66 30; 65 30 66 67 31; 66 31 67 68 32;
67 32 68 69 33; 68 33 69 70 34; 69 34 70 71 35; 70 35 71 72 36;
71 36 72 73 37; 72 37 73 38 2; 73 38 74 75 39; 74 39 75 76 40;
75 40 76 77 41; 76 41 77 78 42; 77 42 78 79 43; 78 43 79 80 44;
79 44 80 81 45; 80 45 81 82 46; 81 46 82 83 47; 82 47 83 84 48;
83 48 84 85 49; 84 49 85 86 50; 85 50 86 87 51; 86 51 87 88 52;
87 52 88 89 53; 88 53 89 90 54; 89 54 90 91 55; 90 55 91 92 56;
91 56 92 93 57; 92 57 93 94 58; 93 58 94 95 59; 94 59 95 96 60;
95 60 96 97 61; 96 61 97 98 62; 97 62 98 99 63; 98 63 99 100 64;
99 64 100 101 65; 100 65 101 102 66; 101 66 102 103 67;
102 67 103 104 68; 103 68 104 105 69; 104 69 105 106 70;
105 70 106 107 71; 106 71 107 108 72; 107 72 108 109 73;
108 73 109 74 38; 109 74 110 111 75; 110 75 111 112 76;
111 76 112 113 77; 112 77 113 114 78; 113 78 114 115 79;
114 79 115 116 80; 115 80 116 117 81; 116 81 117 118 82;
117 82 118 119 83; 118 83 119 120 84; 119 84 120 121 85;
120 85 121 122 86; 121 86 122 123 87; 122 87 123 124 88;
123 88 124 125 89; 124 89 125 126 90; 125 90 126 127 91;
126 91 127 128 92; 127 92 128 129 93; 128 93 129 130 94;
129 94 130 131 95; 130 95 131 132 96; 131 96 132 133 97;
132 97 133 134 98; 133 98 134 135 99; 134 99 135 136 100;
135 100 136 137 101; 136 101 137 138 102; 137 102 138 139 103;
138 103 139 140 104; 139 104 140 141 105; 140 105 141 142 106;
141 106 142 143 107; 142 107 143 144 108; 143 108 144 145 109;
144 109 145 110 74; 145 110 146 147 111; 146 111 147 148 112;
147 112 148 149 113; 148 113 149 150 114; 149 114 150 151 115;
150 115 151 152 116; 151 116 152 153 117; 152 117 153 154 118;
153 118 154 155 119; 154 119 155 156 120; 155 120 156 157 121;
156 121 157 158 122; 157 122 158 159 123; 158 123 159 160 124;
159 124 160 161 125; 160 125 161 162 126; 161 126 162 163 127;
162 127 163 164 128; 163 128 164 165 129; 164 129 165 166 130;
165 130 166 167 131; 166 131 167 168 132; 167 132 168 169 133;
168 133 169 170 134; 169 134 170 171 135; 170 135 171 172 136;
171 136 172 173 137; 172 137 173 174 138; 173 138 174 175 139;
174 139 175 176 140; 175 140 176 177 141; 176 141 177 178 142;
177 142 178 179 143; 178 143 179 180 144; 179 144 180 181 145;
180 145 181 146 110; 181 146 182 183 147; 182 147 183 184 148;
183 148 184 185 149; 184 149 185 186 150; 185 150 186 187 151;
186 151 187 188 152; 187 152 188 189 153; 188 153 189 190 154;
189 154 190 191 155; 190 155 191 192 156; 191 156 192 193 157;
```

```
192 157 193 194 158; 193 158 194 195 159; 194 159 195 196 160;
195 160 196 197 161; 196 161 197 198 162; 197 162 198 199 163;
198 163 199 200 164; 199 164 200 201 165; 200 165 201 202 166;
201 166 202 203 167; 202 167 203 204 168; 203 168 204 205 169;
204 169 205 206 170; 205 170 206 207 171; 206 171 207 208 172;
207 172 208 209 173; 208 173 209 210 174; 209 174 210 211 175;
210 175 211 212 176; 211 176 212 213 177; 212 177 213 214 178;
213 178 214 215 179; 214 179 215 216 180; 215 180 216 217 181;
216 181 217 182 146; 217 182 218 219 183; 218 183 219 220 184;
219 184 220 221 185; 220 185 221 222 186; 221 186 222 223 187;
222 187 223 224 188; 223 188 224 225 189; 224 189 225 226 190;
225 190 226 227 191; 226 191 227 228 192; 227 192 228 229 193;
228 193 229 230 194; 229 194 230 231 195; 230 195 231 232 196;
231 196 232 233 197; 232 197 233 234 198; 233 198 234 235 199;
234 199 235 236 200; 235 200 236 237 201; 236 201 237 238 202;
237 202 238 239 203; 238 203 239 240 204; 239 204 240 241 205;
240 205 241 242 206; 241 206 242 243 207; 242 207 243 244 208;
243 208 244 245 209; 244 209 245 246 210; 245 210 246 247 211;
246 211 247 248 212; 247 212 248 249 213; 248 213 249 250 214;
249 214 250 251 215; 250 215 251 252 216; 251 216 252 253 217;
252 217 253 218 182; 253 218 254 255 219; 254 219 255 256 220;
255 220 256 257 221; 256 221 257 258 222; 257 222 258 259 223;
258 223 259 260 224; 259 224 260 261 225; 260 225 261 262 226;
261 226 262 263 227; 262 227 263 264 228; 263 228 264 265 229;
264 229 265 266 230; 265 230 266 267 231; 266 231 267 268 232;
267 232 268 269 233; 268 233 269 270 234; 269 234 270 271 235;
270 235 271 272 236; 271 236 272 273 237; 272 237 273 274 238;
273 238 274 275 239; 274 239 275 276 240; 275 240 276 277 241;
276 241 277 278 242; 277 242 278 279 243; 278 243 279 280 244;
279 244 280 281 245; 280 245 281 282 246; 281 246 282 283 247;
282 247 283 284 248; 283 248 284 285 249; 284 249 285 286 250;
285 250 286 287 251; 286 251 287 288 252; 287 252 288 289 253;
288 253 289 254 218; 289 254 290 291 255; 290 255 291 292 256;
291 256 292 293 257; 292 257 293 294 258; 293 258 294 295 259;
294 259 295 296 260; 295 260 296 297 261; 296 261 297 298 262;
297 262 298 299 263; 298 263 299 300 264; 299 264 300 301 265;
300 265 301 302 266; 301 266 302 303 267; 302 267 303 304 268;
303 268 304 305 269; 304 269 305 306 270; 305 270 306 307 271;
306 271 307 308 272; 307 272 308 309 273; 308 273 309 310 274;
309 274 310 311 275; 310 275 311 312 276; 311 276 312 313 277;
312 277 313 314 278; 313 278 314 315 279; 314 279 315 316 280;
315 280 316 317 281; 316 281 317 318 282; 317 282 318 319 283;
318 283 319 320 284; 319 284 320 321 285; 320 285 321 322 286;
321 286 322 323 287; 322 287 323 324 288; 323 288 324 325 289;
324 289 325 290 254; 325 290 326 327 291; 326 291 327 328 292;
327 292 328 329 293; 328 293 329 330 294; 329 294 330 331 295;
330 295 331 332 296; 331 296 332 333 297; 332 297 333 334 298;
333 298 334 335 299; 334 299 335 336 300; 335 300 336 337 301;
336 301 337 338 302; 337 302 338 339 303; 338 303 339 340 304;
339 304 340 341 305; 340 305 341 342 306; 341 306 342 343 307;
342 307 343 344 308; 343 308 344 345 309; 344 309 345 346 310;
345 310 346 347 311; 346 311 347 348 312; 347 312 348 349 313;
348 313 349 350 314; 349 314 350 351 315; 350 315 351 352 316;
351 316 352 353 317; 352 317 353 354 318; 353 318 354 355 319;
354 319 355 356 320; 355 320 356 357 321; 356 321 357 358 322;
357 322 358 359 323; 358 323 359 360 324; 359 324 360 361 325;
360 325 361 326 290;
ELEMENT PROPERTY
```

```
1 TO 360 THICKNESS 0.02
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
326 TO 361 FIXED
LOAD 1 UNIFORM PRESSURE
ELEMENT LOAD
1 TO 360 PR 0.1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1
PRINT ELEMENT JOINT STRESSES LIST 1
PRINT SUPPORT REACTION LIST 326
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   2.16005   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
   277. PRINT ELEMENT JOINT STRESSES LIST 1
  ELEMENT  JOINT    STRESSES LIST
      :UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE              -- PAGE NO.
8
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
      1      1        -1.48       -2.18        0.83         0.83         0.00
                   12399.08    12399.08        0.00         0.00         0.00
                   12399.08    12399.08
        TOP : SMAX=   12399.08 SMIN=   12399.07 TMAX=       0.00 ANGLE= 90.0
        BOTT: SMAX=  -12399.07 SMIN=  -12399.08 TMAX=       0.00 ANGLE= 90.0
          JOINT       -1.48       -2.18        0.85         0.83         0.00
            1      12566.42    12566.42        0.00         0.00         0.00
        TOP : SMAX=   12692.08 SMIN=   12436.87 TMAX=     127.61 ANGLE=  0.0
        BOTT: SMAX=  -12436.87 SMIN=  -12692.08 TMAX=     127.61 ANGLE=-90.0
          JOINT       -1.48       -2.18        0.82         0.83         0.00
            2      12344.11    12344.11        0.00         0.00         0.00
        TOP : SMAX=   12436.87 SMIN=   12249.21 TMAX=      93.83 ANGLE= 90.0
        BOTT: SMAX=  -12249.21 SMIN=  -12436.87 TMAX=      93.83 ANGLE= -0.0
          JOINT       -1.48       -2.18        0.82         0.82         0.00
            3      12289.85    12289.85        0.00         0.00         0.00
        TOP : SMAX=   12323.49 SMIN=   12255.94 TMAX=      33.78 ANGLE= 90.0
        BOTT: SMAX=  -12255.94 SMIN=  -12323.49 TMAX=      33.78 ANGLE= -0.0
             **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
                MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
               PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
```

```
            STRESS        STRESS        STRESS        STRESS        STRESS
         1.269208E+04 -1.269208E+04  1.276060E+02  1.239908E+04  1.239908E+04
 PLATE NO.       1             1             1             1             1
 CASE  NO.       1             1             1             1             1
   ********************END OF ELEMENT FORCES********************
   278. PRINT SUPPORT REACTION LIST 326
  SUPPORT  REACTION LIST     326
       :UNIFORM PRESSURE ON A FIXED CIRCULAR PLATE              -- PAGE NO.
9
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    326    1      0.00     -0.87      0.00     -0.01      0.00      2.15
   ************** END OF LATEST ANALYSIS RESULT **************
```

V. 2D Curved Beam Maximum Stress

Using plate/shell elements, find maximum bending stress due to a force couple on a curved cantilever beam.

Reference

Timoshenko, S., *Strength of Materials*, Part I, 3rd Edition, Van Nostrand Co., 1956.

Problem

Find the maximum bending stress

E = 3,000.0 ksi.

Poisson's ratio = 0.3.

t = 1.0 in.

P = 100 lbs

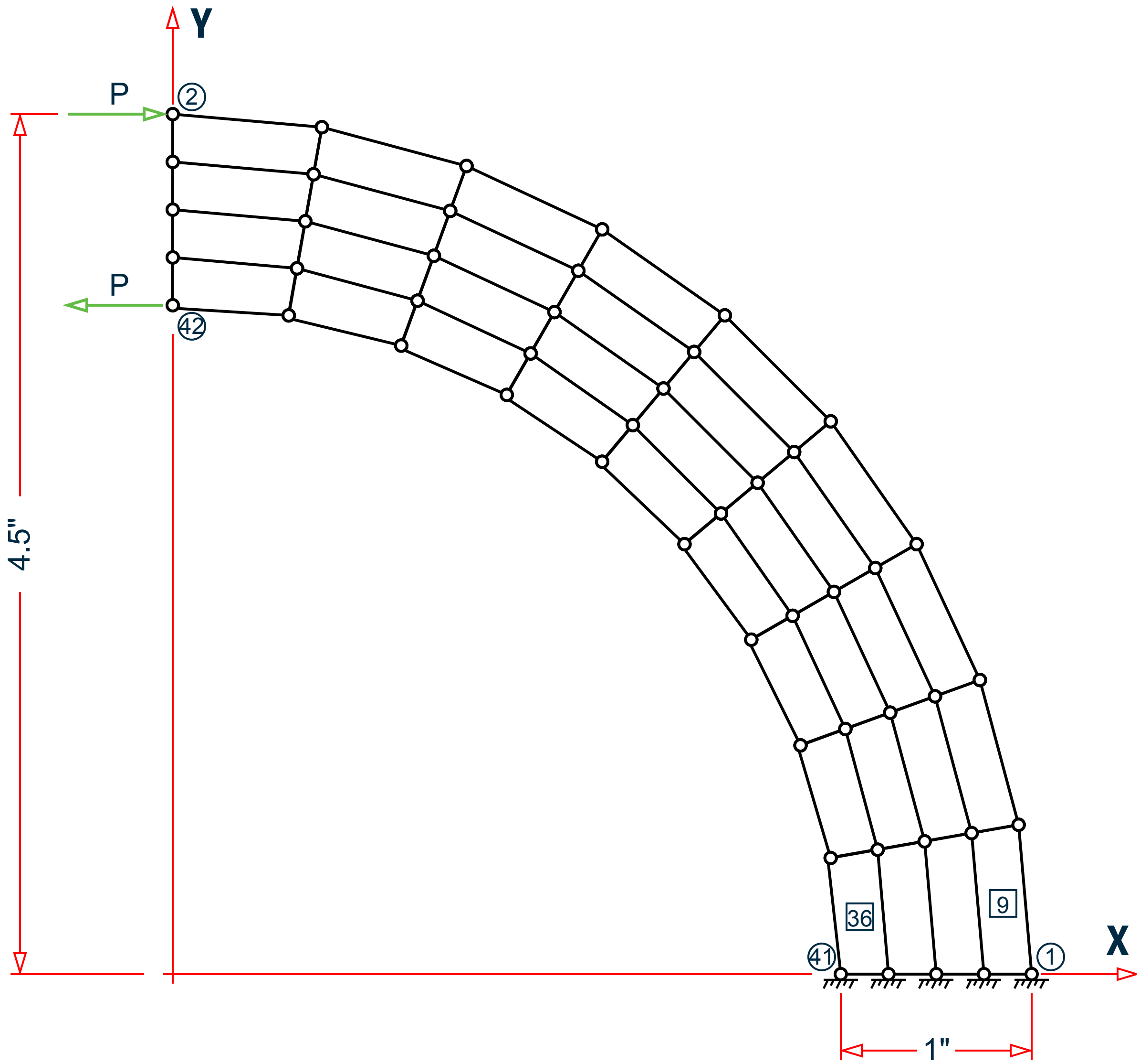


Figure 398: Cantilevered, curved plate with coupling load at free end

Comparison

Table 422: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
Inside stress (psi)	655.0	636.02	2.9%	The result from the Beam theory is compared with the result from a Finite Element Model output, hence the difference in results.

Result Type	Theory	STAAD.Pro	Difference	Comments
Outside stress (psi)	555.0	556.73	negligible	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Curved Beam Maximum Stress.STD` is typically installed with the program.

```
STAAD SPACE :CURVED BEAM WITH PLATE ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE: TIMOSHENKO, S., "STENGTH OF MATERIALS, PART I, ELEMENTARY
*             THEORY AND PROBLEMS", 3RD EDITION, D. VAN NOSTRAND CO.,
*             INC., NEW YORK, 1956.
*
UNIT INCHES POUND
JOINT COORDINATES
1 4.5 0 0; 2 0 4.5 0; 3 0.781 4.432 0; 4 1.539 4.229 0; 5 2.25 3.897 0;
6 2.893 3.447 0; 7 3.447 2.893 0; 8 3.897 2.25 0; 9 4.229 1.539 0;
10 4.432 0.781 0; 11 4.25 0 0; 12 0 4.25 0; 13 0.738 4.185 0;
14 1.454 3.994 0; 15 2.125 3.681 0; 16 2.732 3.256 0; 17 3.256 2.732 0;
18 3.681 2.125 0; 19 3.994 1.454 0; 20 4.185 0.738 0; 21 4 0 0;
22 0 4 0; 23 0.695 3.939 0; 24 1.368 3.759 0; 25 2 3.464 0;
26 2.571 3.064 0; 27 3.064 2.571 0; 28 3.464 2 0; 29 3.759 1.368 0;
30 3.939 0.695 0; 31 3.75 0 0; 32 0 3.75 0; 33 0.651 3.693 0;
34 1.283 3.524 0; 35 1.875 3.248 0; 36 2.41 2.873 0; 37 2.873 2.41 0;
38 3.248 1.875 0; 39 3.524 1.283 0; 40 3.693 0.651 0; 41 3.5 0 0;
42 0 3.5 0; 43 0.608 3.447 0; 44 1.197 3.289 0; 45 1.75 3.031 0;
46 2.25 2.681 0; 47 2.681 2.25 0; 48 3.031 1.75 0; 49 3.289 1.197 0;
50 3.447 0.608 0;
ELEMENT INCIDENCES SHELL
1 2 3 13 12; 2 3 4 14 13; 3 4 5 15 14; 4 5 6 16 15; 5 6 7 17 16;
6 7 8 18 17; 7 8 9 19 18; 8 9 10 20 19; 9 10 1 11 20; 10 12 13 23 22;
11 13 14 24 23; 12 14 15 25 24; 13 15 16 26 25; 14 16 17 27 26;
15 17 18 28 27; 16 18 19 29 28; 17 19 20 30 29; 18 20 11 21 30;
19 22 23 33 32; 20 23 24 34 33; 21 24 25 35 34; 22 25 26 36 35;
23 26 27 37 36; 24 27 28 38 37; 25 28 29 39 38; 26 29 30 40 39;
27 30 21 31 40; 28 32 33 43 42; 29 33 34 44 43; 30 34 35 45 44;
31 35 36 46 45; 32 36 37 47 46; 33 37 38 48 47; 34 38 39 49 48;
35 39 40 50 49; 36 40 31 41 50;
ELEMENT PROPERTY
1 TO 36 THICKNESS 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
```

```
SUPPORTS
1 11 21 31 41 FIXED
LOAD 1 100 IN-IB
JOINT LOAD
2 FX 100
42 FX -100
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT ELEMENT JOINT STRESSES LIST 9 36
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -15.30     61.91      0.00      0.00      0.00     -2.09
     11    1      3.80     69.53      0.00      0.00      0.00      0.72
     21    1     12.47      5.36      0.00      0.00      0.00      1.21
     31    1     11.58    -66.95      0.00      0.00      0.00      1.75
     41    1    -12.55    -69.85      0.00      0.00      0.00     -1.59
   ************** END OF LATEST ANALYSIS RESULT **************
    52. PRINT ELEMENT JOINT STRESSES LIST 9 36
  ELEMENT  JOINT    STRESSES LIST
      :CURVED BEAM WITH PLATE ELEMENTS                         -- PAGE NO.
4
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
     9      1         0.00        0.00        0.00         0.00         0.00
                    432.15      432.15     -433.11        -2.49         8.94
                    433.30      433.30
      TOP : SMAX=      -2.31 SMIN=    -433.30 TMAX=     215.50 ANGLE= 88.8
      BOTT: SMAX=      -2.31 SMIN=    -433.30 TMAX=     215.50 ANGLE= 88.8
        JOINT         0.00        0.00        0.00         0.00         0.00
          10        592.15      592.15     -560.35        58.12        15.37
      TOP : SMAX=      58.50 SMIN=    -560.73 TMAX=     309.62 ANGLE= 88.6
      BOTT: SMAX=      58.50 SMIN=    -560.73 TMAX=     309.62 ANGLE= 88.6
        JOINT         0.00        0.00        0.00         0.00         0.00
           1        527.90      527.90     -556.47       -63.65        11.40
      TOP : SMAX=     -63.39 SMIN=    -556.73 TMAX=     246.67 ANGLE= 88.7
      BOTT: SMAX=     -63.39 SMIN=    -556.73 TMAX=     246.67 ANGLE= 88.7
        JOINT         0.00        0.00        0.00         0.00         0.00
          11        285.05      285.05     -310.91       -64.85        13.47
      TOP : SMAX=     -64.11 SMIN=    -311.65 TMAX=     123.77 ANGLE= 86.9
      BOTT: SMAX=     -64.11 SMIN=    -311.65 TMAX=     123.77 ANGLE= 86.9
        JOINT         0.00        0.00        0.00         0.00         0.00
          20        351.99      351.99     -318.48        58.96         8.48
      TOP : SMAX=      59.15 SMIN=    -318.67 TMAX=     188.91 ANGLE= 88.7
      BOTT: SMAX=      59.15 SMIN=    -318.67 TMAX=     188.91 ANGLE= 88.7
    36      1         0.00        0.00        0.00         0.00         0.00
                    457.10      457.10      473.43        34.66         2.24
                    473.44      473.44
```

```
       TOP : SMAX=     473.44 SMIN=      34.64 TMAX=     219.40 ANGLE=  0.3
       BOTT: SMAX=     473.44 SMIN=      34.64 TMAX=     219.40 ANGLE=  0.3
         JOINT         0.00        0.00        0.00        0.00        0.00
           40        314.58      314.58      298.80      -24.94      -23.18
       TOP : SMAX=     300.45 SMIN=     -26.59 TMAX=     163.52 ANGLE= -4.1
       BOTT: SMAX=     300.45 SMIN=     -26.59 TMAX=     163.52 ANGLE= -4.1
         JOINT         0.00        0.00        0.00        0.00        0.00
           31        247.25      247.25      282.26      108.74       10.51
       TOP : SMAX=     282.90 SMIN=     108.11 TMAX=      87.39 ANGLE=  3.5
       BOTT: SMAX=     282.90 SMIN=     108.11 TMAX=      87.39 ANGLE=  3.5
         JOINT         0.00        0.00        0.00        0.00        0.00
           41        594.55      594.55      634.80       95.42       25.72
       TOP : SMAX=     636.02 SMIN=      94.20 TMAX=     270.91 ANGLE=  2.7
       BOTT: SMAX=     636.02 SMIN=      94.20 TMAX=     270.91 ANGLE=  2.7
         JOINT         0.00        0.00        0.00        0.00        0.00
           50        680.99      680.99      656.97      -43.84      -21.63
       TOP : SMAX=     657.64 SMIN=     -44.51 TMAX=     351.08 ANGLE= -1.8
       BOTT: SMAX=     657.64 SMIN=     -44.51 TMAX=     351.08 ANGLE= -1.8
      :CURVED BEAM WITH PLATE ELEMENTS                         -- PAGE NO.
5
           **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
              MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
             PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
              STRESS        STRESS        STRESS        STRESS        STRESS
           6.576412E+02 -5.607280E+02  3.510752E+02  4.571006E+02  4.734368E+02
 PLATE NO.      36            9            36            36            36
 CASE  NO.       1            1             1             1             1
```

V. 2D Plate Thermal Moment and Stress

To find the bending moment due to thermal load through the thickness of a square plate.

Reference

Timoshenko, S., *Strength of Materials*, D. Van Nostrand Co., 3rd Edition,1956.

Problem

Temperature varies 100°F linearly through the thickness of a square plate that is fixed on the edges. Calculate the bending moment on the edges and the maximum bending stress.

E = 30,000.0 ksi

$\alpha = 70\times(10)^{-7}$ in/in/°F

Size = 5” × 5”

Thickness = 0.5 in

Poisson’s ratio = 0.3

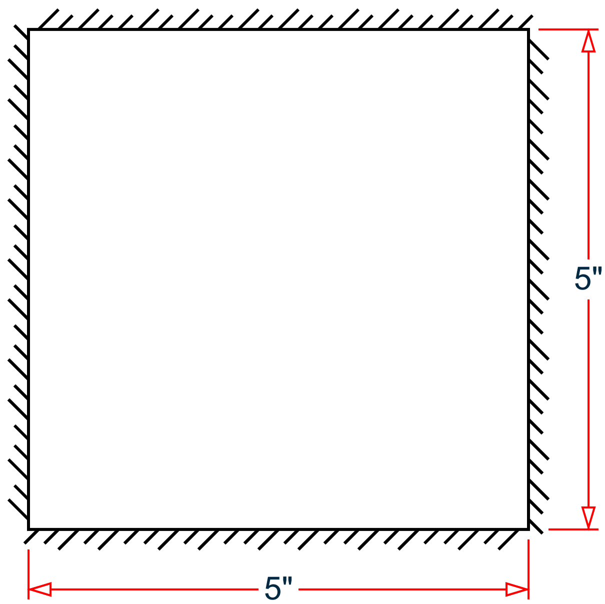


Figure 399: Model

Comparison

Table 423: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Moment (in·lb/in)	625.0	625.0	none
Maximum stress (psi)	15,000	15,000	none

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Plate Thermal Moment and Stress.STD` is typically installed with the program.

```
STAAD SPACE :THERMAL LOADING OF A PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE: TIMOSHENKO, S., "STENGTH OF MATERIALS," 3RD EDITION,
*             D. VAN NOSTRAND CO., 1956.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 5 0 0; 3 0 5 0; 4 5 5 0;
ELEMENT INCIDENCES SHELL
```

```
1 1 2 4 3;
ELEMENT PROPERTY
1 THICKNESS 0.5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
ALPHA 7e-06
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 4 FIXED
LOAD 1 NON UNIFORM HEATING OF THE PLATE
TEMPERATURE LOAD
1 TEMP 0 100
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESSES ALL
FINISH
```

STAAD Output

```
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
      1      1        -0.00       -0.00     -625.00      -625.00         0.00
                   15000.00    15000.00        0.00         0.00         0.00
                   15000.00    15000.00
       TOP : SMAX=  -15000.00 SMIN=  -15000.00 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=   15000.00 SMIN=   15000.00 TMAX=       0.00 ANGLE= 90.0
         JOINT        -0.00       -0.00     -625.00      -625.00         0.00
           1       15000.00    15000.00        0.00         0.00         0.00
       TOP : SMAX=  -15000.00 SMIN=  -15000.00 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=   15000.00 SMIN=   15000.00 TMAX=       0.00 ANGLE= 90.0
         JOINT        -0.00       -0.00     -625.00      -625.00         0.00
           2       15000.00    15000.00        0.00         0.00         0.00
       TOP : SMAX=  -15000.00 SMIN=  -15000.00 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=   15000.00 SMIN=   15000.00 TMAX=       0.00 ANGLE= 90.0
         JOINT        -0.00       -0.00     -625.00      -625.00        -0.00
           4       15000.00    15000.00        0.00         0.00         0.00
       TOP : SMAX=  -15000.00 SMIN=  -15000.00 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=   15000.00 SMIN=   15000.00 TMAX=       0.00 ANGLE= 90.0
         JOINT        -0.00       -0.00     -625.00      -625.00         0.00
           3       15000.00    15000.00        0.00         0.00         0.00
       TOP : SMAX=  -15000.00 SMIN=  -15000.00 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=   15000.00 SMIN=   15000.00 TMAX=       0.00 ANGLE= 90.0
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          1.500000E+04 -1.500000E+04  1.011339E-03  1.500000E+04  1.500000E+04
```

```
PLATE NO.       1             1             1             1             1
CASE  NO.       1             1             1             1             1
```

V. 2D Rectangular Plate with fixed edges

To find the vertical deflection and bending moments at several points due to a unit pressure on a thin rectangular plate simply supported along 4 edges.

Reference

Timoshenko, S. and Woinowsky-Kreiger, S., *Theory of Plates and Shells*, McGraw-Hill, 2nd Edition, 1959, Pages 113-117.

Problem

Loading is unit pressure (1 psi) over entire surface.

$E = 1 \times (10)^6$ psi

Poisson's ratio = 0.3

Length = 16 in.

Width = 10 in.

Thickness = 0.2 in

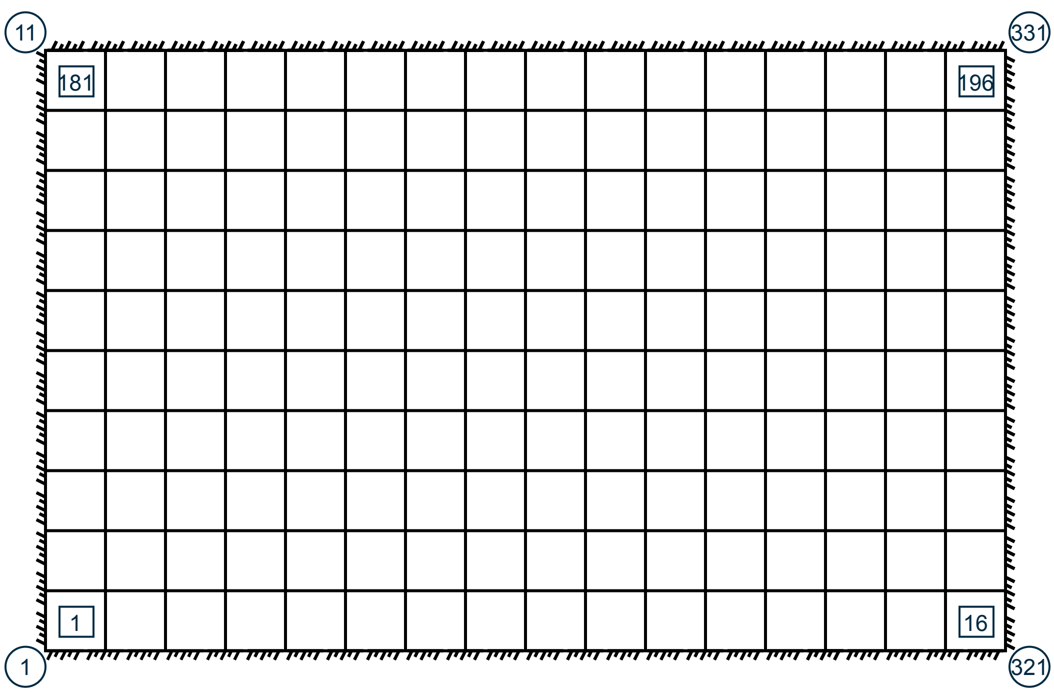


Figure 400: Model

Comparison

Table 424: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Vertical deflection, δ (in) at joint	162	0.036	0.03549	1.2%
	166	0.113	0.11230	1.0%
	306	0.025	0.02478	1.4%
Bending moment, Mx (in·lb) at joint	9	1.763	1.64	7.0%
	89	8.513	8.41	1.2%
	96	1.098	1.05	4.4%
Bending moment, My (in·lb) at joint	9	0.897	0.86	4.1%
	89	4.873	4.81	1.3%
	96	1.108	0.99	10.7%

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Rectangular Plate with fixed edges.std` is typically installed with the program.

```
STAAD SPACE :UNIFORM PRESSURE ON RECTANGULAR PLATE ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* FILE:  2D Rectangular Plate with fixed edges.STD
*
* REFERENCE:  "THEORY OF PLATES AND SHELLS", TIMOSHENKO, S. AND
*             WOINOWSKY-KREIGER, S., SECOND EDITION, MCGRAW-HILL,
*             PAGES 113-117.
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 21 0 0 1; 22 1 0 1; 23 2 0 1; 24 3 0 1;
25 4 0 1; 26 5 0 1; 27 6 0 1; 28 7 0 1; 29 8 0 1; 30 9 0 1; 31 10 0 1;
41 0 0 2; 42 1 0 2; 43 2 0 2; 44 3 0 2; 45 4 0 2; 46 5 0 2; 47 6 0 2;
48 7 0 2; 49 8 0 2; 50 9 0 2; 51 10 0 2; 61 0 0 3; 62 1 0 3; 63 2 0 3;
64 3 0 3; 65 4 0 3; 66 5 0 3; 67 6 0 3; 68 7 0 3; 69 8 0 3; 70 9 0 3;
71 10 0 3; 81 0 0 4; 82 1 0 4; 83 2 0 4; 84 3 0 4; 85 4 0 4; 86 5 0 4;
87 6 0 4; 88 7 0 4; 89 8 0 4; 90 9 0 4; 91 10 0 4; 101 0 0 5; 102 1 0 5;
```

```
103 2 0 5; 104 3 0 5; 105 4 0 5; 106 5 0 5; 107 6 0 5; 108 7 0 5;
109 8 0 5; 110 9 0 5; 111 10 0 5; 121 0 0 6; 122 1 0 6; 123 2 0 6;
124 3 0 6; 125 4 0 6; 126 5 0 6; 127 6 0 6; 128 7 0 6; 129 8 0 6;
130 9 0 6; 131 10 0 6; 141 0 0 7; 142 1 0 7; 143 2 0 7; 144 3 0 7;
145 4 0 7; 146 5 0 7; 147 6 0 7; 148 7 0 7; 149 8 0 7; 150 9 0 7;
151 10 0 7; 161 0 0 8; 162 1 0 8; 163 2 0 8; 164 3 0 8; 165 4 0 8;
166 5 0 8; 167 6 0 8; 168 7 0 8; 169 8 0 8; 170 9 0 8; 171 10 0 8;
181 0 0 9; 182 1 0 9; 183 2 0 9; 184 3 0 9; 185 4 0 9; 186 5 0 9;
187 6 0 9; 188 7 0 9; 189 8 0 9; 190 9 0 9; 191 10 0 9; 201 0 0 10;
202 1 0 10; 203 2 0 10; 204 3 0 10; 205 4 0 10; 206 5 0 10; 207 6 0 10;
208 7 0 10; 209 8 0 10; 210 9 0 10; 211 10 0 10; 221 0 0 11; 222 1 0 11;
223 2 0 11; 224 3 0 11; 225 4 0 11; 226 5 0 11; 227 6 0 11; 228 7 0 11;
229 8 0 11; 230 9 0 11; 231 10 0 11; 241 0 0 12; 242 1 0 12; 243 2 0 12;
244 3 0 12; 245 4 0 12; 246 5 0 12; 247 6 0 12; 248 7 0 12; 249 8 0 12;
250 9 0 12; 251 10 0 12; 261 0 0 13; 262 1 0 13; 263 2 0 13; 264 3 0 13;
265 4 0 13; 266 5 0 13; 267 6 0 13; 268 7 0 13; 269 8 0 13; 270 9 0 13;
271 10 0 13; 281 0 0 14; 282 1 0 14; 283 2 0 14; 284 3 0 14; 285 4 0 14;
286 5 0 14; 287 6 0 14; 288 7 0 14; 289 8 0 14; 290 9 0 14; 291 10 0 14;
301 0 0 15; 302 1 0 15; 303 2 0 15; 304 3 0 15; 305 4 0 15; 306 5 0 15;
307 6 0 15; 308 7 0 15; 309 8 0 15; 310 9 0 15; 311 10 0 15; 321 0 0 16;
322 1 0 16; 323 2 0 16; 324 3 0 16; 325 4 0 16; 326 5 0 16; 327 6 0 16;
328 7 0 16; 329 8 0 16; 330 9 0 16; 331 10 0 16;
ELEMENT INCIDENCES SHELL
1 1 2 22 21; 2 21 22 42 41; 3 41 42 62 61; 4 61 62 82 81;
5 81 82 102 101; 6 101 102 122 121; 7 121 122 142 141;
8 141 142 162 161; 9 161 162 182 181; 10 181 182 202 201;
11 201 202 222 221; 12 221 222 242 241; 13 241 242 262 261;
14 261 262 282 281; 15 281 282 302 301; 16 301 302 322 321;
21 2 3 23 22; 22 22 23 43 42; 23 42 43 63 62; 24 62 63 83 82;
25 82 83 103 102; 26 102 103 123 122; 27 122 123 143 142;
28 142 143 163 162; 29 162 163 183 182; 30 182 183 203 202;
31 202 203 223 222; 32 222 223 243 242; 33 242 243 263 262;
34 262 263 283 282; 35 282 283 303 302; 36 302 303 323 322;
41 3 4 24 23; 42 23 24 44 43; 43 43 44 64 63; 44 63 64 84 83;
45 83 84 104 103; 46 103 104 124 123; 47 123 124 144 143;
48 143 144 164 163; 49 163 164 184 183; 50 183 184 204 203;
51 203 204 224 223; 52 223 224 244 243; 53 243 244 264 263;
54 263 264 284 283; 55 283 284 304 303; 56 303 304 324 323;
61 4 5 25 24; 62 24 25 45 44; 63 44 45 65 64; 64 64 65 85 84;
65 84 85 105 104; 66 104 105 125 124; 67 124 125 145 144;
68 144 145 165 164; 69 164 165 185 184; 70 184 185 205 204;
71 204 205 225 224; 72 224 225 245 244; 73 244 245 265 264;
74 264 265 285 284; 75 284 285 305 304; 76 304 305 325 324;
81 5 6 26 25; 82 25 26 46 45; 83 45 46 66 65; 84 65 66 86 85;
85 85 86 106 105; 86 105 106 126 125; 87 125 126 146 145;
88 145 146 166 165; 89 165 166 186 185; 90 185 186 206 205;
91 205 206 226 225; 92 225 226 246 245; 93 245 246 266 265;
94 265 266 286 285; 95 285 286 306 305; 96 305 306 326 325;
101 6 7 27 26; 102 26 27 47 46; 103 46 47 67 66; 104 66 67 87 86;
105 86 87 107 106; 106 106 107 127 126; 107 126 127 147 146;
108 146 147 167 166; 109 166 167 187 186; 110 186 187 207 206;
111 206 207 227 226; 112 226 227 247 246; 113 246 247 267 266;
114 266 267 287 286; 115 286 287 307 306; 116 306 307 327 326;
121 7 8 28 27; 122 27 28 48 47; 123 47 48 68 67; 124 67 68 88 87;
125 87 88 108 107; 126 107 108 128 127; 127 127 128 148 147;
128 147 148 168 167; 129 167 168 188 187; 130 187 188 208 207;
131 207 208 228 227; 132 227 228 248 247; 133 247 248 268 267;
134 267 268 288 287; 135 287 288 308 307; 136 307 308 328 327;
```

```
141 8 9 29 28; 142 28 29 49 48; 143 48 49 69 68; 144 68 69 89 88;
145 88 89 109 108; 146 108 109 129 128; 147 128 129 149 148;
148 148 149 169 168; 149 168 169 189 188; 150 188 189 209 208;
151 208 209 229 228; 152 228 229 249 248; 153 248 249 269 268;
154 268 269 289 288; 155 288 289 309 308; 156 308 309 329 328;
161 9 10 30 29; 162 29 30 50 49; 163 49 50 70 69; 164 69 70 90 89;
165 89 90 110 109; 166 109 110 130 129; 167 129 130 150 149;
168 149 150 170 169; 169 169 170 190 189; 170 189 190 210 209;
171 209 210 230 229; 172 229 230 250 249; 173 249 250 270 269;
174 269 270 290 289; 175 289 290 310 309; 176 309 310 330 329;
181 10 11 31 30; 182 30 31 51 50; 183 50 51 71 70; 184 70 71 91 90;
185 90 91 111 110; 186 110 111 131 130; 187 130 131 151 150;
188 150 151 171 170; 189 170 171 191 190; 190 190 191 211 210;
191 210 211 231 230; 192 230 231 251 250; 193 250 251 271 270;
194 270 271 291 290; 195 290 291 311 310; 196 310 311 331 330;
ELEMENT PROPERTY
1 TO 16 21 TO 36 41 TO 56 61 TO 76 81 TO 96 THICKNESS 0.2
101 TO 116 121 TO 136 141 TO 156 161 TO 176 181 TO 196 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 11 321 331 FIXED
2 TO 10 322 TO 330 FIXED BUT MX
21 31 41 51 61 71 81 91 101 111 121 131 141 151 161 171 181 191 201 -
211 221 231 241 251 261 271 281 291 301 311 FIXED BUT MZ
LOAD 1 UNIFORM PRESSURE
ELEMENT LOAD
1 TO 16 21 TO 36 41 TO 56 61 TO 76 81 TO 96 PR 1
101 TO 116 121 TO 136 141 TO 156 161 TO 176 181 TO 196 PR 1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 162 166 306
PRINT ELEMENT STRESSES LIST 9 89 96
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    162    1    0.00000  -0.03549   0.00000   0.00000   0.00000  -0.03420
    166    1    0.00000  -0.11230   0.00000   0.00000   0.00000   0.00000
    306    1    0.00000  -0.02478   0.00000  -0.02424   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
   115. PRINT ELEMENT STRESSES LIST 9 89 96
  ELEMENT  STRESSES LIST     9
      :UNIFORM PRESSURE ON RECTANGULAR PLATE ELEMENTS          -- PAGE NO.
5
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY           MX           MY           MXY
```

```
                     VONT        VONB         SX           SY           SXY
                     TRESCAT     TRESCAB
      9      1       19.24        -0.08        1.64         0.86         0.28
                    225.14       225.14        0.00         0.00         0.00
                    259.42       259.42
       TOP : SMAX=     259.42 SMIN=     115.15 TMAX=      72.14 ANGLE= 17.6
       BOTT: SMAX=    -115.15 SMIN=    -259.42 TMAX=      72.14 ANGLE=-72.4
     89      1        1.99        -0.51        8.41         4.81         0.04
                   1096.84      1096.84        0.00         0.00         0.00
                   1262.19      1262.19
       TOP : SMAX=    1262.19 SMIN=     721.80 TMAX=     270.20 ANGLE=  0.7
       BOTT: SMAX=    -721.80 SMIN=   -1262.19 TMAX=     270.20 ANGLE=-89.3
     96      1        0.25       -15.87        1.05         0.99         0.58
                    214.93       214.93        0.00         0.00         0.00
                    240.24       240.24
       TOP : SMAX=     240.24 SMIN=      66.20 TMAX=      87.02 ANGLE= 43.5
       BOTT: SMAX=     -66.20 SMIN=    -240.24 TMAX=      87.02 ANGLE=-46.5
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          1.262189E+03 -1.262189E+03  2.701960E+02  1.096844E+03  1.262189E+03
 PLATE NO.       89            89            89            89            89
 CASE  NO.        1             1             1             1             1
```

V. 2D Retaining Wall

To find the bending moments at various points on a wall fixed along three edges.

Reference

Young, W. C., *Roark's Formulas for Stress and Strain*, McGraw-Hill Inc., 6th Edition, 1989.

Problem

Find the bending moment at the points circled in the figure for 2 load cases:

a. a uniform pressure over the entire wall
b. a hydrostatic pressure varying linearly from 0 at the top to maximum at the bottom.

E =3,150 ksi

Length, a = 60 ft

Height, b = 40 ft

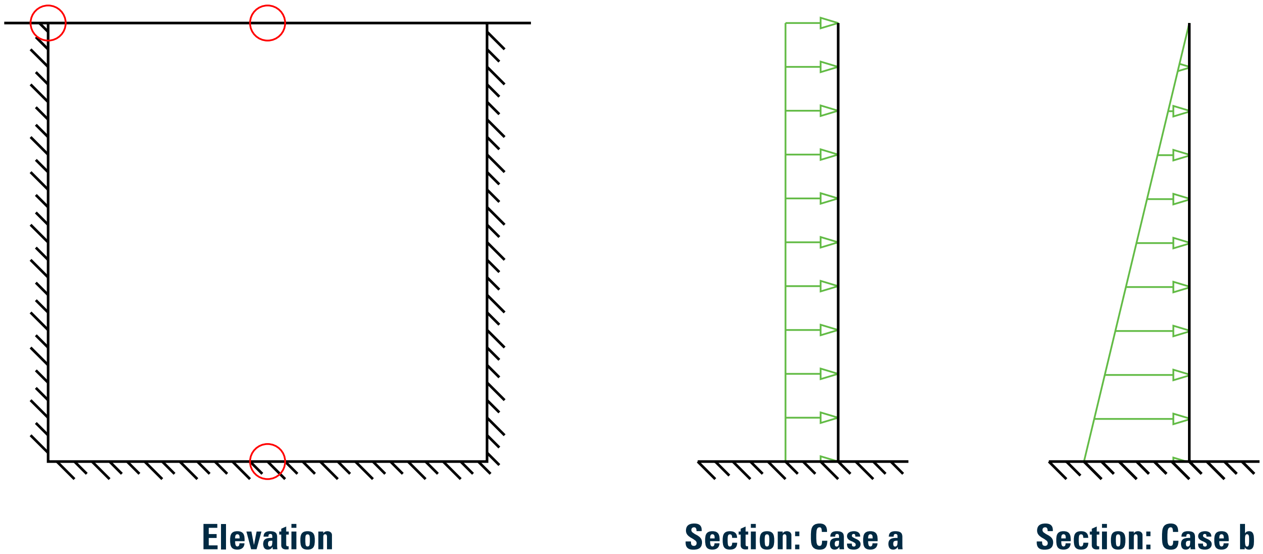


Figure 401: Model

Hand Calculation

$$\frac{a}{b} = \frac{60\text{ft}}{40\text{ft}} = 1.5$$

$$M = \frac{\beta_i q b^2}{6}$$

a. For uniform pressure (load case 1):

where

β_i = = value taken for Case 10a, Table 26, on p.469 of the reference (when a/b = 1.5):

= 0.727 for midpoint of bottom edge (moment about the horizontal axis)

= 0.484 for midpoint of top edge (free edge; moment about the horizontal axis)

= 1.073 for corner of the free edge and fixed edge (moment about the vertical axis)

q = magnitude of the pressure, 1 ksf

b. For linearly varying pressure (load case 2):

Pressure varies from zero intensity at the top to maximum at the base.

where

β_i = value taken for Case 10d, Table 26, on p.470 of the reference (when a/b = 1.5):

= 0.351 for midpoint of bottom edge (moment about the horizontal axis)

= 0.244 for corner of the free edge and fixed edge (moment about the vertical axis)

q = magnitude of the pressure at bottom, 3.5 ksf

Comparison

Table 425: Comparison of results

Result Type		Theory	STAAD.Pro	Difference	Comments
Uniform pressure load case	Bending moment at midpoint of bottom edge (ft·kip/ft)	193.9	393.25/2 = 196.6	1.4%	The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.
	Bending moment at midpoint of top edge (ft·kip/ft)	129	130.6	1.2%	
	Bending moment at corner of free edge and fixed edge (ft·kip/ft)	286.1	294.0	2.8%	
Linearly varying pressure load case	Bending moment at midpoint of bottom edge (ft·kip/ft)	327.6	673.0/2 = 336.5	2.7%	
	Bending moment at corner of free edge and fixed edge (ft·kip/ft)	227.7	212.0	6.9%	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Retaining Wall.STD` is typically installed with the program.

```
STAAD SPACE : A WALL FIXED ALONG 2 EDGES
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
```

```
*
* REFERENCE 'ROARK'S FORMULAS FOR STRESS AND STRAIN', WARREN YOUNG, 6TH ED.
* CASES 10A &amp; 10D, PP.469-470
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 40 0; 3 30 0 0; 4 30 40 0; 5 60 0 0; 6 60 40 0; 7 0 2 0;
8 0 4 0; 9 0 6 0; 10 0 8 0; 11 0 10 0; 12 0 12 0; 13 0 14 0; 14 0 16 0;
15 0 18 0; 16 0 20 0; 17 0 22 0; 18 0 24 0; 19 0 26 0; 20 0 28 0;
21 0 30 0; 22 0 32 0; 23 0 34 0; 24 0 36 0; 25 0 38 0; 26 2 0 0;
27 2 2 0; 28 2 4 0; 29 2 6 0; 30 2 8 0; 31 2 10 0; 32 2 12 0; 33 2 14 0;
34 2 16 0; 35 2 18 0; 36 2 20 0; 37 2 22 0; 38 2 24 0; 39 2 26 0;
40 2 28 0; 41 2 30 0; 42 2 32 0; 43 2 34 0; 44 2 36 0; 45 2 38 0;
46 2 40 0; 47 4 0 0; 48 4 2 0; 49 4 4 0; 50 4 6 0; 51 4 8 0; 52 4 10 0;
53 4 12 0; 54 4 14 0; 55 4 16 0; 56 4 18 0; 57 4 20 0; 58 4 22 0;
59 4 24 0; 60 4 26 0; 61 4 28 0; 62 4 30 0; 63 4 32 0; 64 4 34 0;
65 4 36 0; 66 4 38 0; 67 4 40 0; 68 6 0 0; 69 6 2 0; 70 6 4 0; 71 6 6 0;
72 6 8 0; 73 6 10 0; 74 6 12 0; 75 6 14 0; 76 6 16 0; 77 6 18 0;
78 6 20 0; 79 6 22 0; 80 6 24 0; 81 6 26 0; 82 6 28 0; 83 6 30 0;
84 6 32 0; 85 6 34 0; 86 6 36 0; 87 6 38 0; 88 6 40 0; 89 8 0 0;
90 8 2 0; 91 8 4 0; 92 8 6 0; 93 8 8 0; 94 8 10 0; 95 8 12 0; 96 8 14 0;
97 8 16 0; 98 8 18 0; 99 8 20 0; 100 8 22 0; 101 8 24 0; 102 8 26 0;
103 8 28 0; 104 8 30 0; 105 8 32 0; 106 8 34 0; 107 8 36 0; 108 8 38 0;
109 8 40 0; 110 10 0 0; 111 10 2 0; 112 10 4 0; 113 10 6 0; 114 10 8 0;
115 10 10 0; 116 10 12 0; 117 10 14 0; 118 10 16 0; 119 10 18 0;
120 10 20 0; 121 10 22 0; 122 10 24 0; 123 10 26 0; 124 10 28 0;
125 10 30 0; 126 10 32 0; 127 10 34 0; 128 10 36 0; 129 10 38 0;
130 10 40 0; 131 12 0 0; 132 12 2 0; 133 12 4 0; 134 12 6 0; 135 12 8 0;
136 12 10 0; 137 12 12 0; 138 12 14 0; 139 12 16 0; 140 12 18 0;
141 12 20 0; 142 12 22 0; 143 12 24 0; 144 12 26 0; 145 12 28 0;
146 12 30 0; 147 12 32 0; 148 12 34 0; 149 12 36 0; 150 12 38 0;
151 12 40 0; 152 14 0 0; 153 14 2 0; 154 14 4 0; 155 14 6 0; 156 14 8 0;
157 14 10 0; 158 14 12 0; 159 14 14 0; 160 14 16 0; 161 14 18 0;
162 14 20 0; 163 14 22 0; 164 14 24 0; 165 14 26 0; 166 14 28 0;
167 14 30 0; 168 14 32 0; 169 14 34 0; 170 14 36 0; 171 14 38 0;
172 14 40 0; 173 16 0 0; 174 16 2 0; 175 16 4 0; 176 16 6 0; 177 16 8 0;
178 16 10 0; 179 16 12 0; 180 16 14 0; 181 16 16 0; 182 16 18 0;
183 16 20 0; 184 16 22 0; 185 16 24 0; 186 16 26 0; 187 16 28 0;
188 16 30 0; 189 16 32 0; 190 16 34 0; 191 16 36 0; 192 16 38 0;
193 16 40 0; 194 18 0 0; 195 18 2 0; 196 18 4 0; 197 18 6 0; 198 18 8 0;
199 18 10 0; 200 18 12 0; 201 18 14 0; 202 18 16 0; 203 18 18 0;
204 18 20 0; 205 18 22 0; 206 18 24 0; 207 18 26 0; 208 18 28 0;
209 18 30 0; 210 18 32 0; 211 18 34 0; 212 18 36 0; 213 18 38 0;
214 18 40 0; 215 20 0 0; 216 20 2 0; 217 20 4 0; 218 20 6 0; 219 20 8 0;
220 20 10 0; 221 20 12 0; 222 20 14 0; 223 20 16 0; 224 20 18 0;
225 20 20 0; 226 20 22 0; 227 20 24 0; 228 20 26 0; 229 20 28 0;
230 20 30 0; 231 20 32 0; 232 20 34 0; 233 20 36 0; 234 20 38 0;
235 20 40 0; 236 22 0 0; 237 22 2 0; 238 22 4 0; 239 22 6 0; 240 22 8 0;
241 22 10 0; 242 22 12 0; 243 22 14 0; 244 22 16 0; 245 22 18 0;
246 22 20 0; 247 22 22 0; 248 22 24 0; 249 22 26 0; 250 22 28 0;
251 22 30 0; 252 22 32 0; 253 22 34 0; 254 22 36 0; 255 22 38 0;
256 22 40 0; 257 24 0 0; 258 24 2 0; 259 24 4 0; 260 24 6 0; 261 24 8 0;
262 24 10 0; 263 24 12 0; 264 24 14 0; 265 24 16 0; 266 24 18 0;
267 24 20 0; 268 24 22 0; 269 24 24 0; 270 24 26 0; 271 24 28 0;
272 24 30 0; 273 24 32 0; 274 24 34 0; 275 24 36 0; 276 24 38 0;
277 24 40 0; 278 26 0 0; 279 26 2 0; 280 26 4 0; 281 26 6 0; 282 26 8 0;
283 26 10 0; 284 26 12 0; 285 26 14 0; 286 26 16 0; 287 26 18 0;
288 26 20 0; 289 26 22 0; 290 26 24 0; 291 26 26 0; 292 26 28 0;
```

```
293 26 30 0; 294 26 32 0; 295 26 34 0; 296 26 36 0; 297 26 38 0;
298 26 40 0; 299 28 0 0; 300 28 2 0; 301 28 4 0; 302 28 6 0; 303 28 8 0;
304 28 10 0; 305 28 12 0; 306 28 14 0; 307 28 16 0; 308 28 18 0;
309 28 20 0; 310 28 22 0; 311 28 24 0; 312 28 26 0; 313 28 28 0;
314 28 30 0; 315 28 32 0; 316 28 34 0; 317 28 36 0; 318 28 38 0;
319 28 40 0; 320 30 2 0; 321 30 4 0; 322 30 6 0; 323 30 8 0;
324 30 10 0; 325 30 12 0; 326 30 14 0; 327 30 16 0; 328 30 18 0;
329 30 20 0; 330 30 22 0; 331 30 24 0; 332 30 26 0; 333 30 28 0;
334 30 30 0; 335 30 32 0; 336 30 34 0; 337 30 36 0; 338 30 38 0;
339 32 0 0; 340 32 2 0; 341 32 4 0; 342 32 6 0; 343 32 8 0; 344 32 10 0;
345 32 12 0; 346 32 14 0; 347 32 16 0; 348 32 18 0; 349 32 20 0;
350 32 22 0; 351 32 24 0; 352 32 26 0; 353 32 28 0; 354 32 30 0;
355 32 32 0; 356 32 34 0; 357 32 36 0; 358 32 38 0; 359 32 40 0;
360 34 0 0; 361 34 2 0; 362 34 4 0; 363 34 6 0; 364 34 8 0; 365 34 10 0;
366 34 12 0; 367 34 14 0; 368 34 16 0; 369 34 18 0; 370 34 20 0;
371 34 22 0; 372 34 24 0; 373 34 26 0; 374 34 28 0; 375 34 30 0;
376 34 32 0; 377 34 34 0; 378 34 36 0; 379 34 38 0; 380 34 40 0;
381 36 0 0; 382 36 2 0; 383 36 4 0; 384 36 6 0; 385 36 8 0; 386 36 10 0;
387 36 12 0; 388 36 14 0; 389 36 16 0; 390 36 18 0; 391 36 20 0;
392 36 22 0; 393 36 24 0; 394 36 26 0; 395 36 28 0; 396 36 30 0;
397 36 32 0; 398 36 34 0; 399 36 36 0; 400 36 38 0; 401 36 40 0;
402 38 0 0; 403 38 2 0; 404 38 4 0; 405 38 6 0; 406 38 8 0; 407 38 10 0;
408 38 12 0; 409 38 14 0; 410 38 16 0; 411 38 18 0; 412 38 20 0;
413 38 22 0; 414 38 24 0; 415 38 26 0; 416 38 28 0; 417 38 30 0;
418 38 32 0; 419 38 34 0; 420 38 36 0; 421 38 38 0; 422 38 40 0;
423 40 0 0; 424 40 2 0; 425 40 4 0; 426 40 6 0; 427 40 8 0; 428 40 10 0;
429 40 12 0; 430 40 14 0; 431 40 16 0; 432 40 18 0; 433 40 20 0;
434 40 22 0; 435 40 24 0; 436 40 26 0; 437 40 28 0; 438 40 30 0;
439 40 32 0; 440 40 34 0; 441 40 36 0; 442 40 38 0; 443 40 40 0;
444 42 0 0; 445 42 2 0; 446 42 4 0; 447 42 6 0; 448 42 8 0; 449 42 10 0;
450 42 12 0; 451 42 14 0; 452 42 16 0; 453 42 18 0; 454 42 20 0;
455 42 22 0; 456 42 24 0; 457 42 26 0; 458 42 28 0; 459 42 30 0;
460 42 32 0; 461 42 34 0; 462 42 36 0; 463 42 38 0; 464 42 40 0;
465 44 0 0; 466 44 2 0; 467 44 4 0; 468 44 6 0; 469 44 8 0; 470 44 10 0;
471 44 12 0; 472 44 14 0; 473 44 16 0; 474 44 18 0; 475 44 20 0;
476 44 22 0; 477 44 24 0; 478 44 26 0; 479 44 28 0; 480 44 30 0;
481 44 32 0; 482 44 34 0; 483 44 36 0; 484 44 38 0; 485 44 40 0;
486 46 0 0; 487 46 2 0; 488 46 4 0; 489 46 6 0; 490 46 8 0; 491 46 10 0;
492 46 12 0; 493 46 14 0; 494 46 16 0; 495 46 18 0; 496 46 20 0;
497 46 22 0; 498 46 24 0; 499 46 26 0; 500 46 28 0; 501 46 30 0;
502 46 32 0; 503 46 34 0; 504 46 36 0; 505 46 38 0; 506 46 40 0;
507 48 0 0; 508 48 2 0; 509 48 4 0; 510 48 6 0; 511 48 8 0; 512 48 10 0;
513 48 12 0; 514 48 14 0; 515 48 16 0; 516 48 18 0; 517 48 20 0;
518 48 22 0; 519 48 24 0; 520 48 26 0; 521 48 28 0; 522 48 30 0;
523 48 32 0; 524 48 34 0; 525 48 36 0; 526 48 38 0; 527 48 40 0;
528 50 0 0; 529 50 2 0; 530 50 4 0; 531 50 6 0; 532 50 8 0; 533 50 10 0;
534 50 12 0; 535 50 14 0; 536 50 16 0; 537 50 18 0; 538 50 20 0;
539 50 22 0; 540 50 24 0; 541 50 26 0; 542 50 28 0; 543 50 30 0;
544 50 32 0; 545 50 34 0; 546 50 36 0; 547 50 38 0; 548 50 40 0;
549 52 0 0; 550 52 2 0; 551 52 4 0; 552 52 6 0; 553 52 8 0; 554 52 10 0;
555 52 12 0; 556 52 14 0; 557 52 16 0; 558 52 18 0; 559 52 20 0;
560 52 22 0; 561 52 24 0; 562 52 26 0; 563 52 28 0; 564 52 30 0;
565 52 32 0; 566 52 34 0; 567 52 36 0; 568 52 38 0; 569 52 40 0;
570 54 0 0; 571 54 2 0; 572 54 4 0; 573 54 6 0; 574 54 8 0; 575 54 10 0;
576 54 12 0; 577 54 14 0; 578 54 16 0; 579 54 18 0; 580 54 20 0;
581 54 22 0; 582 54 24 0; 583 54 26 0; 584 54 28 0; 585 54 30 0;
586 54 32 0; 587 54 34 0; 588 54 36 0; 589 54 38 0; 590 54 40 0;
591 56 0 0; 592 56 2 0; 593 56 4 0; 594 56 6 0; 595 56 8 0; 596 56 10 0;
```

```
597 56 12 0; 598 56 14 0; 599 56 16 0; 600 56 18 0; 601 56 20 0;
602 56 22 0; 603 56 24 0; 604 56 26 0; 605 56 28 0; 606 56 30 0;
607 56 32 0; 608 56 34 0; 609 56 36 0; 610 56 38 0; 611 56 40 0;
612 58 0 0; 613 58 2 0; 614 58 4 0; 615 58 6 0; 616 58 8 0; 617 58 10 0;
618 58 12 0; 619 58 14 0; 620 58 16 0; 621 58 18 0; 622 58 20 0;
623 58 22 0; 624 58 24 0; 625 58 26 0; 626 58 28 0; 627 58 30 0;
628 58 32 0; 629 58 34 0; 630 58 36 0; 631 58 38 0; 632 58 40 0;
633 60 2 0; 634 60 4 0; 635 60 6 0; 636 60 8 0; 637 60 10 0;
638 60 12 0; 639 60 14 0; 640 60 16 0; 641 60 18 0; 642 60 20 0;
643 60 22 0; 644 60 24 0; 645 60 26 0; 646 60 28 0; 647 60 30 0;
648 60 32 0; 649 60 34 0; 650 60 36 0; 651 60 38 0;
ELEMENT INCIDENCES SHELL
1 1 7 27 26; 2 7 8 28 27; 3 8 9 29 28; 4 9 10 30 29; 5 10 11 31 30;
6 11 12 32 31; 7 12 13 33 32; 8 13 14 34 33; 9 14 15 35 34;
10 15 16 36 35; 11 16 17 37 36; 12 17 18 38 37; 13 18 19 39 38;
14 19 20 40 39; 15 20 21 41 40; 16 21 22 42 41; 17 22 23 43 42;
18 23 24 44 43; 19 24 25 45 44; 20 25 2 46 45; 21 26 27 48 47;
22 27 28 49 48; 23 28 29 50 49; 24 29 30 51 50; 25 30 31 52 51;
26 31 32 53 52; 27 32 33 54 53; 28 33 34 55 54; 29 34 35 56 55;
30 35 36 57 56; 31 36 37 58 57; 32 37 38 59 58; 33 38 39 60 59;
34 39 40 61 60; 35 40 41 62 61; 36 41 42 63 62; 37 42 43 64 63;
38 43 44 65 64; 39 44 45 66 65; 40 45 46 67 66; 41 47 48 69 68;
42 48 49 70 69; 43 49 50 71 70; 44 50 51 72 71; 45 51 52 73 72;
46 52 53 74 73; 47 53 54 75 74; 48 54 55 76 75; 49 55 56 77 76;
50 56 57 78 77; 51 57 58 79 78; 52 58 59 80 79; 53 59 60 81 80;
54 60 61 82 81; 55 61 62 83 82; 56 62 63 84 83; 57 63 64 85 84;
58 64 65 86 85; 59 65 66 87 86; 60 66 67 88 87; 61 68 69 90 89;
62 69 70 91 90; 63 70 71 92 91; 64 71 72 93 92; 65 72 73 94 93;
66 73 74 95 94; 67 74 75 96 95; 68 75 76 97 96; 69 76 77 98 97;
70 77 78 99 98; 71 78 79 100 99; 72 79 80 101 100; 73 80 81 102 101;
74 81 82 103 102; 75 82 83 104 103; 76 83 84 105 104; 77 84 85 106 105;
78 85 86 107 106; 79 86 87 108 107; 80 87 88 109 108; 81 89 90 111 110;
82 90 91 112 111; 83 91 92 113 112; 84 92 93 114 113; 85 93 94 115 114;
86 94 95 116 115; 87 95 96 117 116; 88 96 97 118 117; 89 97 98 119 118;
90 98 99 120 119; 91 99 100 121 120; 92 100 101 122 121;
93 101 102 123 122; 94 102 103 124 123; 95 103 104 125 124;
96 104 105 126 125; 97 105 106 127 126; 98 106 107 128 127;
99 107 108 129 128; 100 108 109 130 129; 101 110 111 132 131;
102 111 112 133 132; 103 112 113 134 133; 104 113 114 135 134;
105 114 115 136 135; 106 115 116 137 136; 107 116 117 138 137;
108 117 118 139 138; 109 118 119 140 139; 110 119 120 141 140;
111 120 121 142 141; 112 121 122 143 142; 113 122 123 144 143;
114 123 124 145 144; 115 124 125 146 145; 116 125 126 147 146;
117 126 127 148 147; 118 127 128 149 148; 119 128 129 150 149;
120 129 130 151 150; 121 131 132 153 152; 122 132 133 154 153;
123 133 134 155 154; 124 134 135 156 155; 125 135 136 157 156;
126 136 137 158 157; 127 137 138 159 158; 128 138 139 160 159;
129 139 140 161 160; 130 140 141 162 161; 131 141 142 163 162;
132 142 143 164 163; 133 143 144 165 164; 134 144 145 166 165;
135 145 146 167 166; 136 146 147 168 167; 137 147 148 169 168;
138 148 149 170 169; 139 149 150 171 170; 140 150 151 172 171;
141 152 153 174 173; 142 153 154 175 174; 143 154 155 176 175;
144 155 156 177 176; 145 156 157 178 177; 146 157 158 179 178;
147 158 159 180 179; 148 159 160 181 180; 149 160 161 182 181;
150 161 162 183 182; 151 162 163 184 183; 152 163 164 185 184;
153 164 165 186 185; 154 165 166 187 186; 155 166 167 188 187;
156 167 168 189 188; 157 168 169 190 189; 158 169 170 191 190;
159 170 171 192 191; 160 171 172 193 192; 161 173 174 195 194;
```

```
162 174 175 196 195; 163 175 176 197 196; 164 176 177 198 197;
165 177 178 199 198; 166 178 179 200 199; 167 179 180 201 200;
168 180 181 202 201; 169 181 182 203 202; 170 182 183 204 203;
171 183 184 205 204; 172 184 185 206 205; 173 185 186 207 206;
174 186 187 208 207; 175 187 188 209 208; 176 188 189 210 209;
177 189 190 211 210; 178 190 191 212 211; 179 191 192 213 212;
180 192 193 214 213; 181 194 195 216 215; 182 195 196 217 216;
183 196 197 218 217; 184 197 198 219 218; 185 198 199 220 219;
186 199 200 221 220; 187 200 201 222 221; 188 201 202 223 222;
189 202 203 224 223; 190 203 204 225 224; 191 204 205 226 225;
192 205 206 227 226; 193 206 207 228 227; 194 207 208 229 228;
195 208 209 230 229; 196 209 210 231 230; 197 210 211 232 231;
198 211 212 233 232; 199 212 213 234 233; 200 213 214 235 234;
201 215 216 237 236; 202 216 217 238 237; 203 217 218 239 238;
204 218 219 240 239; 205 219 220 241 240; 206 220 221 242 241;
207 221 222 243 242; 208 222 223 244 243; 209 223 224 245 244;
210 224 225 246 245; 211 225 226 247 246; 212 226 227 248 247;
213 227 228 249 248; 214 228 229 250 249; 215 229 230 251 250;
216 230 231 252 251; 217 231 232 253 252; 218 232 233 254 253;
219 233 234 255 254; 220 234 235 256 255; 221 236 237 258 257;
222 237 238 259 258; 223 238 239 260 259; 224 239 240 261 260;
225 240 241 262 261; 226 241 242 263 262; 227 242 243 264 263;
228 243 244 265 264; 229 244 245 266 265; 230 245 246 267 266;
231 246 247 268 267; 232 247 248 269 268; 233 248 249 270 269;
234 249 250 271 270; 235 250 251 272 271; 236 251 252 273 272;
237 252 253 274 273; 238 253 254 275 274; 239 254 255 276 275;
240 255 256 277 276; 241 257 258 279 278; 242 258 259 280 279;
243 259 260 281 280; 244 260 261 282 281; 245 261 262 283 282;
246 262 263 284 283; 247 263 264 285 284; 248 264 265 286 285;
249 265 266 287 286; 250 266 267 288 287; 251 267 268 289 288;
252 268 269 290 289; 253 269 270 291 290; 254 270 271 292 291;
255 271 272 293 292; 256 272 273 294 293; 257 273 274 295 294;
258 274 275 296 295; 259 275 276 297 296; 260 276 277 298 297;
261 278 279 300 299; 262 279 280 301 300; 263 280 281 302 301;
264 281 282 303 302; 265 282 283 304 303; 266 283 284 305 304;
267 284 285 306 305; 268 285 286 307 306; 269 286 287 308 307;
270 287 288 309 308; 271 288 289 310 309; 272 289 290 311 310;
273 290 291 312 311; 274 291 292 313 312; 275 292 293 314 313;
276 293 294 315 314; 277 294 295 316 315; 278 295 296 317 316;
279 296 297 318 317; 280 297 298 319 318; 281 299 300 320 3;
282 300 301 321 320; 283 301 302 322 321; 284 302 303 323 322;
285 303 304 324 323; 286 304 305 325 324; 287 305 306 326 325;
288 306 307 327 326; 289 307 308 328 327; 290 308 309 329 328;
291 309 310 330 329; 292 310 311 331 330; 293 311 312 332 331;
294 312 313 333 332; 295 313 314 334 333; 296 314 315 335 334;
297 315 316 336 335; 298 316 317 337 336; 299 317 318 338 337;
300 318 319 4 338; 301 3 320 340 339; 302 320 321 341 340;
303 321 322 342 341; 304 322 323 343 342; 305 323 324 344 343;
306 324 325 345 344; 307 325 326 346 345; 308 326 327 347 346;
309 327 328 348 347; 310 328 329 349 348; 311 329 330 350 349;
312 330 331 351 350; 313 331 332 352 351; 314 332 333 353 352;
315 333 334 354 353; 316 334 335 355 354; 317 335 336 356 355;
318 336 337 357 356; 319 337 338 358 357; 320 338 4 359 358;
321 339 340 361 360; 322 340 341 362 361; 323 341 342 363 362;
324 342 343 364 363; 325 343 344 365 364; 326 344 345 366 365;
327 345 346 367 366; 328 346 347 368 367; 329 347 348 369 368;
330 348 349 370 369; 331 349 350 371 370; 332 350 351 372 371;
333 351 352 373 372; 334 352 353 374 373; 335 353 354 375 374;
```

```
336 354 355 376 375; 337 355 356 377 376; 338 356 357 378 377;
339 357 358 379 378; 340 358 359 380 379; 341 360 361 382 381;
342 361 362 383 382; 343 362 363 384 383; 344 363 364 385 384;
345 364 365 386 385; 346 365 366 387 386; 347 366 367 388 387;
348 367 368 389 388; 349 368 369 390 389; 350 369 370 391 390;
351 370 371 392 391; 352 371 372 393 392; 353 372 373 394 393;
354 373 374 395 394; 355 374 375 396 395; 356 375 376 397 396;
357 376 377 398 397; 358 377 378 399 398; 359 378 379 400 399;
360 379 380 401 400; 361 381 382 403 402; 362 382 383 404 403;
363 383 384 405 404; 364 384 385 406 405; 365 385 386 407 406;
366 386 387 408 407; 367 387 388 409 408; 368 388 389 410 409;
369 389 390 411 410; 370 390 391 412 411; 371 391 392 413 412;
372 392 393 414 413; 373 393 394 415 414; 374 394 395 416 415;
375 395 396 417 416; 376 396 397 418 417; 377 397 398 419 418;
378 398 399 420 419; 379 399 400 421 420; 380 400 401 422 421;
381 402 403 424 423; 382 403 404 425 424; 383 404 405 426 425;
384 405 406 427 426; 385 406 407 428 427; 386 407 408 429 428;
387 408 409 430 429; 388 409 410 431 430; 389 410 411 432 431;
390 411 412 433 432; 391 412 413 434 433; 392 413 414 435 434;
393 414 415 436 435; 394 415 416 437 436; 395 416 417 438 437;
396 417 418 439 438; 397 418 419 440 439; 398 419 420 441 440;
399 420 421 442 441; 400 421 422 443 442; 401 423 424 445 444;
402 424 425 446 445; 403 425 426 447 446; 404 426 427 448 447;
405 427 428 449 448; 406 428 429 450 449; 407 429 430 451 450;
408 430 431 452 451; 409 431 432 453 452; 410 432 433 454 453;
411 433 434 455 454; 412 434 435 456 455; 413 435 436 457 456;
414 436 437 458 457; 415 437 438 459 458; 416 438 439 460 459;
417 439 440 461 460; 418 440 441 462 461; 419 441 442 463 462;
420 442 443 464 463; 421 444 445 466 465; 422 445 446 467 466;
423 446 447 468 467; 424 447 448 469 468; 425 448 449 470 469;
426 449 450 471 470; 427 450 451 472 471; 428 451 452 473 472;
429 452 453 474 473; 430 453 454 475 474; 431 454 455 476 475;
432 455 456 477 476; 433 456 457 478 477; 434 457 458 479 478;
435 458 459 480 479; 436 459 460 481 480; 437 460 461 482 481;
438 461 462 483 482; 439 462 463 484 483; 440 463 464 485 484;
441 465 466 487 486; 442 466 467 488 487; 443 467 468 489 488;
444 468 469 490 489; 445 469 470 491 490; 446 470 471 492 491;
447 471 472 493 492; 448 472 473 494 493; 449 473 474 495 494;
450 474 475 496 495; 451 475 476 497 496; 452 476 477 498 497;
453 477 478 499 498; 454 478 479 500 499; 455 479 480 501 500;
456 480 481 502 501; 457 481 482 503 502; 458 482 483 504 503;
459 483 484 505 504; 460 484 485 506 505; 461 486 487 508 507;
462 487 488 509 508; 463 488 489 510 509; 464 489 490 511 510;
465 490 491 512 511; 466 491 492 513 512; 467 492 493 514 513;
468 493 494 515 514; 469 494 495 516 515; 470 495 496 517 516;
471 496 497 518 517; 472 497 498 519 518; 473 498 499 520 519;
474 499 500 521 520; 475 500 501 522 521; 476 501 502 523 522;
477 502 503 524 523; 478 503 504 525 524; 479 504 505 526 525;
480 505 506 527 526; 481 507 508 529 528; 482 508 509 530 529;
483 509 510 531 530; 484 510 511 532 531; 485 511 512 533 532;
486 512 513 534 533; 487 513 514 535 534; 488 514 515 536 535;
489 515 516 537 536; 490 516 517 538 537; 491 517 518 539 538;
492 518 519 540 539; 493 519 520 541 540; 494 520 521 542 541;
495 521 522 543 542; 496 522 523 544 543; 497 523 524 545 544;
498 524 525 546 545; 499 525 526 547 546; 500 526 527 548 547;
501 528 529 550 549; 502 529 530 551 550; 503 530 531 552 551;
504 531 532 553 552; 505 532 533 554 553; 506 533 534 555 554;
507 534 535 556 555; 508 535 536 557 556; 509 536 537 558 557;
```

```
510 537 538 559 558; 511 538 539 560 559; 512 539 540 561 560;
513 540 541 562 561; 514 541 542 563 562; 515 542 543 564 563;
516 543 544 565 564; 517 544 545 566 565; 518 545 546 567 566;
519 546 547 568 567; 520 547 548 569 568; 521 549 550 571 570;
522 550 551 572 571; 523 551 552 573 572; 524 552 553 574 573;
525 553 554 575 574; 526 554 555 576 575; 527 555 556 577 576;
528 556 557 578 577; 529 557 558 579 578; 530 558 559 580 579;
531 559 560 581 580; 532 560 561 582 581; 533 561 562 583 582;
534 562 563 584 583; 535 563 564 585 584; 536 564 565 586 585;
537 565 566 587 586; 538 566 567 588 587; 539 567 568 589 588;
540 568 569 590 589; 541 570 571 592 591; 542 571 572 593 592;
543 572 573 594 593; 544 573 574 595 594; 545 574 575 596 595;
546 575 576 597 596; 547 576 577 598 597; 548 577 578 599 598;
549 578 579 600 599; 550 579 580 601 600; 551 580 581 602 601;
552 581 582 603 602; 553 582 583 604 603; 554 583 584 605 604;
555 584 585 606 605; 556 585 586 607 606; 557 586 587 608 607;
558 587 588 609 608; 559 588 589 610 609; 560 589 590 611 610;
561 591 592 613 612; 562 592 593 614 613; 563 593 594 615 614;
564 594 595 616 615; 565 595 596 617 616; 566 596 597 618 617;
567 597 598 619 618; 568 598 599 620 619; 569 599 600 621 620;
570 600 601 622 621; 571 601 602 623 622; 572 602 603 624 623;
573 603 604 625 624; 574 604 605 626 625; 575 605 606 627 626;
576 606 607 628 627; 577 607 608 629 628; 578 608 609 630 629;
579 609 610 631 630; 580 610 611 632 631; 581 612 613 633 5;
582 613 614 634 633; 583 614 615 635 634; 584 615 616 636 635;
585 616 617 637 636; 586 617 618 638 637; 587 618 619 639 638;
588 619 620 640 639; 589 620 621 641 640; 590 621 622 642 641;
591 622 623 643 642; 592 623 624 644 643; 593 624 625 645 644;
594 625 626 646 645; 595 626 627 647 646; 596 627 628 648 647;
597 628 629 649 648; 598 629 630 650 649; 599 630 631 651 650;
600 631 632 6 651;
ELEMENT PROPERTY
1 TO 600 THICKNESS 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 453600
POISSON 0.2
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
UNIT FEET KIP
SUPPORTS
1 2 7 TO 25 FIXED
5 6 633 TO 651 FIXED
1 3 5 26 47 68 89 110 131 152 173 194 215 236 257 278 299 339 360 381 -
402 423 444 465 486 507 528 549 570 591 612 FIXED
LOAD 1
ELEMENT LOAD
1 TO 600 PR 1
LOAD 2 HYDROSTATIC
ELEMENT LOAD
1 21 41 61 81 101 121 141 161 181 201 221 241 261 281 301 321 341 361 -
381 401 421 441 461 481 501 521 541 561 581 TRAP X 3.5 3.325
2 22 42 62 82 102 122 142 162 182 202 222 242 262 282 302 322 342 362 -
382 402 422 442 462 482 502 522 542 562 582 TRAP X 3.325 3.15
3 23 43 63 83 103 123 143 163 183 203 223 243 263 283 303 323 343 363 -
383 403 423 443 463 483 503 523 543 563 583 TRAP X 3.15 2.975
```

```
4 24 44 64 84 104 124 144 164 184 204 224 244 264 284 304 324 344 364 -
384 404 424 444 464 484 504 524 544 564 584 TRAP X 2.975 2.8
5 25 45 65 85 105 125 145 165 185 205 225 245 265 285 305 325 345 365 -
385 405 425 445 465 485 505 525 545 565 585 TRAP X 2.8 2.625
6 26 46 66 86 106 126 146 166 186 206 226 246 266 286 306 326 346 366 -
386 406 426 446 466 486 506 526 546 566 586 TRAP X 2.625 2.45
7 27 47 67 87 107 127 147 167 187 207 227 247 267 287 307 327 347 367 -
387 407 427 447 467 487 507 527 547 567 587 TRAP X 2.45 2.275
8 28 48 68 88 108 128 148 168 188 208 228 248 268 288 308 328 348 368 -
388 408 428 448 468 488 508 528 548 568 588 TRAP X 2.275 2.1
9 29 49 69 89 109 129 149 169 189 209 229 249 269 289 309 329 349 369 -
389 409 429 449 469 489 509 529 549 569 589 TRAP X 2.1 1.925
10 30 50 70 90 110 130 150 170 190 210 230 250 270 290 310 330 350 -
370 390 410 430 450 470 490 510 530 550 570 590 TRAP X 1.925 1.75
11 31 51 71 91 111 131 151 171 191 211 231 251 271 291 311 331 351 -
371 391 411 431 451 471 491 511 531 551 571 591 TRAP X 1.75 1.575
12 32 52 72 92 112 132 152 172 192 212 232 252 272 292 312 332 352 -
372 392 412 432 452 472 492 512 532 552 572 592 TRAP X 1.575 1.4
13 33 53 73 93 113 133 153 173 193 213 233 253 273 293 313 333 353 -
373 393 413 433 453 473 493 513 533 553 573 593 TRAP X 1.4 1.225
14 34 54 74 94 114 134 154 174 194 214 234 254 274 294 314 334 354 -
374 394 414 434 454 474 494 514 534 554 574 594 TRAP X 1.225 1.05
15 35 55 75 95 115 135 155 175 195 215 235 255 275 295 315 335 355 -
375 395 415 435 455 475 495 515 535 555 575 595 TRAP X 1.05 0.875
16 36 56 76 96 116 136 156 176 196 216 236 256 276 296 316 336 356 -
376 396 416 436 456 476 496 516 536 556 576 596 TRAP X 0.875 0.7
17 37 57 77 97 117 137 157 177 197 217 237 257 277 297 317 337 357 -
377 397 417 437 457 477 497 517 537 557 577 597 TRAP X 0.7 0.525
18 38 58 78 98 118 138 158 178 198 218 238 258 278 298 318 338 358 -
378 398 418 438 458 478 498 518 538 558 578 598 TRAP X 0.525 0.35
19 39 59 79 99 119 139 159 179 199 219 239 259 279 299 319 339 359 -
379 399 419 439 459 479 499 519 539 559 579 599 TRAP X 0.35 0.175
20 40 60 80 100 120 140 160 180 200 220 240 260 280 300 320 340 360 -
380 400 420 440 460 480 500 520 540 560 580 600 TRAP X 0.175 0
PERFORM ANALYSIS
PRINT SUPPORT REACTION LIST 2 3
PRINT ELEMENT JOINT STRESSES LIST 300
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     2    1      0.00      0.00     24.50    -59.94   -293.96      0.00
          2      0.00      0.00     -3.73    -42.71   -211.97      0.00
     3    1      0.00      0.00     53.00    393.27      0.00      0.00
          2      0.00      0.00    112.98    673.01      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
   389. PRINT ELEMENT JOINT STRESSES LIST 300
  ELEMENT  JOINT    STRESSES LIST
     : A WALL FIXED ALONG 2 EDGES                             -- PAGE NO.
10
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
```

```
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    300      1        -0.88        1.09        2.76      126.17       -2.01
                     187.30      187.30        0.00        0.00        0.00
                     189.31      189.31
      TOP : SMAX=     189.31 SMIN=       4.09 TMAX=      92.61 ANGLE=-89.1
      BOTT: SMAX=      -4.09 SMIN=    -189.31 TMAX=      92.61 ANGLE=  0.9
        JOINT         -0.92        0.26        4.26      125.10       -2.45
         318         184.65      184.65        0.00        0.00        0.00
      TOP : SMAX=     187.73 SMIN=       6.32 TMAX=      90.70 ANGLE=-88.8
      BOTT: SMAX=      -6.32 SMIN=    -187.73 TMAX=      90.70 ANGLE=  1.2
        JOINT         -0.92        1.92        0.87      124.07       -1.27
         319         185.49      185.49        0.00        0.00        0.00
      TOP : SMAX=     186.13 SMIN=       1.29 TMAX=      92.42 ANGLE=-89.4
      BOTT: SMAX=      -1.29 SMIN=    -186.13 TMAX=      92.42 ANGLE=  0.6
        JOINT         -0.84        1.92        1.43      130.57       -1.57
          4          194.83      194.83        0.00        0.00        0.00
      TOP : SMAX=     195.88 SMIN=       2.11 TMAX=      96.88 ANGLE=-89.3
      BOTT: SMAX=      -2.11 SMIN=    -195.88 TMAX=      96.88 ANGLE=  0.7
        JOINT         -0.84        0.26        4.47      124.94       -2.76
         338         184.30      184.30        0.00        0.00        0.00
      TOP : SMAX=     187.51 SMIN=       6.61 TMAX=      90.45 ANGLE=-88.7
      BOTT: SMAX=      -6.61 SMIN=    -187.51 TMAX=      90.45 ANGLE=  1.3
             2        -0.92        0.78        2.26      116.85       -1.13
                     173.62      173.62        0.00        0.00        0.00
                     175.29      175.29
      TOP : SMAX=     175.29 SMIN=       3.37 TMAX=      85.96 ANGLE=-89.4
      BOTT: SMAX=      -3.37 SMIN=    -175.29 TMAX=      85.96 ANGLE=  0.6
        JOINT         -0.94        0.29        3.71      116.18       -1.21
         318         171.58      171.58        0.00        0.00        0.00
      TOP : SMAX=     174.28 SMIN=       5.55 TMAX=      84.37 ANGLE=-89.4
      BOTT: SMAX=      -5.55 SMIN=    -174.28 TMAX=      84.37 ANGLE=  0.6
        JOINT         -0.94        1.26        0.62      115.23       -0.38
         319         172.39      172.39        0.00        0.00        0.00
      TOP : SMAX=     172.85 SMIN=       0.93 TMAX=      85.96 ANGLE=-89.8
      BOTT: SMAX=      -0.93 SMIN=    -172.85 TMAX=      85.96 ANGLE=  0.2
        JOINT         -0.90        1.26        0.90      119.46       -1.05
          4          178.54      178.54        0.00        0.00        0.00
      TOP : SMAX=     179.20 SMIN=       1.34 TMAX=      88.93 ANGLE=-89.5
      BOTT: SMAX=      -1.34 SMIN=    -179.20 TMAX=      88.93 ANGLE=  0.5
        JOINT         -0.90        0.29        3.81      116.52       -1.88
         338         172.06      172.06        0.00        0.00        0.00
      TOP : SMAX=     174.83 SMIN=       5.67 TMAX=      84.58 ANGLE=-89.0
      BOTT: SMAX=      -5.67 SMIN=    -174.83 TMAX=      84.58 ANGLE=  1.0
     : A WALL FIXED ALONG 2 EDGES                                -- PAGE NO.
11
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          1.958811E+02 -1.958811E+02  9.688409E+01  1.872972E+02  1.893074E+02
 PLATE NO.     300           300           300           300           300
 CASE  NO.       1             1             1             1             1
```

V. 2D Surface Displacements

To find the vertical deflection and bending moments due to an unit pressure in a rectangular plate simply supported along four edges.

References

Martin, H. C., *Stiffness Matrix for a Triangular Sandwich Element in Bending*, NASA Technical Report 32-1158, October, 1967.

Timoshenko, S., *Strength of Materials*, D. Van Nostrand Co., 3rd Edition,1956.

Salerno, V. L., and Goldberg, M. A., *Effect of Shear Deformations on the Bending of Rectangular Plates*, March 1960, Pages 54-58.

Problem

Find the vertical deflection at the points shown in the sketch and the bending stress at the center of a 20 in. square plate subjected to a uniform pressure of 1 ksi. Use a quarter of the plate assuming proper boundary conditions along the lines of symmetry.

E = 10,000.0 ksi

Thickness = 5 in

Poisson's ratio = 0.4

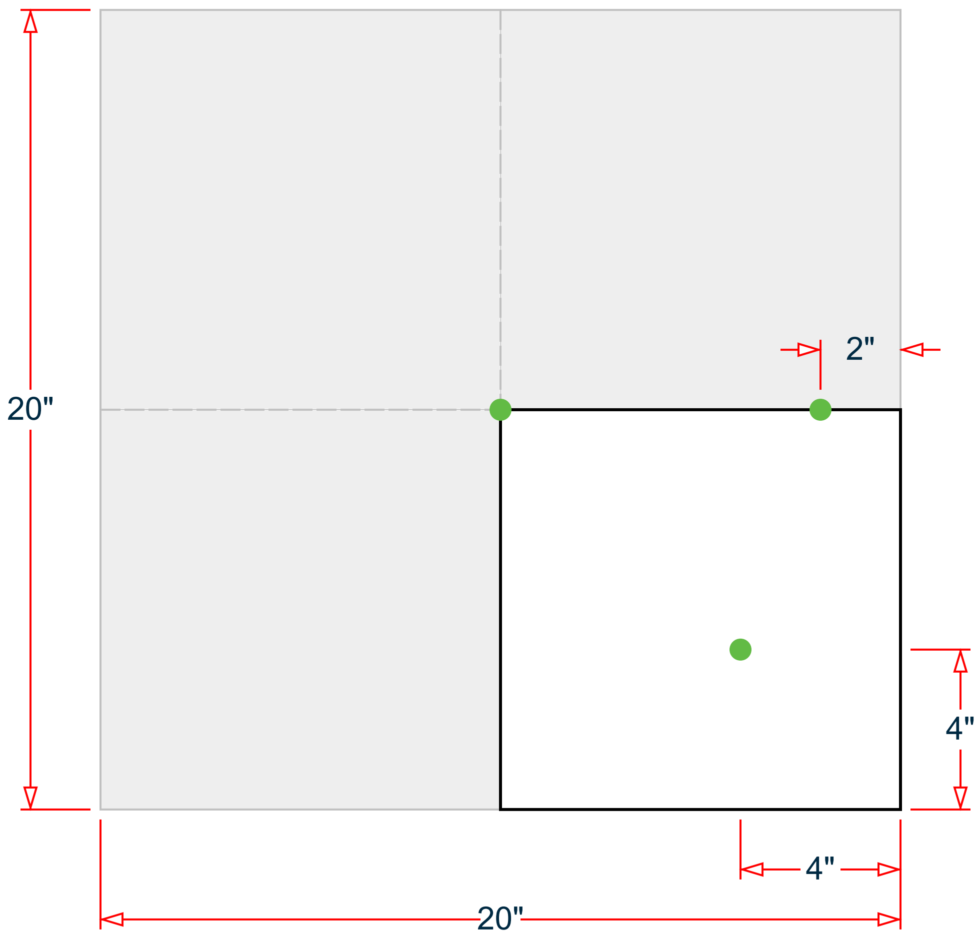


Figure 402: Model

Comparison

Table 426: Comparison of results

Result Type		Theory	STAAD.Pro	Difference	Comments
Vertical deflections due to unit pressure, 10^{-3} in	δ_1	6.826	6.864	0.6%	
	δ_9	2.322	2.343	0.9%	
	δ_{73}	2.684	2.707	0.9%	

Result Type	Theory	STAAD.Pro	Difference	Comments
Maximum bending stress due to unit pressure, σ, element 1 (psi)	5,071	4,908	3.2%	The theoretical results from classical Plate theory was compared with the results from Finite Element model - hence the difference in results.

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Surface Displacements.STD` is typically installed with the program.

```
STAAD SPACE :A SIMPLY SUPPORTED PLATE WITH TRIANGULAR ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 12 0 0 1; 13 1 0 1; 14 2 0 1; 15 3 0 1;
16 4 0 1; 17 5 0 1; 18 6 0 1; 19 7 0 1; 20 8 0 1; 21 9 0 1; 22 10 0 1;
23 0 0 2; 24 1 0 2; 25 2 0 2; 26 3 0 2; 27 4 0 2; 28 5 0 2; 29 6 0 2;
30 7 0 2; 31 8 0 2; 32 9 0 2; 33 10 0 2; 34 0 0 3; 35 1 0 3; 36 2 0 3;
37 3 0 3; 38 4 0 3; 39 5 0 3; 40 6 0 3; 41 7 0 3; 42 8 0 3; 43 9 0 3;
44 10 0 3; 45 0 0 4; 46 1 0 4; 47 2 0 4; 48 3 0 4; 49 4 0 4; 50 5 0 4;
51 6 0 4; 52 7 0 4; 53 8 0 4; 54 9 0 4; 55 10 0 4; 56 0 0 5; 57 1 0 5;
58 2 0 5; 59 3 0 5; 60 4 0 5; 61 5 0 5; 62 6 0 5; 63 7 0 5; 64 8 0 5;
65 9 0 5; 66 10 0 5; 67 0 0 6; 68 1 0 6; 69 2 0 6; 70 3 0 6; 71 4 0 6;
72 5 0 6; 73 6 0 6; 74 7 0 6; 75 8 0 6; 76 9 0 6; 77 10 0 6; 78 0 0 7;
79 1 0 7; 80 2 0 7; 81 3 0 7; 82 4 0 7; 83 5 0 7; 84 6 0 7; 85 7 0 7;
86 8 0 7; 87 9 0 7; 88 10 0 7; 89 0 0 8; 90 1 0 8; 91 2 0 8; 92 3 0 8;
93 4 0 8; 94 5 0 8; 95 6 0 8; 96 7 0 8; 97 8 0 8; 98 9 0 8; 99 10 0 8;
100 0 0 9; 101 1 0 9; 102 2 0 9; 103 3 0 9; 104 4 0 9; 105 5 0 9;
106 6 0 9; 107 7 0 9; 108 8 0 9; 109 9 0 9; 110 10 0 9; 111 0 0 10;
112 1 0 10; 113 2 0 10; 114 3 0 10; 115 4 0 10; 116 5 0 10; 117 6 0 10;
118 7 0 10; 119 8 0 10; 120 9 0 10; 121 10 0 10;
ELEMENT INCIDENCES SHELL
1 1 2 12; 2 2 3 13; 3 3 4 14; 4 4 5 15; 5 5 6 16; 6 6 7 17; 7 7 8 18;
8 8 9 19; 9 9 10 20; 10 10 11 21; 11 12 2 13; 12 13 3 14; 13 14 4 15;
14 15 5 16; 15 16 6 17; 16 17 7 18; 17 18 8 19; 18 19 9 20; 19 20 10 21;
20 21 11 22; 21 12 13 23; 22 13 14 24; 23 14 15 25; 24 15 16 26;
25 16 17 27; 26 17 18 28; 27 18 19 29; 28 19 20 30; 29 20 21 31;
30 21 22 32; 31 23 13 24; 32 24 14 25; 33 25 15 26; 34 26 16 27;
35 27 17 28; 36 28 18 29; 37 29 19 30; 38 30 20 31; 39 31 21 32;
```

```
40 32 22 33; 41 23 24 34; 42 24 25 35; 43 25 26 36; 44 26 27 37;
45 27 28 38; 46 28 29 39; 47 29 30 40; 48 30 31 41; 49 31 32 42;
50 32 33 43; 51 34 24 35; 52 35 25 36; 53 36 26 37; 54 37 27 38;
55 38 28 39; 56 39 29 40; 57 40 30 41; 58 41 31 42; 59 42 32 43;
60 43 33 44; 61 34 35 45; 62 35 36 46; 63 36 37 47; 64 37 38 48;
65 38 39 49; 66 39 40 50; 67 40 41 51; 68 41 42 52; 69 42 43 53;
70 43 44 54; 71 45 35 46; 72 46 36 47; 73 47 37 48; 74 48 38 49;
75 49 39 50; 76 50 40 51; 77 51 41 52; 78 52 42 53; 79 53 43 54;
80 54 44 55; 81 45 46 56; 82 46 47 57; 83 47 48 58; 84 48 49 59;
85 49 50 60; 86 50 51 61; 87 51 52 62; 88 52 53 63; 89 53 54 64;
90 54 55 65; 91 56 46 57; 92 57 47 58; 93 58 48 59; 94 59 49 60;
95 60 50 61; 96 61 51 62; 97 62 52 63; 98 63 53 64; 99 64 54 65;
100 65 55 66; 101 56 57 67; 102 57 58 68; 103 58 59 69; 104 59 60 70;
105 60 61 71; 106 61 62 72; 107 62 63 73; 108 63 64 74; 109 64 65 75;
110 65 66 76; 111 67 57 68; 112 68 58 69; 113 69 59 70; 114 70 60 71;
115 71 61 72; 116 72 62 73; 117 73 63 74; 118 74 64 75; 119 75 65 76;
120 76 66 77; 121 67 68 78; 122 68 69 79; 123 69 70 80; 124 70 71 81;
125 71 72 82; 126 72 73 83; 127 73 74 84; 128 74 75 85; 129 75 76 86;
130 76 77 87; 131 78 68 79; 132 79 69 80; 133 80 70 81; 134 81 71 82;
135 82 72 83; 136 83 73 84; 137 84 74 85; 138 85 75 86; 139 86 76 87;
140 87 77 88; 141 78 79 89; 142 79 80 90; 143 80 81 91; 144 81 82 92;
145 82 83 93; 146 83 84 94; 147 84 85 95; 148 85 86 96; 149 86 87 97;
150 87 88 98; 151 89 79 90; 152 90 80 91; 153 91 81 92; 154 92 82 93;
155 93 83 94; 156 94 84 95; 157 95 85 96; 158 96 86 97; 159 97 87 98;
160 98 88 99; 161 89 90 100; 162 90 91 101; 163 91 92 102;
164 92 93 103; 165 93 94 104; 166 94 95 105; 167 95 96 106;
168 96 97 107; 169 97 98 108; 170 98 99 109; 171 100 90 101;
172 101 91 102; 173 102 92 103; 174 103 93 104; 175 104 94 105;
176 105 95 106; 177 106 96 107; 178 107 97 108; 179 108 98 109;
180 109 99 110; 181 100 101 111; 182 101 102 112; 183 102 103 113;
184 103 104 114; 185 104 105 115; 186 105 106 116; 187 106 107 117;
188 107 108 118; 189 108 109 119; 190 109 110 120; 191 111 101 112;
192 112 102 113; 193 113 103 114; 194 114 104 115; 195 115 105 116;
196 116 106 117; 197 117 107 118; 198 118 108 119; 199 119 109 120;
200 120 110 121;
ELEMENT PROPERTY
1 TO 200 THICKNESS 5
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.4
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED BUT FY
2 TO 10 FIXED BUT FX FY MZ
12 23 34 45 56 67 78 89 100 FIXED BUT FY FZ MX
11 22 33 44 55 66 77 88 99 110 FIXED BUT MZ
111 TO 120 FIXED BUT MX
121 FIXED
LOAD 1 UNIFORM PRESSURE
ELEMENT LOAD
1 TO 200 PR -1000
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1 9 73
```

```
PRINT ELEMENT STRESSES LIST 1
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00686   0.00000   0.00000   0.00000   0.00000
      9    1    0.00000   0.00234   0.00000   0.00000   0.00000  -0.00080
     73    1    0.00000   0.00271   0.00000   0.00040   0.00000  -0.00040
   ************** END OF LATEST ANALYSIS RESULT **************
    90. PRINT ELEMENT STRESSES LIST 1
  ELEMENT  STRESSES LIST     1
      :A SIMPLY SUPPORTED PLATE WITH TRIANGULAR ELEMENTS       -- PAGE NO.
5
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY          MXY
                      VONT       VONB         SX           SY          SXY
                      TRESCAT    TRESCAB
      1      1        33.33       33.33   -20451.07   -20451.07        0.00
                    4908.26     4908.26        0.00        0.00        0.00
                    4908.26     4908.26
       TOP : SMAX=   -4908.26 SMIN=   -4908.26 TMAX=       0.00 ANGLE= 90.0
       BOTT: SMAX=    4908.26 SMIN=    4908.26 TMAX=       0.00 ANGLE= 90.0
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          4.908257E+03 -4.908257E+03  0.000000E+00  4.908257E+03  4.908257E+03
 PLATE NO.       1             1             1             1             1
 CASE  NO.       1             1             1             1             1
```

V. 2D Surface with Hole Edge Stress

To find the normal stress on the edge of a circular hole in the center of a rectangular plate.

Reference

Young, W. C., *Roark's Formulas for Stress and Strain*, McGraw-Hill Inc., 6th Edition, 1989 (Page 732, Type 7).

Problem

Find the normal stress on the edge of the circular hole for the plate shown, when an in-plane load causes tension. Use a one-quarter, doubly symmetric model.

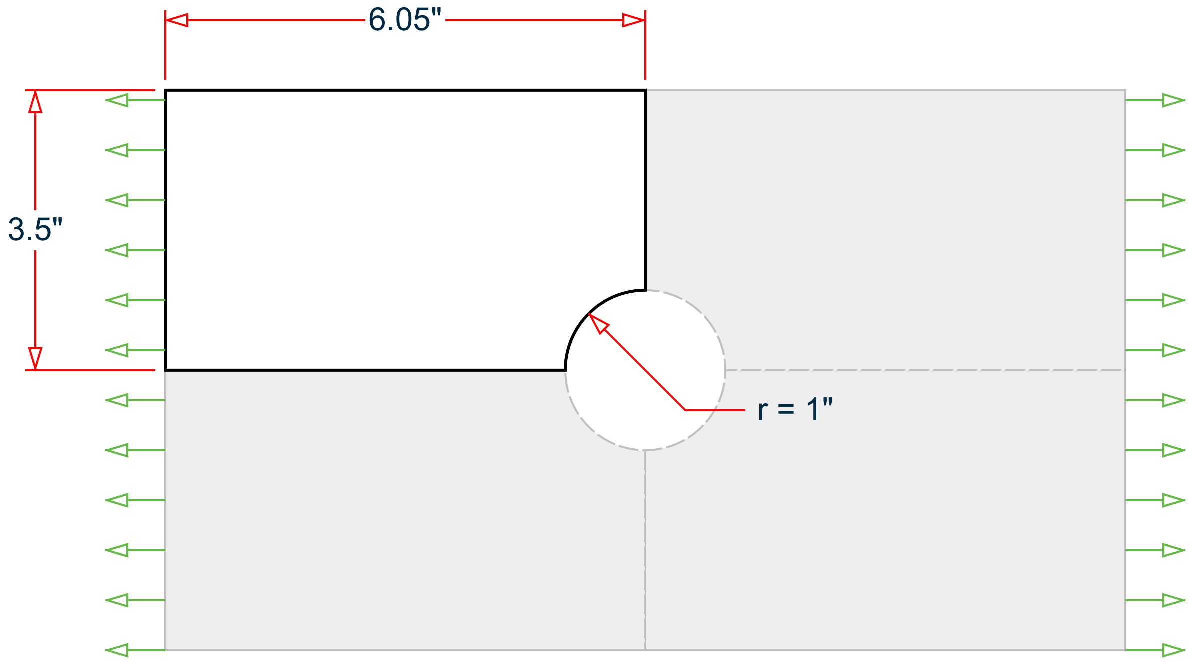


Figure 403: One quarter of rectangular plate with hole

E = 30,000.0 ksi

Size = 12.10 in × 7.0 in

Thickness = 0.1 in

Fillet radius = 1 in

Poisson's ratio = 0.3

P = 2,000 lbs

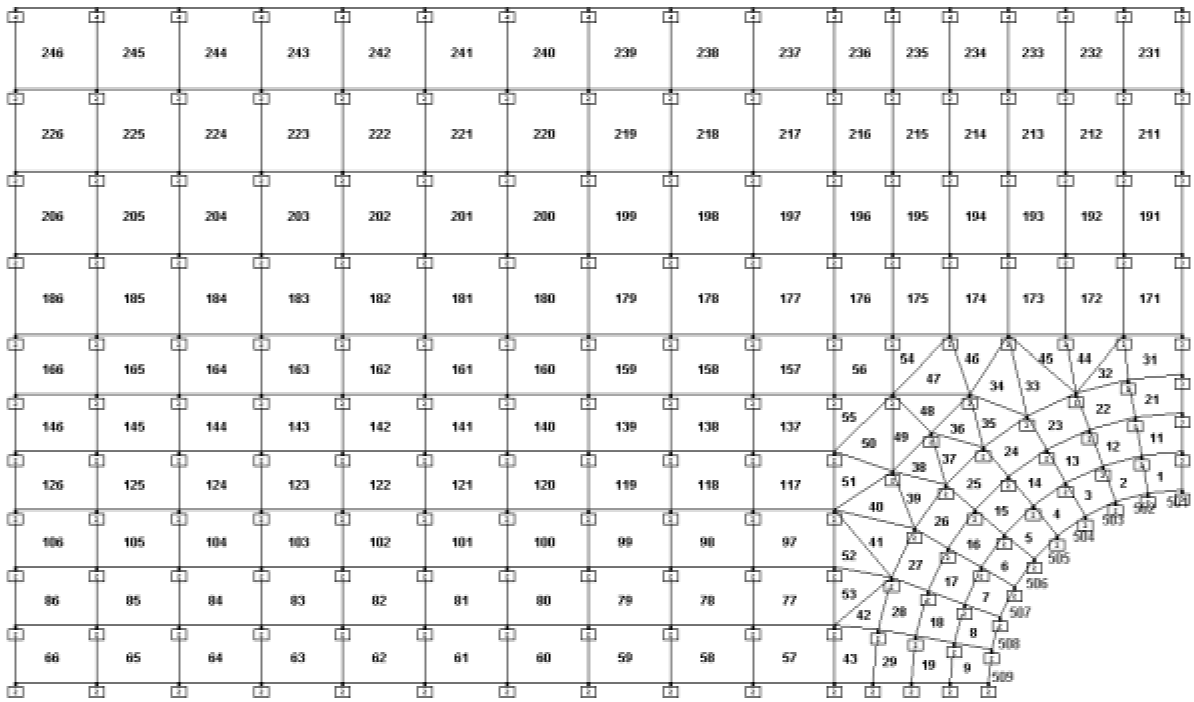


Figure 404: Model with nodes and elements labeled

Comparison

Table 427: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Stress on fillet (node 1, plate 1) (psi)	9.475	9.489	<1%

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Surface with Hole Edge Stress.STD` is typically installed with the program.

```
STAAD PLANE :STRESS CONCENTRATION IN A PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE:  "ROARK'S FORMULAS FOR STRESS AND STRAIN", YOUNG W. C.,
*              SIXTH EDITION, MCGRAW-HILL, PAGE 732.
*   7. GENERAL CIRCULAR HOLE IN CENTER OF A MEMBER OF RECTANGULAR CROSS
SECTION
```

```
*   1/4 MODEL DOUBLY SYMMETRIC
*   D = 7.0 INCHES  WIDTH
*   R = 1.0 INCH    HOLE RADIUS
*   T = 0.1 INCH    THICKNESS
*   L = 12.1 INCH   LENGTH
*  LOAD = 2000 POUNDS
* THEORETICAL RESULTS
*   AVERAGE STRESS AT ENDS  = 2857 PSI (7 INCH WIDTH)
*   AVERAGE NOMINAL STRESS AT CENTER = 4000 PSI  (7-5 INCH WIDTH)
*   K STRESS CONCENTRATION FACTOR = 2.3688
*   MAXIMUM STRESS AT EDGE OF HOLE = 2.3688 * 4000 = 9475 PSI
* STAAD RESULTS
*   MAXIMUM STRESS AT EDGE OF HOLE = 9489 PSI
*   K STRESS CONCENTRATION FACTOR = 2.37225
*   PERCENT ERROR = 0.15%
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 1 0; 2 -0.174 0.985 0; 3 -0.342 0.94 0; 4 -0.5 0.866 0;
5 -0.643 0.766 0; 6 -0.766 0.643 0; 7 -0.866 0.5 0; 8 -0.94 0.342 0;
9 -0.985 0.174 0; 10 -1 0 0; 11 0 1.2 0; 12 -0.208 1.182 0;
13 -0.41 1.128 0; 14 -0.6 1.039 0; 15 -0.771 0.919 0; 16 -0.919 0.771 0;
17 -1.039 0.6 0; 18 -1.128 0.41 0; 19 -1.182 0.208 0; 20 -1.2 0 0;
21 0 1.4 0; 22 -0.243 1.379 0; 23 -0.479 1.316 0; 24 -0.7 1.212 0;
25 -0.9 1.072 0; 26 -1.072 0.9 0; 27 -1.212 0.7 0; 28 -1.316 0.479 0;
29 -1.379 0.243 0; 30 -1.4 0 0; 31 0 1.6 0; 32 -0.278 1.576 0;
33 -0.547 1.504 0; 34 -0.8 1.386 0; 35 -1.028 1.226 0;
36 -1.226 1.028 0; 37 -1.386 0.8 0; 38 -1.504 0.547 0;
39 -1.576 0.278 0; 40 -1.6 0 0; 41 -1.101 1.5 0; 42 -1.5 1.101 0;
43 -1.5 1.5 0; 44 0 1.8 0; 45 -0.3 1.8 0; 46 -0.6 1.8 0; 47 -0.9 1.8 0;
48 -1.2 1.8 0; 49 -1.5 1.8 0; 50 0 2.225 0; 51 -0.3 2.225 0;
52 -0.6 2.225 0; 53 -0.9 2.225 0; 54 -1.2 2.225 0; 55 -1.5 2.225 0;
56 0 2.65 0; 57 -0.3 2.65 0; 58 -0.6 2.65 0; 59 -0.9 2.65 0;
60 -1.2 2.65 0; 61 -1.5 2.65 0; 62 0 3.075 0; 63 -0.3 3.075 0;
64 -0.6 3.075 0; 65 -0.9 3.075 0; 66 -1.2 3.075 0; 67 -1.5 3.075 0;
68 0 3.5 0; 69 -0.3 3.5 0; 70 -0.6 3.5 0; 71 -0.9 3.5 0; 72 -1.2 3.5 0;
73 -1.5 3.5 0; 74 -1.8 0 0; 75 -2.225 0 0; 76 -2.65 0 0; 77 -3.075 0 0;
78 -3.5 0 0; 79 -3.925 0 0; 80 -4.35 0 0; 81 -4.775 0 0; 82 -5.2 0 0;
83 -5.625 0 0; 84 -6.05 0 0; 85 -1.8 0.3 0; 86 -2.225 0.3 0;
87 -2.65 0.3 0; 88 -3.075 0.3 0; 89 -3.5 0.3 0; 90 -3.925 0.3 0;
91 -4.35 0.3 0; 92 -4.775 0.3 0; 93 -5.2 0.3 0; 94 -5.625 0.3 0;
95 -6.05 0.3 0; 96 -1.8 0.6 0; 97 -2.225 0.6 0; 98 -2.65 0.6 0;
99 -3.075 0.6 0; 100 -3.5 0.6 0; 101 -3.925 0.6 0; 102 -4.35 0.6 0;
103 -4.775 0.6 0; 104 -5.2 0.6 0; 105 -5.625 0.6 0; 106 -6.05 0.6 0;
107 -1.8 0.9 0; 108 -2.225 0.9 0; 109 -2.65 0.9 0; 110 -3.075 0.9 0;
111 -3.5 0.9 0; 112 -3.925 0.9 0; 113 -4.35 0.9 0; 114 -4.775 0.9 0;
115 -5.2 0.9 0; 116 -5.625 0.9 0; 117 -6.05 0.9 0; 118 -1.8 1.2 0;
119 -2.225 1.2 0; 120 -2.65 1.2 0; 121 -3.075 1.2 0; 122 -3.5 1.2 0;
123 -3.925 1.2 0; 124 -4.35 1.2 0; 125 -4.775 1.2 0; 126 -5.2 1.2 0;
127 -5.625 1.2 0; 128 -6.05 1.2 0; 129 -1.8 1.5 0; 130 -2.225 1.5 0;
131 -2.65 1.5 0; 132 -3.075 1.5 0; 133 -3.5 1.5 0; 134 -3.925 1.5 0;
135 -4.35 1.5 0; 136 -4.775 1.5 0; 137 -5.2 1.5 0; 138 -5.625 1.5 0;
139 -6.05 1.5 0; 140 -1.8 1.8 0; 141 -2.225 1.8 0; 142 -2.65 1.8 0;
143 -3.075 1.8 0; 144 -3.5 1.8 0; 145 -3.925 1.8 0; 146 -4.35 1.8 0;
147 -4.775 1.8 0; 148 -5.2 1.8 0; 149 -5.625 1.8 0; 150 -6.05 1.8 0;
151 -1.8 2.225 0; 152 -2.225 2.225 0; 153 -2.65 2.225 0;
154 -3.075 2.225 0; 155 -3.5 2.225 0; 156 -3.925 2.225 0;
157 -4.35 2.225 0; 158 -4.775 2.225 0; 159 -5.2 2.225 0;
```

```
160 -5.625 2.225 0; 161 -6.05 2.225 0; 162 -1.8 2.65 0;
163 -2.225 2.65 0; 164 -2.65 2.65 0; 165 -3.075 2.65 0; 166 -3.5 2.65 0;
167 -3.925 2.65 0; 168 -4.35 2.65 0; 169 -4.775 2.65 0; 170 -5.2 2.65 0;
171 -5.625 2.65 0; 172 -6.05 2.65 0; 173 -1.8 3.075 0;
174 -2.225 3.075 0; 175 -2.65 3.075 0; 176 -3.075 3.075 0;
177 -3.5 3.075 0; 178 -3.925 3.075 0; 179 -4.35 3.075 0;
180 -4.775 3.075 0; 181 -5.2 3.075 0; 182 -5.625 3.075 0;
183 -6.05 3.075 0; 184 -1.8 3.5 0; 185 -2.225 3.5 0; 186 -2.65 3.5 0;
187 -3.075 3.5 0; 188 -3.5 3.5 0; 189 -3.925 3.5 0; 190 -4.35 3.5 0;
191 -4.775 3.5 0; 192 -5.2 3.5 0; 193 -5.625 3.5 0; 194 -6.05 3.5 0;
201 -1.301 1.301 0;
ELEMENT INCIDENCES SHELL
1 1 11 12 2; 2 2 12 13 3; 3 3 13 14 4; 4 4 14 15 5; 5 5 15 16 6;
6 6 16 17 7; 7 7 17 18 8; 8 8 18 19 9; 9 9 19 20 10; 11 11 21 22 12;
12 12 22 23 13; 13 13 23 24 14; 14 14 24 25 15; 15 15 25 26 16;
16 16 26 27 17; 17 17 27 28 18; 18 18 28 29 19; 19 19 29 30 20;
21 21 31 32 22; 22 22 32 33 23; 23 23 33 34 24; 24 24 34 35 25;
25 25 35 36 26; 26 26 36 37 27; 27 27 37 38 28; 28 28 38 39 29;
29 29 39 40 30; 31 31 44 45 32; 32 32 45 33; 33 33 47 34; 34 34 47 41;
35 34 41 35; 36 35 41 201; 37 35 201 36; 38 36 201 42; 39 36 42 37;
40 37 42 107; 41 37 107 38; 42 38 85 39; 43 39 85 74 40; 44 45 46 33;
45 46 47 33; 46 47 48 41; 47 41 48 43; 48 41 43 201; 49 201 43 42;
50 42 43 118; 51 42 118 107; 52 38 107 96; 53 38 96 85; 54 48 49 43;
55 43 129 118; 56 43 49 140 129; 57 74 85 86 75; 58 75 86 87 76;
59 76 87 88 77; 60 77 88 89 78; 61 78 89 90 79; 62 79 90 91 80;
63 80 91 92 81; 64 81 92 93 82; 65 82 93 94 83; 66 83 94 95 84;
77 85 96 97 86; 78 86 97 98 87; 79 87 98 99 88; 80 88 99 100 89;
81 89 100 101 90; 82 90 101 102 91; 83 91 102 103 92; 84 92 103 104 93;
85 93 104 105 94; 86 94 105 106 95; 97 96 107 108 97; 98 97 108 109 98;
99 98 109 110 99; 100 99 110 111 100; 101 100 111 112 101;
102 101 112 113 102; 103 102 113 114 103; 104 103 114 115 104;
105 104 115 116 105; 106 105 116 117 106; 117 107 118 119 108;
118 108 119 120 109; 119 109 120 121 110; 120 110 121 122 111;
121 111 122 123 112; 122 112 123 124 113; 123 113 124 125 114;
124 114 125 126 115; 125 115 126 127 116; 126 116 127 128 117;
137 118 129 130 119; 138 119 130 131 120; 139 120 131 132 121;
140 121 132 133 122; 141 122 133 134 123; 142 123 134 135 124;
143 124 135 136 125; 144 125 136 137 126; 145 126 137 138 127;
146 127 138 139 128; 157 129 140 141 130; 158 130 141 142 131;
159 131 142 143 132; 160 132 143 144 133; 161 133 144 145 134;
162 134 145 146 135; 163 135 146 147 136; 164 136 147 148 137;
165 137 148 149 138; 166 138 149 150 139; 171 44 50 51 45;
172 45 51 52 46; 173 46 52 53 47; 174 47 53 54 48; 175 48 54 55 49;
176 49 55 151 140; 177 140 151 152 141; 178 141 152 153 142;
179 142 153 154 143; 180 143 154 155 144; 181 144 155 156 145;
182 145 156 157 146; 183 146 157 158 147; 184 147 158 159 148;
185 148 159 160 149; 186 149 160 161 150; 191 50 56 57 51;
192 51 57 58 52; 193 52 58 59 53; 194 53 59 60 54; 195 54 60 61 55;
196 55 61 162 151; 197 151 162 163 152; 198 152 163 164 153;
199 153 164 165 154; 200 154 165 166 155; 201 155 166 167 156;
202 156 167 168 157; 203 157 168 169 158; 204 158 169 170 159;
205 159 170 171 160; 206 160 171 172 161; 211 56 62 63 57;
212 57 63 64 58; 213 58 64 65 59; 214 59 65 66 60; 215 60 66 67 61;
216 61 67 173 162; 217 162 173 174 163; 218 163 174 175 164;
219 164 175 176 165; 220 165 176 177 166; 221 166 177 178 167;
222 167 178 179 168; 223 168 179 180 169; 224 169 180 181 170;
225 170 181 182 171; 226 171 182 183 172; 231 62 68 69 63;
232 63 69 70 64; 233 64 70 71 65; 234 65 71 72 66; 235 66 72 73 67;
```

```
236 67 73 184 173; 237 173 184 185 174; 238 174 185 186 175;
239 175 186 187 176; 240 176 187 188 177; 241 177 188 189 178;
242 178 189 190 179; 243 179 190 191 180; 244 180 191 192 181;
245 181 192 193 182; 246 182 193 194 183;
ELEMENT PROPERTY
1 TO 9 11 TO 19 21 TO 29 31 TO 66 77 TO 86 THICKNESS 0.1
97 TO 106 117 TO 126 137 TO 146 157 TO 166 171 TO 186 191 TO 205 -
206 THICKNESS 0.1
211 TO 226 231 TO 246 THICKNESS 0.1
ELEMENT PLANE STRESS
1 TO 9 11 TO 19 21 TO 29 31 TO 66 77 TO 86 97 TO 106 117 TO 126 137 -
138 TO 146 157 TO 166 171 TO 186 191 TO 206 211 TO 226 231 TO 246
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
10 20 30 40 74 TO 84 FIXED BUT FX
1 11 21 31 44 50 56 62 68 FIXED BUT FY
LOAD 1 TENSILE LOAD
JOINT LOAD
84 FX -42.857
95 106 117 128 139 FX -85.714
150 FX -103.571
161 172 183 FX -121.428
194 FX -60.714
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT JOINT DISPLACEMENTS LIST 6 84
PRINT ELEMENT JOINT STRESSES LIST 1
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      6    1   -0.00024  -0.00008   0.00000   0.00000   0.00000  -0.00029
     84    1   -0.00066   0.00000   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
   156. PRINT ELEMENT JOINT STRESSES LIST 1
  ELEMENT   JOINT    STRESSES LIST
      :STRESS CONCENTRATION IN A PLATE                         -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
      1      1         0.00        0.00        0.00        0.00        0.00
                    7461.40     7461.40      650.49     7726.25     -439.78
                    7753.48     7753.48
       TOP : SMAX=    7753.48 SMIN=     623.26 TMAX=    3565.11 ANGLE=-86.5
```

```
      BOTT: SMAX=    7753.48 SMIN=     623.26 TMAX=    3565.11 ANGLE=-86.5
        JOINT        0.00        0.00        0.00        0.00        0.00
          1       9229.88     9229.88      751.63     9488.87     -757.04
      TOP : SMAX=    9553.97 SMIN=     686.52 TMAX=    4433.73 ANGLE=-85.1
      BOTT: SMAX=    9553.97 SMIN=     686.52 TMAX=    4433.73 ANGLE=-85.1
        JOINT        0.00        0.00        0.00        0.00        0.00
         11       5912.86     5912.86      608.93     6174.80     -272.64
      TOP : SMAX=    6188.12 SMIN=     595.61 TMAX=    2796.26 ANGLE=-87.2
      BOTT: SMAX=    6188.12 SMIN=     595.61 TMAX=    2796.26 ANGLE=-87.2
        JOINT        0.00        0.00        0.00        0.00        0.00
         12       5802.41     5802.41      548.98     6051.20     -154.55
      TOP : SMAX=    6055.53 SMIN=     544.64 TMAX=    2755.44 ANGLE=-88.4
      BOTT: SMAX=    6055.53 SMIN=     544.64 TMAX=    2755.44 ANGLE=-88.4
        JOINT        0.00        0.00        0.00        0.00        0.00
          2       8940.02     8940.02      666.87     9167.41     -719.10
      TOP : SMAX=    9227.81 SMIN=     606.47 TMAX=    4310.67 ANGLE=-85.2
      BOTT: SMAX=    9227.81 SMIN=     606.47 TMAX=    4310.67 ANGLE=-85.2
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         9.553974E+03  5.446445E+02  4.433728E+03  7.461400E+03  7.753480E+03
PLATE NO.       1             1             1             1             1
CASE  NO.       1             1             1             1             1
```

V. 2D Tapered Beam In-Plane Stress

To find element stress due to joint load at the fixed end of a tapered plate with one end fixed.

Reference

Crandall, S.H., & Dahl, N.C., *An Introduction to the Mechanics of Solids*, McGraw – Hill, Inc., 1959.

Problem

The tapered plate structure is loaded at the free end. Calculate the maximum stress at the midspan.

E = 30,000.0 ksi

Thickness = 2 in

Poisson's ratio = 0.2

P = 4 kips

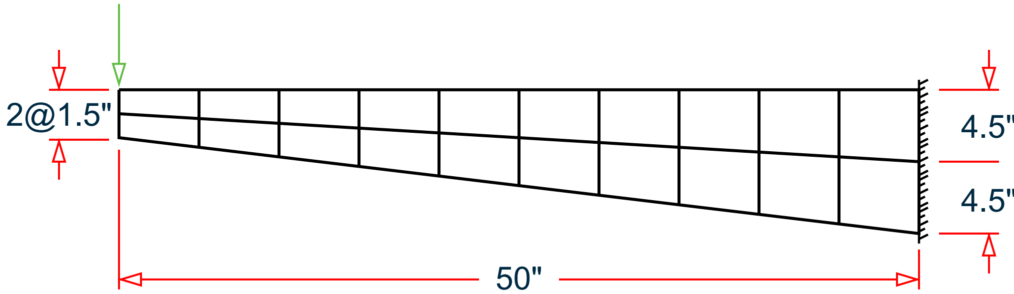


Figure 405: Model

Comparison

The STAAD.Pro result is taken as average of stress in elements 9 and 11 at node 16 = *0.5(8,333.35 + 8,359.62) = 8,346.5*.

Table 428: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Maximum stress at the center (psi)	8,333	8,346.5	0.2%

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Tapered Beam In-Plane Stress.STD` is typically installed with the program.

```
STAAD SPACE :A TAPERED BEAM WITH PLATE ELEMENTS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE: CRANDALL, S. H., AND DAHL, N.C., "AN INTRODUCTION TO THE
*             MECHANICS OF SOLIDS," MCGRAW-HILL BOOK CO., INC.,
*             NEW YORK, 1978
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 50 0 0; 2 50 -4.5 0; 3 50 -9 0; 4 45 0 0; 5 45 -4.2 0; 6 45 -8.4 0; 7 40 0
0;
8 40 -3.9 0; 9 40 -7.8 0; 10 35 0 0; 11 35 -3.6 0; 12 35 -7.2 0; 13 30 0 0;
14 30 -3.3 0; 15 30 -6.6 0; 16 25 0 0; 17 25 -3 0; 18 25 -6 0; 19 20 0 0;
20 20 -2.7 0; 21 20 -5.4 0; 22 15 0 0; 23 15 -2.4 0; 24 15 -4.8 0; 25 10 0 0;
26 10 -2.1 0; 27 10 -4.2 0; 28 5 0 0; 29 5 -1.8 0; 30 5 -3.6 0; 31 0 0 0;
32 0 -1.5 0; 33 0 -3 0;
ELEMENT INCIDENCES SHELL
```

```
1 4 5 2 1; 2 5 6 3 2; 3 7 8 5 4; 4 8 9 6 5; 5 10 11 8 7; 6 11 12 9 8;
7 13 14 11 10; 8 14 15 12 11; 9 16 17 14 13; 10 17 18 15 14; 11 19 20 17 16;
12 20 21 18 17; 13 22 23 20 19; 14 23 24 21 20; 15 25 26 23 22; 16 26 27 24
23;
17 28 29 26 25; 18 29 30 27 26; 19 31 32 29 28; 20 32 33 30 29;
ELEMENT PROPERTY
1 TO 20 THICKNESS 2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 3 FIXED
LOAD 1 POINT LOAD
JOINT LOAD
31 FY -4000
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESSES LIST 9 11
FINISH
```

STAAD Output

```
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY         MX          MY          MXY
                      VONT       VONB        SX          SY          SXY
                      TRESCAT    TRESCAB
      9      1        0.00        0.00        0.00        0.00        0.00
                    4157.59     4157.59       -4.32     4153.58      -71.59
                    4160.36     4160.36
       TOP : SMAX=    4154.81 SMIN=      -5.55 TMAX=    2080.18 ANGLE=-89.0
       BOTT: SMAX=    4154.81 SMIN=      -5.55 TMAX=    2080.18 ANGLE=-89.0
         JOINT        0.00        0.00        0.00        0.00        0.00
           16       8372.99     8372.99      -78.55     8333.17       38.19
       TOP : SMAX=    8333.35 SMIN=     -78.72 TMAX=    4206.03 ANGLE= 89.7
       BOTT: SMAX=    8333.35 SMIN=     -78.72 TMAX=    4206.03 ANGLE= 89.7
         JOINT        0.00        0.00        0.00        0.00        0.00
           17        343.34      343.34       42.49        3.59     -196.82
       TOP : SMAX=     220.82 SMIN=    -174.74 TMAX=     197.78 ANGLE=-42.2
       BOTT: SMAX=     220.82 SMIN=    -174.74 TMAX=     197.78 ANGLE=-42.2
         JOINT        0.00        0.00        0.00        0.00        0.00
           14        295.01      295.01     -107.35      -73.48     -161.24
       TOP : SMAX=      71.72 SMIN=    -252.54 TMAX=     162.13 ANGLE=-48.0
       BOTT: SMAX=      71.72 SMIN=    -252.54 TMAX=     162.13 ANGLE=-48.0
         JOINT        0.00        0.00        0.00        0.00        0.00
           13       8295.66     8295.66       93.65     8341.38       62.12
       TOP : SMAX=    8341.85 SMIN=      93.18 TMAX=    4124.34 ANGLE= 89.6
       BOTT: SMAX=    8341.85 SMIN=      93.18 TMAX=    4124.34 ANGLE= 89.6
     11      1        0.00        0.00        0.00        0.00        0.00
                    4155.63     4155.63       -4.64     4149.10     -107.91
                    4159.35     4159.35
       TOP : SMAX=    4151.91 SMIN=      -7.44 TMAX=    2079.67 ANGLE=-88.5
```

```
       BOTT: SMAX=    4151.91 SMIN=      -7.44 TMAX=    2079.67 ANGLE=-88.5
         JOINT         0.00        0.00        0.00        0.00        0.00
           19       8323.84     8323.84      -72.93     8286.88       37.52
       TOP : SMAX=    8287.05 SMIN=     -73.10 TMAX=    4180.08 ANGLE= 89.7
       BOTT: SMAX=    8287.05 SMIN=     -73.10 TMAX=    4180.08 ANGLE= 89.7
         JOINT         0.00        0.00        0.00        0.00        0.00
           20        410.28      410.28       69.05      -21.19     -232.13
       TOP : SMAX=     260.40 SMIN=    -212.54 TMAX=     236.47 ANGLE=-39.5
       BOTT: SMAX=     260.40 SMIN=    -212.54 TMAX=     236.47 ANGLE=-39.5
         JOINT         0.00        0.00        0.00        0.00        0.00
           17        377.42      377.42     -132.29      -39.10     -207.04
       TOP : SMAX=     126.52 SMIN=    -297.91 TMAX=     212.21 ANGLE=-51.3
       BOTT: SMAX=     126.52 SMIN=    -297.91 TMAX=     212.21 ANGLE=-51.3
         JOINT         0.00        0.00        0.00        0.00        0.00
           16       8319.08     8319.08       82.05     8359.26       54.91
       TOP : SMAX=    8359.62 SMIN=      81.68 TMAX=    4138.97 ANGLE= 89.6
       BOTT: SMAX=    8359.62 SMIN=      81.68 TMAX=    4138.97 ANGLE= 89.6
      :A TAPERED BEAM WITH PLATE ELEMENTS                      -- PAGE NO.
4
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          8.359622E+03 -2.979081E+02  4.206035E+03  4.157588E+03  4.160359E+03
 PLATE NO.      11            11             9             9             9
 CASE  NO.       1             1             1             1             1
```

V. 2D Triangular Surface with Thermal Load

A simply-supported equilateral triangle is subjected to a linear thermal gradient. The deflections and bending moments are calculated.

Reference

Timoshenko, S. and Woinowsky-Krieger, S., *Theory of Plates and Shells*, McGraw Hill, 1959, pp. 92-97.

Problem

The purpose of this example is to demonstrate the ability of STAAD to calculate the response of a structure when thermally-loaded plate elements are utilized. The STAAD.Pro model considers one-half of the plate, modeled using 49 quad elements and seven tri plate elements. For specifying inclined supports, an auxiliary coordinate system is defined along the edge (see second figure below) not parallel to a global axis.

$E = 10\times(10)^6$ lb/in^2

$\upsilon = 0.3$

$\alpha = 12\ (10)^{-6}$ in/in °F

$\Delta T = T_{top} - T_{bottom} = 4500$

$t = 0.1$ inches

$a = 3.0$ inches

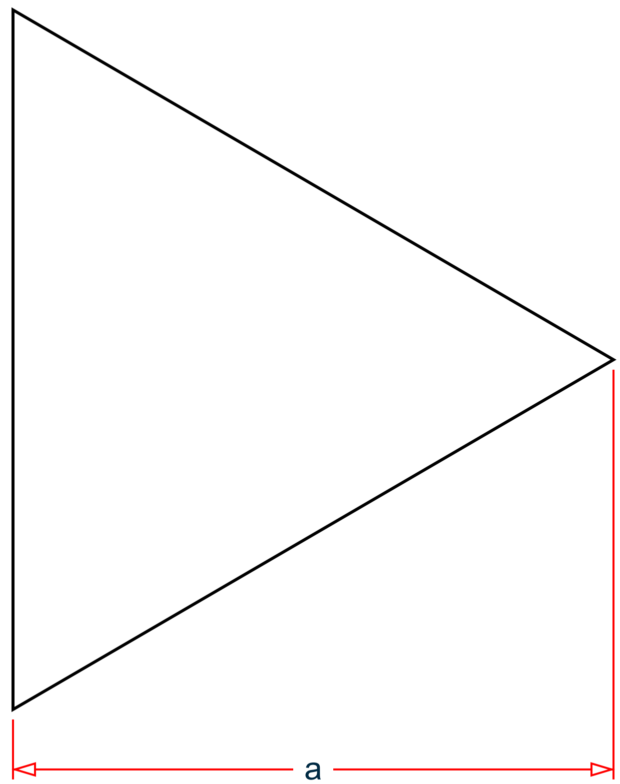


Figure 406: Simply-supported, equilateral triangle with thermal load

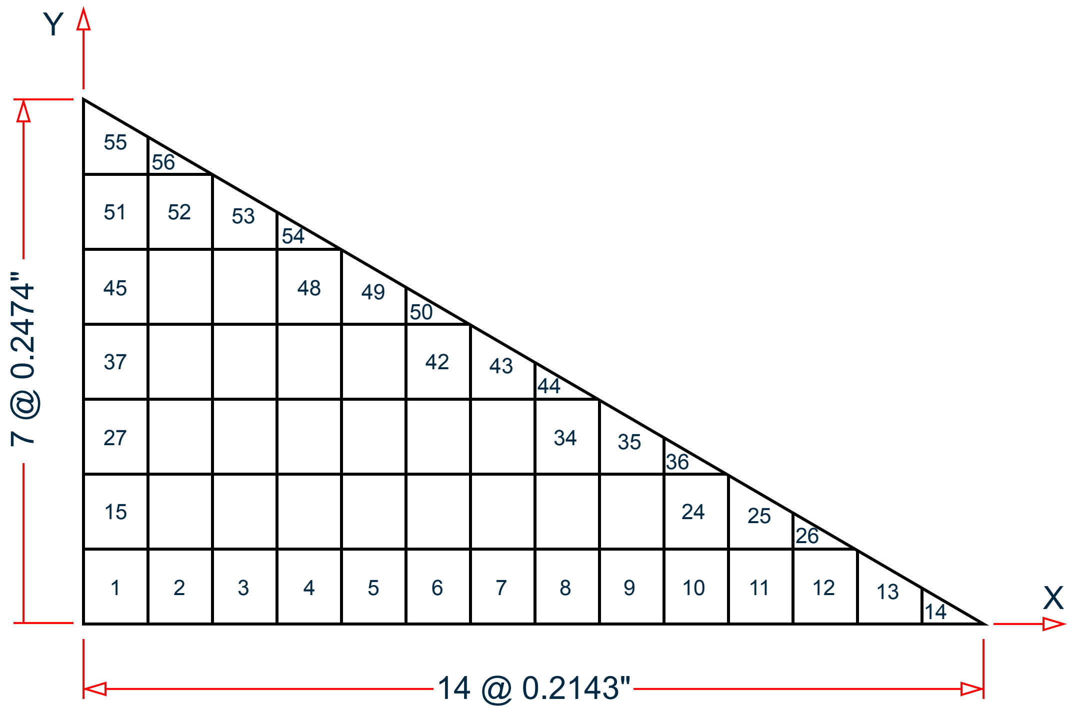


Figure 407: Finite element model of a simply-supported, equilateral triangle

Theoretical Solution

From page 96 of the reference:

$$M_x = -\frac{\alpha \Delta T E t^2}{24}\left(1 + \frac{3x'}{a}\right)$$

$$= -\frac{12(10)^{-6}(450.0)10(10)^6(0.1)^2}{24}\left(1 + \frac{3x'}{3}\right) = -22.5\left(1 + x'\right)$$

$$M_y = -\frac{\alpha \Delta T E t^2}{24}\left(1 - \frac{3x'}{a}\right) = -22.5\left(1 - x'\right)$$

$$M_{xy} = \frac{\alpha \Delta T E t^2 y}{8a}$$

$$= \frac{12(10)^{-6}(450.0)10(10)^6(0.1)^2}{8(3)}y = 22.5y$$

$$w = \frac{\alpha \Delta T(1+\nu)}{4at}\left(x'^3 - 3y^2x' - ax'^2 - ay^2 + \frac{4a^3}{27}\right)$$

$$= 0.00585\left(x'^3 - 3y^2x' - 3x'^2 - 3y^2 + 4\right)$$

The origin of the x' axis in the reference is located at a/3, or 1 inch from the edge. Therefore:

x' = x-1

Upon substitution:

Mx = -22.5x

$My = -22.5 (2 - x)$

$Mxy = 22.5y$

$w = 0.00585 (x3 – 6x^2 + 9x – 3y^2x)$

Comparison

Table 429: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Z deflection at center of element along x-axis (in)	Node 6	0.02331	0.02331	none
Maximum (magnitude) bending moment along x-axis (in·lb/in)	Mx at Element 8	-36.16	-36.15	none
	Mx at Element 14	-64.29	-65.64	2.1%
	My at Element 1	-42.59	-42.59	none
	My at Element 8	-8.839	-8.838	none
Twisting moment along y-axis at center of elements (in·lb/in)	Mxy at Element 55	36.19	36.06	none

The results show excellent agreement for displacements and for quad plate element stresses except in the few instances where the magnitude of the answers is very small. The tri plate elements are lower order elements, and it is therefore not unusual for these elements to show poorer agreement than quads where mesh size is smaller.

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\2D Triangular Surface with Thermal Load.STD` is typically installed with the program.

```
STAAD SPACE :SIMPLY SUPPORTED PLATE THERMAL
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* AN EQUILATERAL TRIANGLE WITH LINEAR THERMAL GRADIENT
*
*     EXPECTED ANSWERS
*         DEFLECTION OF NODE 6, IN THE X3 DIRECTION
*                THEORY      .02331
*                STAAD       .02331
```

```
*
*         INTERNAL MOMENTS IN QUAD PLATE NO 8
*                               MX           MY
*                THEORY     -36.16      -8.864
*                STAAD      -36.15      -8.84
*
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0.214286 0 0; 3 0.428571 0 0; 4 0.642857 0 0; 5 0.857143 0 0;
6 1.07143 0 0; 7 1.28571 0 0; 8 1.5 0 0; 9 1.71429 0 0; 10 1.92857 0 0;
11 2.14286 0 0; 12 2.35714 0 0; 13 2.57143 0 0; 14 2.78571 0 0; 15 3 0 0;
16 2.78571 0.123718 0; 17 0 0.247436 0; 18 0.214286 0.247436 0;
19 0.428571 0.247436 0; 20 0.642857 0.247436 0; 21 0.857143 0.247436 0;
22 1.07143 0.247436 0; 23 1.28571 0.247436 0; 24 1.5 0.247436 0;
25 1.71429 0.247436 0; 26 1.92857 0.247436 0; 27 2.14286 0.247436 0;
28 2.35714 0.247436 0; 29 2.57143 0.247436 0; 30 2.35714 0.371154 0;
31 0 0.494872 0; 32 0.214286 0.494872 0; 33 0.428571 0.494872 0;
34 0.642857 0.494872 0; 35 0.857143 0.494872 0; 36 1.07143 0.494872 0;
37 1.28571 0.494872 0; 38 1.5 0.494872 0; 39 1.71429 0.494872 0;
40 1.92857 0.494872 0; 41 2.14286 0.494872 0; 42 1.92857 0.61859 0;
43 0 0.742307 0; 44 0.214286 0.742307 0; 45 0.428571 0.742307 0;
46 0.642857 0.742307 0; 47 0.857143 0.742307 0; 48 1.07143 0.742307 0;
49 1.28571 0.742307 0; 50 1.5 0.742307 0; 51 1.71429 0.742307 0;
52 1.5 0.866025 0; 53 0 0.989743 0; 54 0.214286 0.989743 0;
55 0.428571 0.989743 0; 56 0.642857 0.989743 0; 57 0.857143 0.989743 0;
58 1.07143 0.989743 0; 59 1.28571 0.989743 0; 60 1.07143 1.11346 0;
61 0 1.23718 0; 62 0.214286 1.23718 0; 63 0.428571 1.23718 0;
64 0.642857 1.23718 0; 65 0.857143 1.23718 0; 66 0.642857 1.3609 0;
67 0 1.48461 0; 68 0.214285 1.48461 0; 69 0.428571 1.48461 0;
70 0.214286 1.60833 0; 71 0 1.73205 0;
*72 3 1.73205 0
ELEMENT INCIDENCES SHELL
1 1 2 18 17; 2 2 3 19 18; 3 3 4 20 19; 4 4 5 21 20; 5 5 6 22 21; 6 6 7 23 22;
7 7 8 24 23; 8 8 9 25 24; 9 9 10 26 25; 10 10 11 27 26; 11 11 12 28 27;
12 12 13 29 28; 13 13 14 16 29; 14 14 15 16; 15 17 18 32 31; 16 18 19 33 32;
17 19 20 34 33; 18 20 21 35 34; 19 21 22 36 35; 20 22 23 37 36; 21 23 24 38
37;
22 24 25 39 38; 23 25 26 40 39; 24 26 27 41 40; 25 27 28 30 41; 26 28 29 30;
27 31 32 44 43; 28 32 33 45 44; 29 33 34 46 45; 30 34 35 47 46; 31 35 36 48
47;
32 36 37 49 48; 33 37 38 50 49; 34 38 39 51 50; 35 39 40 42 51; 36 40 41 42;
37 43 44 54 53; 38 44 45 55 54; 39 45 46 56 55; 40 46 47 57 56; 41 47 48 58
57;
42 48 49 59 58; 43 49 50 52 59; 44 50 51 52; 45 53 54 62 61; 46 54 55 63 62;
47 55 56 64 63; 48 56 57 65 64; 49 57 58 60 65; 50 58 59 60; 51 61 62 68 67;
52 62 63 69 68; 53 63 64 66 69; 54 64 65 66; 55 67 68 70 71; 56 68 69 70;
DEFINE MATERIAL START
ISOTROPIC ALUMINUM
E 1e+07
POISSON 0.3
ALPHA 1.2e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL ALUMINUM ALL
ELEMENT PROPERTY
1 TO 56 THICKNESS 0.1
SUPPORTS
```

```
16 29 30 41 42 51 52 59 60 65 66 69 70 INC REFJT 15 FIXED BUT FX MX MZ
2 TO 14 FIXED BUT FX FZ MY MZ
17 31 43 53 61 67 FIXED BUT FY MY MZ
1 FIXED BUT MY MZ
15 71 FIXED
LOAD 1 UNIFORM BENDING TEMPERATURE
TEMPERATURE LOAD
1 TO 56 TEMP 0 450
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT JOINT DISPLACEMENTS LIST 2 TO 14
PRINT ELEMENT JOINT STRESSES LIST 1 8 14 55
FINISH
```

STAAD Output

```
  JOINT    DISPLACE LIST     2
      :SIMPLY SUPPORTED PLATE THERMAL                          -- PAGE NO.
5
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      2    1    0.00000   0.00000   0.00973   0.00000  -0.03853   0.00000
      3    1    0.00000   0.00000   0.01658   0.00000  -0.02590   0.00000
      4    1    0.00000   0.00000   0.02090   0.00000  -0.01489   0.00000
      5    1    0.00000   0.00000   0.02303   0.00000  -0.00549   0.00000
      6    1    0.00000   0.00000   0.02331   0.00000   0.00230   0.00000
      7    1    0.00000   0.00000   0.02211   0.00000   0.00848   0.00000
      8    1    0.00000   0.00000   0.01975   0.00000   0.01305   0.00000
      9    1    0.00000   0.00000   0.01658   0.00000   0.01600   0.00000
     10    1    0.00000   0.00000   0.01295   0.00000   0.01735   0.00000
     11    1    0.00000   0.00000   0.00921   0.00000   0.01709   0.00000
     12    1    0.00000   0.00000   0.00570   0.00000   0.01518   0.00000
     13    1    0.00000   0.00000   0.00276   0.00000   0.01165   0.00000
     14    1    0.00000   0.00000   0.00075   0.00000   0.00672   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
    77. PRINT ELEMENT JOINT STRESSES LIST 1 8 14 55
  ELEMENT  JOINT    STRESSES LIST
      :SIMPLY SUPPORTED PLATE THERMAL                          -- PAGE NO.
6
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
      1      1        -0.01       -0.01       -2.41      -42.59        2.78
                   25029.83    25029.83        0.00        0.00        0.00
                   25668.88    25668.88
       TOP : SMAX=   -1331.23 SMIN=  -25668.88 TMAX=   12168.82 ANGLE=  3.9
       BOTT: SMAX=   25668.88 SMIN=    1331.23 TMAX=   12168.82 ANGLE=-86.1
         JOINT        -0.00       -0.01       -1.67      -45.50        1.93
             1     26888.39    26888.39        0.00        0.00        0.00
       TOP : SMAX=    -951.54 SMIN=  -27351.53 TMAX=   13200.00 ANGLE=  2.5
       BOTT: SMAX=   27351.53 SMIN=     951.54 TMAX=   13200.00 ANGLE=-87.5
         JOINT        -0.00       -0.00       -3.15      -39.68        1.93
             2     23007.69    23007.69        0.00        0.00        0.00
```

```
       TOP : SMAX=   -1829.24 SMIN=  -23867.71 TMAX=   11019.23 ANGLE=  3.0
       BOTT: SMAX=   23867.71 SMIN=    1829.24 TMAX=   11019.23 ANGLE=-87.0
         JOINT         -0.02        -0.00       -3.15      -39.68        3.64
           18      23229.73    23229.73        0.00        0.00        0.00
       TOP : SMAX=   -1675.24 SMIN=  -24022.00 TMAX=   11173.38 ANGLE=  5.6
       BOTT: SMAX=   24022.00 SMIN=    1675.24 TMAX=   11173.38 ANGLE=-84.4
         JOINT         -0.02        -0.01       -1.67      -45.50        3.64
           17      27078.78    27078.78        0.00        0.00        0.00
       TOP : SMAX=    -822.50 SMIN=  -27480.66 TMAX=   13329.08 ANGLE=  4.7
       BOTT: SMAX=   27480.66 SMIN=     822.50 TMAX=   13329.08 ANGLE=-85.3
      8      1          2.23         1.01      -36.15       -8.84        2.79
                   19799.74    19799.74        0.00        0.00        0.00
                   21861.08    21861.08
       TOP : SMAX=   -5134.01 SMIN=  -21861.08 TMAX=    8363.53 ANGLE= 84.2
       BOTT: SMAX=   21861.08 SMIN=    5134.01 TMAX=    8363.53 ANGLE= -5.8
         JOINT         -4.50        -1.02      -35.37      -11.73        1.92
            8      18829.74    18829.74        0.00        0.00        0.00
       TOP : SMAX=   -6945.23 SMIN=  -21315.87 TMAX=    7185.32 ANGLE= 85.4
       BOTT: SMAX=   21315.87 SMIN=    6945.23 TMAX=    7185.32 ANGLE= -4.6
         JOINT         -4.50         3.05      -36.95       -5.96        1.93
            9      20715.56    20715.56        0.00        0.00        0.00
       TOP : SMAX=   -3504.10 SMIN=  -22244.13 TMAX=    9370.01 ANGLE= 86.4
       BOTT: SMAX=   22244.13 SMIN=    3504.10 TMAX=    9370.01 ANGLE= -3.6
         JOINT          8.97         3.05      -36.79       -5.89        3.65
           25      20883.86    20883.86        0.00        0.00        0.00
       TOP : SMAX=   -3281.80 SMIN=  -22330.46 TMAX=    9524.33 ANGLE= 83.4
       BOTT: SMAX=   22330.46 SMIN=    3281.80 TMAX=    9524.33 ANGLE= -6.6
         JOINT          8.97        -1.02      -35.50      -11.77        3.64
           24      19168.57    19168.57        0.00        0.00        0.00
       TOP : SMAX=   -6732.25 SMIN=  -21626.52 TMAX=    7447.13 ANGLE= 81.5
       BOTT: SMAX=   21626.52 SMIN=    6732.25 TMAX=    7447.13 ANGLE= -8.5
     14      1        -57.76        58.83      -65.64       18.37        1.02
                   45910.85    45910.85        0.00        0.00        0.00
                   50421.84    50421.84
       TOP : SMAX=   11032.26 SMIN=  -39389.58 TMAX=   25210.92 ANGLE= 89.3
       BOTT: SMAX=   39389.58 SMIN=  -11032.26 TMAX=   25210.92 ANGLE= -0.7
      :SIMPLY SUPPORTED PLATE THERMAL                          -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT        -57.76        58.83      -65.22       18.13        1.02
           14      45571.45    45571.45        0.00        0.00        0.00
       TOP : SMAX=   10886.75 SMIN=  -39142.11 TMAX=   25014.43 ANGLE= 89.3
       BOTT: SMAX=   39142.11 SMIN=  -10886.75 TMAX=   25014.43 ANGLE= -0.7
         JOINT        -57.76        58.83      -66.46       18.13        1.02
           15      46298.03    46298.03        0.00        0.00        0.00
       TOP : SMAX=   10886.64 SMIN=  -39884.58 TMAX=   25385.61 ANGLE= 89.3
       BOTT: SMAX=   39884.58 SMIN=  -10886.64 TMAX=   25385.61 ANGLE= -0.7
         JOINT        -57.76        58.83      -65.22       18.86        1.02
           16      45864.36    45864.36        0.00        0.00        0.00
       TOP : SMAX=   11323.39 SMIN=  -39142.05 TMAX=   25232.72 ANGLE= 89.3
       BOTT: SMAX=   39142.05 SMIN=  -11323.39 TMAX=   25232.72 ANGLE= -0.7
     55      1         63.71       -48.81       -2.49      -42.60       36.06
                   44968.46    44968.46        0.00        0.00        0.00
```

```
                   49519.82    49519.82
      TOP : SMAX=   11232.03 SMIN=  -38287.79 TMAX=   24759.91 ANGLE= 30.5
      BOTT: SMAX=   38287.79 SMIN=  -11232.03 TMAX=   24759.91 ANGLE=-59.5
        JOINT      -130.44      -11.74       -0.31      -45.48       35.28
          67      45646.34    45646.34        0.00        0.00        0.00
      TOP : SMAX=   11394.66 SMIN=  -38869.58 TMAX=   25132.12 ANGLE= 28.7
      BOTT: SMAX=   38869.58 SMIN=  -11394.66 TMAX=   25132.12 ANGLE=-61.3
        JOINT       -17.92       11.58       -3.49      -38.96       35.48
          68      43142.03    43142.03        0.00        0.00        0.00
      TOP : SMAX=   11063.97 SMIN=  -36532.57 TMAX=   23798.27 ANGLE= 31.7
      BOTT: SMAX=   36532.57 SMIN=  -11063.97 TMAX=   23798.27 ANGLE=-58.3
        JOINT       165.93      -53.39       -2.02      -39.46       36.61
          70      44500.96    44500.96        0.00        0.00        0.00
      TOP : SMAX=   12222.83 SMIN=  -37112.28 TMAX=   24667.55 ANGLE= 31.5
      BOTT: SMAX=   37112.28 SMIN=  -12222.83 TMAX=   24667.55 ANGLE=-58.5
        JOINT       237.25     -141.67       -5.26      -48.10       36.69
          71      46961.48    46961.48        0.00        0.00        0.00
      TOP : SMAX=    9479.68 SMIN=  -41498.48 TMAX=   25489.08 ANGLE= 29.9
      BOTT: SMAX=   41498.48 SMIN=   -9479.68 TMAX=   25489.08 ANGLE=-60.1
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         4.149848E+04 -4.149848E+04  2.548908E+04  4.591085E+04  5.042184E+04
PLATE NO.      55            55            55            14            14
CASE  NO.       1             1             1             1             1
```

V. Cantilever Tube Stresses and Deflection

To find deflections and element stresses due to loads at the free end of a Cantilever beam of tubular section. The beam is modeled using plate/shell elements.

Reference

Timoshenko, S., *Strength of Materials, Part I, Elementary Theory and Problems*, 2nd Edition, Van Nostrand Company, 1940.

Problem

A cantilever beam is made of a tubular section. Using plate/shell elements calculate the deflection at the free end and axial stress at the center of the beam for the following free end loads:

$P = 1{,}000$ lb

$M_x = 2{,}000$ in·lb

$M_y = 2{,}500$ in·lb

$V = 1{,}000$ lb

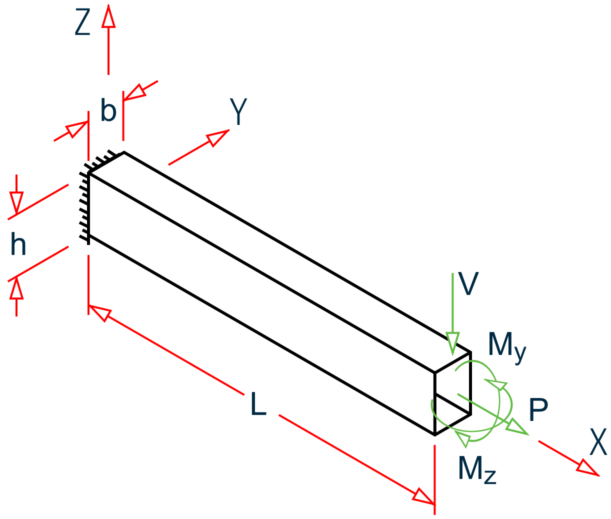


Figure 408: Cantilever beam modeled with elements

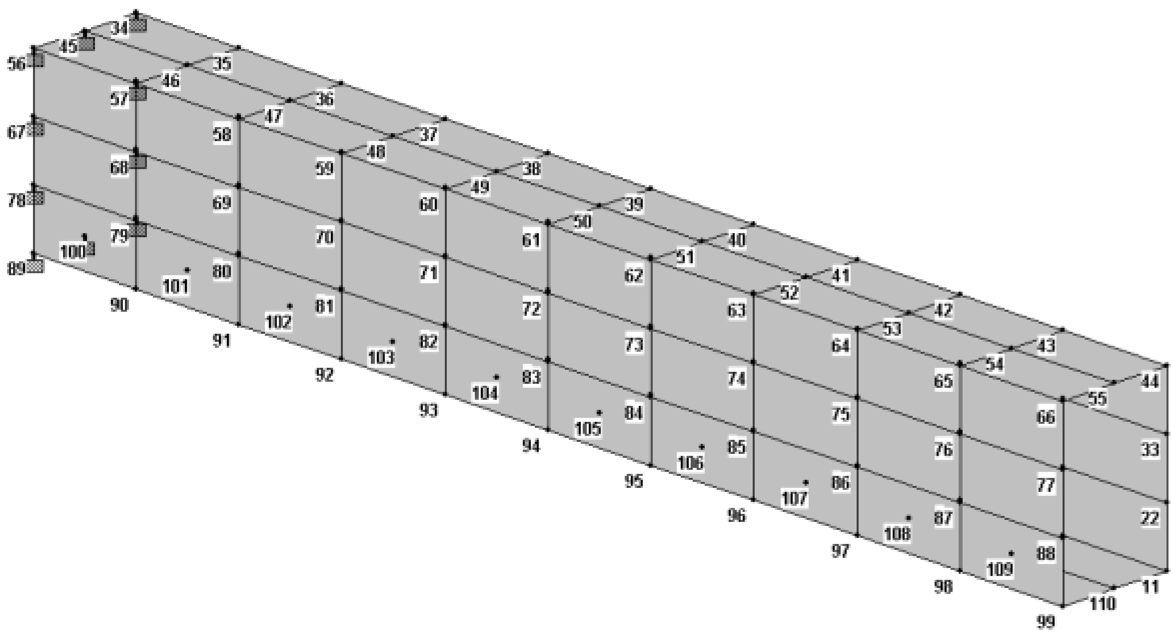


Figure 409: STAAD.Pro Model showing Node numbers

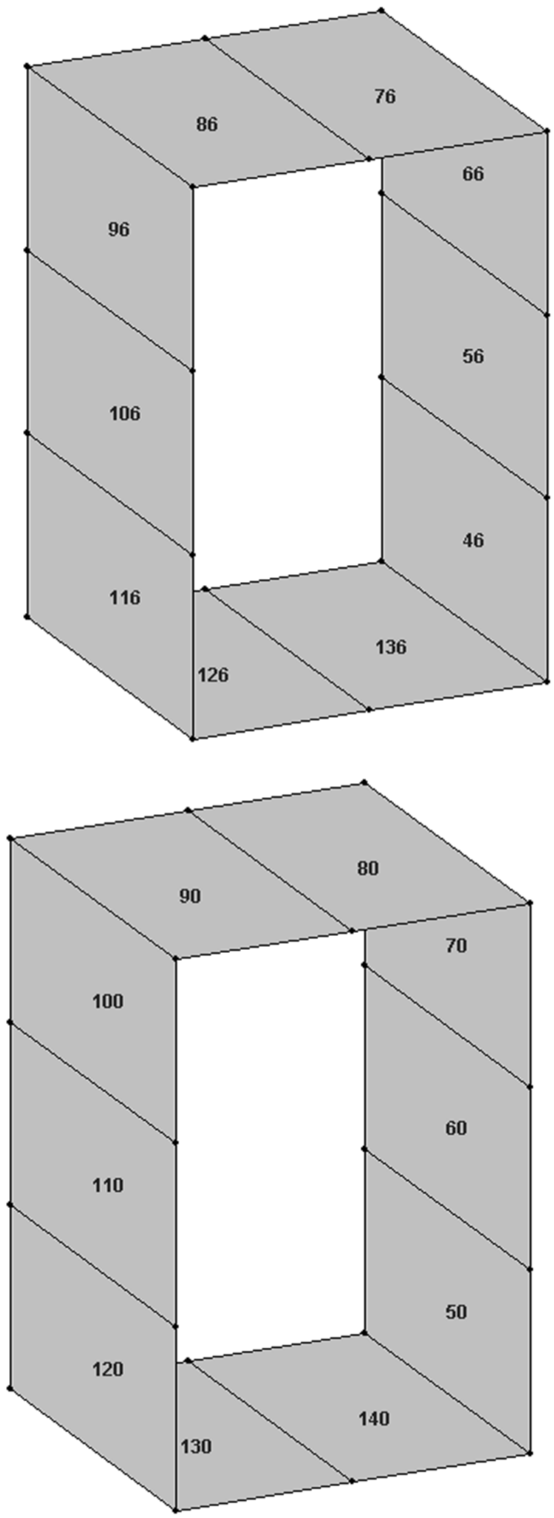


Figure 410: Element numbers at a) the middle section and b) the free end

Average deflection for nodes 11, 22, 33, 44, 55, 66, 77, 88, 99 and 10 due to load case 3 (M_y):

$$d = [4(0.07431) + 4(0.07384) + 2(0.07374)]/10 = 0.074008$$

Average deflection for nodes 11, 22, 33, 44, 55, 66, 77, 88, 99 and 10 due to load case 4 (V):

$$d = [4(-0.41123) + 4(-0.41124) + 2(-0.41057)]/10 = 0.41110$$

Average bending stress for nodes 76, 86, 126, and 136 due to load case 3 (My):

$$\sigma = 4(5551.52)/4 = 5551.52 \text{ psi}$$

Average bending stress for nodes 71, 81, 121, and 131 due to load case 4 (V):

$$\sigma = 4(42074.63)/4 = 42074.63 \text{ psi}$$

Comparison

Table 430: Comparison of results

Result Type		Theory	STAAD.Pro	Difference
Free end deflection due to axial load, P (in)	Nodes 11, 44, 66, 99	0.004	0.00410	2.5%
	Nodes 22,33, 55, 77, 88, 110	0.004	0.00401	negligible
Axial stress at the middle of the beam due to axial load, P (psi)	Elements 46, 66, 96, 116	2,000	2,000.02	negligible
	Elements 56, 106	2,000	1,991.57	negligible
	Elements 76, 86, 126, 136	2,000	2,004,19	negligible
In plane shear stress at the free end due to torque, M_x (psi)	Elements 50,70,100,120	3,333	3,333.55	negligible
	Elements 60, 110	3,333	3,325.54	negligible
	Elements 80,90,130,140	3,333	3,342.23	negligible
Avg. free end deflection at the center due to moment, M_y (in)		0.0741	0.074008	negligible
Avg. free end bending stress at the center due to moment, M_y (psi)		5,647	5,551.52	1.7%
Avg. free end deflection at the center due to shear, V (in)		0.4152	-0.41110	1.0%
Avg. free end bending stress at the center due to shear, V (psi)		42,913	42,074.63	2.0%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Cantilever Tube Stresses and Deflection.STD` is typically installed with the program.

```
STAAD SPACE :A CANTILEVER BEAM OF TUBULAR SECTION
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCES: 1. TIMOSHENKO, S., STRENGTH OF MATERIALS, PART 1,
*                2ND EDITION, D. VAN NOSTRAND COMPANY, 1940
*             2. SEELEY, F. B., AND SMITH, J.O., ADVANCED MECHANICS
*                OF MATERIALS, 2ND EDITION, JOHN WILEY AND SONS, 1955
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 2 0 0; 3 4 0 0; 4 6 0 0; 5 8 0 0; 6 10 0 0; 7 12 0 0;
8 14 0 0; 9 16 0 0; 10 18 0 0; 11 20 0 0; 12 0 1 0; 13 2 1 0; 14 4 1 0;
15 6 1 0; 16 8 1 0; 17 10 1 0; 18 12 1 0; 19 14 1 0; 20 16 1 0;
21 18 1 0; 22 20 1 0; 23 0 2 0; 24 2 2 0; 25 4 2 0; 26 6 2 0; 27 8 2 0;
28 10 2 0; 29 12 2 0; 30 14 2 0; 31 16 2 0; 32 18 2 0; 33 20 2 0;
34 0 3 0; 35 2 3 0; 36 4 3 0; 37 6 3 0; 38 8 3 0; 39 10 3 0; 40 12 3 0;
41 14 3 0; 42 16 3 0; 43 18 3 0; 44 20 3 0; 45 0 3 1; 46 2 3 1;
47 4 3 1; 48 6 3 1; 49 8 3 1; 50 10 3 1; 51 12 3 1; 52 14 3 1;
53 16 3 1; 54 18 3 1; 55 20 3 1; 56 0 3 2; 57 2 3 2; 58 4 3 2; 59 6 3 2;
60 8 3 2; 61 10 3 2; 62 12 3 2; 63 14 3 2; 64 16 3 2; 65 18 3 2;
66 20 3 2; 67 0 2 2; 68 2 2 2; 69 4 2 2; 70 6 2 2; 71 8 2 2; 72 10 2 2;
73 12 2 2; 74 14 2 2; 75 16 2 2; 76 18 2 2; 77 20 2 2; 78 0 1 2;
79 2 1 2; 80 4 1 2; 81 6 1 2; 82 8 1 2; 83 10 1 2; 84 12 1 2; 85 14 1 2;
86 16 1 2; 87 18 1 2; 88 20 1 2; 89 0 0 2; 90 2 0 2; 91 4 0 2; 92 6 0 2;
93 8 0 2; 94 10 0 2; 95 12 0 2; 96 14 0 2; 97 16 0 2; 98 18 0 2;
99 20 0 2; 100 0 0 1; 101 2 0 1; 102 4 0 1; 103 6 0 1; 104 8 0 1;
105 10 0 1; 106 12 0 1; 107 14 0 1; 108 16 0 1; 109 18 0 1; 110 20 0 1;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 34 35; 12 35 36; 13 36 37; 14 37 38; 15 38 39; 16 39 40;
17 40 41; 18 41 42; 19 42 43; 20 43 44; 21 56 57; 22 57 58; 23 58 59;
24 59 60; 25 60 61; 26 61 62; 27 62 63; 28 63 64; 29 64 65; 30 65 66;
31 89 90; 32 90 91; 33 91 92; 34 92 93; 35 93 94; 36 94 95; 37 95 96;
38 96 97; 39 97 98; 40 98 99;
ELEMENT INCIDENCES SHELL
41 1 2 13 12; 42 2 3 14 13; 43 3 4 15 14; 44 4 5 16 15; 45 5 6 17 16;
46 6 7 18 17; 47 7 8 19 18; 48 8 9 20 19; 49 9 10 21 20; 50 10 11 22 21;
51 12 13 24 23; 52 13 14 25 24; 53 14 15 26 25; 54 15 16 27 26;
55 16 17 28 27; 56 17 18 29 28; 57 18 19 30 29; 58 19 20 31 30;
59 20 21 32 31; 60 21 22 33 32; 61 23 24 35 34; 62 24 25 36 35;
63 25 26 37 36; 64 26 27 38 37; 65 27 28 39 38; 66 28 29 40 39;
67 29 30 41 40; 68 30 31 42 41; 69 31 32 43 42; 70 32 33 44 43;
71 34 35 46 45; 72 35 36 47 46; 73 36 37 48 47; 74 37 38 49 48;
75 38 39 50 49; 76 39 40 51 50; 77 40 41 52 51; 78 41 42 53 52;
79 42 43 54 53; 80 43 44 55 54; 81 45 46 57 56; 82 46 47 58 57;
83 47 48 59 58; 84 48 49 60 59; 85 49 50 61 60; 86 50 51 62 61;
87 51 52 63 62; 88 52 53 64 63; 89 53 54 65 64; 90 54 55 66 65;
91 56 57 68 67; 92 57 58 69 68; 93 58 59 70 69; 94 59 60 71 70;
95 60 61 72 71; 96 61 62 73 72; 97 62 63 74 73; 98 63 64 75 74;
```

```
99 64 65 76 75; 100 65 66 77 76; 101 67 68 79 78; 102 68 69 80 79;
103 69 70 81 80; 104 70 71 82 81; 105 71 72 83 82; 106 72 73 84 83;
107 73 74 85 84; 108 74 75 86 85; 109 75 76 87 86; 110 76 77 88 87;
111 78 79 90 89; 112 79 80 91 90; 113 80 81 92 91; 114 81 82 93 92;
115 82 83 94 93; 116 83 84 95 94; 117 84 85 96 95; 118 85 86 97 96;
119 86 87 98 97; 120 87 88 99 98; 121 89 90 101 100; 122 90 91 102 101;
123 91 92 103 102; 124 92 93 104 103; 125 93 94 105 104;
126 94 95 106 105; 127 95 96 107 106; 128 96 97 108 107;
129 97 98 109 108; 130 98 99 110 109; 131 100 101 2 1; 132 101 102 3 2;
133 102 103 4 3; 134 103 104 5 4; 135 104 105 6 5; 136 105 106 7 6;
137 106 107 8 7; 138 107 108 9 8; 139 108 109 10 9; 140 109 110 11 10;
MEMBER TRUSS
1 TO 40
MEMBER PROPERTY AMERICAN
1 TO 40 PRIS AX 0.0005
ELEMENT PROPERTY
41 TO 140 THICKNESS 0.05
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1000
POISSON 0.599866
ISOTROPIC MATERIAL2
E 1e+07
POISSON 0.33
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 MEMB 1 TO 40
MATERIAL MATERIAL2 MEMB 41 TO 140
SUPPORTS
1 12 23 34 45 56 67 78 89 100 FIXED
LOAD 1 AXIAL LOAD
JOINT LOAD
11 22 33 44 55 66 77 88 99 110 FX 100
LOAD 2 TORQUE
JOINT LOAD
11 FY 80.43 FZ -86.21
22 33 FY 169.44
44 FY 80.43 FZ 86.21
55 FZ 162.23
66 FY -80.43 FZ 86.21
77 88 FY -169.44
99 FY -80.43 FZ -86.21
110 FZ -162.23
LOAD 3 END MOMENT
JOINT LOAD
44 55 66 FX -258.6
33 77 FX -86.3
22 88 FX 86.3
11 99 110 FX 258.6
LOAD 4 SHEAR
JOINT LOAD
11 44 66 99 FY -66
22 33 77 88 FY -184
PERFORM ANALYSIS
LOAD LIST 1 TO 4
PRINT JOINT DISPLACEMENTS LIST 11 22 33 44 55 66 77 88 99 110
LOAD LIST 1
PRINT ELEMENT STRESSES LIST 46 56 66 76 86 96 106 116 126 136
```

```
LOAD LIST 2
PRINT ELEMENT STRESSES LIST 50 60 70 80 90 100 110 120 130 140
LOAD LIST 3
PRINT ELEMENT STRESSES LIST 45 46 55 56 65 66 75 76 85 86 95 96 105 -
106 115 116 125 126 135 136
LOAD LIST 4
PRINT ELEMENT STRESSES LIST 41 51 61 71 81 91 101 111 121 131
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     11    1    0.00410   0.00024   0.00020  -0.00005  -0.00082   0.00093
           2   -0.00035   0.01443  -0.02268   0.01498   0.00067   0.00001
           3    0.01142   0.07431   0.00056   0.00039  -0.00233   0.00846
           4   -0.04451  -0.41123  -0.00026  -0.00119   0.00184  -0.02863
     22    1    0.00401   0.00004  -0.00001  -0.00010   0.00013   0.00012
           2   -0.00011   0.01446  -0.00757   0.01505   0.00058   0.00007
           3    0.00371   0.07384   0.00015  -0.00031   0.00034   0.00638
           4   -0.01476  -0.41124  -0.00036   0.00050  -0.00013  -0.02764
     33    1    0.00401  -0.00004  -0.00001   0.00010   0.00013  -0.00012
           2    0.00011   0.01446   0.00757   0.01505  -0.00058   0.00007
           3   -0.00371   0.07384  -0.00015  -0.00031  -0.00034   0.00638
           4    0.01476  -0.41124   0.00036   0.00050   0.00013  -0.02764
     44    1    0.00410  -0.00024   0.00020   0.00005  -0.00082  -0.00093
           2    0.00035   0.01443   0.02268   0.01498  -0.00067   0.00001
           3   -0.01142   0.07431  -0.00056   0.00039   0.00233   0.00846
           4    0.04451  -0.41123   0.00026  -0.00119  -0.00184  -0.02863
     55    1    0.00401  -0.00011   0.00000   0.00000   0.00000   0.00027
           2    0.00000   0.00000   0.02265   0.01404  -0.00059   0.00000
           3   -0.01111   0.07374   0.00000   0.00000   0.00000   0.00696
           4    0.04414  -0.41057   0.00000   0.00000   0.00000  -0.03079
     66    1    0.00410  -0.00024  -0.00020  -0.00005   0.00082  -0.00093
           2   -0.00035  -0.01443   0.02268   0.01498  -0.00067  -0.00001
           3   -0.01142   0.07431   0.00056  -0.00039  -0.00233   0.00846
           4    0.04451  -0.41123  -0.00026   0.00119   0.00184  -0.02863
     77    1    0.00401  -0.00004   0.00001  -0.00010  -0.00013  -0.00012
           2   -0.00011  -0.01446   0.00757   0.01505  -0.00058  -0.00007
           3   -0.00371   0.07384   0.00015   0.00031   0.00034   0.00638
           4    0.01476  -0.41124  -0.00036  -0.00050  -0.00013  -0.02764
     88    1    0.00401   0.00004   0.00001   0.00010  -0.00013   0.00012
           2    0.00011  -0.01446  -0.00757   0.01505   0.00058  -0.00007
           3    0.00371   0.07384  -0.00015   0.00031  -0.00034   0.00638
           4   -0.01476  -0.41124   0.00036  -0.00050   0.00013  -0.02764
     99    1    0.00410   0.00024  -0.00020   0.00005   0.00082   0.00093
           2    0.00035  -0.01443  -0.02268   0.01498   0.00067  -0.00001
           3    0.01142   0.07431  -0.00056  -0.00039   0.00233   0.00846
           4   -0.04451  -0.41123   0.00026   0.00119  -0.00184  -0.02863
    110    1    0.00401   0.00011   0.00000   0.00000   0.00000  -0.00027
           2    0.00000   0.00000  -0.02265   0.01404   0.00059   0.00000
           3    0.01111   0.07374   0.00000   0.00000   0.00000   0.00696
           4   -0.04414  -0.41057   0.00000   0.00000   0.00000  -0.03079
      :A CANTILEVER BEAM OF TUBULAR SECTION                   -- PAGE NO.
6
```

```
   ************** END OF LATEST ANALYSIS RESULT **************
   108. LOAD LIST 1
   109. PRINT ELEMENT STRESSES LIST 46 56 66 76 86 96 106 116 126 136
  ELEMENT  STRESSES LIST     46
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
     46      1         0.02        0.00        0.00        0.00       -0.00
                    1998.93     1998.62     2000.02        2.49       -0.24
                    2000.58     1999.46
      TOP : SMAX=    2000.58 SMIN=       3.31 TMAX=     998.64 ANGLE= -0.0
      BOTT: SMAX=    1999.46 SMIN=       1.67 TMAX=     998.89 ANGLE=  0.0
     56      1        -0.01        0.00       -0.00        0.00       -0.00
                    1994.35     1994.87     1991.57       -6.07       -0.00
                    1997.15     1998.13
      TOP : SMAX=    1991.55 SMIN=      -5.60 TMAX=     998.57 ANGLE= -0.0
      BOTT: SMAX=    1991.59 SMIN=      -6.53 TMAX=     999.06 ANGLE=  0.0
     66      1         0.02       -0.00        0.00        0.00        0.00
                    1998.93     1998.62     2000.02        2.49        0.24
                    2000.58     1999.46
      TOP : SMAX=    2000.58 SMIN=       3.31 TMAX=     998.64 ANGLE=  0.0
      BOTT: SMAX=    1999.46 SMIN=       1.67 TMAX=     998.89 ANGLE= -0.0
     76      1        -0.04        0.00       -0.00       -0.00        0.00
                    2003.32     2003.71     2004.19        1.36       -0.81
                    2003.44     2004.95
      TOP : SMAX=    2003.44 SMIN=       0.24 TMAX=    1001.60 ANGLE=  0.0
      BOTT: SMAX=    2004.95 SMIN=       2.48 TMAX=    1001.24 ANGLE= -0.1
     86      1        -0.04       -0.00       -0.00       -0.00       -0.00
                    2003.32     2003.71     2004.19        1.36        0.81
                    2003.44     2004.95
      TOP : SMAX=    2003.44 SMIN=       0.24 TMAX=    1001.60 ANGLE= -0.0
      BOTT: SMAX=    2004.95 SMIN=       2.48 TMAX=    1001.24 ANGLE=  0.1
     96      1         0.02        0.00        0.00        0.00       -0.00
                    1998.93     1998.62     2000.02        2.49       -0.24
                    2000.58     1999.46
      TOP : SMAX=    2000.58 SMIN=       3.31 TMAX=     998.64 ANGLE= -0.0
      BOTT: SMAX=    1999.46 SMIN=       1.67 TMAX=     998.89 ANGLE=  0.0
    106      1        -0.01        0.00       -0.00        0.00       -0.00
                    1994.35     1994.87     1991.57       -6.07        0.00
                    1997.15     1998.13
      TOP : SMAX=    1991.55 SMIN=      -5.60 TMAX=     998.57 ANGLE= -0.0
      BOTT: SMAX=    1991.59 SMIN=      -6.53 TMAX=     999.06 ANGLE=  0.0
    116      1         0.02       -0.00        0.00        0.00        0.00
                    1998.93     1998.62     2000.02        2.49        0.24
                    2000.58     1999.46
      TOP : SMAX=    2000.58 SMIN=       3.31 TMAX=     998.64 ANGLE=  0.0
      BOTT: SMAX=    1999.46 SMIN=       1.67 TMAX=     998.89 ANGLE= -0.0
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
8
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
```

```
                    VONT       VONB         SX           SY           SXY
                    TRESCAT    TRESCAB
   126      1        -0.04       0.00       -0.00        -0.00         0.00
                  2003.32     2003.71     2004.19         1.36        -0.81
                  2003.44     2004.95
      TOP : SMAX=    2003.44 SMIN=       0.24 TMAX=    1001.60 ANGLE=  0.0
      BOTT: SMAX=    2004.95 SMIN=       2.48 TMAX=    1001.24 ANGLE= -0.1
   136      1        -0.04      -0.00       -0.00        -0.00        -0.00
                  2003.32     2003.71     2004.19         1.36         0.81
                  2003.44     2004.95
      TOP : SMAX=    2003.44 SMIN=       0.24 TMAX=    1001.60 ANGLE= -0.0
      BOTT: SMAX=    2004.95 SMIN=       2.48 TMAX=    1001.24 ANGLE=  0.1
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         2.004949E+03 -6.534180E+00  1.001600E+03  2.003712E+03  2.004949E+03
 PLATE NO.      76            56            76            76            76
 CASE  NO.       1             1             1             1             1
   *******************END OF ELEMENT FORCES********************
   110. LOAD LIST 2
   111. PRINT ELEMENT STRESSES LIST 50 60 70 80 90 100 110 120 130 140
  ELEMENT  STRESSES LIST     50
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
9
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                    VONT       VONB         SX           SY           SXY
                    TRESCAT    TRESCAB
    50      2        -2.88       1.41       -0.01        -0.02        -0.03
                  5636.28     5911.95       -0.36        18.30      3333.55
                  6508.14     6826.34
      TOP : SMAX=    3227.82 SMIN=   -3280.32 TMAX=    3254.07 ANGLE= 44.9
      BOTT: SMAX=    3457.36 SMIN=   -3368.98 TMAX=    3413.17 ANGLE= 45.2
    60      2         0.00       1.42        0.00        -0.00        -0.08
                  5445.08     6074.93        0.00        -0.00      3325.54
                  6287.44     7014.73
      TOP : SMAX=    3143.72 SMIN=   -3143.72 TMAX=    3143.72 ANGLE= 90.0
      BOTT: SMAX=    3507.36 SMIN=   -3507.36 TMAX=    3507.36 ANGLE= 90.0
    70      2         2.88       1.41        0.01         0.02        -0.03
                  5636.28     5911.95        0.36       -18.30      3333.55
                  6508.14     6826.34
      TOP : SMAX=    3280.32 SMIN=   -3227.82 TMAX=    3254.07 ANGLE= 45.1
      BOTT: SMAX=    3368.98 SMIN=   -3457.36 TMAX=    3413.17 ANGLE= 44.8
    80      2        -3.78      -1.42        0.02         0.06        -0.03
                  5674.31     5906.56        1.32       -11.09      3342.23
                  6551.26     6819.29
      TOP : SMAX=    3367.91 SMIN=   -3183.35 TMAX=    3275.63 ANGLE= 45.4
      BOTT: SMAX=    3307.60 SMIN=   -3511.69 TMAX=    3409.64 ANGLE= 44.5
    90      2         3.78      -1.42       -0.02        -0.06        -0.03
                  5674.31     5906.56       -1.32        11.09      3342.23
                  6551.26     6819.29
      TOP : SMAX=    3183.35 SMIN=   -3367.91 TMAX=    3275.63 ANGLE= 44.6
      BOTT: SMAX=    3511.69 SMIN=   -3307.60 TMAX=    3409.64 ANGLE= 45.5
   100      2        -2.88       1.41       -0.01        -0.02        -0.03
                  5636.28     5911.95       -0.36        18.30      3333.55
```

```
                    6508.14     6826.34
       TOP : SMAX=    3227.82 SMIN=   -3280.32 TMAX=    3254.07 ANGLE= 44.9
       BOTT: SMAX=    3457.36 SMIN=   -3368.98 TMAX=    3413.17 ANGLE= 45.2
    110      2         0.00        1.42       -0.00       -0.00       -0.08
                    5445.08     6074.93       -0.00        0.00     3325.54
                    6287.44     7014.73
       TOP : SMAX=    3143.72 SMIN=   -3143.72 TMAX=    3143.72 ANGLE= 90.0
       BOTT: SMAX=    3507.36 SMIN=   -3507.36 TMAX=    3507.36 ANGLE= 90.0
    120      2         2.88        1.41        0.01        0.02       -0.03
                    5636.28     5911.95        0.36      -18.30     3333.55
                    6508.14     6826.34
       TOP : SMAX=    3280.32 SMIN=   -3227.82 TMAX=    3254.07 ANGLE= 45.1
       BOTT: SMAX=    3368.98 SMIN=   -3457.36 TMAX=    3413.17 ANGLE= 44.8
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
10
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    130      2        -3.78       -1.42        0.02        0.06       -0.03
                    5674.31     5906.56        1.32      -11.09     3342.23
                    6551.26     6819.29
       TOP : SMAX=    3367.91 SMIN=   -3183.35 TMAX=    3275.63 ANGLE= 45.4
       BOTT: SMAX=    3307.60 SMIN=   -3511.69 TMAX=    3409.64 ANGLE= 44.5
    140      2         3.78       -1.42       -0.02       -0.06       -0.03
                    5674.31     5906.56       -1.32       11.09     3342.23
                    6551.26     6819.29
       TOP : SMAX=    3183.35 SMIN=   -3367.91 TMAX=    3275.63 ANGLE= 44.6
       BOTT: SMAX=    3511.69 SMIN=   -3307.60 TMAX=    3409.64 ANGLE= 45.5
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          3.511688E+03 -3.511688E+03  3.507364E+03  6.074933E+03  7.014728E+03
 PLATE NO.      90            80            60            60            60
 CASE  NO.       2             2             2             2             2
   ********************END OF ELEMENT FORCES********************
   112. LOAD LIST 3
   113. PRINT ELEMENT STRESSES LIST 45 46 55 56 65 66 75 76 85 86 95 96 105 -
  ELEMENT  STRESSES LIST     45
   114. 106 115 116 125 126 135 136
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
11
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
     45      3        -0.02        0.01       -0.00       -0.00        0.00
                    3703.79     3702.75     3703.80        1.06       -0.17
                    3704.34     3704.36
       TOP : SMAX=    3703.23 SMIN=      -1.11 TMAX=    1852.17 ANGLE=  0.0
       BOTT: SMAX=    3704.36 SMIN=       3.23 TMAX=    1850.57 ANGLE= -0.0
     46      3         0.03        0.03       -0.00       -0.00       -0.00
                    3704.99     3703.08     3709.20       10.35       -1.86
```

```
                  3709.14     3709.27
      TOP : SMAX=    3709.14 SMIN=       8.31 TMAX=    1850.41 ANGLE= -0.0
      BOTT: SMAX=    3709.27 SMIN=      12.39 TMAX=    1848.44 ANGLE= -0.0
    55      3        0.00        0.02        0.00        -0.00       -0.00
                     2.87        1.59       -0.00         0.00       -1.29
                     3.32        1.83
      TOP : SMAX=       1.66 SMIN=      -1.66 TMAX=       1.66 ANGLE= 90.0
      BOTT: SMAX=       0.92 SMIN=      -0.92 TMAX=       0.92 ANGLE= 90.0
    56      3        0.00        0.03        0.00        -0.00        0.00
                     4.19        1.52       -0.00         0.00        1.65
                     4.83        1.75
      TOP : SMAX=       2.42 SMIN=      -2.42 TMAX=       2.42 ANGLE= 90.0
      BOTT: SMAX=       0.88 SMIN=      -0.88 TMAX=       0.88 ANGLE= 90.0
    65      3        0.02        0.01        0.00         0.00        0.00
                  3703.79     3702.75    -3703.80        -1.06       -0.17
                  3704.34     3704.36
      TOP : SMAX=       1.11 SMIN=   -3703.23 TMAX=    1852.17 ANGLE= 90.0
      BOTT: SMAX=      -3.23 SMIN=   -3704.36 TMAX=    1850.57 ANGLE=-90.0
    66      3       -0.03        0.03        0.00         0.00       -0.00
                  3704.99     3703.08    -3709.20       -10.35       -1.86
                  3709.14     3709.27
      TOP : SMAX=      -8.31 SMIN=   -3709.14 TMAX=    1850.41 ANGLE=-90.0
      BOTT: SMAX=     -12.39 SMIN=   -3709.27 TMAX=    1848.44 ANGLE=-90.0
    75      3       -0.81       -0.01        0.04        -0.00        0.01
                  5463.48     5646.25    -5554.57         0.18        1.31
                  5464.40     5647.48
      TOP : SMAX=      -1.84 SMIN=   -5464.40 TMAX=    2731.28 ANGLE= 89.7
      BOTT: SMAX=       2.48 SMIN=   -5645.01 TMAX=    2823.74 ANGLE=-89.7
    76      3       -1.04       -0.03        0.04        -0.00        0.01
                  5459.11     5640.90    -5551.52        -3.75        4.91
                  5462.11     5641.41
      TOP : SMAX=      -6.00 SMIN=   -5462.11 TMAX=    2728.05 ANGLE= 89.6
      BOTT: SMAX=      -1.02 SMIN=   -5641.41 TMAX=    2820.19 ANGLE=-89.7
     :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
12
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX           MY          MXY
                      VONT        VONB         SX           SY          SXY
                      TRESCAT     TRESCAB
    85      3       -0.81        0.01        0.04        -0.00       -0.01
                  5463.48     5646.25    -5554.57         0.18       -1.31
                  5464.40     5647.48
      TOP : SMAX=      -1.84 SMIN=   -5464.40 TMAX=    2731.28 ANGLE=-89.7
      BOTT: SMAX=       2.48 SMIN=   -5645.01 TMAX=    2823.74 ANGLE= 89.7
    86      3       -1.04        0.03        0.04        -0.00       -0.01
                  5459.11     5640.90    -5551.52        -3.75       -4.91
                  5462.11     5641.41
      TOP : SMAX=      -6.00 SMIN=   -5462.11 TMAX=    2728.05 ANGLE=-89.6
      BOTT: SMAX=      -1.02 SMIN=   -5641.41 TMAX=    2820.19 ANGLE= 89.7
    95      3        0.02       -0.01        0.00         0.00       -0.00
                  3703.79     3702.75    -3703.80        -1.06        0.17
                  3704.34     3704.36
      TOP : SMAX=       1.11 SMIN=   -3703.23 TMAX=    1852.17 ANGLE=-90.0
      BOTT: SMAX=      -3.23 SMIN=   -3704.36 TMAX=    1850.57 ANGLE= 90.0
    96      3       -0.03       -0.03        0.00         0.00        0.00
                  3704.99     3703.08    -3709.20       -10.35        1.86
```

```
                    3709.14     3709.27
       TOP : SMAX=       -8.31 SMIN=   -3709.14 TMAX=    1850.41 ANGLE= 90.0
       BOTT: SMAX=      -12.39 SMIN=   -3709.27 TMAX=    1848.44 ANGLE= 90.0
    105      3         -0.00       -0.02       -0.00       -0.00        0.00
                        2.87        1.59        0.00        0.00        1.29
                        3.32        1.83
       TOP : SMAX=        1.66 SMIN=      -1.66 TMAX=       1.66 ANGLE= 90.0
       BOTT: SMAX=        0.92 SMIN=      -0.92 TMAX=       0.92 ANGLE= 90.0
    106      3         -0.00       -0.03       -0.00       -0.00       -0.00
                        4.19        1.52        0.00        0.00       -1.65
                        4.83        1.75
       TOP : SMAX=        2.42 SMIN=      -2.42 TMAX=       2.42 ANGLE= 90.0
       BOTT: SMAX=        0.88 SMIN=      -0.88 TMAX=       0.88 ANGLE= 90.0
    115      3         -0.02       -0.01       -0.00       -0.00       -0.00
                    3703.79     3702.75     3703.80        1.06        0.17
                    3704.34     3704.36
       TOP : SMAX=     3703.23 SMIN=      -1.11 TMAX=    1852.17 ANGLE= -0.0
       BOTT: SMAX=     3704.36 SMIN=       3.23 TMAX=    1850.57 ANGLE=  0.0
    116      3          0.03       -0.03       -0.00       -0.00        0.00
                    3704.99     3703.08     3709.20       10.35        1.86
                    3709.14     3709.27
       TOP : SMAX=     3709.14 SMIN=       8.31 TMAX=    1850.41 ANGLE=  0.0
       BOTT: SMAX=     3709.27 SMIN=      12.39 TMAX=    1848.44 ANGLE=  0.0
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
13
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD        SQX         SQY          MX          MY         MXY
                       VONT        VONB         SX          SY         SXY
                       TRESCAT     TRESCAB
    125      3          0.81        0.01       -0.04        0.00       -0.01
                    5463.48     5646.25     5554.57       -0.18       -1.31
                    5464.40     5647.48
       TOP : SMAX=     5464.40 SMIN=       1.84 TMAX=    2731.28 ANGLE= -0.3
       BOTT: SMAX=     5645.01 SMIN=      -2.48 TMAX=    2823.74 ANGLE=  0.3
    126      3          1.04        0.03       -0.04        0.00       -0.01
                    5459.11     5640.90     5551.52        3.75       -4.91
                    5462.11     5641.41
       TOP : SMAX=     5462.11 SMIN=       6.00 TMAX=    2728.05 ANGLE= -0.4
       BOTT: SMAX=     5641.41 SMIN=       1.02 TMAX=    2820.19 ANGLE=  0.3
    135      3          0.81       -0.01       -0.04        0.00        0.01
                    5463.48     5646.25     5554.57       -0.18        1.31
                    5464.40     5647.48
       TOP : SMAX=     5464.40 SMIN=       1.84 TMAX=    2731.28 ANGLE=  0.3
       BOTT: SMAX=     5645.01 SMIN=      -2.48 TMAX=    2823.74 ANGLE= -0.3
    136      3          1.04       -0.03       -0.04        0.00        0.01
                    5459.11     5640.90     5551.52        3.75        4.91
                    5462.11     5641.41
       TOP : SMAX=     5462.11 SMIN=       6.00 TMAX=    2728.05 ANGLE=  0.4
       BOTT: SMAX=     5641.41 SMIN=       1.02 TMAX=    2820.19 ANGLE= -0.3
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          5.645007E+03 -5.645007E+03  2.823742E+03  5.646246E+03  5.647484E+03
 PLATE NO.     125            75            75            75            75
 CASE  NO.       3             3             3             3             3
```

```
   ********************END OF ELEMENT FORCES********************
   115. LOAD LIST 4
   116. PRINT ELEMENT STRESSES LIST 41 51 61 71 81 91 101 111 121 131
  ELEMENT  STRESSES LIST     41
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
14
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
    41      4       -14.20       10.26       -0.04        -0.03         0.01
                  28183.86    28061.26   -28424.34    -2007.57     -3462.32
                  28963.74    28777.50
      TOP : SMAX=   -1630.57 SMIN=  -28963.74 TMAX=   13666.59 ANGLE=-82.7
      BOTT: SMAX=   -1492.00 SMIN=  -28777.50 TMAX=   13642.75 ANGLE=-82.6
    51      4        -0.00       -1.14        0.00         0.00        -0.02
                   5356.82     5150.93       -0.00        -0.00     -3033.32
                   6185.52     5947.78
      TOP : SMAX=    3092.76 SMIN=   -3092.76 TMAX=    3092.76 ANGLE= 90.0
      BOTT: SMAX=    2973.89 SMIN=   -2973.89 TMAX=    2973.89 ANGLE= 90.0
    61      4        14.20       10.26        0.04         0.03         0.01
                  28183.86    28061.26    28424.34     2007.57     -3462.32
                  28963.74    28777.50
      TOP : SMAX=   28963.74 SMIN=    1630.57 TMAX=   13666.59 ANGLE= -7.3
      BOTT: SMAX=   28777.50 SMIN=    1492.00 TMAX=   13642.75 ANGLE= -7.4
    71      4        21.02      -14.62       -0.33        -0.06        -0.06
                  39671.99    41111.66    42074.63     3650.80       692.96
                  41300.99    42874.24
      TOP : SMAX=   41300.99 SMIN=    3488.38 TMAX=   18906.30 ANGLE=  0.8
      BOTT: SMAX=   42874.24 SMIN=    3787.25 TMAX=   19543.49 ANGLE=  1.2
    81      4        21.02       14.62       -0.33        -0.06         0.06
                  39671.99    41111.66    42074.63     3650.80      -692.96
                  41300.99    42874.24
      TOP : SMAX=   41300.99 SMIN=    3488.38 TMAX=   18906.30 ANGLE= -0.8
      BOTT: SMAX=   42874.24 SMIN=    3787.25 TMAX=   19543.49 ANGLE= -1.2
    91      4        14.20      -10.26        0.04         0.03        -0.01
                  28183.86    28061.26    28424.34     2007.57      3462.32
                  28963.74    28777.50
      TOP : SMAX=   28963.74 SMIN=    1630.57 TMAX=   13666.59 ANGLE=  7.3
      BOTT: SMAX=   28777.50 SMIN=    1492.00 TMAX=   13642.75 ANGLE=  7.4
   101      4         0.00        1.14       -0.00        -0.00         0.02
                   5356.82     5150.93        0.00         0.00      3033.32
                   6185.52     5947.78
      TOP : SMAX=    3092.76 SMIN=   -3092.76 TMAX=    3092.76 ANGLE= 90.0
      BOTT: SMAX=    2973.89 SMIN=   -2973.89 TMAX=    2973.89 ANGLE= 90.0
   111      4       -14.20      -10.26       -0.04        -0.03        -0.01
                  28183.86    28061.26   -28424.34    -2007.57      3462.32
                  28963.74    28777.50
      TOP : SMAX=   -1630.57 SMIN=  -28963.74 TMAX=   13666.59 ANGLE= 82.7
      BOTT: SMAX=   -1492.00 SMIN=  -28777.50 TMAX=   13642.75 ANGLE= 82.6
      :A CANTILEVER BEAM OF TUBULAR SECTION                    -- PAGE NO.
15
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
```

```
                  VONT       VONB        SX         SY         SXY
                  TRESCAT    TRESCAB
   121      4       -21.02      14.62       0.33       0.06       0.06
                 39671.99   41111.66  -42074.63   -3650.80    -692.96
                 41300.99   42874.24
      TOP : SMAX=   -3488.38 SMIN=  -41300.99 TMAX=   18906.30 ANGLE=-89.2
      BOTT: SMAX=   -3787.25 SMIN=  -42874.24 TMAX=   19543.49 ANGLE=-88.8
   131      4       -21.02     -14.62       0.33       0.06      -0.06
                 39671.99   41111.66  -42074.63   -3650.80     692.96
                 41300.99   42874.24
      TOP : SMAX=   -3488.38 SMIN=  -41300.99 TMAX=   18906.30 ANGLE= 89.2
      BOTT: SMAX=   -3787.25 SMIN=  -42874.24 TMAX=   19543.49 ANGLE= 88.8
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM      MINIMUM      MAXIMUM      MAXIMUM      MAXIMUM
           PRINCIPAL    PRINCIPAL      SHEAR      VONMISES      TRESCA
            STRESS       STRESS       STRESS       STRESS       STRESS
         4.287424E+04 -4.287424E+04  1.954349E+04  4.111166E+04  4.287424E+04
 PLATE NO.      71          121           71           71           71
 CASE  NO.       4            4            4            4            4
   *******************END OF ELEMENT FORCES********************
```

V. Curved Roof Displacements and Stresses

A Cylindrical roof is supported along two circular edges. Using plate/shell elements, find the vertical deflection at the center of the free edge, principal stresses at the center of the support and center of the free edge (top and bottom of the roof plate) due to uniformly distributed gravity load.

Reference

Scordelis, A.C. and Lo, K.S., "Computer Analysis of Cylindrical Shells", *Journal of the American Concrete Institute*, Vol. 61, May 1964.

Problem

For the cylindrical roof shell calculate the following deflection and stresses due to the gravity load.

The vertical deflection, δ_y, at the center of the free edge.

Principal stresses, σ_{max} and σ_{min}, at the center line section at the vertical angle (top and bottom of the roof plate element). Principal stresses, σ_{max} and σ_{min}, at the center section of the free edge (top and bottom of the roof plate element).

$E = 4.32\times(10)^8$ psf

t = 3.0 in.

Poisson's ratio = 0.0 in theory (0.01001 in STAAD)

w = 90 psf (uniform on surface).

L = 50 feet.

r = 25 feet, 40° sector either side of vertical

Boundary conditions: simply supported on circular edges

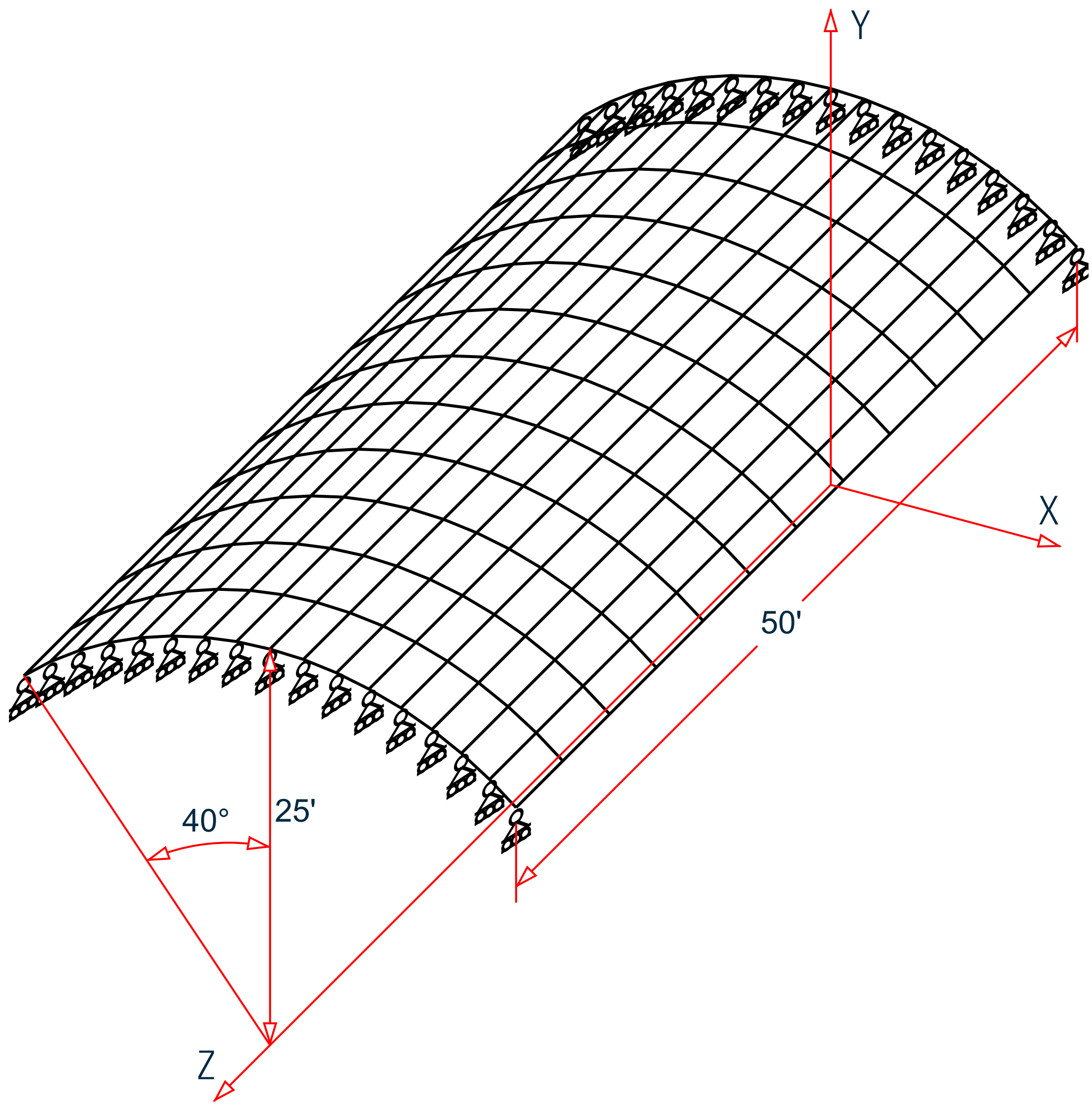


Figure 411: Finite element model of cylindrical roof structure

Comparison

Table 431: Comparison of results

Result Type		Theory	STAAD.Pro	Difference	Comments
δ_y, at the center of the free edge (in) (y translation at node 63)		-3.703	3.650	1.4%	A finer mesh may reduce the difference between the theoretical and software output.
σ at center (ft·kip)	Bottom	191.23	181.55[a]	5.1%	
	Top	218.74	209.16[a]	4.4%	

Result Type		Theory	STAAD.Pro	Difference	Comments
σ at free edge (ft·kip)	Bottom	215.57	208.33[b]	3.4%	
	Top	340.7	334.95[b]	1.7%	

a. Plate 97, Joint 55, SMAX and SMIN
b. Plate 111, Joint 63, SMAX

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Curved Roof Displacements and Stresses.STD` is typically installed with the program.

```
STAAD SPACE SCORDELIS-LO ROOF
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* A CYLINDRICAL ROOF SUPPORTED ALONG TWO CIRCULAR EDGES
*
* REFERENCE: SCORDELIS, A.C., AND LO K. S., COMPUTER ANALYSIS
*            OF CYLINDRICAL SHELLS, JOURNAL OF THE AMERICAN CONCRETE
*            INSTITUTE, VOL. 61, MAY 1964.
*
*  Theory displacement = .3086 feet or 3.7032 inches
*  STAAD displacement with POIS = 0.30
*    Joint 63&173  -3.79217 inches
*    Joint 63&173      2.01523  -3.79217   0.00009  -0.00010   0.00019
0.03358
*  STAAD displacement with POIS = 0.01001
*    Joint 63&173 -3.65022 inches
*    Joint 63&173      1.92962  -3.65022   0.00009  -0.00008   0.00015
0.03148
INPUT WIDTH 72
UNIT FEET POUND
JOINT COORDINATES
1 0 25 0; 2 2.179 24.905 0; 3 4.341 24.62 0; 4 6.471 24.148 0;
5 8.551 23.492 0; 6 10.566 22.658 0; 7 12.5 21.651 0; 8 14.339 20.479 0;
9 16.07 19.151 0; 10 0 25 4.167; 11 2.179 24.905 4.167;
12 4.341 24.62 4.167; 13 6.471 24.148 4.167; 14 8.551 23.492 4.167;
15 10.566 22.658 4.167; 16 12.5 21.651 4.167; 17 14.339 20.479 4.167;
18 16.07 19.151 4.167; 19 0 25 8.334; 20 2.179 24.905 8.334;
21 4.341 24.62 8.334; 22 6.471 24.148 8.334; 23 8.551 23.492 8.334;
24 10.566 22.658 8.334; 25 12.5 21.651 8.334; 26 14.339 20.479 8.334;
27 16.07 19.151 8.334; 28 0 25 12.501; 29 2.179 24.905 12.501;
30 4.341 24.62 12.501; 31 6.471 24.148 12.501; 32 8.551 23.492 12.501;
33 10.566 22.658 12.501; 34 12.5 21.651 12.501; 35 14.339 20.479 12.501;
36 16.07 19.151 12.501; 37 0 25 16.668; 38 2.179 24.905 16.668;
39 4.341 24.62 16.668; 40 6.471 24.148 16.668; 41 8.551 23.492 16.668;
42 10.566 22.658 16.668; 43 12.5 21.651 16.668; 44 14.339 20.479 16.668;
45 16.07 19.151 16.668; 46 0 25 20.835; 47 2.179 24.905 20.835;
48 4.341 24.62 20.835; 49 6.471 24.148 20.835; 50 8.551 23.492 20.835;
```

```
51 10.566 22.658 20.835; 52 12.5 21.651 20.835; 53 14.339 20.479 20.835;
54 16.07 19.151 20.835; 55 0 25 25.002; 56 2.179 24.905 25.002;
57 4.341 24.62 25.002; 58 6.471 24.148 25.002; 59 8.551 23.492 25.002;
60 10.566 22.658 25.002; 61 12.5 21.651 25.002; 62 14.339 20.479 25.002;
63 16.07 19.151 25.002; 64 0 25 29.169; 65 2.179 24.905 29.169;
66 4.341 24.62 29.169; 67 6.471 24.148 29.169; 68 8.551 23.492 29.169;
69 10.566 22.658 29.169; 70 12.5 21.651 29.169; 71 14.339 20.479 29.169;
72 16.07 19.151 29.169; 73 0 25 33.336; 74 2.179 24.905 33.336;
75 4.341 24.62 33.336; 76 6.471 24.148 33.336; 77 8.551 23.492 33.336;
78 10.566 22.658 33.336; 79 12.5 21.651 33.336; 80 14.339 20.479 33.336;
81 16.07 19.151 33.336; 82 0 25 37.503; 83 2.179 24.905 37.503;
84 4.341 24.62 37.503; 85 6.471 24.148 37.503; 86 8.551 23.492 37.503;
87 10.566 22.658 37.503; 88 12.5 21.651 37.503; 89 14.339 20.479 37.503;
90 16.07 19.151 37.503; 91 0 25 41.67; 92 2.179 24.905 41.67;
93 4.341 24.62 41.67; 94 6.471 24.148 41.67; 95 8.551 23.492 41.67;
96 10.566 22.658 41.67; 97 12.5 21.651 41.67; 98 14.339 20.479 41.67;
99 16.07 19.151 41.67; 100 0 25 45.837; 101 2.179 24.905 45.837;
102 4.341 24.62 45.837; 103 6.471 24.148 45.837;
104 8.551 23.492 45.837; 105 10.566 22.658 45.837;
106 12.5 21.651 45.837; 107 14.339 20.479 45.837;
108 16.07 19.151 45.837; 109 0 25 50.004; 110 2.179 24.905 50.004;
111 4.341 24.62 50.004; 112 6.471 24.148 50.004;
113 8.551 23.492 50.004; 114 10.566 22.658 50.004;
115 12.5 21.651 50.004; 116 14.339 20.479 50.004;
117 16.07 19.151 50.004; 118 -2.179 24.905 0; 119 -4.341 24.62 0;
120 -6.471 24.148 0; 121 -8.551 23.492 0; 122 -10.566 22.658 0;
123 -12.5 21.651 0; 124 -14.339 20.479 0; 125 -16.07 19.151 0;
126 -2.179 24.905 4.167; 127 -4.341 24.62 4.167;
128 -6.471 24.148 4.167; 129 -8.551 23.492 4.167;
130 -10.566 22.658 4.167; 131 -12.5 21.651 4.167;
132 -14.339 20.479 4.167; 133 -16.07 19.151 4.167;
134 -2.179 24.905 8.334; 135 -4.341 24.62 8.334;
136 -6.471 24.148 8.334; 137 -8.551 23.492 8.334;
138 -10.566 22.658 8.334; 139 -12.5 21.651 8.334;
140 -14.339 20.479 8.334; 141 -16.07 19.151 8.334;
142 -2.179 24.905 12.501; 143 -4.341 24.62 12.501;
144 -6.471 24.148 12.501; 145 -8.551 23.492 12.501;
146 -10.566 22.658 12.501; 147 -12.5 21.651 12.501;
148 -14.339 20.479 12.501; 149 -16.07 19.151 12.501;
150 -2.179 24.905 16.668; 151 -4.341 24.62 16.668;
152 -6.471 24.148 16.668; 153 -8.551 23.492 16.668;
154 -10.566 22.658 16.668; 155 -12.5 21.651 16.668;
156 -14.339 20.479 16.668; 157 -16.07 19.151 16.668;
158 -2.179 24.905 20.835; 159 -4.341 24.62 20.835;
160 -6.471 24.148 20.835; 161 -8.551 23.492 20.835;
162 -10.566 22.658 20.835; 163 -12.5 21.651 20.835;
164 -14.339 20.479 20.835; 165 -16.07 19.151 20.835;
166 -2.179 24.905 25.002; 167 -4.341 24.62 25.002;
168 -6.471 24.148 25.002; 169 -8.551 23.492 25.002;
170 -10.566 22.658 25.002; 171 -12.5 21.651 25.002;
172 -14.339 20.479 25.002; 173 -16.07 19.151 25.002;
174 -2.179 24.905 29.169; 175 -4.341 24.62 29.169;
176 -6.471 24.148 29.169; 177 -8.551 23.492 29.169;
178 -10.566 22.658 29.169; 179 -12.5 21.651 29.169;
180 -14.339 20.479 29.169; 181 -16.07 19.151 29.169;
182 -2.179 24.905 33.336; 183 -4.341 24.62 33.336;
184 -6.471 24.148 33.336; 185 -8.551 23.492 33.336;
186 -10.566 22.658 33.336; 187 -12.5 21.651 33.336;
```

```
188 -14.339 20.479 33.336; 189 -16.07 19.151 33.336;
190 -2.179 24.905 37.503; 191 -4.341 24.62 37.503;
192 -6.471 24.148 37.503; 193 -8.551 23.492 37.503;
194 -10.566 22.658 37.503; 195 -12.5 21.651 37.503;
196 -14.339 20.479 37.503; 197 -16.07 19.151 37.503;
198 -2.179 24.905 41.67; 199 -4.341 24.62 41.67;
200 -6.471 24.148 41.67; 201 -8.551 23.492 41.67;
202 -10.566 22.658 41.67; 203 -12.5 21.651 41.67;
204 -14.339 20.479 41.67; 205 -16.07 19.151 41.67;
206 -2.179 24.905 45.837; 207 -4.341 24.62 45.837;
208 -6.471 24.148 45.837; 209 -8.551 23.492 45.837;
210 -10.566 22.658 45.837; 211 -12.5 21.651 45.837;
212 -14.339 20.479 45.837; 213 -16.07 19.151 45.837;
214 -2.179 24.905 50.004; 215 -4.341 24.62 50.004;
216 -6.471 24.148 50.004; 217 -8.551 23.492 50.004;
218 -10.566 22.658 50.004; 219 -12.5 21.651 50.004;
220 -14.339 20.479 50.004; 221 -16.07 19.151 50.004;
ELEMENT INCIDENCES SHELL
1 1 2 10; 2 10 2 11; 3 11 2 3; 4 11 3 12; 5 12 3 4; 6 4 13 12; 7 4 5 14;
8 14 13 4; 9 5 15 14; 10 5 6 15; 11 15 7 16; 12 16 7 17; 13 17 7 8;
14 8 18 17; 15 8 9 18; 16 6 7 15; 17 10 11 19; 18 19 11 20; 19 20 11 12;
20 20 12 21; 21 21 12 13; 22 13 22 21; 23 13 14 23; 24 23 22 13;
25 14 24 23; 26 14 15 24; 27 24 16 25; 28 25 16 26; 29 26 16 17;
30 17 27 26; 31 17 18 27; 32 15 16 24; 33 19 20 28; 34 28 20 29;
35 29 20 21; 36 29 21 30; 37 30 21 22; 38 22 31 30; 39 22 23 32;
40 32 31 22; 41 23 33 32; 42 23 24 33; 43 33 25 34; 44 34 25 35;
45 35 25 26; 46 26 36 35; 47 26 27 36; 48 24 25 33; 49 28 29 37;
50 37 29 38; 51 38 29 30; 52 38 30 39; 53 39 30 31; 54 31 40 39;
55 31 32 41; 56 41 40 31; 57 32 42 41; 58 32 33 42; 59 42 34 43;
60 43 34 44; 61 44 34 35; 62 35 45 44; 63 35 36 45; 64 33 34 42;
65 37 38 46; 66 46 38 47; 67 47 38 39; 68 47 39 48; 69 48 39 40;
70 40 49 48; 71 40 41 50; 72 50 49 40; 73 41 51 50; 74 41 42 51;
75 51 43 52; 76 52 43 53; 77 53 43 44; 78 44 54 53; 79 44 45 54;
80 42 43 51; 81 46 47 55; 82 55 47 56; 83 56 47 48; 84 56 48 57;
85 57 48 49; 86 49 58 57; 87 49 50 59; 88 59 58 49; 89 50 60 59;
90 50 51 60; 91 60 52 61; 92 61 52 62; 93 62 52 53; 94 53 63 62;
95 53 54 63; 96 51 52 60; 97 55 56 64; 98 64 56 65; 99 65 56 57;
100 65 57 66; 101 66 57 58; 102 58 67 66; 103 58 59 68; 104 68 67 58;
105 59 69 68; 106 59 60 69; 107 69 61 70; 108 70 61 71; 109 71 61 62;
110 62 72 71; 111 62 63 72; 112 60 61 69; 113 64 65 73; 114 73 65 74;
115 74 65 66; 116 74 66 75; 117 75 66 67; 118 67 76 75; 119 67 68 77;
120 77 76 67; 121 68 78 77; 122 68 69 78; 123 78 70 79; 124 79 70 80;
125 80 70 71; 126 71 81 80; 127 71 72 81; 128 69 70 78; 129 73 74 82;
130 82 74 83; 131 83 74 75; 132 83 75 84; 133 84 75 76; 134 76 85 84;
135 76 77 86; 136 86 85 76; 137 77 87 86; 138 77 78 87; 139 87 79 88;
140 88 79 89; 141 89 79 80; 142 80 90 89; 143 80 81 90; 144 78 79 87;
145 82 83 91; 146 91 83 92; 147 92 83 84; 148 92 84 93; 149 93 84 85;
150 85 94 93; 151 85 86 95; 152 95 94 85; 153 86 96 95; 154 86 87 96;
155 96 88 97; 156 97 88 98; 157 98 88 89; 158 89 99 98; 159 89 90 99;
160 87 88 96; 161 91 92 100; 162 100 92 101; 163 101 92 93;
164 101 93 102; 165 102 93 94; 166 94 103 102; 167 94 95 104;
168 104 103 94; 169 95 105 104; 170 95 96 105; 171 105 97 106;
172 106 97 107; 173 107 97 98; 174 98 108 107; 175 98 99 108;
176 96 97 105; 177 100 101 109; 178 109 101 110; 179 110 101 102;
180 110 102 111; 181 111 102 103; 182 103 112 111; 183 103 104 113;
184 113 112 103; 185 104 114 113; 186 104 105 114; 187 114 106 115;
188 115 106 116; 189 116 106 107; 190 107 117 116; 191 107 108 117;
192 105 106 114; 193 1 118 10; 194 10 118 126; 195 126 118 119;
```

```
196 126 119 127; 197 127 119 120; 198 120 128 127; 199 120 121 129;
200 129 128 120; 201 121 130 129; 202 121 122 130; 203 130 123 131;
204 131 123 132; 205 132 123 124; 206 124 133 132; 207 124 125 133;
208 122 123 130; 209 10 126 19; 210 19 126 134; 211 134 126 127;
212 134 127 135; 213 135 127 128; 214 128 136 135; 215 128 129 137;
216 137 136 128; 217 129 138 137; 218 129 130 138; 219 138 131 139;
220 139 131 140; 221 140 131 132; 222 132 141 140; 223 132 133 141;
224 130 131 138; 225 19 134 28; 226 28 134 142; 227 142 134 135;
228 142 135 143; 229 143 135 136; 230 136 144 143; 231 136 137 145;
232 145 144 136; 233 137 146 145; 234 137 138 146; 235 146 139 147;
236 147 139 148; 237 148 139 140; 238 140 149 148; 239 140 141 149;
240 138 139 146; 241 28 142 37; 242 37 142 150; 243 150 142 143;
244 150 143 151; 245 151 143 144; 246 144 152 151; 247 144 145 153;
248 153 152 144; 249 145 154 153; 250 145 146 154; 251 154 147 155;
252 155 147 156; 253 156 147 148; 254 148 157 156; 255 148 149 157;
256 146 147 154; 257 37 150 46; 258 46 150 158; 259 158 150 151;
260 158 151 159; 261 159 151 152; 262 152 160 159; 263 152 153 161;
264 161 160 152; 265 153 162 161; 266 153 154 162; 267 162 155 163;
268 163 155 164; 269 164 155 156; 270 156 165 164; 271 156 157 165;
272 154 155 162; 273 46 158 55; 274 55 158 166; 275 166 158 159;
276 166 159 167; 277 167 159 160; 278 160 168 167; 279 160 161 169;
280 169 168 160; 281 161 170 169; 282 161 162 170; 283 170 163 171;
284 171 163 172; 285 172 163 164; 286 164 173 172; 287 164 165 173;
288 162 163 170; 289 55 166 64; 290 64 166 174; 291 174 166 167;
292 174 167 175; 293 175 167 168; 294 168 176 175; 295 168 169 177;
296 177 176 168; 297 169 178 177; 298 169 170 178; 299 178 171 179;
300 179 171 180; 301 180 171 172; 302 172 181 180; 303 172 173 181;
304 170 171 178; 305 64 174 73; 306 73 174 182; 307 182 174 175;
308 182 175 183; 309 183 175 176; 310 176 184 183; 311 176 177 185;
312 185 184 176; 313 177 186 185; 314 177 178 186; 315 186 179 187;
316 187 179 188; 317 188 179 180; 318 180 189 188; 319 180 181 189;
320 178 179 186; 321 73 182 82; 322 82 182 190; 323 190 182 183;
324 190 183 191; 325 191 183 184; 326 184 192 191; 327 184 185 193;
328 193 192 184; 329 185 194 193; 330 185 186 194; 331 194 187 195;
332 195 187 196; 333 196 187 188; 334 188 197 196; 335 188 189 197;
336 186 187 194; 337 82 190 91; 338 91 190 198; 339 198 190 191;
340 198 191 199; 341 199 191 192; 342 192 200 199; 343 192 193 201;
344 201 200 192; 345 193 202 201; 346 193 194 202; 347 202 195 203;
348 203 195 204; 349 204 195 196; 350 196 205 204; 351 196 197 205;
352 194 195 202; 353 91 198 100; 354 100 198 206; 355 206 198 199;
356 206 199 207; 357 207 199 200; 358 200 208 207; 359 200 201 209;
360 209 208 200; 361 201 210 209; 362 201 202 210; 363 210 203 211;
364 211 203 212; 365 212 203 204; 366 204 213 212; 367 204 205 213;
368 202 203 210; 369 100 206 109; 370 109 206 214; 371 214 206 207;
372 214 207 215; 373 215 207 208; 374 208 216 215; 375 208 209 217;
376 217 216 208; 377 209 218 217; 378 209 210 218; 379 218 211 219;
380 219 211 220; 381 220 211 212; 382 212 221 220; 383 212 213 221;
384 210 211 218;
ELEMENT PROPERTY
1 TO 384 THICKNESS 0.25
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 432000
POISSON 0.01001
DENSITY 0.36
END DEFINE MATERIAL
UNIT FEET POUND
```

```
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 TO 9 109 TO 125 214 TO 221 FIXED BUT FZ MX MY
55 FIXED BUT FX FY MX MY MZ KFZ 12
LOAD 1 DEAD LOAD (SELFWEIGHT OF 90 PSF)
SELFWEIGHT Y -1
PERFORM ANALYSIS
UNIT FEET KIP
PRINT JOINT DISPLACEMENTS LIST 55 63 173
PRINT ELEMENT JOINT STRESSES LIST 97 111
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     55    1    0.00000   0.54458   0.00000   0.00001   0.00000   0.00000
     63    1   -1.92962  -3.65022   0.00009  -0.00008  -0.00015  -0.03148
    173    1    1.92962  -3.65022   0.00009  -0.00008   0.00015   0.03148
   ************** END OF LATEST ANALYSIS RESULT **************
   221. PRINT ELEMENT JOINT STRESSES LIST 97 111
  ELEMENT  JOINT    STRESSES LIST
      SCORDELIS-LO ROOF                                        -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
     97      1         0.14        0.06       -2.01        -0.10        -0.00
                     200.58      176.28      -14.22        -4.43         1.22
                     207.05      178.61
       TOP : SMAX=     -13.63 SMIN=    -207.05 TMAX=      96.71 ANGLE= 89.7
       BOTT: SMAX=     178.61 SMIN=       4.77 TMAX=      86.92 ANGLE=  0.4
         JOINT         0.14        0.06       -2.03        -0.12        -0.00
           55        202.01      178.11      -13.81        -4.02         1.22
       TOP : SMAX=     -15.15 SMIN=    -209.16 TMAX=      97.00 ANGLE= 89.7
       BOTT: SMAX=     181.55 SMIN=       7.10 TMAX=      87.22 ANGLE=  0.4
         JOINT         0.14        0.06       -1.96        -0.12        -0.00
           56        193.92      171.12      -13.81        -5.25         1.22
       TOP : SMAX=     -16.38 SMIN=    -201.59 TMAX=      92.61 ANGLE= 89.7
       BOTT: SMAX=     173.98 SMIN=       5.88 TMAX=      84.05 ANGLE=  0.5
         JOINT         0.14        0.06       -2.03        -0.06        -0.00
           64        205.87      179.66      -15.04        -4.02         1.22
       TOP : SMAX=      -9.36 SMIN=    -210.40 TMAX=     100.52 ANGLE= 89.7
       BOTT: SMAX=     180.31 SMIN=       1.31 TMAX=      89.50 ANGLE=  0.4
    111      1         0.27       -0.24        0.01         0.58         0.06
                     318.91      210.31      -15.62       254.40       -16.47
                     326.36      218.90
       TOP : SMAX=     310.89 SMIN=     -15.47 TMAX=     163.18 ANGLE=-88.1
       BOTT: SMAX=     200.52 SMIN=     -18.38 TMAX=     109.45 ANGLE=-84.2
         JOINT         0.27       -0.24       -0.09         0.67         0.06
           62        296.45      162.76       -8.74       222.47       -16.47
```

```
       TOP : SMAX=     287.08 SMIN=     -17.93 TMAX=     152.51 ANGLE=-88.0
       BOTT: SMAX=     161.26 SMIN=      -2.96 TMAX=      82.11 ANGLE=-82.2
         JOINT          0.27        -0.24         0.05         0.67         0.06
          63          336.92       216.84        -8.74       270.37       -16.47
       TOP : SMAX=     334.95 SMIN=      -3.91 TMAX=     169.43 ANGLE=-88.2
       BOTT: SMAX=     208.33 SMIN=     -16.10 TMAX=     112.22 ANGLE=-84.3
         JOINT          0.27        -0.24         0.05         0.42         0.06
          72          323.64       252.42       -29.38       270.37       -16.47
       TOP : SMAX=     310.66 SMIN=     -24.56 TMAX=     167.61 ANGLE=-88.1
       BOTT: SMAX=     232.26 SMIN=     -36.38 TMAX=     134.32 ANGLE=-85.3
      SCORDELIS-LO ROOF                                          -- PAGE NO.
8
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR        VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          3.349464E+02 -2.103963E+02  1.694301E+02  3.189089E+02  3.263608E+02
 PLATE NO.     111            97           111           111           111
 CASE  NO.       1             1             1             1             1
```

V. Element Offset Table Top Comparison

A simple table-like structure is composed of four column members and a single plate element. Verify that the element offset command gives similar results as those from other modeling methods.

Problem

The plate element is subjected to a pressure load acting vertically downward along global Y direction of 1 kN/m^2. Also, two concentrated point loads of 10 kN are acting laterally along global X & global Z directions at center of the plate element.

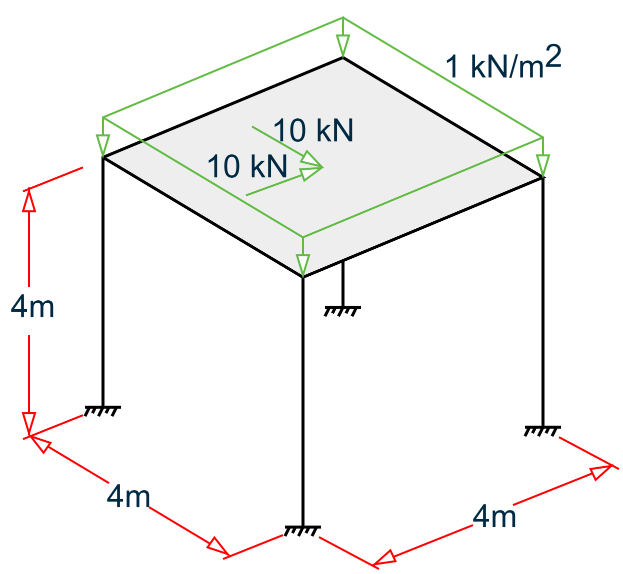


Figure 412: Loads and dimensions of the table-top structure

- Slab thickness = 20 cm
- Columns and beams = 40 cm square
- Concrete materials for all members and elements*

Offset the slab edge by 1/2 the column width (20 cm) in the plan dimensions and 1/2 the slab depth (10 cm) into the thickness of the slab.

Calculations

The model is generated with three different approaches to include the offset effect on the plate. First approach is to use `ELEMENT OFFSET` command wherein offset distances are directly assigned to corner joints of the plate without any need to manually compute co-ordinates for offset joints.

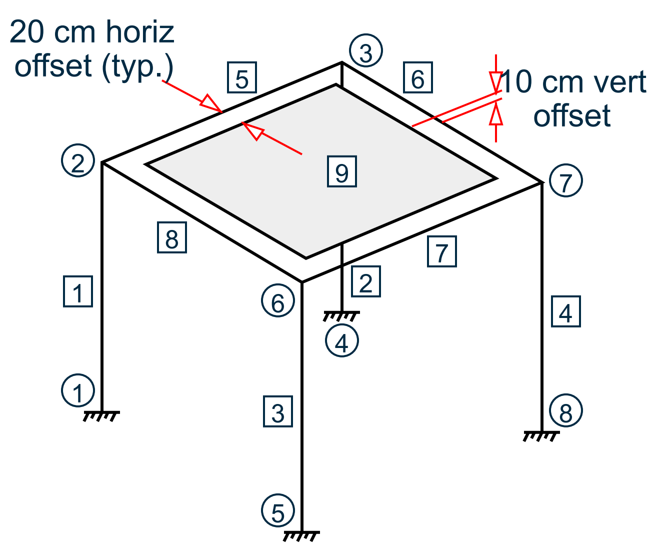
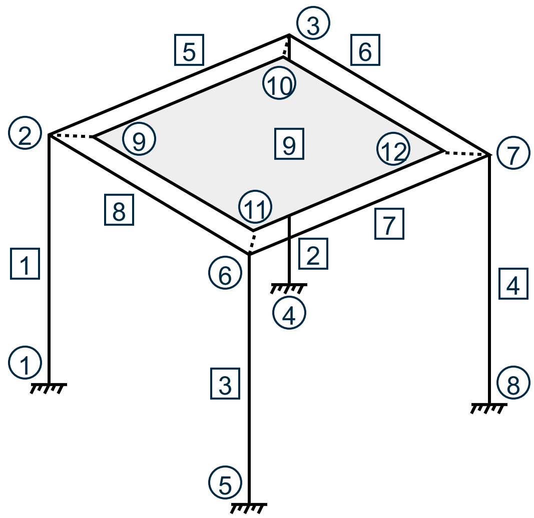


Figure 413: Nominal joint, member, and plate numbers and offset distances

The second approach is to use control-dependent specification instead of offset commands with rigid connections between offset joints of the plate & beam-column junction for load transfer. The third approach is to use short, relatively stiff beam members (*assigned with material `STIFF` having high E & G values) instead of control-dependent specification so that offset joints of plate and beam-column junction are rigidly connected to ensure load transfer.

Note: In both control-dependent approach and short rigid beam approaches, the coordinates for plate offset joints must be manually computed.



The STAAD.Pro model includes three structures to represent these three different methods. The node numbers, member numbers, and plate numbers are prefixed with 1, 2, and 3, respectively:

1. Using element offsets in the global direction (nodes 101, 102, 103, etc.)
2. Using rigid links with a high stiffness value (nodes 201, 202, 203, etc.)
3. Using a control / dependent specification (nodes 301, 302, 303, etc.)

Figure 414: Additional node numbers added for different approaches

Comparison

Results for the model created with element offset command should be identical with the model created with control-dependent approach or short rigid beam approach. Hence, the results from control-dependent and short rigid beam models are treated as reference values against which results from model with element offset command are compared.

Table 432: Comparison of selected results

Result Type		Modeling Method			Difference
		Element Offsets	Rigid Links	Control / Dependent	
Joint deflection (cm)	Joint 2, X	0.0372	0.0372	0.0372	none
	Joint 2, Y	0.0002	0.0002	0.0002	none
	Joint 2, Z	0.0372	0.0372	0.0375	none
	Joint 7, X	0.0373	0.0373	0.0373	none
	Joint 7, Y	-0.0009	-0.0009	-0.0009	none

Result Type		Modeling Method			Difference
		Element Offsets	Rigid Links	Control / Dependent	
	Joint 7, Z	0.0373	0.0373	0.0373	none
Support Reactions	Joint 8, FX (kN)	-2.82	-2.82	-2.82	none
	Joint 8, FY (kN)	8.05	8.05	8.05	none
	Joint 8, FZ (kN)	-2.82	-2.82	-2.82	none
	Joint 8, MX (kN·m)	-5.87	-5.87	-5.87	none
	Joint 8, MZ	5.87	5.87	5.87	none

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\04 Plates Shells\Element Offset Table Top Comparison.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-DEC-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
101 0 0 0; 102 0 4 0; 103 4 4 0; 104 4 0 0; 105 0 0 4; 106 0 4 4; 107 4 4 4;
108 4 0 4; 201 0 0 10; 202 0 4 10; 203 4 4 10; 204 4 0 10; 205 0 0 14;
206 0 4 14; 207 4 4 14; 208 4 0 14; 209 0.2 4.1 10.2; 210 3.8 4.1 10.2;
211 0.2 4.1 13.8; 212 3.8 4.1 13.8; 301 0 0 20; 302 0 4 20; 303 4 4 20;
304 4 0 20; 305 0 0 24; 306 0 4 24; 307 4 4 24; 308 4 0 24; 309 0.2 4.1 20.2;
310 3.8 4.1 20.2; 311 0.2 4.1 23.8; 312 3.8 4.1 23.8;
*Coordinates for Structure 1: Using ELEMENT OFFSETS
*Coordinates for structure 2: Using rigid, linking members
*Coordinates for structure 3: Using master/slave links
MEMBER INCIDENCES
101 101 102; 102 103 104; 103 105 106; 104 107 108; 105 102 103; 106 103 107;
107 107 106; 108 106 102; 201 201 202; 202 203 204; 203 205 206; 204 207 208;
205 202 203; 206 203 207; 207 207 206; 208 206 202; 210 202 209; 211 203 210;
212 207 212; 213 206 211; 301 301 302; 302 303 304; 303 305 306; 304 307 308;
305 302 303; 306 303 307; 307 307 306; 308 306 302;
*Structure 1
*Structure 2
*Structure 3
* 310 302 309; 311 303 310; 312 307 312; 313 306 311;
ELEMENT INCIDENCES SHELL
109 102 103 107 106; 209 209 210 212 211; 309 309 310 312 311;
```

```
*Structure 1
*Structure 2
*Structure 3
ELEMENT PROPERTY
109 209 309 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STIFF
E 1e+14
POISSON 0.17
DENSITY 0.0001
G 5e+13
END DEFINE MATERIAL
MEMBER PROPERTY
101 TO 108 PRIS YD 0.4 ZD 0.4
201 TO 208 PRIS YD 0.4 ZD 0.4
210 TO 213 PRIS YD 0.4 ZD 0.4
301 TO 308 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE MEMB 101 TO 109 201 TO 209 301 TO 309
*A material with very high E and G is used for rigid links structure 2
MATERIAL STIFF MEMB 210 TO 213
ELEMENT OFFSET
***Global Offsets for structure 1
109 JT1 0.2 0.1 0.2
109 JT2 -0.2 0.1 0.2
109 JT3 -0.2 0.1 -0.2
109 JT4 0.2 0.1 -0.2
***Local Offset: an alternate method of specifying the same resulting offset
*9 JT1 LOCAL 0.2 0.2 -0.1
*9 JT2 LOCAL -0.2 0.2 -0.1
*9 JT3 LOCAL -0.2 -0.2 -0.1
*9 JT4 LOCAL 0.2 -0.2 -0.1
SUPPORTS
101 104 105 108 FIXED
201 204 205 208 FIXED
301 304 305 308 FIXED
* Master / slave specification for structure 3
DEPENDENT RIGID CONTROL 302 JOINT 309
DEPENDENT RIGID CONTROL 303 JOINT 310
DEPENDENT RIGID CONTROL 306 JOINT 311
DEPENDENT RIGID CONTROL 307 JOINT 312
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
* The elements in each of the models is loaded identically
ELEMENT LOAD
109 209 309 PR GY -1
109 209 309 PR GX 10 0 0
109 209 309 PR GZ 10 0 0
PERFORM ANALYSIS PRINT ALL
PRINT ANALYSIS RESULTS
```

```
PRINT ELEMENT JOINT STRESSES
PRINT ELEMENT FORCE
FINISH
```

STAAD Output

```
  ANALYSIS RESULTS
      STAAD SPACE                                               -- PAGE NO.
7
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     101    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    102    1     0.0372    0.0002    0.0372    0.0000    0.0000   -0.0000
    103    1     0.0372   -0.0004    0.0371    0.0001   -0.0000   -0.0000
    104    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    105    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    106    1     0.0371   -0.0004    0.0372    0.0000    0.0000   -0.0001
    107    1     0.0373   -0.0009    0.0373    0.0000    0.0000   -0.0000
    108    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    201    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    202    1     0.0372    0.0002    0.0372    0.0000    0.0000   -0.0000
    203    1     0.0372   -0.0004    0.0371    0.0001   -0.0000   -0.0000
    204    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    205    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    206    1     0.0371   -0.0004    0.0372    0.0000    0.0000   -0.0001
    207    1     0.0373   -0.0009    0.0373    0.0000    0.0000   -0.0000
    208    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    209    1     0.0375   -0.0011    0.0375    0.0000    0.0000   -0.0000
    210    1     0.0376   -0.0006    0.0376    0.0001   -0.0000   -0.0000
    211    1     0.0376   -0.0006    0.0376    0.0000    0.0000   -0.0001
    212    1     0.0375   -0.0001    0.0375    0.0000    0.0000   -0.0000
    301    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    302    1     0.0372    0.0002    0.0372    0.0000    0.0000   -0.0000
    303    1     0.0372   -0.0004    0.0371    0.0001   -0.0000   -0.0000
    304    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    305    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    306    1     0.0371   -0.0004    0.0372    0.0000    0.0000   -0.0001
    307    1     0.0373   -0.0009    0.0373    0.0000    0.0000   -0.0000
    308    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    309    1     0.0375   -0.0011    0.0375    0.0000    0.0000   -0.0000
    310    1     0.0376   -0.0006    0.0376    0.0001   -0.0000   -0.0000
    311    1     0.0376   -0.0006    0.0376    0.0000    0.0000   -0.0001
    312    1     0.0375   -0.0001    0.0375    0.0000    0.0000   -0.0000
      STAAD SPACE                                               -- PAGE NO.
8
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    101    1      -2.60     -1.57     -2.60     -5.57      0.00      5.57
    104    1      -2.40      3.24     -2.18     -5.01      0.00      5.31
    105    1      -2.18      3.24     -2.40     -5.31     -0.00      5.01
    108    1      -2.82      8.05     -2.82     -5.87      0.00      5.87
    201    1      -2.60     -1.57     -2.60     -5.57      0.00      5.57
    204    1      -2.40      3.24     -2.18     -5.01      0.00      5.31
    205    1      -2.18      3.24     -2.40     -5.31     -0.00      5.01
```

```
   208     1      -2.82      8.05      -2.82      -5.87      0.00      5.87
   301     1      -2.60     -1.57      -2.60      -5.57      0.00      5.57
   304     1      -2.40      3.24      -2.18      -5.01      0.00      5.31
```

Related Links

- *TR.25.2 Element Offset Specification* (on page 2298)

V. Element Offset Water Tank Comparison

A concrete water tank structure is composed of members and plate elements. Verify that the element offset command gives similar results as those from other modeling methods.

Problem

A water tank structure supported by peripheral beams & columns is to be modelled in STAAD.Pro. Objective is to introduce offsets in elevation to all four side walls of water tank so that corner joints of side walls flushes with top surface of bottom slab and offsets in plan to two side walls spanning parallel to global Z direction so that corner joints flushes with the edge of side walls spanning parallel to global X direction.

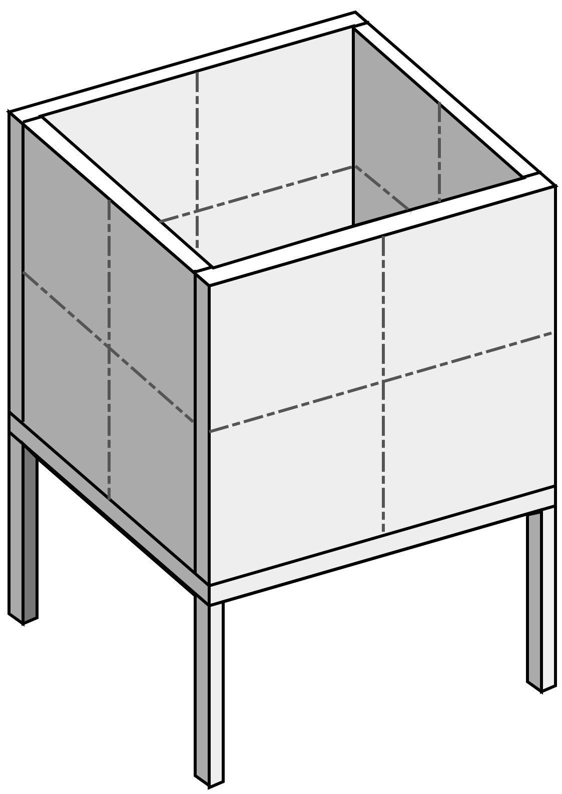


Figure 415: Concrete water tank model showing element offsets

Selfweight acting vertically downward along global Y direction.

- Tank wall and bottom thickness = 20 cm

- Columns and beams = 40 cm square
- Concrete materials for all members and elements*

Validation

The model is generated with three different approaches to include the offset effect on the plate. First approach is to use `ELEMENT OFFSET` command wherein offset distances are directly assigned to corner joints of the plate without any need to manually compute co-ordinates for offset joints.

The second approach is to use control-dependent specification instead of offset commands with rigid connections between offset joints of the plate & beam-column junction for load transfer. The third approach is to use short, relatively stiff beam members (*assigned with material `STIFF` having high E & G values) instead of control-dependent specification so that offset joints of plate and beam-column junction are rigidly connected to ensure load transfer.

Note: In both control-dependent approach and short rigid beam approaches, the coordinates for plate offset joints must be manually computed.

The STAAD.Pro model includes three structures to represent these three different methods. The node numbers, member numbers, and plate numbers are prefixed with 1, 2, and 3, respectively:

1. Using element offsets in the global direction (nodes 101, 102, 103, etc.)
2. Using rigid links with a high stiffness value (nodes 201, 202, 203, etc.)
3. Using a control / dependent specification (nodes 301, 302, 303, etc.)

Comparison

Results for the model created with element offset command should be identical with the model created with control-dependent approach or short rigid beam approach. Hence, the results from control-dependent and short rigid beam models are treated as reference values against which results from model with element offset command are compared.

Table 433: Comparison of selected results

Result Type		Modeling Method			Difference
		Element Offsets	Rigid Links	Control / Dependent	
Joint displacement (cm)	Joint 23, X	0.0007	0.0007	0.0007	none
	Joint 23, Y	-0.1238	-0.1238	-0.1238	none
	Joint 23, Z	0.0007	0.0007	0.0007	none
Support Reactions	Joint 1, FX (kN)	0.78	0.78	0.78	none
	Joint 1, FY (kN)	749.31	749.31	749.31	none
	Joint 1, FZ (kN)	0.66	0.66	0.66	none
	Joint 1, MX (kN·m)	1.16	1.16	1.16	none

Result Type		Modeling Method			Difference
		Element Offsets	Rigid Links	Control / Dependent	
	Join 1, MY (kN·m)	-0.02	-0.02	-0.02	none
	Joint 1, MZ (kN·m)	-1.39	-1.39	-1.39	none

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Element Offset Water Tank Comparison.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
101 0 -5 0; 102 10 -5 0; 103 0 -5 10; 104 10 -5 10; 105 0 0 0; 106 10 0 0;
107 0 0 10; 108 10 0 10; 109 5 0 0; 110 5 0 10; 111 0 0 5; 112 10 0 5;
113 5 5 0; 114 0 5 0; 115 10 5 0; 116 5 5 10; 117 0 5 10; 118 10 5 10;
119 0 5 5; 120 10 5 5; 121 10 10 0; 122 0 10 0; 123 10 10 10; 124 0 10 10;
125 5 10 0; 126 5 10 10; 127 0 10 5; 128 10 10 5; 201 0 -5 20; 202 10 -5 20;
203 0 -5 30; 204 10 -5 30; 205 0 0 20; 206 10 0 20; 207 0 0 30; 208 10 0 30;
209 5 0 20; 210 5 0 30; 211 0 0 25; 212 10 0 25; 213 5 5 20; 214 0 5 20;
215 10 5 20; 216 5 5 30; 217 0 5 30; 218 10 5 30; 219 0 5 25; 220 10 5 25;
221 10 10 20; 222 0 10 20; 223 10 10 30; 224 0 10 30; 225 5 10 20; 226 5 10
30;
227 0 10 25; 228 10 10 25; 229 0.1 0.1 20.1; 230 0 0.1 20; 231 0.1 0.1 25;
232 0.1 0.1 29.9; 233 0 0.1 30; 234 0.1 5 20.1; 235 0.1 5 25; 236 0.1 5 29.9;
237 0.1 10 20.1; 238 0.1 10 25; 239 0.1 10 29.9; 240 10 0.1 20; 241 5 0.1 20;
242 9.9 0.1 20.1; 243 9.9 0.1 25; 244 9.9 0.1 29.9; 245 10 0.1 30;
246 9.9 10 20.1; 247 9.9 5 20.1; 248 9.9 5 25; 249 9.9 10 25; 250 9.9 10 29.9;
251 9.9 5 29.9; 252 5 0.1 30; 301 0 -5 40; 302 10 -5 40; 303 0 -5 50;
304 10 -5 50; 305 0 0 40; 306 10 0 40; 307 0 0 50; 308 10 0 50; 309 5 0 40;
310 5 0 50; 311 0 0 45; 312 10 0 45; 313 5 5 40; 314 0 5 40; 315 10 5 40;
316 5 5 50; 317 0 5 50; 318 10 5 50; 319 0 5 45; 320 10 5 45; 321 10 10 40;
322 0 10 40; 323 10 10 50; 324 0 10 50; 325 5 10 40; 326 5 10 50; 327 0 10 45;
328 10 10 45; 329 0.1 0.1 40.1; 330 0 0.1 40; 331 0.1 0.1 45; 332 0.1 0.1
49.9;
333 0 0.1 50; 334 0.1 5 40.1; 335 0.1 5 45; 336 0.1 5 49.9; 337 0.1 10 40.1;
338 0.1 10 45; 339 0.1 10 49.9; 340 10 0.1 40; 341 5 0.1 40; 342 9.9 0.1 40.1;
343 9.9 0.1 45; 344 9.9 0.1 49.9; 345 10 0.1 50; 346 9.9 10 40.1;
347 9.9 5 40.1; 348 9.9 5 45; 349 9.9 10 45; 350 9.9 10 49.9; 351 9.9 5 49.9;
352 5 0.1 50;
* Structure 1: Element offsets
```

```
* Structure 2: Rigid linking beams
* Structure 3: Master / slave specification
MEMBER INCIDENCES
1001 105 109; 1002 107 110; 1003 105 111; 1004 106 112; 1005 105 101;
1006 106 102; 1007 107 103; 1008 108 104; 1009 109 106; 1010 110 108;
1011 111 107; 1012 112 108; 1013 114 113; 1014 113 115; 1015 115 120;
1016 120 118; 1017 118 116; 1018 116 117; 1019 117 119; 1020 119 114;
1021 122 125; 1022 125 121; 1023 121 128; 1024 128 123; 1025 123 126;
1026 126 124; 1027 124 127; 1028 127 122; 1029 105 114; 1030 114 122;
1031 106 115; 1032 115 121; 1033 108 118; 1034 118 123; 1035 107 117;
1036 117 124; 2001 205 209; 2002 207 210; 2003 205 211; 2004 206 212;
2005 205 201; 2006 206 202; 2007 207 203; 2008 208 204; 2009 209 206;
2010 210 208; 2011 211 207; 2012 212 208; 2013 215 220; 2014 220 218;
2015 217 219; 2016 219 214; 2017 221 228; 2018 228 223; 2019 224 227;
2020 227 222; 2021 214 213; 2022 213 215; 2023 218 216; 2024 216 217;
2025 222 225; 2026 225 221; 2027 223 226; 2028 226 224; 2029 205 214;
2030 214 222; 2031 206 215; 2032 215 221; 2033 208 218; 2034 218 223;
2035 207 217; 2036 217 224; 2037 205 229; 2038 205 230; 2039 206 242;
2040 206 240; 2041 208 244; 2042 208 245; 2043 207 232; 2044 207 233;
2045 209 241; 2046 210 252; 2047 211 231; 2048 212 243; 2049 214 234;
2050 215 247; 2051 218 251; 2052 217 236; 2053 219 235; 2054 220 248;
2055 222 237; 2056 221 246; 2057 223 250; 2058 224 239; 2059 227 238;
2060 228 249; 3001 305 309; 3002 307 310; 3003 305 311; 3004 306 312;
3005 305 301; 3006 306 302; 3007 307 303; 3008 308 304; 3009 309 306;
3010 310 308; 3011 311 307; 3012 312 308; 3013 315 320; 3014 320 318;
3015 317 319; 3016 319 314; 3017 321 328; 3018 328 323; 3019 324 327;
3020 327 322; 3021 314 313; 3022 313 315; 3023 318 316; 3024 316 317;
3025 322 325; 3026 325 321; 3027 323 326; 3028 326 324; 3029 305 314;
3030 314 322; 3031 306 315; 3032 315 321; 3033 308 318; 3034 318 323;
3035 307 317; 3036 317 324;
*Structure 1
* Structure 2
* Structure 3
ELEMENT INCIDENCES SHELL
114 105 106 108 107; 115 105 109 113 114; 116 109 106 115 113;
117 114 113 125 122; 118 113 115 121 125; 119 107 110 116 117;
120 110 108 118 116; 121 117 116 126 124; 122 116 118 123 126;
123 105 111 119 114; 124 111 107 117 119; 125 114 119 127 122;
126 119 117 124 127; 127 106 112 120 115; 128 112 108 118 120;
129 115 120 128 121; 130 120 118 123 128; 214 205 206 208 207;
215 230 241 213 214; 216 241 240 215 213; 217 214 213 225 222;
218 213 215 221 225; 219 233 252 216 217; 220 252 245 218 216;
221 217 216 226 224; 222 216 218 223 226; 223 229 231 235 234;
224 231 232 236 235; 225 234 235 238 237; 226 235 236 239 238;
227 242 243 248 247; 228 243 244 251 248; 229 247 248 249 246;
230 248 251 250 249; 314 305 306 308 307; 315 330 341 313 314;
316 341 340 315 313; 317 314 313 325 322; 318 313 315 321 325;
319 333 352 316 317; 320 352 345 318 316; 321 317 316 326 324;
322 316 318 323 326; 323 329 331 335 334; 324 331 332 336 335;
325 334 335 338 337; 326 335 336 339 338; 327 342 343 348 347;
328 343 344 351 348; 329 347 348 349 346; 330 348 351 350 349;
* Structure 1
* Structure 2
* Structure 3
ELEMENT PROPERTY
114 TO 130 214 TO 230 314 TO 330 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC CONCRETE
```

```
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STIFF
E 1e+12
POISSON 0.17
DENSITY 0.0001
G 5e+11
END DEFINE MATERIAL
MEMBER PROPERTY
1001 TO 1036 PRIS YD 0.4 ZD 0.4
2001 TO 2036 PRIS YD 0.4 ZD 0.4
2037 TO 2060 PRIS YD 0.4 ZD 0.4
3001 TO 3036 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE MEMB 114 TO 130 214 TO 230 314 TO 330 1001 TO 1036 2001 -
2002 TO 2036 3001 TO 3036
MATERIAL STIFF MEMB 2037 TO 2060
**Global Offsets
ELEMENT OFFSET
115 JT1 0 0.1 0
115 JT2 0 0.1 0
116 JT1 0 0.1 0
116 JT2 0 0.1 0
119 JT1 0 0.1 0
119 JT2 0 0.1 0
120 JT1 0 0.1 0
120 JT2 0 0.1 0
123 JT1 0.1 0.1 0.1
123 JT2 0.1 0.1 0
123 JT3 0.1 0 0
123 JT4 0.1 0 0.1
124 JT1 0.1 0.1 0
124 JT2 0.1 0.1 -0.1
124 JT3 0.1 0 -0.1
124 JT4 0.1 0 0
125 JT1 0.1 0 0.1
125 JT2 0.1 0 0
125 JT3 0.1 0 0
125 JT4 0.1 0 0.1
126 JT1 0.1 0 0
126 JT2 0.1 0 -0.1
126 JT3 0.1 0 -0.1
126 JT4 0.1 0 0
127 JT1 -0.1 0.1 0.1
127 JT2 -0.1 0.1 0
127 JT3 -0.1 0 0
127 JT4 -0.1 0 0.1
128 JT1 -0.1 0.1 0
128 JT2 -0.1 0.1 -0.1
128 JT3 -0.1 0 -0.1
128 JT4 -0.1 0 0
129 JT1 -0.1 0 0.1
```

```
129 JT2 -0.1 0 0
129 JT3 -0.1 0 0
129 JT4 -0.1 0 0.1
130 JT1 -0.1 0 0
130 JT2 -0.1 0 -0.1
130 JT3 -0.1 0 -0.1
130 JT4 -0.1 0 0
SUPPORTS
101 TO 104 FIXED
201 TO 204 FIXED
301 TO 304 FIXED
* Master / slave specifications for Structure 3
DEPENDENT RIGID CONTROL 305 JOINT 329 330
DEPENDENT RIGID CONTROL 309 JOINT 341
DEPENDENT RIGID CONTROL 306 JOINT 340 342
DEPENDENT RIGID CONTROL 312 JOINT 343
DEPENDENT RIGID CONTROL 308 JOINT 344 345
DEPENDENT RIGID CONTROL 310 JOINT 352
DEPENDENT RIGID CONTROL 307 JOINT 332 333
DEPENDENT RIGID CONTROL 311 JOINT 331
DEPENDENT RIGID CONTROL 314 JOINT 334
DEPENDENT RIGID CONTROL 315 JOINT 347
DEPENDENT RIGID CONTROL 318 JOINT 351
DEPENDENT RIGID CONTROL 317 JOINT 336
DEPENDENT RIGID CONTROL 322 JOINT 337
DEPENDENT RIGID CONTROL 321 JOINT 346
DEPENDENT RIGID CONTROL 323 JOINT 350
DEPENDENT RIGID CONTROL 324 JOINT 339
DEPENDENT RIGID CONTROL 319 JOINT 335
DEPENDENT RIGID CONTROL 320 JOINT 348
DEPENDENT RIGID CONTROL 327 JOINT 338
DEPENDENT RIGID CONTROL 328 JOINT 349
LOAD 1 LOADTYPE None  TITLE SELFWEIGHT
SELFWEIGHT Y -1
PERFORM ANALYSIS PRINT ALL
PRINT ANALYSIS RESULTS
PRINT ELEMENT JOINT STRESSES
PRINT ELEMENT FORCE
FINISH
```

STAAD Output

```
  ANALYSIS RESULTS
      STAAD SPACE                                                -- PAGE NO.
12
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    101    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    102    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    103    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    104    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    105    1    -0.0027   -0.1065   -0.0017    0.0001    0.0000   -0.0001
    106    1     0.0027   -0.1065   -0.0017    0.0001   -0.0000    0.0001
    107    1    -0.0027   -0.1065    0.0017   -0.0001   -0.0000   -0.0001
    108    1     0.0027   -0.1065    0.0017   -0.0001    0.0000    0.0001
```

```
    109    1      0.0000   -0.1237   -0.0022    0.0000    0.0000    0.0000
    110    1      0.0000   -0.1237    0.0022   -0.0000    0.0000    0.0000
    111    1      0.0025   -0.1236    0.0000    0.0000    0.0000    0.0000
    112    1     -0.0025   -0.1236    0.0000    0.0000    0.0000   -0.0000
    113    1      0.0000   -0.1217    0.0029   -0.0000    0.0000    0.0000
    114    1      0.0004   -0.1214    0.0003   -0.0000   -0.0000    0.0000
    115    1     -0.0004   -0.1214    0.0003   -0.0000    0.0000   -0.0000
    116    1      0.0000   -0.1217   -0.0029    0.0000    0.0000    0.0000
    117    1      0.0004   -0.1214   -0.0003    0.0000    0.0000    0.0000
    118    1     -0.0004   -0.1214   -0.0003    0.0000   -0.0000   -0.0000
    119    1      0.0029   -0.1217    0.0000    0.0000    0.0000    0.0000
    120    1     -0.0029   -0.1217    0.0000    0.0000    0.0000   -0.0000
    121    1      0.0007   -0.1238   -0.0007   -0.0000    0.0000   -0.0000
    122    1     -0.0007   -0.1238   -0.0007   -0.0000   -0.0000    0.0000
    123    1      0.0007   -0.1238    0.0007    0.0000   -0.0000   -0.0000
    124    1     -0.0007   -0.1238    0.0007    0.0000    0.0000    0.0000
    125    1      0.0000   -0.1232   -0.0003   -0.0000    0.0000    0.0000
    126    1      0.0000   -0.1232    0.0003    0.0000    0.0000    0.0000
    127    1     -0.0062   -0.1235    0.0000    0.0000    0.0000    0.0000
    128    1      0.0062   -0.1235    0.0000    0.0000    0.0000   -0.0000
    201    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    202    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    203    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    204    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    205    1     -0.0027   -0.1065   -0.0017    0.0001    0.0000   -0.0001
    206    1      0.0027   -0.1065   -0.0017    0.0001   -0.0000    0.0001
    207    1     -0.0027   -0.1065    0.0017   -0.0001   -0.0000   -0.0001
    208    1      0.0027   -0.1065    0.0017   -0.0001    0.0000    0.0001
    209    1     -0.0000   -0.1237   -0.0022    0.0000    0.0000    0.0000
    210    1     -0.0000   -0.1237    0.0022   -0.0000    0.0000    0.0000
    211    1      0.0025   -0.1236   -0.0000   -0.0000    0.0000    0.0000
    212    1     -0.0025   -0.1236   -0.0000   -0.0000    0.0000   -0.0000
    213    1     -0.0000   -0.1217    0.0029   -0.0000    0.0000    0.0000
    214    1      0.0004   -0.1214    0.0003   -0.0000   -0.0000    0.0000
    215    1     -0.0004   -0.1214    0.0003   -0.0000    0.0000   -0.0000
    216    1     -0.0000   -0.1217   -0.0029    0.0000    0.0000    0.0000
    217    1      0.0004   -0.1214   -0.0003    0.0000    0.0000    0.0000
    218    1     -0.0004   -0.1214   -0.0003    0.0000   -0.0000   -0.0000
      STAAD SPACE                                                -- PAGE NO.
13
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    219    1      0.0029   -0.1217   -0.0000   -0.0000    0.0000    0.0000
    220    1     -0.0029   -0.1217   -0.0000   -0.0000    0.0000   -0.0000
    221    1      0.0007   -0.1238   -0.0007   -0.0000    0.0000   -0.0000
    222    1     -0.0007   -0.1238   -0.0007   -0.0000   -0.0000    0.0000
    223    1      0.0007   -0.1238    0.0007    0.0000   -0.0000   -0.0000
    224    1     -0.0007   -0.1238    0.0007    0.0000    0.0000    0.0000
    225    1     -0.0000   -0.1232   -0.0003   -0.0000    0.0000    0.0000
    226    1     -0.0000   -0.1232    0.0003    0.0000    0.0000    0.0000
    227    1     -0.0062   -0.1235   -0.0000   -0.0000    0.0000    0.0000
    228    1      0.0062   -0.1235   -0.0000   -0.0000    0.0000   -0.0000
    229    1     -0.0020   -0.1076   -0.0012    0.0001    0.0000   -0.0001
    230    1     -0.0021   -0.1065   -0.0012    0.0001    0.0000   -0.0001
    231    1      0.0023   -0.1234   -0.0000   -0.0000    0.0000    0.0000
    232    1     -0.0020   -0.1076    0.0012   -0.0001   -0.0000   -0.0001
    233    1     -0.0021   -0.1065    0.0012   -0.0001   -0.0000   -0.0001
```

```
    234    1     0.0004   -0.1212    0.0003   -0.0000   -0.0000    0.0000
    235    1     0.0029   -0.1215   -0.0000   -0.0000    0.0000    0.0000
    236    1     0.0004   -0.1212   -0.0003    0.0000    0.0000    0.0000
    237    1    -0.0008   -0.1235   -0.0007   -0.0000   -0.0000    0.0000
    238    1    -0.0062   -0.1230   -0.0000   -0.0000    0.0000    0.0000
    239    1    -0.0008   -0.1235    0.0007    0.0000    0.0000    0.0000
    240    1     0.0021   -0.1065   -0.0012    0.0001   -0.0000    0.0001
    241    1    -0.0000   -0.1237   -0.0021    0.0000    0.0000    0.0000
    242    1     0.0020   -0.1076   -0.0012    0.0001   -0.0000    0.0001
    243    1    -0.0023   -0.1234   -0.0000   -0.0000    0.0000   -0.0000
    244    1     0.0020   -0.1076    0.0012   -0.0001    0.0000    0.0001
    245    1     0.0021   -0.1065    0.0012   -0.0001    0.0000    0.0001
    246    1     0.0008   -0.1235   -0.0007   -0.0000    0.0000   -0.0000
    247    1    -0.0004   -0.1212    0.0003   -0.0000    0.0000   -0.0000
    248    1    -0.0029   -0.1215   -0.0000   -0.0000    0.0000   -0.0000
    249    1     0.0062   -0.1230   -0.0000   -0.0000    0.0000   -0.0000
    250    1     0.0008   -0.1235    0.0007    0.0000   -0.0000   -0.0000
    251    1    -0.0004   -0.1212   -0.0003    0.0000   -0.0000   -0.0000
    252    1    -0.0000   -0.1237    0.0021   -0.0000    0.0000    0.0000
    301    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    302    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    303    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    304    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    305    1    -0.0027   -0.1065   -0.0017    0.0001    0.0000   -0.0001
    306    1     0.0027   -0.1065   -0.0017    0.0001   -0.0000    0.0001
    307    1    -0.0027   -0.1065    0.0017   -0.0001   -0.0000   -0.0001
    308    1     0.0027   -0.1065    0.0017   -0.0001    0.0000    0.0001
    309    1    -0.0000   -0.1237   -0.0022    0.0000    0.0000    0.0000
    310    1    -0.0000   -0.1237    0.0022   -0.0000    0.0000    0.0000
    311    1     0.0025   -0.1236   -0.0000   -0.0000    0.0000    0.0000
    312    1    -0.0025   -0.1236   -0.0000   -0.0000    0.0000   -0.0000
      STAAD SPACE                                               -- PAGE NO.
14
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
  JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    313    1    -0.0000   -0.1217    0.0029   -0.0000    0.0000    0.0000
    314    1     0.0004   -0.1214    0.0003   -0.0000   -0.0000    0.0000
    315    1    -0.0004   -0.1214    0.0003   -0.0000    0.0000   -0.0000
    316    1    -0.0000   -0.1217   -0.0029    0.0000    0.0000    0.0000
    317    1     0.0004   -0.1214   -0.0003    0.0000    0.0000    0.0000
    318    1    -0.0004   -0.1214   -0.0003    0.0000   -0.0000   -0.0000
    319    1     0.0029   -0.1217   -0.0000   -0.0000    0.0000    0.0000
    320    1    -0.0029   -0.1217   -0.0000   -0.0000    0.0000   -0.0000
    321    1     0.0007   -0.1238   -0.0007   -0.0000    0.0000   -0.0000
    322    1    -0.0007   -0.1238   -0.0007   -0.0000   -0.0000    0.0000
    323    1     0.0007   -0.1238    0.0007    0.0000   -0.0000   -0.0000
    324    1    -0.0007   -0.1238    0.0007    0.0000    0.0000    0.0000
    325    1    -0.0000   -0.1232   -0.0003   -0.0000    0.0000    0.0000
    326    1    -0.0000   -0.1232    0.0003    0.0000    0.0000    0.0000
    327    1    -0.0062   -0.1235   -0.0000   -0.0000    0.0000    0.0000
    328    1     0.0062   -0.1235   -0.0000   -0.0000    0.0000   -0.0000
    329    1    -0.0020   -0.1076   -0.0012    0.0001    0.0000   -0.0001
    330    1    -0.0021   -0.1065   -0.0012    0.0001    0.0000   -0.0001
    331    1     0.0023   -0.1234   -0.0000   -0.0000    0.0000    0.0000
    332    1    -0.0020   -0.1076    0.0012   -0.0001   -0.0000   -0.0001
    333    1    -0.0021   -0.1065    0.0012   -0.0001   -0.0000   -0.0001
    334    1     0.0004   -0.1212    0.0003   -0.0000   -0.0000    0.0000
```

```
    335     1     0.0029   -0.1215   -0.0000   -0.0000    0.0000    0.0000
    336     1     0.0004   -0.1212   -0.0003    0.0000    0.0000    0.0000
    337     1    -0.0008   -0.1235   -0.0007   -0.0000   -0.0000    0.0000
    338     1    -0.0062   -0.1230   -0.0000   -0.0000    0.0000    0.0000
    339     1    -0.0008   -0.1235    0.0007    0.0000    0.0000    0.0000
    340     1     0.0021   -0.1065   -0.0012    0.0001   -0.0000    0.0001
    341     1    -0.0000   -0.1237   -0.0021    0.0000    0.0000    0.0000
    342     1     0.0020   -0.1076   -0.0012    0.0001   -0.0000    0.0001
    343     1    -0.0023   -0.1234   -0.0000   -0.0000    0.0000   -0.0000
    344     1     0.0020   -0.1076    0.0012   -0.0001    0.0000    0.0001
    345     1     0.0021   -0.1065    0.0012   -0.0001    0.0000    0.0001
    346     1     0.0008   -0.1235   -0.0007   -0.0000    0.0000   -0.0000
    347     1    -0.0004   -0.1212    0.0003   -0.0000    0.0000   -0.0000
    348     1    -0.0029   -0.1215   -0.0000   -0.0000    0.0000   -0.0000
    349     1     0.0062   -0.1230   -0.0000   -0.0000    0.0000   -0.0000
    350     1     0.0008   -0.1235    0.0007    0.0000   -0.0000   -0.0000
    351     1    -0.0004   -0.1212   -0.0003    0.0000   -0.0000   -0.0000
    352     1    -0.0000   -0.1237    0.0021   -0.0000    0.0000    0.0000
      STAAD SPACE                                                -- PAGE NO.
15
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    101     1      0.78    749.31      0.66      1.16     -0.02     -1.39
    102     1     -0.78    749.31      0.66      1.16      0.02      1.39
    103     1      0.78    749.31     -0.66     -1.16      0.02     -1.39
    104     1     -0.78    749.31     -0.66     -1.16     -0.02      1.39
    201     1      0.78    749.31      0.66      1.16     -0.02     -1.39
    202     1     -0.78    749.31      0.66      1.16      0.02      1.39
    203     1      0.78    749.31     -0.66     -1.16      0.02     -1.39
    204     1     -0.78    749.31     -0.66     -1.16     -0.02      1.39
    301     1      0.78    749.31      0.66      1.16     -0.02     -1.39
    302     1     -0.78    749.31      0.66      1.16      0.02      1.39
```

Related Links

- *TR.25.2 Element Offset Specification* (on page 2298)

V. Response Spectrum Using Element Offset

Validate the response spectrum analysis results using element offsets with a similar model that uses control-dependent connections.

Details

A 2 story building with overall dimensions of 20 m in the global X direction and 8 m in the global Z direction is modelled in STAAD.Pro.

A custom defined response spectrum loading containing time-acceleration pairs along global X, Z, & Y directions is used for the loading.

Validation

The model is generated with two different approaches to include the effect on the plate offsets. The first approach is to use the `ELEMENT OFFSET` command wherein offset distances are directly assigned to corner joints of the plate without any need to manually compute coordinates for offset joints. The second approach is to use control-dependent specification instead with rigid connections between offset joints of the plate & beam-column junction for load transfer. In the control-dependent approach, one must note that coordinates for plate offset joints need to be manually computed.

Results for the model created with `ELEMENT OFFSET` command should be identical with the model created with control-dependent approach. Hence results from control-dependent approach are treated as reference values against which results from model with element offset command will be compared.

Results

Table 434: Eigen solution frequencies (cycles/sec.)

Mode Number	Control-Dependent Method (Reference)	Element Offset Method	Difference	Comments
1	3.299	3.299	none	
2	4.513	4.513	none	
3	8.747	8.747	none	
4	10.076	10.076	none	
5	12.601	12.601	none	
6	32.152	32.152	none	
7	67.750	67.750	none	
8	67.990	67.990	none	
9	68.843	68.843	none	
10	70.679	70.679	none	
11	71.621	71.621	none	
12	71.712	71.712	none	
13	73.050	73.050	none	
14	75.299	75.299	none	
15	78.879	78.879	none	
16	85.007	85.007	none	
17	91.029	91.029	none	
18	91.989	91.989	none	
19	92.101	92.101	none	

Mode Number	Control-Dependent Method (Reference)	Element Offset Method	Difference	Comments
20	92.960	92.960	none	
21	93.797	93.797	none	
22	94.286	94.286	none	
23	95.411	95.411	none	

Table 435: Joint displacement at node 45 for load case 2

Quantity	Control-Dependent Method (Reference)	Element Offset Method	Difference	Remarks
X Translation (cm)	0.0014	0.0014	none	
Y Translation (cm)	0.0152	0.0152	none	
Z Translation (cm)	6.7039	6.7039	none	
X Rotation (radians)	0.0007	0.0007	none	
Y Rotation (radians)	0.0000	0.0000	none	
Z Rotation (radians)	0.0001	0.0001	none	

Table 436: Support reactions at node 21 for load case 1

Quantity	Control-Dependent Method (Reference)	Element Offset Method	Difference	Remarks
FX (kN)	69.83	69.83	none	
FY (kN)	54.99	54.99	none	
FZ (kN)	12.43	12.43	none	
MX (kN·m)	25.85	25.85	none	
MY (kN·m)	0.02	0.02	none	
MZ (kN·m)	210.76	210.76	none	

Table 437: Member end forces on member 39 for load case 3

Quantity		Control-Dependent Method (Reference)	Element Offset Method	Difference	Remarks
Node 23	FX (kN)	15.56	15.56	none	
	FY (kN)	0.17	0.17	none	
	FZ (kN)	0.07	0.07	none	
	MX (kN·m)	0.00	0.00	none	
	MY (kN·m)	0.15	0.15	none	
	MZ (kN·m)	0.39	0.39	none	
Node 24	FX (kN)	15.56	15.56	none	
	FY (kN)	0.17	0.17	none	
	FZ (kN)	0.07	0.07	none	
	MX (kN·m)	0.00	0.00	none	
	MY (kN·m)	0.12	0.12	none	
	MZ (kN·m)	0.31	0.31	none	

STAAD.Pro Input

Input using the `ELEMENT OFFSET` method:

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Response Spectrum Using Element Offset.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 09-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
SET PRINT 17
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 5 4 0; 4 5 0 0; 5 10 4 0; 6 10 0 0; 7 15 4 0; 8 15 0 0;
9 20 4 0; 10 20 0 0; 11 0 0 4; 12 0 4 4; 13 5 4 4; 14 5 0 4; 15 10 4 4;
16 10 0 4; 17 15 4 4; 18 15 0 4; 19 20 4 4; 20 20 0 4; 21 0 0 8; 22 0 4 8;
23 5 4 8; 24 5 0 8; 25 10 4 8; 26 10 0 8; 27 15 4 8; 28 15 0 8; 29 20 4 8;
30 20 0 8; 31 0 8 0; 32 5 8 0; 33 10 8 0; 34 15 8 0; 35 20 8 0; 36 0 8 4;
37 5 8 4; 38 10 8 4; 39 15 8 4; 40 20 8 4; 41 0 8 8; 42 5 8 8; 43 10 8 8;
```

```
44 15 8 8; 45 20 8 8;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 3 5; 5 5 6; 6 5 7; 9 7 8; 10 7 9; 13 9 10; 14 2 12;
15 3 13; 16 5 15; 17 7 17; 18 9 19; 19 11 12; 20 12 13; 21 13 14; 22 13 15;
23 15 16; 24 15 17; 27 17 18; 28 17 19; 31 19 20; 32 12 22; 33 13 23; 34 15
25;
35 17 27; 36 19 29; 37 21 22; 38 22 23; 39 23 24; 40 23 25; 41 25 26; 42 25
27;
45 27 28; 46 27 29; 49 29 30; 58 2 31; 59 3 32; 60 5 33; 61 7 34; 62 9 35;
63 12 36; 64 13 37; 65 15 38; 66 17 39; 67 19 40; 68 22 41; 69 23 42; 70 25
43;
71 27 44; 72 29 45; 73 31 32; 74 32 33; 75 33 34; 77 34 35; 79 31 36; 80 32
37;
81 33 38; 82 34 39; 83 35 40; 84 36 37; 85 37 38; 86 38 39; 88 39 40; 90 36
41;
91 37 42; 92 38 43; 93 39 44; 94 40 45; 95 41 42; 96 42 43; 97 43 44; 99 44
45;
ELEMENT INCIDENCES SHELL
50 2 3 13 12; 51 3 5 15 13; 52 5 7 17 15; 53 7 9 19 17; 54 12 13 23 22;
55 13 15 25 23; 56 15 17 27 25; 57 17 19 29 27; 101 31 32 37 36;
102 32 33 38 37; 103 33 34 39 38; 104 34 35 40 39; 105 36 37 42 41;
106 37 38 43 42; 107 38 39 44 43; 108 39 40 45 44;
ELEMENT PROPERTY
50 TO 57 101 TO 108 THICKNESS 0.3
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
G 7.88462e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 6 9 10 13 TO 24 27 28 31 TO 36 38 40 42 46 58 TO 67 73 TO 75 77 -
79 TO 86 88 90 TO 97 99 TABLE ST W12X50
37 39 41 45 49 68 TO 72 TABLE ST W30X90
CONSTANTS
MATERIAL STEEL MEMB 1 TO 6 9 10 13 TO 24 27 28 31 TO 42 45 46 49 58 TO 75 -
77 79 TO 86 88 90 TO 97 99
MATERIAL CONCRETE MEMB 50 TO 57 101 TO 108
SUPPORTS
1 4 6 8 10 11 14 16 18 20 21 24 26 28 30 FIXED
ELEMENT OFFSET
50 101 JT1 0.1 0 0.1
50 101 JT2 0 0 0.1
50 101 JT4 0.1 0 0
```

```
54 105 JT1 0.1 0 0
54 105 JT3 0 0 -0.1
54 105 JT4 0.1 0 -0.1
51 102 JT1 0 0 0.1
51 102 JT2 0 0 0.1
55 106 JT3 0 0 -0.1
55 106 JT4 0 0 -0.1
52 103 JT1 0 0 0.1
52 103 JT2 0 0 0.1
56 107 JT3 0 0 -0.1
56 107 JT4 0 0 -0.1
53 104 JT1 0 0 0.1
53 104 JT2 -0.1 0 0.1
53 104 JT3 -0.1 0 0
57 108 JT2 -0.1 0 0
57 108 JT3 -0.1 0 -0.1
57 108 JT4 0 0 -0.1
CUT OFF MODE SHAPE 50
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 3 5 7 9 12 13 15 17 19 22 23 25 27 29 31 TO 45 FX 10 FY -10 FZ 10
END DEFINE REFERENCE LOADS
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS X 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
LOAD 2 LOADTYPE None  TITLE RS_Z
SPECTRUM SRSS Z 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
```

```
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
LOAD 3 LOADTYPE None  TITLE RS_Y
SPECTRUM SRSS Y 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
PERFORM ANALYSIS PRINT ALL
PRINT ANALYSIS RESULTS
PRINT ELEMENT JOINT STRESSES
FINISH
```

Input using the Control-Dependent method:

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Response Spectrum Using Control Dependent.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 09-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
SET PRINT 17
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 5 4 0; 4 5 0 0; 5 10 4 0; 6 10 0 0; 7 15 4 0; 8 15 0 0;
9 20 4 0; 10 20 0 0; 11 0 0 4; 12 0 4 4; 13 5 4 4; 14 5 0 4; 15 10 4 4;
16 10 0 4; 17 15 4 4; 18 15 0 4; 19 20 4 4; 20 20 0 4; 21 0 0 8; 22 0 4 8;
23 5 4 8; 24 5 0 8; 25 10 4 8; 26 10 0 8; 27 15 4 8; 28 15 0 8; 29 20 4 8;
30 20 0 8; 31 0 8 0; 32 5 8 0; 33 10 8 0; 34 15 8 0; 35 20 8 0; 36 0 8 4;
```

```
37 5 8 4; 38 10 8 4; 39 15 8 4; 40 20 8 4; 41 0 8 8; 42 5 8 8; 43 10 8 8;
44 15 8 8; 45 20 8 8; 46 0.1 4 0.1; 47 5 4 0.1; 48 10 4 0.1; 49 15 4 0.1;
50 19.9 4 0.1; 51 19.9 4 4; 52 19.9 4 7.9; 53 5 4 7.9; 54 10 4 7.9;
55 15 4 7.9; 56 0.1 4 7.9; 57 0.1 4 4; 58 0.1 8 0.1; 59 5 8 0.1; 60 10 8 0.1;
61 15 8 0.1; 62 19.9 8 0.1; 63 19.9 8 4; 64 19.9 8 7.9; 65 5 8 7.9;
66 10 8 7.9; 67 15 8 7.9; 68 0.1 8 7.9; 69 0.1 8 4;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 3 5; 5 5 6; 6 5 7; 9 7 8; 10 7 9; 13 9 10; 14 2 12;
15 3 13; 16 5 15; 17 7 17; 18 9 19; 19 11 12; 20 12 13; 21 13 14; 22 13 15;
23 15 16; 24 15 17; 27 17 18; 28 17 19; 31 19 20; 32 12 22; 33 13 23; 34 15
25;
35 17 27; 36 19 29; 37 21 22; 38 22 23; 39 23 24; 40 23 25; 41 25 26; 42 25
27;
45 27 28; 46 27 29; 49 29 30; 58 2 31; 59 3 32; 60 5 33; 61 7 34; 62 9 35;
63 12 36; 64 13 37; 65 15 38; 66 17 39; 67 19 40; 68 22 41; 69 23 42; 70 25
43;
71 27 44; 72 29 45; 73 31 32; 74 32 33; 75 33 34; 77 34 35; 79 31 36; 80 32
37;
81 33 38; 82 34 39; 83 35 40; 84 36 37; 85 37 38; 86 38 39; 88 39 40; 90 36
41;
91 37 42; 92 38 43; 93 39 44; 94 40 45; 95 41 42; 96 42 43; 97 43 44; 99 44
45;
ELEMENT INCIDENCES SHELL
50 46 47 13 57; 51 47 48 15 13; 52 48 49 17 15; 53 49 50 51 17; 54 57 13 53
56;
55 13 15 54 53; 56 15 17 55 54; 57 17 51 52 55; 101 58 59 37 69;
102 59 60 38 37; 103 60 61 39 38; 104 61 62 63 39; 105 69 37 65 68;
106 37 38 66 65; 107 38 39 67 66; 108 39 63 64 67;
ELEMENT PROPERTY
50 TO 57 101 TO 108 THICKNESS 0.3
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
G 7.88462e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 6 9 10 13 TO 24 27 28 31 TO 36 38 40 42 46 58 TO 67 73 TO 75 77 -
79 TO 86 88 90 TO 97 99 TABLE ST W12X50
37 39 41 45 49 68 TO 72 TABLE ST W30X90
CONSTANTS
MATERIAL STEEL MEMB 1 TO 6 9 10 13 TO 24 27 28 31 TO 42 45 46 49 58 TO 75 -
77 79 TO 86 88 88 90 TO 97 99
MATERIAL CONCRETE MEMB 50 TO 57 101 TO 108
```

```
SUPPORTS
1 4 6 8 10 11 14 16 18 20 21 24 26 28 30 FIXED
*ELEMENT OFFSET
*50 101 JT1 0.1 0 0.1
*50 101 JT2 0 0 0.1
*50 101 JT4 0.1 0 0
*54 105 JT1 0.1 0 0
*54 105 JT3 0 0 -0.1
*54 105 JT4 0.1 0 -0.1
*51 102 JT1 0 0 0.1
*51 102 JT2 0 0 0.1
*55 106 JT3 0 0 -0.1
*55 106 JT4 0 0 -0.1
*52 103 JT1 0 0 0.1
*52 103 JT2 0 0 0.1
*56 107 JT3 0 0 -0.1
*56 107 JT4 0 0 -0.1
*53 104 JT1 0 0 0.1
*53 104 JT2 -0.1 0 0.1
*53 104 JT3 -0.1 0 0
*57 108 JT2 -0.1 0 0
*57 108 JT3 -0.1 0 -0.1
*57 108 JT4 0 0 -0.1
CONTROL RIGID CONTROL 2 JOINT 46
CONTROL RIGID CONTROL 3 JOINT 47
CONTROL RIGID CONTROL 5 JOINT 48
CONTROL RIGID CONTROL 7 JOINT 49
CONTROL RIGID CONTROL 9 JOINT 50
CONTROL RIGID CONTROL 19 JOINT 51
CONTROL RIGID CONTROL 29 JOINT 52
CONTROL RIGID CONTROL 27 JOINT 55
CONTROL RIGID CONTROL 25 JOINT 54
CONTROL RIGID CONTROL 23 JOINT 53
CONTROL RIGID CONTROL 22 JOINT 56
CONTROL RIGID CONTROL 12 JOINT 57
CONTROL RIGID CONTROL 31 JOINT 58
CONTROL RIGID CONTROL 32 JOINT 59
CONTROL RIGID CONTROL 33 JOINT 60
CONTROL RIGID CONTROL 34 JOINT 61
CONTROL RIGID CONTROL 35 JOINT 62
CONTROL RIGID CONTROL 40 JOINT 63
CONTROL RIGID CONTROL 45 JOINT 64
CONTROL RIGID CONTROL 44 JOINT 67
CONTROL RIGID CONTROL 43 JOINT 66
CONTROL RIGID CONTROL 42 JOINT 65
CONTROL RIGID CONTROL 41 JOINT 68
CONTROL RIGID CONTROL 36 JOINT 69
CUT OFF MODE SHAPE 50
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 3 5 7 9 12 13 15 17 19 22 23 25 27 29 31 TO 45 FX 10 FY -10 FZ 10
END DEFINE REFERENCE LOADS
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS X 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
```

```
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
LOAD 2 LOADTYPE None  TITLE RS_Z
SPECTRUM SRSS Z 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
LOAD 3 LOADTYPE None  TITLE RS_Y
SPECTRUM SRSS Y 1 ACC DAMP 0.05 LIN
0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167; 0.3
24.5167;
0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223;
1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778;
1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815;
1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985;
2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611;
2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321;
2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06 3.2048;
3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865;
3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942;
3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643;
4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167;
```

```
4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167;
4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167;
4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167;
5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167;
5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167;
5.82 2.45167; 5.88 2.45167; 5.94 2.45167;
PERFORM ANALYSIS PRINT ALL
PRINT ANALYSIS RESULTS
PRINT ELEMENT JOINT STRESSES
FINISH
```

STAAD.Pro Output

Output using the `ELEMENT OFFSET` method:

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:49:10                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: Response Spectrum Using Element Offset.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 09-MAR-21
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. SET PRINT 17
     7. UNIT METER KN
     8. JOINT COORDINATES
     9. 1 0 0 0; 2 0 4 0; 3 5 4 0; 4 5 0 0; 5 10 4 0; 6 10 0 0; 7 15 4 0; 8
15 0 0
    10. 9 20 4 0; 10 20 0 0; 11 0 0 4; 12 0 4 4; 13 5 4 4; 14 5 0 4; 15 10 4 4
    11. 16 10 0 4; 17 15 4 4; 18 15 0 4; 19 20 4 4; 20 20 0 4; 21 0 0 8; 22 0
4 8
    12. 23 5 4 8; 24 5 0 8; 25 10 4 8; 26 10 0 8; 27 15 4 8; 28 15 0 8; 29 20
4 8
    13. 30 20 0 8; 31 0 8 0; 32 5 8 0; 33 10 8 0; 34 15 8 0; 35 20 8 0; 36 0
8 4
    14. 37 5 8 4; 38 10 8 4; 39 15 8 4; 40 20 8 4; 41 0 8 8; 42 5 8 8; 43 10
8 8
    15. 44 15 8 8; 45 20 8 8
    16. MEMBER INCIDENCES
    17. 1 1 2; 2 2 3; 3 3 4; 4 3 5; 5 5 6; 6 5 7; 9 7 8; 10 7 9; 13 9 10; 14
2 12
    18. 15 3 13; 16 5 15; 17 7 17; 18 9 19; 19 11 12; 20 12 13; 21 13 14; 22
13 15
    19. 23 15 16; 24 15 17; 27 17 18; 28 17 19; 31 19 20; 32 12 22; 33 13 23;
34 15 25
```

```
    20. 35 17 27; 36 19 29; 37 21 22; 38 22 23; 39 23 24; 40 23 25; 41 25 26;
42 25 27
    21. 45 27 28; 46 27 29; 49 29 30; 58 2 31; 59 3 32; 60 5 33; 61 7 34; 62
9 35
    22. 63 12 36; 64 13 37; 65 15 38; 66 17 39; 67 19 40; 68 22 41; 69 23 42;
70 25 43
    23. 71 27 44; 72 29 45; 73 31 32; 74 32 33; 75 33 34; 77 34 35; 79 31 36;
80 32 37
    24. 81 33 38; 82 34 39; 83 35 40; 84 36 37; 85 37 38; 86 38 39; 88 39 40;
90 36 41
    25. 91 37 42; 92 38 43; 93 39 44; 94 40 45; 95 41 42; 96 42 43; 97 43 44;
99 44 45
    26. ELEMENT INCIDENCES SHELL
    27. 50 2 3 13 12; 51 3 5 15 13; 52 5 7 17 15; 53 7 9 19 17; 54 12 13 23 22
    28. 55 13 15 25 23; 56 15 17 27 25; 57 17 19 29 27; 101 31 32 37 36
    29. 102 32 33 38 37; 103 33 34 39 38; 104 34 35 40 39; 105 36 37 42 41
    30. 106 37 38 43 42; 107 38 39 44 43; 108 39 40 45 44
    31. ELEMENT PROPERTY
    32. 50 TO 57 101 TO 108 THICKNESS 0.3
    33. DEFINE MATERIAL START
    34. ISOTROPIC STEEL
    35. E 2.05E+08
    36. POISSON 0.3
    37. DENSITY 76.8195
    38. ALPHA 1.2E-05
      STAAD SPACE                                               -- PAGE NO.
2
    39. DAMP 0.03
    40. G 7.88462E+07
    41. TYPE STEEL
    42. STRENGTH RY 1.5 RT 1.2
    43. ISOTROPIC CONCRETE
    44. E 2.17185E+07
    45. POISSON 0.17
    46. DENSITY 23.5616
    47. ALPHA 1E-05
    48. DAMP 0.05
    49. G 9.28139E+06
    50. TYPE CONCRETE
    51. STRENGTH FCU 27579
    52. END DEFINE MATERIAL
    53. MEMBER PROPERTY AMERICAN
    54. 1 TO 6 9 10 13 TO 24 27 28 31 TO 36 38 40 42 46 58 TO 67 73 TO 75 77 -
    55. 79 TO 86 88 90 TO 97 99 TABLE ST W12X50
    56. 37 39 41 45 49 68 TO 72 TABLE ST W30X90
    57. CONSTANTS
    58. MATERIAL STEEL MEMB 1 TO 6 9 10 13 TO 24 27 28 31 TO 42 45 46 49 58
TO 75 -
    59. 77 79 TO 86 88 90 TO 97 99
    60. MATERIAL CONCRETE MEMB 50 TO 57 101 TO 108
    61. SUPPORTS
    62. 1 4 6 8 10 11 14 16 18 20 21 24 26 28 30 FIXED
    63. ELEMENT OFFSET
    64. 50 101 JT1 0.1 0 0.1
    65. 50 101 JT2 0 0 0.1
    66. 50 101 JT4 0.1 0 0
    67. 54 105 JT1 0.1 0 0
    68. 54 105 JT3 0 0 -0.1
```

```
    69. 54 105 JT4 0.1 0 -0.1
    70. 51 102 JT1 0 0 0.1
    71. 51 102 JT2 0 0 0.1
    72. 55 106 JT3 0 0 -0.1
    73. 55 106 JT4 0 0 -0.1
    74. 52 103 JT1 0 0 0.1
    75. 52 103 JT2 0 0 0.1
    76. 56 107 JT3 0 0 -0.1
    77. 56 107 JT4 0 0 -0.1
    78. 53 104 JT1 0 0 0.1
    79. 53 104 JT2 -0.1 0 0.1
    80. 53 104 JT3 -0.1 0 0
    81. 57 108 JT2 -0.1 0 0
    82. 57 108 JT3 -0.1 0 -0.1
    83. 57 108 JT4 0 0 -0.1
    84. CUT OFF MODE SHAPE 50
    85. DEFINE REFERENCE LOADS
    86. LOAD R1 LOADTYPE MASS  TITLE REF LOAD CASE 1
    87. JOINT LOAD
    88. 2 3 5 7 9 12 13 15 17 19 22 23 25 27 29 31 TO 45 FX 10 FY -10 FZ 10
    89. END DEFINE REFERENCE LOADS
   *** NOTE: MASS MODEL FORMED WILL BE USED IN SEISMIC/RESPONSE/TIME HISTORY
LOADING, IF ANY.
             IF MASS MODEL IS SEPARATELY PROVIDED IN INDIVIDUAL LOADING, THE
GENERATED MASS
             WILL BE REPLACED BY THE MASS PROVIDED IN INDIVIDUAL LOADING.
      STAAD SPACE                                              -- PAGE NO.
3
    90. LOAD 1 LOADTYPE NONE  TITLE RS_X
    91. SPECTRUM SRSS X 1 ACC DAMP 0.05 LIN
    92. 0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167;
0.3 24.5167
    93. 0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444
    94. 0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963
    95. 0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223
    96. 1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778
    97. 1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815
    98. 1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985
    99. 2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611
   100. 2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321
   101. 2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06
3.2048
   102. 3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865
   103. 3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942
   104. 3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643
   105. 4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167
   106. 4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167
   107. 4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167
   108. 4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167
   109. 5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167
   110. 5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167
   111. 5.82 2.45167; 5.88 2.45167; 5.94 2.45167
   112. LOAD 2 LOADTYPE NONE  TITLE RS_Z
   113. SPECTRUM SRSS Z 1 ACC DAMP 0.05 LIN
   114. 0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167;
0.3 24.5167
   115. 0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444
   116. 0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963
```

```
   117. 0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223
   118. 1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778
   119. 1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815
   120. 1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985
   121. 2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611
   122. 2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321
   123. 2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06
3.2048
   124. 3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865
   125. 3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942
   126. 3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643
   127. 4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167
   128. 4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167
   129. 4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167
   130. 4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167
   131. 5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167
   132. 5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167
   133. 5.82 2.45167; 5.88 2.45167; 5.94 2.45167
   134. LOAD 3 LOADTYPE NONE  TITLE RS_Y
   135. SPECTRUM SRSS Y 1 ACC DAMP 0.05 LIN
   136. 0 9.80667; 0.06 18.6327; 0.12 24.5167; 0.18 24.5167; 0.24 24.5167;
0.3 24.5167
   137. 0.36 24.5167; 0.42 23.3492; 0.48 20.4306; 0.54 18.1605; 0.6 16.3444
   138. 0.66 14.8586; 0.72 13.6204; 0.78 12.5727; 0.84 11.6746; 0.9 10.8963
   139. 0.96 10.2153; 1.02 9.61438; 1.08 9.08025; 1.14 8.60234; 1.2 8.17223
   140. 1.26 7.78307; 1.32 7.4293; 1.38 7.10628; 1.44 6.81019; 1.5 6.53778
   141. 1.56 6.28633; 1.62 6.0535; 1.68 5.83731; 1.74 5.63602; 1.8 5.44815
   142. 1.86 5.27241; 1.92 5.10764; 1.98 4.95287; 2.04 4.80719; 2.1 4.66985
   143. 2.16 4.54013; 2.22 4.41742; 2.28 4.30117; 2.34 4.19089; 2.4 4.08611
   144. 2.46 3.98645; 2.52 3.89154; 2.58 3.80104; 2.64 3.71465; 2.7 3.6321
   145. 2.76 3.55314; 2.82 3.47754; 2.88 3.4051; 2.94 3.3356; 3 3.26889; 3.06
3.2048
      STAAD SPACE                                                -- PAGE NO.
4
   146. 3.12 3.14317; 3.18 3.08386; 3.24 3.02675; 3.3 2.97172; 3.36 2.91865
   147. 3.42 2.86745; 3.48 2.81801; 3.54 2.77025; 3.6 2.72408; 3.66 2.67942
   148. 3.72 2.6362; 3.78 2.59436; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643
   149. 4.02 2.45167; 4.08 2.45167; 4.14 2.45167; 4.2 2.45167; 4.26 2.45167
   150. 4.32 2.45167; 4.38 2.45167; 4.44 2.45167; 4.5 2.45167; 4.56 2.45167
   151. 4.62 2.45167; 4.68 2.45167; 4.74 2.45167; 4.8 2.45167; 4.86 2.45167
   152. 4.92 2.45167; 4.98 2.45167; 5.04 2.45167; 5.1 2.45167; 5.16 2.45167
   153. 5.22 2.45167; 5.28 2.45167; 5.34 2.45167; 5.4 2.45167; 5.46 2.45167
   154. 5.52 2.45167; 5.58 2.45167; 5.64 2.45167; 5.7 2.45167; 5.76 2.45167
   155. 5.82 2.45167; 5.88 2.45167; 5.94 2.45167
   156. PERFORM ANALYSIS PRINT ALL
             P R O B L E M   S T A T I S T I C S
             -----------------------------------
     NUMBER OF JOINTS         45  NUMBER OF MEMBERS      74
     NUMBER OF PLATES         16  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     15
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =     180
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
      STAAD SPACE                                                -- PAGE NO.
5
   *** NOTE: CAPACITY FOR MAXIMUM #    256 LOAD CASES IS ASSIGNED FOR PLATE
LOAD.
```

```
   LOADING     1  LOADTYPE NONE  TITLE RS_X
   -----------
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
           DIRECTIONAL VALUES:                 SCALE FACTOR =    1.00
    X =  1.00   Y =  0.00   Z =  0.00          DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           0.0010          9.8067
           0.0600         18.6327
           0.1200         24.5167
           0.1800         24.5167
           0.2400         24.5167
           0.3000         24.5167
           0.3600         24.5167
           0.4200         23.3492
           0.4800         20.4306
           0.5400         18.1605
           0.6000         16.3444
           0.6600         14.8586
           0.7200         13.6204
           0.7800         12.5727
           0.8400         11.6746
           0.9000         10.8963
           0.9600         10.2153
           1.0200          9.6144
           1.0800          9.0802
           1.1400          8.6023
           1.2000          8.1722
           1.2600          7.7831
           1.3200          7.4293
           1.3800          7.1063
           1.4400          6.8102
           1.5000          6.5378
           1.5600          6.2863
           1.6200          6.0535
           1.6800          5.8373
           1.7400          5.6360
           1.8000          5.4482
           1.8600          5.2724
           1.9200          5.1076
           1.9800          4.9529
           2.0400          4.8072
           2.1000          4.6698
           2.1600          4.5401
           2.2200          4.4174
           2.2800          4.3012
      STAAD SPACE                                                -- PAGE NO.
6
           2.3400          4.1909
           2.4000          4.0861
           2.4600          3.9864
           2.5200          3.8915
           2.5800          3.8010
           2.6400          3.7146
           2.7000          3.6321
           2.7600          3.5531
           2.8200          3.4775
           2.8800          3.4051
```

```
          2.9400           3.3356
          3.0000           3.2689
          3.0600           3.2048
          3.1200           3.1432
          3.1800           3.0839
          3.2400           3.0268
          3.3000           2.9717
          3.3600           2.9186
          3.4200           2.8674
          3.4800           2.8180
          3.5400           2.7703
          3.6000           2.7241
          3.6600           2.6794
          3.7200           2.6362
          3.7800           2.5944
          3.8400           2.5538
          3.9000           2.5145
          3.9600           2.4764
          4.0200           2.4517
          4.0800           2.4517
          4.1400           2.4517
          4.2000           2.4517
          4.2600           2.4517
          4.3200           2.4517
          4.3800           2.4517
          4.4400           2.4517
          4.5000           2.4517
          4.5600           2.4517
          4.6200           2.4517
          4.6800           2.4517
          4.7400           2.4517
          4.8000           2.4517
          4.8600           2.4517
          4.9200           2.4517
          4.9800           2.4517
          5.0400           2.4517
          5.1000           2.4517
          5.1600           2.4517
          5.2200           2.4517
          5.2800           2.4517
          5.3400           2.4517
          5.4000           2.4517
          5.4600           2.4517
          5.5200           2.4517
          5.5800           2.4517
          5.6400           2.4517
     STAAD SPACE                                               -- PAGE NO.
7
          5.7000           2.4517
          5.7600           2.4517
          5.8200           2.4517
          5.8800           2.4517
          5.9400           2.4517
   ***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM
            THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
   LOADING     2  LOADTYPE NONE  TITLE RS_Z
   -----------
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
```

```
   ------------------------
           DIRECTIONAL VALUES:                  SCALE FACTOR =    1.00
    X =  0.00   Y =  0.00   Z =  1.00          DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           0.0010          9.8067
           0.0600         18.6327
           0.1200         24.5167
           0.1800         24.5167
           0.2400         24.5167
           0.3000         24.5167
           0.3600         24.5167
           0.4200         23.3492
           0.4800         20.4306
           0.5400         18.1605
           0.6000         16.3444
           0.6600         14.8586
           0.7200         13.6204
           0.7800         12.5727
           0.8400         11.6746
           0.9000         10.8963
           0.9600         10.2153
           1.0200          9.6144
           1.0800          9.0802
           1.1400          8.6023
           1.2000          8.1722
           1.2600          7.7831
           1.3200          7.4293
           1.3800          7.1063
           1.4400          6.8102
           1.5000          6.5378
           1.5600          6.2863
           1.6200          6.0535
           1.6800          5.8373
           1.7400          5.6360
           1.8000          5.4482
      STAAD SPACE                                               -- PAGE NO.
8
           1.8600          5.2724
           1.9200          5.1076
           1.9800          4.9529
           2.0400          4.8072
           2.1000          4.6698
           2.1600          4.5401
           2.2200          4.4174
           2.2800          4.3012
           2.3400          4.1909
           2.4000          4.0861
           2.4600          3.9864
           2.5200          3.8915
           2.5800          3.8010
           2.6400          3.7146
           2.7000          3.6321
           2.7600          3.5531
           2.8200          3.4775
           2.8800          3.4051
           2.9400          3.3356
           3.0000          3.2689
           3.0600          3.2048
```

```
          3.1200           3.1432
          3.1800           3.0839
          3.2400           3.0268
          3.3000           2.9717
          3.3600           2.9186
          3.4200           2.8674
          3.4800           2.8180
          3.5400           2.7703
          3.6000           2.7241
          3.6600           2.6794
          3.7200           2.6362
          3.7800           2.5944
          3.8400           2.5538
          3.9000           2.5145
          3.9600           2.4764
          4.0200           2.4517
          4.0800           2.4517
          4.1400           2.4517
          4.2000           2.4517
          4.2600           2.4517
          4.3200           2.4517
          4.3800           2.4517
          4.4400           2.4517
          4.5000           2.4517
          4.5600           2.4517
          4.6200           2.4517
          4.6800           2.4517
          4.7400           2.4517
          4.8000           2.4517
          4.8600           2.4517
          4.9200           2.4517
          4.9800           2.4517
          5.0400           2.4517
          5.1000           2.4517
          5.1600           2.4517
     STAAD SPACE                                             -- PAGE NO.
9
          5.2200           2.4517
          5.2800           2.4517
          5.3400           2.4517
          5.4000           2.4517
          5.4600           2.4517
          5.5200           2.4517
          5.5800           2.4517
          5.6400           2.4517
          5.7000           2.4517
          5.7600           2.4517
          5.8200           2.4517
          5.8800           2.4517
          5.9400           2.4517
   LOADING     3  LOADTYPE NONE  TITLE RS_Y
   -----------
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
          DIRECTIONAL VALUES:                 SCALE FACTOR =    1.00
    X =  0.00   Y =  1.00   Z =  0.00         DAMPING FACTOR = 0.050
          PERIOD  VS.  ACCELERATION
          0.0010           9.8067
```

```
          0.0600          18.6327
          0.1200          24.5167
          0.1800          24.5167
          0.2400          24.5167
          0.3000          24.5167
          0.3600          24.5167
          0.4200          23.3492
          0.4800          20.4306
          0.5400          18.1605
          0.6000          16.3444
          0.6600          14.8586
          0.7200          13.6204
          0.7800          12.5727
          0.8400          11.6746
          0.9000          10.8963
          0.9600          10.2153
          1.0200           9.6144
          1.0800           9.0802
          1.1400           8.6023
          1.2000           8.1722
          1.2600           7.7831
          1.3200           7.4293
          1.3800           7.1063
          1.4400           6.8102
          1.5000           6.5378
          1.5600           6.2863
      STAAD SPACE                                               -- PAGE NO.
10
          1.6200           6.0535
          1.6800           5.8373
          1.7400           5.6360
          1.8000           5.4482
          1.8600           5.2724
          1.9200           5.1076
          1.9800           4.9529
          2.0400           4.8072
          2.1000           4.6698
          2.1600           4.5401
          2.2200           4.4174
          2.2800           4.3012
          2.3400           4.1909
          2.4000           4.0861
          2.4600           3.9864
          2.5200           3.8915
          2.5800           3.8010
          2.6400           3.7146
          2.7000           3.6321
          2.7600           3.5531
          2.8200           3.4775
          2.8800           3.4051
          2.9400           3.3356
          3.0000           3.2689
          3.0600           3.2048
          3.1200           3.1432
          3.1800           3.0839
          3.2400           3.0268
          3.3000           2.9717
          3.3600           2.9186
```

```
           3.4200           2.8674
           3.4800           2.8180
           3.5400           2.7703
           3.6000           2.7241
           3.6600           2.6794
           3.7200           2.6362
           3.7800           2.5944
           3.8400           2.5538
           3.9000           2.5145
           3.9600           2.4764
           4.0200           2.4517
           4.0800           2.4517
           4.1400           2.4517
           4.2000           2.4517
           4.2600           2.4517
           4.3200           2.4517
           4.3800           2.4517
           4.4400           2.4517
           4.5000           2.4517
           4.5600           2.4517
           4.6200           2.4517
           4.6800           2.4517
           4.7400           2.4517
           4.8000           2.4517
           4.8600           2.4517
           4.9200           2.4517
      STAAD SPACE                                                -- PAGE NO.
11
           4.9800           2.4517
           5.0400           2.4517
           5.1000           2.4517
           5.1600           2.4517
           5.2200           2.4517
           5.2800           2.4517
           5.3400           2.4517
           5.4000           2.4517
           5.4600           2.4517
           5.5200           2.4517
           5.5800           2.4517
           5.6400           2.4517
           5.7000           2.4517
           5.7600           2.4517
           5.8200           2.4517
           5.8800           2.4517
           5.9400           2.4517
 EIGEN METHOD   : SUBSPACE
 -------------------------
 NUMBER OF MODES REQUESTED             =    50
 NUMBER OF EXISTING MASSES IN THE MODEL =    90
 NUMBER OF MODES THAT WILL BE USED     =    50
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      STAAD SPACE                                                -- PAGE NO.
12
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                       3.299                 0.30314
         2                       4.513                 0.22157
         3                       8.747                 0.11432
```

```
         4                      10.076                  0.09924
         5                      12.601                  0.07936
         6                      32.152                  0.03110
         7                      67.750                  0.01476
         8                      67.990                  0.01471
         9                      68.843                  0.01453
        10                      70.679                  0.01415
        11                      71.621                  0.01396
        12                      71.712                  0.01394
        13                      73.050                  0.01369
        14                      75.299                  0.01328
        15                      78.879                  0.01268
        16                      85.007                  0.01176
        17                      91.029                  0.01099
        18                      91.989                  0.01087
        19                      92.101                  0.01086
        20                      92.960                  0.01076
        21                      93.797                  0.01066
        22                      94.286                  0.01061
        23                      95.411                  0.01048
    C O M P O S I T E   D A M P I N G   SUMMARY
    MODE  STRAIN ENERGY   DAMP*ENERGY  COMPOSITE DAMPING
       1   2.565972E+01   7.850347E-01    0.0306
       2   3.095850E+01   9.899908E-01    0.0320
       3   1.802559E+02   5.449556E+00    0.0302
       4   1.768272E+02   6.119833E+00    0.0346
       5   2.445134E+02   7.521179E+00    0.0308
       6   1.800416E+03   5.736233E+01    0.0319
       7   4.479752E+03   1.343998E+02    0.0300
       8   2.413502E+03   7.257655E+01    0.0301
       9   2.476470E+03   7.508756E+01    0.0303
      10   2.368591E+03   7.320725E+01    0.0309
      11   2.206916E+03   6.889431E+01    0.0312
      12   2.062698E+03   6.442215E+01    0.0312
      13   2.022382E+03   6.397979E+01    0.0316
      14   2.200408E+03   7.196828E+01    0.0327
      15   2.302982E+03   7.767730E+01    0.0337
      16   2.385968E+03   8.530492E+01    0.0358
      17   1.473957E+04   6.710149E+02    0.0455
      18   4.121837E+03   1.266205E+02    0.0307
      19   3.325657E+03   1.010278E+02    0.0304
      20   3.560806E+03   1.207387E+02    0.0339
      STAAD SPACE                                               -- PAGE NO.
13
    MODE  STRAIN ENERGY   DAMP*ENERGY  COMPOSITE DAMPING
      21   1.035648E+04   4.316491E+02    0.0417
      22   3.490804E+03   1.076316E+02    0.0308
      23   3.566267E+03   1.110562E+02    0.0311
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       5.984143E-24  1.308434E-04  2.830457E+02    2.051552E+02
         2       2.674250E+01  1.755244E-29  7.301247E-23    1.322364E+02
         3       5.065707E-24  5.100442E-04  1.695390E+01    2.049607E+02
         4       2.450581E+02  5.450377E-28  4.994469E-27    1.515248E+02
         5       5.653800E-01  4.194184E-27  1.131665E-25    1.339801E+02
         6       2.762138E+01  1.415896E-21  6.831347E-22    1.515326E+02
```

```
        7       6.469897E-19  1.709744E+02  2.191617E-04    8.491271E+01
        8       5.191913E-04  7.860724E-19  2.581121E-22    4.542528E+01
        9       2.700640E-19  3.956472E-03  1.269517E-06    4.546258E+01
       10       2.182148E-04  1.841933E-22  3.151962E-23    4.125279E+01
       11       2.053212E-18  1.742969E+01  3.337675E-06    3.743278E+01
       12       1.262537E-03  1.280337E-17  1.592862E-21    3.489792E+01
       13       6.567201E-19  5.102359E-01  2.157681E-07    3.297345E+01
       14       1.920060E-17  1.225476E+01  4.658799E-06    3.376513E+01
       15       1.813976E-04  1.101497E-20  1.075361E-21    3.220428E+01
       16       6.906754E-17  5.351847E-01  1.310061E-06    2.872755E+01
       17       5.837981E-15  3.860367E-01  7.409204E-08    1.547635E+02
       18       1.179917E-16  7.473509E+01  1.455537E-04    4.238037E+01
       19       3.492625E-04  5.984215E-21  1.433175E-24    3.411115E+01
       20       4.208206E-16  7.160740E+00  1.785032E-05    3.585090E+01
       21       5.498390E-16  1.703037E-01  1.462900E-07    1.024190E+02
       22       7.887138E-05  9.206345E-21  2.039361E-24    3.416440E+01
       23       1.080083E-18  6.280113E-03  1.077932E-09    3.408452E+01
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   3.000000E+02  3.000000E+02  3.000000E+02 KN
     MISSING WEIGHT X Y Z  -1.010015E-02 -1.583268E+01 -2.927266E-05 KN
       MODAL WEIGHT X Y Z   2.999899E+02  2.841673E+02  3.000000E+02 KN
      STAAD SPACE                                              -- PAGE NO.
14
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     2.50001       0.05000
              2                     2.50001       0.05000
              3                     2.44320       0.05000
              4                     2.29243       0.05000
              5                     2.09360       0.05000
              6                     1.45919       0.05000
              7                     1.20990       0.05000
              8                     1.20911       0.05000
              9                     1.20633       0.05000
             10                     1.20057       0.05000
             11                     1.19774       0.05000
             12                     1.19747       0.05000
             13                     1.19357       0.05000
             14                     1.18733       0.05000
             15                     1.17814       0.05000
             16                     1.16420       0.05000
             17                     1.15232       0.05000
             18                     1.15058       0.05000
             19                     1.15037       0.05000
             20                     1.14884       0.05000
             21                     1.14738       0.05000
             22                     1.14654       0.05000
             23                     1.14463       0.05000
      STAAD SPACE                                              -- PAGE NO.
15
 GENERALIZED WEIGHT & MODAL DISPLACEMENT
  ADDITIONAL MODAL VALUES      WEIGHT IN KN    LENGTH IN METE
  -----------------------------------------------------------
      MODE   GENERALIZED WEIGHT  MODAL DISPLACEMENT
         1       2.0515518E+02     9.7465049E-15
         2       1.3223642E+02     1.3710564E-02
         3       2.0496067E+02     1.2469291E-15
         4       1.5152478E+02     7.1324722E-03
```

```
         5       1.3398013E+02    -2.1276414E-04
         6       1.5153264E+02     1.4970403E-04
         7       8.4912708E+01    -5.7154246E-15
         8       4.5425281E+01     2.1965811E-07
         9       4.5462584E+01    -4.8731541E-15
        10       4.1252788E+01     1.3730443E-07
        11       3.7432782E+01    -1.3584235E-14
        12       3.4897917E+01    -3.4790946E-07
        13       3.2973453E+01     7.8410346E-15
        14       3.3765129E+01     3.9226302E-14
        15       3.2204276E+01    -1.1163363E-07
        16       2.8727553E+01     6.2053309E-14
        17       1.5476347E+02    -2.1216448E-13
        18       4.2380370E+01    -5.6357254E-14
        19       3.4111151E+01     1.0779616E-07
        20       3.5850904E+01     1.1314317E-13
        21       1.0241904E+02     7.5062030E-14
        22       3.4164403E+01     4.8677230E-08
        23       3.4084516E+01    -5.5599976E-15
 FLOOR MODAL BASE ACTIONS
  FLOOR MODAL BASE ACTIONS  FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX           FY           FZ           MX
MY          MZ
   STORY NUMBER    3  HEIGHT       8.00
      1      0.303        0.00         0.00         0.00         0.00
-0.00        0.00
      2      0.222       41.35        -0.00        -0.00        -0.00
-857.44     -331.40
      3      0.114       -0.00        -0.00        -0.00        -0.00
0.00       -0.00
      4      0.099      359.99        -0.00         0.00         0.00
2616.88    -2901.05
      5      0.079        5.86        -0.00        -0.00        -0.00
-38.80      -47.32
      6      0.031      -38.50        -0.00        -0.00         0.00
-262.72      319.15
      7      0.015       -0.00        -0.00         0.00         0.00
-0.00       -0.00
      8      0.015        0.00        -0.00         0.00         0.00
0.02        1.55
      9      0.015       -0.00        -0.00         0.00         0.00
-0.00       -0.00
     10      0.014        0.00         0.00         0.00        -0.00
0.00        0.00
     11      0.014       -0.00        -0.00         0.00         0.00
-0.00       -0.00
     12      0.014        0.01        -0.00         0.00         0.00
0.05        0.88
     13      0.014       -0.00         0.00         0.00        -0.00
-0.00        0.00
     14      0.013       -0.00         0.00        -0.00        -0.00
0.00        0.00
     15      0.013        0.00         0.00         0.00        -0.00
0.00       -0.12
     16      0.012       -0.00        -0.00         0.00         0.00
```

```
-0.00       -0.00
     17      0.011       -0.00        0.00        0.00       -0.00
-0.00        0.00
     18      0.011       -0.00       -0.00       -0.00        0.00
0.00       -0.00
     19      0.011        0.00       -0.00       -0.00        0.00
0.01        0.85
     20      0.011       -0.00        0.00        0.00       -0.00
-0.00        0.00
     21      0.011       -0.00       -0.00        0.00        0.00
-0.00       -0.00
     22      0.011        0.00       -0.00       -0.00        0.00
0.00       -0.01
     23      0.010       -0.00       -0.00        0.00        0.00
-0.00       -0.00
   STORY NUMBER    2  HEIGHT       4.00
      1      0.303        0.00        0.00        0.00        0.00
-0.00        0.00
      2      0.222       25.51       -0.00       -0.00       -0.00
-473.80     -102.52
      3      0.114        0.00       -0.00        0.00        0.00
-0.00       -0.00
      4      0.099      201.79        0.00       -0.00       -0.00
827.20     -822.77
      5      0.079       -4.68        0.00       -0.00       -0.00
104.83       18.44
      6      0.031       78.80        0.00       -0.00       -0.00
552.34     -309.02
      7      0.015        0.00       -0.00       -0.00        0.00
0.00       -0.00
      8      0.015       -0.00       -0.00        0.00        0.00
-0.01        0.98
      9      0.015        0.00       -0.00       -0.00        0.00
0.00       -0.00
     10      0.014       -0.00       -0.00        0.00        0.00
-0.00        0.01
      STAAD SPACE                                              -- PAGE NO.
16
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD         FX          FY          FZ          MX
MY          MZ
     11      0.014        0.00       -0.00       -0.00        0.00
0.00       -0.00
     12      0.014       -0.01       -0.00        0.00        0.00
-0.04        0.61
     13      0.014        0.00        0.00       -0.00       -0.00
0.00        0.00
     14      0.013        0.00        0.00        0.00       -0.00
-0.00        0.00
     15      0.013       -0.00       -0.00        0.00        0.00
-0.00       -0.07
     16      0.012        0.00       -0.00       -0.00        0.00
0.00       -0.00
     17      0.011        0.00        0.00       -0.00       -0.00
0.00        0.00
```

```
     18      0.011         0.00        -0.00         0.00         0.00
-0.00      -0.00
     19      0.011        -0.00        -0.00         0.00         0.00
-0.01       0.54
     20      0.011         0.00         0.00        -0.00        -0.00
0.00       0.00
     21      0.011         0.00        -0.00        -0.00         0.00
0.00      -0.00
     22      0.011        -0.00        -0.00         0.00         0.00
-0.00       0.00
     23      0.010         0.00        -0.00        -0.00         0.00
0.00      -0.00
   STORY NUMBER    1  HEIGHT       0.00
      1      0.303         0.00         0.00         0.00         0.00
0.00       0.00
      2      0.222         0.00         0.00         0.00         0.00
0.00       0.00
      3      0.114         0.00         0.00         0.00         0.00
0.00       0.00
      4      0.099         0.00         0.00         0.00         0.00
0.00       0.00
      5      0.079         0.00         0.00         0.00         0.00
0.00       0.00
      6      0.031         0.00         0.00         0.00         0.00
0.00       0.00
      7      0.015         0.00         0.00         0.00         0.00
0.00       0.00
      8      0.015         0.00         0.00         0.00         0.00
0.00       0.00
      9      0.015         0.00         0.00         0.00         0.00
0.00       0.00
     10      0.014         0.00         0.00         0.00         0.00
0.00       0.00
     11      0.014         0.00         0.00         0.00         0.00
0.00       0.00
     12      0.014         0.00         0.00         0.00         0.00
0.00       0.00
     13      0.014         0.00         0.00         0.00         0.00
0.00       0.00
     14      0.013         0.00         0.00         0.00         0.00
0.00       0.00
     15      0.013         0.00         0.00         0.00         0.00
0.00       0.00
     16      0.012         0.00         0.00         0.00         0.00
0.00       0.00
     17      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     18      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     19      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     20      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     21      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     22      0.011         0.00         0.00         0.00         0.00
0.00       0.00
     23      0.010         0.00         0.00         0.00         0.00
```

```
0.00        0.00
      STAAD SPACE                                                -- PAGE NO.
17
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                              MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.303        0.00        0.00        0.00        0.00
-0.00        0.00
      2      0.222       66.86       -0.00       -0.00       -0.00
-1331.24     -433.92
      3      0.114        0.00       -0.00        0.00       -0.00
-0.00       -0.00
      4      0.099      561.78        0.00       -0.00        0.00
3444.08    -3723.82
      5      0.079        1.18        0.00       -0.00       -0.00
66.03      -28.88
      6      0.031       40.30        0.00       -0.00       -0.00
289.62       10.13
      7      0.015        0.00       -0.00        0.00        0.00
-0.00       -0.00
      8      0.015        0.00       -0.00        0.00        0.00
0.00        2.53
      9      0.015        0.00       -0.00        0.00        0.00
-0.00       -0.00
     10      0.014        0.00       -0.00        0.00        0.00
-0.00        0.01
     11      0.014        0.00       -0.00        0.00        0.00
-0.00       -0.00
     12      0.014        0.00       -0.00        0.00        0.00
0.01        1.49
     13      0.014        0.00        0.00        0.00       -0.00
-0.00        0.00
     14      0.013        0.00        0.00       -0.00       -0.00
0.00        0.00
     15      0.013        0.00        0.00        0.00       -0.00
0.00       -0.19
     16      0.012        0.00       -0.00        0.00        0.00
-0.00       -0.00
     17      0.011        0.00        0.00       -0.00       -0.00
0.00        0.00
     18      0.011        0.00       -0.00       -0.00        0.00
0.00       -0.00
     19      0.011        0.00       -0.00        0.00        0.00
0.00        1.39
     20      0.011        0.00        0.00        0.00       -0.00
-0.00        0.00
     21      0.011        0.00       -0.00       -0.00        0.00
0.00       -0.00
     22      0.011        0.00       -0.00        0.00        0.00
0.00       -0.01
     23      0.010        0.00       -0.00        0.00        0.00
-0.00       -0.00
      STAAD SPACE                                                -- PAGE NO.
18
```

```
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE     X      Y      Z     SUMM-X    SUMM-Y   SUMM-Z        X         Y        Z
   1    0.00   0.00 94.35    0.000     0.000    94.349      0.00      0.00
0.00
   2    8.91   0.00  0.00    8.914     0.000    94.349     66.86      0.00
0.00
   3    0.00   0.00  5.65    8.914     0.000   100.000      0.00      0.00
0.00
   4   81.69   0.00  0.00   90.600     0.000   100.000    561.78      0.00
0.00
   5    0.19   0.00  0.00   90.789     0.000   100.000      1.18      0.00
0.00
   6    9.21   0.00  0.00   99.996     0.000   100.000     40.30      0.00
0.00
   7    0.00  56.99  0.00   99.996    56.992   100.000      0.00      0.00
0.00
   8    0.00   0.00  0.00   99.996    56.992   100.000      0.00      0.00
0.00
   9    0.00   0.00  0.00   99.996    56.993   100.000      0.00      0.00
0.00
  10    0.00   0.00  0.00   99.996    56.993   100.000      0.00      0.00
0.00
  11    0.00   5.81  0.00   99.996    62.803   100.000      0.00      0.00
0.00
  12    0.00   0.00  0.00   99.996    62.803   100.000      0.00      0.00
0.00
  13    0.00   0.17  0.00   99.996    62.973   100.000      0.00      0.00
0.00
  14    0.00   4.08  0.00   99.996    67.058   100.000      0.00      0.00
0.00
  15    0.00   0.00  0.00   99.996    67.058   100.000      0.00      0.00
0.00
  16    0.00   0.18  0.00   99.996    67.236   100.000      0.00      0.00
0.00
  17    0.00   0.13  0.00   99.996    67.365   100.000      0.00      0.00
0.00
  18    0.00  24.91  0.00   99.996    92.277   100.000      0.00      0.00
0.00
  19    0.00   0.00  0.00   99.997    92.277   100.000      0.00      0.00
0.00
  20    0.00   2.39  0.00   99.997    94.664   100.000      0.00      0.00
0.00
  21    0.00   0.06  0.00   99.997    94.720   100.000      0.00      0.00
0.00
  22    0.00   0.00  0.00   99.997    94.720   100.000      0.00      0.00
0.00
  23    0.00   0.00  0.00   99.997    94.722   100.000      0.00      0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR    567.18      0.00
0.00
                                  TOTAL 10PCT SHEAR    567.18      0.00
0.00
                                  TOTAL ABS   SHEAR    670.13      0.00
0.00
```

```
  RESPONSE SPECTRUM LOAD     2
     RESPONSE LOAD CASE      2
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN          GENERALIZED
      MODE           X              Y              Z             WEIGHT
         1       5.984143E-24  1.308434E-04  2.830457E+02    2.051552E+02
         2       2.674250E+01  1.755244E-29  7.301247E-23    1.322364E+02
         3       5.065707E-24  5.100442E-04  1.695390E+01    2.049607E+02
         4       2.450581E+02  5.450377E-28  4.994469E-27    1.515248E+02
         5       5.653800E-01  4.194184E-27  1.131665E-25    1.339801E+02
      STAAD SPACE                                                  -- PAGE NO.
19
         6       2.762138E+01  1.415896E-21  6.831347E-22    1.515326E+02
         7       6.469897E-19  1.709744E+02  2.191617E-04    8.491271E+01
         8       5.191913E-04  7.860724E-19  2.581121E-22    4.542528E+01
         9       2.700640E-19  3.956472E-03  1.269517E-06    4.546258E+01
        10       2.182148E-04  1.841933E-22  3.151962E-23    4.125279E+01
        11       2.053212E-18  1.742969E+01  3.337675E-06    3.743278E+01
        12       1.262537E-03  1.280337E-17  1.592862E-21    3.489792E+01
        13       6.567201E-19  5.102359E-01  2.157681E-07    3.297345E+01
        14       1.920060E-17  1.225476E+01  4.658799E-06    3.376513E+01
        15       1.813976E-04  1.101497E-20  1.075361E-21    3.220428E+01
        16       6.906754E-17  5.351847E-01  1.310061E-06    2.872755E+01
        17       5.837981E-15  3.860367E-01  7.409204E-08    1.547635E+02
        18       1.179917E-16  7.473509E+01  1.455537E-04    4.238037E+01
        19       3.492625E-04  5.984215E-21  1.433175E-24    3.411115E+01
        20       4.208206E-16  7.160740E+00  1.785032E-05    3.585090E+01
        21       5.498390E-16  1.703037E-01  1.462900E-07    1.024190E+02
        22       7.887138E-05  9.206345E-21  2.039361E-24    3.416440E+01
        23       1.080083E-18  6.280113E-03  1.077932E-09    3.408452E+01
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   3.000000E+02  3.000000E+02  3.000000E+02 KN
     MISSING WEIGHT X Y Z  -1.010015E-02 -1.583268E+01 -2.927266E-05 KN
       MODAL WEIGHT X Y Z   2.999899E+02  2.841673E+02  3.000000E+02 KN
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     2.50001      0.05000
              2                     2.50001      0.05000
              3                     2.44320      0.05000
              4                     2.29243      0.05000
              5                     2.09360      0.05000
              6                     1.45919      0.05000
              7                     1.20990      0.05000
              8                     1.20911      0.05000
              9                     1.20633      0.05000
             10                     1.20057      0.05000
             11                     1.19774      0.05000
             12                     1.19747      0.05000
             13                     1.19357      0.05000
             14                     1.18733      0.05000
             15                     1.17814      0.05000
             16                     1.16420      0.05000
             17                     1.15232      0.05000
             18                     1.15058      0.05000
             19                     1.15037      0.05000
             20                     1.14884      0.05000
             21                     1.14738      0.05000
             22                     1.14654      0.05000
             23                     1.14463      0.05000
```

```
      STAAD SPACE                                              -- PAGE NO.
20
 GENERALIZED WEIGHT & MODAL DISPLACEMENT
  ADDITIONAL MODAL VALUES      WEIGHT IN KN    LENGTH IN METE
  -----------------------------------------------------------
      MODE   GENERALIZED WEIGHT  MODAL DISPLACEMENT
         1       2.0515518E+02     6.7031081E-02
         2       1.3223642E+02    -2.2654413E-14
         3       2.0496067E+02     2.2811639E-03
         4       1.5152478E+02    -3.2199584E-17
         5       1.3398013E+02     9.5189103E-17
         6       1.5153264E+02    -7.4449874E-16
         7       8.4912708E+01    -1.0519186E-07
         8       4.5425281E+01     1.5487729E-16
         9       4.5462584E+01    -1.0565649E-08
        10       4.1252788E+01     5.2183466E-17
        11       3.7432782E+01    -1.7319698E-08
        12       3.4897917E+01    -3.9078081E-16
        13       3.2973453E+01     4.4944533E-09
        14       3.3765129E+01    -1.9322218E-08
        15       3.2204276E+01    -2.7180429E-16
        16       2.8727553E+01     8.5462116E-09
        17       1.5476347E+02     7.5583518E-10
        18       4.2380370E+01     6.2594434E-08
        19       3.4111151E+01     6.9052098E-18
        20       3.5850904E+01     2.3302519E-08
        21       1.0241904E+02    -1.2243621E-09
        22       3.4164403E+01     7.8273208E-18
        23       3.4084516E+01    -1.7564736E-10
 FLOOR MODAL BASE ACTIONS
  FLOOR MODAL BASE ACTIONS  FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
   STORY NUMBER    3  HEIGHT       8.00
      1      0.303        0.00        0.27      440.40     3524.70
-4403.96        2.70
      2      0.222       -0.00        0.00        0.00        0.00
0.00        0.00
      3      0.114       -0.00       -0.14      -63.91     -512.08
639.12       -1.44
      4      0.099       -0.00        0.00       -0.00       -0.00
-0.00        0.00
      5      0.079       -0.00        0.00        0.00        0.00
0.00        0.00
      6      0.031        0.00        0.00        0.00       -0.00
0.00       -0.00
      7      0.015       -0.00       -0.14        0.00        0.20
-0.01       -1.45
      8      0.015        0.00       -0.00        0.00        0.00
0.00        0.00
      9      0.015       -0.00       -0.00        0.00        0.00
-0.00       -0.00
     10      0.014        0.00        0.00        0.00       -0.00
0.00        0.00
     11      0.014       -0.00       -0.01        0.00        0.04
```

```
-0.00       -0.06
     12      0.014        0.00        -0.00        0.00        0.00
0.00        0.00
     13      0.014       -0.00         0.00        0.00       -0.00
-0.00        0.00
     14      0.013        0.00        -0.01        0.00        0.04
-0.00       -0.06
     15      0.013        0.00         0.00        0.00       -0.00
0.00       -0.00
     16      0.012       -0.00        -0.00        0.00        0.00
-0.00       -0.01
     17      0.011        0.00        -0.00       -0.00        0.00
0.00       -0.00
     18      0.011        0.00         0.07        0.00       -0.59
-0.00        0.74
     19      0.011        0.00        -0.00       -0.00        0.00
0.00        0.00
     20      0.011       -0.00         0.01        0.00       -0.06
-0.00        0.08
     21      0.011        0.00         0.00       -0.00       -0.00
0.00        0.00
     22      0.011        0.00        -0.00       -0.00        0.00
0.00       -0.00
     23      0.010       -0.00        -0.00        0.00        0.00
-0.00       -0.00
   STORY NUMBER    2  HEIGHT       4.00
      1      0.303        0.00         0.21      267.22     1070.06
-2672.20        2.12
      2      0.222       -0.00         0.00        0.00        0.00
0.00        0.00
      3      0.114        0.00        -0.08      105.33      420.88
-1053.34       -0.83
      4      0.099       -0.00        -0.00        0.00        0.00
-0.00        0.00
      5      0.079        0.00        -0.00        0.00        0.00
-0.00       -0.00
      6      0.031       -0.00        -0.00        0.00        0.00
-0.00        0.00
      7      0.015        0.00        -0.09       -0.00        0.12
0.00       -0.89
      8      0.015       -0.00        -0.00        0.00        0.00
-0.00        0.00
      9      0.015        0.00        -0.00       -0.00        0.00
0.00       -0.00
     10      0.014       -0.00        -0.00        0.00        0.00
-0.00        0.00
      STAAD SPACE                                                -- PAGE NO.
21
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                              MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD         FX          FY          FZ          MX
MY          MZ
     11      0.014        0.00        -0.00       -0.00        0.03
0.00       -0.03
     12      0.014       -0.00        -0.00        0.00        0.00
-0.00        0.00
```

```
     13      0.014         0.00         0.00        -0.00        -0.00
0.00        0.00
     14      0.013        -0.00        -0.00        -0.00         0.03
0.00       -0.03
     15      0.013        -0.00        -0.00         0.00         0.00
-0.00       -0.00
     16      0.012         0.00        -0.00        -0.00         0.00
0.00       -0.00
     17      0.011        -0.00        -0.00         0.00         0.00
-0.00       -0.00
     18      0.011        -0.00         0.05        -0.00        -0.37
0.00        0.46
     19      0.011        -0.00        -0.00         0.00         0.00
-0.00        0.00
     20      0.011         0.00         0.00        -0.00        -0.04
0.00        0.05
     21      0.011        -0.00         0.00         0.00        -0.00
-0.00        0.00
     22      0.011        -0.00        -0.00         0.00         0.00
-0.00        0.00
     23      0.010         0.00        -0.00        -0.00         0.00
0.00       -0.00
   STORY NUMBER    1  HEIGHT       0.00
      1      0.303         0.00         0.00         0.00         0.00
0.00        0.00
      2      0.222         0.00         0.00         0.00         0.00
0.00        0.00
      3      0.114         0.00         0.00         0.00         0.00
0.00        0.00
      4      0.099         0.00         0.00         0.00         0.00
0.00        0.00
      5      0.079         0.00         0.00         0.00         0.00
0.00        0.00
      6      0.031         0.00         0.00         0.00         0.00
0.00        0.00
      7      0.015         0.00         0.00         0.00         0.00
0.00        0.00
      8      0.015         0.00         0.00         0.00         0.00
0.00        0.00
      9      0.015         0.00         0.00         0.00         0.00
0.00        0.00
     10      0.014         0.00         0.00         0.00         0.00
0.00        0.00
     11      0.014         0.00         0.00         0.00         0.00
0.00        0.00
     12      0.014         0.00         0.00         0.00         0.00
0.00        0.00
     13      0.014         0.00         0.00         0.00         0.00
0.00        0.00
     14      0.013         0.00         0.00         0.00         0.00
0.00        0.00
     15      0.013         0.00         0.00         0.00         0.00
0.00        0.00
     16      0.012         0.00         0.00         0.00         0.00
0.00        0.00
     17      0.011         0.00         0.00         0.00         0.00
0.00        0.00
     18      0.011         0.00         0.00         0.00         0.00
```

```
0.00        0.00
     19      0.011        0.00        0.00        0.00        0.00
0.00        0.00
     20      0.011        0.00        0.00        0.00        0.00
0.00        0.00
     21      0.011        0.00        0.00        0.00        0.00
0.00        0.00
     22      0.011        0.00        0.00        0.00        0.00
0.00        0.00
     23      0.010        0.00        0.00        0.00        0.00
0.00        0.00
      STAAD SPACE                                          -- PAGE NO.
22
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                        MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.303        0.00        0.48      707.62     4594.76
-7076.16        4.81
      2      0.222       -0.00        0.00        0.00        0.00
0.00        0.00
      3      0.114        0.00       -0.23       41.42      -91.21
-414.22       -2.27
      4      0.099       -0.00       -0.00        0.00       -0.00
-0.00        0.00
      5      0.079       -0.00       -0.00        0.00        0.00
-0.00        0.00
      6      0.031       -0.00       -0.00        0.00        0.00
-0.00       -0.00
      7      0.015        0.00       -0.23        0.00        0.32
-0.00       -2.34
      8      0.015        0.00       -0.00        0.00        0.00
0.00        0.00
      9      0.015        0.00       -0.00        0.00        0.00
-0.00       -0.00
     10      0.014        0.00       -0.00        0.00        0.00
-0.00        0.00
     11      0.014        0.00       -0.01        0.00        0.07
-0.00       -0.09
     12      0.014        0.00       -0.00        0.00        0.00
0.00        0.00
     13      0.014        0.00        0.00        0.00       -0.00
-0.00        0.00
     14      0.013       -0.00       -0.01        0.00        0.07
-0.00       -0.09
     15      0.013        0.00        0.00        0.00       -0.00
0.00       -0.00
     16      0.012        0.00       -0.00        0.00        0.01
-0.00       -0.01
     17      0.011       -0.00       -0.00        0.00        0.00
-0.00       -0.00
     18      0.011       -0.00        0.12        0.00       -0.96
-0.00        1.20
     19      0.011        0.00       -0.00        0.00        0.00
0.00        0.00
```

```
     20      0.011        0.00        0.01        0.00       -0.10
-0.00        0.13
     21      0.011       -0.00        0.00        0.00       -0.00
-0.00        0.00
     22      0.011        0.00       -0.00        0.00        0.00
0.00       -0.00
     23      0.010        0.00       -0.00        0.00        0.00
-0.00       -0.00
      STAAD SPACE                                                -- PAGE NO.
23
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KN
           --------------------------------------         ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z        X        Y        Z
   1    0.00  0.00 94.35    0.000    0.000   94.349      0.00     0.00
707.62
   2    8.91  0.00  0.00    8.914    0.000   94.349      0.00     0.00
0.00
   3    0.00  0.00  5.65    8.914    0.000  100.000      0.00     0.00
41.42
   4   81.69  0.00  0.00   90.600    0.000  100.000      0.00     0.00
0.00
   5    0.19  0.00  0.00   90.789    0.000  100.000      0.00     0.00
0.00
   6    9.21  0.00  0.00   99.996    0.000  100.000      0.00     0.00
0.00
   7    0.00 56.99  0.00   99.996   56.992  100.000      0.00     0.00
0.00
   8    0.00  0.00  0.00   99.996   56.992  100.000      0.00     0.00
0.00
   9    0.00  0.00  0.00   99.996   56.993  100.000      0.00     0.00
0.00
  10    0.00  0.00  0.00   99.996   56.993  100.000      0.00     0.00
0.00
  11    0.00  5.81  0.00   99.996   62.803  100.000      0.00     0.00
0.00
  12    0.00  0.00  0.00   99.996   62.803  100.000      0.00     0.00
0.00
  13    0.00  0.17  0.00   99.996   62.973  100.000      0.00     0.00
0.00
  14    0.00  4.08  0.00   99.996   67.058  100.000      0.00     0.00
0.00
  15    0.00  0.00  0.00   99.996   67.058  100.000      0.00     0.00
0.00
  16    0.00  0.18  0.00   99.996   67.236  100.000      0.00     0.00
0.00
  17    0.00  0.13  0.00   99.996   67.365  100.000      0.00     0.00
0.00
  18    0.00 24.91  0.00   99.996   92.277  100.000      0.00     0.00
0.00
  19    0.00  0.00  0.00   99.997   92.277  100.000      0.00     0.00
0.00
  20    0.00  2.39  0.00   99.997   94.664  100.000      0.00     0.00
0.00
  21    0.00  0.06  0.00   99.997   94.720  100.000      0.00     0.00
0.00
  22    0.00  0.00  0.00   99.997   94.720  100.000      0.00     0.00
0.00
```

```
  23    0.00  0.00  0.00   99.997   94.722  100.000      0.00     0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR      0.00     0.00
708.83
                                  TOTAL 10PCT SHEAR      0.00     0.00
708.83
                                  TOTAL ABS   SHEAR      0.00     0.00
749.04
  RESPONSE SPECTRUM LOAD     3
     RESPONSE LOAD CASE      3
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN          GENERALIZED
      MODE           X             Y             Z             WEIGHT
         1       5.984143E-24  1.308434E-04  2.830457E+02    2.051552E+02
         2       2.674250E+01  1.755244E-29  7.301247E-23    1.322364E+02
         3       5.065707E-24  5.100442E-04  1.695390E+01    2.049607E+02
         4       2.450581E+02  5.450377E-28  4.994469E-27    1.515248E+02
         5       5.653800E-01  4.194184E-27  1.131665E-25    1.339801E+02
      STAAD SPACE                                              -- PAGE NO.
24
         6       2.762138E+01  1.415896E-21  6.831347E-22    1.515326E+02
         7       6.469897E-19  1.709744E+02  2.191617E-04    8.491271E+01
         8       5.191913E-04  7.860724E-19  2.581121E-22    4.542528E+01
         9       2.700640E-19  3.956472E-03  1.269517E-06    4.546258E+01
        10       2.182148E-04  1.841933E-22  3.151962E-23    4.125279E+01
        11       2.053212E-18  1.742969E+01  3.337675E-06    3.743278E+01
        12       1.262537E-03  1.280337E-17  1.592862E-21    3.489792E+01
        13       6.567201E-19  5.102359E-01  2.157681E-07    3.297345E+01
        14       1.920060E-17  1.225476E+01  4.658799E-06    3.376513E+01
        15       1.813976E-04  1.101497E-20  1.075361E-21    3.220428E+01
        16       6.906754E-17  5.351847E-01  1.310061E-06    2.872755E+01
        17       5.837981E-15  3.860367E-01  7.409204E-08    1.547635E+02
        18       1.179917E-16  7.473509E+01  1.455537E-04    4.238037E+01
        19       3.492625E-04  5.984215E-21  1.433175E-24    3.411115E+01
        20       4.208206E-16  7.160740E+00  1.785032E-05    3.585090E+01
        21       5.498390E-16  1.703037E-01  1.462900E-07    1.024190E+02
        22       7.887138E-05  9.206345E-21  2.039361E-24    3.416440E+01
        23       1.080083E-18  6.280113E-03  1.077932E-09    3.408452E+01
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   3.000000E+02  3.000000E+02  3.000000E+02 KN
     MISSING WEIGHT X Y Z  -1.010015E-02 -1.583268E+01 -2.927266E-05 KN
       MODAL WEIGHT X Y Z   2.999899E+02  2.841673E+02  3.000000E+02 KN
           MODE                   ACCELERATION-G     DAMPING
           ----                   --------------     -------
              1                        2.50001       0.05000
              2                        2.50001       0.05000
              3                        2.44320       0.05000
              4                        2.29243       0.05000
              5                        2.09360       0.05000
              6                        1.45919       0.05000
              7                        1.20990       0.05000
              8                        1.20911       0.05000
              9                        1.20633       0.05000
             10                        1.20057       0.05000
             11                        1.19774       0.05000
             12                        1.19747       0.05000
             13                        1.19357       0.05000
```

```
             14                       1.18733       0.05000
             15                       1.17814       0.05000
             16                       1.16420       0.05000
             17                       1.15232       0.05000
             18                       1.15058       0.05000
             19                       1.15037       0.05000
             20                       1.14884       0.05000
             21                       1.14738       0.05000
             22                       1.14654       0.05000
             23                       1.14463       0.05000
      STAAD SPACE                                                -- PAGE NO.
25
 GENERALIZED WEIGHT & MODAL DISPLACEMENT
  ADDITIONAL MODAL VALUES      WEIGHT IN KN    LENGTH IN METE
  ------------------------------------------------------------
      MODE   GENERALIZED WEIGHT  MODAL DISPLACEMENT
         1       2.0515518E+02     4.5574707E-05
         2       1.3223642E+02    -1.1107674E-17
         3       2.0496067E+02    -1.2511969E-05
         4       1.5152478E+02     1.0636993E-17
         5       1.3398013E+02    -1.8325336E-17
         6       1.5153264E+02     1.0718312E-15
         7       8.4912708E+01     9.2910634E-05
         8       4.5425281E+01    -8.5470223E-15
         9       4.5462584E+01     5.8983549E-07
        10       4.1252788E+01    -1.2614770E-16
        11       3.7432782E+01     3.9578840E-05
        12       3.4897917E+01     3.5035334E-14
        13       3.2973453E+01     6.9114431E-06
        14       3.3765129E+01     3.1338075E-05
        15       3.2204276E+01    -8.6990314E-16
        16       2.8727553E+01    -5.4623531E-06
        17       1.5476347E+02    -1.7252644E-06
        18       4.2380370E+01     4.4852468E-05
        19       3.4111151E+01    -4.4620138E-16
        20       3.5850904E+01     1.4759063E-05
        21       1.0241904E+02    -1.3210358E-06
        22       3.4164403E+01    -5.2590798E-16
        23       3.4084516E+01     4.2396474E-07
 FLOOR MODAL BASE ACTIONS
  FLOOR MODAL BASE ACTIONS  FORCES IN KN   LENGTH IN METE
  ------------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
   STORY NUMBER    3  HEIGHT       8.00
      1      0.303        0.00        0.00        0.30        2.40
-2.99        0.00
      2      0.222       -0.00        0.00        0.00        0.00
0.00        0.00
      3      0.114        0.00        0.00        0.35        2.81
-3.51        0.01
      4      0.099        0.00       -0.00        0.00        0.00
0.00       -0.00
      5      0.079        0.00       -0.00       -0.00       -0.00
-0.00       -0.00
      6      0.031       -0.00       -0.00       -0.00        0.00
```

```
-0.00        0.00
      7      0.015        0.00       127.86        -0.64     -176.80
6.41     1278.56
      8      0.015       -0.00         0.00        -0.00       -0.00
-0.00       -0.00
      9      0.015        0.00         0.00        -0.00       -0.00
0.00        0.03
     10      0.014       -0.00        -0.00        -0.00        0.00
-0.00       -0.00
     11      0.014        0.00        12.91        -0.03     -102.73
0.30      129.09
     12      0.014       -0.00         0.00        -0.00       -0.00
-0.00       -0.00
     13      0.014       -0.00         0.38         0.00       -3.03
-0.02        3.77
     14      0.013       -0.00         9.00        -0.03      -71.80
0.25       89.97
     15      0.013        0.00         0.00         0.00       -0.00
0.00       -0.00
     16      0.012        0.00         0.39        -0.00       -3.08
0.03        3.85
     17      0.011       -0.00         0.27         0.00       -2.20
-0.00        2.75
     18      0.011        0.00        53.12         0.33     -424.47
-3.25      531.23
     19      0.011       -0.00         0.00         0.00       -0.00
-0.00       -0.00
     20      0.011       -0.00         5.08         0.04      -40.59
-0.38       50.83
     21      0.011        0.00         0.12        -0.00       -0.97
0.00        1.21
     22      0.011       -0.00         0.00         0.00       -0.00
-0.00        0.00
     23      0.010        0.00         0.00        -0.00       -0.04
0.00        0.04
   STORY NUMBER    2  HEIGHT       4.00
      1      0.303        0.00         0.00         0.18        0.73
-1.82        0.00
      2      0.222       -0.00         0.00         0.00        0.00
0.00        0.00
      3      0.114       -0.00         0.00        -0.58       -2.31
5.78        0.00
      4      0.099        0.00         0.00        -0.00       -0.00
0.00       -0.00
      5      0.079       -0.00         0.00        -0.00       -0.00
0.00        0.00
      6      0.031        0.00         0.00        -0.00       -0.00
0.00       -0.00
      7      0.015       -0.00        79.01         0.41     -104.53
-4.06      790.06
      8      0.015        0.00         0.00        -0.00       -0.00
0.00       -0.00
      9      0.015       -0.00         0.00         0.00       -0.00
-0.00        0.02
     10      0.014        0.00         0.00        -0.00       -0.00
0.00       -0.00
      STAAD SPACE                                              -- PAGE NO.
26
```

```
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                               MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD         FX           FY           FZ           MX
MY          MZ
     11      0.014       -0.00         7.97         0.02       -63.19
-0.20       79.67
     12      0.014        0.00         0.00        -0.00        -0.00
0.00       -0.00
     13      0.014        0.00         0.23        -0.00        -1.88
0.01        2.32
     14      0.013        0.00         5.55         0.02       -44.14
-0.17       55.54
     15      0.013       -0.00        -0.00         0.00         0.00
-0.00       -0.00
     16      0.012       -0.00         0.24         0.00        -1.88
-0.02        2.38
     17      0.011        0.00         0.17        -0.00        -1.36
0.00        1.70
     18      0.011       -0.00        32.87        -0.21      -264.86
2.05      328.65
     19      0.011        0.00         0.00        -0.00        -0.00
0.00       -0.00
     20      0.011        0.00         3.14        -0.03       -25.38
0.25       31.44
     21      0.011       -0.00         0.07         0.00        -0.60
-0.00        0.75
     22      0.011        0.00         0.00        -0.00        -0.00
0.00       -0.00
     23      0.010       -0.00         0.00         0.00        -0.02
-0.00        0.03
   STORY NUMBER    1  HEIGHT       0.00
      1      0.303        0.00         0.00         0.00         0.00
0.00        0.00
      2      0.222        0.00         0.00         0.00         0.00
0.00        0.00
      3      0.114        0.00         0.00         0.00         0.00
0.00        0.00
      4      0.099        0.00         0.00         0.00         0.00
0.00        0.00
      5      0.079        0.00         0.00         0.00         0.00
0.00        0.00
      6      0.031        0.00         0.00         0.00         0.00
0.00        0.00
      7      0.015        0.00         0.00         0.00         0.00
0.00        0.00
      8      0.015        0.00         0.00         0.00         0.00
0.00        0.00
      9      0.015        0.00         0.00         0.00         0.00
0.00        0.00
     10      0.014        0.00         0.00         0.00         0.00
0.00        0.00
     11      0.014        0.00         0.00         0.00         0.00
0.00        0.00
     12      0.014        0.00         0.00         0.00         0.00
0.00        0.00
     13      0.014        0.00         0.00         0.00         0.00
```

```
0.00         0.00
     14      0.013         0.00         0.00         0.00         0.00
0.00         0.00
     15      0.013         0.00         0.00         0.00         0.00
0.00         0.00
     16      0.012         0.00         0.00         0.00         0.00
0.00         0.00
     17      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     18      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     19      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     20      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     21      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     22      0.011         0.00         0.00         0.00         0.00
0.00         0.00
     23      0.010         0.00         0.00         0.00         0.00
0.00         0.00
      STAAD SPACE                                              -- PAGE NO.
27
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD         FX           FY           FZ           MX
MY           MZ
      1      0.303         0.00         0.00         0.48         3.12
-4.81         0.00
      2      0.222        -0.00         0.00         0.00         0.00
0.00         0.00
      3      0.114        -0.00         0.00        -0.23         0.50
2.27         0.01
      4      0.099         0.00         0.00        -0.00         0.00
0.00        -0.00
      5      0.079         0.00         0.00        -0.00        -0.00
0.00        -0.00
      6      0.031         0.00         0.00        -0.00        -0.00
0.00         0.00
      7      0.015        -0.00       206.86        -0.23      -281.33
2.34      2068.62
      8      0.015        -0.00         0.00        -0.00        -0.00
-0.00         -0.00
      9      0.015        -0.00         0.00        -0.00        -0.01
0.00         0.05
     10      0.014        -0.00         0.00        -0.00        -0.00
0.00        -0.00
     11      0.014        -0.00        20.88        -0.01      -165.92
0.09       208.76
     12      0.014        -0.00         0.00        -0.00        -0.00
-0.00         -0.00
     13      0.014         0.00         0.61         0.00        -4.91
-0.00         6.09
     14      0.013         0.00        14.55        -0.01      -115.94
0.09       145.50
```

```
     15      0.013        0.00        0.00        0.00       -0.00
0.00       -0.00
     16      0.012       -0.00        0.62       -0.00       -4.96
0.01        6.23
     17      0.011        0.00        0.44       -0.00       -3.56
0.00        4.45
     18      0.011       -0.00       85.99        0.12     -689.33
-1.20      859.88
     19      0.011       -0.00        0.00       -0.00       -0.00
-0.00       -0.00
     20      0.011        0.00        8.23        0.01      -65.97
-0.13       82.27
     21      0.011       -0.00        0.20        0.00       -1.56
-0.00        1.95
     22      0.011       -0.00        0.00       -0.00       -0.00
-0.00        0.00
     23      0.010       -0.00        0.01       -0.00       -0.06
0.00        0.07
      STAAD SPACE                                             -- PAGE NO.
28
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1    0.00  0.00 94.35    0.000    0.000   94.349      0.00     0.00
0.00
   2    8.91  0.00  0.00    8.914    0.000   94.349      0.00     0.00
0.00
   3    0.00  0.00  5.65    8.914    0.000  100.000      0.00     0.00
0.00
   4   81.69  0.00  0.00   90.600    0.000  100.000      0.00     0.00
0.00
   5    0.19  0.00  0.00   90.789    0.000  100.000      0.00     0.00
0.00
   6    9.21  0.00  0.00   99.996    0.000  100.000      0.00     0.00
0.00
   7    0.00 56.99  0.00   99.996   56.992  100.000      0.00   206.86
0.00
   8    0.00  0.00  0.00   99.996   56.992  100.000      0.00     0.00
0.00
   9    0.00  0.00  0.00   99.996   56.993  100.000      0.00     0.00
0.00
  10    0.00  0.00  0.00   99.996   56.993  100.000      0.00     0.00
0.00
  11    0.00  5.81  0.00   99.996   62.803  100.000      0.00    20.88
0.00
  12    0.00  0.00  0.00   99.996   62.803  100.000      0.00     0.00
0.00
  13    0.00  0.17  0.00   99.996   62.973  100.000      0.00     0.61
0.00
  14    0.00  4.08  0.00   99.996   67.058  100.000      0.00    14.55
0.00
  15    0.00  0.00  0.00   99.996   67.058  100.000      0.00     0.00
0.00
  16    0.00  0.18  0.00   99.996   67.236  100.000      0.00     0.62
0.00
  17    0.00  0.13  0.00   99.996   67.365  100.000      0.00     0.44
0.00
```

```
  18    0.00 24.91  0.00   99.996   92.277  100.000      0.00    85.99
0.00
  19    0.00  0.00  0.00   99.997   92.277  100.000      0.00     0.00
0.00
  20    0.00  2.39  0.00   99.997   94.664  100.000      0.00     8.23
0.00
  21    0.00  0.06  0.00   99.997   94.720  100.000      0.00     0.20
0.00
  22    0.00  0.00  0.00   99.997   94.720  100.000      0.00     0.00
0.00
  23    0.00  0.00  0.00   99.997   94.722  100.000      0.00     0.01
0.00

---------------------------
                                  TOTAL SRSS  SHEAR      0.00   225.62
0.00
                                  TOTAL 10PCT SHEAR      0.00   249.20
0.00
                                  TOTAL ABS   SHEAR      0.00   338.39
0.00
   ************ END OF DATA FROM INTERNAL STORAGE ************
   157. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                                -- PAGE NO.
29
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1    0.6109    0.0039    0.7722    0.0004    0.0008    0.0007
           2    0.0002    0.0102    4.0730    0.0009    0.0000    0.0003
           3    0.0005    0.0059    0.0031    0.0000    0.0000    0.0000
      3    1    0.6110    0.0027    0.3864    0.0000    0.0008    0.0002
           2    0.0001    0.0123    4.0731    0.0003    0.0000    0.0001
           3    0.0003    0.0057    0.0030    0.0000    0.0000    0.0000
      4    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1    0.6111    0.0000    0.0000    0.0000    0.0008    0.0003
           2    0.0000    0.0106    4.0732    0.0004    0.0000    0.0000
           3    0.0000    0.0058    0.0031    0.0000    0.0000    0.0000
      6    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      7    1    0.6110    0.0027    0.3864    0.0000    0.0008    0.0002
           2    0.0001    0.0123    4.0731    0.0003    0.0000    0.0001
           3    0.0003    0.0057    0.0030    0.0000    0.0000    0.0000
      8    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      9    1    0.6109    0.0039    0.7722    0.0004    0.0008    0.0007
           2    0.0002    0.0102    4.0730    0.0009    0.0000    0.0003
           3    0.0005    0.0059    0.0031    0.0000    0.0000    0.0000
     10    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
```

```
     11    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     12    1      0.3894    0.0117    0.7722    0.0002    0.0008    0.0001
           2      0.0000    0.0079    4.0728    0.0001    0.0000    0.0001
           3      0.0000    0.0038    0.0031    0.0000    0.0000    0.0000
     13    1      0.3893    0.0032    0.3864    0.0000    0.0008    0.0001
           2      0.0000    0.0136    4.0731    0.0002    0.0000    0.0000
           3      0.0000    0.0032    0.0030    0.0000    0.0000    0.0000
     14    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     15    1      0.3894    0.0000    0.0000    0.0000    0.0008    0.0001
           2      0.0000    0.0098    4.0732    0.0001    0.0000    0.0000
           3      0.0000    0.0034    0.0031    0.0000    0.0000    0.0000
     16    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      STAAD SPACE                                                -- PAGE NO.
30
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     17    1      0.3893    0.0032    0.3864    0.0000    0.0008    0.0001
           2      0.0000    0.0136    4.0731    0.0002    0.0000    0.0000
           3      0.0000    0.0032    0.0030    0.0000    0.0000    0.0000
     18    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     19    1      0.3894    0.0117    0.7722    0.0002    0.0008    0.0001
           2      0.0000    0.0079    4.0728    0.0001    0.0000    0.0001
           3      0.0000    0.0038    0.0031    0.0000    0.0000    0.0000
     20    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     21    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     22    1      0.3454    0.0063    0.7719    0.0006    0.0008    0.0009
           2      0.0008    0.0119    4.0722    0.0015    0.0000    0.0001
           3      0.0005    0.0028    0.0031    0.0000    0.0000    0.0000
     23    1      0.3466    0.0014    0.3864    0.0001    0.0008    0.0007
           2      0.0002    0.0120    4.0729    0.0007    0.0000    0.0000
           3      0.0003    0.0018    0.0030    0.0000    0.0000    0.0000
     24    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     25    1      0.3468    0.0000    0.0000    0.0000    0.0008    0.0007
           2      0.0000    0.0124    4.0731    0.0008    0.0000    0.0000
           3      0.0000    0.0016    0.0031    0.0000    0.0000    0.0000
     26    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     27    1      0.3466    0.0014    0.3864    0.0001    0.0008    0.0007
           2      0.0002    0.0120    4.0729    0.0007    0.0000    0.0000
           3      0.0003    0.0018    0.0030    0.0000    0.0000    0.0000
     28    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
```

```
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     29   1     0.3454    0.0063    0.7719    0.0006    0.0008    0.0009
          2     0.0008    0.0119    4.0722    0.0015    0.0000    0.0001
          3     0.0005    0.0028    0.0031    0.0000    0.0000    0.0000
     30   1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     31   1     0.9975    0.0050    1.4072    0.0002    0.0014    0.0003
          2     0.0012    0.0133    6.7045    0.0003    0.0000    0.0001
      STAAD SPACE                                              -- PAGE NO.
31
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
          3     0.0002    0.0095    0.0046    0.0000    0.0000    0.0000
     32   1     0.9972    0.0035    0.7038    0.0000    0.0014    0.0001
          2     0.0005    0.0160    6.7031    0.0001    0.0000    0.0000
          3     0.0001    0.0093    0.0046    0.0000    0.0000    0.0000
     33   1     0.9972    0.0000    0.0000    0.0000    0.0014    0.0001
          2     0.0000    0.0139    6.7027    0.0002    0.0000    0.0000
          3     0.0000    0.0094    0.0046    0.0000    0.0000    0.0000
     34   1     0.9972    0.0035    0.7038    0.0000    0.0014    0.0001
          2     0.0005    0.0160    6.7031    0.0001    0.0000    0.0000
          3     0.0001    0.0093    0.0046    0.0000    0.0000    0.0000
     35   1     0.9975    0.0050    1.4072    0.0002    0.0014    0.0003
          2     0.0012    0.0133    6.7045    0.0003    0.0000    0.0001
          3     0.0002    0.0095    0.0046    0.0000    0.0000    0.0000
     36   1     0.6767    0.0157    1.4070    0.0001    0.0014    0.0001
          2     0.0000    0.0101    6.7041    0.0000    0.0000    0.0000
          3     0.0000    0.0061    0.0046    0.0000    0.0000    0.0000
     37   1     0.6766    0.0039    0.7037    0.0000    0.0014    0.0000
          2     0.0001    0.0167    6.7029    0.0000    0.0000    0.0000
          3     0.0000    0.0052    0.0046    0.0000    0.0000    0.0000
     38   1     0.6766    0.0000    0.0000    0.0000    0.0014    0.0001
          2     0.0000    0.0124    6.7026    0.0000    0.0000    0.0000
          3     0.0000    0.0056    0.0046    0.0000    0.0000    0.0000
     39   1     0.6766    0.0039    0.7037    0.0000    0.0014    0.0000
          2     0.0001    0.0167    6.7029    0.0000    0.0000    0.0000
          3     0.0000    0.0052    0.0046    0.0000    0.0000    0.0000
     40   1     0.6767    0.0157    1.4070    0.0001    0.0014    0.0001
          2     0.0000    0.0101    6.7041    0.0000    0.0000    0.0000
          3     0.0000    0.0061    0.0046    0.0000    0.0000    0.0000
     41   1     0.7441    0.0088    1.4070    0.0005    0.0014    0.0007
          2     0.0014    0.0152    6.7039    0.0007    0.0000    0.0001
          3     0.0002    0.0045    0.0046    0.0000    0.0000    0.0000
     42   1     0.7434    0.0020    0.7037    0.0000    0.0014    0.0005
          2     0.0006    0.0153    6.7028    0.0003    0.0000    0.0000
          3     0.0001    0.0029    0.0046    0.0000    0.0000    0.0000
     43   1     0.7434    0.0000    0.0000    0.0000    0.0014    0.0005
          2     0.0000    0.0158    6.7024    0.0004    0.0000    0.0000
          3     0.0000    0.0025    0.0046    0.0000    0.0000    0.0000
     44   1     0.7434    0.0020    0.7037    0.0000    0.0014    0.0005
          2     0.0006    0.0153    6.7028    0.0003    0.0000    0.0000
          3     0.0001    0.0029    0.0046    0.0000    0.0000    0.0000
     45   1     0.7441    0.0088    1.4070    0.0005    0.0014    0.0007
          2     0.0014    0.0152    6.7039    0.0007    0.0000    0.0001
          3     0.0002    0.0045    0.0046    0.0000    0.0000    0.0000
      STAAD SPACE                                              -- PAGE NO.
```

```
32
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     27.02     18.77      6.09     12.71      0.01     59.53
           2      3.28     49.47     34.78     70.60      0.00      4.14
           3      0.03     28.33      0.03      0.06      0.00      0.06
      4    1     32.14     13.14      3.47      6.92      0.01     66.00
           2      0.60     59.33     35.75     71.88      0.00      0.75
           3      0.03     27.60      0.03      0.06      0.00      0.05
      6    1     31.17      0.00      0.00      0.00      0.01     64.78
           2      0.00     51.34     35.55     71.62      0.00      0.00
           3      0.00     27.96      0.04      0.06      0.00      0.00
      8    1     32.14     13.14      3.47      6.92      0.01     66.00
           2      0.60     59.33     35.75     71.88      0.00      0.75
           3      0.03     27.60      0.03      0.06      0.00      0.05
     10    1     27.02     18.77      6.09     12.71      0.01     59.53
           2      3.28     49.47     34.78     70.60      0.00      4.14
           3      0.03     28.33      0.03      0.06      0.00      0.06
     11    1     20.77     56.69      7.11     14.07      0.01     42.41
           2      1.16     38.37     36.55     72.94      0.00      1.46
           3      0.06     18.25      0.03      0.06      0.00      0.08
     14    1     22.40     15.56      3.43      6.87      0.01     44.48
           2      0.57     65.69     36.58     72.99      0.00      0.71
           3      0.02     15.37      0.03      0.06      0.00      0.03
     16    1     22.36      0.00      0.00      0.00      0.01     44.42
           2      0.00     47.19     36.52     72.91      0.00      0.00
           3      0.00     16.62      0.03      0.06      0.00      0.00
     18    1     22.40     15.56      3.43      6.87      0.01     44.48
           2      0.57     65.69     36.58     72.99      0.00      0.71
           3      0.02     15.37      0.03      0.06      0.00      0.03
     20    1     20.77     56.69      7.11     14.07      0.01     42.41
           2      1.16     38.37     36.55     72.94      0.00      1.46
           3      0.06     18.25      0.03      0.06      0.00      0.08
     21    1     69.83     54.99     12.43     25.85      0.02    210.76
           2     12.47    103.61     68.22    140.13      0.00     14.20
           3      0.24     24.40      0.07      0.12      0.00      0.46
     24    1     88.23     12.51      6.79     13.71      0.02    231.78
           2      0.56    104.37     71.09    143.94      0.00      0.56
           3      0.17     15.56      0.07      0.12      0.00      0.31
     26    1     87.68      0.00      0.00      0.00      0.02    231.27
           2      0.00    107.80     70.79    143.54      0.00      0.00
           3      0.00     13.59      0.07      0.12      0.00      0.00
     28    1     88.23     12.51      6.79     13.71      0.02    231.78
           2      0.56    104.37     71.09    143.94      0.00      0.56
           3      0.17     15.56      0.07      0.12      0.00      0.31
     30    1     69.83     54.99     12.43     25.85      0.02    210.76
           2     12.47    103.61     68.22    140.13      0.00     14.20
           3      0.24     24.40      0.07      0.12      0.00      0.46
      STAAD SPACE                                                -- PAGE NO.
33
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     18.77     27.02     6.09      0.01     12.71      59.53
                 2     18.77     27.02     6.09      0.01     11.64      48.57
           2     1     49.47      3.28    34.78      0.00     70.60       4.14
```

```
                 2     49.47       3.28    34.78       0.00      68.53         8.97
           3     1     28.33       0.03     0.03       0.00       0.06         0.06
                 2     28.33       0.03     0.03       0.00       0.08         0.07
      2    1     2      1.05       6.34     0.06       0.01       0.16        18.92
                 3      1.05       6.34     0.06       0.01       0.16        12.77
           2     2      0.27       1.71     0.00       0.01       0.01         6.56
                 3      0.27       1.71     0.00       0.01       0.00         1.97
           3     2      0.78       0.04     0.00       0.00       0.00         0.10
                 3      0.78       0.04     0.00       0.00       0.00         0.12
      3    1     3     13.14      32.14     3.47       0.01       6.96        62.56
                 4     13.14      32.14     3.47       0.01       6.92        66.00
           2     3     59.33       0.60    35.75       0.00      71.12         1.65
                 4     59.33       0.60    35.75       0.00      71.88         0.75
           3     3     27.60       0.03     0.03       0.00       0.07         0.06
                 4     27.60       0.03     0.03       0.00       0.06         0.05
      4    1     3      0.92       3.82     0.07       0.00       0.17         8.96
                 5      0.92       3.82     0.07       0.00       0.16        10.11
           2     3      0.41       0.35     0.00       0.00       0.00         1.24
                 5      0.41       0.35     0.00       0.00       0.00         0.52
           3     3      1.23       0.03     0.00       0.00       0.00         0.08
                 5      1.23       0.03     0.00       0.00       0.00         0.07
      5    1     5      0.00      31.17     0.00       0.01       0.00        59.90
                 6      0.00      31.17     0.00       0.01       0.00        64.78
           2     5     51.34       0.00    35.55       0.00      70.59         0.00
                 6     51.34       0.00    35.55       0.00      71.62         0.00
           3     5     27.96       0.00     0.04       0.00       0.08         0.00
                 6     27.96       0.00     0.04       0.00       0.06         0.00
      6    1     5      0.92       3.82     0.07       0.00       0.16        10.11
                 7      0.92       3.82     0.07       0.00       0.17         8.96
           2     5      0.41       0.35     0.00       0.00       0.00         0.52
                 7      0.41       0.35     0.00       0.00       0.00         1.24
           3     5      1.23       0.03     0.00       0.00       0.00         0.07
                 7      1.23       0.03     0.00       0.00       0.00         0.08
      9    1     7     13.14      32.14     3.47       0.01       6.96        62.56
                 8     13.14      32.14     3.47       0.01       6.92        66.00
      STAAD SPACE                                                -- PAGE NO.
34
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y       MOM-Z
           2     7     59.33       0.60    35.75       0.00      71.12         1.65
                 8     59.33       0.60    35.75       0.00      71.88         0.75
           3     7     27.60       0.03     0.03       0.00       0.07         0.06
                 8     27.60       0.03     0.03       0.00       0.06         0.05
     10    1     7      1.05       6.34     0.06       0.01       0.16        12.77
                 9      1.05       6.34     0.06       0.01       0.16        18.92
           2     7      0.27       1.71     0.00       0.01       0.00         1.97
                 9      0.27       1.71     0.00       0.01       0.01         6.56
           3     7      0.78       0.04     0.00       0.00       0.00         0.12
                 9      0.78       0.04     0.00       0.00       0.00         0.10
     13    1     9     18.77      27.02     6.09       0.01      11.64        48.57
                10     18.77      27.02     6.09       0.01      12.71        59.53
           2     9     49.47       3.28    34.78       0.00      68.53         8.97
                10     49.47       3.28    34.78       0.00      70.60         4.14
           3     9     28.33       0.03     0.03       0.00       0.08         0.07
                10     28.33       0.03     0.03       0.00       0.06         0.06
     14    1     2      0.63       3.95     0.10       0.01       0.19        12.36
```

```
               12       0.63       3.95      0.10       0.01       0.19        4.30
          2     2       0.90       8.07      0.00       0.00       0.01       24.45
               12       0.90       8.07      0.00       0.00       0.01        7.84
          3     2       0.46       0.09      0.00       0.00       0.00        0.09
               12       0.46       0.09      0.00       0.00       0.00        0.26
    15    1     3       0.56       0.45      0.10       0.00       0.20        1.25
               13       0.56       0.45      0.10       0.00       0.20        0.59
          2     3       0.22       1.95      0.00       0.00       0.00        7.81
               13       0.22       1.95      0.00       0.00       0.00        0.03
          3     3       0.14       0.04      0.00       0.00       0.00        0.02
               13       0.14       0.04      0.00       0.00       0.00        0.14
    16    1     5       0.00       0.00      0.10       0.00       0.19        0.00
               15       0.00       0.00      0.10       0.00       0.20        0.00
          2     5       0.33       3.49      0.00       0.00       0.00       11.55
               15       0.33       3.49      0.00       0.00       0.00        2.41
          3     5       0.11       0.04      0.00       0.00       0.00        0.04
               15       0.11       0.04      0.00       0.00       0.00        0.16
    17    1     7       0.56       0.45      0.10       0.00       0.20        1.25
               17       0.56       0.45      0.10       0.00       0.20        0.59
          2     7       0.22       1.95      0.00       0.00       0.00        7.81
               17       0.22       1.95      0.00       0.00       0.00        0.03
          3     7       0.14       0.04      0.00       0.00       0.00        0.02
               17       0.14       0.04      0.00       0.00       0.00        0.14
      STAAD SPACE                                                  -- PAGE NO.
35
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y       MOM-Z
    18    1     9       0.63       3.95      0.10       0.01       0.19       12.36
               19       0.63       3.95      0.10       0.01       0.19        4.30
          2     9       0.90       8.07      0.00       0.00       0.01       24.45
               19       0.90       8.07      0.00       0.00       0.01        7.84
          3     9       0.46       0.09      0.00       0.00       0.00        0.09
               19       0.46       0.09      0.00       0.00       0.00        0.26
    19    1    11      56.69      20.77      7.11       0.01      14.07       42.41
               12      56.69      20.77      7.11       0.01      14.38       40.67
          2    11      38.37       1.16     36.55       0.00      72.94        1.46
               12      38.37       1.16     36.55       0.00      73.26        3.19
          3    11      18.25       0.06      0.03       0.00       0.06        0.08
               12      18.25       0.06      0.03       0.00       0.07        0.17
    20    1    12       0.42       1.02      0.06       0.00       0.15        3.05
               13       0.42       1.02      0.06       0.00       0.16        2.42
          2    12       0.09       0.23      0.00       0.00       0.00        1.60
               13       0.09       0.23      0.00       0.00       0.00        0.45
          3    12       0.01       0.02      0.00       0.00       0.00        0.05
               13       0.01       0.02      0.00       0.00       0.00        0.08
    21    1    13      15.56      22.40      3.43       0.01       6.84       45.14
               14      15.56      22.40      3.43       0.01       6.87       44.48
          2    13      65.69       0.57     36.58       0.00      73.35        1.55
               14      65.69       0.57     36.58       0.00      72.99        0.71
          3    13      15.37       0.02      0.03       0.00       0.07        0.06
               14      15.37       0.02      0.03       0.00       0.06        0.03
    22    1    13       0.10       1.17      0.07       0.00       0.17        2.71
               15       0.10       1.17      0.07       0.00       0.17        3.14
          2    13       0.11       0.26      0.00       0.00       0.00        0.98
               15       0.11       0.26      0.00       0.00       0.00        0.32
          3    13       0.02       0.02      0.00       0.00       0.00        0.05
```

```
                15      0.02      0.02      0.00      0.00      0.00      0.06
    23     1    15      0.00     22.36      0.00      0.01      0.00     45.04
                16      0.00     22.36      0.00      0.01      0.00     44.42
           2    15     47.19      0.00     36.52      0.00     73.19      0.00
                16     47.19      0.00     36.52      0.00     72.91      0.00
           3    15     16.62      0.00      0.03      0.00      0.07      0.00
                16     16.62      0.00      0.03      0.00      0.06      0.00
    24     1    15      0.10      1.17      0.07      0.00      0.17      3.14
                17      0.10      1.17      0.07      0.00      0.17      2.71
     STAAD SPACE                                              -- PAGE NO.
36
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    15      0.11      0.26      0.00      0.00      0.00      0.32
                17      0.11      0.26      0.00      0.00      0.00      0.98
           3    15      0.02      0.02      0.00      0.00      0.00      0.06
                17      0.02      0.02      0.00      0.00      0.00      0.05
    27     1    17     15.56     22.40      3.43      0.01      6.84     45.14
                18     15.56     22.40      3.43      0.01      6.87     44.48
           2    17     65.69      0.57     36.58      0.00     73.35      1.55
                18     65.69      0.57     36.58      0.00     72.99      0.71
           3    17     15.37      0.02      0.03      0.00      0.07      0.06
                18     15.37      0.02      0.03      0.00      0.06      0.03
    28     1    17      0.42      1.02      0.06      0.00      0.16      2.42
                19      0.42      1.02      0.06      0.00      0.15      3.05
           2    17      0.09      0.23      0.00      0.00      0.00      0.45
                19      0.09      0.23      0.00      0.00      0.00      1.60
           3    17      0.01      0.02      0.00      0.00      0.00      0.08
                19      0.01      0.02      0.00      0.00      0.00      0.05
    31     1    19     56.69     20.77      7.11      0.01     14.38     40.67
                20     56.69     20.77      7.11      0.01     14.07     42.41
           2    19     38.37      1.16     36.55      0.00     73.26      3.19
                20     38.37      1.16     36.55      0.00     72.94      1.46
           3    19     18.25      0.06      0.03      0.00      0.07      0.17
                20     18.25      0.06      0.03      0.00      0.06      0.08
    32     1    12      2.75      5.01      0.09      0.01      0.18      3.30
                22      2.75      5.01      0.09      0.01      0.20     16.77
           2    12      3.36     14.39      0.00      0.00      0.00     15.08
                22      3.36     14.39      0.00      0.00      0.02     42.49
           3    12      0.41      0.08      0.00      0.00      0.00      0.23
                22      0.41      0.08      0.00      0.00      0.00      0.11
    33     1    13      0.38      0.65      0.09      0.01      0.19      0.82
                23      0.38      0.65      0.09      0.01      0.19      1.76
           2    13      0.81      4.93      0.00      0.00      0.00      2.64
                23      0.81      4.93      0.00      0.00      0.00     17.10
           3    13      0.13      0.04      0.00      0.00      0.00      0.13
                23      0.13      0.04      0.00      0.00      0.00      0.04
    34     1    15      0.00      0.00      0.10      0.01      0.19      0.00
                25      0.00      0.00      0.10      0.01      0.19      0.00
           2    15      0.49      6.47      0.00      0.00      0.00      5.28
                25      0.49      6.47      0.00      0.00      0.00     20.60
           3    15      0.11      0.04      0.00      0.00      0.00      0.15
                25      0.11      0.04      0.00      0.00      0.00      0.03
     STAAD SPACE                                              -- PAGE NO.
37
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
```

```
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     35     1   17      0.38      0.65     0.09      0.01      0.19       0.82
                27      0.38      0.65     0.09      0.01      0.19       1.76
            2   17      0.81      4.93     0.00      0.00      0.00       2.64
                27      0.81      4.93     0.00      0.00      0.00      17.10
            3   17      0.13      0.04     0.00      0.00      0.00       0.13
                27      0.13      0.04     0.00      0.00      0.00       0.04
     36     1   19      2.75      5.01     0.09      0.01      0.18       3.30
                29      2.75      5.01     0.09      0.01      0.20      16.77
            2   19      3.36     14.39     0.00      0.00      0.00      15.08
                29      3.36     14.39     0.00      0.00      0.02      42.49
            3   19      0.41      0.08     0.00      0.00      0.00       0.23
                29      0.41      0.08     0.00      0.00      0.00       0.11
     37     1   21     54.99     69.83    12.43      0.02     25.85     210.76
                22     54.99     69.83    12.43      0.02     23.92      69.21
            2   21    103.61     12.47    68.22      0.00    140.13      14.20
                22    103.61     12.47    68.22      0.00    132.75      35.69
            3   21     24.40      0.24     0.07      0.00      0.12       0.46
                22     24.40      0.24     0.07      0.00      0.15       0.51
     38     1   22      4.63     12.12     0.07      0.01      0.17      31.69
                23      4.63     12.12     0.07      0.01      0.16      28.92
            2   22      2.20      0.99     0.00      0.01      0.01       3.45
                23      2.20      0.99     0.00      0.01      0.00       1.49
            3   22      0.74      0.04     0.00      0.00      0.00       0.09
                23      0.74      0.04     0.00      0.00      0.00       0.09
     39     1   23     12.51     88.23     6.79      0.02     13.43     121.29
                24     12.51     88.23     6.79      0.02     13.71     231.78
            2   23    104.37      0.56    71.09      0.00    140.44       1.70
                24    104.37      0.56    71.09      0.00    143.94       0.56
            3   23     15.56      0.17     0.07      0.00      0.15       0.39
                24     15.56      0.17     0.07      0.00      0.12       0.31
     40     1   23      1.08     10.87     0.06      0.00      0.16      27.13
                25      1.08     10.87     0.06      0.00      0.16      27.24
            2   23      0.74      0.07     0.00      0.00      0.00       0.22
                25      0.74      0.07     0.00      0.00      0.00       0.12
            3   23      1.20      0.02     0.00      0.00      0.00       0.04
                25      1.20      0.02     0.00      0.00      0.00       0.05
     41     1   25      0.00     87.68     0.00      0.02      0.00     119.61
                26      0.00     87.68     0.00      0.02      0.00     231.27
      STAAD SPACE                                              -- PAGE NO.
38
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
            2   25    107.80      0.00    70.79      0.00    139.62       0.00
                26    107.80      0.00    70.79      0.00    143.54       0.00
            3   25     13.59      0.00     0.07      0.00      0.15       0.00
                26     13.59      0.00     0.07      0.00      0.12       0.00
     42     1   25      1.08     10.87     0.06      0.00      0.16      27.24
                27      1.08     10.87     0.06      0.00      0.16      27.13
            2   25      0.74      0.07     0.00      0.00      0.00       0.12
                27      0.74      0.07     0.00      0.00      0.00       0.22
            3   25      1.20      0.02     0.00      0.00      0.00       0.05
                27      1.20      0.02     0.00      0.00      0.00       0.04
     45     1   27     12.51     88.23     6.79      0.02     13.43     121.29
```

```
                28     12.51     88.23     6.79      0.02     13.71      231.78
           2    27    104.37      0.56    71.09      0.00    140.44        1.70
                28    104.37      0.56    71.09      0.00    143.94        0.56
           3    27     15.56      0.17     0.07      0.00      0.15        0.39
                28     15.56      0.17     0.07      0.00      0.12        0.31
     46    1    27      4.63     12.12     0.07      0.01      0.16       28.92
                29      4.63     12.12     0.07      0.01      0.17       31.69
           2    27      2.20      0.99     0.00      0.01      0.00        1.49
                29      2.20      0.99     0.00      0.01      0.01        3.45
           3    27      0.74      0.04     0.00      0.00      0.00        0.09
                29      0.74      0.04     0.00      0.00      0.00        0.09
     49    1    29     54.99     69.83    12.43      0.02     23.92       69.21
                30     54.99     69.83    12.43      0.02     25.85      210.76
           2    29    103.61     12.47    68.22      0.00    132.75       35.69
                30    103.61     12.47    68.22      0.00    140.13       14.20
           3    29     24.40      0.24     0.07      0.00      0.15        0.51
                30     24.40      0.24     0.07      0.00      0.12        0.46
     58    1     2      5.23     11.56     5.06      0.01      9.92       19.99
                31      5.23     11.56     5.06      0.01     10.33       26.29
           2     2     14.55      4.66    21.61      0.00     42.59       10.66
                31     14.55      4.66    21.61      0.00     43.84        8.00
           3     2     17.53      0.07     0.06      0.00      0.12        0.12
                31     17.53      0.07     0.06      0.00      0.14        0.15
     59    1     3      3.79     18.93     3.00      0.01      6.00       36.89
                32      3.79     18.93     3.00      0.01      6.02       38.83
           2     3     17.85      0.90    22.95      0.00     45.66        2.05
                32     17.85      0.90    22.95      0.00     46.15        1.53
           3     3     17.07      0.04     0.05      0.00      0.10        0.08
                32     17.07      0.04     0.05      0.00      0.11        0.09
      STAAD SPACE                                               -- PAGE NO.
39
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     60    1     5      0.00     17.82     0.00      0.01      0.00       34.16
                33      0.00     17.82     0.00      0.01      0.00       37.12
           2     5     15.91      0.00    22.67      0.00     45.02        0.00
                33     15.91      0.00    22.67      0.00     45.68        0.00
           3     5     17.30      0.00     0.07      0.00      0.12        0.00
                33     17.30      0.00     0.07      0.00      0.14        0.00
     61    1     7      3.79     18.93     3.00      0.01      6.00       36.89
                34      3.79     18.93     3.00      0.01      6.02       38.83
           2     7     17.85      0.90    22.95      0.00     45.66        2.05
                34     17.85      0.90    22.95      0.00     46.15        1.53
           3     7     17.07      0.04     0.05      0.00      0.10        0.08
                34     17.07      0.04     0.05      0.00      0.11        0.09
     62    1     9      5.23     11.56     5.06      0.01      9.92       19.99
                35      5.23     11.56     5.06      0.01     10.33       26.29
           2     9     14.55      4.66    21.61      0.00     42.59       10.66
                35     14.55      4.66    21.61      0.00     43.84        8.00
           3     9     17.53      0.07     0.06      0.00      0.12        0.12
                35     17.53      0.07     0.06      0.00      0.14        0.15
     63    1    12     19.06     14.82     6.52      0.01     13.11       29.17
                36     19.06     14.82     6.52      0.01     12.96       30.11
           2    12     10.64      1.67    24.02      0.00     48.16        3.83
                36     10.64      1.67    24.02      0.00     47.91        2.86
           3    12     11.35      0.18     0.05      0.00      0.09        0.32
```

```
                 36      11.35       0.18      0.05       0.00       0.10        0.40
    64     1     13       3.50      17.17      2.97       0.01       5.95       34.44
                 37       3.50      17.17      2.97       0.01       5.94       34.25
           2     13      15.09       0.77     24.04       0.00      48.22        1.80
                 37      15.09       0.77     24.04       0.00      47.94        1.28
           3     13       9.54       0.06      0.04       0.00       0.08        0.11
                 37       9.54       0.06      0.04       0.00       0.09        0.13
    65     1     15       0.00      17.42      0.00       0.01       0.00       34.78
                 38       0.00      17.42      0.00       0.01       0.00       34.90
           2     15      12.71       0.00     23.96       0.00      48.03        0.00
                 38      12.71       0.00     23.96       0.00      47.80        0.00
           3     15      10.32       0.00      0.04       0.00       0.08        0.00
                 38      10.32       0.00      0.04       0.00       0.09        0.00
    66     1     17       3.50      17.17      2.97       0.01       5.95       34.44
                 39       3.50      17.17      2.97       0.01       5.94       34.25
     STAAD SPACE                                                  -- PAGE NO.
40
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2     17      15.09       0.77     24.04       0.00      48.22        1.80
                 39      15.09       0.77     24.04       0.00      47.94        1.28
           3     17       9.54       0.06      0.04       0.00       0.08        0.11
                 39       9.54       0.06      0.04       0.00       0.09        0.13
    67     1     19      19.06      14.82      6.52       0.01      13.11       29.17
                 40      19.06      14.82      6.52       0.01      12.96       30.11
           2     19      10.64       1.67     24.02       0.00      48.16        3.83
                 40      10.64       1.67     24.02       0.00      47.91        2.86
           3     19      11.35       0.18      0.05       0.00       0.09        0.32
                 40      11.35       0.18      0.05       0.00       0.10        0.40
    68     1     22      21.82      32.07      8.78       0.02      17.02       49.11
                 41      21.82      32.07      8.78       0.02      18.11       80.15
           2     22      28.71      16.68     40.36       0.00      78.66       39.23
                 41      28.71      16.68     40.36       0.00      82.77       27.55
           3     22      15.14       0.16      0.13       0.00       0.24        0.25
                 41      15.14       0.16      0.13       0.00       0.28        0.50
    69     1     23       4.80      69.20      5.92       0.02      11.81      119.83
                 42       4.80      69.20      5.92       0.02      11.88      157.08
           2     23      28.84       1.29     44.29       0.00      87.69        2.89
                 42      28.84       1.29     44.29       0.00      89.48        2.27
           3     23       9.63       0.24      0.11       0.00       0.20        0.36
                 42       9.63       0.24      0.11       0.00       0.23        0.59
    70     1     25       0.00      67.95      0.00       0.02       0.00      117.15
                 43       0.00      67.95      0.00       0.02       0.00      154.77
           2     25      29.47       0.00     43.85       0.00      86.69        0.00
                 43      29.47       0.00     43.85       0.00      88.72        0.00
           3     25       8.41       0.00      0.11       0.00       0.21        0.00
                 43       8.41       0.00      0.11       0.00       0.24        0.00
    71     1     27       4.80      69.20      5.92       0.02      11.81      119.83
                 44       4.80      69.20      5.92       0.02      11.88      157.08
           2     27      28.84       1.29     44.29       0.00      87.69        2.89
                 44      28.84       1.29     44.29       0.00      89.48        2.27
           3     27       9.63       0.24      0.11       0.00       0.20        0.36
                 44       9.63       0.24      0.11       0.00       0.23        0.59
    72     1     29      21.82      32.07      8.78       0.02      17.02       49.11
                 45      21.82      32.07      8.78       0.02      18.11       80.15
           2     29      28.71      16.68     40.36       0.00      78.66       39.23
```

```
                45     28.71      16.68     40.36       0.00      82.77      27.55
           3    29     15.14       0.16      0.13       0.00       0.24       0.25
                45     15.14       0.16      0.13       0.00       0.28       0.50
      STAAD SPACE                                                  -- PAGE NO.
41
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     73    1    31       1.47      2.49      0.12       0.00       0.30       7.55
                32       1.47      2.49      0.12       0.00       0.30       4.93
           2    31       2.42      0.73      0.00       0.00       0.01       2.85
                32       2.42      0.73      0.00       0.00       0.00       0.80
           3    31       0.35      0.07      0.00       0.00       0.00       0.15
                32       0.35      0.07      0.00       0.00       0.00       0.19
     74    1    32       0.48      1.55      0.12       0.00       0.31       3.75
                33       0.48      1.55      0.12       0.00       0.31       4.00
           2    32       2.09      0.11      0.00       0.00       0.00       0.42
                33       2.09      0.11      0.00       0.00       0.00       0.12
           3    32       0.55      0.05      0.00       0.00       0.00       0.12
                33       0.55      0.05      0.00       0.00       0.00       0.11
     75    1    33       0.48      1.55      0.12       0.00       0.31       4.00
                34       0.48      1.55      0.12       0.00       0.31       3.75
           2    33       2.09      0.11      0.00       0.00       0.00       0.12
                34       2.09      0.11      0.00       0.00       0.00       0.42
           3    33       0.55      0.05      0.00       0.00       0.00       0.11
                34       0.55      0.05      0.00       0.00       0.00       0.12
     77    1    34       1.47      2.49      0.12       0.00       0.30       4.93
                35       1.47      2.49      0.12       0.00       0.30       7.55
           2    34       2.42      0.73      0.00       0.00       0.00       0.80
                35       2.42      0.73      0.00       0.00       0.01       2.85
           3    34       0.35      0.07      0.00       0.00       0.00       0.19
                35       0.35      0.07      0.00       0.00       0.00       0.15
     79    1    31       1.13      1.56      0.18       0.00       0.36       5.30
                36       1.13      1.56      0.18       0.00       0.36       1.72
           2    31       2.21      3.31      0.01       0.00       0.02       9.67
                36       2.21      3.31      0.01       0.00       0.00       3.59
           3    31       0.20      0.13      0.00       0.00       0.00       0.14
                36       0.20      0.13      0.00       0.00       0.00       0.41
     80    1    32       0.14      0.27      0.18       0.00       0.36       0.52
                37       0.14      0.27      0.18       0.00       0.36       0.58
           2    32       0.91      0.98      0.00       0.00       0.00       3.17
                37       0.91      0.98      0.00       0.00       0.00       0.75
           3    32       0.06      0.06      0.00       0.00       0.00       0.03
                37       0.06      0.06      0.00       0.00       0.00       0.21
     81    1    33       0.00      0.00      0.18       0.00       0.36       0.00
                38       0.00      0.00      0.18       0.00       0.36       0.00
      STAAD SPACE                                                  -- PAGE NO.
42
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    33       0.44      1.44      0.00       0.00       0.00       4.34
                38       0.44      1.44      0.00       0.00       0.00       1.42
           3    33       0.05      0.06      0.00       0.00       0.00       0.06
                38       0.05      0.06      0.00       0.00       0.00       0.25
     82    1    34       0.14      0.27      0.18       0.00       0.36       0.52
```

```
               39       0.14      0.27     0.18      0.00      0.36       0.58
          2    34       0.91      0.98     0.00      0.00      0.00       3.17
               39       0.91      0.98     0.00      0.00      0.00       0.75
          3    34       0.06      0.06     0.00      0.00      0.00       0.03
               39       0.06      0.06     0.00      0.00      0.00       0.21
     83   1    35       1.13      1.56     0.18      0.00      0.36       5.30
               40       1.13      1.56     0.18      0.00      0.36       1.72
          2    35       2.21      3.31     0.01      0.00      0.02       9.67
               40       2.21      3.31     0.01      0.00      0.00       3.59
          3    35       0.20      0.13     0.00      0.00      0.00       0.14
               40       0.20      0.13     0.00      0.00      0.00       0.41
     84   1    36       0.24      0.88     0.12      0.00      0.30       2.16
               37       0.24      0.88     0.12      0.00      0.30       2.35
          2    36       0.26      0.02     0.00      0.00      0.00       0.42
               37       0.26      0.02     0.00      0.00      0.00       0.43
          3    36       0.02      0.03     0.00      0.00      0.00       0.07
               37       0.02      0.03     0.00      0.00      0.00       0.12
     85   1    37       0.35      0.94     0.12      0.00      0.31       2.18
               38       0.35      0.94     0.12      0.00      0.31       2.54
          2    37       0.30      0.02     0.00      0.00      0.00       0.16
               38       0.30      0.02     0.00      0.00      0.00       0.10
          3    37       0.02      0.03     0.00      0.00      0.00       0.08
               38       0.02      0.03     0.00      0.00      0.00       0.10
     86   1    38       0.35      0.94     0.12      0.00      0.31       2.54
               39       0.35      0.94     0.12      0.00      0.31       2.18
          2    38       0.30      0.02     0.00      0.00      0.00       0.10
               39       0.30      0.02     0.00      0.00      0.00       0.16
          3    38       0.02      0.03     0.00      0.00      0.00       0.10
               39       0.02      0.03     0.00      0.00      0.00       0.08
     88   1    39       0.24      0.88     0.12      0.00      0.30       2.35
               40       0.24      0.88     0.12      0.00      0.30       2.16
          2    39       0.26      0.02     0.00      0.00      0.00       0.43
               40       0.26      0.02     0.00      0.00      0.00       0.42
          3    39       0.02      0.03     0.00      0.00      0.00       0.12
               40       0.02      0.03     0.00      0.00      0.00       0.07
      STAAD SPACE                                                -- PAGE NO.
43
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     90   1    36       1.74      3.43     0.18      0.01      0.36       2.11
               41       1.74      3.43     0.18      0.01      0.35      11.66
          2    36       0.73      5.68     0.00      0.00      0.01       5.65
               41       0.73      5.68     0.00      0.00      0.01      17.08
          3    36       0.20      0.12     0.00      0.00      0.00       0.37
               41       0.20      0.12     0.00      0.00      0.00       0.14
     91   1    37       0.17      0.22     0.18      0.01      0.35       0.18
               42       0.17      0.22     0.18      0.01      0.36       0.72
          2    37       0.51      1.76     0.00      0.00      0.00       0.38
               42       0.51      1.76     0.00      0.00      0.00       6.68
          3    37       0.06      0.06     0.00      0.00      0.00       0.21
               42       0.06      0.06     0.00      0.00      0.00       0.04
     92   1    38       0.00      0.00     0.18      0.01      0.36       0.00
               43       0.00      0.00     0.18      0.01      0.36       0.00
          2    38       0.58      2.49     0.00      0.00      0.00       1.63
               43       0.58      2.49     0.00      0.00      0.00       8.32
          3    38       0.05      0.06     0.00      0.00      0.00       0.23
```

```
                43      0.05      0.06      0.00      0.00      0.00      0.04
    93     1    39      0.17      0.22      0.18      0.01      0.35      0.18
                44      0.17      0.22      0.18      0.01      0.36      0.72
           2    39      0.51      1.76      0.00      0.00      0.00      0.38
                44      0.51      1.76      0.00      0.00      0.00      6.68
           3    39      0.06      0.06      0.00      0.00      0.00      0.21
                44      0.06      0.06      0.00      0.00      0.00      0.04
    94     1    40      1.74      3.43      0.18      0.01      0.36      2.11
                45      1.74      3.43      0.18      0.01      0.35     11.66
           2    40      0.73      5.68      0.00      0.00      0.01      5.65
                45      0.73      5.68      0.00      0.00      0.01     17.08
           3    40      0.20      0.12      0.00      0.00      0.00      0.37
                45      0.20      0.12      0.00      0.00      0.00      0.14
    95     1    41      2.54      8.57      0.12      0.01      0.29     22.98
                42      2.54      8.57      0.12      0.01      0.30     19.86
           2    41      2.89      0.45      0.00      0.00      0.00      1.58
                42      2.89      0.45      0.00      0.00      0.00      0.68
           3    41      0.34      0.04      0.00      0.00      0.00      0.08
                42      0.34      0.04      0.00      0.00      0.00      0.11
    96     1    42      0.13      7.20      0.13      0.00      0.32     17.97
                43      0.13      7.20      0.13      0.00      0.32     18.04
      STAAD SPACE                                              -- PAGE NO.
44
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    42      2.42      0.04      0.00      0.00      0.00      0.12
                43      2.42      0.04      0.00      0.00      0.00      0.08
           3    42      0.55      0.02      0.00      0.00      0.00      0.06
                43      0.55      0.02      0.00      0.00      0.00      0.06
    97     1    43      0.13      7.20      0.13      0.00      0.32     18.04
                44      0.13      7.20      0.13      0.00      0.32     17.97
           2    43      2.42      0.04      0.00      0.00      0.00      0.08
                44      2.42      0.04      0.00      0.00      0.00      0.12
           3    43      0.55      0.02      0.00      0.00      0.00      0.06
                44      0.55      0.02      0.00      0.00      0.00      0.06
    99     1    44      2.54      8.57      0.12      0.01      0.30     19.86
                45      2.54      8.57      0.12      0.01      0.29     22.98
           2    44      2.89      0.45      0.00      0.00      0.00      0.68
                45      2.89      0.45      0.00      0.00      0.00      1.58
           3    44      0.34      0.04      0.00      0.00      0.00      0.11
                45      0.34      0.04      0.00      0.00      0.00      0.08
   ************** END OF LATEST ANALYSIS RESULT **************
   158. PRINT ELEMENT JOINT STRESSES
  ELEMENT  JOINT    STRESSES
      STAAD SPACE                                              -- PAGE NO.
45
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    50      1         25.03        6.33        2.65        3.23        1.19
                     249.33      234.33        6.64        3.39        3.88
                     286.20      269.40
     TOP : SMAX=     286.20 SMIN=     116.11 TMAX=      85.05 ANGLE= 51.0
```

```
       BOTT: SMAX=    -112.84 SMIN=    -269.40 TMAX=      78.28 ANGLE=-37.2
         JOINT        39.41        14.77       25.92       13.01        6.19
            2       1669.46      1647.55        7.81        8.24        6.50
       TOP : SMAX=    1906.34 SMIN=     705.19 TMAX=     600.58 ANGLE= 22.1
       BOTT: SMAX=    -697.82 SMIN=   -1881.62 TMAX=     591.90 ANGLE=-68.3
         JOINT        39.41         2.44       18.64        0.97        6.84
            3       1459.30      1433.47       12.85        5.00        4.32
       TOP : SMAX=    1413.17 SMIN=     -88.26 TMAX=     750.72 ANGLE= 18.9
       BOTT: SMAX=      94.61 SMIN=   -1383.82 TMAX=     739.21 ANGLE=-71.2
         JOINT        10.87         2.44        2.55        2.68        3.81
           13        481.26       465.08        6.84        3.67        3.85
       TOP : SMAX=     437.37 SMIN=     -78.20 TMAX=     257.79 ANGLE= 45.3
       BOTT: SMAX=      81.07 SMIN=    -419.22 TMAX=     250.15 ANGLE=-44.3
         JOINT        10.87        14.77        2.99        5.50        4.46
           12        615.39       595.53        3.59       10.64        3.49
       TOP : SMAX=     603.48 SMIN=     -23.16 TMAX=     313.32 ANGLE= 53.1
       BOTT: SMAX=      28.77 SMIN=    -580.63 TMAX=     304.70 ANGLE=-37.4
             2         1.16        34.32        0.95        9.01        0.70
                     581.76       573.12        1.47        4.81        0.76
                     609.62       599.79
       TOP : SMAX=     609.62 SMIN=      60.44 TMAX=     274.59 ANGLE= 85.0
       BOTT: SMAX=     -57.71 SMIN=    -599.79 TMAX=     271.04 ANGLE= -4.9
         JOINT         2.51        51.80        1.28       38.53        5.82
            2       2619.90      2610.02        2.81        5.20        2.70
       TOP : SMAX=    2633.75 SMIN=      27.93 TMAX=    1302.91 ANGLE= 81.3
       BOTT: SMAX=     -23.83 SMIN=   -2621.85 TMAX=    1299.01 ANGLE= -8.6
         JOINT         2.51        16.85        2.99       11.86        3.96
            3        854.00       839.31        3.01        2.73        6.22
       TOP : SMAX=     898.48 SMIN=      97.30 TMAX=     400.59 ANGLE= 68.8
       BOTT: SMAX=     -99.63 SMIN=    -884.68 TMAX=     392.53 ANGLE=-20.5
         JOINT         0.19        16.85        2.60        0.09        4.41
           13        543.59       531.45        0.37        4.43        4.03
       TOP : SMAX=     401.33 SMIN=    -217.30 TMAX=     309.31 ANGLE= 37.3
       BOTT: SMAX=     215.45 SMIN=    -389.89 TMAX=     302.67 ANGLE=-53.2
         JOINT         0.19        51.80        2.90       14.29        5.37
           12       1084.31      1055.92        3.08       12.30        5.16
       TOP : SMAX=    1109.32 SMIN=      51.88 TMAX=     528.72 ANGLE= 68.3
       BOTT: SMAX=     -50.27 SMIN=   -1080.15 TMAX=     514.94 ANGLE=-21.6
             3         0.08         0.21        0.02        0.10        0.04
                       9.93         7.71        2.63        0.05        1.81
                      10.21         8.34
       TOP : SMAX=      10.21 SMIN=       0.58 TMAX=       4.81 ANGLE= 53.9
       BOTT: SMAX=       1.46 SMIN=      -6.88 TMAX=       4.17 ANGLE= -6.7
      STAAD SPACE                                               -- PAGE NO.
46
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY         MX          MY          MXY
                      VONT       VONB        SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT         0.15         0.43        0.12        0.13        0.07
            2         13.83         9.73        4.12        1.65        0.05
       TOP : SMAX=      15.88 SMIN=       6.49 TMAX=       4.70 ANGLE= 38.2
       BOTT: SMAX=      -0.75 SMIN=     -10.08 TMAX=       4.67 ANGLE=-36.5
         JOINT         0.15         0.26        0.15        0.09        0.07
            3         19.77         7.86        9.66        2.15        0.77
       TOP : SMAX=      22.04 SMIN=       5.89 TMAX=       8.08 ANGLE= 21.8
```

```
      BOTT: SMAX=       2.22 SMIN=      -6.52 TMAX=       4.37 ANGLE=-33.8
        JOINT        0.28        0.26        0.20        0.24        0.04
          13        19.47       13.30        1.16        1.57        3.65
      TOP : SMAX=      22.37 SMIN=       9.24 TMAX=       6.57 ANGLE= 51.6
      BOTT: SMAX=     -11.65 SMIN=     -14.48 TMAX=       1.41 ANGLE= 20.1
        JOINT        0.28        0.43        0.15        0.37        0.02
          12        25.50       20.90        4.40        2.24        4.38
      TOP : SMAX=      29.27 SMIN=      11.89 TMAX=       8.69 ANGLE= 68.5
      BOTT: SMAX=      -4.96 SMIN=     -22.94 TMAX=       8.99 ANGLE=  9.2
    51       1      18.44        0.70        0.51        0.23        1.48
                   189.37      158.03        3.14        0.55        8.90
                   216.49      180.55
      TOP : SMAX=     134.88 SMIN=     -81.61 TMAX=     108.24 ANGLE= 42.2
      BOTT: SMAX=      67.33 SMIN=    -113.22 TMAX=      90.27 ANGLE=-47.5
        JOINT       28.02        3.32       14.69        3.57        5.50
           3      1103.98     1074.23       11.13        2.83        6.67
      TOP : SMAX=    1144.80 SMIN=      86.76 TMAX=     529.02 ANGLE= 22.4
      BOTT: SMAX=     -88.08 SMIN=   -1115.56 TMAX=     513.74 ANGLE=-67.8
        JOINT       28.02        3.69       16.36        3.30        5.90
           5      1220.29     1198.50        5.66        2.76        7.22
      TOP : SMAX=    1252.39 SMIN=      66.95 TMAX=     592.72 ANGLE= 21.3
      BOTT: SMAX=     -70.36 SMIN=   -1232.13 TMAX=     580.89 ANGLE=-69.2
        JOINT       11.15        3.69        4.97        1.25        3.11
          15       484.89      449.25        5.14        2.12       11.27
      TOP : SMAX=     463.03 SMIN=     -41.10 TMAX=     252.06 ANGLE= 30.0
      BOTT: SMAX=      27.48 SMIN=    -434.88 TMAX=     231.18 ANGLE=-61.0
        JOINT       11.15        3.32        4.38        1.44        3.45
          13       490.35      459.13        1.08        3.52       10.61
      TOP : SMAX=     455.82 SMIN=     -62.98 TMAX=     259.40 ANGLE= 34.1
      BOTT: SMAX=      49.02 SMIN=    -432.65 TMAX=     240.83 ANGLE=-57.1
             2       1.50       21.29        1.64        6.64        0.30
                   403.14      398.99        2.31        2.80        0.09
                   446.79      441.17
      TOP : SMAX=     446.79 SMIN=     110.23 TMAX=     168.28 ANGLE= 86.6
      BOTT: SMAX=    -105.62 SMIN=    -441.17 TMAX=     167.78 ANGLE= -3.4
        JOINT        1.05       16.70        3.77       13.53        3.21
           3       888.88      886.23        5.77        2.39        0.74
      TOP : SMAX=     969.20 SMIN=     192.04 TMAX=     388.58 ANGLE= 73.2
      BOTT: SMAX=    -181.86 SMIN=    -963.06 TMAX=     390.60 ANGLE=-16.6
     STAAD SPACE                                                -- PAGE NO.
47
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
        JOINT        1.05       25.88        2.92       19.64        3.24
           5      1281.06     1277.39        4.66        3.47        0.44
      TOP : SMAX=    1353.09 SMIN=     158.88 TMAX=     597.10 ANGLE= 79.4
      BOTT: SMAX=    -149.96 SMIN=   -1345.74 TMAX=     597.89 ANGLE=-10.6
        JOINT        1.95       25.88        1.17        5.62        3.81
          15       559.69      554.70        1.16        3.26        0.56
      TOP : SMAX=     523.35 SMIN=     -66.70 TMAX=     295.02 ANGLE= 60.2
      BOTT: SMAX=      69.09 SMIN=    -516.91 TMAX=     293.00 ANGLE=-29.9
        JOINT        1.95       16.70        1.02        0.98        2.64
          13       313.46      311.86        0.67        2.26        0.29
      TOP : SMAX=     244.95 SMIN=    -108.30 TMAX=     176.63 ANGLE= 44.9
```

```
      BOTT: SMAX=     110.66 SMIN=    -241.44 TMAX=     176.05 ANGLE=-45.4
            3          0.11        0.25        0.03        0.12        0.01
                       8.91       10.28        6.39        0.34        0.96
                      10.18       11.74
      TOP : SMAX=      10.18 SMIN=       6.33 TMAX=       1.93 ANGLE= 42.3
      BOTT: SMAX=       4.35 SMIN=      -7.39 TMAX=       5.87 ANGLE=  0.0
        JOINT          0.13        0.27        0.09        0.04        0.06
            3         19.21        9.42       12.60        0.39        0.93
      TOP : SMAX=      20.09 SMIN=       1.90 TMAX=       9.09 ANGLE= 15.8
      BOTT: SMAX=       7.31 SMIN=      -3.33 TMAX=       5.32 ANGLE=-16.5
        JOINT          0.13        0.29        0.09        0.06        0.04
            5         18.46       10.57       13.61        0.87        0.20
      TOP : SMAX=      20.16 SMIN=       4.07 TMAX=       8.05 ANGLE= 11.3
      BOTT: SMAX=       8.32 SMIN=      -3.58 TMAX=       5.95 ANGLE=-13.4
        JOINT          0.20        0.29        0.14        0.27        0.04
           15         17.08       15.63        0.25        0.30        0.99
      TOP : SMAX=      19.65 SMIN=       8.34 TMAX=       5.65 ANGLE= 70.2
      BOTT: SMAX=      -8.84 SMIN=     -18.04 TMAX=       4.60 ANGLE=-10.2
        JOINT          0.20        0.27        0.12        0.23        0.06
           13         16.89       13.87        0.84        0.19        1.73
      TOP : SMAX=      18.92 SMIN=       5.35 TMAX=       6.78 ANGLE= 60.8
      BOTT: SMAX=      -6.30 SMIN=     -15.90 TMAX=       4.80 ANGLE=-14.5
    52      1         18.44        0.70        0.51        0.23        1.48
                     189.37      158.03        3.14        0.55        8.90
                     216.49      180.55
      TOP : SMAX=     134.88 SMIN=     -81.61 TMAX=     108.24 ANGLE= 42.2
      BOTT: SMAX=      67.33 SMIN=    -113.22 TMAX=      90.27 ANGLE=-47.5
        JOINT         28.02        3.69       16.36        3.30        5.90
            5       1220.29     1198.50        5.66        2.76        7.22
      TOP : SMAX=    1252.39 SMIN=      66.95 TMAX=     592.72 ANGLE= 21.3
      BOTT: SMAX=     -70.36 SMIN=   -1232.13 TMAX=     580.89 ANGLE=-69.2
        JOINT         28.02        3.32       14.69        3.57        5.50
            7       1103.98     1074.23       11.13        2.83        6.67
      TOP : SMAX=    1144.80 SMIN=      86.76 TMAX=     529.02 ANGLE= 22.4
      BOTT: SMAX=     -88.08 SMIN=   -1115.56 TMAX=     513.74 ANGLE=-67.8
     STAAD SPACE                                                -- PAGE NO.
48
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD        SQX         SQY          MX          MY          MXY
                       VONT        VONB         SX          SY          SXY
                       TRESCAT     TRESCAB
        JOINT         11.15        3.32        4.38        1.44        3.45
           17        490.35      459.13        1.08        3.52       10.61
      TOP : SMAX=     455.82 SMIN=     -62.98 TMAX=     259.40 ANGLE= 34.1
      BOTT: SMAX=      49.02 SMIN=    -432.65 TMAX=     240.83 ANGLE=-57.1
        JOINT         11.15        3.69        4.97        1.25        3.11
           15        484.89      449.25        5.14        2.12       11.27
      TOP : SMAX=     463.03 SMIN=     -41.10 TMAX=     252.06 ANGLE= 30.0
      BOTT: SMAX=      27.48 SMIN=    -434.88 TMAX=     231.18 ANGLE=-61.0
            2          1.50       21.29        1.64        6.64        0.30
                     403.14      398.99        2.31        2.80        0.09
                     446.79      441.17
      TOP : SMAX=     446.79 SMIN=     110.23 TMAX=     168.28 ANGLE= 86.6
      BOTT: SMAX=    -105.62 SMIN=    -441.17 TMAX=     167.78 ANGLE= -3.4
        JOINT          1.05       25.88        2.92       19.64        3.24
            5       1281.06     1277.39        4.66        3.47        0.44
```

```
      TOP : SMAX=    1353.09 SMIN=     158.88 TMAX=     597.10 ANGLE= 79.4
      BOTT: SMAX=    -149.96 SMIN=   -1345.74 TMAX=     597.89 ANGLE=-10.6
        JOINT         1.05       16.70         3.77      13.53         3.21
          7         888.88      886.23         5.77       2.39         0.74
      TOP : SMAX=     969.20 SMIN=     192.04 TMAX=     388.58 ANGLE= 73.2
      BOTT: SMAX=    -181.86 SMIN=    -963.06 TMAX=     390.60 ANGLE=-16.6
        JOINT         1.95       16.70         1.02       0.98         2.64
         17         313.46      311.86         0.67       2.26         0.29
      TOP : SMAX=     244.95 SMIN=    -108.30 TMAX=     176.63 ANGLE= 44.9
      BOTT: SMAX=     110.66 SMIN=    -241.44 TMAX=     176.05 ANGLE=-45.4
        JOINT         1.95       25.88         1.17       5.62         3.81
         15         559.69      554.70         1.16       3.26         0.56
      TOP : SMAX=     523.35 SMIN=     -66.70 TMAX=     295.02 ANGLE= 60.2
      BOTT: SMAX=      69.09 SMIN=    -516.91 TMAX=     293.00 ANGLE=-29.9
           3          0.11        0.25         0.03       0.12         0.01
                      8.91       10.28         6.39       0.34         0.96
                     10.18       11.74
      TOP : SMAX=      10.18 SMIN=       6.33 TMAX=       1.93 ANGLE= 42.3
      BOTT: SMAX=       4.35 SMIN=      -7.39 TMAX=       5.87 ANGLE=  0.0
        JOINT         0.13        0.29         0.09       0.06         0.04
          5          18.46       10.57        13.61       0.87         0.20
      TOP : SMAX=      20.16 SMIN=       4.07 TMAX=       8.05 ANGLE= 11.3
      BOTT: SMAX=       8.32 SMIN=      -3.58 TMAX=       5.95 ANGLE=-13.4
        JOINT         0.13        0.27         0.09       0.04         0.06
          7          19.21        9.42        12.60       0.39         0.93
      TOP : SMAX=      20.09 SMIN=       1.90 TMAX=       9.09 ANGLE= 15.8
      BOTT: SMAX=       7.31 SMIN=      -3.33 TMAX=       5.32 ANGLE=-16.5
        JOINT         0.20        0.27         0.12       0.23         0.06
         17          16.89       13.87         0.84       0.19         1.73
      TOP : SMAX=      18.92 SMIN=       5.35 TMAX=       6.78 ANGLE= 60.8
      BOTT: SMAX=      -6.30 SMIN=     -15.90 TMAX=       4.80 ANGLE=-14.5
        JOINT         0.20        0.29         0.14       0.27         0.04
         15          17.08       15.63         0.25       0.30         0.99
      TOP : SMAX=      19.65 SMIN=       8.34 TMAX=       5.65 ANGLE= 70.2
      BOTT: SMAX=      -8.84 SMIN=     -18.04 TMAX=       4.60 ANGLE=-10.2
     STAAD SPACE                                                -- PAGE NO.
49
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    53      1         25.03        6.33         2.65       3.23         1.19
                     249.33      234.33         6.64       3.39         3.88
                     286.20      269.40
      TOP : SMAX=     286.20 SMIN=     116.11 TMAX=      85.05 ANGLE= 51.0
      BOTT: SMAX=    -112.84 SMIN=    -269.40 TMAX=      78.28 ANGLE=-37.2
        JOINT        39.41        2.44        18.64       0.97         6.84
          7        1459.30     1433.47        12.85       5.00         4.32
      TOP : SMAX=    1413.17 SMIN=     -88.26 TMAX=     750.72 ANGLE= 18.9
      BOTT: SMAX=      94.61 SMIN=   -1383.82 TMAX=     739.21 ANGLE=-71.2
        JOINT        39.41       14.77        25.92      13.01         6.19
          9        1669.46     1647.55         7.81       8.24         6.50
      TOP : SMAX=    1906.34 SMIN=     705.19 TMAX=     600.58 ANGLE= 22.1
      BOTT: SMAX=    -697.82 SMIN=   -1881.62 TMAX=     591.90 ANGLE=-68.3
        JOINT        10.87       14.77         2.99       5.50         4.46
         19         615.39      595.53         3.59      10.64         3.49
```

```
       TOP : SMAX=     603.48 SMIN=     -23.16 TMAX=     313.32 ANGLE= 53.1
       BOTT: SMAX=      28.77 SMIN=    -580.63 TMAX=     304.70 ANGLE=-37.4
         JOINT        10.87         2.44         2.55         2.68         3.81
           17        481.26       465.08         6.84         3.67         3.85
       TOP : SMAX=     437.37 SMIN=     -78.20 TMAX=     257.79 ANGLE= 45.3
       BOTT: SMAX=      81.07 SMIN=    -419.22 TMAX=     250.15 ANGLE=-44.3
             2         1.16        34.32         0.95         9.01         0.70
                     581.76       573.12         1.47         4.81         0.76
                     609.62       599.79
       TOP : SMAX=     609.62 SMIN=      60.44 TMAX=     274.59 ANGLE= 85.0
       BOTT: SMAX=     -57.71 SMIN=    -599.79 TMAX=     271.04 ANGLE= -4.9
         JOINT         2.51        16.85         2.99        11.86         3.96
            7        854.00       839.31         3.01         2.73         6.22
       TOP : SMAX=     898.48 SMIN=      97.30 TMAX=     400.59 ANGLE= 68.8
       BOTT: SMAX=     -99.63 SMIN=    -884.68 TMAX=     392.53 ANGLE=-20.5
         JOINT         2.51        51.80         1.28        38.53         5.82
            9       2619.90      2610.02         2.81         5.20         2.70
       TOP : SMAX=    2633.75 SMIN=      27.93 TMAX=    1302.91 ANGLE= 81.3
       BOTT: SMAX=     -23.83 SMIN=   -2621.85 TMAX=    1299.01 ANGLE= -8.6
         JOINT         0.19        51.80         2.90        14.29         5.37
           19       1084.31      1055.92         3.08        12.30         5.16
       TOP : SMAX=    1109.32 SMIN=      51.88 TMAX=     528.72 ANGLE= 68.3
       BOTT: SMAX=     -50.27 SMIN=   -1080.15 TMAX=     514.94 ANGLE=-21.6
         JOINT         0.19        16.85         2.60         0.09         4.41
           17        543.59       531.45         0.37         4.43         4.03
       TOP : SMAX=     401.33 SMIN=    -217.30 TMAX=     309.31 ANGLE= 37.3
       BOTT: SMAX=     215.45 SMIN=    -389.89 TMAX=     302.67 ANGLE=-53.2
             3         0.08         0.21         0.02         0.10         0.04
                       9.93         7.71         2.63         0.05         1.81
                      10.21         8.34
       TOP : SMAX=      10.21 SMIN=       0.58 TMAX=       4.81 ANGLE= 53.9
       BOTT: SMAX=       1.46 SMIN=      -6.88 TMAX=       4.17 ANGLE= -6.7
      STAAD SPACE                                                 -- PAGE NO.
50
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
         JOINT         0.15         0.26         0.15         0.09         0.07
            7         19.77         7.86         9.66         2.15         0.77
       TOP : SMAX=      22.04 SMIN=       5.89 TMAX=       8.08 ANGLE= 21.8
       BOTT: SMAX=       2.22 SMIN=      -6.52 TMAX=       4.37 ANGLE=-33.8
         JOINT         0.15         0.43         0.12         0.13         0.07
            9         13.83         9.73         4.12         1.65         0.05
       TOP : SMAX=      15.88 SMIN=       6.49 TMAX=       4.70 ANGLE= 38.2
       BOTT: SMAX=      -0.75 SMIN=     -10.08 TMAX=       4.67 ANGLE=-36.5
         JOINT         0.28         0.43         0.15         0.37         0.02
           19         25.50        20.90         4.40         2.24         4.38
       TOP : SMAX=      29.27 SMIN=      11.89 TMAX=       8.69 ANGLE= 68.5
       BOTT: SMAX=      -4.96 SMIN=     -22.94 TMAX=       8.99 ANGLE=  9.2
         JOINT         0.28         0.26         0.20         0.24         0.04
           17         19.47        13.30         1.16         1.57         3.65
       TOP : SMAX=      22.37 SMIN=       9.24 TMAX=       6.57 ANGLE= 51.6
       BOTT: SMAX=     -11.65 SMIN=     -14.48 TMAX=       1.41 ANGLE= 20.1
     54      1        43.06        13.99         1.24         5.15         3.66
                     543.11       506.30        12.17         3.54        13.21
```

```
                    572.96      534.50
       TOP : SMAX=     507.29 SMIN=     -65.67 TMAX=     286.48 ANGLE= 58.1
       BOTT: SMAX=      62.15 SMIN=    -472.35 TMAX=     267.25 ANGLE=-29.9
         JOINT       12.54       21.19        7.06        3.30        6.99
           12       923.40      886.20       23.06       11.02        6.21
       TOP : SMAX=     852.57 SMIN=    -128.24 TMAX=     490.41 ANGLE= 37.2
       BOTT: SMAX=     147.20 SMIN=    -803.38 TMAX=     475.29 ANGLE=-52.3
         JOINT       12.54        6.92        7.10        0.98        5.17
           13       756.83      730.91        7.83       15.59        8.57
       TOP : SMAX=     686.75 SMIN=    -124.68 TMAX=     405.72 ANGLE= 30.2
       BOTT: SMAX=     137.29 SMIN=    -652.53 TMAX=     394.91 ANGLE=-60.9
         JOINT       76.73        6.92       44.34        5.64       14.24
           23      3288.69     3183.61       47.35       17.88       20.59
       TOP : SMAX=    3324.36 SMIN=      72.55 TMAX=    1625.91 ANGLE= 18.3
       BOTT: SMAX=     -55.47 SMIN=   -3210.98 TMAX=    1577.75 ANGLE=-72.0
         JOINT       76.73       21.19       45.29       18.73       11.74
           22      2996.06     2917.13       32.15        8.74       17.88
       TOP : SMAX=    3356.45 SMIN=     952.23 TMAX=    1202.11 ANGLE= 20.9
       BOTT: SMAX=    -952.56 SMIN=   -3274.34 TMAX=    1160.89 ANGLE=-69.4
             2        8.55       67.56        2.79       16.20        1.23
                   1016.68     1003.64        2.14        7.40        0.14
                   1094.82     1080.04
       TOP : SMAX=    1094.82 SMIN=     180.44 TMAX=     457.19 ANGLE= 84.8
       BOTT: SMAX=    -176.13 SMIN=   -1080.04 TMAX=     451.96 ANGLE= -5.2
         JOINT        7.66       95.23        1.11       26.38        8.29
           12      1995.38     1946.49       19.14       25.55       11.83
       TOP : SMAX=    1955.25 SMIN=     -77.98 TMAX=    1016.62 ANGLE= 73.1
       BOTT: SMAX=     104.33 SMIN=   -1892.22 TMAX=     998.28 ANGLE=-16.4
      STAAD SPACE                                                 -- PAGE NO.
51
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT        7.66       39.89        3.33        4.71       11.25
           13      1333.47     1324.56        1.51        3.73        2.25
       TOP : SMAX=    1024.67 SMIN=    -482.99 TMAX=     753.83 ANGLE= 46.8
       BOTT: SMAX=     483.60 SMIN=   -1014.80 TMAX=     749.20 ANGLE=-43.3
         JOINT        9.45       39.89        3.15       26.41        5.83
           23      1806.20     1786.74       21.66       11.02       11.81
       TOP : SMAX=    1869.30 SMIN=     133.60 TMAX=     867.85 ANGLE= 76.3
       BOTT: SMAX=    -102.00 SMIN=   -1835.55 TMAX=     866.78 ANGLE=-12.9
         JOINT        9.45       95.23       10.22       69.49       13.71
           22      4628.34     4596.05        5.26       17.88        2.31
       TOP : SMAX=    4851.88 SMIN=     485.32 TMAX=    2183.28 ANGLE= 77.6
       BOTT: SMAX=    -475.58 SMIN=   -4815.35 TMAX=    2169.89 ANGLE=-12.4
             3        0.23        0.27        0.02        0.09        0.03
                      8.59        7.05        2.51        0.05        1.88
                      8.98        7.67
       TOP : SMAX=       8.98 SMIN=       0.83 TMAX=       4.07 ANGLE= 54.9
       BOTT: SMAX=       1.49 SMIN=      -6.18 TMAX=       3.84 ANGLE= -0.5
         JOINT        0.29        0.45        0.12        0.35        0.04
           12        24.51       19.80        4.21        1.86        4.25
       TOP : SMAX=      27.85 SMIN=       9.57 TMAX=       9.14 ANGLE= 66.6
       BOTT: SMAX=      -3.84 SMIN=     -21.44 TMAX=       8.80 ANGLE=  6.0
         JOINT        0.29        0.26        0.18        0.22        0.08
```

```
          13          21.18       12.37        1.07         1.47         3.71
      TOP : SMAX=       23.41 SMIN=       5.58 TMAX=        8.91 ANGLE= 50.4
      BOTT: SMAX=      -10.11 SMIN=     -13.80 TMAX=        1.85 ANGLE=-23.3
        JOINT          0.25        0.26        0.13         0.08         0.08
          23          19.00        9.30        9.23         1.90         0.51
      TOP : SMAX=       20.82 SMIN=       4.41 TMAX=        8.20 ANGLE= 23.6
      BOTT: SMAX=        3.81 SMIN=      -6.78 TMAX=        5.30 ANGLE=-35.3
        JOINT          0.25        0.45        0.14         0.15         0.09
          22          16.28       12.51        3.96         1.43         0.11
      TOP : SMAX=       18.56 SMIN=       6.67 TMAX=        5.95 ANGLE= 40.7
      BOTT: SMAX=       -1.32 SMIN=     -13.11 TMAX=        5.90 ANGLE=-37.0
    55      1         43.80        2.76        0.16         0.34         4.30
                     521.26      472.46        5.53         0.43        14.09
                     601.46      545.31
      TOP : SMAX=      320.50 SMIN=    -280.96 TMAX=      300.73 ANGLE= 45.3
      BOTT: SMAX=      258.85 SMIN=    -286.46 TMAX=      272.66 ANGLE=-44.1
        JOINT         12.04        8.85        4.43         1.71         6.14
          13         785.45      722.99        5.29         1.22        18.19
      TOP : SMAX=      645.32 SMIN=    -229.24 TMAX=      437.28 ANGLE= 38.9
      BOTT: SMAX=      199.34 SMIN=    -602.41 TMAX=      400.88 ANGLE=-51.4
        JOINT         12.04        7.52        5.17         1.13         6.12
          15         802.64      744.85        4.93         1.88        17.17
      TOP : SMAX=      660.05 SMIN=    -233.43 TMAX=      446.74 ANGLE= 36.1
      BOTT: SMAX=      206.68 SMIN=    -619.69 TMAX=      413.18 ANGLE=-54.4
     STAAD SPACE                                                -- PAGE NO.
52
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX           MY          MXY
                      VONT        VONB         SX           SY          SXY
                      TRESCAT     TRESCAB
        JOINT         75.82        7.52       43.56         7.61        14.74
          25        3194.55     3166.08        6.09         1.11        10.24
      TOP : SMAX=     3267.53 SMIN=     151.34 TMAX=     1558.09 ANGLE= 19.8
      BOTT: SMAX=     -160.98 SMIN=   -3243.50 TMAX=     1541.26 ANGLE=-70.4
        JOINT         75.82        8.85       43.50         8.62        14.69
          23        3177.37     3130.97       15.92         1.68        11.25
      TOP : SMAX=     3278.60 SMIN=     213.19 TMAX=     1532.70 ANGLE= 20.1
      BOTT: SMAX=     -220.98 SMIN=   -3235.60 TMAX=     1507.31 ANGLE=-70.0
            2          1.67       45.35        1.74        11.48         0.10
                     720.11      708.83        3.82         7.14         0.08
                     772.57      758.29
      TOP : SMAX=      772.57 SMIN=     120.00 TMAX=      326.29 ANGLE= 89.4
      BOTT: SMAX=     -112.36 SMIN=    -758.29 TMAX=      322.97 ANGLE= -0.6
        JOINT          1.87       40.93        2.96         7.46         7.31
          13         953.49      945.71        0.88         9.11         0.07
      TOP : SMAX=      863.92 SMIN=    -159.12 TMAX=      511.52 ANGLE= 53.8
      BOTT: SMAX=      166.55 SMIN=    -851.37 TMAX=      508.96 ANGLE=-36.7
        JOINT          1.87       49.78        1.29        11.77         6.62
          15        1072.10     1063.42        0.69         5.55         0.55
      TOP : SMAX=     1003.32 SMIN=    -126.37 TMAX=      564.84 ANGLE= 64.3
      BOTT: SMAX=      128.73 SMIN=    -993.19 TMAX=      560.96 ANGLE=-25.9
        JOINT          1.47       49.78        6.15        35.61         7.50
          25        2365.06     2359.80        6.85         5.19         0.18
      TOP : SMAX=     2499.31 SMIN=     296.47 TMAX=     1101.42 ANGLE= 76.5
      BOTT: SMAX=     -283.11 SMIN=   -2488.58 TMAX=     1102.74 ANGLE=-13.5
        JOINT          1.47       40.93        5.07        29.55         6.43
```

```
          23       1975.71     1963.92        7.75         8.73         0.70
      TOP : SMAX=    2084.73 SMIN=     239.95 TMAX=     922.39 ANGLE= 76.1
      BOTT: SMAX=    -224.98 SMIN=   -2066.73 TMAX=     920.87 ANGLE=-13.8
             3         0.11        0.26        0.03         0.10         0.01
                       8.00        9.21        6.15         0.33         0.99
                       9.17       10.58
      TOP : SMAX=       9.17 SMIN=       5.56 TMAX=       1.80 ANGLE= 36.2
      BOTT: SMAX=       4.40 SMIN=      -6.18 TMAX=       5.29 ANGLE=  1.4
        JOINT          0.19        0.28        0.11         0.22         0.04
          13          14.64       12.36        1.01         0.20         1.75
      TOP : SMAX=      16.72 SMIN=       6.19 TMAX=       5.26 ANGLE= 63.1
      BOTT: SMAX=      -6.25 SMIN=     -14.24 TMAX=       3.99 ANGLE= -5.4
        JOINT          0.19        0.29        0.13         0.25         0.05
          15          16.63       14.80        0.18         0.27         1.01
      TOP : SMAX=      18.97 SMIN=       6.91 TMAX=       6.03 ANGLE= 64.7
      BOTT: SMAX=      -7.92 SMIN=     -17.08 TMAX=       4.58 ANGLE=-17.7
        JOINT          0.10        0.29        0.07         0.06         0.04
          25          16.77       11.51       13.30         0.84         0.25
      TOP : SMAX=      18.50 SMIN=       4.31 TMAX=       7.10 ANGLE= 12.1
      BOTT: SMAX=       9.19 SMIN=      -3.71 TMAX=       6.45 ANGLE=-11.0
        JOINT          0.10        0.28        0.06         0.07         0.07
          23          17.62       12.60       12.32         0.38         0.98
      TOP : SMAX=      18.92 SMIN=       2.98 TMAX=       7.97 ANGLE= 22.2
      BOTT: SMAX=       8.96 SMIN=      -5.46 TMAX=       7.21 ANGLE=-15.1
     STAAD SPACE                                                -- PAGE NO.
53
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX           MY          MXY
                      VONT        VONB         SX           SY          SXY
                      TRESCAT     TRESCAB
    56      1         43.80        2.76        0.16         0.34         4.30
                     521.26      472.46        5.53         0.43        14.09
                     601.46      545.31
      TOP : SMAX=     320.50 SMIN=    -280.96 TMAX=     300.73 ANGLE= 45.3
      BOTT: SMAX=     258.85 SMIN=    -286.46 TMAX=     272.66 ANGLE=-44.1
        JOINT         12.04        7.52        5.17         1.13         6.12
          15         802.64      744.85        4.93         1.88        17.17
      TOP : SMAX=     660.05 SMIN=    -233.43 TMAX=     446.74 ANGLE= 36.1
      BOTT: SMAX=     206.68 SMIN=    -619.69 TMAX=     413.18 ANGLE=-54.4
        JOINT         12.04        8.85        4.43         1.71         6.14
          17         785.45      722.99        5.29         1.22        18.19
      TOP : SMAX=     645.32 SMIN=    -229.24 TMAX=     437.28 ANGLE= 38.9
      BOTT: SMAX=     199.34 SMIN=    -602.41 TMAX=     400.88 ANGLE=-51.4
        JOINT         75.82        8.85       43.50         8.62        14.69
          27        3177.37     3130.97       15.92         1.68        11.25
      TOP : SMAX=    3278.60 SMIN=     213.19 TMAX=    1532.70 ANGLE= 20.1
      BOTT: SMAX=    -220.98 SMIN=   -3235.60 TMAX=    1507.31 ANGLE=-70.0
        JOINT         75.82        7.52       43.56         7.61        14.74
          25        3194.55     3166.08        6.09         1.11        10.24
      TOP : SMAX=    3267.53 SMIN=     151.34 TMAX=    1558.09 ANGLE= 19.8
      BOTT: SMAX=    -160.98 SMIN=   -3243.50 TMAX=    1541.26 ANGLE=-70.4
             2         1.67       45.35        1.74        11.48         0.10
                     720.11      708.83        3.82         7.14         0.08
                     772.57      758.29
      TOP : SMAX=     772.57 SMIN=     120.00 TMAX=     326.29 ANGLE= 89.4
      BOTT: SMAX=    -112.36 SMIN=    -758.29 TMAX=     322.97 ANGLE= -0.6
```

```
        JOINT         1.87       49.78        1.29       11.77        6.62
          15       1072.10     1063.42        0.69        5.55        0.55
      TOP : SMAX=    1003.32 SMIN=    -126.37 TMAX=     564.84 ANGLE= 64.3
      BOTT: SMAX=     128.73 SMIN=    -993.19 TMAX=     560.96 ANGLE=-25.9
        JOINT         1.87       40.93        2.96        7.46        7.31
          17        953.49      945.71        0.88        9.11        0.07
      TOP : SMAX=     863.92 SMIN=    -159.12 TMAX=     511.52 ANGLE= 53.8
      BOTT: SMAX=     166.55 SMIN=    -851.37 TMAX=     508.96 ANGLE=-36.7
        JOINT         1.47       40.93        5.07       29.55        6.43
          27       1975.71     1963.92        7.75        8.73        0.70
      TOP : SMAX=    2084.73 SMIN=     239.95 TMAX=     922.39 ANGLE= 76.1
      BOTT: SMAX=    -224.98 SMIN=   -2066.73 TMAX=     920.87 ANGLE=-13.8
        JOINT         1.47       49.78        6.15       35.61        7.50
          25       2365.06     2359.80        6.85        5.19        0.18
      TOP : SMAX=    2499.31 SMIN=     296.47 TMAX=    1101.42 ANGLE= 76.5
      BOTT: SMAX=    -283.11 SMIN=   -2488.58 TMAX=    1102.74 ANGLE=-13.5
            3         0.11        0.26        0.03        0.10        0.01
                      8.00        9.21        6.15        0.33        0.99
                      9.17       10.58
      TOP : SMAX=       9.17 SMIN=       5.56 TMAX=       1.80 ANGLE= 36.2
      BOTT: SMAX=       4.40 SMIN=      -6.18 TMAX=       5.29 ANGLE=  1.4
     STAAD SPACE                                             -- PAGE NO.
54
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY         MX          MY          MXY
                      VONT       VONB        SX          SY          SXY
                      TRESCAT    TRESCAB
        JOINT         0.19        0.29        0.13        0.25        0.05
          15         16.63       14.80        0.18        0.27        1.01
      TOP : SMAX=      18.97 SMIN=       6.91 TMAX=       6.03 ANGLE= 64.7
      BOTT: SMAX=      -7.92 SMIN=     -17.08 TMAX=       4.58 ANGLE=-17.7
        JOINT         0.19        0.28        0.11        0.22        0.04
          17         14.64       12.36        1.01        0.20        1.75
      TOP : SMAX=      16.72 SMIN=       6.19 TMAX=       5.26 ANGLE= 63.1
      BOTT: SMAX=      -6.25 SMIN=     -14.24 TMAX=       3.99 ANGLE= -5.4
        JOINT         0.10        0.28        0.06        0.07        0.07
          27         17.62       12.60       12.32        0.38        0.98
      TOP : SMAX=      18.92 SMIN=       2.98 TMAX=       7.97 ANGLE= 22.2
      BOTT: SMAX=       8.96 SMIN=      -5.46 TMAX=       7.21 ANGLE=-15.1
        JOINT         0.10        0.29        0.07        0.06        0.04
          25         16.77       11.51       13.30        0.84        0.25
      TOP : SMAX=      18.50 SMIN=       4.31 TMAX=       7.10 ANGLE= 12.1
      BOTT: SMAX=       9.19 SMIN=      -3.71 TMAX=       6.45 ANGLE=-11.0
    57      1        43.06       13.99        1.24        5.15        3.66
                    543.11      506.30       12.17        3.54       13.21
                    572.96      534.50
      TOP : SMAX=     507.29 SMIN=     -65.67 TMAX=     286.48 ANGLE= 58.1
      BOTT: SMAX=      62.15 SMIN=    -472.35 TMAX=     267.25 ANGLE=-29.9
        JOINT        12.54        6.92        7.10        0.98        5.17
          17        756.83      730.91        7.83       15.59        8.57
      TOP : SMAX=     686.75 SMIN=    -124.68 TMAX=     405.72 ANGLE= 30.2
      BOTT: SMAX=     137.29 SMIN=    -652.53 TMAX=     394.91 ANGLE=-60.9
        JOINT        12.54       21.19        7.06        3.30        6.99
          19        923.40      886.20       23.06       11.02        6.21
      TOP : SMAX=     852.57 SMIN=    -128.24 TMAX=     490.41 ANGLE= 37.2
      BOTT: SMAX=     147.20 SMIN=    -803.38 TMAX=     475.29 ANGLE=-52.3
```

```
        JOINT        76.73       21.19       45.29       18.73       11.74
          29       2996.06     2917.13       32.15        8.74       17.88
      TOP : SMAX=    3356.45 SMIN=     952.23 TMAX=    1202.11 ANGLE= 20.9
      BOTT: SMAX=    -952.56 SMIN=   -3274.34 TMAX=    1160.89 ANGLE=-69.4
        JOINT        76.73        6.92       44.34        5.64       14.24
          27       3288.69     3183.61       47.35       17.87       20.59
      TOP : SMAX=    3324.36 SMIN=      72.55 TMAX=    1625.91 ANGLE= 18.3
      BOTT: SMAX=     -55.47 SMIN=   -3210.98 TMAX=    1577.75 ANGLE=-72.0
            2         8.55       67.56        2.79       16.20        1.23
                   1016.68     1003.64        2.14        7.40        0.14
                   1094.82     1080.04
      TOP : SMAX=    1094.82 SMIN=     180.44 TMAX=     457.19 ANGLE= 84.8
      BOTT: SMAX=    -176.13 SMIN=   -1080.04 TMAX=     451.96 ANGLE= -5.2
        JOINT         7.66       39.89        3.33        4.71       11.25
          17       1333.47     1324.56        1.51        3.73        2.25
      TOP : SMAX=    1024.67 SMIN=    -482.99 TMAX=     753.83 ANGLE= 46.8
      BOTT: SMAX=     483.60 SMIN=   -1014.80 TMAX=     749.20 ANGLE=-43.3
     STAAD SPACE                                              -- PAGE NO.
55
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
        JOINT         7.66       95.23        1.11       26.38        8.29
          19       1995.38     1946.49       19.14       25.55       11.83
      TOP : SMAX=    1955.25 SMIN=     -77.98 TMAX=    1016.62 ANGLE= 73.1
      BOTT: SMAX=     104.33 SMIN=   -1892.22 TMAX=     998.28 ANGLE=-16.4
        JOINT         9.45       95.23       10.22       69.49       13.71
          29       4628.34     4596.05        5.26       17.88        2.31
      TOP : SMAX=    4851.88 SMIN=     485.32 TMAX=    2183.28 ANGLE= 77.6
      BOTT: SMAX=    -475.58 SMIN=   -4815.35 TMAX=    2169.89 ANGLE=-12.4
        JOINT         9.45       39.89        3.15       26.41        5.83
          27       1806.20     1786.74       21.66       11.02       11.81
      TOP : SMAX=    1869.30 SMIN=     133.60 TMAX=     867.85 ANGLE= 76.3
      BOTT: SMAX=    -102.00 SMIN=   -1835.55 TMAX=     866.78 ANGLE=-12.9
            3         0.23        0.27        0.02        0.09        0.03
                      8.59        7.05        2.51        0.05        1.88
                      8.98        7.67
      TOP : SMAX=       8.98 SMIN=       0.83 TMAX=       4.07 ANGLE= 54.9
      BOTT: SMAX=       1.49 SMIN=      -6.18 TMAX=       3.84 ANGLE= -0.5
        JOINT         0.29        0.26        0.18        0.22        0.08
          17         21.18       12.37        1.07        1.47        3.71
      TOP : SMAX=      23.41 SMIN=       5.58 TMAX=       8.91 ANGLE= 50.4
      BOTT: SMAX=     -10.11 SMIN=     -13.80 TMAX=       1.85 ANGLE=-23.3
        JOINT         0.29        0.45        0.12        0.35        0.04
          19         24.51       19.80        4.21        1.86        4.25
      TOP : SMAX=      27.85 SMIN=       9.57 TMAX=       9.14 ANGLE= 66.6
      BOTT: SMAX=      -3.84 SMIN=     -21.44 TMAX=       8.80 ANGLE=  6.0
        JOINT         0.25        0.45        0.14        0.15        0.09
          29         16.28       12.51        3.96        1.43        0.11
      TOP : SMAX=      18.56 SMIN=       6.67 TMAX=       5.95 ANGLE= 40.7
      BOTT: SMAX=      -1.32 SMIN=     -13.11 TMAX=       5.90 ANGLE=-37.0
        JOINT         0.25        0.26        0.13        0.08        0.08
          27         19.00        9.30        9.23        1.90        0.51
      TOP : SMAX=      20.82 SMIN=       4.41 TMAX=       8.20 ANGLE= 23.6
      BOTT: SMAX=       3.81 SMIN=      -6.78 TMAX=       5.30 ANGLE=-35.3
```

```
   101      1         9.51        2.69        1.17        1.62        0.48
                    127.31       97.35        8.74        4.65       12.09
                    145.42      112.30
      TOP : SMAX=     145.42 SMIN=      54.08 TMAX=      45.67 ANGLE= 53.2
      BOTT: SMAX=     -60.44 SMIN=    -112.30 TMAX=      25.93 ANGLE=-24.6
        JOINT        15.30        5.84       10.31        5.90        2.31
          31        674.55      633.77       13.47        7.85       13.73
      TOP : SMAX=     775.63 SMIN=     326.01 TMAX=     224.81 ANGLE= 24.1
      BOTT: SMAX=    -328.42 SMIN=    -730.58 TMAX=     201.08 ANGLE=-67.9
        JOINT        15.30        0.48        7.24        0.10        2.61
          32        589.27      543.00       16.85        6.04       12.34
      TOP : SMAX=     562.65 SMIN=     -50.06 TMAX=     306.35 ANGLE= 18.7
      BOTT: SMAX=      49.66 SMIN=    -516.46 TMAX=     283.06 ANGLE=-72.6
     STAAD SPACE                                              -- PAGE NO.
56
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY         MX          MY          MXY
                      VONT        VONB        SX          SY          SXY
                      TRESCAT     TRESCAB
        JOINT         5.03        0.48        2.99        0.75        1.37
          37        254.03      224.55        4.74        3.36       10.50
      TOP : SMAX=     255.22 SMIN=       2.41 TMAX=     126.41 ANGLE= 26.7
      BOTT: SMAX=     -11.41 SMIN=    -230.03 TMAX=     109.31 ANGLE=-66.3
        JOINT         5.03        5.84        2.24        2.17        1.66
          36        264.17      220.10        4.62       10.58       12.74
      TOP : SMAX=     278.27 SMIN=      30.93 TMAX=     123.67 ANGLE= 45.1
      BOTT: SMAX=     -41.07 SMIN=    -237.74 TMAX=      98.34 ANGLE=-46.6
            2         0.16       14.37        0.56        3.10        0.26
                    196.43      190.19        6.55        4.54        3.48
                    213.73      203.20
      TOP : SMAX=     213.73 SMIN=      41.12 TMAX=      86.31 ANGLE= 83.0
      BOTT: SMAX=     -29.46 SMIN=    -203.20 TMAX=      86.87 ANGLE= -4.6
        JOINT         1.11       20.93        0.89       15.05        2.47
          31       1019.67     1011.64        9.90        7.58        2.17
      TOP : SMAX=    1039.46 SMIN=      40.79 TMAX=     499.33 ANGLE= 80.2
      BOTT: SMAX=     -22.56 SMIN=   -1022.73 TMAX=     500.09 ANGLE= -9.5
        JOINT         1.11        7.81        1.19        4.73        1.59
          32        347.92      331.48       27.30        7.45        9.06
      TOP : SMAX=     372.79 SMIN=      56.71 TMAX=     158.04 ANGLE= 66.6
      BOTT: SMAX=     -19.30 SMIN=    -340.71 TMAX=     160.71 ANGLE=-18.6
        JOINT         0.85        7.81        1.67        1.18        1.94
          37        254.02      236.59        3.21        1.53        4.81
      TOP : SMAX=     232.82 SMIN=     -38.09 TMAX=     135.46 ANGLE= 41.4
      BOTT: SMAX=      33.06 SMIN=    -218.32 TMAX=     125.69 ANGLE=-48.5
        JOINT         0.85       20.93        0.86        6.21        2.11
          36        483.55      435.61       14.20       16.52       16.01
      TOP : SMAX=     489.76 SMIN=      12.67 TMAX=     238.55 ANGLE= 69.4
      BOTT: SMAX=      -3.60 SMIN=    -437.39 TMAX=     216.90 ANGLE=-17.6
            3         0.10        0.28        0.04        0.17        0.08
                     14.02       12.67        1.20        0.05        0.81
                     14.25       12.82
      TOP : SMAX=      14.25 SMIN=       0.47 TMAX=       6.89 ANGLE= 61.2
      BOTT: SMAX=       0.30 SMIN=     -12.52 TMAX=       6.41 ANGLE=-20.5
        JOINT         0.22        0.59        0.18        0.16        0.12
          31         18.54       16.53        1.90        0.71        0.08
      TOP : SMAX=      20.37 SMIN=       4.47 TMAX=       7.95 ANGLE= 40.1
```

```
       BOTT: SMAX=      -2.13 SMIN=     -17.50 TMAX=       7.68 ANGLE=-45.7
         JOINT        0.22         0.41        0.24         0.14        0.10
           32        21.56        15.38        4.37         0.96        0.33
       TOP : SMAX=      24.01 SMIN=       6.31 TMAX=       8.85 ANGLE= 27.8
       BOTT: SMAX=      -3.00 SMIN=     -16.66 TMAX=       6.83 ANGLE=-51.7
         JOINT        0.38         0.41        0.29         0.37        0.06
           37        25.15        22.16        0.53         0.73        1.65
       TOP : SMAX=      28.97 SMIN=      16.22 TMAX=       6.37 ANGLE= 57.2
       BOTT: SMAX=     -17.84 SMIN=     -24.81 TMAX=       3.48 ANGLE=-22.9
         JOINT        0.38         0.59        0.18         0.55        0.05
           36        34.32        32.14        1.99         0.94        1.94
       TOP : SMAX=      38.85 SMIN=      12.65 TMAX=      13.10 ANGLE= 77.8
       BOTT: SMAX=      -9.76 SMIN=     -35.88 TMAX=      13.06 ANGLE= -3.4
      STAAD SPACE                                               -- PAGE NO.
57
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    102      1         8.05        0.46        0.13        0.10        0.70
                     108.25       54.75        4.24        2.07       15.32
                     124.38       63.01
       TOP : SMAX=      73.00 SMIN=     -51.37 TMAX=      62.19 ANGLE= 43.9
       BOTT: SMAX=      27.00 SMIN=     -36.01 TMAX=      31.50 ANGLE=-45.2
         JOINT       11.30         1.87        6.11         1.39        2.23
           32       470.42       431.74        5.44         2.31       15.72
       TOP : SMAX=     482.56 SMIN=      25.29 TMAX=     228.63 ANGLE= 23.0
       BOTT: SMAX=     -41.39 SMIN=    -450.95 TMAX=     204.78 ANGLE=-69.7
         JOINT       11.30         1.71        6.47         1.30        2.41
           33       502.58       464.68        5.00         1.72       15.45
       TOP : SMAX=     509.93 SMIN=      15.03 TMAX=     247.45 ANGLE= 22.7
       BOTT: SMAX=     -31.77 SMIN=    -479.75 TMAX=     223.99 ANGLE=-69.8
         JOINT        7.33         1.71        4.10         0.93        1.68
           38       333.98       297.04        3.71         1.87       15.07
       TOP : SMAX=     336.28 SMIN=       4.64 TMAX=     165.82 ANGLE= 25.0
       BOTT: SMAX=     -22.22 SMIN=    -307.53 TMAX=     142.65 ANGLE=-68.6
         JOINT        7.33         1.87        3.65         1.38        1.80
           37       318.45       276.51        4.20         2.58       15.27
       TOP : SMAX=     326.03 SMIN=      15.75 TMAX=     155.14 ANGLE= 30.2
       BOTT: SMAX=     -35.61 SMIN=    -292.59 TMAX=     128.49 ANGLE=-62.8
             2         0.10        9.37        0.56         2.11        0.10
                     129.81       123.34        8.14         5.51        0.32
                     146.34       135.22
       TOP : SMAX=     146.34 SMIN=      45.09 TMAX=      50.63 ANGLE= 86.1
       BOTT: SMAX=     -28.91 SMIN=    -135.22 TMAX=      53.16 ANGLE= -3.4
         JOINT        0.13         8.02        1.42         5.51        1.31
           32       366.30       361.71       21.16         7.39        1.34
       TOP : SMAX=     402.35 SMIN=      88.20 TMAX=     157.07 ANGLE= 72.7
       BOTT: SMAX=     -49.52 SMIN=    -383.92 TMAX=     167.20 ANGLE=-15.5
         JOINT        0.13        10.71        1.23         7.44        1.53
           33       488.78       498.68       21.52         1.83        0.98
       TOP : SMAX=     523.04 SMIN=      77.87 TMAX=     222.58 ANGLE= 76.2
       BOTT: SMAX=     -37.74 SMIN=    -516.48 TMAX=     239.37 ANGLE=-12.5
         JOINT        0.08        10.71        0.24         2.87        1.51
           38       255.80       251.95        4.87         3.62        0.70
       TOP : SMAX=     241.86 SMIN=     -25.91 TMAX=     133.88 ANGLE= 65.3
```

```
      BOTT: SMAX=      34.17 SMIN=    -233.12 TMAX=     133.65 ANGLE=-24.3
        JOINT        0.08        8.02        0.18        1.66        1.34
          37       192.99      180.49        5.23        9.20        1.62
      TOP : SMAX=     172.60 SMIN=     -35.77 TMAX=     104.19 ANGLE= 59.8
      BOTT: SMAX=      45.45 SMIN=    -153.42 TMAX=      99.43 ANGLE=-30.8
           3         0.17        0.37        0.05        0.19        0.02
                    11.55       12.30        2.88        0.17        0.44
                    13.29       12.47
      TOP : SMAX=      13.29 SMIN=       5.65 TMAX=       3.82 ANGLE= 73.9
      BOTT: SMAX=      -0.36 SMIN=     -12.47 TMAX=       6.06 ANGLE= -5.5
     STAAD SPACE                                             -- PAGE NO.
58
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
        JOINT        0.20        0.40        0.15        0.06        0.09
          32        17.63       10.11        5.68        0.20        0.43
      TOP : SMAX=      18.34 SMIN=       1.54 TMAX=       8.40 ANGLE= 23.7
      BOTT: SMAX=       1.28 SMIN=      -9.41 TMAX=       5.34 ANGLE=-46.0
        JOINT        0.20        0.43        0.14        0.08        0.06
          33        15.40        8.52        6.14        0.40        0.09
      TOP : SMAX=      17.10 SMIN=       4.32 TMAX=       6.39 ANGLE= 21.4
      BOTT: SMAX=       0.12 SMIN=      -8.46 TMAX=       4.29 ANGLE=-37.9
        JOINT        0.30        0.43        0.22        0.42        0.06
          38        25.52       24.87        0.26        0.15        0.45
      TOP : SMAX=      29.45 SMIN=      13.86 TMAX=       7.80 ANGLE= 73.9
      BOTT: SMAX=     -13.79 SMIN=     -28.71 TMAX=       7.46 ANGLE=-12.9
        JOINT        0.30        0.40        0.18        0.36        0.09
          37        24.24       22.87        0.45        0.09        0.78
      TOP : SMAX=      27.47 SMIN=       9.10 TMAX=       9.18 ANGLE= 65.2
      BOTT: SMAX=      -9.41 SMIN=     -26.08 TMAX=       8.33 ANGLE=-20.3
    103      1       8.05        0.46        0.13        0.10        0.70
                   108.25       54.75        4.24        2.07       15.32
                   124.38       63.01
      TOP : SMAX=      73.00 SMIN=     -51.37 TMAX=      62.19 ANGLE= 43.9
      BOTT: SMAX=      27.00 SMIN=     -36.01 TMAX=      31.50 ANGLE=-45.2
        JOINT       11.30        1.71        6.47        1.30        2.41
          33       502.58      464.68        5.00        1.72       15.45
      TOP : SMAX=     509.93 SMIN=      15.03 TMAX=     247.45 ANGLE= 22.7
      BOTT: SMAX=     -31.77 SMIN=    -479.75 TMAX=     223.99 ANGLE=-69.8
        JOINT       11.30        1.87        6.11        1.39        2.23
          34       470.42      431.74        5.44        2.31       15.72
      TOP : SMAX=     482.56 SMIN=      25.29 TMAX=     228.63 ANGLE= 23.0
      BOTT: SMAX=     -41.39 SMIN=    -450.95 TMAX=     204.78 ANGLE=-69.7
        JOINT        7.33        1.87        3.65        1.38        1.80
          39       318.45      276.51        4.20        2.58       15.27
      TOP : SMAX=     326.03 SMIN=      15.75 TMAX=     155.14 ANGLE= 30.2
      BOTT: SMAX=     -35.61 SMIN=    -292.59 TMAX=     128.49 ANGLE=-62.8
        JOINT        7.33        1.71        4.10        0.93        1.68
          38       333.98      297.04        3.71        1.87       15.07
      TOP : SMAX=     336.28 SMIN=       4.64 TMAX=     165.82 ANGLE= 25.0
      BOTT: SMAX=     -22.22 SMIN=    -307.53 TMAX=     142.65 ANGLE=-68.6
           2         0.10        9.37        0.56        2.11        0.10
                   129.81      123.34        8.14        5.51        0.32
                   146.34      135.22
```

```
       TOP : SMAX=     146.34 SMIN=      45.09 TMAX=      50.63 ANGLE= 86.1
       BOTT: SMAX=     -28.91 SMIN=    -135.22 TMAX=      53.16 ANGLE= -3.4
         JOINT         0.13        10.71        1.23        7.44         1.53
           33        488.78       498.68       21.52        1.83         0.98
       TOP : SMAX=     523.04 SMIN=      77.87 TMAX=     222.58 ANGLE= 76.2
       BOTT: SMAX=     -37.74 SMIN=    -516.48 TMAX=     239.37 ANGLE=-12.5
      STAAD SPACE                                                -- PAGE NO.
59
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT         0.13         8.02        1.42        5.51         1.31
           34        366.30       361.71       21.16        7.39         1.34
       TOP : SMAX=     402.35 SMIN=      88.20 TMAX=     157.07 ANGLE= 72.7
       BOTT: SMAX=     -49.52 SMIN=    -383.92 TMAX=     167.20 ANGLE=-15.5
         JOINT         0.08         8.02        0.18        1.66         1.34
           39        192.99       180.49        5.23        9.20         1.62
       TOP : SMAX=     172.60 SMIN=     -35.77 TMAX=     104.19 ANGLE= 59.8
       BOTT: SMAX=      45.45 SMIN=    -153.42 TMAX=      99.43 ANGLE=-30.8
         JOINT         0.08        10.71        0.24        2.87         1.51
           38        255.80       251.95        4.87        3.62         0.70
       TOP : SMAX=     241.86 SMIN=     -25.91 TMAX=     133.88 ANGLE= 65.3
       BOTT: SMAX=      34.17 SMIN=    -233.12 TMAX=     133.65 ANGLE=-24.3
             3         0.17         0.37        0.05        0.19         0.02
                      11.55        12.30        2.88        0.17         0.44
                      13.29        12.47
       TOP : SMAX=      13.29 SMIN=       5.65 TMAX=       3.82 ANGLE= 73.9
       BOTT: SMAX=      -0.36 SMIN=     -12.47 TMAX=       6.06 ANGLE= -5.5
         JOINT         0.20         0.43        0.14        0.08         0.06
           33         15.40         8.52        6.14        0.40         0.09
       TOP : SMAX=      17.10 SMIN=       4.32 TMAX=       6.39 ANGLE= 21.4
       BOTT: SMAX=       0.12 SMIN=      -8.46 TMAX=       4.29 ANGLE=-37.9
         JOINT         0.20         0.40        0.15        0.06         0.09
           34         17.63        10.11        5.68        0.20         0.43
       TOP : SMAX=      18.34 SMIN=       1.54 TMAX=       8.40 ANGLE= 23.7
       BOTT: SMAX=       1.28 SMIN=      -9.41 TMAX=       5.34 ANGLE=-46.0
         JOINT         0.30         0.40        0.18        0.36         0.09
           39         24.24        22.87        0.45        0.09         0.78
       TOP : SMAX=      27.47 SMIN=       9.10 TMAX=       9.18 ANGLE= 65.2
       BOTT: SMAX=      -9.41 SMIN=     -26.08 TMAX=       8.33 ANGLE=-20.3
         JOINT         0.30         0.43        0.22        0.42         0.06
           38         25.52        24.87        0.26        0.15         0.45
       TOP : SMAX=      29.45 SMIN=      13.86 TMAX=       7.80 ANGLE= 73.9
       BOTT: SMAX=     -13.79 SMIN=     -28.71 TMAX=       7.46 ANGLE=-12.9
    104      1         9.51         2.69        1.17        1.62         0.48
                     127.31        97.35        8.74        4.65        12.09
                     145.42       112.30
       TOP : SMAX=     145.42 SMIN=      54.08 TMAX=      45.67 ANGLE= 53.2
       BOTT: SMAX=     -60.44 SMIN=    -112.30 TMAX=      25.93 ANGLE=-24.6
         JOINT        15.30         0.48        7.24        0.10         2.61
           34        589.27       543.00       16.85        6.04        12.34
       TOP : SMAX=     562.65 SMIN=     -50.06 TMAX=     306.35 ANGLE= 18.7
       BOTT: SMAX=      49.66 SMIN=    -516.46 TMAX=     283.06 ANGLE=-72.6
         JOINT        15.30         5.84       10.31        5.90         2.31
           35        674.55       633.77       13.47        7.85        13.73
```

```
       TOP : SMAX=     775.63 SMIN=     326.01 TMAX=     224.81 ANGLE= 24.1
       BOTT: SMAX=    -328.42 SMIN=    -730.58 TMAX=     201.08 ANGLE=-67.9
      STAAD SPACE                                              -- PAGE NO.
60
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT         5.03        5.84        2.24        2.17        1.66
           40        264.17      220.10        4.62       10.58       12.74
       TOP : SMAX=     278.27 SMIN=      30.93 TMAX=     123.67 ANGLE= 45.1
       BOTT: SMAX=     -41.07 SMIN=    -237.74 TMAX=      98.34 ANGLE=-46.6
         JOINT         5.03        0.48        2.99        0.75        1.37
           39        254.03      224.55        4.74        3.36       10.50
       TOP : SMAX=     255.22 SMIN=       2.41 TMAX=     126.41 ANGLE= 26.7
       BOTT: SMAX=     -11.41 SMIN=    -230.03 TMAX=     109.31 ANGLE=-66.3
             2         0.16       14.37        0.56        3.10        0.26
                     196.43      190.19        6.55        4.54        3.48
                     213.73      203.20
       TOP : SMAX=     213.73 SMIN=      41.12 TMAX=      86.31 ANGLE= 83.0
       BOTT: SMAX=     -29.46 SMIN=    -203.20 TMAX=      86.87 ANGLE= -4.6
         JOINT         1.11        7.81        1.19        4.73        1.59
           34        347.92      331.48       27.30        7.45        9.06
       TOP : SMAX=     372.79 SMIN=      56.71 TMAX=     158.04 ANGLE= 66.6
       BOTT: SMAX=     -19.30 SMIN=    -340.71 TMAX=     160.71 ANGLE=-18.6
         JOINT         1.11       20.93        0.89       15.05        2.47
           35       1019.67     1011.64        9.90        7.58        2.17
       TOP : SMAX=    1039.46 SMIN=      40.79 TMAX=     499.33 ANGLE= 80.2
       BOTT: SMAX=     -22.56 SMIN=   -1022.73 TMAX=     500.09 ANGLE= -9.5
         JOINT         0.85       20.93        0.86        6.21        2.11
           40        483.55      435.61       14.20       16.52       16.01
       TOP : SMAX=     489.76 SMIN=      12.67 TMAX=     238.55 ANGLE= 69.4
       BOTT: SMAX=      -3.60 SMIN=    -437.39 TMAX=     216.90 ANGLE=-17.6
         JOINT         0.85        7.81        1.67        1.18        1.94
           39        254.02      236.59        3.21        1.53        4.81
       TOP : SMAX=     232.82 SMIN=     -38.09 TMAX=     135.46 ANGLE= 41.4
       BOTT: SMAX=      33.06 SMIN=    -218.32 TMAX=     125.69 ANGLE=-48.5
             3         0.10        0.28        0.04        0.17        0.08
                      14.02       12.67        1.20        0.05        0.81
                      14.25       12.82
       TOP : SMAX=      14.25 SMIN=       0.47 TMAX=       6.89 ANGLE= 61.2
       BOTT: SMAX=       0.30 SMIN=     -12.52 TMAX=       6.41 ANGLE=-20.5
         JOINT         0.22        0.41        0.24        0.14        0.10
           34         21.56       15.38        4.37        0.96        0.33
       TOP : SMAX=      24.01 SMIN=       6.31 TMAX=       8.85 ANGLE= 27.8
       BOTT: SMAX=      -3.00 SMIN=     -16.66 TMAX=       6.83 ANGLE=-51.7
         JOINT         0.22        0.59        0.18        0.16        0.12
           35         18.54       16.53        1.90        0.71        0.08
       TOP : SMAX=      20.37 SMIN=       4.47 TMAX=       7.95 ANGLE= 40.1
       BOTT: SMAX=      -2.13 SMIN=     -17.50 TMAX=       7.68 ANGLE=-45.7
         JOINT         0.38        0.59        0.18        0.55        0.05
           40         34.32       32.14        1.99        0.94        1.94
       TOP : SMAX=      38.85 SMIN=      12.65 TMAX=      13.10 ANGLE= 77.8
       BOTT: SMAX=      -9.76 SMIN=     -35.88 TMAX=      13.06 ANGLE= -3.4
         JOINT         0.38        0.41        0.29        0.37        0.06
           39         25.15       22.16        0.53        0.73        1.65
```

```
       TOP : SMAX=      28.97 SMIN=      16.22 TMAX=       6.37 ANGLE= 57.2
       BOTT: SMAX=     -17.84 SMIN=     -24.81 TMAX=       3.48 ANGLE=-22.9
      STAAD SPACE                                              -- PAGE NO.
61
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    105      1        27.56        7.22        0.94        3.43        2.66
                     394.39      344.46        8.03        3.57       16.78
                     420.47      363.48
       TOP : SMAX=     361.72 SMIN=     -58.74 TMAX=     210.23 ANGLE= 56.3
       BOTT: SMAX=      41.85 SMIN=    -321.63 TMAX=     181.74 ANGLE=-31.0
         JOINT         3.97       11.61        5.62        1.25        3.78
           36        584.73      523.89        6.26        8.73       20.72
       TOP : SMAX=     545.28 SMIN=     -72.20 TMAX=     308.74 ANGLE= 31.1
       BOTT: SMAX=      52.55 SMIN=    -495.64 TMAX=     274.09 ANGLE=-61.2
         JOINT         3.97        2.91        6.03        0.64        2.83
           37        519.87      486.99        3.35        8.82       14.85
       TOP : SMAX=     497.97 SMIN=     -41.34 TMAX=     269.65 ANGLE= 24.5
       BOTT: SMAX=      35.82 SMIN=    -468.10 TMAX=     251.96 ANGLE=-68.2
         JOINT        52.59        2.91       30.09        2.98        9.07
           42       2207.57     2156.93       22.18       13.99       13.35
       TOP : SMAX=    2218.43 SMIN=      21.88 TMAX=    1098.28 ANGLE= 17.1
       BOTT: SMAX=      -7.36 SMIN=   -2160.60 TMAX=    1076.62 ANGLE=-73.3
         JOINT        52.59       11.61       32.58       11.32        7.92
           41       2147.59     2087.75       19.37        3.54       18.79
       TOP : SMAX=    2376.12 SMIN=     573.50 TMAX=     901.31 ANGLE= 18.7
       BOTT: SMAX=    -585.73 SMIN=   -2318.06 TMAX=     866.16 ANGLE=-72.0
             2         2.67       25.76        1.15        6.86        0.44
                     434.06      420.55       15.22       12.23        0.88
                     471.82      447.06
       TOP : SMAX=     471.82 SMIN=      89.45 TMAX=     191.18 ANGLE= 85.5
       BOTT: SMAX=     -59.32 SMIN=    -447.06 TMAX=     193.87 ANGLE= -4.2
         JOINT         2.17       37.23        0.22        9.92        3.22
           36        768.02      737.12        3.85       16.25        3.44
       TOP : SMAX=     743.33 SMIN=     -47.20 TMAX=     395.27 ANGLE= 73.2
       BOTT: SMAX=      53.17 SMIN=    -709.09 TMAX=     381.13 ANGLE=-16.8
         JOINT         2.17       14.28        0.71        0.62        4.16
           37        490.35      475.64        0.51       14.08        3.95
       TOP : SMAX=     333.38 SMIN=    -229.66 TMAX=     281.52 ANGLE= 45.4
       BOTT: SMAX=     236.50 SMIN=    -311.03 TMAX=     273.76 ANGLE=-46.0
         JOINT         3.19       14.28        1.36       10.21        2.35
           42        692.33      699.03       33.99        8.36        1.80
       TOP : SMAX=     730.19 SMIN=      83.26 TMAX=     323.47 ANGLE= 75.4
       BOTT: SMAX=     -19.95 SMIN=    -708.79 TMAX=     344.42 ANGLE=-13.3
         JOINT         3.19       37.23        3.76       27.76        5.04
           41       1835.84     1832.66       30.93       10.46        5.71
       TOP : SMAX=    1932.20 SMIN=     210.93 TMAX=     860.63 ANGLE= 78.3
       BOTT: SMAX=    -155.09 SMIN=   -1905.27 TMAX=     875.09 ANGLE=-11.1
             3         0.26        0.34        0.03        0.16        0.06
                      12.25       11.15        1.15        0.08        0.85
                      12.79       11.27
       TOP : SMAX=      12.79 SMIN=       1.17 TMAX=       5.81 ANGLE= 64.5
       BOTT: SMAX=      -0.25 SMIN=     -11.27 TMAX=       5.51 ANGLE=-15.4
      STAAD SPACE                                              -- PAGE NO.
```

```
62
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
         JOINT         0.39        0.59        0.15        0.51        0.04
           36         31.85       30.04        1.93        0.96        1.97
       TOP : SMAX=      35.85 SMIN=      10.80 TMAX=      12.52 ANGLE= 79.7
       BOTT: SMAX=      -7.73 SMIN=     -33.15 TMAX=      12.71 ANGLE= -1.0
         JOINT         0.39        0.39        0.26        0.35        0.10
           37         26.04       22.19        0.59        0.66        1.67
       TOP : SMAX=      29.90 SMIN=      12.26 TMAX=       8.82 ANGLE= 54.5
       BOTT: SMAX=     -14.10 SMIN=     -25.57 TMAX=       5.74 ANGLE=-30.3
         JOINT         0.22        0.39        0.13        0.11        0.11
           42         17.78       13.65        4.23        0.90        0.35
       TOP : SMAX=      19.00 SMIN=       2.76 TMAX=       8.12 ANGLE= 36.7
       BOTT: SMAX=       1.39 SMIN=     -12.90 TMAX=       7.15 ANGLE=-41.0
         JOINT         0.22        0.59        0.11        0.16        0.12
           41         17.45       16.09        1.78        0.71        0.12
       TOP : SMAX=      18.51 SMIN=       2.38 TMAX=       8.07 ANGLE= 49.4
       BOTT: SMAX=       0.11 SMIN=     -16.03 TMAX=       8.07 ANGLE=-36.7
    106      1        28.30        0.90        0.14        0.23        2.88
                     378.68      286.32        1.52        1.20       26.66
                     436.97      330.36
       TOP : SMAX=     232.21 SMIN=    -204.76 TMAX=     218.48 ANGLE= 45.4
       BOTT: SMAX=     154.18 SMIN=    -176.18 TMAX=     165.18 ANGLE=-44.4
         JOINT         6.75        4.75        3.46        0.42        3.97
           37        548.34      467.67        9.68        2.07       23.38
       TOP : SMAX=     442.08 SMIN=    -171.52 TMAX=     306.80 ANGLE= 35.0
       BOTT: SMAX=     136.90 SMIN=    -383.94 TMAX=     260.42 ANGLE=-56.0
         JOINT         6.75        5.33        4.09        0.92        3.75
           38        531.18      466.81        2.04        2.24       21.00
       TOP : SMAX=     459.87 SMIN=    -121.55 TMAX=     290.71 ANGLE= 34.3
       BOTT: SMAX=      87.27 SMIN=    -417.02 TMAX=     252.14 ANGLE=-57.4
         JOINT        49.99        5.33       28.92        5.14        9.73
           43       2139.49     2073.12        6.75        0.38       29.96
       TOP : SMAX=    2184.42 SMIN=      92.89 TMAX=    1045.76 ANGLE= 20.2
       BOTT: SMAX=    -128.62 SMIN=   -2134.43 TMAX=    1002.91 ANGLE=-71.0
         JOINT        49.99        4.75       28.82        5.30        9.50
           42       2117.08     2050.04        5.05        0.65       32.35
       TOP : SMAX=    2170.06 SMIN=     110.28 TMAX=    1029.89 ANGLE= 20.1
       BOTT: SMAX=    -148.63 SMIN=   -2120.31 TMAX=     985.84 ANGLE=-71.2
             2         0.25       17.04        0.78        5.02        0.04
                     314.44      309.76       16.76        8.63        0.32
                     343.06      325.79
       TOP : SMAX=     343.06 SMIN=      68.54 TMAX=     137.26 ANGLE= 89.4
       BOTT: SMAX=     -35.03 SMIN=    -325.79 TMAX=     145.38 ANGLE= -0.4
         JOINT         0.13       14.89        0.63        1.82        2.66
           37        328.48      322.24        5.50        7.81        0.63
       TOP : SMAX=     270.96 SMIN=     -94.37 TMAX=     182.67 ANGLE= 51.4
       BOTT: SMAX=     105.95 SMIN=    -255.92 TMAX=     180.94 ANGLE=-38.9
      STAAD SPACE                                                -- PAGE NO.
63
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
```

```
  ELEMENT  LOAD      SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
         JOINT       0.13        19.19        0.73        4.01        2.58
           38      391.43      381.11        4.65        8.46        0.65
       TOP : SMAX=     369.58 SMIN=     -40.54 TMAX=     205.06 ANGLE= 61.4
       BOTT: SMAX=      50.52 SMIN=    -353.33 TMAX=     201.92 ANGLE=-28.9
         JOINT        0.42       19.19        2.45       14.36        2.73
           43      941.13      942.31       28.03        9.46        0.01
       TOP : SMAX=    1007.30 SMIN=     150.47 TMAX=     428.41 ANGLE= 77.4
       BOTT: SMAX=     -96.12 SMIN=    -986.68 TMAX=     445.28 ANGLE=-12.1
         JOINT        0.42       14.89        2.02       11.52        2.51
           42      767.09      768.11       28.88        8.80        1.29
       TOP : SMAX=     820.16 SMIN=     120.37 TMAX=     349.90 ANGLE= 75.6
       BOTT: SMAX=     -66.14 SMIN=    -799.04 TMAX=     366.45 ANGLE=-13.4
             3        0.16        0.37        0.05        0.17        0.02
                     10.28       10.91        2.84        0.15        0.44
                     11.87       11.14
       TOP : SMAX=      11.87 SMIN=       5.71 TMAX=       3.08 ANGLE= 73.7
       BOTT: SMAX=      -0.47 SMIN=     -11.14 TMAX=       5.34 ANGLE= -4.2
         JOINT        0.30        0.39        0.17        0.34        0.05
           37        21.02       19.98        0.48        0.12        0.78
       TOP : SMAX=      24.17 SMIN=      10.21 TMAX=       6.98 ANGLE= 70.6
       BOTT: SMAX=     -10.15 SMIN=     -23.02 TMAX=       6.43 ANGLE=-12.9
         JOINT        0.30        0.42        0.21        0.39        0.07
           38        24.58       23.79        0.26        0.13        0.45
       TOP : SMAX=      28.30 SMIN=      12.38 TMAX=       7.96 ANGLE= 68.7
       BOTT: SMAX=     -12.46 SMIN=     -27.44 TMAX=       7.49 ANGLE=-18.4
         JOINT        0.09        0.42        0.09        0.08        0.06
           43        12.77        8.52        6.10        0.39        0.13
       TOP : SMAX=      14.19 SMIN=       3.65 TMAX=       5.27 ANGLE= 26.6
       BOTT: SMAX=       2.28 SMIN=      -7.14 TMAX=       4.71 ANGLE=-28.6
         JOINT        0.09        0.39        0.07        0.07        0.10
           42        14.76       11.53        5.64        0.17        0.43
       TOP : SMAX=      14.83 SMIN=       0.15 TMAX=       7.34 ANGLE= 34.3
       BOTT: SMAX=       4.91 SMIN=      -8.26 TMAX=       6.59 ANGLE=-32.5
    107      1       28.30        0.90        0.14        0.23        2.88
                    378.68      286.32        1.52        1.20       26.66
                    436.97      330.36
       TOP : SMAX=     232.21 SMIN=    -204.76 TMAX=     218.48 ANGLE= 45.4
       BOTT: SMAX=     154.18 SMIN=    -176.18 TMAX=     165.18 ANGLE=-44.4
         JOINT        6.75        5.33        4.09        0.92        3.75
           38       531.18      466.81        2.04        2.24       21.00
       TOP : SMAX=     459.87 SMIN=    -121.55 TMAX=     290.71 ANGLE= 34.3
       BOTT: SMAX=      87.27 SMIN=    -417.02 TMAX=     252.14 ANGLE=-57.4
         JOINT        6.75        4.75        3.46        0.42        3.97
           39       548.34      467.67        9.68        2.07       23.38
       TOP : SMAX=     442.08 SMIN=    -171.52 TMAX=     306.80 ANGLE= 35.0
       BOTT: SMAX=     136.90 SMIN=    -383.94 TMAX=     260.42 ANGLE=-56.0
      STAAD SPACE                                                -- PAGE NO.
64
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD      SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
         JOINT       49.99        4.75       28.82        5.30        9.50
```

```
          44       2117.08     2050.04        5.05         0.65        32.35
       TOP : SMAX=    2170.06 SMIN=     110.28 TMAX=    1029.89 ANGLE= 20.1
       BOTT: SMAX=    -148.63 SMIN=   -2120.31 TMAX=     985.84 ANGLE=-71.2
         JOINT        49.99        5.33       28.92         5.14         9.73
          43       2139.49     2073.12        6.75         0.38        29.96
       TOP : SMAX=    2184.42 SMIN=      92.89 TMAX=    1045.76 ANGLE= 20.2
       BOTT: SMAX=    -128.62 SMIN=   -2134.43 TMAX=    1002.91 ANGLE=-71.0
             2         0.25       17.04        0.78         5.02         0.04
                     314.44      309.76       16.76         8.63         0.32
                     343.06      325.79
       TOP : SMAX=     343.06 SMIN=      68.54 TMAX=     137.26 ANGLE= 89.4
       BOTT: SMAX=     -35.03 SMIN=    -325.79 TMAX=     145.38 ANGLE= -0.4
         JOINT         0.13       19.19        0.73         4.01         2.58
          38         391.43      381.11        4.65         8.46         0.65
       TOP : SMAX=     369.58 SMIN=     -40.54 TMAX=     205.06 ANGLE= 61.4
       BOTT: SMAX=      50.52 SMIN=    -353.33 TMAX=     201.92 ANGLE=-28.9
         JOINT         0.13       14.89        0.63         1.82         2.66
          39         328.48      322.24        5.50         7.81         0.63
       TOP : SMAX=     270.96 SMIN=     -94.37 TMAX=     182.67 ANGLE= 51.4
       BOTT: SMAX=     105.95 SMIN=    -255.92 TMAX=     180.94 ANGLE=-38.9
         JOINT         0.42       14.89        2.02        11.52         2.51
          44         767.09      768.11       28.88         8.80         1.29
       TOP : SMAX=     820.16 SMIN=     120.37 TMAX=     349.90 ANGLE= 75.6
       BOTT: SMAX=     -66.14 SMIN=    -799.04 TMAX=     366.45 ANGLE=-13.4
         JOINT         0.42       19.19        2.45        14.36         2.73
          43         941.13      942.31       28.03         9.46         0.01
       TOP : SMAX=    1007.30 SMIN=     150.47 TMAX=     428.41 ANGLE= 77.4
       BOTT: SMAX=     -96.12 SMIN=    -986.68 TMAX=     445.28 ANGLE=-12.1
             3         0.16        0.37        0.05         0.17         0.02
                      10.28       10.91        2.84         0.15         0.44
                      11.87       11.14
       TOP : SMAX=      11.87 SMIN=       5.71 TMAX=       3.08 ANGLE= 73.7
       BOTT: SMAX=      -0.47 SMIN=     -11.14 TMAX=       5.34 ANGLE= -4.2
         JOINT         0.30        0.42        0.21         0.39         0.07
          38          24.58       23.79        0.26         0.13         0.45
       TOP : SMAX=      28.30 SMIN=      12.38 TMAX=       7.96 ANGLE= 68.7
       BOTT: SMAX=     -12.46 SMIN=     -27.44 TMAX=       7.49 ANGLE=-18.4
         JOINT         0.30        0.39        0.17         0.34         0.05
          39          21.02       19.98        0.48         0.12         0.78
       TOP : SMAX=      24.17 SMIN=      10.21 TMAX=       6.98 ANGLE= 70.6
       BOTT: SMAX=     -10.15 SMIN=     -23.02 TMAX=       6.43 ANGLE=-12.9
         JOINT         0.09        0.39        0.07         0.07         0.10
          44          14.76       11.53        5.64         0.17         0.43
       TOP : SMAX=      14.83 SMIN=       0.15 TMAX=       7.34 ANGLE= 34.3
       BOTT: SMAX=       4.91 SMIN=      -8.26 TMAX=       6.59 ANGLE=-32.5
         JOINT         0.09        0.42        0.09         0.08         0.06
          43          12.77        8.52        6.10         0.39         0.13
       TOP : SMAX=      14.19 SMIN=       3.65 TMAX=       5.27 ANGLE= 26.6
       BOTT: SMAX=       2.28 SMIN=      -7.14 TMAX=       4.71 ANGLE=-28.6
      STAAD SPACE                                                -- PAGE NO.
65
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                      VONT       VONB         SX           SY           SXY
                      TRESCAT    TRESCAB
    108      1        27.56        7.22        0.94         3.43         2.66
```

```
                    394.39      344.46        8.03         3.57        16.78
                    420.47      363.48
       TOP : SMAX=     361.72 SMIN=     -58.74 TMAX=     210.23 ANGLE= 56.3
       BOTT: SMAX=      41.85 SMIN=    -321.63 TMAX=     181.74 ANGLE=-31.0
         JOINT        3.97        2.91        6.03         0.64         2.83
           39       519.87      486.99        3.35         8.82        14.84
       TOP : SMAX=     497.97 SMIN=     -41.34 TMAX=     269.65 ANGLE= 24.5
       BOTT: SMAX=      35.82 SMIN=    -468.10 TMAX=     251.96 ANGLE=-68.2
         JOINT        3.97       11.61        5.62         1.25         3.78
           40       584.73      523.89        6.26         8.73        20.72
       TOP : SMAX=     545.28 SMIN=     -72.20 TMAX=     308.74 ANGLE= 31.1
       BOTT: SMAX=      52.55 SMIN=    -495.64 TMAX=     274.09 ANGLE=-61.2
         JOINT       52.59       11.61       32.58        11.32         7.92
           45      2147.59     2087.75       19.37         3.54        18.79
       TOP : SMAX=    2376.12 SMIN=     573.50 TMAX=     901.31 ANGLE= 18.7
       BOTT: SMAX=    -585.73 SMIN=   -2318.06 TMAX=     866.16 ANGLE=-72.0
         JOINT       52.59        2.91       30.09         2.98         9.07
           44      2207.57     2156.93       22.18        13.99        13.35
       TOP : SMAX=    2218.43 SMIN=      21.88 TMAX=    1098.28 ANGLE= 17.1
       BOTT: SMAX=      -7.36 SMIN=   -2160.60 TMAX=    1076.62 ANGLE=-73.3
              2       2.67       25.76        1.15         6.86         0.44
                    434.06      420.55       15.22        12.23         0.88
                    471.82      447.06
       TOP : SMAX=     471.82 SMIN=      89.45 TMAX=     191.18 ANGLE= 85.5
       BOTT: SMAX=     -59.32 SMIN=    -447.06 TMAX=     193.87 ANGLE= -4.2
         JOINT        2.17       14.28        0.71         0.62         4.16
           39       490.35      475.64        0.51        14.08         3.95
       TOP : SMAX=     333.38 SMIN=    -229.66 TMAX=     281.52 ANGLE= 45.4
       BOTT: SMAX=     236.50 SMIN=    -311.03 TMAX=     273.76 ANGLE=-46.0
         JOINT        2.17       37.23        0.22         9.92         3.22
           40       768.02      737.12        3.85        16.25         3.44
       TOP : SMAX=     743.33 SMIN=     -47.20 TMAX=     395.27 ANGLE= 73.2
       BOTT: SMAX=      53.17 SMIN=    -709.09 TMAX=     381.13 ANGLE=-16.8
         JOINT        3.19       37.23        3.76        27.76         5.04
           45      1835.84     1832.66       30.93        10.46         5.71
       TOP : SMAX=    1932.20 SMIN=     210.93 TMAX=     860.63 ANGLE= 78.3
       BOTT: SMAX=    -155.09 SMIN=   -1905.27 TMAX=     875.09 ANGLE=-11.1
         JOINT        3.19       14.28        1.36        10.21         2.35
           44       692.33      699.03       33.99         8.36         1.80
       TOP : SMAX=     730.19 SMIN=      83.26 TMAX=     323.47 ANGLE= 75.4
       BOTT: SMAX=     -19.95 SMIN=    -708.79 TMAX=     344.42 ANGLE=-13.3
              3       0.26        0.34        0.03         0.16         0.06
                     12.25       11.15        1.15         0.08         0.85
                     12.79       11.27
       TOP : SMAX=      12.79 SMIN=       1.17 TMAX=       5.81 ANGLE= 64.5
       BOTT: SMAX=      -0.25 SMIN=     -11.27 TMAX=       5.51 ANGLE=-15.4
      STAAD SPACE                                                -- PAGE NO.
66
   ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX         SQY          MX           MY           MXY
                      VONT        VONB         SX           SY           SXY
                      TRESCAT     TRESCAB
         JOINT        0.39        0.39        0.26         0.35         0.10
           39        26.04       22.19        0.59         0.66         1.67
       TOP : SMAX=      29.90 SMIN=      12.26 TMAX=       8.82 ANGLE= 54.5
       BOTT: SMAX=     -14.10 SMIN=     -25.57 TMAX=       5.74 ANGLE=-30.3
```

```
        JOINT         0.39        0.59        0.15        0.51        0.04
          40         31.85       30.04        1.93        0.96        1.97
       TOP : SMAX=      35.85 SMIN=      10.80 TMAX=      12.52 ANGLE= 79.7
       BOTT: SMAX=      -7.73 SMIN=     -33.15 TMAX=      12.71 ANGLE= -1.0
         JOINT         0.22        0.59        0.11        0.16        0.12
          45         17.45       16.09        1.78        0.71        0.12
       TOP : SMAX=      18.51 SMIN=       2.38 TMAX=       8.07 ANGLE= 49.4
       BOTT: SMAX=       0.11 SMIN=     -16.03 TMAX=       8.07 ANGLE=-36.7
         JOINT         0.22        0.39        0.13        0.11        0.11
          44         17.78       13.65        4.23        0.90        0.35
       TOP : SMAX=      19.00 SMIN=       2.76 TMAX=       8.12 ANGLE= 36.7
       BOTT: SMAX=       1.39 SMIN=     -12.90 TMAX=       7.15 ANGLE=-41.0
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          4.851880E+03 -4.815351E+03  2.183280E+03  1.016677E+03  1.094815E+03
 PLATE NO.      54            54            54            54            54
 CASE  NO.       2             2             2             2             2
   ********************END OF ELEMENT FORCES********************
   159. FINISH
      STAAD SPACE                                              -- PAGE NO.
67
      STAAD SPACE                                              -- PAGE NO.
68
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:49:11 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Output using the Control-Dependent method:

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:49: 8                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: Response Spectrum Using Control Dependent.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 09-MAR-21
     4. END JOB INFORMATION
```

```
      5. INPUT WIDTH 79
      6. SET PRINT 17
      7. UNIT METER KN
      8. JOINT COORDINATES
      9. 1 0 0 0; 2 0 4 0; 3 5 4 0; 4 5 0 0; 5 10 4 0; 6 10 0 0; 7 15 4 0; 8
15 0 0
     10. 9 20 4 0; 10 20 0 0; 11 0 0 4; 12 0 4 4; 13 5 4 4; 14 5 0 4; 15 10 4 4
     11. 16 10 0 4; 17 15 4 4; 18 15 0 4; 19 20 4 4; 20 20 0 4; 21 0 0 8; 22 0
4 8
     12. 23 5 4 8; 24 5 0 8; 25 10 4 8; 26 10 0 8; 27 15 4 8; 28 15 0 8; 29 20
4 8
     13. 30 20 0 8; 31 0 8 0; 32 5 8 0; 33 10 8 0; 34 15 8 0; 35 20 8 0; 36 0
8 4
     14. 37 5 8 4; 38 10 8 4; 39 15 8 4; 40 20 8 4; 41 0 8 8; 42 5 8 8; 43 10
8 8
     15. 44 15 8 8; 45 20 8 8; 46 0.1 4 0.1; 47 5 4 0.1; 48 10 4 0.1; 49 15 4
0.1
     16. 50 19.9 4 0.1; 51 19.9 4 4; 52 19.9 4 7.9; 53 5 4 7.9; 54 10 4 7.9
     17. 55 15 4 7.9; 56 0.1 4 7.9; 57 0.1 4 4; 58 0.1 8 0.1; 59 5 8 0.1; 60
10 8 0.1
     18. 61 15 8 0.1; 62 19.9 8 0.1; 63 19.9 8 4; 64 19.9 8 7.9; 65 5 8 7.9
     19. 66 10 8 7.9; 67 15 8 7.9; 68 0.1 8 7.9; 69 0.1 8 4
     20. MEMBER INCIDENCES
     21. 1 1 2; 2 2 3; 3 3 4; 4 3 5; 5 5 6; 6 5 7; 9 7 8; 10 7 9; 13 9 10; 14
2 12
     22. 15 3 13; 16 5 15; 17 7 17; 18 9 19; 19 11 12; 20 12 13; 21 13 14; 22
13 15
     23. 23 15 16; 24 15 17; 27 17 18; 28 17 19; 31 19 20; 32 12 22; 33 13 23;
34 15 25
     24. 35 17 27; 36 19 29; 37 21 22; 38 22 23; 39 23 24; 40 23 25; 41 25 26;
42 25 27
     25. 45 27 28; 46 27 29; 49 29 30; 58 2 31; 59 3 32; 60 5 33; 61 7 34; 62
9 35
     26. 63 12 36; 64 13 37; 65 15 38; 66 17 39; 67 19 40; 68 22 41; 69 23 42;
70 25 43
     27. 71 27 44; 72 29 45; 73 31 32; 74 32 33; 75 33 34; 77 34 35; 79 31 36;
80 32 37
     28. 81 33 38; 82 34 39; 83 35 40; 84 36 37; 85 37 38; 86 38 39; 88 39 40;
90 36 41
     29. 91 37 42; 92 38 43; 93 39 44; 94 40 45; 95 41 42; 96 42 43; 97 43 44;
99 44 45
     30. ELEMENT INCIDENCES SHELL
     31. 50 46 47 13 57; 51 47 48 15 13; 52 48 49 17 15; 53 49 50 51 17; 54 57
13 53 56
     32. 55 13 15 54 53; 56 15 17 55 54; 57 17 51 52 55; 101 58 59 37 69
     33. 102 59 60 38 37; 103 60 61 39 38; 104 61 62 63 39; 105 69 37 65 68
     34. 106 37 38 66 65; 107 38 39 67 66; 108 39 63 64 67
     35. ELEMENT PROPERTY
     36. 50 TO 57 101 TO 108 THICKNESS 0.3
     37. DEFINE MATERIAL START
     38. ISOTROPIC STEEL
      STAAD SPACE                                               -- PAGE NO.
2
     39. E 2.05E+08
     40. POISSON 0.3
     41. DENSITY 76.8195
     42. ALPHA 1.2E-05
     43. DAMP 0.03
```

```
    44. G 7.88462E+07
    45. TYPE STEEL
    46. STRENGTH RY 1.5 RT 1.2
    47. ISOTROPIC CONCRETE
    48. E 2.17185E+07
    49. POISSON 0.17
    50. DENSITY 23.5616
    51. ALPHA 1E-05
    52. DAMP 0.05
    53. G 9.28139E+06
    54. TYPE CONCRETE
    55. STRENGTH FCU 27579
    56. END DEFINE MATERIAL
    57. MEMBER PROPERTY AMERICAN
    58. 1 TO 6 9 10 13 TO 24 27 28 31 TO 36 38 40 42 46 58 TO 67 73 TO 75 77 -
    59. 79 TO 86 88 90 TO 97 99 TABLE ST W12X50
    60. 37 39 41 45 49 68 TO 72 TABLE ST W30X90
    61. CONSTANTS
    62. MATERIAL STEEL MEMB 1 TO 6 9 10 13 TO 24 27 28 31 TO 42 45 46 49 58
TO 75 -
    63. 77 79 TO 86 88 88 90 TO 97 99
    64. MATERIAL CONCRETE MEMB 50 TO 57 101 TO 108
    65. SUPPORTS
    66. 1 4 6 8 10 11 14 16 18 20 21 24 26 28 30 FIXED
    67. *ELEMENT OFFSET
    68. *50 101 JT1 0.1 0 0.1
    69. *50 101 JT2 0 0 0.1
    70. *50 101 JT4 0.1 0 0
    71. *54 105 JT1 0.1 0 0
    72. *54 105 JT3 0 0 -0.1
    73. *54 105 JT4 0.1 0 -0.1
    74. *51 102 JT1 0 0 0.1
    75. *51 102 JT2 0 0 0.1
    76. *55 106 JT3 0 0 -0.1
    77. *55 106 JT4 0 0 -0.1
    78. *52 103 JT1 0 0 0.1
    79. *52 103 JT2 0 0 0.1
    80. *56 107 JT3 0 0 -0.1
    81. *56 107 JT4 0 0 -0.1
    82. *53 104 JT1 0 0 0.1
    83. *53 104 JT2 -0.1 0 0.1
    84. *53 104 JT3 -0.1 0 0
    85. *57 108 JT2 -0.1 0 0
    86. *57 108 JT3 -0.1 0 -0.1
    87. *57 108 JT4 0 0 -0.1
    88. CONTROL RIGID CONTROL 2 JOINT 46
    89. CONTROL RIGID CONTROL 3 JOINT 47
   *** STAAD.Pro ERROR MESSAGE ***
   DATA SHOULD FOLLOW AFTER CONSTANT COMMAND AND NOT
   ANY OTHER TYPE OF COMMAND. DATA-CHECK MODE ENTERED.
      STAAD SPACE                                              -- PAGE NO.
3
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:49: 8 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
```

```
         *   Details about additional assistance from              *
         *   Bentley and Partners can be found at program menu     *
         *   Help->Technical Support                               *
         *                                                         *
         *           Copyright (c) Bentley Systems, Inc.           *
         *                http://www.bentley.com                   *
         ***********************************************************
```

V. Spherical Shell Displacements

To find the displacement in the direction of the load due to a unit load applied at the quadrants of a quarter of a spherical shell.

Reference

MacNeal, R.H. and Harder, R.C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*, 1985.

Problem

For the quarter of a spherical shell find the displacement in the direction of the load.

$E = 6.825(10)^7$ psi

Poisson's ratio = 0.3

t = 0.04 inches

r = 10 in.

Unit forces on quadrants

Boundary conditions:

Vertical restraint at center of free edge
Symmetry defines boundary conditions

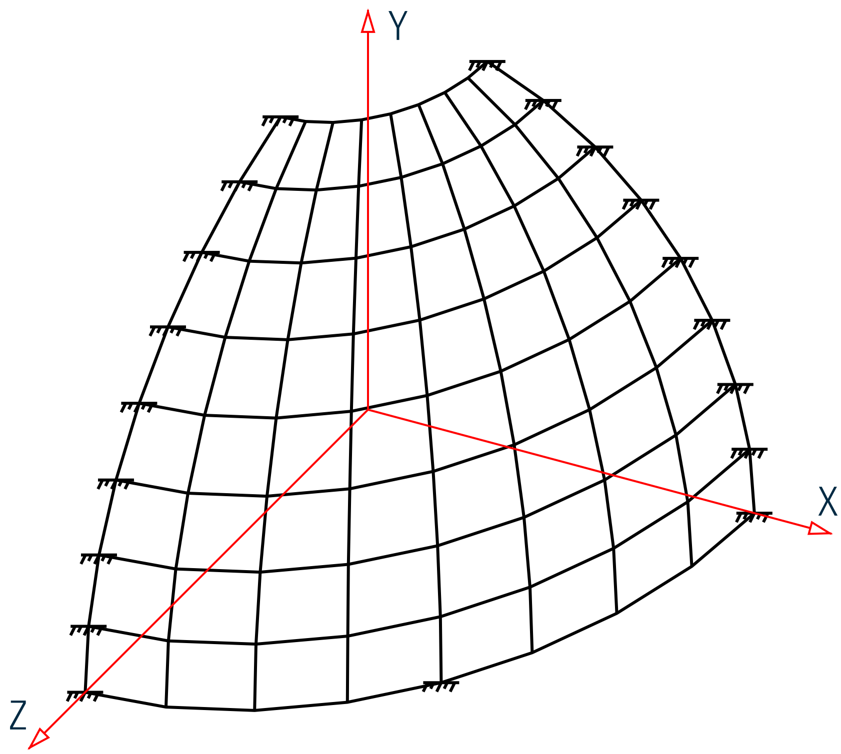


Figure 416: Model

Comparison

Table 438: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Deflection, δ (in)	0.094	0.09342	<1%

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Spherical Shell Displacements.STD` is typically installed with the program.

```
STAAD SPACE :A QUARTER OF A SHERICAL SHELL
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
*             TO TEST FINITE ELEMENT ACCURACY,
*             FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
```

```
*           1985
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 10 0 0; 2 0 0 10; 3 1.951 0 9.808; 4 3.827 0 9.239; 5 5.556 0 8.315;
6 7.071 0 7.071; 7 8.315 0 5.556; 8 9.239 0 3.827; 9 9.808 0 1.951;
10 9.877 1.564 0; 11 0 1.564 9.877; 12 1.927 1.564 9.687;
13 3.78 1.564 9.125; 14 5.487 1.564 8.212; 15 6.984 1.564 6.984;
16 8.212 1.564 5.487; 17 9.125 1.564 3.78; 18 9.687 1.564 1.927;
19 9.511 3.09 0; 20 0 3.09 9.511; 21 1.856 3.09 9.328;
22 3.64 3.09 8.787; 23 5.284 3.09 7.908; 24 6.725 3.09 6.725;
25 7.908 3.09 5.284; 26 8.787 3.09 3.64; 27 9.328 3.09 1.855;
28 8.91 4.54 0; 29 0 4.54 8.91; 30 1.738 4.54 8.739; 31 3.41 4.54 8.232;
32 4.95 4.54 7.408; 33 6.3 4.54 6.3; 34 7.408 4.54 4.95;
35 8.232 4.54 3.41; 36 8.739 4.54 1.738; 37 8.09 5.878 0;
38 0 5.878 8.09; 39 1.578 5.878 7.935; 40 3.096 5.878 7.474;
41 4.495 5.878 6.727; 42 5.721 5.878 5.72; 43 6.727 5.878 4.495;
44 7.474 5.878 3.096; 45 7.935 5.878 1.578; 46 7.071 7.071 0;
47 0 7.071 7.071; 48 1.379 7.071 6.935; 49 2.706 7.071 6.533;
50 3.928 7.071 5.879; 51 5 7.071 5; 52 5.879 7.071 3.928;
53 6.533 7.071 2.706; 54 6.935 7.071 1.379; 55 5.878 8.09 0;
56 0 8.09 5.878; 57 1.147 8.09 5.765; 58 2.249 8.09 5.431;
59 3.266 8.09 4.887; 60 4.156 8.09 4.156; 61 4.887 8.09 3.266;
62 5.431 8.09 2.249; 63 5.765 8.09 1.147; 64 4.54 8.91 0;
65 0 8.91 4.54; 66 0.886 8.91 4.453; 67 1.737 8.91 4.194;
68 2.522 8.91 3.775; 69 3.21 8.91 3.21; 70 3.775 8.91 2.522;
71 4.194 8.91 1.737; 72 4.453 8.91 0.886; 73 3.09 9.511 0;
74 0 9.511 3.09; 75 0.603 9.511 3.031; 76 1.182 9.511 2.855;
77 1.717 9.511 2.569; 78 2.185 9.511 2.185; 79 2.569 9.511 1.717;
80 2.855 9.511 1.182; 81 3.031 9.511 0.603;
ELEMENT INCIDENCES SHELL
1 2 3 12 11; 2 3 4 13 12; 3 4 5 14 13; 4 5 6 15 14; 5 6 7 16 15;
6 7 8 17 16; 7 8 9 18 17; 8 9 1 10 18; 9 11 12 21 20; 10 12 13 22 21;
11 13 14 23 22; 12 14 15 24 23; 13 15 16 25 24; 14 16 17 26 25;
15 17 18 27 26; 16 18 10 19 27; 17 20 21 30 29; 18 21 22 31 30;
19 22 23 32 31; 20 23 24 33 32; 21 24 25 34 33; 22 25 26 35 34;
23 26 27 36 35; 24 27 19 28 36; 25 29 30 39 38; 26 30 31 40 39;
27 31 32 41 40; 28 32 33 42 41; 29 33 34 43 42; 30 34 35 44 43;
31 35 36 45 44; 32 36 28 37 45; 33 38 39 48 47; 34 39 40 49 48;
35 40 41 50 49; 36 41 42 51 50; 37 42 43 52 51; 38 43 44 53 52;
39 44 45 54 53; 40 45 37 46 54; 41 47 48 57 56; 42 48 49 58 57;
43 49 50 59 58; 44 50 51 60 59; 45 51 52 61 60; 46 52 53 62 61;
47 53 54 63 62; 48 54 46 55 63; 49 56 57 66 65; 50 57 58 67 66;
51 58 59 68 67; 52 59 60 69 68; 53 60 61 70 69; 54 61 62 71 70;
55 62 63 72 71; 56 63 55 64 72; 57 65 66 75 74; 58 66 67 76 75;
59 67 68 77 76; 60 68 69 78 77; 61 69 70 79 78; 62 70 71 80 79;
63 71 72 81 80; 64 72 64 73 81;
ELEMENT PROPERTY
1 TO 64 THICKNESS 0.04
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 6.825e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
```

```
1 10 19 28 37 46 55 64 73 FIXED BUT FX FY MZ
2 11 20 29 38 47 56 65 74 FIXED BUT FY FZ MX
6 FIXED BUT FX FZ MX MY MZ
LOAD 1 UNIT FORCE ON QUADRANT
JOINT LOAD
2 FZ 1
1 FX -1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 1 2
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1   -0.09335  -0.04663   0.00000   0.00000   0.00000  -0.01893
      2    1    0.00000   0.04665   0.09342  -0.01898   0.00000   0.00000
```

V. Thermal Load on a Plate

Find deflections and moments due to thermal loading and compare theoretical answers to the STAAD solution.

Reference

Timoshenko, S., and Woinowsky-Krieger, S., *Theory of Plates and Shells*, Second Edition, McGraw-Hill, 1959, pages 162 - 165.

Problem

A rectangular plate is simply supported on all four sides. The transverse and longitudinal bending moments as well as the deflections at several points on the plate are computed.

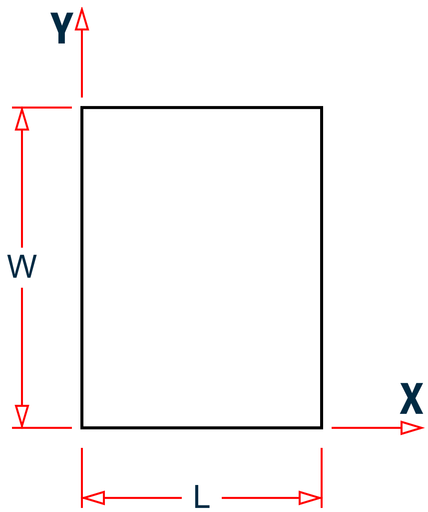


Figure 417: Rectangular plate model

L = 12 in., W = 16 in.

The plate is modeled using 1 in. X 1 in. size elements. At the corner nodes, all the degrees of freedom are considered restrained. For the nodes along the four edges, rotation is permitted about that edge.

Theoretical Solution

From the Reference, equation (j), the expression for deflection normal to the plate surface is:

$$w = -\frac{at(1+\nu)4a^2}{\pi^3 h}\sum_{m=1,3,5\ldots}^{\infty}\frac{\sin\frac{m\pi x}{a}}{m^3}\left(1-\frac{\cosh\frac{m\pi y}{a}}{\cosh a_m}\right)$$

where

$$a_m = \frac{m\pi b}{2a}$$

From the Reference, equation (k), the expressions for bending moment per unit width are

$$M_x = \frac{4Dat(1-\nu^2)}{\pi h}\sum_{m=1,3,5\ldots}^{\infty}\frac{\sin\frac{m\pi x}{a}\cosh\frac{m\pi y}{a}}{m\cdot\cosh a_m}$$

$$M_y = \frac{at(1-\nu^2)D}{h} - \frac{4Dat(1-\nu^2)}{\pi h}\sum_{m=1,3,5\ldots}^{\infty}\frac{\sin\frac{m\pi x}{a}\cosh\frac{m\pi y}{a}}{m\cdot\cosh a_m}$$

where

a	=	Coefficient of Thermal Expansion
t	=	Difference between the temperatures of the upper and lower surfaces of the plate
ν	=	Poisson's ratio
h	=	Plate thickness
a	=	Dimension of the plate along the x1 axis
b	=	Dimension of the plate along the x2 axis
E	=	Elastic Modulus
D	=	$Eh^3/[12(1 - \nu^2)$

The numerical values used for this example are:

$\alpha = 12.0E\text{-}06$ / °F
$t = 450$ °F
$\nu = 0.3$
$h = 0.3$ in.
$a = 12$ in.
$b = 16$ in.
$E = 10.0E6$ psi

Comparison

Table 439: Comparison of results

Node Number	X	Y	Theoretical Deflection	STAAD.ProDeflection
7	6	-8	0.00	0.00
20	6	-7	0.0897	0.0897
33	6	-6	0.1597	0.1597
46	6	-5	0.2132	0.2132
59	6	-4	0.2531	0.2531
72	6	-3	0.2818	0.2818
85	6	-2	0.3011	0.3011
98	6	-1	0.3122	0.3122
111	6	0	0.3158	0.3158
124	6	1	0.3122	0.3122

Table 440: Comparison of results

Node Number	X	Y	Theoretical Deflection	STAAD.ProDeflection
118	0	1	0	0
119	1	1	0.1004	0.1004
120	2	1	0.1794	0.1794
121	3	1	0.2387	0.2387
122	4	1	0.2799	0.2799

Node Number	X	Y	Theoretical Deflection	STAAD.ProDeflection
123	5	1	0.3042	0.3042
124	6	1	0.3122	0.3122
125	7	1	0.3042	0.3042
126	8	1	0.2799	0.2799
127	9	1	0.2387	0.2387
128	10	1	0.1794	0.1794
129	11	1	0.1004	0.1004
130	12	1	0	0

Table 441: Comparison of results

Element Number	X	Y	Theoretical Moment (Pound-in/in)		STAAD STAAD.Pro Moment (Pound-in/in)	
			Mx	My	Mx	My
97	0.5	0.5	16.74	388.26	16.85	388.37
98	1.5	0.5	48.93	356.07	49.25	356.37
99	2.5	0.5	77.45	327.55	77.92	328.01
100	3.5	0.5	100.36	304.64	100.93	305.20
101	4.5	0.5	116.31	288.69	116.93	289.32
102	5.5	0.5	124.47	280.54	125.11	281.19
103	6.5	0.5	124.47	280.54	125.11	281.19
104	7.5	0.5	116.31	288.69	116.93	289.32
105	8.5	0.5	100.36	304.64	100.93	305.20
106	9.5	0.5	77.45	327.55	77.92	328.01
107	10.5	0.5	48.93	356.07	49.25	356.37
108	11.5	0.5	16.74	388.26	16.85	388.37

Table 442: Comparison of results

Element Number	X	Y	Theoretical Moment (Pound-in/in)		STAAD.ProMoment (Pound-in/in)	
			Mx	My	Mx	My
6	5.5	-7.5	373.88	31.12	373.27	32.17
18	5.5	-6.5	311.64	93.36	312.19	93.95
30	5.5	-5.5	256.83	148.17	257.59	148.94
42	5.5	-4.5	211.14	193.87	211.95	194.66
54	5.5	-3.5	175.41	229.60	176.20	230.35
66	5.5	-2.5	149.47	255.53	150.19	256.25
78	5.5	-1.5	132.68	272.32	133.36	273.00
90	5.5	-0.5	124.47	280.54	125.11	281.19
102	5.5	0.5	124.47	280.54	125.11	281.19
114	5.5	1.5	132.68	272.32	133.36	273.00
126	5.5	2.5	149.47	255.53	150.19	256.25
138	5.5	3.5	175.41	229.59	176.20	230.35
150	5.5	4.5	211.14	193.87	211.95	194.66
162	5.5	5.5	256.83	148.17	257.59	148.94

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Thermal Load on a Plate.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* Thermal loading on a simply supported rectangular plate
UNIT INCHES POUND
JOINT COORDINATES
1 0 -8 0; 2 1 -8 0; 3 2 -8 0; 4 3 -8 0; 5 4 -8 0; 6 5 -8 0; 7 6 -8 0;
8 7 -8 0; 9 8 -8 0; 10 9 -8 0; 11 10 -8 0; 12 11 -8 0; 13 12 -8 0;
14 0 -7 0; 15 1 -7 0; 16 2 -7 0; 17 3 -7 0; 18 4 -7 0; 19 5 -7 0;
20 6 -7 0; 21 7 -7 0; 22 8 -7 0; 23 9 -7 0; 24 10 -7 0; 25 11 -7 0;
26 12 -7 0; 27 0 -6 0; 28 1 -6 0; 29 2 -6 0; 30 3 -6 0; 31 4 -6 0;
32 5 -6 0; 33 6 -6 0; 34 7 -6 0; 35 8 -6 0; 36 9 -6 0; 37 10 -6 0;
```

```
38 11 -6 0; 39 12 -6 0; 40 0 -5 0; 41 1 -5 0; 42 2 -5 0; 43 3 -5 0;
44 4 -5 0; 45 5 -5 0; 46 6 -5 0; 47 7 -5 0; 48 8 -5 0; 49 9 -5 0;
50 10 -5 0; 51 11 -5 0; 52 12 -5 0; 53 0 -4 0; 54 1 -4 0; 55 2 -4 0;
56 3 -4 0; 57 4 -4 0; 58 5 -4 0; 59 6 -4 0; 60 7 -4 0; 61 8 -4 0;
62 9 -4 0; 63 10 -4 0; 64 11 -4 0; 65 12 -4 0; 66 0 -3 0; 67 1 -3 0;
68 2 -3 0; 69 3 -3 0; 70 4 -3 0; 71 5 -3 0; 72 6 -3 0; 73 7 -3 0;
74 8 -3 0; 75 9 -3 0; 76 10 -3 0; 77 11 -3 0; 78 12 -3 0; 79 0 -2 0;
80 1 -2 0; 81 2 -2 0; 82 3 -2 0; 83 4 -2 0; 84 5 -2 0; 85 6 -2 0;
86 7 -2 0; 87 8 -2 0; 88 9 -2 0; 89 10 -2 0; 90 11 -2 0; 91 12 -2 0;
92 0 -1 0; 93 1 -1 0; 94 2 -1 0; 95 3 -1 0; 96 4 -1 0; 97 5 -1 0;
98 6 -1 0; 99 7 -1 0; 100 8 -1 0; 101 9 -1 0; 102 10 -1 0; 103 11 -1 0;
104 12 -1 0; 105 0 0 0; 106 1 0 0; 107 2 0 0; 108 3 0 0; 109 4 0 0;
110 5 0 0; 111 6 0 0; 112 7 0 0; 113 8 0 0; 114 9 0 0; 115 10 0 0;
116 11 0 0; 117 12 0 0; 118 0 1 0; 119 1 1 0; 120 2 1 0; 121 3 1 0;
122 4 1 0; 123 5 1 0; 124 6 1 0; 125 7 1 0; 126 8 1 0; 127 9 1 0;
128 10 1 0; 129 11 1 0; 130 12 1 0; 131 0 2 0; 132 1 2 0; 133 2 2 0;
134 3 2 0; 135 4 2 0; 136 5 2 0; 137 6 2 0; 138 7 2 0; 139 8 2 0;
140 9 2 0; 141 10 2 0; 142 11 2 0; 143 12 2 0; 144 0 3 0; 145 1 3 0;
146 2 3 0; 147 3 3 0; 148 4 3 0; 149 5 3 0; 150 6 3 0; 151 7 3 0;
152 8 3 0; 153 9 3 0; 154 10 3 0; 155 11 3 0; 156 12 3 0; 157 0 4 0;
158 1 4 0; 159 2 4 0; 160 3 4 0; 161 4 4 0; 162 5 4 0; 163 6 4 0;
164 7 4 0; 165 8 4 0; 166 9 4 0; 167 10 4 0; 168 11 4 0; 169 12 4 0;
170 0 5 0; 171 1 5 0; 172 2 5 0; 173 3 5 0; 174 4 5 0; 175 5 5 0;
176 6 5 0; 177 7 5 0; 178 8 5 0; 179 9 5 0; 180 10 5 0; 181 11 5 0;
182 12 5 0; 183 0 6 0; 184 1 6 0; 185 2 6 0; 186 3 6 0; 187 4 6 0;
188 5 6 0; 189 6 6 0; 190 7 6 0; 191 8 6 0; 192 9 6 0; 193 10 6 0;
194 11 6 0; 195 12 6 0; 196 0 7 0; 197 1 7 0; 198 2 7 0; 199 3 7 0;
200 4 7 0; 201 5 7 0; 202 6 7 0; 203 7 7 0; 204 8 7 0; 205 9 7 0;
206 10 7 0; 207 11 7 0; 208 12 7 0; 209 0 8 0; 210 1 8 0; 211 2 8 0;
212 3 8 0; 213 4 8 0; 214 5 8 0; 215 6 8 0; 216 7 8 0; 217 8 8 0;
218 9 8 0; 219 10 8 0; 220 11 8 0; 221 12 8 0;
ELEMENT INCIDENCES SHELL
1 1 2 15 14; 2 2 3 16 15; 3 3 4 17 16; 4 4 5 18 17; 5 5 6 19 18;
6 6 7 20 19; 7 7 8 21 20; 8 8 9 22 21; 9 9 10 23 22; 10 10 11 24 23;
11 11 12 25 24; 12 12 13 26 25; 13 14 15 28 27; 14 15 16 29 28;
15 16 17 30 29; 16 17 18 31 30; 17 18 19 32 31; 18 19 20 33 32;
19 20 21 34 33; 20 21 22 35 34; 21 22 23 36 35; 22 23 24 37 36;
23 24 25 38 37; 24 25 26 39 38; 25 27 28 41 40; 26 28 29 42 41;
27 29 30 43 42; 28 30 31 44 43; 29 31 32 45 44; 30 32 33 46 45;
31 33 34 47 46; 32 34 35 48 47; 33 35 36 49 48; 34 36 37 50 49;
35 37 38 51 50; 36 38 39 52 51; 37 40 41 54 53; 38 41 42 55 54;
39 42 43 56 55; 40 43 44 57 56; 41 44 45 58 57; 42 45 46 59 58;
43 46 47 60 59; 44 47 48 61 60; 45 48 49 62 61; 46 49 50 63 62;
47 50 51 64 63; 48 51 52 65 64; 49 53 54 67 66; 50 54 55 68 67;
51 55 56 69 68; 52 56 57 70 69; 53 57 58 71 70; 54 58 59 72 71;
55 59 60 73 72; 56 60 61 74 73; 57 61 62 75 74; 58 62 63 76 75;
59 63 64 77 76; 60 64 65 78 77; 61 66 67 80 79; 62 67 68 81 80;
63 68 69 82 81; 64 69 70 83 82; 65 70 71 84 83; 66 71 72 85 84;
67 72 73 86 85; 68 73 74 87 86; 69 74 75 88 87; 70 75 76 89 88;
71 76 77 90 89; 72 77 78 91 90; 73 79 80 93 92; 74 80 81 94 93;
75 81 82 95 94; 76 82 83 96 95; 77 83 84 97 96; 78 84 85 98 97;
79 85 86 99 98; 80 86 87 100 99; 81 87 88 101 100; 82 88 89 102 101;
83 89 90 103 102; 84 90 91 104 103; 85 92 93 106 105; 86 93 94 107 106;
87 94 95 108 107; 88 95 96 109 108; 89 96 97 110 109; 90 97 98 111 110;
91 98 99 112 111; 92 99 100 113 112; 93 100 101 114 113;
94 101 102 115 114; 95 102 103 116 115; 96 103 104 117 116;
97 105 106 119 118; 98 106 107 120 119; 99 107 108 121 120;
100 108 109 122 121; 101 109 110 123 122; 102 110 111 124 123;
```

```
103 111 112 125 124; 104 112 113 126 125; 105 113 114 127 126;
106 114 115 128 127; 107 115 116 129 128; 108 116 117 130 129;
109 118 119 132 131; 110 119 120 133 132; 111 120 121 134 133;
112 121 122 135 134; 113 122 123 136 135; 114 123 124 137 136;
115 124 125 138 137; 116 125 126 139 138; 117 126 127 140 139;
118 127 128 141 140; 119 128 129 142 141; 120 129 130 143 142;
121 131 132 145 144; 122 132 133 146 145; 123 133 134 147 146;
124 134 135 148 147; 125 135 136 149 148; 126 136 137 150 149;
127 137 138 151 150; 128 138 139 152 151; 129 139 140 153 152;
130 140 141 154 153; 131 141 142 155 154; 132 142 143 156 155;
133 144 145 158 157; 134 145 146 159 158; 135 146 147 160 159;
136 147 148 161 160; 137 148 149 162 161; 138 149 150 163 162;
139 150 151 164 163; 140 151 152 165 164; 141 152 153 166 165;
142 153 154 167 166; 143 154 155 168 167; 144 155 156 169 168;
145 157 158 171 170; 146 158 159 172 171; 147 159 160 173 172;
148 160 161 174 173; 149 161 162 175 174; 150 162 163 176 175;
151 163 164 177 176; 152 164 165 178 177; 153 165 166 179 178;
154 166 167 180 179; 155 167 168 181 180; 156 168 169 182 181;
157 170 171 184 183; 158 171 172 185 184; 159 172 173 186 185;
160 173 174 187 186; 161 174 175 188 187; 162 175 176 189 188;
163 176 177 190 189; 164 177 178 191 190; 165 178 179 192 191;
166 179 180 193 192; 167 180 181 194 193; 168 181 182 195 194;
169 183 184 197 196; 170 184 185 198 197; 171 185 186 199 198;
172 186 187 200 199; 173 187 188 201 200; 174 188 189 202 201;
175 189 190 203 202; 176 190 191 204 203; 177 191 192 205 204;
178 192 193 206 205; 179 193 194 207 206; 180 194 195 208 207;
181 196 197 210 209; 182 197 198 211 210; 183 198 199 212 211;
184 199 200 213 212; 185 200 201 214 213; 186 201 202 215 214;
187 202 203 216 215; 188 203 204 217 216; 189 204 205 218 217;
190 205 206 219 218; 191 206 207 220 219; 192 207 208 221 220;
ELEMENT PROPERTY
1 TO 192 THICKNESS 0.3
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
ALPHA 1.2e-05
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 13 209 221 FIXED
2 TO 12 210 TO 220 FIXED BUT MX
14 26 27 39 40 52 53 65 66 78 79 91 92 104 105 117 118 130 131 143 -
144 156 157 169 170 182 183 195 196 208 FIXED BUT MY
LOAD 1
TEMPERATURE LOAD
1 TO 192 TEMP 0 450
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 7 20 33 46 59 72 85 98 111 118 TO 130 -
137 150 163 176 189 202 215
PRINT ELEMENT STRESSES LIST 6 18 30 42 54 66 78 90 97 TO 108 114 126 -
138 150 162 174 186
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      7    1    0.00000   0.00000   0.00000   0.10098   0.00000   0.00000
     20    1    0.00000   0.00000   0.08974   0.07940   0.00000   0.00000
     33    1    0.00000   0.00000   0.15969   0.06132   0.00000   0.00000
     46    1    0.00000   0.00000   0.21318   0.04637   0.00000   0.00000
     59    1    0.00000   0.00000   0.25310   0.03406   0.00000   0.00000
     72    1    0.00000   0.00000   0.28181   0.02380   0.00000   0.00000
     85    1    0.00000   0.00000   0.30109   0.01506   0.00000   0.00000
     98    1    0.00000   0.00000   0.31218   0.00729   0.00000   0.00000
    111    1    0.00000   0.00000   0.31579   0.00000   0.00000   0.00000
    118    1    0.00000   0.00000   0.00000   0.00000  -0.11185   0.00000
    119    1    0.00000   0.00000   0.10040  -0.00193  -0.08944   0.00000
    120    1    0.00000   0.00000   0.17937  -0.00371  -0.06893   0.00000
    121    1    0.00000   0.00000   0.23869  -0.00521  -0.05010   0.00000
    122    1    0.00000   0.00000   0.27991  -0.00635  -0.03262   0.00000
    123    1    0.00000   0.00000   0.30417  -0.00705  -0.01607   0.00000
    124    1    0.00000   0.00000   0.31218  -0.00729   0.00000   0.00000
    125    1    0.00000   0.00000   0.30417  -0.00705   0.01607   0.00000
    126    1    0.00000   0.00000   0.27991  -0.00635   0.03262   0.00000
    127    1    0.00000   0.00000   0.23869  -0.00521   0.05010   0.00000
    128    1    0.00000   0.00000   0.17937  -0.00371   0.06893   0.00000
    129    1    0.00000   0.00000   0.10040  -0.00193   0.08944   0.00000
    130    1    0.00000   0.00000   0.00000   0.00000   0.11185   0.00000
    137    1    0.00000   0.00000   0.30109  -0.01506   0.00000   0.00000
    150    1    0.00000   0.00000   0.28181  -0.02380   0.00000   0.00000
    163    1    0.00000   0.00000   0.25310  -0.03406   0.00000   0.00000
    176    1    0.00000   0.00000   0.21318  -0.04637   0.00000   0.00000
    189    1    0.00000   0.00000   0.15969  -0.06132   0.00000   0.00000
    202    1    0.00000   0.00000   0.08974  -0.07940   0.00000   0.00000
    215    1    0.00000   0.00000   0.00000  -0.10098   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
   123. PRINT ELEMENT STRESSES LIST 6 18 30 42 54 66 78 90 97 TO 108 114 126 -
  ELEMENT  STRESSES LIST     6
   124. 138 150 162 174 186
      STAAD SPACE                                              -- PAGE NO.
5
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
      6      1         0.34       -4.36     -373.27      -32.17      -31.51
                   24160.11    24160.11        0.00        0.00        0.00
                   25076.87    25076.87
       TOP : SMAX=   -1951.93 SMIN=  -25076.87 TMAX=   11562.47 ANGLE=-84.8
       BOTT: SMAX=   25076.87 SMIN=    1951.93 TMAX=   11562.47 ANGLE=  5.2
     18      1         0.74       -2.75     -312.19      -93.95      -29.19
                   18798.88    18798.88        0.00        0.00        0.00
                   21068.45    21068.45
       TOP : SMAX=   -6007.83 SMIN=  -21068.45 TMAX=    7530.31 ANGLE=-82.5
       BOTT: SMAX=   21068.45 SMIN=    6007.83 TMAX=    7530.31 ANGLE=  7.5
```

```
    30      1         0.67       -1.03     -257.59     -148.94      -25.19
                  15213.03    15213.03        0.00        0.00        0.00
                  17542.96    17542.96
      TOP : SMAX=   -9558.83 SMIN=  -17542.96 TMAX=    3992.07 ANGLE=-77.6
      BOTT: SMAX=   17542.96 SMIN=    9558.83 TMAX=    3992.07 ANGLE= 12.4
    42      1         0.38       -0.06     -211.95     -194.66      -20.36
                  13792.44    13792.44        0.00        0.00        0.00
                  15028.62    15028.62
      TOP : SMAX=  -12079.03 SMIN=  -15028.62 TMAX=    1474.80 ANGLE=-56.5
      BOTT: SMAX=   15028.62 SMIN=   12079.03 TMAX=    1474.80 ANGLE= 33.5
    54      1         0.12        0.39     -176.20     -230.35      -15.40
                  14020.85    14020.85        0.00        0.00        0.00
                  15628.28    15628.28
      TOP : SMAX=  -11474.99 SMIN=  -15628.28 TMAX=    2076.64 ANGLE=-14.8
      BOTT: SMAX=   15628.28 SMIN=   11474.99 TMAX=    2076.64 ANGLE= 75.2
    66      1        -0.02        0.49     -150.19     -256.25      -10.66
                  14918.57    14918.57        0.00        0.00        0.00
                  17154.23    17154.23
      TOP : SMAX=   -9942.06 SMIN=  -17154.23 TMAX=    3606.09 ANGLE= -5.7
      BOTT: SMAX=   17154.23 SMIN=    9942.06 TMAX=    3606.09 ANGLE= 84.3
    78      1        -0.09        0.36     -133.36     -273.00       -6.23
                  15779.24    15779.24        0.00        0.00        0.00
                  18218.24    18218.24
      TOP : SMAX=   -8871.89 SMIN=  -18218.24 TMAX=    4673.17 ANGLE= -2.5
      BOTT: SMAX=   18218.24 SMIN=    8871.89 TMAX=    4673.17 ANGLE= 87.5
    90      1        -0.13        0.13     -125.11     -281.19       -2.05
                  16269.07    16269.07        0.00        0.00        0.00
                  18747.84    18747.84
      TOP : SMAX=   -8338.78 SMIN=  -18747.84 TMAX=    5204.53 ANGLE= -0.8
      BOTT: SMAX=   18747.84 SMIN=    8338.78 TMAX=    5204.53 ANGLE= 89.2
     STAAD SPACE                                             -- PAGE NO.
6
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
    97      1        -2.00       -0.08      -16.85     -388.37       16.53
                  25419.74    25419.74        0.00        0.00        0.00
                  25939.96    25939.96
      TOP : SMAX=   -1074.52 SMIN=  -25939.96 TMAX=   12432.72 ANGLE=  2.5
      BOTT: SMAX=   25939.96 SMIN=    1074.52 TMAX=   12432.72 ANGLE=-87.5
    98      1        -1.69       -0.20      -49.25     -356.37       15.27
                  22367.84    22367.84        0.00        0.00        0.00
                  23808.31    23808.31
      TOP : SMAX=   -3232.73 SMIN=  -23808.31 TMAX=   10287.79 ANGLE=  2.8
      BOTT: SMAX=   23808.31 SMIN=    3232.73 TMAX=   10287.79 ANGLE=-87.2
    99      1        -1.23       -0.23      -77.92     -328.01       12.92
                  19844.15    19844.15        0.00        0.00        0.00
                  21911.50    21911.50
      TOP : SMAX=   -5150.16 SMIN=  -21911.50 TMAX=    8380.67 ANGLE=  3.0
      BOTT: SMAX=   21911.50 SMIN=    5150.16 TMAX=    8380.67 ANGLE=-87.0
   100      1        -0.78       -0.21     -100.93     -305.20        9.76
                  17989.87    17989.87        0.00        0.00        0.00
                  20377.92    20377.92
      TOP : SMAX=   -6697.43 SMIN=  -20377.92 TMAX=    6840.24 ANGLE=  2.7
      BOTT: SMAX=   20377.92 SMIN=    6697.43 TMAX=    6840.24 ANGLE=-87.3
```

```
   101      1        -0.42       -0.16     -116.93     -289.32        6.05
                  16820.56    16820.56        0.00        0.00        0.00
                  19302.28    19302.28
      TOP : SMAX=   -7780.89 SMIN=  -19302.28 TMAX=    5760.70 ANGLE=  2.0
      BOTT: SMAX=   19302.28 SMIN=    7780.89 TMAX=    5760.70 ANGLE=-88.0
   102      1        -0.13       -0.13     -125.11     -281.19        2.05
                  16269.07    16269.07        0.00        0.00        0.00
                  18747.84    18747.84
      TOP : SMAX=   -8338.78 SMIN=  -18747.84 TMAX=    5204.53 ANGLE=  0.8
      BOTT: SMAX=   18747.84 SMIN=    8338.78 TMAX=    5204.53 ANGLE=-89.2
   103      1         0.13       -0.13     -125.11     -281.19       -2.05
                  16269.07    16269.07        0.00        0.00        0.00
                  18747.84    18747.84
      TOP : SMAX=   -8338.78 SMIN=  -18747.84 TMAX=    5204.53 ANGLE= -0.8
      BOTT: SMAX=   18747.84 SMIN=    8338.78 TMAX=    5204.53 ANGLE= 89.2
   104      1         0.42       -0.16     -116.93     -289.32       -6.05
                  16820.56    16820.56        0.00        0.00        0.00
                  19302.28    19302.28
      TOP : SMAX=   -7780.89 SMIN=  -19302.28 TMAX=    5760.70 ANGLE= -2.0
      BOTT: SMAX=   19302.28 SMIN=    7780.89 TMAX=    5760.70 ANGLE= 88.0
     STAAD SPACE                                             -- PAGE NO.
7
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                      VONT       VONB         SX          SY          SXY
                      TRESCAT    TRESCAB
   105      1         0.78       -0.21     -100.93     -305.20       -9.76
                  17989.87    17989.87        0.00        0.00        0.00
                  20377.92    20377.92
      TOP : SMAX=   -6697.43 SMIN=  -20377.92 TMAX=    6840.24 ANGLE= -2.7
      BOTT: SMAX=   20377.92 SMIN=    6697.43 TMAX=    6840.24 ANGLE= 87.3
   106      1         1.23       -0.23      -77.92     -328.01      -12.92
                  19844.15    19844.15        0.00        0.00        0.00
                  21911.50    21911.50
      TOP : SMAX=   -5150.16 SMIN=  -21911.50 TMAX=    8380.67 ANGLE= -3.0
      BOTT: SMAX=   21911.50 SMIN=    5150.16 TMAX=    8380.67 ANGLE= 87.0
   107      1         1.69       -0.20      -49.25     -356.37      -15.27
                  22367.84    22367.84        0.00        0.00        0.00
                  23808.31    23808.31
      TOP : SMAX=   -3232.73 SMIN=  -23808.31 TMAX=   10287.79 ANGLE= -2.8
      BOTT: SMAX=   23808.31 SMIN=    3232.73 TMAX=   10287.79 ANGLE= 87.2
   108      1         2.00       -0.08      -16.85     -388.37      -16.53
                  25419.74    25419.74        0.00        0.00        0.00
                  25939.96    25939.96
      TOP : SMAX=   -1074.52 SMIN=  -25939.96 TMAX=   12432.72 ANGLE= -2.5
      BOTT: SMAX=   25939.96 SMIN=    1074.52 TMAX=   12432.72 ANGLE= 87.5
   114      1        -0.09       -0.36     -133.36     -273.00        6.23
                  15779.24    15779.24        0.00        0.00        0.00
                  18218.24    18218.24
      TOP : SMAX=   -8871.89 SMIN=  -18218.24 TMAX=    4673.17 ANGLE=  2.5
      BOTT: SMAX=   18218.24 SMIN=    8871.89 TMAX=    4673.17 ANGLE=-87.5
   126      1        -0.02       -0.49     -150.19     -256.25       10.66
                  14918.57    14918.57        0.00        0.00        0.00
                  17154.23    17154.23
      TOP : SMAX=   -9942.06 SMIN=  -17154.23 TMAX=    3606.09 ANGLE=  5.7
      BOTT: SMAX=   17154.23 SMIN=    9942.06 TMAX=    3606.09 ANGLE=-84.3
```

```
    138      1         0.12        -0.39     -176.20     -230.35       15.40
                   14020.85     14020.85        0.00        0.00        0.00
                   15628.28     15628.28
       TOP : SMAX=  -11474.99 SMIN=  -15628.28 TMAX=    2076.64 ANGLE= 14.8
       BOTT: SMAX=   15628.28 SMIN=   11474.99 TMAX=    2076.64 ANGLE=-75.2
    150      1         0.38         0.06     -211.95     -194.66       20.36
                   13792.44     13792.44        0.00        0.00        0.00
                   15028.62     15028.62
       TOP : SMAX=  -12079.03 SMIN=  -15028.62 TMAX=    1474.80 ANGLE= 56.5
       BOTT: SMAX=   15028.62 SMIN=   12079.03 TMAX=    1474.80 ANGLE=-33.5
      STAAD SPACE                                             -- PAGE NO.
8
   ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
   ----------------
            STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
  ELEMENT  LOAD       SQX        SQY          MX           MY          MXY
                      VONT       VONB         SX           SY          SXY
                      TRESCAT    TRESCAB
    162      1         0.67         1.03     -257.59     -148.94       25.19
                   15213.03     15213.03        0.00        0.00        0.00
                   17542.96     17542.96
       TOP : SMAX=   -9558.83 SMIN=  -17542.96 TMAX=    3992.07 ANGLE= 77.6
       BOTT: SMAX=   17542.96 SMIN=    9558.83 TMAX=    3992.07 ANGLE=-12.4
    174      1         0.74         2.75     -312.19      -93.95       29.19
                   18798.88     18798.88        0.00        0.00        0.00
                   21068.45     21068.45
       TOP : SMAX=   -6007.83 SMIN=  -21068.45 TMAX=    7530.31 ANGLE= 82.5
       BOTT: SMAX=   21068.45 SMIN=    6007.83 TMAX=    7530.31 ANGLE= -7.5
    186      1         0.34         4.36     -373.27      -32.17       31.51
                   24160.11     24160.11        0.00        0.00        0.00
                   25076.87     25076.87
       TOP : SMAX=   -1951.93 SMIN=  -25076.87 TMAX=   11562.47 ANGLE= 84.8
       BOTT: SMAX=   25076.87 SMIN=    1951.93 TMAX=   11562.47 ANGLE= -5.2
          **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
            PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
             STRESS        STRESS        STRESS        STRESS        STRESS
          2.593996E+04 -2.593996E+04  1.243272E+04  2.541974E+04  2.593996E+04
 PLATE NO.       97            97            97            97            97
 CASE  NO.        1             1             1             1             1
```

V. Warped Surface Displacements

To find the displacements at the free end of a warped cantilever plate due to in-plane load and out of plane loads.

Reference

MacNeal, R.H. and Harder, R.C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*, 1985.

Problem

The finite element model is as shown below: Find the displacements at the tip in the direction of the loads. Loading is unit forces at the free end: in plane and out of plane.

E = 29,000.0 ksi.

L = 12.0 in.

B = 1.1 in.

t = 0.22 in.

Twist = 90° (root to tip)

Poisson's ratio = 0.22

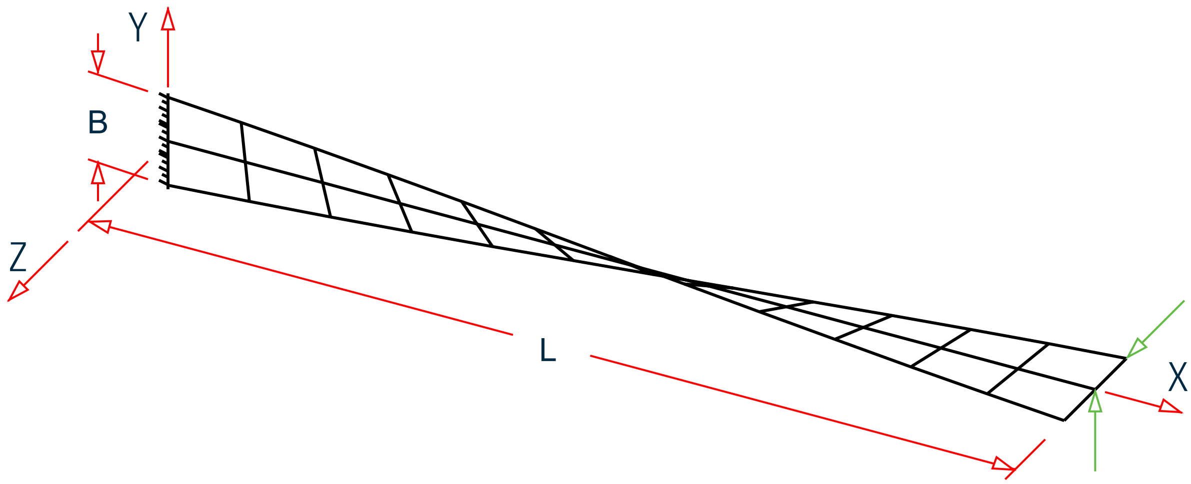


Figure 418: Model of warped, cantilever plate

Comparison

Table 443: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
δ due to in-plane load (in)	$5.424(10)^{-3}$	$5.590(10)^{-3}$	3.1%	Instead of using triangular element, a more advanced element type could to be used. Also the mesh size could be reduced to get closer result in comparison the theoretical value.
δ due to out-of-plane load (in)	$1.754(10)^{-3}$	$1.950(10)^{-3}$	11.2%	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\04 Plates Shells\Warped Surface Displacements.STD` is typically installed with the program.

```
STAAD SPACE :A WARPED CANTILEVER PLATE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
* REFERENCE: MACNEAL AND HARDER, A PROPOSED STANDARD SET OF PROBLEMS
*            TO TEST FINITE ELEMENT ACCURACY,
*            FINITE ELEMENT IN ANALYSIS AND DESIGN 1, NORTH HOLLAND
*            1985
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 -0.55 0; 2 1 -0.545 -0.072; 3 2 -0.531 -0.142; 4 3 -0.508 -0.21;
5 4 -0.476 -0.275; 6 5 -0.436 -0.335; 7 6 -0.389 -0.389;
8 7 -0.335 -0.436; 9 8 -0.275 -0.476; 10 9 -0.21 -0.508;
11 10 -0.142 -0.531; 12 11 -0.072 -0.545; 13 12 0 -0.55; 14 0 0 0;
15 1 0 0; 16 2 0 0; 17 3 0 0; 18 4 0 0; 19 5 0 0; 20 6 0 0; 21 7 0 0;
22 8 0 0; 23 9 0 0; 24 10 0 0; 25 11 0 0; 26 12 0 0; 27 0 0.55 0;
28 1 0.545 0.072; 29 2 0.531 0.142; 30 3 0.508 0.21; 31 4 0.476 0.275;
32 5 0.436 0.335; 33 6 0.389 0.389; 34 7 0.335 0.436; 35 8 0.275 0.476;
36 9 0.21 0.508; 37 10 0.142 0.531; 38 11 0.072 0.545; 39 12 0 0.55;
ELEMENT INCIDENCES SHELL
1 1 2 15; 2 15 14 1; 3 14 15 28; 4 28 27 14; 5 2 3 16; 6 16 15 2;
7 15 16 29; 8 29 28 15; 9 3 4 17; 10 17 16 3; 11 16 17 30; 12 30 29 16;
13 4 5 18; 14 18 17 4; 15 17 18 31; 16 31 30 17; 17 5 6 19; 18 19 18 5;
19 18 19 32; 20 32 31 18; 21 6 7 20; 22 20 19 6; 23 19 20 33;
24 33 32 19; 25 7 8 21; 26 21 20 7; 27 20 21 34; 28 34 33 20; 29 8 9 22;
30 22 21 8; 31 21 22 35; 32 35 34 21; 33 9 10 23; 34 23 22 9;
35 22 23 36; 36 36 35 22; 37 36 37 24; 38 24 23 36; 39 23 24 11;
40 11 10 23; 41 37 38 25; 42 25 24 37; 43 24 25 12; 44 12 11 24;
45 38 39 26; 46 26 25 38; 47 25 26 13; 48 13 12 25;
ELEMENT PROPERTY
1 TO 48 THICKNESS 0.32
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.9e+07
POISSON 0.22
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 14 27 FIXED
LOAD 1 UNIT LOAD AT TIP, OUT OF PLANE
JOINT LOAD
13 39 FY 0.25
26 FY 0.5
LOAD 2 UNIT LOAD AT TIP, IN PLANE
JOINT LOAD
13 39 FZ 0.25
26 FZ 0.5
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 13 26 39
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     13    1  -0.00015   0.00202  -0.00195  -0.00000   0.00022   0.00036
           2   0.00035  -0.00195   0.00559   0.00000  -0.00060  -0.00028
     26    1  -0.00000   0.00202  -0.00195  -0.00000   0.00022   0.00036
           2   0.00000  -0.00195   0.00559   0.00000  -0.00060  -0.00028
     39    1   0.00015   0.00202  -0.00195  -0.00000   0.00022   0.00035
           2  -0.00034  -0.00195   0.00559   0.00000  -0.00060  -0.00028
```

V.05 Solids

V. Cantilever Beam End Displacement 1

To find the displacement at the free end of a cantilever beam modeled with solid elements.

Reference

Hand calculation.

Problem

Calculate the maximum displacement of a cantilever beam due to a concentrated load at the free end

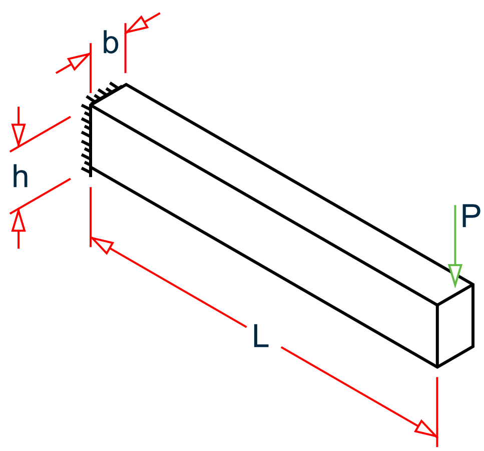


Figure 419: Entire model

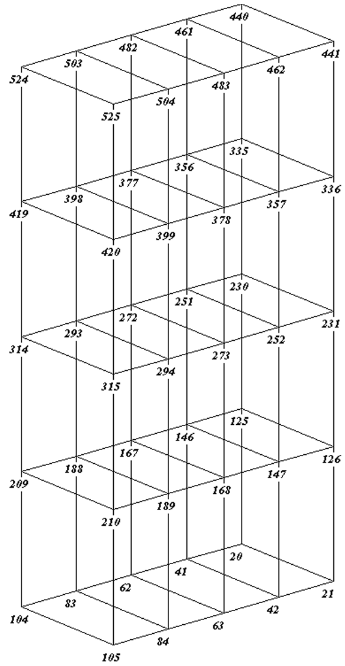


Figure 420: Free end section with node numbers

L = 10 in

A = 2 in^2

P = 300 lb

l = 2/3 in.^{4}

E = 29,000 ksi

v = 0.3

Hand Calculation

$\delta_{bend} = PL^3/(3EI) = 300(10)^3/\{3[29(10)^6](2/3)\} = 0.00517$ in

$\delta_{shear} = 12/5*(1+v)PL/AE = 12/5*(1+0.3)(300)(10)/[29(10)^6(2)] = 0.00016$ in

$\delta = \delta_{bend} + \delta_{shear} = 0.00517 + 0.00016 = 0.00533$ in

Comparison

Table 444: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Deflection, δ, (in)	0.00533	0.00529	< 1.0%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\05 Solids\Cantilever Beam End Displacement 1.STD` is typically installed with the program.

```
STAAD SPACE : A CANTILEVER BEAM WITH SOLID ELEMENTS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0.5 0 0; 3 1 0 0; 4 1.5 0 0; 5 2 0 0; 6 2.5 0 0; 7 3 0 0;
8 3.5 0 0; 9 4 0 0; 10 4.5 0 0; 11 5 0 0; 12 5.5 0 0; 13 6 0 0;
14 6.5 0 0; 15 7 0 0; 16 7.5 0 0; 17 8 0 0; 18 8.5 0 0; 19 9 0 0;
20 9.5 0 0; 21 10 0 0; 22 0 0 0.25; 23 0.5 0 0.25; 24 1 0 0.25;
25 1.5 0 0.25; 26 2 0 0.25; 27 2.5 0 0.25; 28 3 0 0.25; 29 3.5 0 0.25;
30 4 0 0.25; 31 4.5 0 0.25; 32 5 0 0.25; 33 5.5 0 0.25; 34 6 0 0.25;
35 6.5 0 0.25; 36 7 0 0.25; 37 7.5 0 0.25; 38 8 0 0.25; 39 8.5 0 0.25;
40 9 0 0.25; 41 9.5 0 0.25; 42 10 0 0.25; 43 0 0 0.5; 44 0.5 0 0.5;
45 1 0 0.5; 46 1.5 0 0.5; 47 2 0 0.5; 48 2.5 0 0.5; 49 3 0 0.5;
50 3.5 0 0.5; 51 4 0 0.5; 52 4.5 0 0.5; 53 5 0 0.5; 54 5.5 0 0.5;
55 6 0 0.5; 56 6.5 0 0.5; 57 7 0 0.5; 58 7.5 0 0.5; 59 8 0 0.5;
60 8.5 0 0.5; 61 9 0 0.5; 62 9.5 0 0.5; 63 10 0 0.5; 64 0 0 0.75;
65 0.5 0 0.75; 66 1 0 0.75; 67 1.5 0 0.75; 68 2 0 0.75; 69 2.5 0 0.75;
70 3 0 0.75; 71 3.5 0 0.75; 72 4 0 0.75; 73 4.5 0 0.75; 74 5 0 0.75;
75 5.5 0 0.75; 76 6 0 0.75; 77 6.5 0 0.75; 78 7 0 0.75; 79 7.5 0 0.75;
80 8 0 0.75; 81 8.5 0 0.75; 82 9 0 0.75; 83 9.5 0 0.75; 84 10 0 0.75;
85 0 0 1; 86 0.5 0 1; 87 1 0 1; 88 1.5 0 1; 89 2 0 1; 90 2.5 0 1;
91 3 0 1; 92 3.5 0 1; 93 4 0 1; 94 4.5 0 1; 95 5 0 1; 96 5.5 0 1;
97 6 0 1; 98 6.5 0 1; 99 7 0 1; 100 7.5 0 1; 101 8 0 1; 102 8.5 0 1;
103 9 0 1; 104 9.5 0 1; 105 10 0 1; 106 0 0.5 0; 107 0.5 0.5 0;
108 1 0.5 0; 109 1.5 0.5 0; 110 2 0.5 0; 111 2.5 0.5 0; 112 3 0.5 0;
113 3.5 0.5 0; 114 4 0.5 0; 115 4.5 0.5 0; 116 5 0.5 0; 117 5.5 0.5 0;
118 6 0.5 0; 119 6.5 0.5 0; 120 7 0.5 0; 121 7.5 0.5 0; 122 8 0.5 0;
123 8.5 0.5 0; 124 9 0.5 0; 125 9.5 0.5 0; 126 10 0.5 0; 127 0 0.5 0.25;
128 0.5 0.5 0.25; 129 1 0.5 0.25; 130 1.5 0.5 0.25; 131 2 0.5 0.25;
132 2.5 0.5 0.25; 133 3 0.5 0.25; 134 3.5 0.5 0.25; 135 4 0.5 0.25;
136 4.5 0.5 0.25; 137 5 0.5 0.25; 138 5.5 0.5 0.25; 139 6 0.5 0.25;
140 6.5 0.5 0.25; 141 7 0.5 0.25; 142 7.5 0.5 0.25; 143 8 0.5 0.25;
144 8.5 0.5 0.25; 145 9 0.5 0.25; 146 9.5 0.5 0.25; 147 10 0.5 0.25;
148 0 0.5 0.5; 149 0.5 0.5 0.5; 150 1 0.5 0.5; 151 1.5 0.5 0.5;
152 2 0.5 0.5; 153 2.5 0.5 0.5; 154 3 0.5 0.5; 155 3.5 0.5 0.5;
156 4 0.5 0.5; 157 4.5 0.5 0.5; 158 5 0.5 0.5; 159 5.5 0.5 0.5;
160 6 0.5 0.5; 161 6.5 0.5 0.5; 162 7 0.5 0.5; 163 7.5 0.5 0.5;
164 8 0.5 0.5; 165 8.5 0.5 0.5; 166 9 0.5 0.5; 167 9.5 0.5 0.5;
168 10 0.5 0.5; 169 0 0.5 0.75; 170 0.5 0.5 0.75; 171 1 0.5 0.75;
172 1.5 0.5 0.75; 173 2 0.5 0.75; 174 2.5 0.5 0.75; 175 3 0.5 0.75;
176 3.5 0.5 0.75; 177 4 0.5 0.75; 178 4.5 0.5 0.75; 179 5 0.5 0.75;
180 5.5 0.5 0.75; 181 6 0.5 0.75; 182 6.5 0.5 0.75; 183 7 0.5 0.75;
184 7.5 0.5 0.75; 185 8 0.5 0.75; 186 8.5 0.5 0.75; 187 9 0.5 0.75;
188 9.5 0.5 0.75; 189 10 0.5 0.75; 190 0 0.5 1; 191 0.5 0.5 1;
192 1 0.5 1; 193 1.5 0.5 1; 194 2 0.5 1; 195 2.5 0.5 1; 196 3 0.5 1;
197 3.5 0.5 1; 198 4 0.5 1; 199 4.5 0.5 1; 200 5 0.5 1; 201 5.5 0.5 1;
202 6 0.5 1; 203 6.5 0.5 1; 204 7 0.5 1; 205 7.5 0.5 1; 206 8 0.5 1;
207 8.5 0.5 1; 208 9 0.5 1; 209 9.5 0.5 1; 210 10 0.5 1; 211 0 1 0;
```

```
212 0.5 1 0; 213 1 1 0; 214 1.5 1 0; 215 2 1 0; 216 2.5 1 0; 217 3 1 0;
218 3.5 1 0; 219 4 1 0; 220 4.5 1 0; 221 5 1 0; 222 5.5 1 0; 223 6 1 0;
224 6.5 1 0; 225 7 1 0; 226 7.5 1 0; 227 8 1 0; 228 8.5 1 0; 229 9 1 0;
230 9.5 1 0; 231 10 1 0; 232 0 1 0.25; 233 0.5 1 0.25; 234 1 1 0.25;
235 1.5 1 0.25; 236 2 1 0.25; 237 2.5 1 0.25; 238 3 1 0.25;
239 3.5 1 0.25; 240 4 1 0.25; 241 4.5 1 0.25; 242 5 1 0.25;
243 5.5 1 0.25; 244 6 1 0.25; 245 6.5 1 0.25; 246 7 1 0.25;
247 7.5 1 0.25; 248 8 1 0.25; 249 8.5 1 0.25; 250 9 1 0.25;
251 9.5 1 0.25; 252 10 1 0.25; 253 0 1 0.5; 254 0.5 1 0.5; 255 1 1 0.5;
256 1.5 1 0.5; 257 2 1 0.5; 258 2.5 1 0.5; 259 3 1 0.5; 260 3.5 1 0.5;
261 4 1 0.5; 262 4.5 1 0.5; 263 5 1 0.5; 264 5.5 1 0.5; 265 6 1 0.5;
266 6.5 1 0.5; 267 7 1 0.5; 268 7.5 1 0.5; 269 8 1 0.5; 270 8.5 1 0.5;
271 9 1 0.5; 272 9.5 1 0.5; 273 10 1 0.5; 274 0 1 0.75; 275 0.5 1 0.75;
276 1 1 0.75; 277 1.5 1 0.75; 278 2 1 0.75; 279 2.5 1 0.75;
280 3 1 0.75; 281 3.5 1 0.75; 282 4 1 0.75; 283 4.5 1 0.75;
284 5 1 0.75; 285 5.5 1 0.75; 286 6 1 0.75; 287 6.5 1 0.75;
288 7 1 0.75; 289 7.5 1 0.75; 290 8 1 0.75; 291 8.5 1 0.75;
292 9 1 0.75; 293 9.5 1 0.75; 294 10 1 0.75; 295 0 1 1; 296 0.5 1 1;
297 1 1 1; 298 1.5 1 1; 299 2 1 1; 300 2.5 1 1; 301 3 1 1; 302 3.5 1 1;
303 4 1 1; 304 4.5 1 1; 305 5 1 1; 306 5.5 1 1; 307 6 1 1; 308 6.5 1 1;
309 7 1 1; 310 7.5 1 1; 311 8 1 1; 312 8.5 1 1; 313 9 1 1; 314 9.5 1 1;
315 10 1 1; 316 0 1.5 0; 317 0.5 1.5 0; 318 1 1.5 0; 319 1.5 1.5 0;
320 2 1.5 0; 321 2.5 1.5 0; 322 3 1.5 0; 323 3.5 1.5 0; 324 4 1.5 0;
325 4.5 1.5 0; 326 5 1.5 0; 327 5.5 1.5 0; 328 6 1.5 0; 329 6.5 1.5 0;
330 7 1.5 0; 331 7.5 1.5 0; 332 8 1.5 0; 333 8.5 1.5 0; 334 9 1.5 0;
335 9.5 1.5 0; 336 10 1.5 0; 337 0 1.5 0.25; 338 0.5 1.5 0.25;
339 1 1.5 0.25; 340 1.5 1.5 0.25; 341 2 1.5 0.25; 342 2.5 1.5 0.25;
343 3 1.5 0.25; 344 3.5 1.5 0.25; 345 4 1.5 0.25; 346 4.5 1.5 0.25;
347 5 1.5 0.25; 348 5.5 1.5 0.25; 349 6 1.5 0.25; 350 6.5 1.5 0.25;
351 7 1.5 0.25; 352 7.5 1.5 0.25; 353 8 1.5 0.25; 354 8.5 1.5 0.25;
355 9 1.5 0.25; 356 9.5 1.5 0.25; 357 10 1.5 0.25; 358 0 1.5 0.5;
359 0.5 1.5 0.5; 360 1 1.5 0.5; 361 1.5 1.5 0.5; 362 2 1.5 0.5;
363 2.5 1.5 0.5; 364 3 1.5 0.5; 365 3.5 1.5 0.5; 366 4 1.5 0.5;
367 4.5 1.5 0.5; 368 5 1.5 0.5; 369 5.5 1.5 0.5; 370 6 1.5 0.5;
371 6.5 1.5 0.5; 372 7 1.5 0.5; 373 7.5 1.5 0.5; 374 8 1.5 0.5;
375 8.5 1.5 0.5; 376 9 1.5 0.5; 377 9.5 1.5 0.5; 378 10 1.5 0.5;
379 0 1.5 0.75; 380 0.5 1.5 0.75; 381 1 1.5 0.75; 382 1.5 1.5 0.75;
383 2 1.5 0.75; 384 2.5 1.5 0.75; 385 3 1.5 0.75; 386 3.5 1.5 0.75;
387 4 1.5 0.75; 388 4.5 1.5 0.75; 389 5 1.5 0.75; 390 5.5 1.5 0.75;
391 6 1.5 0.75; 392 6.5 1.5 0.75; 393 7 1.5 0.75; 394 7.5 1.5 0.75;
395 8 1.5 0.75; 396 8.5 1.5 0.75; 397 9 1.5 0.75; 398 9.5 1.5 0.75;
399 10 1.5 0.75; 400 0 1.5 1; 401 0.5 1.5 1; 402 1 1.5 1; 403 1.5 1.5 1;
404 2 1.5 1; 405 2.5 1.5 1; 406 3 1.5 1; 407 3.5 1.5 1; 408 4 1.5 1;
409 4.5 1.5 1; 410 5 1.5 1; 411 5.5 1.5 1; 412 6 1.5 1; 413 6.5 1.5 1;
414 7 1.5 1; 415 7.5 1.5 1; 416 8 1.5 1; 417 8.5 1.5 1; 418 9 1.5 1;
419 9.5 1.5 1; 420 10 1.5 1; 421 0 2 0; 422 0.5 2 0; 423 1 2 0;
424 1.5 2 0; 425 2 2 0; 426 2.5 2 0; 427 3 2 0; 428 3.5 2 0; 429 4 2 0;
430 4.5 2 0; 431 5 2 0; 432 5.5 2 0; 433 6 2 0; 434 6.5 2 0; 435 7 2 0;
436 7.5 2 0; 437 8 2 0; 438 8.5 2 0; 439 9 2 0; 440 9.5 2 0; 441 10 2 0;
442 0 2 0.25; 443 0.5 2 0.25; 444 1 2 0.25; 445 1.5 2 0.25;
446 2 2 0.25; 447 2.5 2 0.25; 448 3 2 0.25; 449 3.5 2 0.25;
450 4 2 0.25; 451 4.5 2 0.25; 452 5 2 0.25; 453 5.5 2 0.25;
454 6 2 0.25; 455 6.5 2 0.25; 456 7 2 0.25; 457 7.5 2 0.25;
458 8 2 0.25; 459 8.5 2 0.25; 460 9 2 0.25; 461 9.5 2 0.25;
462 10 2 0.25; 463 0 2 0.5; 464 0.5 2 0.5; 465 1 2 0.5; 466 1.5 2 0.5;
467 2 2 0.5; 468 2.5 2 0.5; 469 3 2 0.5; 470 3.5 2 0.5; 471 4 2 0.5;
472 4.5 2 0.5; 473 5 2 0.5; 474 5.5 2 0.5; 475 6 2 0.5; 476 6.5 2 0.5;
477 7 2 0.5; 478 7.5 2 0.5; 479 8 2 0.5; 480 8.5 2 0.5; 481 9 2 0.5;
```

```
482 9.5 2 0.5; 483 10 2 0.5; 484 0 2 0.75; 485 0.5 2 0.75; 486 1 2 0.75;
487 1.5 2 0.75; 488 2 2 0.75; 489 2.5 2 0.75; 490 3 2 0.75;
491 3.5 2 0.75; 492 4 2 0.75; 493 4.5 2 0.75; 494 5 2 0.75;
495 5.5 2 0.75; 496 6 2 0.75; 497 6.5 2 0.75; 498 7 2 0.75;
499 7.5 2 0.75; 500 8 2 0.75; 501 8.5 2 0.75; 502 9 2 0.75;
503 9.5 2 0.75; 504 10 2 0.75; 505 0 2 1; 506 0.5 2 1; 507 1 2 1;
508 1.5 2 1; 509 2 2 1; 510 2.5 2 1; 511 3 2 1; 512 3.5 2 1; 513 4 2 1;
514 4.5 2 1; 515 5 2 1; 516 5.5 2 1; 517 6 2 1; 518 6.5 2 1; 519 7 2 1;
520 7.5 2 1; 521 8 2 1; 522 8.5 2 1; 523 9 2 1; 524 9.5 2 1; 525 10 2 1;
ELEMENT INCIDENCES SOLID
1 22 127 106 1 23 128 107 2; 2 23 128 107 2 24 129 108 3;
3 24 129 108 3 25 130 109 4; 4 25 130 109 4 26 131 110 5;
5 26 131 110 5 27 132 111 6; 6 27 132 111 6 28 133 112 7;
7 28 133 112 7 29 134 113 8; 8 29 134 113 8 30 135 114 9;
9 30 135 114 9 31 136 115 10; 10 31 136 115 10 32 137 116 11;
11 32 137 116 11 33 138 117 12; 12 33 138 117 12 34 139 118 13;
13 34 139 118 13 35 140 119 14; 14 35 140 119 14 36 141 120 15;
15 36 141 120 15 37 142 121 16; 16 37 142 121 16 38 143 122 17;
17 38 143 122 17 39 144 123 18; 18 39 144 123 18 40 145 124 19;
19 40 145 124 19 41 146 125 20; 20 41 146 125 20 42 147 126 21;
21 43 148 127 22 44 149 128 23; 22 44 149 128 23 45 150 129 24;
23 45 150 129 24 46 151 130 25; 24 46 151 130 25 47 152 131 26;
25 47 152 131 26 48 153 132 27; 26 48 153 132 27 49 154 133 28;
27 49 154 133 28 50 155 134 29; 28 50 155 134 29 51 156 135 30;
29 51 156 135 30 52 157 136 31; 30 52 157 136 31 53 158 137 32;
31 53 158 137 32 54 159 138 33; 32 54 159 138 33 55 160 139 34;
33 55 160 139 34 56 161 140 35; 34 56 161 140 35 57 162 141 36;
35 57 162 141 36 58 163 142 37; 36 58 163 142 37 59 164 143 38;
37 59 164 143 38 60 165 144 39; 38 60 165 144 39 61 166 145 40;
39 61 166 145 40 62 167 146 41; 40 62 167 146 41 63 168 147 42;
41 64 169 148 43 65 170 149 44; 42 65 170 149 44 66 171 150 45;
43 66 171 150 45 67 172 151 46; 44 67 172 151 46 68 173 152 47;
45 68 173 152 47 69 174 153 48; 46 69 174 153 48 70 175 154 49;
47 70 175 154 49 71 176 155 50; 48 71 176 155 50 72 177 156 51;
49 72 177 156 51 73 178 157 52; 50 73 178 157 52 74 179 158 53;
51 74 179 158 53 75 180 159 54; 52 75 180 159 54 76 181 160 55;
53 76 181 160 55 77 182 161 56; 54 77 182 161 56 78 183 162 57;
55 78 183 162 57 79 184 163 58; 56 79 184 163 58 80 185 164 59;
57 80 185 164 59 81 186 165 60; 58 81 186 165 60 82 187 166 61;
59 82 187 166 61 83 188 167 62; 60 83 188 167 62 84 189 168 63;
61 85 190 169 64 86 191 170 65; 62 86 191 170 65 87 192 171 66;
63 87 192 171 66 88 193 172 67; 64 88 193 172 67 89 194 173 68;
65 89 194 173 68 90 195 174 69; 66 90 195 174 69 91 196 175 70;
67 91 196 175 70 92 197 176 71; 68 92 197 176 71 93 198 177 72;
69 93 198 177 72 94 199 178 73; 70 94 199 178 73 95 200 179 74;
71 95 200 179 74 96 201 180 75; 72 96 201 180 75 97 202 181 76;
73 97 202 181 76 98 203 182 77; 74 98 203 182 77 99 204 183 78;
75 99 204 183 78 100 205 184 79; 76 100 205 184 79 101 206 185 80;
77 101 206 185 80 102 207 186 81; 78 102 207 186 81 103 208 187 82;
79 103 208 187 82 104 209 188 83; 80 104 209 188 83 105 210 189 84;
81 127 232 211 106 128 233 212 107; 82 128 233 212 107 129 234 213 108;
83 129 234 213 108 130 235 214 109; 84 130 235 214 109 131 236 215 110;
85 131 236 215 110 132 237 216 111; 86 132 237 216 111 133 238 217 112;
87 133 238 217 112 134 239 218 113; 88 134 239 218 113 135 240 219 114;
89 135 240 219 114 136 241 220 115; 90 136 241 220 115 137 242 221 116;
91 137 242 221 116 138 243 222 117; 92 138 243 222 117 139 244 223 118;
93 139 244 223 118 140 245 224 119; 94 140 245 224 119 141 246 225 120;
95 141 246 225 120 142 247 226 121; 96 142 247 226 121 143 248 227 122;
```

```
97 143 248 227 122 144 249 228 123; 98 144 249 228 123 145 250 229 124;
99 145 250 229 124 146 251 230 125; 100 146 251 230 125 147 252 231 126;
101 148 253 232 127 149 254 233 128;
102 149 254 233 128 150 255 234 129;
103 150 255 234 129 151 256 235 130;
104 151 256 235 130 152 257 236 131;
105 152 257 236 131 153 258 237 132;
106 153 258 237 132 154 259 238 133;
107 154 259 238 133 155 260 239 134;
108 155 260 239 134 156 261 240 135;
109 156 261 240 135 157 262 241 136;
110 157 262 241 136 158 263 242 137;
111 158 263 242 137 159 264 243 138;
112 159 264 243 138 160 265 244 139;
113 160 265 244 139 161 266 245 140;
114 161 266 245 140 162 267 246 141;
115 162 267 246 141 163 268 247 142;
116 163 268 247 142 164 269 248 143;
117 164 269 248 143 165 270 249 144;
118 165 270 249 144 166 271 250 145;
119 166 271 250 145 167 272 251 146;
120 167 272 251 146 168 273 252 147;
121 169 274 253 148 170 275 254 149;
122 170 275 254 149 171 276 255 150;
123 171 276 255 150 172 277 256 151;
124 172 277 256 151 173 278 257 152;
125 173 278 257 152 174 279 258 153;
126 174 279 258 153 175 280 259 154;
127 175 280 259 154 176 281 260 155;
128 176 281 260 155 177 282 261 156;
129 177 282 261 156 178 283 262 157;
130 178 283 262 157 179 284 263 158;
131 179 284 263 158 180 285 264 159;
132 180 285 264 159 181 286 265 160;
133 181 286 265 160 182 287 266 161;
134 182 287 266 161 183 288 267 162;
135 183 288 267 162 184 289 268 163;
136 184 289 268 163 185 290 269 164;
137 185 290 269 164 186 291 270 165;
138 186 291 270 165 187 292 271 166;
139 187 292 271 166 188 293 272 167;
140 188 293 272 167 189 294 273 168;
141 190 295 274 169 191 296 275 170;
142 191 296 275 170 192 297 276 171;
143 192 297 276 171 193 298 277 172;
144 193 298 277 172 194 299 278 173;
145 194 299 278 173 195 300 279 174;
146 195 300 279 174 196 301 280 175;
147 196 301 280 175 197 302 281 176;
148 197 302 281 176 198 303 282 177;
149 198 303 282 177 199 304 283 178;
150 199 304 283 178 200 305 284 179;
151 200 305 284 179 201 306 285 180;
152 201 306 285 180 202 307 286 181;
153 202 307 286 181 203 308 287 182;
154 203 308 287 182 204 309 288 183;
155 204 309 288 183 205 310 289 184;
156 205 310 289 184 206 311 290 185;
```

```
157 206 311 290 185 207 312 291 186;
158 207 312 291 186 208 313 292 187;
159 208 313 292 187 209 314 293 188;
160 209 314 293 188 210 315 294 189;
161 232 337 316 211 233 338 317 212;
162 233 338 317 212 234 339 318 213;
163 234 339 318 213 235 340 319 214;
164 235 340 319 214 236 341 320 215;
165 236 341 320 215 237 342 321 216;
166 237 342 321 216 238 343 322 217;
167 238 343 322 217 239 344 323 218;
168 239 344 323 218 240 345 324 219;
169 240 345 324 219 241 346 325 220;
170 241 346 325 220 242 347 326 221;
171 242 347 326 221 243 348 327 222;
172 243 348 327 222 244 349 328 223;
173 244 349 328 223 245 350 329 224;
174 245 350 329 224 246 351 330 225;
175 246 351 330 225 247 352 331 226;
176 247 352 331 226 248 353 332 227;
177 248 353 332 227 249 354 333 228;
178 249 354 333 228 250 355 334 229;
179 250 355 334 229 251 356 335 230;
180 251 356 335 230 252 357 336 231;
181 253 358 337 232 254 359 338 233;
182 254 359 338 233 255 360 339 234;
183 255 360 339 234 256 361 340 235;
184 256 361 340 235 257 362 341 236;
185 257 362 341 236 258 363 342 237;
186 258 363 342 237 259 364 343 238;
187 259 364 343 238 260 365 344 239;
188 260 365 344 239 261 366 345 240;
189 261 366 345 240 262 367 346 241;
190 262 367 346 241 263 368 347 242;
191 263 368 347 242 264 369 348 243;
192 264 369 348 243 265 370 349 244;
193 265 370 349 244 266 371 350 245;
194 266 371 350 245 267 372 351 246;
195 267 372 351 246 268 373 352 247;
196 268 373 352 247 269 374 353 248;
197 269 374 353 248 270 375 354 249;
198 270 375 354 249 271 376 355 250;
199 271 376 355 250 272 377 356 251;
200 272 377 356 251 273 378 357 252;
201 274 379 358 253 275 380 359 254;
202 275 380 359 254 276 381 360 255;
203 276 381 360 255 277 382 361 256;
204 277 382 361 256 278 383 362 257;
205 278 383 362 257 279 384 363 258;
206 279 384 363 258 280 385 364 259;
207 280 385 364 259 281 386 365 260;
208 281 386 365 260 282 387 366 261;
209 282 387 366 261 283 388 367 262;
210 283 388 367 262 284 389 368 263;
211 284 389 368 263 285 390 369 264;
212 285 390 369 264 286 391 370 265;
213 286 391 370 265 287 392 371 266;
214 287 392 371 266 288 393 372 267;
```

```
215 288 393 372 267 289 394 373 268;
216 289 394 373 268 290 395 374 269;
217 290 395 374 269 291 396 375 270;
218 291 396 375 270 292 397 376 271;
219 292 397 376 271 293 398 377 272;
220 293 398 377 272 294 399 378 273;
221 295 400 379 274 296 401 380 275;
222 296 401 380 275 297 402 381 276;
223 297 402 381 276 298 403 382 277;
224 298 403 382 277 299 404 383 278;
225 299 404 383 278 300 405 384 279;
226 300 405 384 279 301 406 385 280;
227 301 406 385 280 302 407 386 281;
228 302 407 386 281 303 408 387 282;
229 303 408 387 282 304 409 388 283;
230 304 409 388 283 305 410 389 284;
231 305 410 389 284 306 411 390 285;
232 306 411 390 285 307 412 391 286;
233 307 412 391 286 308 413 392 287;
234 308 413 392 287 309 414 393 288;
235 309 414 393 288 310 415 394 289;
236 310 415 394 289 311 416 395 290;
237 311 416 395 290 312 417 396 291;
238 312 417 396 291 313 418 397 292;
239 313 418 397 292 314 419 398 293;
240 314 419 398 293 315 420 399 294;
241 337 442 421 316 338 443 422 317;
242 338 443 422 317 339 444 423 318;
243 339 444 423 318 340 445 424 319;
244 340 445 424 319 341 446 425 320;
245 341 446 425 320 342 447 426 321;
246 342 447 426 321 343 448 427 322;
247 343 448 427 322 344 449 428 323;
248 344 449 428 323 345 450 429 324;
249 345 450 429 324 346 451 430 325;
250 346 451 430 325 347 452 431 326;
251 347 452 431 326 348 453 432 327;
252 348 453 432 327 349 454 433 328;
253 349 454 433 328 350 455 434 329;
254 350 455 434 329 351 456 435 330;
255 351 456 435 330 352 457 436 331;
256 352 457 436 331 353 458 437 332;
257 353 458 437 332 354 459 438 333;
258 354 459 438 333 355 460 439 334;
259 355 460 439 334 356 461 440 335;
260 356 461 440 335 357 462 441 336;
261 358 463 442 337 359 464 443 338;
262 359 464 443 338 360 465 444 339;
263 360 465 444 339 361 466 445 340;
264 361 466 445 340 362 467 446 341;
265 362 467 446 341 363 468 447 342;
266 363 468 447 342 364 469 448 343;
267 364 469 448 343 365 470 449 344;
268 365 470 449 344 366 471 450 345;
269 366 471 450 345 367 472 451 346;
270 367 472 451 346 368 473 452 347;
271 368 473 452 347 369 474 453 348;
272 369 474 453 348 370 475 454 349;
```

```
273 370 475 454 349 371 476 455 350;
274 371 476 455 350 372 477 456 351;
275 372 477 456 351 373 478 457 352;
276 373 478 457 352 374 479 458 353;
277 374 479 458 353 375 480 459 354;
278 375 480 459 354 376 481 460 355;
279 376 481 460 355 377 482 461 356;
280 377 482 461 356 378 483 462 357;
281 379 484 463 358 380 485 464 359;
282 380 485 464 359 381 486 465 360;
283 381 486 465 360 382 487 466 361;
284 382 487 466 361 383 488 467 362;
285 383 488 467 362 384 489 468 363;
286 384 489 468 363 385 490 469 364;
287 385 490 469 364 386 491 470 365;
288 386 491 470 365 387 492 471 366;
289 387 492 471 366 388 493 472 367;
290 388 493 472 367 389 494 473 368;
291 389 494 473 368 390 495 474 369;
292 390 495 474 369 391 496 475 370;
293 391 496 475 370 392 497 476 371;
294 392 497 476 371 393 498 477 372;
295 393 498 477 372 394 499 478 373;
296 394 499 478 373 395 500 479 374;
297 395 500 479 374 396 501 480 375;
298 396 501 480 375 397 502 481 376;
299 397 502 481 376 398 503 482 377;
300 398 503 482 377 399 504 483 378;
301 400 505 484 379 401 506 485 380;
302 401 506 485 380 402 507 486 381;
303 402 507 486 381 403 508 487 382;
304 403 508 487 382 404 509 488 383;
305 404 509 488 383 405 510 489 384;
306 405 510 489 384 406 511 490 385;
307 406 511 490 385 407 512 491 386;
308 407 512 491 386 408 513 492 387;
309 408 513 492 387 409 514 493 388;
310 409 514 493 388 410 515 494 389;
311 410 515 494 389 411 516 495 390;
312 411 516 495 390 412 517 496 391;
313 412 517 496 391 413 518 497 392;
314 413 518 497 392 414 519 498 393;
315 414 519 498 393 415 520 499 394;
316 415 520 499 394 416 521 500 395;
317 416 521 500 395 417 522 501 396;
318 417 522 501 396 418 523 502 397;
319 418 523 502 397 419 524 503 398;
320 419 524 503 398 420 525 504 399;
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.9e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 106 211 316 421 FIXED
22 127 232 337 442 FIXED
```

```
43 148 253 358 463 FIXED
64 169 274 379 484 FIXED
85 190 295 400 505 FIXED
LOAD 1
JOINT LOAD
21 105 441 525 FY -75
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 21 105 273 441 525
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     21    1   -0.00078  -0.00529   0.00001   0.00000   0.00000   0.00000
    105    1   -0.00078  -0.00529  -0.00001   0.00000   0.00000   0.00000
    273    1    0.00000  -0.00525   0.00000   0.00000   0.00000   0.00000
    441    1    0.00078  -0.00529  -0.00001   0.00000   0.00000   0.00000
    525    1    0.00078  -0.00529   0.00001   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
```

V. Cantilever Beam End Displacement 2

To find the displacement at the free end and normal stresses at mid-span of a cantilever beam modeled with solid elements.

Hand Calculation

Displacement due to Load 1:

$$\delta_{LL} = \frac{PL}{AE} = \frac{1,\ 200(15)}{10 \times (10)^6(6)} = 0.0003 \text{ in}$$

Rotate due to Load 2:

$$\varphi_{L2} = TL/(c_2ab^3G)$$

where

a	=	long side of the cross section = 3 in
b	=	short side of the cross section = 2 in
$c2$	=	0.1958 for $a/b = 1.5$
G	=	$E/[2(1+v)] = 10(10)^3/(2(1+0.3)] = 3,846$ ksi

$$\varphi_{L2} = 2000(15)//[0.1958(3)(2)^3 3.846(10)^6] = 0.00166 \text{ rad}$$

Displacement due to Load 3:

$$\delta_{L3} = ML^2/(2EI) = 2500(15)^2/[2(10)(10)^7(4.5)] = 0.00625 \text{ in}$$

Displacement due to Load 4:

$$\delta_{bend} = PL^3/(3EI) = 1000(15)^3/\{3[10(10)^6](4.5)\} = 0.025 \text{ in}$$

$$\delta_{shear} = 12/5*(1+v)PL/AE = 12/5*(1+0.3)(1000)(15)/[(6)10(10)^6] = 0.00078 \text{ in}$$

$$\delta_{L4} = \delta_{bend} + \delta_{shear} = 0.025 + 0.00078 = 0.02578 \text{ in}$$

Stress at midspan due to Load 1:

$$\sigma_a = P/A = 1200/6 = 200 \text{ psi}$$

Stress at midspan due to Load 3:

$$\sigma_b = My/I = 2500(1.5)/4.5 = 833.33 \text{ psi}$$

Stress at midspan due to Load 4:

$$\sigma_b = My/I = 7.5(1000)(1.5)/4.5 = 2,500 \text{ psi}$$

Comparison

Table 445: Comparison of results

Result Type		Theory	STAAD.Pro	Difference	Comments
Maximum Displacement, δ (in)	LC1	0.00030	0.00033	10%	The theoretical results from classical Beam theory was compared with the results from the model with Solid Elements - hence the difference in results.
	LC3	0.00625	0.00621	0.6%	
	LC4	0.02578	0.02537	1.6%	
Maximum Rotation, φ (rad)	LC2	0.00166	0.00149	10.2%	
Normal Stress at Midspan* (psi)	LC1	200.0	200.004	none	
	LC3	833.3	833.325	none	
	LC4	2,500	2,630.409	5.2%	

Note: (*) Stresses computed at Node no. 32 of solid no. 10.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\05 Solids\Cantilever Beam End Displacement 2.STD` is typically installed with the program.

```
STAAD SPACE : A CANTILEVER BEAM WITH SOLID ELEMENTS
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 0.75 0 0; 3 1.5 0 0; 4 2.25 0 0; 5 3 0 0; 6 3.75 0 0;
7 4.5 0 0; 8 5.25 0 0; 9 6 0 0; 10 6.75 0 0; 11 7.5 0 0; 12 8.25 0 0;
13 9 0 0; 14 9.75 0 0; 15 10.5 0 0; 16 11.25 0 0; 17 12 0 0;
18 12.75 0 0; 19 13.5 0 0; 20 14.25 0 0; 21 15 0 0; 22 0 0 0.5;
23 0.75 0 0.5; 24 1.5 0 0.5; 25 2.25 0 0.5; 26 3 0 0.5; 27 3.75 0 0.5;
```

```
28 4.5 0 0.5; 29 5.25 0 0.5; 30 6 0 0.5; 31 6.75 0 0.5; 32 7.5 0 0.5;
33 8.25 0 0.5; 34 9 0 0.5; 35 9.75 0 0.5; 36 10.5 0 0.5; 37 11.25 0 0.5;
38 12 0 0.5; 39 12.75 0 0.5; 40 13.5 0 0.5; 41 14.25 0 0.5; 42 15 0 0.5;
43 0 0 1; 44 0.75 0 1; 45 1.5 0 1; 46 2.25 0 1; 47 3 0 1; 48 3.75 0 1;
49 4.5 0 1; 50 5.25 0 1; 51 6 0 1; 52 6.75 0 1; 53 7.5 0 1; 54 8.25 0 1;
55 9 0 1; 56 9.75 0 1; 57 10.5 0 1; 58 11.25 0 1; 59 12 0 1;
60 12.75 0 1; 61 13.5 0 1; 62 14.25 0 1; 63 15 0 1; 64 0 0 1.5;
65 0.75 0 1.5; 66 1.5 0 1.5; 67 2.25 0 1.5; 68 3 0 1.5; 69 3.75 0 1.5;
70 4.5 0 1.5; 71 5.25 0 1.5; 72 6 0 1.5; 73 6.75 0 1.5; 74 7.5 0 1.5;
75 8.25 0 1.5; 76 9 0 1.5; 77 9.75 0 1.5; 78 10.5 0 1.5; 79 11.25 0 1.5;
80 12 0 1.5; 81 12.75 0 1.5; 82 13.5 0 1.5; 83 14.25 0 1.5; 84 15 0 1.5;
85 0 0 2; 86 0.75 0 2; 87 1.5 0 2; 88 2.25 0 2; 89 3 0 2; 90 3.75 0 2;
91 4.5 0 2; 92 5.25 0 2; 93 6 0 2; 94 6.75 0 2; 95 7.5 0 2; 96 8.25 0 2;
97 9 0 2; 98 9.75 0 2; 99 10.5 0 2; 100 11.25 0 2; 101 12 0 2;
102 12.75 0 2; 103 13.5 0 2; 104 14.25 0 2; 105 15 0 2; 106 0 0.75 0;
107 0.75 0.75 0; 108 1.5 0.75 0; 109 2.25 0.75 0; 110 3 0.75 0;
111 3.75 0.75 0; 112 4.5 0.75 0; 113 5.25 0.75 0; 114 6 0.75 0;
115 6.75 0.75 0; 116 7.5 0.75 0; 117 8.25 0.75 0; 118 9 0.75 0;
119 9.75 0.75 0; 120 10.5 0.75 0; 121 11.25 0.75 0; 122 12 0.75 0;
123 12.75 0.75 0; 124 13.5 0.75 0; 125 14.25 0.75 0; 126 15 0.75 0;
127 0 0.75 0.5; 128 0.75 0.75 0.5; 129 1.5 0.75 0.5; 130 2.25 0.75 0.5;
131 3 0.75 0.5; 132 3.75 0.75 0.5; 133 4.5 0.75 0.5; 134 5.25 0.75 0.5;
135 6 0.75 0.5; 136 6.75 0.75 0.5; 137 7.5 0.75 0.5; 138 8.25 0.75 0.5;
139 9 0.75 0.5; 140 9.75 0.75 0.5; 141 10.5 0.75 0.5;
142 11.25 0.75 0.5; 143 12 0.75 0.5; 144 12.75 0.75 0.5;
145 13.5 0.75 0.5; 146 14.25 0.75 0.5; 147 15 0.75 0.5; 148 0 0.75 1;
149 0.75 0.75 1; 150 1.5 0.75 1; 151 2.25 0.75 1; 152 3 0.75 1;
153 3.75 0.75 1; 154 4.5 0.75 1; 155 5.25 0.75 1; 156 6 0.75 1;
157 6.75 0.75 1; 158 7.5 0.75 1; 159 8.25 0.75 1; 160 9 0.75 1;
161 9.75 0.75 1; 162 10.5 0.75 1; 163 11.25 0.75 1; 164 12 0.75 1;
165 12.75 0.75 1; 166 13.5 0.75 1; 167 14.25 0.75 1; 168 15 0.75 1;
169 0 0.75 1.5; 170 0.75 0.75 1.5; 171 1.5 0.75 1.5; 172 2.25 0.75 1.5;
173 3 0.75 1.5; 174 3.75 0.75 1.5; 175 4.5 0.75 1.5; 176 5.25 0.75 1.5;
177 6 0.75 1.5; 178 6.75 0.75 1.5; 179 7.5 0.75 1.5; 180 8.25 0.75 1.5;
181 9 0.75 1.5; 182 9.75 0.75 1.5; 183 10.5 0.75 1.5;
184 11.25 0.75 1.5; 185 12 0.75 1.5; 186 12.75 0.75 1.5;
187 13.5 0.75 1.5; 188 14.25 0.75 1.5; 189 15 0.75 1.5; 190 0 0.75 2;
191 0.75 0.75 2; 192 1.5 0.75 2; 193 2.25 0.75 2; 194 3 0.75 2;
195 3.75 0.75 2; 196 4.5 0.75 2; 197 5.25 0.75 2; 198 6 0.75 2;
199 6.75 0.75 2; 200 7.5 0.75 2; 201 8.25 0.75 2; 202 9 0.75 2;
203 9.75 0.75 2; 204 10.5 0.75 2; 205 11.25 0.75 2; 206 12 0.75 2;
207 12.75 0.75 2; 208 13.5 0.75 2; 209 14.25 0.75 2; 210 15 0.75 2;
211 0 1.5 0; 212 0.75 1.5 0; 213 1.5 1.5 0; 214 2.25 1.5 0; 215 3 1.5 0;
216 3.75 1.5 0; 217 4.5 1.5 0; 218 5.25 1.5 0; 219 6 1.5 0;
220 6.75 1.5 0; 221 7.5 1.5 0; 222 8.25 1.5 0; 223 9 1.5 0;
224 9.75 1.5 0; 225 10.5 1.5 0; 226 11.25 1.5 0; 227 12 1.5 0;
228 12.75 1.5 0; 229 13.5 1.5 0; 230 14.25 1.5 0; 231 15 1.5 0;
232 0 1.5 0.5; 233 0.75 1.5 0.5; 234 1.5 1.5 0.5; 235 2.25 1.5 0.5;
236 3 1.5 0.5; 237 3.75 1.5 0.5; 238 4.5 1.5 0.5; 239 5.25 1.5 0.5;
240 6 1.5 0.5; 241 6.75 1.5 0.5; 242 7.5 1.5 0.5; 243 8.25 1.5 0.5;
244 9 1.5 0.5; 245 9.75 1.5 0.5; 246 10.5 1.5 0.5; 247 11.25 1.5 0.5;
248 12 1.5 0.5; 249 12.75 1.5 0.5; 250 13.5 1.5 0.5; 251 14.25 1.5 0.5;
252 15 1.5 0.5; 253 0 1.5 1; 254 0.75 1.5 1; 255 1.5 1.5 1;
256 2.25 1.5 1; 257 3 1.5 1; 258 3.75 1.5 1; 259 4.5 1.5 1;
260 5.25 1.5 1; 261 6 1.5 1; 262 6.75 1.5 1; 263 7.5 1.5 1;
264 8.25 1.5 1; 265 9 1.5 1; 266 9.75 1.5 1; 267 10.5 1.5 1;
268 11.25 1.5 1; 269 12 1.5 1; 270 12.75 1.5 1; 271 13.5 1.5 1;
272 14.25 1.5 1; 273 15 1.5 1; 274 0 1.5 1.5; 275 0.75 1.5 1.5;
```

```
276 1.5 1.5 1.5; 277 2.25 1.5 1.5; 278 3 1.5 1.5; 279 3.75 1.5 1.5;
280 4.5 1.5 1.5; 281 5.25 1.5 1.5; 282 6 1.5 1.5; 283 6.75 1.5 1.5;
284 7.5 1.5 1.5; 285 8.25 1.5 1.5; 286 9 1.5 1.5; 287 9.75 1.5 1.5;
288 10.5 1.5 1.5; 289 11.25 1.5 1.5; 290 12 1.5 1.5; 291 12.75 1.5 1.5;
292 13.5 1.5 1.5; 293 14.25 1.5 1.5; 294 15 1.5 1.5; 295 0 1.5 2;
296 0.75 1.5 2; 297 1.5 1.5 2; 298 2.25 1.5 2; 299 3 1.5 2;
300 3.75 1.5 2; 301 4.5 1.5 2; 302 5.25 1.5 2; 303 6 1.5 2;
304 6.75 1.5 2; 305 7.5 1.5 2; 306 8.25 1.5 2; 307 9 1.5 2;
308 9.75 1.5 2; 309 10.5 1.5 2; 310 11.25 1.5 2; 311 12 1.5 2;
312 12.75 1.5 2; 313 13.5 1.5 2; 314 14.25 1.5 2; 315 15 1.5 2;
316 0 2.25 0; 317 0.75 2.25 0; 318 1.5 2.25 0; 319 2.25 2.25 0;
320 3 2.25 0; 321 3.75 2.25 0; 322 4.5 2.25 0; 323 5.25 2.25 0;
324 6 2.25 0; 325 6.75 2.25 0; 326 7.5 2.25 0; 327 8.25 2.25 0;
328 9 2.25 0; 329 9.75 2.25 0; 330 10.5 2.25 0; 331 11.25 2.25 0;
332 12 2.25 0; 333 12.75 2.25 0; 334 13.5 2.25 0; 335 14.25 2.25 0;
336 15 2.25 0; 337 0 2.25 0.5; 338 0.75 2.25 0.5; 339 1.5 2.25 0.5;
340 2.25 2.25 0.5; 341 3 2.25 0.5; 342 3.75 2.25 0.5; 343 4.5 2.25 0.5;
344 5.25 2.25 0.5; 345 6 2.25 0.5; 346 6.75 2.25 0.5; 347 7.5 2.25 0.5;
348 8.25 2.25 0.5; 349 9 2.25 0.5; 350 9.75 2.25 0.5; 351 10.5 2.25 0.5;
352 11.25 2.25 0.5; 353 12 2.25 0.5; 354 12.75 2.25 0.5;
355 13.5 2.25 0.5; 356 14.25 2.25 0.5; 357 15 2.25 0.5; 358 0 2.25 1;
359 0.75 2.25 1; 360 1.5 2.25 1; 361 2.25 2.25 1; 362 3 2.25 1;
363 3.75 2.25 1; 364 4.5 2.25 1; 365 5.25 2.25 1; 366 6 2.25 1;
367 6.75 2.25 1; 368 7.5 2.25 1; 369 8.25 2.25 1; 370 9 2.25 1;
371 9.75 2.25 1; 372 10.5 2.25 1; 373 11.25 2.25 1; 374 12 2.25 1;
375 12.75 2.25 1; 376 13.5 2.25 1; 377 14.25 2.25 1; 378 15 2.25 1;
379 0 2.25 1.5; 380 0.75 2.25 1.5; 381 1.5 2.25 1.5; 382 2.25 2.25 1.5;
383 3 2.25 1.5; 384 3.75 2.25 1.5; 385 4.5 2.25 1.5; 386 5.25 2.25 1.5;
387 6 2.25 1.5; 388 6.75 2.25 1.5; 389 7.5 2.25 1.5; 390 8.25 2.25 1.5;
391 9 2.25 1.5; 392 9.75 2.25 1.5; 393 10.5 2.25 1.5;
394 11.25 2.25 1.5; 395 12 2.25 1.5; 396 12.75 2.25 1.5;
397 13.5 2.25 1.5; 398 14.25 2.25 1.5; 399 15 2.25 1.5; 400 0 2.25 2;
401 0.75 2.25 2; 402 1.5 2.25 2; 403 2.25 2.25 2; 404 3 2.25 2;
405 3.75 2.25 2; 406 4.5 2.25 2; 407 5.25 2.25 2; 408 6 2.25 2;
409 6.75 2.25 2; 410 7.5 2.25 2; 411 8.25 2.25 2; 412 9 2.25 2;
413 9.75 2.25 2; 414 10.5 2.25 2; 415 11.25 2.25 2; 416 12 2.25 2;
417 12.75 2.25 2; 418 13.5 2.25 2; 419 14.25 2.25 2; 420 15 2.25 2;
421 0 3 0; 422 0.75 3 0; 423 1.5 3 0; 424 2.25 3 0; 425 3 3 0;
426 3.75 3 0; 427 4.5 3 0; 428 5.25 3 0; 429 6 3 0; 430 6.75 3 0;
431 7.5 3 0; 432 8.25 3 0; 433 9 3 0; 434 9.75 3 0; 435 10.5 3 0;
436 11.25 3 0; 437 12 3 0; 438 12.75 3 0; 439 13.5 3 0; 440 14.25 3 0;
441 15 3 0; 442 0 3 0.5; 443 0.75 3 0.5; 444 1.5 3 0.5; 445 2.25 3 0.5;
446 3 3 0.5; 447 3.75 3 0.5; 448 4.5 3 0.5; 449 5.25 3 0.5; 450 6 3 0.5;
451 6.75 3 0.5; 452 7.5 3 0.5; 453 8.25 3 0.5; 454 9 3 0.5;
455 9.75 3 0.5; 456 10.5 3 0.5; 457 11.25 3 0.5; 458 12 3 0.5;
459 12.75 3 0.5; 460 13.5 3 0.5; 461 14.25 3 0.5; 462 15 3 0.5;
463 0 3 1; 464 0.75 3 1; 465 1.5 3 1; 466 2.25 3 1; 467 3 3 1;
468 3.75 3 1; 469 4.5 3 1; 470 5.25 3 1; 471 6 3 1; 472 6.75 3 1;
473 7.5 3 1; 474 8.25 3 1; 475 9 3 1; 476 9.75 3 1; 477 10.5 3 1;
478 11.25 3 1; 479 12 3 1; 480 12.75 3 1; 481 13.5 3 1; 482 14.25 3 1;
483 15 3 1; 484 0 3 1.5; 485 0.75 3 1.5; 486 1.5 3 1.5; 487 2.25 3 1.5;
488 3 3 1.5; 489 3.75 3 1.5; 490 4.5 3 1.5; 491 5.25 3 1.5; 492 6 3 1.5;
493 6.75 3 1.5; 494 7.5 3 1.5; 495 8.25 3 1.5; 496 9 3 1.5;
497 9.75 3 1.5; 498 10.5 3 1.5; 499 11.25 3 1.5; 500 12 3 1.5;
501 12.75 3 1.5; 502 13.5 3 1.5; 503 14.25 3 1.5; 504 15 3 1.5;
505 0 3 2; 506 0.75 3 2; 507 1.5 3 2; 508 2.25 3 2; 509 3 3 2;
510 3.75 3 2; 511 4.5 3 2; 512 5.25 3 2; 513 6 3 2; 514 6.75 3 2;
515 7.5 3 2; 516 8.25 3 2; 517 9 3 2; 518 9.75 3 2; 519 10.5 3 2;
```

```
520 11.25 3 2; 521 12 3 2; 522 12.75 3 2; 523 13.5 3 2; 524 14.25 3 2;
525 15 3 2;
*
* SINCE SOLID-ONLY MODELS HAVE NO ROTATIONAL DOF AT THEIR NODES, ADD
* DUMMY MEMBERS TO CREATE THOSE DEGREES OF FREEDOM.
*
MEMBER INCIDENCES
1001 525 273; 1002 105 273; 1003 21 273; 1004 441 273;
ELEMENT INCIDENCES SOLID
1 22 127 106 1 23 128 107 2; 2 23 128 107 2 24 129 108 3;
3 24 129 108 3 25 130 109 4; 4 25 130 109 4 26 131 110 5;
5 26 131 110 5 27 132 111 6; 6 27 132 111 6 28 133 112 7;
7 28 133 112 7 29 134 113 8; 8 29 134 113 8 30 135 114 9;
9 30 135 114 9 31 136 115 10; 10 31 136 115 10 32 137 116 11;
11 32 137 116 11 33 138 117 12; 12 33 138 117 12 34 139 118 13;
13 34 139 118 13 35 140 119 14; 14 35 140 119 14 36 141 120 15;
15 36 141 120 15 37 142 121 16; 16 37 142 121 16 38 143 122 17;
17 38 143 122 17 39 144 123 18; 18 39 144 123 18 40 145 124 19;
19 40 145 124 19 41 146 125 20; 20 41 146 125 20 42 147 126 21;
21 43 148 127 22 44 149 128 23; 22 44 149 128 23 45 150 129 24;
23 45 150 129 24 46 151 130 25; 24 46 151 130 25 47 152 131 26;
25 47 152 131 26 48 153 132 27; 26 48 153 132 27 49 154 133 28;
27 49 154 133 28 50 155 134 29; 28 50 155 134 29 51 156 135 30;
29 51 156 135 30 52 157 136 31; 30 52 157 136 31 53 158 137 32;
31 53 158 137 32 54 159 138 33; 32 54 159 138 33 55 160 139 34;
33 55 160 139 34 56 161 140 35; 34 56 161 140 35 57 162 141 36;
35 57 162 141 36 58 163 142 37; 36 58 163 142 37 59 164 143 38;
37 59 164 143 38 60 165 144 39; 38 60 165 144 39 61 166 145 40;
39 61 166 145 40 62 167 146 41; 40 62 167 146 41 63 168 147 42;
41 64 169 148 43 65 170 149 44; 42 65 170 149 44 66 171 150 45;
43 66 171 150 45 67 172 151 46; 44 67 172 151 46 68 173 152 47;
45 68 173 152 47 69 174 153 48; 46 69 174 153 48 70 175 154 49;
47 70 175 154 49 71 176 155 50; 48 71 176 155 50 72 177 156 51;
49 72 177 156 51 73 178 157 52; 50 73 178 157 52 74 179 158 53;
51 74 179 158 53 75 180 159 54; 52 75 180 159 54 76 181 160 55;
53 76 181 160 55 77 182 161 56; 54 77 182 161 56 78 183 162 57;
55 78 183 162 57 79 184 163 58; 56 79 184 163 58 80 185 164 59;
57 80 185 164 59 81 186 165 60; 58 81 186 165 60 82 187 166 61;
59 82 187 166 61 83 188 167 62; 60 83 188 167 62 84 189 168 63;
61 85 190 169 64 86 191 170 65; 62 86 191 170 65 87 192 171 66;
63 87 192 171 66 88 193 172 67; 64 88 193 172 67 89 194 173 68;
65 89 194 173 68 90 195 174 69; 66 90 195 174 69 91 196 175 70;
67 91 196 175 70 92 197 176 71; 68 92 197 176 71 93 198 177 72;
69 93 198 177 72 94 199 178 73; 70 94 199 178 73 95 200 179 74;
71 95 200 179 74 96 201 180 75; 72 96 201 180 75 97 202 181 76;
73 97 202 181 76 98 203 182 77; 74 98 203 182 77 99 204 183 78;
75 99 204 183 78 100 205 184 79; 76 100 205 184 79 101 206 185 80;
77 101 206 185 80 102 207 186 81; 78 102 207 186 81 103 208 187 82;
79 103 208 187 82 104 209 188 83; 80 104 209 188 83 105 210 189 84;
81 127 232 211 106 128 233 212 107; 82 128 233 212 107 129 234 213 108;
83 129 234 213 108 130 235 214 109; 84 130 235 214 109 131 236 215 110;
85 131 236 215 110 132 237 216 111; 86 132 237 216 111 133 238 217 112;
87 133 238 217 112 134 239 218 113; 88 134 239 218 113 135 240 219 114;
89 135 240 219 114 136 241 220 115; 90 136 241 220 115 137 242 221 116;
91 137 242 221 116 138 243 222 117; 92 138 243 222 117 139 244 223 118;
93 139 244 223 118 140 245 224 119; 94 140 245 224 119 141 246 225 120;
95 141 246 225 120 142 247 226 121; 96 142 247 226 121 143 248 227 122;
97 143 248 227 122 144 249 228 123; 98 144 249 228 123 145 250 229 124;
```

```
99 145 250 229 124 146 251 230 125; 100 146 251 230 125 147 252 231 126;
101 148 253 232 127 149 254 233 128;
102 149 254 233 128 150 255 234 129;
103 150 255 234 129 151 256 235 130;
104 151 256 235 130 152 257 236 131;
105 152 257 236 131 153 258 237 132;
106 153 258 237 132 154 259 238 133;
107 154 259 238 133 155 260 239 134;
108 155 260 239 134 156 261 240 135;
109 156 261 240 135 157 262 241 136;
110 157 262 241 136 158 263 242 137;
111 158 263 242 137 159 264 243 138;
112 159 264 243 138 160 265 244 139;
113 160 265 244 139 161 266 245 140;
114 161 266 245 140 162 267 246 141;
115 162 267 246 141 163 268 247 142;
116 163 268 247 142 164 269 248 143;
117 164 269 248 143 165 270 249 144;
118 165 270 249 144 166 271 250 145;
119 166 271 250 145 167 272 251 146;
120 167 272 251 146 168 273 252 147;
121 169 274 253 148 170 275 254 149;
122 170 275 254 149 171 276 255 150;
123 171 276 255 150 172 277 256 151;
124 172 277 256 151 173 278 257 152;
125 173 278 257 152 174 279 258 153;
126 174 279 258 153 175 280 259 154;
127 175 280 259 154 176 281 260 155;
128 176 281 260 155 177 282 261 156;
129 177 282 261 156 178 283 262 157;
130 178 283 262 157 179 284 263 158;
131 179 284 263 158 180 285 264 159;
132 180 285 264 159 181 286 265 160;
133 181 286 265 160 182 287 266 161;
134 182 287 266 161 183 288 267 162;
135 183 288 267 162 184 289 268 163;
136 184 289 268 163 185 290 269 164;
137 185 290 269 164 186 291 270 165;
138 186 291 270 165 187 292 271 166;
139 187 292 271 166 188 293 272 167;
140 188 293 272 167 189 294 273 168;
141 190 295 274 169 191 296 275 170;
142 191 296 275 170 192 297 276 171;
143 192 297 276 171 193 298 277 172;
144 193 298 277 172 194 299 278 173;
145 194 299 278 173 195 300 279 174;
146 195 300 279 174 196 301 280 175;
147 196 301 280 175 197 302 281 176;
148 197 302 281 176 198 303 282 177;
149 198 303 282 177 199 304 283 178;
150 199 304 283 178 200 305 284 179;
151 200 305 284 179 201 306 285 180;
152 201 306 285 180 202 307 286 181;
153 202 307 286 181 203 308 287 182;
154 203 308 287 182 204 309 288 183;
155 204 309 288 183 205 310 289 184;
156 205 310 289 184 206 311 290 185;
157 206 311 290 185 207 312 291 186;
```

```
158 207 312 291 186 208 313 292 187;
159 208 313 292 187 209 314 293 188;
160 209 314 293 188 210 315 294 189;
161 232 337 316 211 233 338 317 212;
162 233 338 317 212 234 339 318 213;
163 234 339 318 213 235 340 319 214;
164 235 340 319 214 236 341 320 215;
165 236 341 320 215 237 342 321 216;
166 237 342 321 216 238 343 322 217;
167 238 343 322 217 239 344 323 218;
168 239 344 323 218 240 345 324 219;
169 240 345 324 219 241 346 325 220;
170 241 346 325 220 242 347 326 221;
171 242 347 326 221 243 348 327 222;
172 243 348 327 222 244 349 328 223;
173 244 349 328 223 245 350 329 224;
174 245 350 329 224 246 351 330 225;
175 246 351 330 225 247 352 331 226;
176 247 352 331 226 248 353 332 227;
177 248 353 332 227 249 354 333 228;
178 249 354 333 228 250 355 334 229;
179 250 355 334 229 251 356 335 230;
180 251 356 335 230 252 357 336 231;
181 253 358 337 232 254 359 338 233;
182 254 359 338 233 255 360 339 234;
183 255 360 339 234 256 361 340 235;
184 256 361 340 235 257 362 341 236;
185 257 362 341 236 258 363 342 237;
186 258 363 342 237 259 364 343 238;
187 259 364 343 238 260 365 344 239;
188 260 365 344 239 261 366 345 240;
189 261 366 345 240 262 367 346 241;
190 262 367 346 241 263 368 347 242;
191 263 368 347 242 264 369 348 243;
192 264 369 348 243 265 370 349 244;
193 265 370 349 244 266 371 350 245;
194 266 371 350 245 267 372 351 246;
195 267 372 351 246 268 373 352 247;
196 268 373 352 247 269 374 353 248;
197 269 374 353 248 270 375 354 249;
198 270 375 354 249 271 376 355 250;
199 271 376 355 250 272 377 356 251;
200 272 377 356 251 273 378 357 252;
201 274 379 358 253 275 380 359 254;
202 275 380 359 254 276 381 360 255;
203 276 381 360 255 277 382 361 256;
204 277 382 361 256 278 383 362 257;
205 278 383 362 257 279 384 363 258;
206 279 384 363 258 280 385 364 259;
207 280 385 364 259 281 386 365 260;
208 281 386 365 260 282 387 366 261;
209 282 387 366 261 283 388 367 262;
210 283 388 367 262 284 389 368 263;
211 284 389 368 263 285 390 369 264;
212 285 390 369 264 286 391 370 265;
213 286 391 370 265 287 392 371 266;
214 287 392 371 266 288 393 372 267;
215 288 393 372 267 289 394 373 268;
```

```
216 289 394 373 268 290 395 374 269;
217 290 395 374 269 291 396 375 270;
218 291 396 375 270 292 397 376 271;
219 292 397 376 271 293 398 377 272;
220 293 398 377 272 294 399 378 273;
221 295 400 379 274 296 401 380 275;
222 296 401 380 275 297 402 381 276;
223 297 402 381 276 298 403 382 277;
224 298 403 382 277 299 404 383 278;
225 299 404 383 278 300 405 384 279;
226 300 405 384 279 301 406 385 280;
227 301 406 385 280 302 407 386 281;
228 302 407 386 281 303 408 387 282;
229 303 408 387 282 304 409 388 283;
230 304 409 388 283 305 410 389 284;
231 305 410 389 284 306 411 390 285;
232 306 411 390 285 307 412 391 286;
233 307 412 391 286 308 413 392 287;
234 308 413 392 287 309 414 393 288;
235 309 414 393 288 310 415 394 289;
236 310 415 394 289 311 416 395 290;
237 311 416 395 290 312 417 396 291;
238 312 417 396 291 313 418 397 292;
239 313 418 397 292 314 419 398 293;
240 314 419 398 293 315 420 399 294;
241 337 442 421 316 338 443 422 317;
242 338 443 422 317 339 444 423 318;
243 339 444 423 318 340 445 424 319;
244 340 445 424 319 341 446 425 320;
245 341 446 425 320 342 447 426 321;
246 342 447 426 321 343 448 427 322;
247 343 448 427 322 344 449 428 323;
248 344 449 428 323 345 450 429 324;
249 345 450 429 324 346 451 430 325;
250 346 451 430 325 347 452 431 326;
251 347 452 431 326 348 453 432 327;
252 348 453 432 327 349 454 433 328;
253 349 454 433 328 350 455 434 329;
254 350 455 434 329 351 456 435 330;
255 351 456 435 330 352 457 436 331;
256 352 457 436 331 353 458 437 332;
257 353 458 437 332 354 459 438 333;
258 354 459 438 333 355 460 439 334;
259 355 460 439 334 356 461 440 335;
260 356 461 440 335 357 462 441 336;
261 358 463 442 337 359 464 443 338;
262 359 464 443 338 360 465 444 339;
263 360 465 444 339 361 466 445 340;
264 361 466 445 340 362 467 446 341;
265 362 467 446 341 363 468 447 342;
266 363 468 447 342 364 469 448 343;
267 364 469 448 343 365 470 449 344;
268 365 470 449 344 366 471 450 345;
269 366 471 450 345 367 472 451 346;
270 367 472 451 346 368 473 452 347;
271 368 473 452 347 369 474 453 348;
272 369 474 453 348 370 475 454 349;
273 370 475 454 349 371 476 455 350;
```

```
274 371 476 455 350 372 477 456 351;
275 372 477 456 351 373 478 457 352;
276 373 478 457 352 374 479 458 353;
277 374 479 458 353 375 480 459 354;
278 375 480 459 354 376 481 460 355;
279 376 481 460 355 377 482 461 356;
280 377 482 461 356 378 483 462 357;
281 379 484 463 358 380 485 464 359;
282 380 485 464 359 381 486 465 360;
283 381 486 465 360 382 487 466 361;
284 382 487 466 361 383 488 467 362;
285 383 488 467 362 384 489 468 363;
286 384 489 468 363 385 490 469 364;
287 385 490 469 364 386 491 470 365;
288 386 491 470 365 387 492 471 366;
289 387 492 471 366 388 493 472 367;
290 388 493 472 367 389 494 473 368;
291 389 494 473 368 390 495 474 369;
292 390 495 474 369 391 496 475 370;
293 391 496 475 370 392 497 476 371;
294 392 497 476 371 393 498 477 372;
295 393 498 477 372 394 499 478 373;
296 394 499 478 373 395 500 479 374;
297 395 500 479 374 396 501 480 375;
298 396 501 480 375 397 502 481 376;
299 397 502 481 376 398 503 482 377;
300 398 503 482 377 399 504 483 378;
301 400 505 484 379 401 506 485 380;
302 401 506 485 380 402 507 486 381;
303 402 507 486 381 403 508 487 382;
304 403 508 487 382 404 509 488 383;
305 404 509 488 383 405 510 489 384;
306 405 510 489 384 406 511 490 385;
307 406 511 490 385 407 512 491 386;
308 407 512 491 386 408 513 492 387;
309 408 513 492 387 409 514 493 388;
310 409 514 493 388 410 515 494 389;
311 410 515 494 389 411 516 495 390;
312 411 516 495 390 412 517 496 391;
313 412 517 496 391 413 518 497 392;
314 413 518 497 392 414 519 498 393;
315 414 519 498 393 415 520 499 394;
316 415 520 499 394 416 521 500 395;
317 416 521 500 395 417 522 501 396;
318 417 522 501 396 418 523 502 397;
319 418 523 502 397 419 524 503 398;
320 419 524 503 398 420 525 504 399;
MEMBER PROPERTY AMERICAN
1001 TO 1004 PRIS YD 0.2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 106 211 316 421 PINNED
```

```
22 127 232 337 442 PINNED
43 148 253 358 463 PINNED
64 169 274 379 484 PINNED
85 190 295 400 505 PINNED
LOAD 1
JOINT LOAD
21 FX -48
42 FX -48
63 FX -48
84 FX -48
105 FX -48
126 FX -48
147 FX -48
168 FX -48
189 FX -48
210 FX -48
231 FX -48
252 FX -48
273 FX -48
294 FX -48
315 FX -48
336 FX -48
357 FX -48
378 FX -48
399 FX -48
420 FX -48
441 FX -48
462 FX -48
483 FX -48
504 FX -48
525 FX -48
LOAD 2
JOINT LOAD
231 FY 1000
315 FY -1000
LOAD 3
JOINT LOAD
21 FX 133.33
42 FX 133.33
63 FX 133.33
84 FX 133.33
105 FX 133.33
126 FX 66.67
147 FX 66.67
168 FX 66.67
189 FX 66.67
210 FX 66.67
336 FX -66.67
357 FX -66.67
378 FX -66.67
399 FX -66.67
420 FX -66.67
441 FX -133.33
462 FX -133.33
483 FX -133.33
504 FX -133.33
525 FX -133.33
LOAD 4
```

```
JOINT LOAD
21 FY 40
42 FY 40
63 FY 40
84 FY 40
105 FY 40
126 FY 40
147 FY 40
168 FY 40
189 FY 40
210 FY 40
231 FY 40
252 FY 40
273 FY 40
294 FY 40
315 FY 40
336 FY 40
357 FY 40
378 FY 40
399 FY 40
420 FY 40
441 FY 40
462 FY 40
483 FY 40
504 FY 40
525 FY 40
PERFORM ANALYSIS
LOAD LIST 1
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 -
231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST 2
PRINT JOINT DISPLACEMENTS LIST 21 105 273 441 525
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 -
231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST 4
PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 -
231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
LOAD LIST ALL
PRINT ELEMENT JOINT STRESSES SOLID LIST 10
*PRINT ELEMENT JOINT STRESSES SOLID LIST 10 TO 70 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 90 TO 150 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 170 TO 230 BY 20
*PRINT ELEMENT JOINT STRESSES SOLID LIST 250 TO 310 BY 20
FINISH
```

STAAD Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     21    1   -0.00033  -0.00002  -0.00001   0.00000   0.00002  -0.00003
     42    1   -0.00031  -0.00002  -0.00001   0.00000   0.00000   0.00000
     63    1   -0.00031  -0.00002   0.00000   0.00000   0.00000   0.00000
     84    1   -0.00031  -0.00002   0.00001   0.00000   0.00000   0.00000
    105    1   -0.00033  -0.00002   0.00001  -0.00000  -0.00002  -0.00003
```

```
    126    1   -0.00030  -0.00001  -0.00001   0.00000   0.00000   0.00000
    147    1   -0.00029  -0.00001  -0.00000   0.00000   0.00000   0.00000
    168    1   -0.00029  -0.00001   0.00000   0.00000   0.00000   0.00000
    189    1   -0.00029  -0.00001   0.00000   0.00000   0.00000   0.00000
    210    1   -0.00030  -0.00001   0.00001   0.00000   0.00000   0.00000
    231    1   -0.00030   0.00000  -0.00001   0.00000   0.00000   0.00000
    252    1   -0.00029   0.00000  -0.00000   0.00000   0.00000   0.00000
    273    1   -0.00029   0.00000   0.00000   0.00000   0.00000   0.00000
    294    1   -0.00029   0.00000   0.00000   0.00000   0.00000   0.00000
    315    1   -0.00030   0.00000   0.00001   0.00000   0.00000   0.00000
    336    1   -0.00030   0.00001  -0.00001   0.00000   0.00000   0.00000
    357    1   -0.00029   0.00001  -0.00000   0.00000   0.00000   0.00000
    378    1   -0.00029   0.00001   0.00000   0.00000   0.00000   0.00000
    399    1   -0.00029   0.00001   0.00000   0.00000   0.00000   0.00000
    420    1   -0.00030   0.00001   0.00001   0.00000   0.00000   0.00000
    441    1   -0.00033   0.00002  -0.00001  -0.00000   0.00002   0.00003
    462    1   -0.00031   0.00002  -0.00001   0.00000   0.00000   0.00000
    483    1   -0.00031   0.00002   0.00000   0.00000   0.00000   0.00000
    504    1   -0.00031   0.00002   0.00001   0.00000   0.00000   0.00000
    525    1   -0.00033   0.00002   0.00001   0.00000  -0.00002   0.00003
   ************** END OF LATEST ANALYSIS RESULT **************
   507. LOAD LIST 2
   508. PRINT JOINT DISPLACEMENTS LIST 21 105 273 441 525
  JOINT    DISPLACE LIST     21
      : A CANTILEVER BEAM WITH SOLID ELEMENTS                  -- PAGE NO.
12
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     21    2   -0.00002   0.00149  -0.00224   0.00149   0.00001  -0.00001
    105    2    0.00002  -0.00149  -0.00224   0.00149   0.00001   0.00001
    273    2    0.00000   0.00000   0.00000   0.00149   0.00000   0.00000
    441    2    0.00002   0.00149   0.00224   0.00149  -0.00001  -0.00001
    525    2   -0.00002  -0.00149   0.00224   0.00149  -0.00001   0.00001
   ************** END OF LATEST ANALYSIS RESULT **************
   509. LOAD LIST 3
   510. PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 -
  JOINT    DISPLACE LIST     21
   511. 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
      : A CANTILEVER BEAM WITH SOLID ELEMENTS                  -- PAGE NO.
13
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     21    3    0.00132   0.00621   0.00004  -0.00002   0.00000   0.00088
     42    3    0.00125   0.00621   0.00002   0.00000   0.00000   0.00000
     63    3    0.00125   0.00621   0.00000   0.00000   0.00000   0.00000
     84    3    0.00125   0.00621  -0.00002   0.00000   0.00000   0.00000
    105    3    0.00132   0.00621  -0.00004   0.00002   0.00000   0.00088
    126    3    0.00061   0.00619   0.00001   0.00000   0.00000   0.00000
    147    3    0.00060   0.00619   0.00001   0.00000   0.00000   0.00000
    168    3    0.00060   0.00619   0.00000   0.00000   0.00000   0.00000
    189    3    0.00060   0.00619  -0.00001   0.00000   0.00000   0.00000
    210    3    0.00061   0.00619  -0.00001   0.00000   0.00000   0.00000
    231    3    0.00000   0.00618   0.00000   0.00000   0.00000   0.00000
    252    3    0.00000   0.00618   0.00000   0.00000   0.00000   0.00000
    273    3    0.00000   0.00619   0.00000   0.00000   0.00000   0.00088
    294    3    0.00000   0.00618   0.00000   0.00000   0.00000   0.00000
```

```
    315    3    0.00000   0.00618   0.00000   0.00000   0.00000   0.00000
    336    3   -0.00061   0.00619  -0.00001   0.00000   0.00000   0.00000
    357    3   -0.00060   0.00619  -0.00001   0.00000   0.00000   0.00000
    378    3   -0.00060   0.00619   0.00000   0.00000   0.00000   0.00000
    399    3   -0.00060   0.00619   0.00001   0.00000   0.00000   0.00000
    420    3   -0.00061   0.00619   0.00001   0.00000   0.00000   0.00000
    441    3   -0.00132   0.00621  -0.00004  -0.00002   0.00000   0.00088
    462    3   -0.00125   0.00621  -0.00002   0.00000   0.00000   0.00000
    483    3   -0.00125   0.00621   0.00000   0.00000   0.00000   0.00000
    504    3   -0.00125   0.00621   0.00002   0.00000   0.00000   0.00000
    525    3   -0.00132   0.00621   0.00004   0.00002   0.00000   0.00088
   ************** END OF LATEST ANALYSIS RESULT **************
   512. LOAD LIST 4
   513. PRINT JOINT DISPLACEMENTS LIST 21 42 63 84 105 126 147 168 189 210 -
  JOINT    DISPLACE LIST     21
   514. 231 252 273 294 315 336 357 378 399 420 441 462 483 504 525
      : A CANTILEVER BEAM WITH SOLID ELEMENTS                  -- PAGE NO.
14
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     21    4    0.00372   0.02537  -0.00001   0.00003   0.00000   0.00248
     42    4    0.00373   0.02536  -0.00001   0.00000   0.00000   0.00000
     63    4    0.00373   0.02535   0.00000   0.00000   0.00000   0.00000
     84    4    0.00373   0.02536   0.00001   0.00000   0.00000   0.00000
    105    4    0.00372   0.02537   0.00001  -0.00003   0.00000   0.00248
    126    4    0.00184   0.02535  -0.00000   0.00000   0.00000   0.00000
    147    4    0.00185   0.02534  -0.00000   0.00000   0.00000   0.00000
    168    4    0.00185   0.02534   0.00000   0.00000   0.00000   0.00000
    189    4    0.00185   0.02534   0.00000   0.00000   0.00000   0.00000
    210    4    0.00184   0.02535   0.00000   0.00000   0.00000   0.00000
    231    4    0.00000   0.02534   0.00000   0.00000   0.00000   0.00000
    252    4    0.00000   0.02533   0.00000   0.00000   0.00000   0.00000
    273    4    0.00000   0.02533   0.00000   0.00000   0.00000   0.00248
    294    4    0.00000   0.02533   0.00000   0.00000   0.00000   0.00000
    315    4    0.00000   0.02534   0.00000   0.00000   0.00000   0.00000
    336    4   -0.00184   0.02535   0.00000   0.00000   0.00000   0.00000
    357    4   -0.00185   0.02534   0.00000   0.00000   0.00000   0.00000
    378    4   -0.00185   0.02534   0.00000   0.00000   0.00000   0.00000
    399    4   -0.00185   0.02534  -0.00000   0.00000   0.00000   0.00000
    420    4   -0.00184   0.02535  -0.00000   0.00000   0.00000   0.00000
    441    4   -0.00372   0.02537   0.00001   0.00003   0.00000   0.00248
    462    4   -0.00373   0.02536   0.00001   0.00000   0.00000   0.00000
    483    4   -0.00373   0.02535   0.00000   0.00000   0.00000   0.00000
    504    4   -0.00373   0.02536  -0.00001   0.00000   0.00000   0.00000
    525    4   -0.00372   0.02537  -0.00001  -0.00003   0.00000   0.00248
   ************** END OF LATEST ANALYSIS RESULT **************
   515. LOAD LIST ALL
   516. PRINT ELEMENT JOINT STRESSES SOLID LIST 10
  ELEMENT  JOINT    STRESSES SOLID
      : A CANTILEVER BEAM WITH SOLID ELEMENTS                  -- PAGE NO.
15
 ELEMENT STRESSES        UNITS= POUNINCH

------------------------------------------------------------------------------
-
             NODE/           NORMAL STRESSES              SHEAR STRESSES
 ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ
```

```
SZX

--------------------------------------------------------------------------------
-
    10    1     31   -200.005    -0.003     0.000     0.001     0.000
-0.001
    10    1    136   -199.999    -0.003     0.001     0.001    -0.000
-0.000
    10    1    115   -200.000    -0.003     0.002     0.001    -0.000
-0.000
    10    1     10   -200.007    -0.003    -0.000     0.001    -0.000
-0.001
    10    1     32   -200.004    -0.001     0.001     0.001    -0.000
-0.000
    10    1    137   -199.999    -0.001     0.000     0.001     0.000
-0.001
    10    1    116   -200.000    -0.002     0.000     0.001     0.000
-0.000
    10    1     11   -200.007    -0.002     0.001     0.001     0.000
-0.001
    10    1 CENTER   -200.003    -0.002     0.001     0.001    -0.000
-0.000
              S1=      0.001   S2=     -0.002   S3=   -200.003   SE=
200.002
              DC=     -0.000    -0.017     1.000     0.000     1.000
0.017
    10    2     31      0.006    -0.006    -0.008   137.619     0.001
-279.913
    10    2    136      0.006    -0.004    -0.004   137.618     0.003
-53.160
    10    2    115      0.006    -0.009    -0.006   390.964     0.002
-53.159
    10    2     10      0.007    -0.010    -0.009   390.962    -0.000
-279.913
    10    2     32      0.005     0.004     0.008   137.619     0.000
-279.913
    10    2    137      0.002     0.002     0.003   137.618    -0.002
-53.160
    10    2    116      0.006     0.007     0.005   390.963    -0.001
-53.159
    10    2     11      0.011     0.009     0.010   390.962     0.001
-279.914
    10    2 CENTER      0.006    -0.001    -0.000   264.291     0.001
-166.536
              S1=    312.387   S2=      0.000   S3=   -312.382   SE=
541.066
              DC=      0.707     0.598    -0.377     0.000     0.533
0.846
      : A CANTILEVER BEAM WITH SOLID ELEMENTS                  -- PAGE NO.
16
 ELEMENT STRESSES        UNITS= POUNINCH

--------------------------------------------------------------------------------
-
              NODE/          NORMAL STRESSES              SHEAR STRESSES
 ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ
SZX
```

```
--------------------------------------------------------------------------------
-
    10    3     31    833.325    -0.002    -0.000     0.000     0.000
0.000
    10    3    136    416.662    -0.002    -0.000    -0.000     0.000
0.000
    10    3    115    416.662    -0.002    -0.001     0.000     0.000
0.000
    10    3     10    833.325    -0.002    -0.000     0.000     0.000
0.000
    10    3     32    833.325     0.000     0.000     0.000     0.000
0.000
    10    3    137    416.662     0.001     0.000    -0.000    -0.000
0.000
    10    3    116    416.662     0.001     0.000     0.000    -0.000
0.000
    10    3     11    833.325     0.001     0.000    -0.000     0.000
0.000
    10    3 CENTER    624.994    -0.001    -0.000     0.000     0.000
0.000
              S1=    624.994   S2=     -0.000   S3=     -0.001   SE=
624.994
              DC=       1.000     0.000     0.000     0.000     0.000
1.000
    10    4     31   2619.592   -45.597   -52.800   101.363     2.404
-4.541
    10    4    136   1317.907   -34.780   -27.561   101.362     2.404
-2.110
    10    4    115   1317.908   -34.781   -27.562   115.768    -2.403
2.697
    10    4     10   2619.594   -45.598   -52.801   115.768    -2.404
-9.349
    10    4     32   2630.409    45.589    52.800   101.363    -2.404
-4.541
    10    4    137   1307.089    34.772    27.558   101.363    -2.404
-2.110
    10    4    116   1307.091    34.773    27.559   115.768     2.404
2.698
    10    4     11   2630.412    45.590    52.800   115.767     2.404
-9.349
    10    4 CENTER   1968.750    -0.004    -0.001   108.565     0.000
-3.326
              S1=   1974.724   S2=     -0.001   S3=     -5.978   SE=
1977.721
              DC=       0.998     0.055    -0.002    -0.000     0.031
1.000
```

V.06 Loading

V. EN 1998-1-2004

V. EN 1998-1-2004 Response Spectrum

Calculation of base shear for response spectrum method per EN 1998-1-2004.

Details

A three story building is modelled in STAAD.Pro considering the following:

Type 1
Ground type A
Soil factor, S = 1
Lower limit of period, T_B = 0.15 sec
Upper limit of period, T_C = 0.4 sec
Value defining the beginning of constant displacement response range, T_D = 2 sec
Damping: 5%
Design ground acceleration, a_g = 0.981 m/sec^2
Damping correction factor, η = 1
Behavior factor, q = 3
Lower bound factor for horizontal design spectrum, β = 0.2

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 446: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 447: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 448: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.5175
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight Mk × g		227.66			16.35			1.173		
Modal Weight Participation (In %)		92.86			6.667			0.479		

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Mode Participation Factor $\prod \sum Wi \times \phi \,/\, \sum Wi \times \phi^2$		1.244			0.333			0.089		

	Mode 1	Mode 2	Mode 3
Time Period T	0.30014	0.10986	0.08042
Design response spectrum Sd(T):	As $T_B \le T \le T_C$,hence- a_g x S x (2.5/q) = 0.981 x 1 x (2.5/3) = 0.8175	As $0 \le T \le T_B$,hence- a_g x S x [(2/3) + (T/T_B) x {(2.5/q)-(2/3)}] = 0.981 x 1 x [(2/3) + (0.10986/0.15) x {(2.5/3)-(2/3)}] = 0.7737	As $0 \le T \le T_B$,hence- a_g x S x [(2/3) + (T/T_B) x {(2.5/q)-(2/3)}] = 0.981 x 1 x [(2/3) + (0.08042/0.15) x {(2.5/3)-(2/3)}] = 0.7417
Sd/g =	0.083	0.079	0.076

Design lateral force at each floor in each mode, $Q_i = P_k \times s_i \times (Sd/g) \times W_i$

Table 449: Base Shear Calculation and its Distribution Across Height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		s_i	Qi	Vi	s_i	Qi	Vi	s_i	Qi	Vi	
Roof	49.035	1	5.08	5.08	-1	-1.29	-1.29	1	0.33	0.33	5.25
2nd Floor	98.070	0.866603	8.80	13.89	0	0.00	-1.29	-0.866603	-0.57	-0.24	13.95
1st Floor	98.070	0.5	5.08	18.97	1	2.58	1.29	0.5	0.33	0.09	19.02
Base Shear, V											19.02

Comparison

Table 450: Comparison of results

Parameter	Reference	STAAD.Pro	Difference	Comments
Base shear (kN)	19.02	19.02	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\EN 1998-1-2004\EN 1998-1-2004 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
```

```
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS EURO 2004 DESIGN RS1 X 1 ACC DAMP 0.05
SOIL TYPE A ALPHA 0.1 Q 3
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX           FY           FZ           MX
MY          MZ
      1      0.300       18.97        0.00         0.00         0.00
0.00     -113.83
      2      0.110        1.29        0.00         0.00         0.00
0.00        3.87
      3      0.080        0.09        0.00         0.00         0.00
0.00       -0.53
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X      Y      Z      SUMM-X   SUMM-Y   SUMM-Z       X         Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000     18.97     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      1.29     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.09     0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR     19.02     0.00
0.00
                                  TOTAL 10PCT SHEAR     19.02     0.00
0.00
                                  TOTAL ABS   SHEAR     20.35     0.00
0.00
```

V. GB 50011

V. GB 50011 2010 Response Spectrum

Calculation of base shear and for response spectrum method in GB 50011-2010.

Details

A three story building is modelled in STAAD.Pro considering the following:

Classification of seismic design = Group 2
Design seismic intensity = 8
Site class = II
Damping = 3.5%

The seismic weight of 420 kN applied as a joint weight of 70 kN to the joints at each floor (joints 2, 3, 4, 6,7, and 8); or 140 kN at each story above ground.

The generated model is subjected to a response spectrum load along global X direction. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 451: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	1.114	0.897
2	3.122	0.320
3	4.512	0.222

Table 452: Mode Shapes

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 4 & 8	1	0	0	0	0	0
	2nd Floor/Joints 3 & 7	0.802	0	0	0	0	0
	1st Floor/Joints 2 & 6	0.445	0	0	0	0	0
2	Roof/Joints 4 & 8	-0.802	0	0	0	0	0
	2nd Floor/Joints 3 & 7	0 .445	0	0	0	0	0
	1st Floor/Joints 2 & 6	1	0	0	0	0	0
3	Roof/Joints 4 & 8	-0.445	0	0	0	0	0

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
	2nd Floor/Joints 3 & 7	1	0	0	0	0	0
	1st Floor/Joints 2 & 6	-0.802	0	0	0	0	0

<u>Horizontal Acceleration Spectrum Value</u>

The max value of horizontal seismic influence, $a_{max} = 0.16$.

The characteristic period, $T_g = 0.4s$.

The damping ratio, $\zeta = 0.035$.

The power index of the curvilinear decrease section:

$$= 0.9 + \frac{0.05 - \zeta}{0.3 + 6\zeta} = 0.929$$

The damping adjustment factor is determined as:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.08 + 1.6\zeta} = 1.11$$

	Mode 1	Mode 2	Mode 3
Time Period T	0.897 (Tg<T<5Tg)	0.320 (0.1<T < Tg)	0.222 (0.1<T < Tg)
Formula	$a = \left(\frac{T_g}{T}\right)^{\gamma} \eta_2 a_{max}$	$a = \eta_2 a_{max}$	$a = \eta_2 a_{max}$
Seismic Coefficient a	0.0839	0.1776	0.1776

Table 453: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	140	1	140	140	-0.802	-112.3	90.05	-0.445	-62.3	27.72
2nd Floor	140	0.802	112.3	90.05	0.445	62.3	27.72	1	140	140
1st Floor	140	0.445	62.3	27.72	1	140	140	-0.802	-112.3	90.05
Summation	420		314.6	257.77		90.02	257.77		-34.58	257.77

<table>
<tr><th rowspan="2">Story Level</th><th rowspan="2">Weight Wi (KN)</th><th colspan="3">Mode 1</th><th colspan="3">Mode 2</th><th colspan="3">Mode 3</th></tr>
<tr><th>ϕ</th><th>$Wi \times \phi$</th><th>$Wi \times \phi^2$</th><th>ϕ</th><th>$Wi \times \phi$</th><th>$Wi \times \phi^2$</th><th>ϕ</th><th>$Wi \times \phi$</th><th>$Wi \times \phi^2$</th></tr>
<tr><td colspan="2">Mode Participation Factor $Pk \sum Wi \times \phi$ / $\sum Wi \times \phi^2$</td><td colspan="3">1.22</td><td colspan="3">0.349</td><td colspan="3">-0.134</td></tr>
<tr><td colspan="2">Seismic Coefficient a</td><td colspan="3">0.0839</td><td colspan="3">0.1776</td><td colspan="3">0.1776</td></tr>
<tr><td rowspan="3">Horizontal seismic action</td><td>Roof</td><td colspan="3">14.33</td><td colspan="3">-6.959</td><td colspan="3">1.483</td></tr>
<tr><td>2nd Floor</td><td colspan="3">11.49</td><td colspan="3">3.862</td><td colspan="3">-3.332</td></tr>
<tr><td>1st Floor</td><td colspan="3">6.377</td><td colspan="3">8.678</td><td colspan="3">2.672</td></tr>
<tr><td colspan="2">Base shear</td><td colspan="3">32.197</td><td colspan="3">5.581</td><td colspan="3">0.823</td></tr>
</table>

Summation by SRSS:

$$\sqrt{(32.197)^2 + (5.581)^2 + (0.823)^2} = 32.69$$

Comparison

Table 454: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Base shear (kN)	32.69	32.67	negligible	

Note: The complete quadratic combination method (CQC) in STAAD.Pro gives essentially the same value as the calculation above, as well: 32.69 kN.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\GB 50011\GB 50011 2010 Response Spectrum.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5.00001 0; 3 0 10 0; 4 0 15 0; 5 5.00001 0 0; 6 5.00001 5.00001
0;
7 5.00001 10 0; 8 5.00001 15 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 2 6; 8 3 7; 9 4 8;
START USER TABLE
```

```
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 6 UPTABLE 1 COLLUMN
7 TO 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 5 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE MASS
JOINT LOAD
2 TO 4 6 TO 8 FX 70
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
LOAD 1 LOADTYPE Seismic-H  TITLE EL-X DIR
SPECTRUM CQC GB50011 2010 X 1 ALPHA DAMP 0.035 LOGARITHMIC INTENSITY 8 -
FREQUENT GROUP 2 SCLASS 2
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
  GB50011 RESPONSE SPECTRUM LOAD     1
     GB50011 RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
```

```
         1       3.839138E+02  0.000000E+00  0.000000E+00    2.577637E+02
         2       3.144839E+01  0.000000E+00  0.000000E+00    2.577634E+02
         3       4.638204E+00  0.000000E+00  0.000000E+00    2.577630E+02
     CQC           MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   4.200000E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z   4.013087E-04  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   4.200004E+02  0.000000E+00  0.000000E+00 KN
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     0.08383       0.03500
              2                     0.17765       0.03500
              3                     0.17765       0.03500
      STAAD SPACE                                                    -- PAGE NO.
5
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE      PERIOD         FX          FY          FZ          MX
MY          MZ
      1       0.897       32.18        0.00        0.00        0.00
0.00     -361.56
      2       0.320        5.59        0.00        0.00        0.00
0.00       22.40
      3       0.222        0.82        0.00        0.00        0.00
0.00       -2.29
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE     X      Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1    91.41  0.00  0.00   91.408    0.000    0.000     32.18     0.00
0.00
   2     7.49  0.00  0.00   98.896    0.000    0.000      5.59     0.00
0.00
   3     1.10  0.00  0.00  100.000    0.000    0.000      0.82     0.00
0.00

---------------------------
                                  TOTAL CQC   SHEAR     32.68     0.00
0.00
```

Related Links

- *TR.32.10.1.6 Response Spectrum Specification per GB 50011 2010* (on page 2531)

V. GB 50011-2001 Static Seismic - Case1

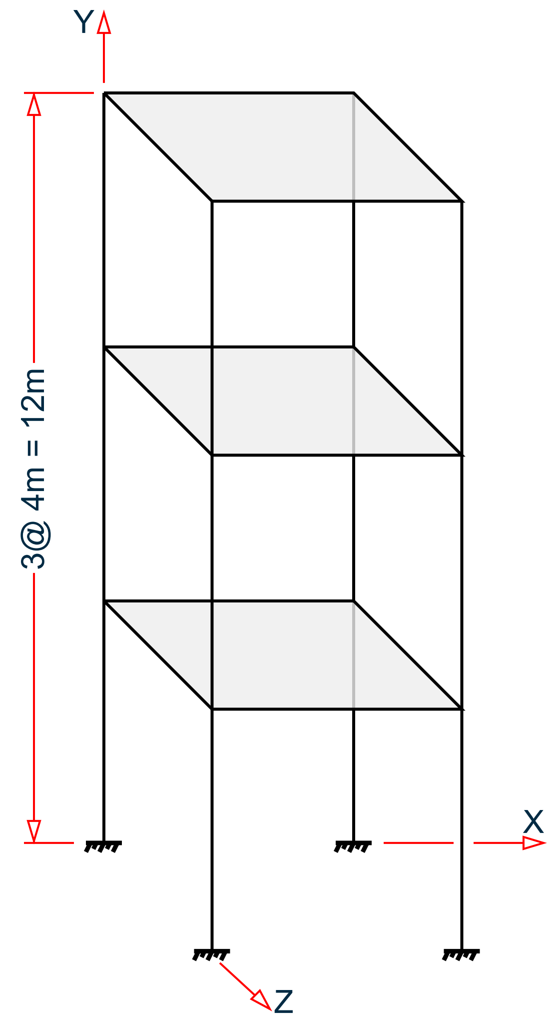
Calculate the base shear and its distribution along the height using the equivalent lateral force method in GB 50011-2001 using the default equivalent gravity load factor for horizontal seismic action.

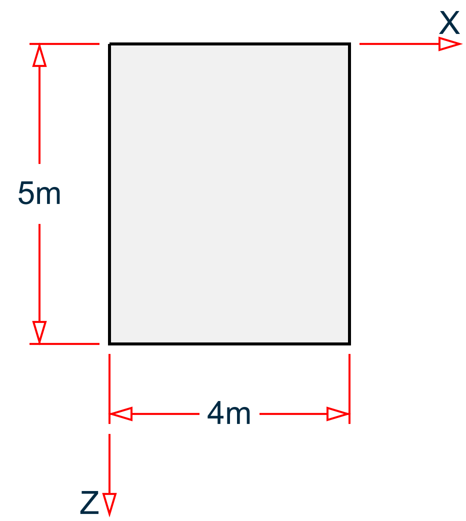
Details

Structure to be modelled:

height = 3× 4m = 12m
plan dimension X = 4m

plan dimension Z = 5m





Input parameters:

Frequency of seismic event: Frequent
Classification of seismic design: Group 2
Seismic design intensity: 8
Site class: II
Period in the X direction: $T = 0.467s$
Equivalent gravity load factor for horizontal seismic action: 0.85 (default in STAAD.Pro)
Damping ratio, $\zeta = 0.05$ (default in STAAD.Pro)

The section of all members is HW300X300, and the seismic weight for each floor is assumed as 280 kN applied as joint weight of 70 kN to nodes 2, 3, 5, 6, 8, 9, and 11 to 16. S

Validation

The effective weight of the structure:

$$G_{Eq} = 0.85 \times 280 \text{ kN} \times 3 \text{ floors} = 714 \text{ kN}$$

The max value of the horizontal seismic influence for a “frequent” earthquake with intensity of 8, $a_{max} = 0.16$ (Table 5.1.4-1)

The characteristic period, $T_g = 0.4s$ (Table 5.1.4-2)

The power index of the curvilinear decrease section:

$$\gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} = 0.9$$

The damping adjustment factor:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} = 1.0$$

$$T_g < T < 5T_g$$

The horizontal seismic influence coefficient corresponding to the fundamental period of the structure:

$$a_1 = \left(\frac{T_g}{T}\right)^{\gamma} \eta_2 \times a_{max} = \left(\frac{0.4}{0.467}\right)^{0.9} 1.0 \times 0.16 = 0.139$$

The characteristic value of the total horizontal seismic action of the structure:

$$F_{Ek} = a_1 \times G_{Eq} = 0.139 \times 714 = 99.25 \text{ kN} \tag{Eq. 2.5}$$

$$T = 0.467 < 1.4T_g = 0.56$$

Additional seismic action factors at the top of the building $\delta_n = 0$

Additional horizontal seismic action applied at top of the building $\Delta F_n = \delta_n \times F_{Ek} = 0$

Characteristic value of horizontal seismic action applied at the i[th] mass:

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n)$$

Story Level	G_i (kN)	H_i (m)	G_iH_i (kN·m)	$G_iH_i / \Sigma G_iH_i$	F_i (kN)	ΔF_n (kN)	V_i (kN)
3rd	280	12	3,360	0.5	49.623	0	49.623
2nd	280	8	2,240	0.333	33.082		82.705
1st	280	4	1,120	0.167	16.541		99.246
Σ	840		6,720	1			

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
3rd	49.623	280	17.72	3.2	1
2nd	82.705	560	14.77	3.2	1

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
1st	99.246	840	11.82	3.2	1

Results

Table 455: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Story Shear, F_i (kN)	1st	16.541	16.563	negligible	
	2nd	33.082	33.126	negligible	
	3rd	49.623	49.689	negligible	
Base Shear, V (kN)		99.246	99.377	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\GB 50011\GB 50011 2001 Static Seismic - Case1.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
       1 0 0 0
       2 0 4 0
       3 4 4 0
       4 4 0 0
       5 0 8 0
       6 4 8 0
       7 0 0 5
       8 0 4 5
       9 4 4 5
      10 4 0 5
      11 0 8 5
      12 4 8 5
      13 0 12 0
      14 4 12 0
      15 0 12 5
      16 4 12 5
MEMBER INCIDENCES
       1         1         2
       2         2         3
       3         3         4
       4         2         5
```

```
       5         5         6
       6         6         3
       7         2         8
       8         3         9
       9         5        11
      10         6        12
      11         7         8
      12         8         9
      13         9        10
      14         8        11
      15        11        12
      16        12         9
      17         5        13
      18         6        14
      19        11        15
      20        12        16
      21        13        14
      22        13        15
      23        14        16
      24        15        16
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 24 TABLE ST HW300X300
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE DL
JOINT LOAD
2 3 5 6 8 9 11 TO 16 FY -70
END DEFINE REFERENCE LOADS
DEFINE GB50011 2001 LOAD
INTENSITY 8 FREQUENT GROUP 2 SCLASS 2 PX 0.467
REFERENCE LOAD Y
R1 1.0
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
GB50011 LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
```

```
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------

*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR
BUILDINGS              *
   *                               (GB50011:2001) OF CHINA ALONG
X            *
   *  T CALCULATED =  0.685 SEC.      T USER PROVIDED =  0.467
SEC.            *
   *  T USED =  0.467
SEC.                                        *
   *  MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT =
0.160                  *
   *  CHARACTERISTIC PERIOD =  0.400
SEC.                         *
   *  DAMPING RATIO =  0.050    POWER INDEX (GAMMA) =
0.900                   *
   *  DAMPING ADJUSTMENT FACTOR (ETA2) =
1.000                               *
   *  ADJUSTING FACTOR (ETA1) =
0.020                                        *
   *  HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) =  0.139
(13.918%)    *
   *  MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) =   0.032
(  3.200%)    *
   *  EQUIVALENT GRAVITY LOAD FACTOR FOR HORIZONTAL SEISMIC ACTION =
0.850    *
   *  TOTAL HORIZONTAL SEISMIC ACTION
=                                        *
   *          =  0.139 X      0.850 X    840.000 =       99.377
KN            *
   *  DESIGN BASE SHEAR =  1.000 X
99.377                                 *
   *                    =       99.377
KN                                      *
   *  ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) =
0.000                      *

*****************************************************************************
    CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011:5.2.5]
                                                           LOAD -    1
FACTOR -   1.000
    FLOOR            LATERAL          GRAVITY         LAMBDA   LAMBDA
ADJUSTMENT
    HEIGHT (METE)    LOAD (KN  )      LOAD (KN  )      (%)     MIN (%)
FACTOR
    -------------    -----------      ------------    ------   ------
-----------
        12.000          49.689         280.000       17.75     3.20     1.00
```

```
         8.000           82.814           560.000       14.79     3.20     1.00
         4.000           99.377           840.000       11.83     3.20     1.00
    JOINT            LATERAL          TORSIONAL            VERTICAL      LOAD
-    1
                     LOAD (KN  )      MOMENT (KN  -METE)   LOAD (KN  )
FACTOR -   1.000
    -----            -------          ---------           -----------
        2     FX        4.141    MY        0.000           FY        0.000
        3     FX        4.141    MY        0.000           FY        0.000
        8     FX        4.141    MY        0.000           FY        0.000
        9     FX        4.141    MY        0.000           FY        0.000
             -----------        -----------           ----------------
          TOTAL =     16.563             0.000                     0.000
AT LEVEL     4.000 METE
      STAAD SPACE                                                -- PAGE NO.
5
        5     FX        8.281    MY        0.000           FY        0.000
        6     FX        8.281    MY        0.000           FY        0.000
       11     FX        8.281    MY        0.000           FY        0.000
       12     FX        8.281    MY        0.000           FY        0.000
             -----------        -----------           ----------------
          TOTAL =     33.126             0.000                     0.000
AT LEVEL     8.000 METE
       13     FX       12.422    MY        0.000           FY        0.000
       14     FX       12.422    MY        0.000           FY        0.000
       15     FX       12.422    MY        0.000           FY        0.000
       16     FX       12.422    MY        0.000           FY        0.000
             -----------        -----------           ----------------
          TOTAL =     49.689             0.000                     0.000
AT LEVEL    12.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    85. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                                -- PAGE NO.
6
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     0.6474    0.0138    0.0000    0.0000    0.0000   -0.0014
      3    1     0.6474   -0.0138    0.0000    0.0000    0.0000   -0.0014
      4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1     1.4829    0.0215    0.0000    0.0000    0.0000   -0.0012
      6    1     1.4829   -0.0215    0.0000    0.0000    0.0000   -0.0012
      7    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      8    1     0.6474    0.0138    0.0000    0.0000    0.0000   -0.0014
      9    1     0.6474   -0.0138    0.0000    0.0000    0.0000   -0.0014
     10    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     11    1     1.4829    0.0215    0.0000    0.0000    0.0000   -0.0012
     12    1     1.4829   -0.0215    0.0000    0.0000    0.0000   -0.0012
     13    1     2.0484    0.0240    0.0000    0.0000    0.0000   -0.0007
     14    1     2.0484   -0.0240    0.0000    0.0000    0.0000   -0.0007
     15    1     2.0484    0.0240    0.0000    0.0000    0.0000   -0.0007
     16    1     2.0484   -0.0240    0.0000    0.0000    0.0000   -0.0007
      STAAD SPACE                                                -- PAGE NO.
7
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
```

```
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -24.84    -83.72      0.00      0.00      0.00     64.44
      4    1    -24.84     83.72      0.00      0.00      0.00     64.44
      7    1    -24.84    -83.72      0.00      0.00      0.00     64.44
     10    1    -24.84     83.72      0.00      0.00      0.00     64.44
      STAAD SPACE                                                -- PAGE NO.
8
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1    -83.72     24.84    -0.00     -0.00      0.00      64.44
                 2     83.72    -24.84     0.00      0.00      0.00      34.94
      2    1     2     -0.00    -37.23    -0.00      0.00      0.00     -74.45
                 3      0.00     37.23     0.00     -0.00      0.00     -74.45
      3    1     3     83.72     24.84    -0.00     -0.00      0.00      34.94
                 4    -83.72    -24.84     0.00      0.00      0.00      64.44
      4    1     2    -46.49     20.70    -0.00     -0.00      0.00      39.51
                 5     46.49    -20.70     0.00      0.00      0.00      43.30
      5    1     5      0.00    -31.16    -0.00      0.00      0.00     -62.31
                 6      0.00     31.16     0.00     -0.00      0.00     -62.31
      6    1     6     46.49     20.70    -0.00     -0.00      0.00      43.30
                 3    -46.49    -20.70     0.00      0.00      0.00      39.51
      7    1     2     -0.00     -0.00     0.00      0.00     -0.00      -0.00
                 8      0.00      0.00    -0.00     -0.00     -0.00      -0.00
      8    1     3      0.00      0.00     0.00      0.00     -0.00       0.00
                 9     -0.00     -0.00    -0.00     -0.00     -0.00       0.00
      9    1     5      0.00     -0.00     0.00      0.00     -0.00      -0.00
                11     -0.00      0.00    -0.00     -0.00     -0.00      -0.00
     10    1     6     -0.00      0.00     0.00      0.00     -0.00       0.00
                12      0.00     -0.00    -0.00     -0.00     -0.00       0.00
     11    1     7    -83.72     24.84    -0.00     -0.00      0.00      64.44
                 8     83.72    -24.84     0.00      0.00      0.00      34.94
     12    1     8      0.00    -37.23    -0.00      0.00      0.00     -74.45
                 9      0.00     37.23     0.00     -0.00      0.00     -74.45
     13    1     9     83.72     24.84    -0.00     -0.00      0.00      34.94
                10    -83.72    -24.84     0.00      0.00      0.00      64.44
     14    1     8    -46.49     20.70    -0.00     -0.00      0.00      39.51
                11     46.49    -20.70     0.00      0.00      0.00      43.30
     15    1    11      0.00    -31.16    -0.00      0.00      0.00     -62.31
                12      0.00     31.16     0.00     -0.00      0.00     -62.31
      STAAD SPACE                                                -- PAGE NO.
9
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     16    1    12     46.49     20.70    -0.00     -0.00      0.00      43.30
                 9    -46.49    -20.70     0.00      0.00      0.00      39.51
     17    1     5    -15.34     12.42    -0.00     -0.00      0.00      19.01
                13     15.34    -12.42     0.00      0.00      0.00      30.68
     18    1     6     15.34     12.42     0.00     -0.00     -0.00      19.01
                14    -15.34    -12.42    -0.00      0.00     -0.00      30.68
     19    1    11    -15.34     12.42    -0.00     -0.00      0.00      19.01
                15     15.34    -12.42     0.00      0.00      0.00      30.68
     20    1    12     15.34     12.42     0.00     -0.00     -0.00      19.01
                16    -15.34    -12.42    -0.00      0.00     -0.00      30.68
     21    1    13     -0.00    -15.34    -0.00      0.00      0.00     -30.68
```

```
               14      0.00     15.34     0.00     -0.00      0.00    -30.68
     22    1   13      0.00     -0.00     0.00     -0.00     -0.00     -0.00
               15     -0.00      0.00    -0.00      0.00     -0.00     -0.00
     23    1   14     -0.00      0.00     0.00     -0.00     -0.00      0.00
               16      0.00     -0.00    -0.00      0.00     -0.00      0.00
     24    1   15      0.00    -15.34    -0.00      0.00      0.00    -30.68
               16      0.00     15.34     0.00     -0.00      0.00    -30.68
   ************** END OF LATEST ANALYSIS RESULT **************
```

Related Links

- *TR.31.2.5 Chinese Static Seismic per GB50011-2001* (on page 2369)

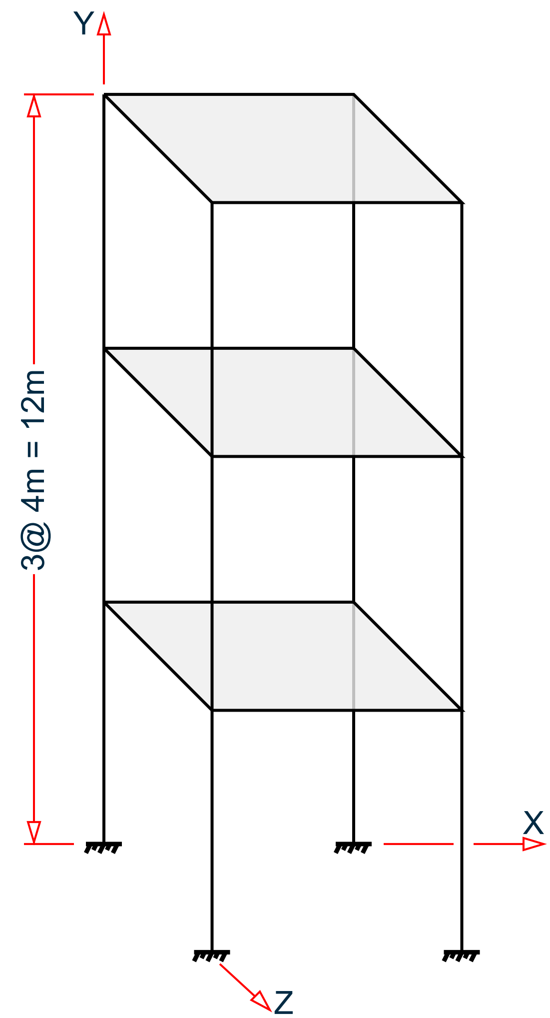
V. GB 50011-2001 Static Seismic - Case2

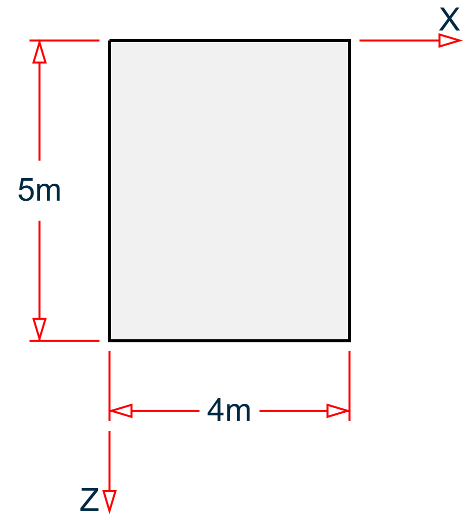
Calculate the base shear and its distribution along the height using the equivalent lateral force method in GB 50011-2001 using a equivalent gravity load factor for horizontal seismic action of 1.0.

Details

Structure to be modelled:

height = 3× 4m = 12m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Frequency of seismic event: Frequent
Classification of seismic design: Group 2
Seismic design intensity: 8
Site class: II
Period in the X direction: $T = 0.467s$
Equivalent gravity load factor for horizontal seismic action: 1.0
Damping ratio, $\zeta = 0.05$ (default in STAAD.Pro)

The section of all members is HW300X300, and the seismic weight for each floor is assumed as 280 kN applied as joint weight of 70 kN to nodes 2, 3, 5, 6, 8, 9, and 11 to 16. S

Validation

The effective weight of the structure:

$$G_{\mathrm{Eq}} = 1.0 \times 280 \text{ kN} \times 3 \text{ floors} = 840 \text{ kN}$$

The max value of the horizontal seismic influence for a “frequent” earthquake with intensity of 8, $a_{max} = 0.16$ (Table 5.1.4-1)

The characteristic period, $T_g = 0.4s$ (Table 5.1.4-2)

The power index of the curvilinear decrease section:

$$\gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} = 0.9$$

The damping adjustment factor:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} = 1.0$$

$$\mathrm{T_g} < \mathrm{T} < 5\mathrm{T_g}$$

The horizontal seismic influence coefficient corresponding to the fundamental period of the structure:

$$a_1 = \left(\frac{T_\mathrm{g}}{T}\right)^\gamma \eta_2 \times a_\mathrm{max} = \left(\frac{0.4}{0.467}\right)^{0.9} 1.0 \times 0.16 = 0.139$$

The characteristic value of the total horizontal seismic action of the structure:

$$F_\mathrm{Ek} = a_1 \times G_\mathrm{Eq} = 0.139 \times 840 = 116.76 \text{ kN} \qquad \text{(Eq. 2.5)}$$

$$\mathrm{T} = 0.467 < 1.4\mathrm{T_g} = 0.56$$

Additional seismic action factors at the top of the building $\delta_n = 0$

Additional horizontal seismic action applied at top of the building $\Delta F_n = \delta_n \times F_{Ek} = 0$

Characteristic value of horizontal seismic action applied at the ith mass:

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n)$$

Story Level	G_i (kN)	H_i (m)	G_iH_i (kN·m)	G_iH_i/ ΣG_iH_i	F_i (kN)	ΔF_n (kN)	V_i (kN)
3rd	280	12	3,360	0.5	58.457	0	58.457
2nd	280	8	2,240	0.333	38.971		97.428
1st	280	4	1,120	0.167	19.486		116.914
Σ	840		6,720	1			

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
3rd	58.457	280	20.88	3.2	1
2nd	97.428	560	17.4	3.2	1
1st	116.914	840	13.92	3.2	1

Results

Table 456: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Story Shear, F_i (kN)	1st	19.486	19.486	negligible	
	2nd	38.971	38.971	negligible	
	3rd	58.457	58.457	negligible	
Base Shear, V (kN)		116.914	116.914	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\GB 50011\GB 50011 2001 Static Seismic - Case2.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 24 TABLE ST HW300X300
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 4 7 10 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE DL
JOINT LOAD
2 3 5 6 8 9 11 TO 16 FY -70
END DEFINE REFERENCE LOADS
DEFINE GB50011 2001 LOAD
INTENSITY 8 FREQUENT GROUP 2 SCLASS 2 GFA 1.0 PX 0.467
REFERENCE LOAD Y
R1 1.0
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
GB50011 LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------

*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR
BUILDINGS            *
   *                              (GB50011:2001) OF CHINA ALONG
X            *
   *  T CALCULATED =  0.685 SEC.      T USER PROVIDED =  0.467
SEC.           *
   *  T USED =  0.467
SEC.                                                       *
   *  MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT =
0.160                   *
   *  CHARACTERISTIC PERIOD =  0.400
SEC.                                        *
   *  DAMPING RATIO =  0.050    POWER INDEX (GAMMA) =
0.900                    *
   *  DAMPING ADJUSTMENT FACTOR (ETA2) =
1.000                                *
   *  ADJUSTING FACTOR (ETA1) =
0.020                                         *
   *  HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) =  0.139
(13.918%)    *
   *  MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) =   0.032
(  3.200%)    *
   *  EQUIVALENT GRAVITY LOAD FACTOR FOR HORIZONTAL SEISMIC ACTION =
1.000    *
   *  TOTAL HORIZONTAL SEISMIC ACTION
=                                         *
   *          =  0.139 X      1.000 X    840.000 =      116.914
KN            *
   *  DESIGN BASE SHEAR =  1.000 X
116.914                                 *
   *                   =      116.914
KN                                      *
```

```
   *  ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) =
0.000                      *

*****************************************************************************
    CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011:5.2.5]
                                                           LOAD -    1
FACTOR -   1.000
    FLOOR            LATERAL          GRAVITY         LAMBDA   LAMBDA
ADJUSTMENT
    HEIGHT (METE)    LOAD (KN  )      LOAD (KN  )      (%)     MIN (%)
FACTOR
    -------------    -----------      ------------    ------   ------
-----------
        12.000           58.457          280.000       20.88     3.20     1.00
         8.000           97.429          560.000       17.40     3.20     1.00
         4.000          116.914          840.000       13.92     3.20     1.00
    JOINT            LATERAL          TORSIONAL            VERTICAL      LOAD
-    1
                     LOAD (KN  )      MOMENT (KN  -METE)   LOAD (KN  )
FACTOR -   1.000
    -----            -------          ---------           -----------
        2     FX        4.871    MY        0.000          FY        0.000
        3     FX        4.871    MY        0.000          FY        0.000
        8     FX        4.871    MY        0.000          FY        0.000
        9     FX        4.871    MY        0.000          FY        0.000
             -----------       -----------          ----------------
          TOTAL =     19.486               0.000                    0.000
AT LEVEL     4.000 METE
      STAAD SPACE                                               -- PAGE NO.
4
        5     FX        9.743    MY        0.000          FY        0.000
        6     FX        9.743    MY        0.000          FY        0.000
       11     FX        9.743    MY        0.000          FY        0.000
       12     FX        9.743    MY        0.000          FY        0.000
             -----------       -----------          ----------------
          TOTAL =     38.971               0.000                    0.000
AT LEVEL     8.000 METE
       13     FX       14.614    MY        0.000          FY        0.000
       14     FX       14.614    MY        0.000          FY        0.000
       15     FX       14.614    MY        0.000          FY        0.000
       16     FX       14.614    MY        0.000          FY        0.000
             -----------       -----------          ----------------
          TOTAL =     58.457               0.000                    0.000
AT LEVEL    12.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    51. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                               -- PAGE NO.
5
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1    0.7616    0.0162    0.0000    0.0000    0.0000   -0.0017
      3    1    0.7616   -0.0162    0.0000    0.0000    0.0000   -0.0017
      4    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1    1.7446    0.0252    0.0000    0.0000    0.0000   -0.0015
      6    1    1.7446   -0.0252    0.0000    0.0000    0.0000   -0.0015
```

```
      7    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      8    1      0.7616    0.0162    0.0000    0.0000    0.0000   -0.0017
      9    1      0.7616   -0.0162    0.0000    0.0000    0.0000   -0.0017
     10    1      0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     11    1      1.7446    0.0252    0.0000    0.0000    0.0000   -0.0015
     12    1      1.7446   -0.0252    0.0000    0.0000    0.0000   -0.0015
     13    1      2.4099    0.0282    0.0000    0.0000    0.0000   -0.0008
     14    1      2.4099   -0.0282    0.0000    0.0000    0.0000   -0.0008
     15    1      2.4099    0.0282    0.0000    0.0000    0.0000   -0.0008
     16    1      2.4099   -0.0282    0.0000    0.0000    0.0000   -0.0008
      STAAD SPACE                                              -- PAGE NO.
6
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT   LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -29.23    -98.49      0.00      0.00      0.00     75.81
      4    1    -29.23     98.49      0.00      0.00      0.00     75.81
      7    1    -29.23    -98.49      0.00      0.00      0.00     75.81
     10    1    -29.23     98.49      0.00      0.00      0.00     75.81
      STAAD SPACE                                              -- PAGE NO.
7
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     -98.49     29.23    -0.00     -0.00      0.00      75.81
                 2      98.49    -29.23     0.00      0.00      0.00      41.10
      2    1     2       0.00    -43.80    -0.00      0.00      0.00     -87.59
                 3      -0.00     43.80     0.00     -0.00      0.00     -87.59
      3    1     3      98.49     29.23    -0.00     -0.00      0.00      41.10
                 4     -98.49    -29.23     0.00      0.00      0.00      75.81
      4    1     2     -54.70     24.36    -0.00     -0.00      0.00      46.49
                 5      54.70    -24.36     0.00      0.00      0.00      50.94
      5    1     5      -0.00    -36.65    -0.00      0.00      0.00     -73.31
                 6       0.00     36.65     0.00     -0.00      0.00     -73.31
      6    1     6      54.70     24.36    -0.00     -0.00      0.00      50.94
                 3     -54.70    -24.36     0.00      0.00      0.00      46.49
      7    1     2      -0.00     -0.00     0.00      0.00     -0.00      -0.00
                 8       0.00      0.00    -0.00     -0.00     -0.00      -0.00
      8    1     3      -0.00      0.00     0.00      0.00     -0.00       0.00
                 9       0.00     -0.00    -0.00     -0.00     -0.00       0.00
      9    1     5       0.00     -0.00     0.00      0.00     -0.00      -0.00
                11      -0.00      0.00    -0.00     -0.00     -0.00      -0.00
     10    1     6      -0.00      0.00     0.00      0.00     -0.00       0.00
                12       0.00     -0.00    -0.00     -0.00     -0.00       0.00
     11    1     7     -98.49     29.23    -0.00     -0.00      0.00      75.81
                 8      98.49    -29.23     0.00      0.00      0.00      41.10
     12    1     8       0.00    -43.80    -0.00      0.00      0.00     -87.59
                 9       0.00     43.80     0.00     -0.00      0.00     -87.59
     13    1     9      98.49     29.23    -0.00     -0.00      0.00      41.10
                10     -98.49    -29.23     0.00      0.00      0.00      75.81
     14    1     8     -54.70     24.36    -0.00     -0.00      0.00      46.49
                11      54.70    -24.36     0.00      0.00      0.00      50.94
     15    1    11      -0.00    -36.65    -0.00      0.00      0.00     -73.31
                12       0.00     36.65     0.00     -0.00      0.00     -73.31
      STAAD SPACE                                              -- PAGE NO.
8
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
```

```
  -----------------
  ALL UNITS ARE -- KN   METE      (LOCAL )
 MEMBER  LOAD  JT      AXIAL    SHEAR-Y  SHEAR-Z    TORSION     MOM-Y       MOM-Z
    16     1   12      54.70      24.36    -0.00      -0.00      0.00       50.94
                9     -54.70     -24.36     0.00       0.00      0.00       46.49
    17     1    5     -18.05      14.61    -0.00      -0.00      0.00       22.37
               13      18.05     -14.61     0.00       0.00      0.00       36.09
    18     1    6      18.05      14.61     0.00      -0.00     -0.00       22.37
               14     -18.05     -14.61    -0.00       0.00     -0.00       36.09
    19     1   11     -18.05      14.61    -0.00      -0.00      0.00       22.37
               15      18.05     -14.61     0.00       0.00      0.00       36.09
    20     1   12      18.05      14.61     0.00      -0.00     -0.00       22.37
               16     -18.05     -14.61    -0.00       0.00     -0.00       36.09
    21     1   13       0.00     -18.05    -0.00       0.00      0.00      -36.09
               14       0.00      18.05     0.00      -0.00      0.00      -36.09
    22     1   13       0.00      -0.00     0.00      -0.00     -0.00       -0.00
               15      -0.00       0.00    -0.00       0.00     -0.00       -0.00
    23     1   14      -0.00       0.00     0.00      -0.00     -0.00        0.00
               16       0.00      -0.00    -0.00       0.00     -0.00        0.00
    24     1   15       0.00     -18.05    -0.00       0.00      0.00      -36.09
               16       0.00      18.05     0.00      -0.00      0.00      -36.09
  ************** END OF LATEST ANALYSIS RESULT **************
```

Related Links

- *TR.31.2.5 Chinese Static Seismic per GB50011-2001* (on page 2369)

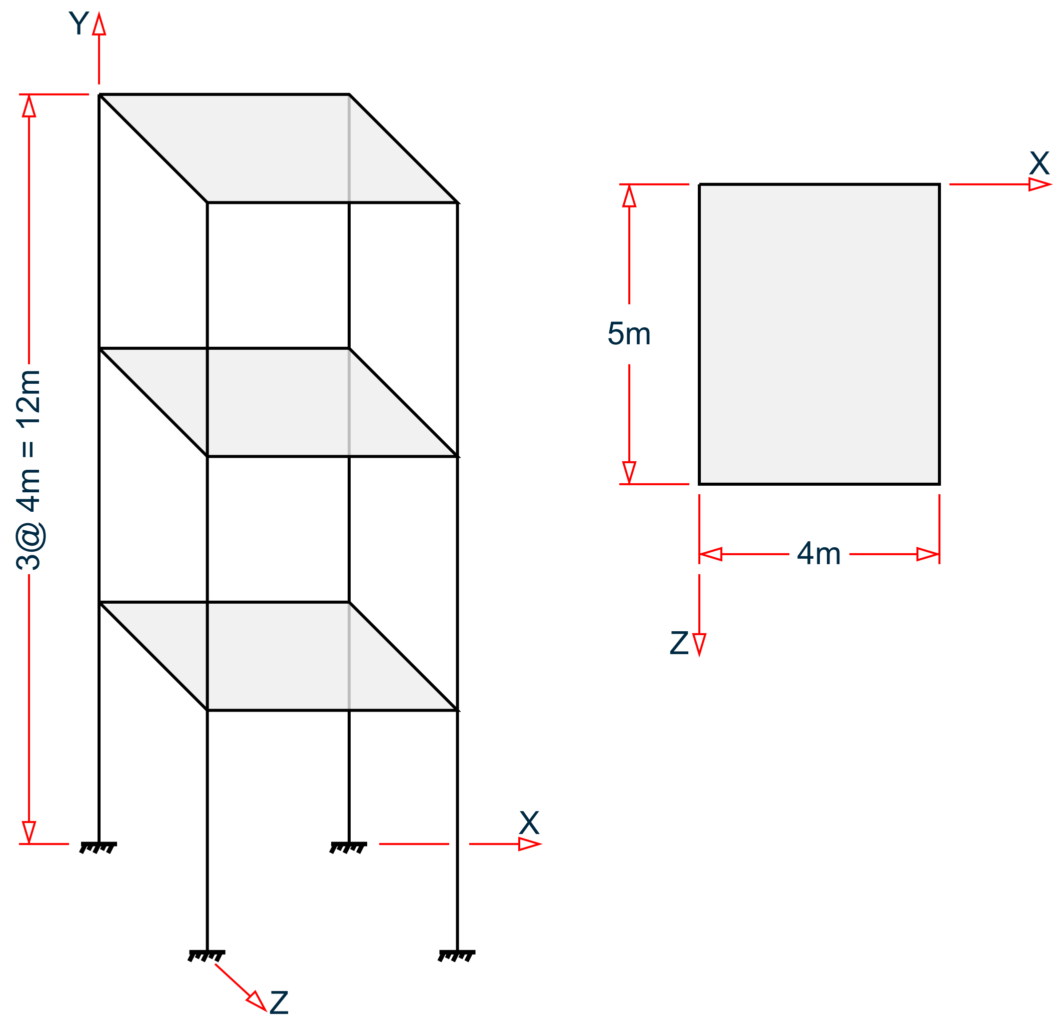
V. GB 50011-2010 Static Seismic - Case1

Calculate the base shear and its distribution along the height using the equivalent lateral force method in GB 50011-2010.

Details

Structure to be modelled:

height = 3× 4m = 12m
plan dimension X = 4m
plan dimension Z = 5m



Input parameters:

Frequency of seismic event: Frequent
Classification of seismic design: Group 2
Seismic design intensity: 8
Site class: II
Equivalent gravity load factor for horizontal seismic action: 0.85 (default in STAAD.Pro)
Damping ratio, $\zeta = 0.05$ (default in STAAD.Pro)

The section of all members is HW300X300, and the seismic weight for each floor is assumed as 280 kN applied as joint weight of 70 kN to nodes 2, 3, 5, 6, 8, 9, and 11 to 16. S

Validation

The effective weight of the structure:

$$G_{\mathrm{Eq}} = 0.85 \times 280\ \mathrm{kN} \times 3\ \mathrm{floors} = 714\ \mathrm{kN}$$

The structure period (as calculated by STAAD.Pro, $T_{calc} = 0.685s$

The max value of the horizontal seismic influence for a “frequent” earthquake with intensity of 8, $a_{max} = 0.16$ (Table 5.1.4-1)

The characteristic period, $T_g = 0.4s$ (Table 5.1.4-2)

The power index of the curvilinear decrease section:

$$\gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} = 0.9$$

The damping adjustment factor:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} = 1.0$$

$$T_g < T < 5T_g$$

The horizontal seismic influence coefficient corresponding to the fundamental period of the structure:

$$a_1 = \left(\frac{T_g}{T}\right)^{\gamma} \eta_2 \times a_{max} = \left(\frac{0.4}{0.685}\right)^{0.9} 1.0 \times 0.16 = 0.099$$

The characteristic value of the total horizontal seismic action of the structure:

$$F_{Ek} = a_1 \times G_{Eq} = 0.099 \times 714 = 70.69 \text{ kN} \quad \text{(Eq. 2.5)}$$

$$T = 0.685 > 1.4T_g = 0.56$$

$$T = 0.35 < T_g < 0.55$$

Additional seismic action factors at the top of the building $\delta_n = 0.08 \times T + 0.01 = 0.065$

Additional horizontal seismic action applied at top of the building $\Delta F_n = \delta_n \times F_{Ek} = 0.065 \times 70.69 = 4.59\ kN$

Characteristic value of horizontal seismic action applied at the ith mass:

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n)$$

Story Level	G_i (kN)	H_i (m)	G_iH_i (kN·m)	$G_iH_i / \Sigma G_iH_i$	F_i (kN)	ΔF_n (kN)	V_i (kN)
3rd	280	12	3,360	0.5	33.053	4.59	37.633
2nd	280	8	2,240	0.333	22.030		59.633
1st	280	4	1,120	0.167	11.015		70.687
Σ	840		6,720	1			

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
3rd	37.633	280	13.44	3.2	1
2nd	59.633	560	10.65	3.2	1

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
1st	70.687	840	8.42	3.2	1

Results

Table 457: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Story Shear, F_i (kN)	1st	11.015	10.971	negligible	
	2nd	22.030	21.943	negligible	
	3rd	37.633	37.476	negligible	
Base Shear, V (kN)		70.687	70.390	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\GB 50011\GB 50011-2010 Static Seismic - Case1.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
       1 0 0 0
       2 0 4 0
       3 4 4 0
       4 4 0 0
       5 0 8 0
       6 4 8 0
       7 0 0 5
       8 0 4 5
       9 4 4 5
      10 4 0 5
      11 0 8 5
      12 4 8 5
      13 0 12 0
      14 4 12 0
      15 0 12 5
      16 4 12 5
MEMBER INCIDENCES
       1         1         2
       2         2         3
       3         3         4
       4         2         5
```

```
       5         5         6
       6         6         3
       7         2         8
       8         3         9
       9         5        11
      10         6        12
      11         7         8
      12         8         9
      13         9        10
      14         8        11
      15        11        12
      16        12         9
      17         5        13
      18         6        14
      19        11        15
      20        12        16
      21        13        14
      22        13        15
      23        14        16
      24        15        16
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 24 TABLE ST HW300X300
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE DL
JOINT LOAD
2 3 5 6 8 9 11 TO 16 FY -70
END DEFINE REFERENCE LOADS
DEFINE GB50011 2010 LOAD
INTENSITY 8 FREQUENT GROUP 2 SCLASS 2 GFA 0.85
REFERENCE LOAD Y
R1 1.0
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
GB50011 LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
```

```
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------

*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR
BUILDINGS             *
   *                              (GB50011:2010) OF CHINA ALONG
X            *
   *  T CALCULATED =  0.685 SEC.      T USER PROVIDED =  0.000
SEC.          *
   *  T USED =  0.685
SEC.                                                  *
   *  MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT =
0.160                  *
   *  CHARACTERISTIC PERIOD =  0.400
SEC.                                  *
   *  DAMPING RATIO =  0.050    POWER INDEX (GAMMA) =
0.900                   *
   *  DAMPING ADJUSTMENT FACTOR (ETA2) =
1.000                               *
   *  ADJUSTING FACTOR (ETA1) =
0.020                                        *
   *  HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) =  0.099
( 9.859%)    *
   *  MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) =   0.032
(  3.200%)    *
   *  EQUIVALENT GRAVITY LOAD FACTOR FOR HORIZONTAL SEISMIC ACTION =
0.850    *
   *  TOTAL HORIZONTAL SEISMIC ACTION
=                                   *
   *          =  0.099 X      0.850 X    840.000 =       70.390
KN            *
   *  DESIGN BASE SHEAR =  1.000 X
70.390                                *
   *                    =       70.390
KN                                     *
   *  ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) =
0.065                      *

*****************************************************************************
    CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011:5.2.5]
                                                           LOAD -    1
FACTOR -   1.000
    FLOOR            LATERAL          GRAVITY         LAMBDA   LAMBDA
ADJUSTMENT
    HEIGHT (METE)    LOAD (KN  )      LOAD (KN  )      (%)     MIN (%)
FACTOR
    -------------    -----------      ------------    ------   ------
-----------
        12.000           37.476          280.000       13.38     3.20     1.00
```

```
         8.000           59.418           560.000       10.61     3.20     1.00
         4.000           70.390           840.000        8.38     3.20     1.00
    JOINT           LATERAL          TORSIONAL            VERTICAL      LOAD
-    1
                    LOAD (KN  )      MOMENT (KN  -METE)   LOAD (KN  )
FACTOR -   1.000
    -----           -------          ---------           -----------
        2     FX        2.743    MY        0.000          FY        0.000
        3     FX        2.743    MY        0.000          FY        0.000
        8     FX        2.743    MY        0.000          FY        0.000
        9     FX        2.743    MY        0.000          FY        0.000
             -----------        -----------          ----------------
          TOTAL =     10.971              0.000                     0.000
AT LEVEL     4.000 METE
      STAAD SPACE                                                -- PAGE NO.
5
        5     FX        5.486    MY        0.000          FY        0.000
        6     FX        5.486    MY        0.000          FY        0.000
       11     FX        5.486    MY        0.000          FY        0.000
       12     FX        5.486    MY        0.000          FY        0.000
             -----------        -----------          ----------------
          TOTAL =     21.943              0.000                     0.000
AT LEVEL     8.000 METE
       13     FX        9.369    MY        0.000          FY        0.000
       14     FX        9.369    MY        0.000          FY        0.000
       15     FX        9.369    MY        0.000          FY        0.000
       16     FX        9.369    MY        0.000          FY        0.000
             -----------        -----------          ----------------
          TOTAL =     37.476              0.000                     0.000
AT LEVEL    12.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    85. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                                -- PAGE NO.
6
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     0.4604    0.0100    0.0000    0.0000    0.0000   -0.0010
      3    1     0.4604   -0.0100    0.0000    0.0000    0.0000   -0.0010
      4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1     1.0622    0.0156    0.0000    0.0000    0.0000   -0.0009
      6    1     1.0622   -0.0156    0.0000    0.0000    0.0000   -0.0009
      7    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      8    1     0.4604    0.0100    0.0000    0.0000    0.0000   -0.0010
      9    1     0.4604   -0.0100    0.0000    0.0000    0.0000   -0.0010
     10    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     11    1     1.0622    0.0156    0.0000    0.0000    0.0000   -0.0009
     12    1     1.0622   -0.0156    0.0000    0.0000    0.0000   -0.0009
     13    1     1.4811    0.0175    0.0000    0.0000    0.0000   -0.0005
     14    1     1.4811   -0.0175    0.0000    0.0000    0.0000   -0.0005
     15    1     1.4811    0.0175    0.0000    0.0000    0.0000   -0.0005
     16    1     1.4811   -0.0175    0.0000    0.0000    0.0000   -0.0005
      STAAD SPACE                                                -- PAGE NO.
7
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
```

```
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -17.60    -60.77      0.00      0.00      0.00     45.74
      4    1    -17.60     60.77      0.00      0.00      0.00     45.74
      7    1    -17.60    -60.77      0.00      0.00      0.00     45.74
     10    1    -17.60     60.77      0.00      0.00      0.00     45.74
      STAAD SPACE                                                -- PAGE NO.
8
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1    -60.77     17.60    -0.00     -0.00      0.00      45.74
                 2     60.77    -17.60     0.00      0.00      0.00      24.65
      2    1     2      0.00    -26.60    -0.00      0.00      0.00     -53.19
                 3      0.00     26.60     0.00     -0.00      0.00     -53.19
      3    1     3     60.77     17.60    -0.00     -0.00      0.00      24.65
                 4    -60.77    -17.60     0.00      0.00      0.00      45.74
      4    1     2    -34.18     14.85    -0.00     -0.00      0.00      28.54
                 5     34.18    -14.85     0.00      0.00      0.00      30.88
      5    1     5      0.00    -22.73    -0.00      0.00      0.00     -45.46
                 6      0.00     22.73     0.00     -0.00      0.00     -45.46
      6    1     6     34.18     14.85    -0.00     -0.00      0.00      30.88
                 3    -34.18    -14.85     0.00      0.00      0.00      28.54
      7    1     2     -0.00     -0.00     0.00      0.00     -0.00      -0.00
                 8      0.00      0.00    -0.00     -0.00     -0.00      -0.00
      8    1     3     -0.00      0.00     0.00      0.00     -0.00       0.00
                 9      0.00     -0.00    -0.00     -0.00     -0.00       0.00
      9    1     5     -0.00     -0.00     0.00      0.00     -0.00      -0.00
                11      0.00      0.00    -0.00     -0.00     -0.00      -0.00
     10    1     6     -0.00      0.00     0.00      0.00     -0.00       0.00
                12      0.00     -0.00    -0.00     -0.00     -0.00       0.00
     11    1     7    -60.77     17.60    -0.00     -0.00      0.00      45.74
                 8     60.77    -17.60     0.00      0.00      0.00      24.65
     12    1     8     -0.00    -26.60    -0.00      0.00      0.00     -53.19
                 9      0.00     26.60     0.00     -0.00      0.00     -53.19
     13    1     9     60.77     17.60    -0.00     -0.00      0.00      24.65
                10    -60.77    -17.60     0.00      0.00      0.00      45.74
     14    1     8    -34.18     14.85    -0.00     -0.00      0.00      28.54
                11     34.18    -14.85     0.00      0.00      0.00      30.88
     15    1    11      0.00    -22.73    -0.00      0.00      0.00     -45.46
                12      0.00     22.73     0.00     -0.00      0.00     -45.46
      STAAD SPACE                                                -- PAGE NO.
9
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     16    1    12     34.18     14.85    -0.00     -0.00      0.00      30.88
                 9    -34.18    -14.85     0.00      0.00      0.00      28.54
     17    1     5    -11.45      9.37    -0.00     -0.00      0.00      14.58
                13     11.45     -9.37     0.00      0.00      0.00      22.89
     18    1     6     11.45      9.37     0.00     -0.00     -0.00      14.58
                14    -11.45     -9.37    -0.00      0.00     -0.00      22.89
     19    1    11    -11.45      9.37    -0.00     -0.00      0.00      14.58
                15     11.45     -9.37     0.00      0.00      0.00      22.89
     20    1    12     11.45      9.37     0.00     -0.00     -0.00      14.58
                16    -11.45     -9.37    -0.00      0.00     -0.00      22.89
     21    1    13      0.00    -11.45    -0.00      0.00      0.00     -22.89
```

```
              14     -0.00      11.45      0.00      -0.00      0.00     -22.89
     22    1  13      0.00      -0.00      0.00      -0.00     -0.00      -0.00
              15     -0.00       0.00     -0.00       0.00     -0.00      -0.00
     23    1  14     -0.00       0.00      0.00      -0.00     -0.00       0.00
              16      0.00      -0.00     -0.00       0.00     -0.00       0.00
     24    1  15      0.00     -11.45     -0.00       0.00      0.00     -22.89
              16      0.00      11.45      0.00      -0.00      0.00     -22.89
   ************** END OF LATEST ANALYSIS RESULT **************
```

Related Links

- *TR.31.2.6 Chinese Static Seismic per GB50011-2010* (on page 2375)

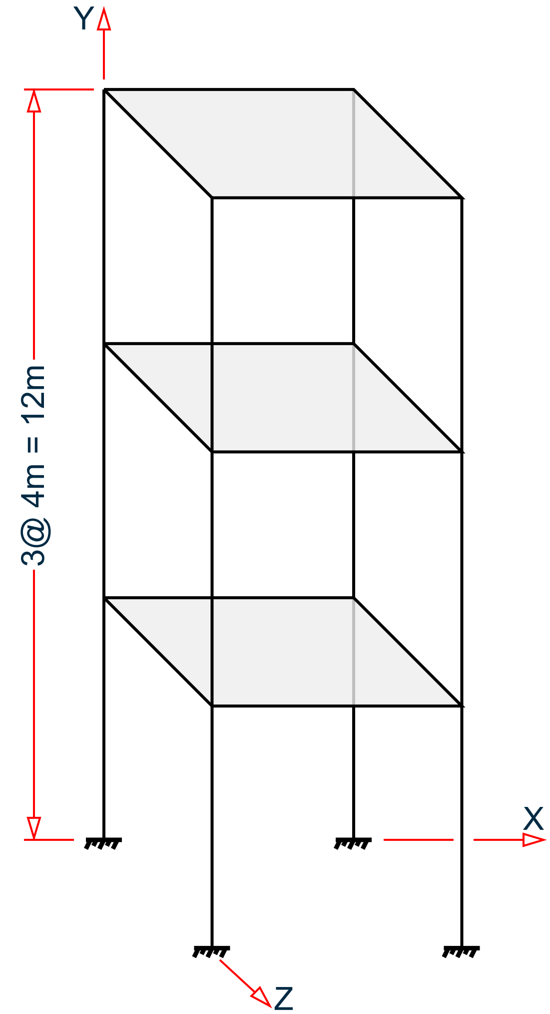
V. GB 50011-2010 Static Seismic - Case2

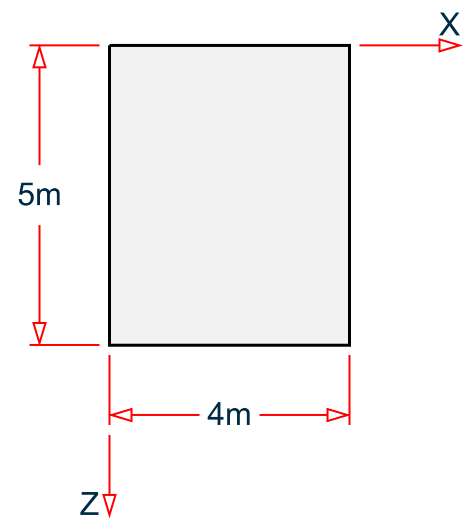
Calculate the base shear and its distribution along the height using the equivalent lateral force method in GB 50011-2010.

Details

Structure to be modelled:

height = 3× 4m = 12m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Frequency of seismic event: Rare
Classification of seismic design: Group 3
Seismic design intensity: 6
Site class: I0
Period in the X direction: $T = 0.40s$
Equivalent gravity load factor for horizontal seismic action: 0.85 (default in STAAD.Pro)
Damping ratio, $\zeta = 0.05$ (default in STAAD.Pro)

The section of all members is HW300X300, and the seismic weight for each floor is assumed as 280 kN applied as joint weight of 70 kN to nodes 2, 3, 5, 6, 8, 9, and 11 to 16. S

Validation

The effective weight of the structure:

$$G_{\mathrm{Eq}} = 0.85 \times 280 \text{ kN} \times 3 \text{ floors} = 714 \text{ kN}$$

The max value of the horizontal seismic influence for a “rare” earthquake with intensity of 6, $a_{max} = 0.28$ (Table 5.1.4-1)

The characteristic period, $T_g = 0.30 + 0.05 = 0.35s$ (Table 5.1.4-2)

The power index of the curvilinear decrease section:

$$\gamma = 0.9 + \frac{0.05 - \zeta}{0.5 + 5\zeta} = 0.9$$

The damping adjustment factor:

$$\eta_2 = 1 + \frac{0.05 - \zeta}{0.06 + 1.7\zeta} = 1.0$$

$$T_g < T < 5T_g$$

The horizontal seismic influence coefficient corresponding to the fundamental period of the structure:

$$a_1 = \left(\frac{T_g}{T}\right)^{\gamma} \eta_2 \times a_{max} = \left(\frac{0.35}{0.4}\right)^{0.9} 1.0 \times 0.28 = 0.248$$

The characteristic value of the total horizontal seismic action of the structure:

$$F_{Ek} = a_1 \times G_{Eq} = 0.248 \times 714 = 177.28 \text{ kN} \qquad \text{(Eq. 2.5)}$$

$$T = 0.4 < 1.4T_g = 0.49$$

Additional seismic action factors at the top of the building $\delta_n = 0$

Additional horizontal seismic action applied at top of the building $\Delta F_n = \delta_n \times F_{Ek} = 0$

Characteristic value of horizontal seismic action applied at the i[th] mass:

$$F_i = \frac{G_i H_i}{\sum_{j=1}^{n} G_j H_j} F_{Ek}(1 - \delta_n)$$

Story Level	G_i (kN)	H_i (m)	G_iH_i (kN·m)	$G_iH_i / \Sigma G_iH_i$	F_i (kN)	ΔF_n (kN)	V_i (kN)
3rd	280	12	3,360	0.5	88.640	0	88.640
2nd	280	8	2,240	0.333	59.093		147.73
1st	280	4	1,120	0.167	29.547		177.28
Σ	840		6,720	1			

Story Level	Lateral Load (kN)	Gravity Load (kN)	λ (%)	λ_{min} (%)	Adjustment Factor
3rd	88.640	280	31.66	0.8	1
2nd	147.73	560	26.38	0.8	1
1st	177.28	840	21.10	0.8	1

Results

Table 458: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Story Shear, F_i (kN)	1st	29.547	29.547	none	
	2nd	59.093	59.094	negligible	
	3rd	88.640	88.641	negligible	
Base Shear, V (kN)		117.28	117.282	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\GB 50011\GB 50011-2010 Static Seismic - Case2.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 24 TABLE ST HW300X300
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 4 7 10 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE DL
JOINT LOAD
2 3 5 6 8 9 11 TO 16 FY -70
END DEFINE REFERENCE LOADS
DEFINE GB50011 2010 LOAD
INTENSITY 6 RARE GROUP 3 SCLASS 0 GFA 0.85 PX 0.4
REFERENCE LOAD Y
R1 1.0
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
GB50011 LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------

*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER SEISMIC DESIGN CODE FOR
BUILDINGS            *
   *                              (GB50011:2010) OF CHINA ALONG
X             *
   *  T CALCULATED =  0.685 SEC.      T USER PROVIDED =  0.400
SEC.          *
   *  T USED =  0.400
SEC.                                                  *
   *  MAX. HORIZONTAL SEISMIC INFLUENCE COEFFICIENT =
0.280                  *
   *  CHARACTERISTIC PERIOD =  0.350
SEC.                                  *
   *  DAMPING RATIO =  0.050    POWER INDEX (GAMMA) =
0.900                   *
   *  DAMPING ADJUSTMENT FACTOR (ETA2) =
1.000                               *
   *  ADJUSTING FACTOR (ETA1) =
0.020                                        *
   *  HORIZONTAL SEISMIC INFLUENCE COEFFICIENT (ALPHA1) =  0.248
(24.829%)    *
   *  MINIMUM SHEAR FACTOR AS PER SEC. 5.2.5 (LAMBDA) =   0.008
(  0.800%)    *
   *  EQUIVALENT GRAVITY LOAD FACTOR FOR HORIZONTAL SEISMIC ACTION =
0.850    *
   *  TOTAL HORIZONTAL SEISMIC ACTION
=                                        *
   *          =  0.248 X      0.850 X    840.000 =      177.282
KN            *
   *  DESIGN BASE SHEAR =  1.000 X
177.282                             *
   *                   =      177.282
KN                                  *
```

```
   *  ADDITIONAL SEISMIC ACTION FACTOR (DELTAN) =
0.000                      *

*****************************************************************************
    CHECK FOR MINIMUM LATERAL FORCE AT EACH FLOOR [GB50011:5.2.5]
                                                           LOAD -    1
FACTOR -   1.000
    FLOOR            LATERAL          GRAVITY         LAMBDA   LAMBDA
ADJUSTMENT
    HEIGHT (METE)    LOAD (KN  )      LOAD (KN  )      (%)     MIN (%)
FACTOR
    -------------    -----------      ------------    ------   ------
-----------
        12.000           88.641          280.000       31.66     0.80     1.00
         8.000          147.735          560.000       26.38     0.80     1.00
         4.000          177.282          840.000       21.10     0.80     1.00
    JOINT            LATERAL          TORSIONAL           VERTICAL      LOAD
-    1
                     LOAD (KN  )      MOMENT (KN  -METE)   LOAD (KN  )
FACTOR -   1.000
    -----            -------          ---------          -----------
       2     FX        7.387    MY        0.000          FY        0.000
       3     FX        7.387    MY        0.000          FY        0.000
       8     FX        7.387    MY        0.000          FY        0.000
       9     FX        7.387    MY        0.000          FY        0.000
                 -----------        -----------          ----------------
          TOTAL =     29.547              0.000                    0.000
AT LEVEL     4.000 METE
      STAAD SPACE                                               -- PAGE NO.
4
       5     FX       14.773    MY        0.000          FY        0.000
       6     FX       14.773    MY        0.000          FY        0.000
      11     FX       14.773    MY        0.000          FY        0.000
      12     FX       14.773    MY        0.000          FY        0.000
                 -----------        -----------          ----------------
          TOTAL =     59.094              0.000                    0.000
AT LEVEL     8.000 METE
      13     FX       22.160    MY        0.000          FY        0.000
      14     FX       22.160    MY        0.000          FY        0.000
      15     FX       22.160    MY        0.000          FY        0.000
      16     FX       22.160    MY        0.000          FY        0.000
                 -----------        -----------          ----------------
          TOTAL =     88.641              0.000                    0.000
AT LEVEL    12.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    51. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                               -- PAGE NO.
5
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     1.1549    0.0246    0.0000    0.0000    0.0000   -0.0025
      3    1     1.1549   -0.0246    0.0000    0.0000    0.0000   -0.0025
      4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1     2.6455    0.0383    0.0000    0.0000    0.0000   -0.0022
      6    1     2.6455   -0.0383    0.0000    0.0000    0.0000   -0.0022
```

```
      7    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      8    1     1.1549    0.0246    0.0000    0.0000    0.0000   -0.0025
      9    1     1.1549   -0.0246    0.0000    0.0000    0.0000   -0.0025
     10    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     11    1     2.6455    0.0383    0.0000    0.0000    0.0000   -0.0022
     12    1     2.6455   -0.0383    0.0000    0.0000    0.0000   -0.0022
     13    1     3.6542    0.0428    0.0000    0.0000    0.0000   -0.0012
     14    1     3.6542   -0.0428    0.0000    0.0000    0.0000   -0.0012
     15    1     3.6542    0.0428    0.0000    0.0000    0.0000   -0.0012
     16    1     3.6542   -0.0428    0.0000    0.0000    0.0000   -0.0012
      STAAD SPACE                                              -- PAGE NO.
6
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT   LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -44.32   -149.35      0.00      0.00      0.00    114.95
      4    1    -44.32    149.35      0.00      0.00      0.00    114.95
      7    1    -44.32   -149.35      0.00      0.00      0.00    114.95
     10    1    -44.32    149.35      0.00      0.00      0.00    114.95
      STAAD SPACE                                              -- PAGE NO.
7
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1     1     1   -149.35     44.32    -0.00     -0.00      0.00     114.95
                  2    149.35    -44.32     0.00      0.00      0.00      62.33
      2     1     2      0.00    -66.41    -0.00      0.00      0.00    -132.82
                  3     -0.00     66.41     0.00     -0.00      0.00    -132.82
      3     1     3    149.35     44.32    -0.00     -0.00      0.00      62.33
                  4   -149.35    -44.32     0.00      0.00      0.00     114.95
      4     1     2    -82.94     36.93    -0.00     -0.00      0.00      70.49
                  5     82.94    -36.93     0.00      0.00      0.00      77.24
      5     1     5     -0.00    -55.58    -0.00      0.00      0.00    -111.16
                  6      0.00     55.58     0.00     -0.00      0.00    -111.16
      6     1     6     82.94     36.93    -0.00     -0.00      0.00      77.24
                  3    -82.94    -36.93     0.00      0.00      0.00      70.49
      7     1     2     -0.00     -0.00     0.00      0.00     -0.00      -0.00
                  8      0.00      0.00    -0.00     -0.00     -0.00      -0.00
      8     1     3     -0.00      0.00     0.00      0.00     -0.00       0.00
                  9      0.00     -0.00    -0.00     -0.00     -0.00       0.00
      9     1     5     -0.00     -0.00     0.00      0.00     -0.00      -0.00
                 11      0.00      0.00    -0.00     -0.00     -0.00      -0.00
     10     1     6      0.00      0.00     0.00      0.00     -0.00       0.00
                 12     -0.00     -0.00    -0.00     -0.00     -0.00       0.00
     11     1     7   -149.35     44.32    -0.00     -0.00      0.00     114.95
                  8    149.35    -44.32     0.00      0.00      0.00      62.33
     12     1     8     -0.00    -66.41    -0.00      0.00      0.00    -132.82
                  9      0.00     66.41     0.00     -0.00      0.00    -132.82
     13     1     9    149.35     44.32    -0.00     -0.00      0.00      62.33
                 10   -149.35    -44.32     0.00      0.00      0.00     114.95
     14     1     8    -82.94     36.93    -0.00     -0.00      0.00      70.49
                 11     82.94    -36.93     0.00      0.00      0.00      77.24
     15     1    11      0.00    -55.58    -0.00      0.00      0.00    -111.16
                 12      0.00     55.58     0.00     -0.00      0.00    -111.16
      STAAD SPACE                                              -- PAGE NO.
8
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
```

```
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     16     1    12     82.94    36.93    -0.00     -0.00      0.00      77.24
                  9    -82.94   -36.93     0.00      0.00      0.00      70.49
     17     1     5    -27.36    22.16    -0.00     -0.00      0.00      33.91
                 13     27.36   -22.16     0.00      0.00      0.00      54.73
     18     1     6     27.36    22.16     0.00     -0.00     -0.00      33.91
                 14    -27.36   -22.16    -0.00      0.00     -0.00      54.73
     19     1    11    -27.36    22.16    -0.00     -0.00      0.00      33.91
                 15     27.36   -22.16     0.00      0.00      0.00      54.73
     20     1    12     27.36    22.16     0.00     -0.00     -0.00      33.91
                 16    -27.36   -22.16    -0.00      0.00     -0.00      54.73
     21     1    13      0.00   -27.36    -0.00      0.00      0.00     -54.73
                 14      0.00    27.36     0.00     -0.00      0.00     -54.73
     22     1    13      0.00    -0.00     0.00     -0.00     -0.00      -0.00
                 15     -0.00     0.00    -0.00      0.00     -0.00      -0.00
     23     1    14      0.00     0.00     0.00     -0.00     -0.00       0.00
                 16     -0.00    -0.00    -0.00      0.00     -0.00       0.00
     24     1    15      0.00   -27.36    -0.00      0.00      0.00     -54.73
                 16     -0.00    27.36     0.00     -0.00      0.00     -54.73
   ************** END OF LATEST ANALYSIS RESULT **************
    52. FINISH
      STAAD SPACE                                                 -- PAGE NO.
9
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= JAN  3,2022   TIME= 17: 8:14 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *TR.31.2.6 Chinese Static Seismic per GB50011-2010* (on page 2375)

V. IBC / ASCE 7

V. ASCE 7 Geometric Irregularity

Verify geometric irregularity (reentrant corners in plan) check in accordance to the ASCE 7-16 specifications.

Details

A two story structure is modelled with diaphragms at the story levels.

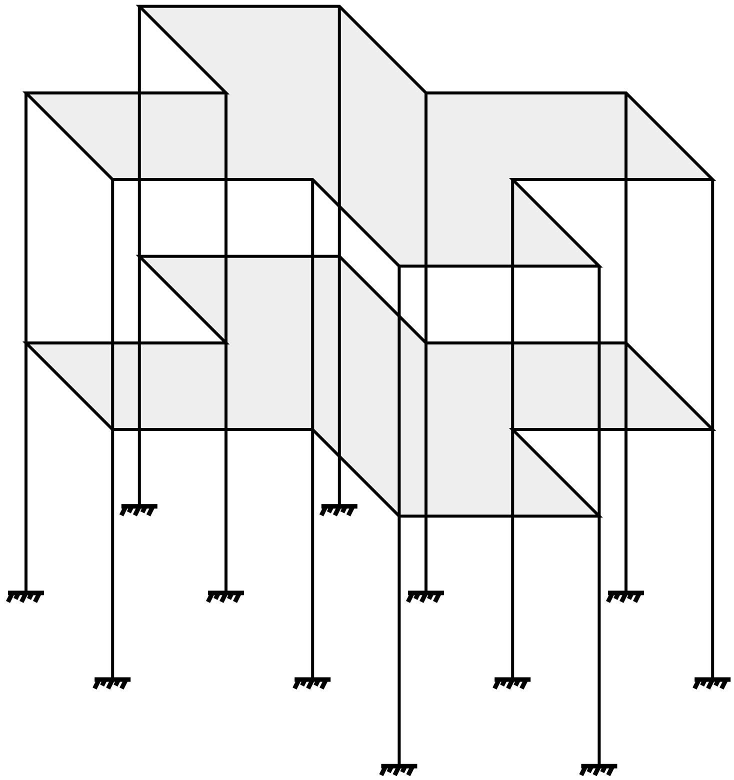


Figure 421: Isometric View

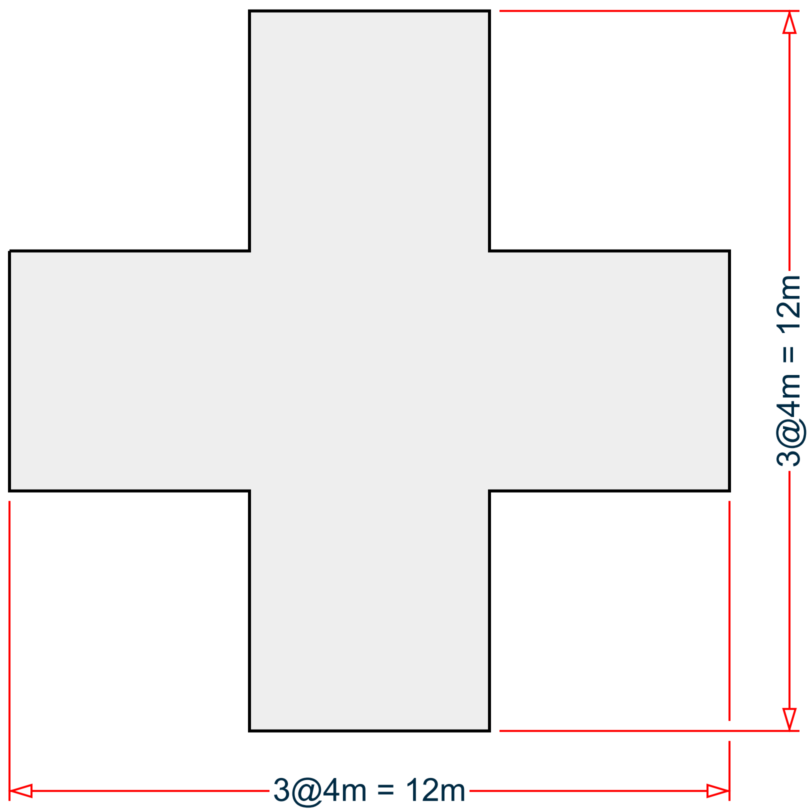


Figure 422: Plan View

Seismic parameters:

Zip code = 92887
Importance factor, I = 1.0
Response reduction factor in X, Rx = 3
Response reduction factor in Z, Rz = 4
Site class A
Long period, TL = 8 sec

Verification

Both diaphragms have the same dimensions and are doubly-symmetric. The overall building width in either direction is 12 m and the potential reentrant corner sides are 4 m. Therefore, the ratio of the $A/L = 4\ m\ /\ 12\ m = 0.333 > 0.15$.

Thus, each of the four corners are reentrant corners per the ASCE 7-16 code.

Results

Table 459: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 1	Reentrant	Reentrant	none	
Node 4	Reentrant	Reentrant	none	
Node 7	Reentrant	Reentrant	none	
Node 10	Reentrant	Reentrant	none	
Node 25	Reentrant	Reentrant	none	
Node 28	Reentrant	Reentrant	none	
Node 31	Reentrant	Reentrant	none	
Node 34	Reentrant	Reentrant	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Geometric Irregularity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 3 0 0; 2 7 0 0; 3 7 0 4; 4 3 0 4; 5 3 0 -4; 6 -1 0 -4; 7 -1 0 0; 8 -5 0 0;
9 -5 0 4; 10 -1 0 4; 11 -1 0 8; 12 3 0 8; 13 3 -5 0; 14 7 -5 0; 15 7 -5 4;
16 3 -5 4; 17 3 -5 -4; 18 -1 -5 -4; 19 -1 -5 0; 20 -5 -5 0; 21 -5 -5 4;
22 -1 -5 4; 23 -1 -5 8; 24 3 -5 8; 25 3 5 0; 26 7 5 0; 27 7 5 4; 28 3 5 4;
29 3 5 -4; 30 -1 5 -4; 31 -1 5 0; 32 -5 5 0; 33 -5 5 4; 34 -1 5 4; 35 -1 5 8;
36 3 5 8;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 1 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10; 10 10 11;
11 11 12; 12 12 4; 13 1 13; 14 2 14; 15 3 15; 16 4 16; 17 5 17; 18 6 18;
19 7 19; 20 8 20; 21 9 21; 22 10 22; 23 11 23; 24 12 24; 25 1 25; 26 2 26;
27 3 27; 28 4 28; 29 5 29; 30 6 30; 31 7 31; 32 8 32; 33 9 33; 34 10 34;
35 11 35; 36 12 36; 37 25 26; 38 26 27; 39 27 28; 40 25 29; 41 29 30; 42 30
31;
43 31 32; 44 32 33; 45 33 34; 46 34 35; 47 35 36; 48 36 28;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
```

```
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 48 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
13 TO 24 FIXED
MEMBER CRACKED
13 TO 36 REDUCTION RIY 0.7 RIZ 0.7
1 TO 12 37 TO 48 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 0
DIA 2 TYPE RIG HEI 5
*CHECK IRREGULARITIES CODE IS1893 2016
CHECK IRREGULARITIES CODE ASCE7
DEFINE IBC 2018
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 1 TL 8
LOAD 1 LOADTYPE None  TITLE STATIC_X
IBC LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
IBC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
 -IRREGULARITY CHECKS
                    STAAD.PRO IRREGULARITIES CHECK - (  ASCE7-2016   )   v1.0
                    ********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
 Ref: Fig. C12.3-1 T1- Ratio Limit(s): 1.20, 1.40
 ------------------------------------------------
 Dia.   Extreme Points of Dia in X        Extreme Points of Dia in Z
       Node    Disp.    Node    Disp.    Node    Disp.     Node    Disp.
               (mm)             (mm)             (mm)              (mm)
 ------------------------------------------------------------------------
    1     12   0.07676      6   0.07676      2   0.07676      8   0.07676
    2     36   0.26545     30   0.26545     26   0.26545     32   0.26545
             Diaphragm  D X-max/avg  D Z-max/avg Status
             --------------------------------------
                   1     1.0000   1.0000        OK
                   2     1.0000   1.0000        OK
 --GEOMETRY IRREGULARITY CHECKS
 Re-Entrant Corner Check.
 (Ref: Fig. C12.3-1 T2- Ratio Limit: 0.15 )
 ------------------------------------------
```

```
    Node      Re-Entrant X-Proj  X-Proj/Lx Z-Proj  Z-Proj/Lz  Status
 Connectivilty    Node    ( m)             ( m)
----------------------------------------------------------------------
        2->        1      4.0000  0.3333    0.0000  0.0000 Re-Entrant
        5                 0.0000  0.0000    4.0000  0.3333
        6->        7      0.0000  0.0000    4.0000  0.3333 Re-Entrant
        8                 4.0000  0.3333    0.0000  0.0000
        9->       10      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       11                 0.0000  0.0000    4.0000  0.3333
       12->        4      0.0000  0.0000    4.0000  0.3333 Re-Entrant
        3                 4.0000  0.3333    0.0000  0.0000
       26->       25      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       29                 0.0000  0.0000    4.0000  0.3333
       30->       31      0.0000  0.0000    4.0000  0.3333 Re-Entrant
       32                 4.0000  0.3333    0.0000  0.0000
       33->       34      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       35                 0.0000  0.0000    4.0000  0.3333
       36->       28      0.0000  0.0000    4.0000  0.3333 Re-Entrant
       27                 4.0000  0.3333    0.0000  0.0000
                  Diaphragm:    Lx:       Lz:
                               ( m)     ( m)
                  ----------------------------
                       1     12.0000   12.0000
                       2     12.0000   12.0000
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Fig. C12.3-2 T2- Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     0.000    636.163    459.451    Base      1.385   N/A     OK
   2     5.000    459.451    Top        636.163    N/A   0.722    OK
```

V. ASCE 7 Mass Irregularity

Verify mass irregularity check in accordance to the ASCE 7-16 specifications.

Details

A three story structure is modelled with floor diaphragms defined at the story levels.

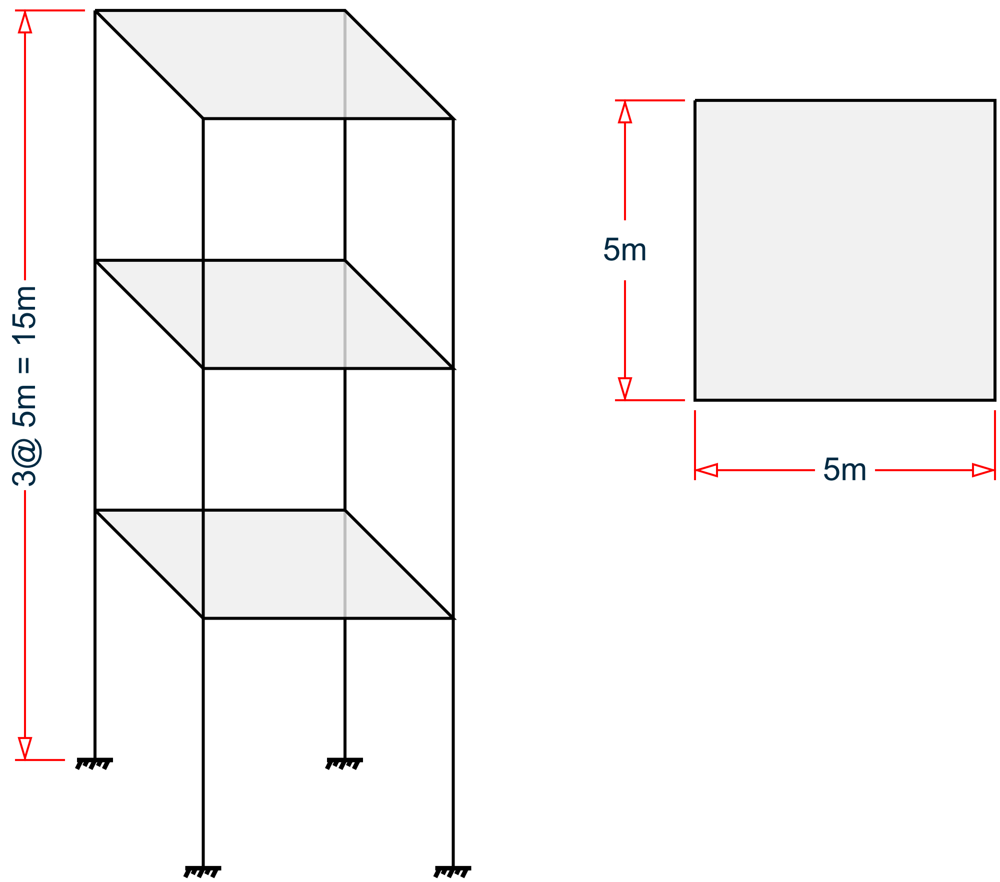


Figure 423: Isometric and Plan View of Structure

Seismic parameters:

Zip code = 92887
Importance factor, I = 1.0
Response reduction factor in X, Rx = 3
Response reduction factor in Z, Rz = 4
Site class A
Long period, TL = 8 sec

Verification

Diaphragm 1 = 1st Floor: Weight = 10 x 4 = 40 kN

Diaphragm 2 = 2nd Floor: Weight = 16 x 4 = 64 kN

Diaphragm 3 = 3rd Floor: Weight = 23 x 4 = 92 kN

Diaphragm	Current Floor Weight (KN)	Above Floor Weight (KN)	Below Floor Weight (KN)	Current Floor Weight/Above Floor Weight	Current Floor Weight/Below Floor Weight	Status (FAIL if ratio > 1.5)
1	40	64	N.A.	0.625	N.A.	OK
2	64	92	40	0.696	1.6	FAIL
3	92	N.A.	64	N.A.	1.4375	OK

Results

Table 460: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Diaphragm 1	Ratio of 1st/2nd floor weights	0.625	0.625	none	
	Check	OK	OK	none	
Diaphragm 2	Ratio of 2nd/3rd floor weights	0.696	0.696	none	
	Check	OK	OK	none	
	Ratio of 2nd/1st floor weights	1.6	1.6	none	
	Check	Fail	Fail	none	
Diaphragm 3	Ratio of 3rd/2nd floor weights	1.438	1.437	negligible	
	Check	OK	OK	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Mass Irregularity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
```

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
MEMBER CRACKED
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 3 6 7 FY -10
9 TO 12 FY -16
13 TO 16 FY -23
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
*CHECK IRREGULARITIES CODE IS1893 2016
CHECK IRREGULARITIES CODE ASCE7
DEFINE IBC 2018
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 1 TL 8
LOAD 1 LOADTYPE None  TITLE STATIC_X
IBC LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
IBC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
 -IRREGULARITY CHECKS
                    STAAD.PRO IRREGULARITIES CHECK - (  ASCE7-2016   )   v1.0
                    *********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
 Ref: Fig. C12.3-1 T1- Ratio Limit(s): 1.20, 1.40
 -------------------------------------------------
 Dia.   Extreme Points of Dia in X        Extreme Points of Dia in Z
       Node    Disp.    Node    Disp.    Node    Disp.     Node    Disp.
```

```
              (mm)             (mm)             (mm)              (mm)
------------------------------------------------------------------------
   1      7   0.24199      2   0.24199      3   0.24199      2   0.24199
   2     12   0.85035      9   0.85035     10   0.85035      9   0.85035
   3     16   1.62595     13   1.62595     14   1.62595     13   1.62595
            Diaphragm  D X-max/avg  D Z-max/avg Status
            --------------------------------------
                  1     1.0000   1.0000        OK
                  2     1.0000   1.0000        OK
                  3     1.0000   1.0000        OK
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Fig. C12.3-1 T2- Ratio Limit: 0.15 )
------------------------------------------
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Fig. C12.3-2 T2- Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     5.000     40.000     64.000    Base      0.625   N/A     OK
   2    10.000     64.000     92.000     40.000   0.696  1.600   FAIL
   3    15.000     92.000    Top         64.000    N/A   1.437    OK
```

V.ASCE 7 Torsion Irregularity

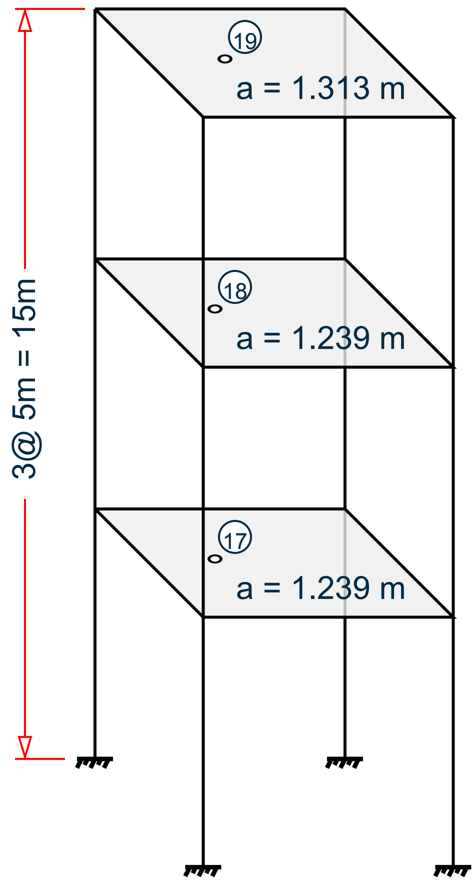
Verify the torsional irregularity check per the ASCE 7-16 specifications.

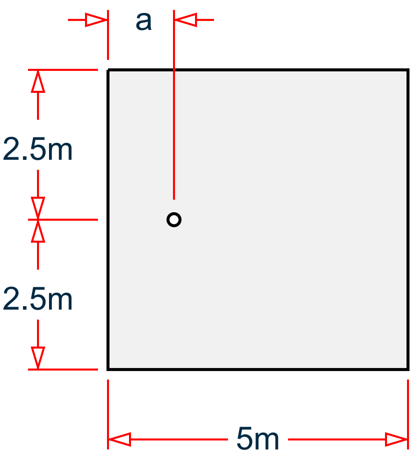
Details

A three story building is modelled with floor diaphragms defined at each floor level.

A 1 kip (4.4482 kN) load is applied to each diaphragm in both the global X and Z directions. In STAAD.Pro, this is modelled using an analytical node in each diaphragm. This is used to obtain the displacements in the nodes of the diaphragm.

Nodes 17, 18, and 19 are the control nodes in each diaphragm.





Validation

The displacements induced because of the applied loads on the control nodes for the nodes situated on the extremities of each diaphragm is taken from the STAAD.Pro output for each lateral direction.

Table 461: X direction checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)		∆max/ ∆min	Status
			Extreme Node 1	Extreme Node 2	∆x Extreme Node 1	∆x Extreme Node 2		
1	17	1	2 or 3	6 or 7	0.0482 × 10 = 0.482	0.0482 × 10 = 0.482	0.482/0.482 = 1	OK
2	18	2	9 or 10	11 or 12	0.2443 × 10 = 2.443	0.2443 × 10 = 2.443	2.443/2.443 = 1	OK
3	19	3	13 or 14	15 or 16	0.6090 × 10 = 6.090	0.6090 × 10 = 6.090	6.090/6.090 = 1	OK

Table 462: Z direction checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)		Δmax/ Δmin	Status
			Extreme Node 1	Extreme Node 2	Δz Extreme Node 1	Δz Extreme Node 2		
1	17	4	2 or 6	3 or 7	0.0234 × 10 = 0.234	0.0361 × 10 = 0.361	0.361/0.234 = 1.543	FAIL
2	18	5	9 or 11	10 or 12	0.0919 × 10 = 0.919	0.1243 × 10 = 1.243	1.243/0.919 = 1.353	OK
3	19	6	13 or 15	14 or 16	0.1884 × 10 = 1.884	0.2441 × 10 = 2.441	2.441/1.884 = 1.296	OK

Results

Table 463: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
X Direction	1	OK	OK	none	
	2	OK	OK	none	
	3	OK	OK	none	
Z Direction	1	Fail	Fail	none	
	2	OK	OK	none	
	3	OK	OK	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Torsion Irregularity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5; 17 1.239 5 2.5; 18 1.239 10 2.5; 19 1.313 15 2.5
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
```

```
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
2 3 5 7 8 10 12 13 15 16 18 20 21 23 24 PRIS YD 0.35 ZD 0.25
MEMBER PROPERTY AMERICAN
1 4 6 9 11 14 17 19 22 PRIS YD 0.5 ZD 0.65
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
MEMBER CRACKED
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
*CHECK IRREGULARITIES CODE IS1893 2016
CHECK IRREGULARITIES CODE ASCE7
DEFINE IBC 2018
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 1 TL 8
LOAD 1 LOADTYPE None  TITLE Diaphragm 1 Unit Load FX
JOINT LOAD
17 FX 4.4482
LOAD 2 LOADTYPE None  TITLE Diaphragm 2 Unit Load FX
JOINT LOAD
18 FX 4.4482
LOAD 3 LOADTYPE None  TITLE Diaphragm 3 Unit Load FX
JOINT LOAD
19 FX 4.4482
LOAD 4 LOADTYPE None  TITLE Diaphragm 1 Unit Load FZ
JOINT LOAD
17 FZ 4.4482
LOAD 5 LOADTYPE None  TITLE Diaphragm 2 Unit Load FZ
JOINT LOAD
18 FZ 4.4482
LOAD 6 LOADTYPE None  TITLE Diaphragm 3 Unit Load FZ
JOINT LOAD
19 FZ 4.4482
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
```

```
PRINT DIA CR
FINISH
```

STAAD.Pro Output

```
                    STAAD.PRO IRREGULARITIES CHECK - (  ASCE7-2016   )   v1.0
                    *********************************************************
--TORSION IRREGULARITY CHECKS
Torsion Irregularity Check
Ref: Fig. C12.3-1 T1- Ratio Limit(s): 1.20, 1.40
------------------------------------------------
Dia.   Extreme Points of Dia in X        Extreme Points of Dia in Z
      Node    Disp.    Node    Disp.    Node    Disp.     Node    Disp.
              (mm)             (mm)             (mm)              (mm)
------------------------------------------------------------------------
   1      7   0.48231      2   0.48231      3   0.36107      2   0.23351
   2     12   2.44302      9   2.44302     10   1.24284      9   0.91874
   3     16   6.08970     13   6.08970     14   2.44124     13   1.88424
            Diaphragm  D X-max/avg  D Z-max/avg Status
            --------------------------------------
                  1     1.0000   1.2145       FAIL
                  2     1.0000   1.1499        OK
                  3     1.0000   1.1288        OK
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Fig. C12.3-1 T2- Ratio Limit: 0.15 )
------------------------------------------
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Fig. C12.3-2 T2- Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     5.000    166.404    166.404    Base      1.000   N/A     OK
   2    10.000    166.404    117.808    166.404   1.412  1.000    OK
   3    15.000    117.808    Top        166.404    N/A   0.708    OK
```

V.IBC 2018 Static Seismic T 1.2

Verify the program-calculated base shear and its distribution along the height of a three-story frame by using the equivalent lateral force method per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

Details

A three-story structure is subject to a seismic load from the +X direction.

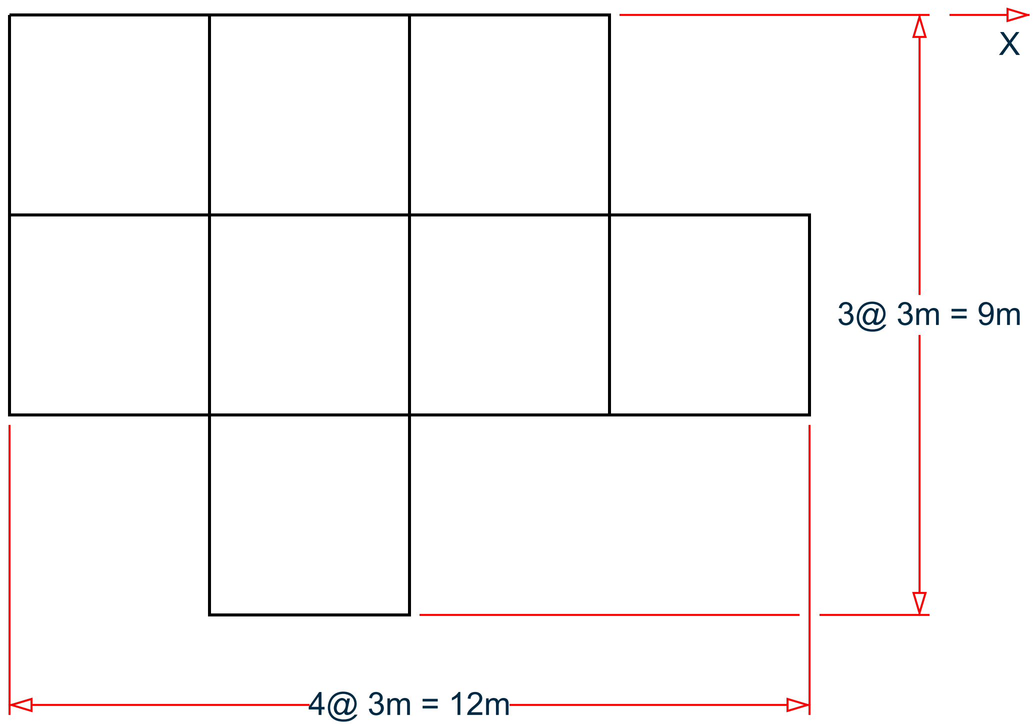


Figure 424: Plan View

Assumptions

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 2.02$
ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.795$
iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor I = 1)
iv. Site Class – D (SCLASS 4)
v. Response Modification Factor (RX & RZ) = 3
vi. Long-period Transition Time (TL)=12 s
vii. Seismic weight is composed of UDLs (magnitude -5 kN/m, direction GY) (defined in Reference Load Definition), incident on the beams
viii. Time Period of the Structure in both directions (PX & PZ) = 1.2 s
ix. Time period coefficients, CTX = CTZ = 0.28
x. Exponents in time period equation: XX = XZ = 1
xi. The effect of shear deformation is neglected

Validation

Calculation of Base Shear

Based on S_s and S_1, $F_a = 1$ (per table 11.4-1) and $F_v = 1.7$ (per table 11.4-2).

Hence:

$$S_{MS} = F_a \text{ x } S_s = 1 \text{ x } 2.02 = 2.02 \qquad \text{Eq. 11.4-1}$$

$$S_{DS} = 2/3 \times S_{MS} = 1.347 \qquad \text{Eq. 11.4-3}$$

$S_{M1} = F_v \times S_1 = 1.7 \times 0.795 = 1.382$ Eq. 11.4-3

$S_{D1} = 2/3 \times S_{M1} = 0.901$ Eq. 11.4-4

$$(C_s)_{\text{initial}} = \frac{S_{DS}}{R / I} = 0.4489$$ Eq. 12.8-2

The natural period of the structure, $T_N = 1.2\ s$ (`TX 1.2`, provides as user input).

Height of the structure, $h = 9\ m$.

So the approximate time period, $T_a = C_t \times h^x = 0.28(9m)^{1.0} = 2.52\ s$

From Table 12.8-1, $C_u = 1.4$ (for $S_{D1} > 0.4$).

$C_u \times T_a = 3.528\ s$

The time period used, T_R , is the lesser of T_N and $C_u \times T_a$, which is $1.2\ s$ ($< T_L = 12\ s$).

$$(C_s)_{\max} = \frac{S_{D1}}{T \times (R / I)} = 0.25028$$ Eq. 12.8-3

$(C_S)_{\min1} = 0.044 \times S_{DS} \times I = 0.059$ Eq. 12.8-5

Since $S_1 = 0.795 > 0.6g$, equation 12.8-6 also needs to be considered for calculating the lower limit of C_s :

$$(C_s)_{\min2} = \frac{0.5 \times S_1}{R / I} = 0.1325$$ Eq. 12.8-6

Therefore, $(C_s)_{min} = 0.1325 < C_s$.

$C_s = 1.5 \times 0.2503 = 0.3754$

The seismic weight, W is taken as the total seismic weight of the beams: $= 5 \times 3 \times 69 = 1{,}035\ kN$

So the total base shear, $V = C_S \times W = 0.3754 \times 1{,}035 = 388.56\ kN$

<u>Vertical Distribution of Lateral Forces</u>

Since, time period of the structure $T = 1.2\ s > 0.5\ s$ and $T < 2.5\ s$, as per clause 12.8.3, value of k needs to be linearly interpolated.

k = 1.35

Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

Table 464: Vertical force distribution

Story Level	W_x **(kN)**	h_x **(m)**	$W_x \times h_x^{\ k}$	$\frac{W_x \times h_x^k}{\sum_{i=1}^{3}\left(W_i \times h_i^k\right)}$	**Lateral force at story level (kN)**
Roof	5 × 3 × 23 = 345	9	6,700	0.554	1.5×143.48 = 215.22
2nd	345	6	3,875	0.320	1.5×82.999 = 124.50
1st	345	3	1,520	0.126	1.5×32.560 = 48.840
Σ	1,035	-	12,095	1	388.56

<u>Consideration of Inherent Torsion (Clause 12.8.4.1)</u>

Floor Level – Roof (9 m)

From output file, CRZ = 3. 831 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = 0.017 m

Floor Level – 2nd (6 m)

From output file , CRZ = 3.858 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = -0.01 m

Floor Level – 1st (3 m)

From output file , CRZ = 3.902 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = -0.054 m

<u>Consideration of Accidental Torsion (Clause 12.8.4.2)</u>

At all floor levels:

$$0.05 \times L_z = 0.05 \times 9\text{m} = 0.45 \text{ m}$$

where

L_z = the dimension of the structure along the global Z axis

Hence, total eccentricity to inherent and accidental torsion at roof level $e_r = (0.45 + 0.017) = 0.467\ m$

Total eccentricity to inherent and accidental torsion at 2nd floor level $e_2 = (0.45 - 0.01) = 0.44\ m$

Total eccentricity to inherent and accidental torsion at 1st floor level $e_1 = (0.45 - 0.054) = 0.396\ m$

Total torsional moment at roof level = $F_{roof} \times e_r = 215.20 \times 0.467 = 100.51\ kN{\cdot}m$

Total Torsional moment at 2nd floor level = $F_{2nd} \times e_2 = 124.50 \times 0.440 = 54.779$ kN·m

Total torsional moment at 1st floor level = $F_{1st} \times e_1 = 48.84 \times 0.396 = 12.893$ kN·m

Results

Table 465: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Base shear, V (kN)	388.56	388.56	none	
Lateral force at roof level (kN)	215.22	215.220	none	
Lateral force at 2nd floor (kN)	124.50	124.497	negligible	
Lateral force at 1st floor (kN)	48.84	48.839	negligible	
Torsional moment at roof level (kN·m)	100.51	100.544	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Torsional moment at 2nd floor (kN·m)	54.779	54.739	negligible	
Torsional moment at 1st floor (kN·m)	19.340	19.312	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Static Seismic T 1.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Mar-19
END JOB INFORMATION
*****************************************************************************
*This problem is created to verify the base shear, distribution of base shear
*And Inherent and Accidental Torsional Moment at different floor levels
*Of the Structure
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 0 3; 6 0 3 3; 7 3 3 3; 8 3 0 3;
9 0 0 6; 10 0 3 6; 11 3 3 6; 12 3 0 6; 13 0 0 9; 14 0 3 9; 15 3 3 9; 16 3 0 9;
17 6 3 0; 18 6 0 0; 19 6 3 3; 20 6 0 3; 21 6 3 6; 22 6 0 6; 25 9 3 3; 26 9 0
3;
27 9 3 6; 28 9 0 6; 33 0 6 0; 34 3 6 0; 35 0 6 3; 36 3 6 3; 37 0 6 6; 38 3 6
6;
39 0 6 9; 40 3 6 9; 41 6 6 0; 42 6 6 3; 43 6 6 6; 45 9 6 3; 46 9 6 6; 49 0 9
0;
50 3 9 0; 51 0 9 3; 52 3 9 3; 53 0 9 6; 54 3 9 6; 55 0 9 9; 56 3 9 9; 57 6 9
0;
58 6 9 3; 59 6 9 6; 70 9 9 3; 71 9 9 6; 72 -3 0 0; 73 -3 3 0; 74 -3 0 3;
75 -3 3 3; 76 -3 0 6; 77 -3 3 6; 80 -3 6 0; 81 -3 6 3; 82 -3 6 6; 84 -3 9 0;
85 -3 9 3; 86 -3 9 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 10 14; 15 11 15; 16 13 14; 17 14 15; 18 15 16;
19 3 17; 20 7 19; 21 11 21; 23 17 18; 24 17 19; 25 19 20; 26 19 21; 27 21 22;
30 19 25; 31 21 27; 32 25 26; 33 25 27; 34 27 28; 40 2 33; 41 3 34; 42 6 35;
43 7 36; 44 10 37; 45 11 38; 46 14 39; 47 15 40; 48 17 41; 49 19 42; 50 21 43;
52 25 45; 53 27 46; 56 33 34; 57 33 35; 58 34 36; 59 35 36; 60 35 37; 61 36
38;
62 37 38; 63 37 39; 64 38 40; 65 39 40; 66 34 41; 67 36 42; 68 38 43; 70 41
42;
71 42 43; 73 42 45; 74 43 46; 75 45 46; 79 33 49; 80 34 50; 81 35 51; 82 36
52;
83 37 53; 84 38 54; 85 39 55; 86 40 56; 87 41 57; 88 42 58; 89 43 59; 95 49
50;
96 49 51; 97 50 52; 98 51 52; 99 51 53; 100 52 54; 101 53 54; 102 53 55;
103 54 56; 104 55 56; 105 50 57; 106 52 58; 107 54 59; 109 57 58; 110 58 59;
```

```
128 45 70; 129 46 71; 132 58 70; 133 59 71; 134 70 71; 135 2 73; 136 6 75;
137 10 77; 139 33 80; 140 35 81; 141 37 82; 143 49 84; 144 51 85; 145 53 86;
147 72 73; 148 73 75; 149 74 75; 150 75 77; 151 76 77; 154 73 80; 155 75 81;
156 77 82; 158 80 81; 159 81 82; 161 80 84; 162 81 85; 163 82 86; 165 84 85;
166 85 86;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 21 23 TO 27 30 TO 34 40 TO 50 52 53 56 TO 68 70 71 73 TO 75 79 TO 89 -
95 TO 107 109 110 128 129 132 TO 137 139 TO 141 143 TO 145 147 TO 151 154 -
155 TO 156 158 159 161 TO 163 165 166 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 9 12 13 16 18 20 22 26 28 72 74 76 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
MEMBER LOAD
2 4 5 7 9 10 12 14 15 17 19 TO 21 24 26 30 31 33 56 TO 68 70 71 73 TO 75 95 -
96 TO 107 109 110 132 TO 137 139 TO 141 143 TO 145 148 150 158 159 165 -
166 UNI GY -5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
DEFINE IBC 2018
SS 2.02 S1 0.795 I 1 RX 3 RZ 3 SCLASS 4 CTX 0.28 CTZ 0.28 PX 1.2 PZ 1.2 TL -
12 XX 1 XZ 1
LOAD 1 LOADTYPE Seismic  TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05
PERFORM ANALYSIS PRINT LOAD DATA
PRINT DIA CR
FINISH
```

STAAD.Pro Output

```
*****************************************************************************
   * EQUIV. SEISMIC LOADS AS PER IBC
2018                                    *
   * PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD
GENERATION                    *
   *  SS =  2.020 S1 =  0.795 FA =  1.000 FV =
1.700                        *
   *  SDS =  1.347 SD1 =
0.901                                            *

*****************************************************************************
```

```
   73. LOAD 1 LOADTYPE SEISMIC  TITLE SL +X
   74. IBC LOAD X 1 DEC 2 ACC 0.05
   75. PERFORM ANALYSIS PRINT LOAD DATA
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         67  NUMBER OF MEMBERS     117
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     162
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
     STAAD SPACE                                               -- PAGE NO.
4
  LOADING     1  LOADTYPE SEISMIC  TITLE SL +X
  -----------
  ************************************************************
  *  IBC 2018 SEISMIC LOAD ALONG X  :                        *
  *     CT =  0.280 Cu =  1.400 x =  1.0000                  *
  *  TIME PERIODS :                                          *
  *     Ta =  2.520 T =  1.200 Tuser = 1.200                 *
  *  TIME PERIOD USED (T) =  1.200                           *
  *  Cs LIMITS : LOWER =  0.133 UPPER =  0.375               *
  *  LOAD FACTOR          =  1.000                           *
  *  DESIGN BASE SHEAR    =  1.000 X  0.375 X      1035.00   *
  *                       =       388.56  KN                 *
  ************************************************************
  ************************************************************************
  ***NOTE: SEISMIC LOAD IS ACTING AT CENTER OF MASS FOR RIGID DIAPHRAGM.
           TORSION FROM STATIC ECCENTRICITY (esi) IS INCLUDED IN ANALYSIS.
           DYNAMIC ECCENTRICITY APPLIED = DEC - 1
  LOAD NO.:     1  DIRECTION : X   UNIT - METE
  STORY  LEVEL     DYN. ECC. (dec)     ACC. ECC. (aec)     DESIGN     ECC.
  -----  -----     ---------------     ---------------     ---------------
                     X         Z         X         Z         X         Z
                                                       dec + aec dec + aec
    1     3.00     -0.05     -0.05      0.60      0.45      0.00      0.40
    2     6.00      0.01     -0.01      0.60      0.45      0.00      0.44
    3     9.00      0.05      0.02      0.60      0.45      0.00      0.47
  ************************************************************************
   JOINT            LATERAL          TORSIONAL           LOAD -    1
                    LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
   -----            -------          ---------
                                     DEC + AEC
     STAAD SPACE                                               -- PAGE NO.
5
      2     FX        3.185    MY        1.260
      3     FX        3.185    MY        1.260
      6     FX        4.247    MY        1.679
      7     FX        4.247    MY        1.679
     10     FX        4.247    MY        1.679
     11     FX        4.247    MY        1.679
     14     FX        2.123    MY        0.840
     15     FX        2.123    MY        0.840
     17     FX        2.123    MY        0.840
     19     FX        4.247    MY        1.679
     21     FX        3.185    MY        1.260
     25     FX        2.123    MY        0.840
```

```
     27     FX        2.123    MY        0.840
     73     FX        2.123    MY        0.840
     75     FX        3.185    MY        1.260
     77     FX        2.123    MY        0.840
                -----------        -----------
         TOTAL =     48.839             19.312 AT LEVEL     3.000 METE
     33     FX        8.119    MY        3.570
     34     FX        8.119    MY        3.570
     35     FX       10.826    MY        4.760
     36     FX       10.826    MY        4.760
     37     FX       10.826    MY        4.760
     38     FX       10.826    MY        4.760
     39     FX        5.413    MY        2.380
     40     FX        5.413    MY        2.380
     41     FX        5.413    MY        2.380
     42     FX       10.826    MY        4.760
     43     FX        8.119    MY        3.570
     45     FX        5.413    MY        2.380
     46     FX        5.413    MY        2.380
     80     FX        5.413    MY        2.380
     81     FX        8.119    MY        3.570
     82     FX        5.413    MY        2.380
                -----------        -----------
         TOTAL =    124.497             54.739 AT LEVEL     6.000 METE
     49     FX       14.036    MY        6.557
     50     FX       14.036    MY        6.557
     51     FX       18.715    MY        8.743
     52     FX       18.715    MY        8.743
     53     FX       18.715    MY        8.743
     54     FX       18.715    MY        8.743
     55     FX        9.357    MY        4.371
     56     FX        9.357    MY        4.371
     57     FX        9.357    MY        4.371
     58     FX       18.715    MY        8.743
     59     FX       14.036    MY        6.557
     70     FX        9.357    MY        4.371
     71     FX        9.357    MY        4.371
     84     FX        9.357    MY        4.371
     85     FX       14.036    MY        6.557
     86     FX        9.357    MY        4.371
     STAAD SPACE                                               -- PAGE NO.
6
                -----------        -----------
         TOTAL =    215.220            100.544 AT LEVEL     9.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    76. PRINT DIA CR
  DIA      CR
   ************************************************************
   CENTRE OF RIGIDITY     UNIT - METE
   ------------------     -----------
   DIAPHRAM      FL. LEVEL      X-COORDINATE      Z-COORDINATE
       1           3.000           2.393             3.902
       2           6.000           2.335             3.858
       3           9.000           2.299             3.831
   ************************************************************
```

Related Links

- *TR.31.2.17 IBC 2018 Seismic Load Definition* (on page 2416)

V.IBC 2018 Static Seismic T Greater Than 2.5

Verify the program-calculated base shear and its distribution along the height of a 22-story frame by using equivalent lateral force method as per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

Details

A 22-story structure is subject to a seismic load from the +X direction (see following figure). Each story is 10' in height (total building height of 220'). The column bases are assumed fixed. The time period of the structure uses is its Rayleigh frequency.

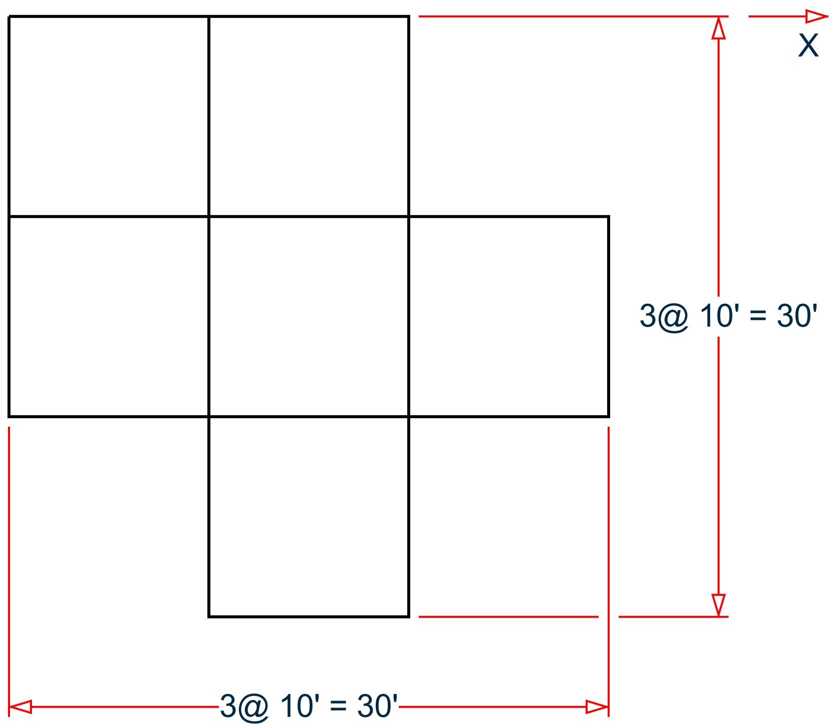


Figure 427: Plan View

<u>Assumptions</u>

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 1.5$
ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.5$
iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor I = 1)
iv. Site Class – D (SCLASS 4)
v. Response Modification Factor (RX & RZ) = 3
vi. Long-period Transition Time (TL)=12 s
vii. Seismic weight is composed of UDLs (magnitude -22.481 kip/ft, direction GY) (defined in Reference Load Definition), incident on the beams
viii. The effect of shear deformation is neglected

Validation

Calculation of Base Shear

Based on S_s and S_1, $F_a = 1$ (per table 11.4-1) and $F_v = 1.8$ (per table 11.4-2).

Hence:

$S_{MS} = F_a \text{ x } S_s = 1 \text{ x } 1.5 = 1.5$ Eq. 11.4-1

$S_{DS} = 2/3 \times S_{MS} = 1.0$ Eq. 11.4-3

$S_{M1} = F_v \text{ x } S_1 = 1.8 \times 0.5 = 0.9$ Eq. 11.4-3

$S_{D1} = 2/3 \times S_{M1} = 0.60$ Eq. 11.4-4

$$(C_s)_{\text{initial}} = \frac{S_{DS}}{R/I} = 0.333$$ Eq. 12.8-2

The time period of the structure is taken as the Rayleigh frequency of the structure (taken from the output file), $T_R = 18.12$ s.

Height of the structure, $h = 220$ ft.

For steel moment-resisting frames, $C_t = 0.028$ and $x = 0.8$ (per Table 12.8-2).

So the approximate time period, $T_a = C_t \times h^x = 0.028(220 \text{ ft})^{0.8} = 2.0946$ s

From Table 12.8-1, $C_u = 1.4$ (for $S_{D1} > 0.4$).

$C_u \times T_a = 2.9324$ s

The time period used, T, is the lesser of T_R and $C_u \times T_a$, which is 2.9324 s ($< T_L = 12$ s).

$$(C_s)_{\max} = \frac{S_{D1}}{T \times (R/I)} = 0.0682$$ Eq. 12.8-3

$(C_S)_{\min1} = 0.044 \times S_{DS} \times I = 0.044$ Eq. 12.8-5

Since $S_1 = 0.5 < 0.6g$, equation 12.8-6 does *not* need to be considered for calculating the lower limit of C_s :

Therefore, $C_s = 1.5 \times 0.0682 = 0.1023$ [*Clause 11.4.8, ii*].

The seismic weight, W is taken as the total seismic weight of the beams: $= 396 \times 22.481 \times 10 = 89{,}025$ *kip*

So the total base shear, $V = C_S \times W = 0.1023 \times 89{,}025 = 9{,}108$ *kip*

Vertical Distribution of Lateral Forces

Since, time period of the structure $T > 2.5$ s, as per clause 12.8.3, $k = 2$.

Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

Table 466: Vertical force distribution

Story Level	W_x (kip)	h_x (ft)	$W_x \times h_x^{\ k}$	$\frac{W_x \times h_x^k}{\sum_{i=1}^{22}(W_i \times h_i^k)}$	Lateral force at story level (kip)
1	4,046.58	10	404,658	0.00026	2.4

Story Level	W_x (kip)	h_x (ft)	$W_x \times h_x{}^k$	$\frac{W_x \times h_x^k}{\sum_{i=1}^{22}\left(W_i \times h_i^k\right)}$	Lateral force at story level (kip)
2	4,046.58	20	1,618,632	0.00105	9.6
3	4,046.58	30	3,641,922	0.00237	21.6
4	4,046.58	40	6,474,528	0.00422	38.4
5	4,046.58	50	10,116,450	0.00659	60.0
6	4,046.58	60	14,567,688	0.00949	86.4
7	4,046.58	70	19,828,242	0.01291	117.6
8	4,046.58	80	25,898,112	0.01686	153.6
9	4,046.58	90	32,777,298	0.02134	194.4
10	4,046.58	100	40,465,800	0.02635	240.0
11	4,046.58	110	48,963,618	0.03188	290.4
12	4,046.58	120	58,270,752	0.03794	345.6
13	4,046.58	130	68,387,202	0.04453	405.6
14	4,046.58	140	79,312,968	0.05165	470.4
15	4,046.58	150	91,048,050	0.05929	540.0
16	4,046.58	160	103,592,448	0.06746	614.4
17	4,046.58	170	116,946,162	0.07615	693.6
18	4,046.58	180	131,109,192	0.08538	777.6
19	4,046.58	190	146,081,538	0.09513	866.4
20	4,046.58	200	161,863,200	0.10540	960.0
21	4,046.58	210	178,454,178	0.11621	1,058
22	4,046.58	220	195,854,472	0.12754	1,162
Σ	89,024.76		1,535,677,110	1	9,108

Consideration of Inherent and Accidental Torsion (as per Clause 12.8.4.1 and 12.8.4.2)

From the output, we can find the Center of Mass and Center of Rigidity values for different story of the structure, numerical difference between the two gives us the static eccentricity. For dynamic eccentricity calculation, the static eccentricity values are to be multiplied with a factor (DEC – 1) (DEC being the dynamic eccentricity factor, provided in the seismic load item in the model = 2), so (DEC-1) = 1. For accidental eccentricity, the multiplying factor ACC (ACC being the multiplication factor for Accidental Torsional Moment, provided in the seismic load item in the model = 0.05). These two eccentricities add up to give the total eccentricity due to inherent as well as accidental torsion. This eccentricity multiplied by the Lateral Force in a particular story level, gives the total torsional moment in that level. Calculations show the results to be:

Floor Level	Center of Mass, CM (ft)	Center of Rigidity CR (ft)	Static Eccentricity $SE = CM-CR$ (ft)	Dynamic Eccentricity $DE = SE \times DEC$ (ft)	Accidental Torsional Eccentricity AE (ft)	DE+AE (ft)	Lateral Force in the Story (kip)	Torsional Moment (kip-ft)
1	13.611	13.76	-0.149	-0.149	0.05 × 30 = 1.5	1.351	2.4	3.24
2	13.611	13.643	-0.032	-0.032	1.5	1.468	9.6	14.1
3	13.611	13.575	0.036	0.036	1.5	1.536	21.6	33.2
4	13.611	13.524	0.087	0.087	1.5	1.587	38.4	60.9
5	13.611	13.481	0.13	0.13	1.5	1.63	60.0	97.8
6	13.611	13.443	0.168	0.168	1.5	1.668	86.4	144.1
7	13.611	13.408	0.203	0.203	1.5	1.703	117.6	200.3
8	13.611	13.378	0.233	0.233	1.5	1.733	153.6	266.2
9	13.611	13.350	0.261	0.261	1.5	1.761	194.4	342.3
10	13.611	13.326	0.285	0.285	1.5	1.785	240.0	428.4
11	13.611	13.305	0.306	0.306	1.5	1.806	290.4	524.4
12	13.611	13.286	0.325	0.325	1.5	1.825	345.6	630.7
13	13.611	13.269	0.342	0.342	1.5	1.842	405.6	747.1
14	13.611	13.254	0.357	0.357	1.5	1.857	470.4	873.5
15	13.611	13.241	0.37	0.37	1.5	1.87	540.0	1,010
16	13.611	13.229	0.382	0.382	1.5	1.882	614.4	1,156
17	13.611	13.217	0.394	0.394	1.5	1.894	693.6	1,314
18	13.611	13.206	0.405	0.405	1.5	1.905	777.6	1,481
19	13.611	13.195	0.416	0.416	1.5	1.916	866.4	1,660

Floor Level	Center of Mass, CM (ft)	Center of Rigidity CR (ft)	Static Eccentricity $SE = CM-CR$ (ft)	Dynamic Eccentricity $DE = SE \times DEC$ (ft)	Accidental Torsional Eccentricity AE (ft)	DE+AE (ft)	Lateral Force in the Story (kip)	Torsional Moment (kip-ft)
20	13.611	13.182	0.429	0.429	1.5	1.929	960.0	1,852
21	13.611	13.167	0.444	0.444	1.5	1.944	1,058.4	2,058
22	13.611	13.146	0.465	0.465	1.5	1.965	1,161.6	2,283

Results

Table 467: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Base shear, V (kN)		9,108	9,107.68	negligible	
Lateral force (kN)	1st Floor	2.4	2.400	none	
	2nd Floor	9.6	9.600	none	
	3rd Floor	21.6	21.599	negligible	
	4th Floor	38.4	38.399	negligible	
	5th Floor	60.0	59.998	negligible	
	6th Floor	86.4	86.397	negligible	
	7th Floor	117.6	117.596	negligible	
	8th Floor	153.6	153.595	negligible	
	9th Floor	194.4	194.393	negligible	
	10th Floor	240.0	239.992	negligible	
	11th Floor	290.4	290.390	negligible	
	12th Floor	345.6	345.588	negligible	
	13th Floor	405.6	405.586	negligible	
	14th Floor	470.4	470.384	negligible	
	15th Floor	540.0	539.981	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	16th Floor	614.4	614.379	negligible	
	17th Floor	693.6	693.576	negligible	
	18th Floor	777.6	777.573	negligible	
	19th Floor	866.4	866.370	negligible	
	20th Floor	960.0	959.967	negligible	
	21st Floor	1,058.4	1,058.363	negligible	
	22nd Floor	1,161.6	1,161.560	negligible	
Torsional moment (kN·m)	1st Floor	3.24	3.243	negligible	
	2nd Floor	14.1	14.095	negligible	
	3rd Floor	33.2	33.185	negligible	
	4th Floor	60.9	60.938	negligible	
	5th Floor	97.8	97.809	negligible	
	6th Floor	144.1	144.158	negligible	
	7th Floor	200.3	200.252	negligible	
	8th Floor	266.2	266.265	negligible	
	9th Floor	342.3	342.301	negligible	
	10th Floor	428.4	428.407	negligible	
	11th Floor	524.4	524.593	negligible	
	12th Floor	630.7	630.846	negligible	
	13th Floor	747.1	747.149	negligible	
	14th Floor	873.5	873.494	negligible	
	15th Floor	1,010	1,009.910	negligible	
	16th Floor	1,156	1,156.487	negligible	
	17th Floor	1,314	1,313.423	negligible	
	18th Floor	1,481	1,481.089	negligible	
	19th Floor	1,660	1,660.120	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	20th Floor	1,852	1,851.580	negligible	
	21st Floor	2,058	2,057.184	negligible	
	22nd Floor	2,283	2,282.172	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Static Seismic T Greater Than 2.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Mar-19
END JOB INFORMATION
*****************************************************************************
*This problem is created to verify the base shear, distribution of base shear
*And Inherent and Accidental Torsional Moment at different floor levels
*Of the Structure
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 0 20; 10 0 10 20; 11 10 10 20; 12 10 0 20; 13 20 10 0;
14 20 0 0; 15 20 10 10; 16 20 0 10; 17 20 10 20; 18 20 0 20; 21 30 10 10;
22 30 0 10; 23 30 10 20; 24 30 0 20; 33 10 10 30; 34 10 0 30; 35 20 10 30;
36 20 0 30; 37 0 20 0; 38 10 20 0; 39 0 20 10; 40 10 20 10; 41 0 20 20;
42 10 20 20; 43 20 20 0; 44 20 20 10; 45 20 20 20; 46 30 20 10; 47 30 20 20;
48 10 20 30; 49 20 20 30; 50 0 30 0; 51 10 30 0; 52 0 30 10; 53 10 30 10;
54 0 30 20; 55 10 30 20; 56 20 30 0; 57 20 30 10; 58 20 30 20; 59 30 30 10;
60 30 30 20; 61 10 30 30; 62 20 30 30; 63 0 40 0; 64 10 40 0; 65 0 40 10;
66 10 40 10; 67 0 40 20; 68 10 40 20; 69 20 40 0; 70 20 40 10; 71 20 40 20;
72 30 40 10; 73 30 40 20; 74 10 40 30; 75 20 40 30; 76 0 50 0; 77 10 50 0;
78 0 50 10; 79 10 50 10; 80 0 50 20; 81 10 50 20; 82 20 50 0; 83 20 50 10;
84 20 50 20; 85 30 50 10; 86 30 50 20; 87 10 50 30; 88 20 50 30; 89 0 60 0;
90 10 60 0; 91 0 60 10; 92 10 60 10; 93 0 60 20; 94 10 60 20; 95 20 60 0;
96 20 60 10; 97 20 60 20; 98 30 60 10; 99 30 60 20; 100 10 60 30; 101 20 60
30;
102 0 70 0; 103 10 70 0; 104 0 70 10; 105 10 70 10; 106 0 70 20; 107 10 70 20;
108 20 70 0; 109 20 70 10; 110 20 70 20; 111 30 70 10; 112 30 70 20;
113 10 70 30; 114 20 70 30; 115 0 80 0; 116 10 80 0; 117 0 80 10; 118 10 80
10;
119 0 80 20; 120 10 80 20; 121 20 80 0; 122 20 80 10; 123 20 80 20;
124 30 80 10; 125 30 80 20; 126 10 80 30; 127 20 80 30; 128 0 90 0;
129 10 90 0; 130 0 90 10; 131 10 90 10; 132 0 90 20; 133 10 90 20; 134 20 90
0;
135 20 90 10; 136 20 90 20; 137 30 90 10; 138 30 90 20; 139 10 90 30;
140 20 90 30; 141 0 100 0; 142 10 100 0; 143 0 100 10; 144 10 100 10;
145 0 100 20; 146 10 100 20; 147 20 100 0; 148 20 100 10; 149 20 100 20;
150 30 100 10; 151 30 100 20; 152 10 100 30; 153 20 100 30; 154 0 110 0;
155 10 110 0; 156 0 110 10; 157 10 110 10; 158 0 110 20; 159 10 110 20;
```

```
160 20 110 0; 161 20 110 10; 162 20 110 20; 163 30 110 10; 164 30 110 20;
165 10 110 30; 166 20 110 30; 167 0 120 0; 168 10 120 0; 169 0 120 10;
170 10 120 10; 171 0 120 20; 172 10 120 20; 173 20 120 0; 174 20 120 10;
175 20 120 20; 176 30 120 10; 177 30 120 20; 178 10 120 30; 179 20 120 30;
180 0 130 0; 181 10 130 0; 182 0 130 10; 183 10 130 10; 184 0 130 20;
185 10 130 20; 186 20 130 0; 187 20 130 10; 188 20 130 20; 189 30 130 10;
190 30 130 20; 191 10 130 30; 192 20 130 30; 193 0 140 0; 194 10 140 0;
195 0 140 10; 196 10 140 10; 197 0 140 20; 198 10 140 20; 199 20 140 0;
200 20 140 10; 201 20 140 20; 202 30 140 10; 203 30 140 20; 204 10 140 30;
205 20 140 30; 206 0 150 0; 207 10 150 0; 208 0 150 10; 209 10 150 10;
210 0 150 20; 211 10 150 20; 212 20 150 0; 213 20 150 10; 214 20 150 20;
215 30 150 10; 216 30 150 20; 217 10 150 30; 218 20 150 30; 219 0 160 0;
220 10 160 0; 221 0 160 10; 222 10 160 10; 223 0 160 20; 224 10 160 20;
225 20 160 0; 226 20 160 10; 227 20 160 20; 228 30 160 10; 229 30 160 20;
230 10 160 30; 231 20 160 30; 232 0 170 0; 233 10 170 0; 234 0 170 10;
235 10 170 10; 236 0 170 20; 237 10 170 20; 238 20 170 0; 239 20 170 10;
240 20 170 20; 241 30 170 10; 242 30 170 20; 243 10 170 30; 244 20 170 30;
245 0 180 0; 246 10 180 0; 247 0 180 10; 248 10 180 10; 249 0 180 20;
250 10 180 20; 251 20 180 0; 252 20 180 10; 253 20 180 20; 254 30 180 10;
255 30 180 20; 256 10 180 30; 257 20 180 30; 258 0 190 0; 259 10 190 0;
260 0 190 10; 261 10 190 10; 262 0 190 20; 263 10 190 20; 264 20 190 0;
265 20 190 10; 266 20 190 20; 267 30 190 10; 268 30 190 20; 269 10 190 30;
270 20 190 30; 271 0 200 0; 272 10 200 0; 273 0 200 10; 274 10 200 10;
275 0 200 20; 276 10 200 20; 277 20 200 0; 278 20 200 10; 279 20 200 20;
280 30 200 10; 281 30 200 20; 282 10 200 30; 283 20 200 30; 284 0 210 0;
285 10 210 0; 286 0 210 10; 287 10 210 10; 288 0 210 20; 289 10 210 20;
290 20 210 0; 291 20 210 10; 292 20 210 20; 293 30 210 10; 294 30 210 20;
295 10 210 30; 296 20 210 30; 297 0 220 0; 298 10 220 0; 299 0 220 10;
300 10 220 10; 301 0 220 20; 302 10 220 20; 303 20 220 0; 304 20 220 10;
305 20 220 20; 306 30 220 10; 307 30 220 20; 308 10 220 30; 309 20 220 30;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 3 13; 15 7 15; 16 11 17; 17 13 14; 18 13 15;
19 15 16; 20 15 17; 21 17 18; 23 15 21; 24 17 23; 27 21 22; 28 21 23; 29 23
24;
39 11 33; 40 17 35; 45 33 34; 46 33 35; 47 35 36; 48 2 37; 49 3 38; 50 6 39;
51 7 40; 52 10 41; 53 11 42; 54 13 43; 55 15 44; 56 17 45; 57 21 46; 58 23 47;
59 33 48; 60 35 49; 61 37 38; 62 37 39; 63 38 40; 64 39 40; 65 39 41; 66 40
42;
67 41 42; 68 38 43; 69 40 44; 70 42 45; 71 43 44; 72 44 45; 73 44 46; 74 45
47;
75 46 47; 76 42 48; 77 45 49; 78 48 49; 79 37 50; 80 38 51; 81 39 52; 82 40
53;
83 41 54; 84 42 55; 85 43 56; 86 44 57; 87 45 58; 88 46 59; 89 47 60; 90 48
61;
91 49 62; 92 50 51; 93 50 52; 94 51 53; 95 52 53; 96 52 54; 97 53 55; 98 54
55;
99 51 56; 100 53 57; 101 55 58; 102 56 57; 103 57 58; 104 57 59; 105 58 60;
106 59 60; 107 55 61; 108 58 62; 109 61 62; 110 50 63; 111 51 64; 112 52 65;
113 53 66; 114 54 67; 115 55 68; 116 56 69; 117 57 70; 118 58 71; 119 59 72;
120 60 73; 121 61 74; 122 62 75; 123 63 64; 124 63 65; 125 64 66; 126 65 66;
127 65 67; 128 66 68; 129 67 68; 130 64 69; 131 66 70; 132 68 71; 133 69 70;
134 70 71; 135 70 72; 136 71 73; 137 72 73; 138 68 74; 139 71 75; 140 74 75;
141 63 76; 142 64 77; 143 65 78; 144 66 79; 145 67 80; 146 68 81; 147 69 82;
148 70 83; 149 71 84; 150 72 85; 151 73 86; 152 74 87; 153 75 88; 154 76 77;
155 76 78; 156 77 79; 157 78 79; 158 78 80; 159 79 81; 160 80 81; 161 77 82;
162 79 83; 163 81 84; 164 82 83; 165 83 84; 166 83 85; 167 84 86; 168 85 86;
169 81 87; 170 84 88; 171 87 88; 172 76 89; 173 77 90; 174 78 91; 175 79 92;
```

```
176 80 93; 177 81 94; 178 82 95; 179 83 96; 180 84 97; 181 85 98; 182 86 99;
183 87 100; 184 88 101; 185 89 90; 186 89 91; 187 90 92; 188 91 92; 189 91 93;
190 92 94; 191 93 94; 192 90 95; 193 92 96; 194 94 97; 195 95 96; 196 96 97;
197 96 98; 198 97 99; 199 98 99; 200 94 100; 201 97 101; 202 100 101;
203 89 102; 204 90 103; 205 91 104; 206 92 105; 207 93 106; 208 94 107;
209 95 108; 210 96 109; 211 97 110; 212 98 111; 213 99 112; 214 100 113;
215 101 114; 216 102 103; 217 102 104; 218 103 105; 219 104 105; 220 104 106;
221 105 107; 222 106 107; 223 103 108; 224 105 109; 225 107 110; 226 108 109;
227 109 110; 228 109 111; 229 110 112; 230 111 112; 231 107 113; 232 110 114;
233 113 114; 234 102 115; 235 103 116; 236 104 117; 237 105 118; 238 106 119;
239 107 120; 240 108 121; 241 109 122; 242 110 123; 243 111 124; 244 112 125;
245 113 126; 246 114 127; 247 115 116; 248 115 117; 249 116 118; 250 117 118;
251 117 119; 252 118 120; 253 119 120; 254 116 121; 255 118 122; 256 120 123;
257 121 122; 258 122 123; 259 122 124; 260 123 125; 261 124 125; 262 120 126;
263 123 127; 264 126 127; 265 115 128; 266 116 129; 267 117 130; 268 118 131;
269 119 132; 270 120 133; 271 121 134; 272 122 135; 273 123 136; 274 124 137;
275 125 138; 276 126 139; 277 127 140; 278 128 129; 279 128 130; 280 129 131;
281 130 131; 282 130 132; 283 131 133; 284 132 133; 285 129 134; 286 131 135;
287 133 136; 288 134 135; 289 135 136; 290 135 137; 291 136 138; 292 137 138;
293 133 139; 294 136 140; 295 139 140; 296 128 141; 297 129 142; 298 130 143;
299 131 144; 300 132 145; 301 133 146; 302 134 147; 303 135 148; 304 136 149;
305 137 150; 306 138 151; 307 139 152; 308 140 153; 309 141 142; 310 141 143;
311 142 144; 312 143 144; 313 143 145; 314 144 146; 315 145 146; 316 142 147;
317 144 148; 318 146 149; 319 147 148; 320 148 149; 321 148 150; 322 149 151;
323 150 151; 324 146 152; 325 149 153; 326 152 153; 327 141 154; 328 142 155;
329 143 156; 330 144 157; 331 145 158; 332 146 159; 333 147 160; 334 148 161;
335 149 162; 336 150 163; 337 151 164; 338 152 165; 339 153 166; 340 154 155;
341 154 156; 342 155 157; 343 156 157; 344 156 158; 345 157 159; 346 158 159;
347 155 160; 348 157 161; 349 159 162; 350 160 161; 351 161 162; 352 161 163;
353 162 164; 354 163 164; 355 159 165; 356 162 166; 357 165 166; 358 154 167;
359 155 168; 360 156 169; 361 157 170; 362 158 171; 363 159 172; 364 160 173;
365 161 174; 366 162 175; 367 163 176; 368 164 177; 369 165 178; 370 166 179;
371 167 168; 372 167 169; 373 168 170; 374 169 170; 375 169 171; 376 170 172;
377 171 172; 378 168 173; 379 170 174; 380 172 175; 381 173 174; 382 174 175;
383 174 176; 384 175 177; 385 176 177; 386 172 178; 387 175 179; 388 178 179;
389 167 180; 390 168 181; 391 169 182; 392 170 183; 393 171 184; 394 172 185;
395 173 186; 396 174 187; 397 175 188; 398 176 189; 399 177 190; 400 178 191;
401 179 192; 402 180 181; 403 180 182; 404 181 183; 405 182 183; 406 182 184;
407 183 185; 408 184 185; 409 181 186; 410 183 187; 411 185 188; 412 186 187;
413 187 188; 414 187 189; 415 188 190; 416 189 190; 417 185 191; 418 188 192;
419 191 192; 420 180 193; 421 181 194; 422 182 195; 423 183 196; 424 184 197;
425 185 198; 426 186 199; 427 187 200; 428 188 201; 429 189 202; 430 190 203;
431 191 204; 432 192 205; 433 193 194; 434 193 195; 435 194 196; 436 195 196;
437 195 197; 438 196 198; 439 197 198; 440 194 199; 441 196 200; 442 198 201;
443 199 200; 444 200 201; 445 200 202; 446 201 203; 447 202 203; 448 198 204;
449 201 205; 450 204 205; 451 193 206; 452 194 207; 453 195 208; 454 196 209;
455 197 210; 456 198 211; 457 199 212; 458 200 213; 459 201 214; 460 202 215;
461 203 216; 462 204 217; 463 205 218; 464 206 207; 465 206 208; 466 207 209;
467 208 209; 468 208 210; 469 209 211; 470 210 211; 471 207 212; 472 209 213;
473 211 214; 474 212 213; 475 213 214; 476 213 215; 477 214 216; 478 215 216;
479 211 217; 480 214 218; 481 217 218; 482 206 219; 483 207 220; 484 208 221;
485 209 222; 486 210 223; 487 211 224; 488 212 225; 489 213 226; 490 214 227;
491 215 228; 492 216 229; 493 217 230; 494 218 231; 495 219 220; 496 219 221;
497 220 222; 498 221 222; 499 221 223; 500 222 224; 501 223 224; 502 220 225;
503 222 226; 504 224 227; 505 225 226; 506 226 227; 507 226 228; 508 227 229;
509 228 229; 510 224 230; 511 227 231; 512 230 231; 513 219 232; 514 220 233;
515 221 234; 516 222 235; 517 223 236; 518 224 237; 519 225 238; 520 226 239;
521 227 240; 522 228 241; 523 229 242; 524 230 243; 525 231 244; 526 232 233;
```

```
527 232 234; 528 233 235; 529 234 235; 530 234 236; 531 235 237; 532 236 237;
533 233 238; 534 235 239; 535 237 240; 536 238 239; 537 239 240; 538 239 241;
539 240 242; 540 241 242; 541 237 243; 542 240 244; 543 243 244; 544 232 245;
545 233 246; 546 234 247; 547 235 248; 548 236 249; 549 237 250; 550 238 251;
551 239 252; 552 240 253; 553 241 254; 554 242 255; 555 243 256; 556 244 257;
557 245 246; 558 245 247; 559 246 248; 560 247 248; 561 247 249; 562 248 250;
563 249 250; 564 246 251; 565 248 252; 566 250 253; 567 251 252; 568 252 253;
569 252 254; 570 253 255; 571 254 255; 572 250 256; 573 253 257; 574 256 257;
575 245 258; 576 246 259; 577 247 260; 578 248 261; 579 249 262; 580 250 263;
581 251 264; 582 252 265; 583 253 266; 584 254 267; 585 255 268; 586 256 269;
587 257 270; 588 258 259; 589 258 260; 590 259 261; 591 260 261; 592 260 262;
593 261 263; 594 262 263; 595 259 264; 596 261 265; 597 263 266; 598 264 265;
599 265 266; 600 265 267; 601 266 268; 602 267 268; 603 263 269; 604 266 270;
605 269 270; 606 258 271; 607 259 272; 608 260 273; 609 261 274; 610 262 275;
611 263 276; 612 264 277; 613 265 278; 614 266 279; 615 267 280; 616 268 281;
617 269 282; 618 270 283; 619 271 272; 620 271 273; 621 272 274; 622 273 274;
623 273 275; 624 274 276; 625 275 276; 626 272 277; 627 274 278; 628 276 279;
629 277 278; 630 278 279; 631 278 280; 632 279 281; 633 280 281; 634 276 282;
635 279 283; 636 282 283; 637 271 284; 638 272 285; 639 273 286; 640 274 287;
641 275 288; 642 276 289; 643 277 290; 644 278 291; 645 279 292; 646 280 293;
647 281 294; 648 282 295; 649 283 296; 650 284 285; 651 284 286; 652 285 287;
653 286 287; 654 286 288; 655 287 289; 656 288 289; 657 285 290; 658 287 291;
659 289 292; 660 290 291; 661 291 292; 662 291 293; 663 292 294; 664 293 294;
665 289 295; 666 292 296; 667 295 296; 668 284 297; 669 285 298; 670 286 299;
671 287 300; 672 288 301; 673 289 302; 674 290 303; 675 291 304; 676 292 305;
677 293 306; 678 294 307; 679 295 308; 680 296 309; 681 297 298; 682 297 299;
683 298 300; 684 299 300; 685 299 301; 686 300 302; 687 301 302; 688 298 303;
689 300 304; 690 302 305; 691 303 304; 692 304 305; 693 304 306; 694 305 307;
695 306 307; 696 302 308; 697 305 309; 698 308 309;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 21 23 24 27 TO 29 39 40 45 TO 698 TABLE ST W12X58
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 5 8 9 12 14 16 18 22 24 34 36 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
MEMBER LOAD
2 4 5 7 9 10 12 14 TO 16 18 20 23 24 28 39 40 46 61 TO 78 92 TO 109 -
123 TO 140 154 TO 171 185 TO 202 216 TO 233 247 TO 264 278 TO 295 -
309 TO 326 340 TO 357 371 TO 388 402 TO 419 433 TO 450 464 TO 481 -
495 TO 512 526 TO 543 557 TO 574 588 TO 605 619 TO 636 650 TO 667 -
681 TO 698 UNI GY -22.481
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 10
DIA 2 TYPE RIG HEI 20
DIA 3 TYPE RIG HEI 30
```

```
DIA 4 TYPE RIG HEI 40
DIA 5 TYPE RIG HEI 50
DIA 6 TYPE RIG HEI 60
DIA 7 TYPE RIG HEI 70
DIA 8 TYPE RIG HEI 80
DIA 9 TYPE RIG HEI 90
DIA 10 TYPE RIG HEI 100
DIA 11 TYPE RIG HEI 110
DIA 12 TYPE RIG HEI 120
DIA 13 TYPE RIG HEI 130
DIA 14 TYPE RIG HEI 140
DIA 15 TYPE RIG HEI 150
DIA 16 TYPE RIG HEI 160
DIA 17 TYPE RIG HEI 170
DIA 18 TYPE RIG HEI 180
DIA 19 TYPE RIG HEI 190
DIA 20 TYPE RIG HEI 200
DIA 21 TYPE RIG HEI 210
DIA 22 TYPE RIG HEI 220
DEFINE IBC 2018
SS 1.5 S1 0.5 I 1 RX 3 RZ 3 SCLASS 4 TL 12
LOAD 1 LOADTYPE Seismic  TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05
PERFORM ANALYSIS PRINT LOAD DATA
PRI DIA CR
FINISH
```

STAAD.Pro Output

```
*****************************************************************************
   * EQUIV. SEISMIC LOADS AS PER IBC
2018                                  *
   * PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD
GENERATION                  *
   *  SS =  1.500 S1 =  0.500 FA =  1.000 FV =
1.800                         *
   *  SDS =  1.000 SD1 =
0.600                                            *

*****************************************************************************
   230. LOAD 1 LOADTYPE SEISMIC  TITLE SL +X
   231. IBC LOAD X 1 DEC 2 ACC 0.05
   232. PERFORM ANALYSIS PRINT LOAD DATA
      STAAD SPACE                                              -- PAGE NO.
7
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS        321  NUMBER OF MEMBERS     682
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     13
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     990
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
      STAAD SPACE                                              -- PAGE NO.
8
```

```
   LOADING     1  LOADTYPE SEISMIC  TITLE SL +X
   -----------
   ************************************************************
   *  IBC 2018 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.028 Cu =  1.400 x =  0.8000                  *
   *  TIME PERIODS :                                          *
   *     Ta =  2.095 T = 18.120 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  2.932                           *
   *  Cs LIMITS : LOWER =  0.044 UPPER =  0.102               *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.102 X     89024.77   *
   *                       =      9107.68  KIP                *
   ************************************************************
   ************************************************************************
      STAAD SPACE                                              -- PAGE NO.
9
   ***NOTE: SEISMIC LOAD IS ACTING AT CENTER OF MASS FOR RIGID DIAPHRAGM.
            TORSION FROM STATIC ECCENTRICITY (esi) IS INCLUDED IN ANALYSIS.
            DYNAMIC ECCENTRICITY APPLIED = DEC - 1
   LOAD NO.:     1  DIRECTION : X   UNIT - FEET
   STORY  LEVEL     DYN. ECC. (dec)     ACC. ECC. (aec)     DESIGN     ECC.
   -----  -----     ---------------     ---------------     ---------------
                     X         Z         X         Z         X         Z
                                                           dec + aec dec + aec
     1    10.00     -0.22     -0.15      1.50      1.50      0.00      1.35
     2    20.00     -0.19     -0.03      1.50      1.50      0.00      1.47
     3    30.00     -0.17      0.04      1.50      1.50      0.00      1.54
     4    40.00     -0.15      0.09      1.50      1.50      0.00      1.59
     5    50.00     -0.14      0.13      1.50      1.50      0.00      1.63
     6    60.00     -0.12      0.17      1.50      1.50      0.00      1.67
     7    70.00     -0.11      0.20      1.50      1.50      0.00      1.70
     8    80.00     -0.10      0.23      1.50      1.50      0.00      1.73
     9    90.00     -0.09      0.26      1.50      1.50      0.00      1.76
    10   100.00     -0.08      0.29      1.50      1.50      0.00      1.79
    11   110.00     -0.07      0.31      1.50      1.50      0.00      1.81
    12   120.00     -0.06      0.33      1.50      1.50      0.00      1.83
    13   130.00     -0.05      0.34      1.50      1.50      0.00      1.84
    14   140.00     -0.04      0.36      1.50      1.50      0.00      1.86
    15   150.00     -0.04      0.37      1.50      1.50      0.00      1.87
    16   160.00     -0.03      0.38      1.50      1.50      0.00      1.88
    17   170.00     -0.02      0.39      1.50      1.50      0.00      1.89
    18   180.00     -0.02      0.40      1.50      1.50      0.00      1.90
    19   190.00     -0.01      0.42      1.50      1.50      0.00      1.92
    20   200.00     -0.01      0.43      1.50      1.50      0.00      1.93
    21   210.00      0.00      0.44      1.50      1.50      0.00      1.94
    22   220.00      0.01      0.46      1.50      1.50      0.00      1.96
   ************************************************************************
    JOINT            LATERAL          TORSIONAL           LOAD -    1
                     LOAD (KIP )      MOMENT (KIP -FEET) FACTOR -   1.000
    -----            -------          ---------
                                      DEC + AEC
      2     FX        0.133    MY        0.180
      3     FX        0.200    MY        0.270
      6     FX        0.200    MY        0.270
      7     FX        0.267    MY        0.360
     10     FX        0.133    MY        0.180
     11     FX        0.267    MY        0.360
     13     FX        0.133    MY        0.180
```

```
      15     FX        0.267    MY        0.360
      17     FX        0.267    MY        0.360
      21     FX        0.133    MY        0.180
      23     FX        0.133    MY        0.180
      33     FX        0.133    MY        0.180
      35     FX        0.133    MY        0.180
                 -----------      -----------
          TOTAL =      2.400              3.243 AT LEVEL    10.000 FEET
      37     FX        0.533    MY        0.783
      38     FX        0.800    MY        1.175
      39     FX        0.800    MY        1.175
      40     FX        1.067    MY        1.566
      41     FX        0.533    MY        0.783
      42     FX        1.067    MY        1.566
      43     FX        0.533    MY        0.783
      44     FX        1.067    MY        1.566
      45     FX        1.067    MY        1.566
      46     FX        0.533    MY        0.783
      47     FX        0.533    MY        0.783
      48     FX        0.533    MY        0.783
      49     FX        0.533    MY        0.783
                 -----------      -----------
          TOTAL =      9.600             14.095 AT LEVEL    20.000 FEET
      50     FX        1.200    MY        1.844
      51     FX        1.800    MY        2.765
      52     FX        1.800    MY        2.765
      53     FX        2.400    MY        3.687
      54     FX        1.200    MY        1.844
      STAAD SPACE                                              -- PAGE NO.
10
      55     FX        2.400    MY        3.687
      56     FX        1.200    MY        1.844
      57     FX        2.400    MY        3.687
      58     FX        2.400    MY        3.687
      59     FX        1.200    MY        1.844
      60     FX        1.200    MY        1.844
      61     FX        1.200    MY        1.844
      62     FX        1.200    MY        1.844
                 -----------      -----------
          TOTAL =     21.599             33.185 AT LEVEL    30.000 FEET
      63     FX        2.133    MY        3.385
      64     FX        3.200    MY        5.078
      65     FX        3.200    MY        5.078
      66     FX        4.267    MY        6.771
      67     FX        2.133    MY        3.385
      68     FX        4.267    MY        6.771
      69     FX        2.133    MY        3.385
      70     FX        4.267    MY        6.771
      71     FX        4.267    MY        6.771
      72     FX        2.133    MY        3.385
      73     FX        2.133    MY        3.385
      74     FX        2.133    MY        3.385
      75     FX        2.133    MY        3.385
                 -----------      -----------
          TOTAL =     38.399             60.938 AT LEVEL    40.000 FEET
      76     FX        3.333    MY        5.434
      77     FX        5.000    MY        8.151
      78     FX        5.000    MY        8.151
```

```
      79      FX         6.666    MY        10.868
      80      FX         3.333    MY         5.434
      81      FX         6.666    MY        10.868
      82      FX         3.333    MY         5.434
      83      FX         6.666    MY        10.868
      84      FX         6.666    MY        10.868
      85      FX         3.333    MY         5.434
      86      FX         3.333    MY         5.434
      87      FX         3.333    MY         5.434
      88      FX         3.333    MY         5.434
                   -----------        -----------
          TOTAL =     59.998             97.809 AT LEVEL    50.000 FEET
      89      FX         4.800    MY         8.009
      90      FX         7.200    MY        12.013
      91      FX         7.200    MY        12.013
      92      FX         9.600    MY        16.018
      93      FX         4.800    MY         8.009
      94      FX         9.600    MY        16.018
      95      FX         4.800    MY         8.009
      96      FX         9.600    MY        16.018
      97      FX         9.600    MY        16.018
      98      FX         4.800    MY         8.009
      99      FX         4.800    MY         8.009
     100      FX         4.800    MY         8.009
     101      FX         4.800    MY         8.009
      STAAD SPACE                                               -- PAGE NO.
11
                   -----------        -----------
          TOTAL =     86.397            144.158 AT LEVEL    60.000 FEET
     102      FX         6.533    MY        11.125
     103      FX         9.800    MY        16.688
     104      FX         9.800    MY        16.688
     105      FX        13.066    MY        22.250
     106      FX         6.533    MY        11.125
     107      FX        13.066    MY        22.250
     108      FX         6.533    MY        11.125
     109      FX        13.066    MY        22.250
     110      FX        13.066    MY        22.250
     111      FX         6.533    MY        11.125
     112      FX         6.533    MY        11.125
     113      FX         6.533    MY        11.125
     114      FX         6.533    MY        11.125
                   -----------        -----------
          TOTAL =    117.596            200.252 AT LEVEL    70.000 FEET
     115      FX         8.533    MY        14.793
     116      FX        12.800    MY        22.189
     117      FX        12.800    MY        22.189
     118      FX        17.066    MY        29.585
     119      FX         8.533    MY        14.793
     120      FX        17.066    MY        29.585
     121      FX         8.533    MY        14.793
     122      FX        17.066    MY        29.585
     123      FX        17.066    MY        29.585
     124      FX         8.533    MY        14.793
     125      FX         8.533    MY        14.793
     126      FX         8.533    MY        14.793
     127      FX         8.533    MY        14.793
                   -----------        -----------
```

```
          TOTAL =    153.595             266.265 AT LEVEL    80.000 FEET
     128      FX        10.800    MY        19.017
     129      FX        16.199    MY        28.525
     130      FX        16.199    MY        28.525
     131      FX        21.599    MY        38.033
     132      FX        10.800    MY        19.017
     133      FX        21.599    MY        38.033
     134      FX        10.800    MY        19.017
     135      FX        21.599    MY        38.033
     136      FX        21.599    MY        38.033
     137      FX        10.800    MY        19.017
     138      FX        10.800    MY        19.017
     139      FX        10.800    MY        19.017
     140      FX        10.800    MY        19.017
                   -----------         -----------
          TOTAL =    194.393             342.301 AT LEVEL    90.000 FEET
     141      FX        13.333    MY        23.800
     142      FX        19.999    MY        35.701
     143      FX        19.999    MY        35.701
     144      FX        26.666    MY        47.601
     145      FX        13.333    MY        23.800
      STAAD SPACE                                               -- PAGE NO.
12
     146      FX        26.666    MY        47.601
     147      FX        13.333    MY        23.800
     148      FX        26.666    MY        47.601
     149      FX        26.666    MY        47.601
     150      FX        13.333    MY        23.800
     151      FX        13.333    MY        23.800
     152      FX        13.333    MY        23.800
     153      FX        13.333    MY        23.800
                   -----------         -----------
          TOTAL =    239.992             428.407 AT LEVEL   100.000 FEET
     154      FX        16.133    MY        29.144
     155      FX        24.199    MY        43.716
     156      FX        24.199    MY        43.716
     157      FX        32.266    MY        58.288
     158      FX        16.133    MY        29.144
     159      FX        32.266    MY        58.288
     160      FX        16.133    MY        29.144
     161      FX        32.266    MY        58.288
     162      FX        32.266    MY        58.288
     163      FX        16.133    MY        29.144
     164      FX        16.133    MY        29.144
     165      FX        16.133    MY        29.144
     166      FX        16.133    MY        29.144
                   -----------         -----------
          TOTAL =    290.390             524.593 AT LEVEL   110.000 FEET
     167      FX        19.199    MY        35.047
     168      FX        28.799    MY        52.571
     169      FX        28.799    MY        52.571
     170      FX        38.399    MY        70.094
     171      FX        19.199    MY        35.047
     172      FX        38.399    MY        70.094
     173      FX        19.199    MY        35.047
     174      FX        38.399    MY        70.094
     175      FX        38.399    MY        70.094
     176      FX        19.199    MY        35.047
```

```
   177     FX       19.199    MY       35.047
   178     FX       19.199    MY       35.047
   179     FX       19.199    MY       35.047
                 -----------        -----------
        TOTAL =    345.588           630.846 AT LEVEL   120.000 FEET
   180     FX       22.533    MY       41.508
   181     FX       33.799    MY       62.262
   182     FX       33.799    MY       62.262
   183     FX       45.065    MY       83.017
   184     FX       22.533    MY       41.508
   185     FX       45.065    MY       83.017
   186     FX       22.533    MY       41.508
   187     FX       45.065    MY       83.017
   188     FX       45.065    MY       83.017
   189     FX       22.533    MY       41.508
   190     FX       22.533    MY       41.508
   191     FX       22.533    MY       41.508
   192     FX       22.533    MY       41.508
    STAAD SPACE                                                -- PAGE NO.
13
                 -----------        -----------
        TOTAL =    405.586           747.149 AT LEVEL   130.000 FEET
   193     FX       26.132    MY       48.527
   194     FX       39.199    MY       72.791
   195     FX       39.199    MY       72.791
   196     FX       52.265    MY       97.055
   197     FX       26.132    MY       48.527
   198     FX       52.265    MY       97.055
   199     FX       26.132    MY       48.527
   200     FX       52.265    MY       97.055
   201     FX       52.265    MY       97.055
   202     FX       26.132    MY       48.527
   203     FX       26.132    MY       48.527
   204     FX       26.132    MY       48.527
   205     FX       26.132    MY       48.527
                 -----------        -----------
        TOTAL =    470.384           873.494 AT LEVEL   140.000 FEET
   206     FX       29.999    MY       56.106
   207     FX       44.998    MY       84.159
   208     FX       44.998    MY       84.159
   209     FX       59.998    MY      112.212
   210     FX       29.999    MY       56.106
   211     FX       59.998    MY      112.212
   212     FX       29.999    MY       56.106
   213     FX       59.998    MY      112.212
   214     FX       59.998    MY      112.212
   215     FX       29.999    MY       56.106
   216     FX       29.999    MY       56.106
   217     FX       29.999    MY       56.106
   218     FX       29.999    MY       56.106
                 -----------        -----------
        TOTAL =    539.981          1009.910 AT LEVEL   150.000 FEET
   219     FX       34.132    MY       64.249
   220     FX       51.198    MY       96.374
   221     FX       51.198    MY       96.374
   222     FX       68.264    MY      128.499
   223     FX       34.132    MY       64.249
   224     FX       68.264    MY      128.499
```

```
     225     FX        34.132    MY       64.249
     226     FX        68.264    MY      128.499
     227     FX        68.264    MY      128.499
     228     FX        34.132    MY       64.249
     229     FX        34.132    MY       64.249
     230     FX        34.132    MY       64.249
     231     FX        34.132    MY       64.249
                  -----------       -----------
          TOTAL =    614.379          1156.487 AT LEVEL   160.000 FEET
     232     FX        38.532    MY       72.968
     233     FX        57.798    MY      109.452
     234     FX        57.798    MY      109.452
     235     FX        77.064    MY      145.936
     236     FX        38.532    MY       72.968
      STAAD SPACE                                        -- PAGE NO.
14
     237     FX        77.064    MY      145.936
     238     FX        38.532    MY       72.968
     239     FX        77.064    MY      145.936
     240     FX        77.064    MY      145.936
     241     FX        38.532    MY       72.968
     242     FX        38.532    MY       72.968
     243     FX        38.532    MY       72.968
     244     FX        38.532    MY       72.968
                  -----------       -----------
          TOTAL =    693.576          1313.423 AT LEVEL   170.000 FEET
     245     FX        43.198    MY       82.283
     246     FX        64.798    MY      123.424
     247     FX        64.798    MY      123.424
     248     FX        86.397    MY      164.565
     249     FX        43.198    MY       82.283
     250     FX        86.397    MY      164.565
     251     FX        43.198    MY       82.283
     252     FX        86.397    MY      164.565
     253     FX        86.397    MY      164.565
     254     FX        43.198    MY       82.283
     255     FX        43.198    MY       82.283
     256     FX        43.198    MY       82.283
     257     FX        43.198    MY       82.283
                  -----------       -----------
          TOTAL =    777.573          1481.089 AT LEVEL   180.000 FEET
     258     FX        48.132    MY       92.229
     259     FX        72.197    MY      138.343
     260     FX        72.197    MY      138.343
     261     FX        96.263    MY      184.458
     262     FX        48.132    MY       92.229
     263     FX        96.263    MY      184.458
     264     FX        48.132    MY       92.229
     265     FX        96.263    MY      184.458
     266     FX        96.263    MY      184.458
     267     FX        48.132    MY       92.229
     268     FX        48.132    MY       92.229
     269     FX        48.132    MY       92.229
     270     FX        48.132    MY       92.229
                  -----------       -----------
          TOTAL =    866.370          1660.120 AT LEVEL   190.000 FEET
     271     FX        53.331    MY      102.866
     272     FX        79.997    MY      154.298
```

```
     273      FX       79.997    MY      154.298
     274      FX      106.663    MY      205.731
     275      FX       53.331    MY      102.866
     276      FX      106.663    MY      205.731
     277      FX       53.331    MY      102.866
     278      FX      106.663    MY      205.731
     279      FX      106.663    MY      205.731
     280      FX       53.331    MY      102.866
     281      FX       53.331    MY      102.866
     282      FX       53.331    MY      102.866
     283      FX       53.331    MY      102.866
      STAAD SPACE                                        -- PAGE NO.
15
                 -----------        -----------
         TOTAL =    959.967            1851.580 AT LEVEL   200.000 FEET
     284      FX       58.798    MY      114.288
     285      FX       88.197    MY      171.432
     286      FX       88.197    MY      171.432
     287      FX      117.596    MY      228.576
     288      FX       58.798    MY      114.288
     289      FX      117.596    MY      228.576
     290      FX       58.798    MY      114.288
     291      FX      117.596    MY      228.576
     292      FX      117.596    MY      228.576
     293      FX       58.798    MY      114.288
     294      FX       58.798    MY      114.288
     295      FX       58.798    MY      114.288
     296      FX       58.798    MY      114.288
                 -----------        -----------
         TOTAL =   1058.363            2057.184 AT LEVEL   210.000 FEET
     297      FX       64.531    MY      126.787
     298      FX       96.797    MY      190.181
     299      FX       96.797    MY      190.181
     300      FX      129.062    MY      253.575
     301      FX       64.531    MY      126.787
     302      FX      129.062    MY      253.575
     303      FX       64.531    MY      126.787
     304      FX      129.062    MY      253.575
     305      FX      129.062    MY      253.575
     306      FX       64.531    MY      126.787
     307      FX       64.531    MY      126.787
     308      FX       64.531    MY      126.787
     309      FX       64.531    MY      126.787
                 -----------        -----------
         TOTAL =   1161.560            2282.172 AT LEVEL   220.000 FEET
   ************ END OF DATA FROM INTERNAL STORAGE ************
   233. PRI DIA CR
  DIA      CR
      STAAD SPACE                                        -- PAGE NO.
16
   ************************************************************
   CENTRE OF RIGIDITY     UNIT - FEET
   ------------------     -----------
   DIAPHRAM      FL. LEVEL      X-COORDINATE      Z-COORDINATE
       1           10.000          13.829            13.760
       2           20.000          13.800            13.643
       3           30.000          13.779            13.575
       4           40.000          13.763            13.524
```

```
       5            50.000            13.748            13.481
       6            60.000            13.734            13.443
       7            70.000            13.721            13.408
       8            80.000            13.709            13.378
       9            90.000            13.698            13.350
      10           100.000            13.688            13.326
      11           110.000            13.679            13.305
      12           120.000            13.670            13.286
      13           130.000            13.663            13.269
      14           140.000            13.655            13.254
      15           150.000            13.649            13.241
      16           160.000            13.642            13.229
      17           170.000            13.636            13.217
      18           180.000            13.630            13.206
      19           190.000            13.624            13.195
      20           200.000            13.617            13.182
      21           210.000            13.609            13.167
      22           220.000            13.602            13.146
   **********************************************************
```

Related Links

- *TR.31.2.17 IBC 2018 Seismic Load Definition* (on page 2416)

V.IBC 2018 Static Seismic T Less Than 0.5

Verify the program-calculated base shear and its distribution along the height of a three-story frame by using the equivalent lateral force method per IBC 2018. Also, verify the torsional moments to which the floors are subjected to, considering inherent as well as accidental torsion.

Details

A three-story structure is subject to a seismic load from the +X direction. The time period of the structure used is its calculated Rayleigh frequency (taken from STAAD.Pro output).

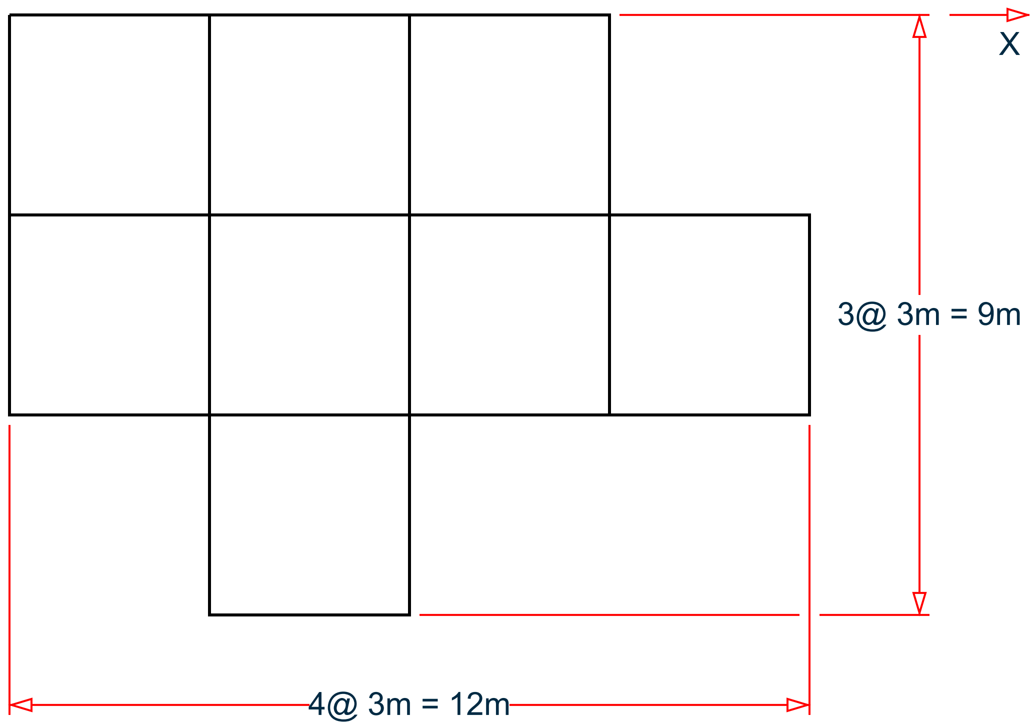


Figure 428: Plan View

Assumptions

i. Mapped MCER spectral response acceleration parameter at short period, $S_s = 2.02$
ii. Mapped MCER spectral response acceleration parameter at a period of 1 s, $S_1 = 0.795$
iii. Risk Category – I (Hence, From Table 1.5-2, Importance Factor I = 1)
iv. Site Class – D (SCLASS 4)
v. Response Modification Factor (RX & RZ) = 3
vi. Long-period Transition Time (TL)=12 s
vii. Seismic weight is composed of UDLs (magnitude -5 kN/m, direction GY) (defined in Reference Load Definition), incident on the beams
viii. The effect of shear deformation is neglected

Validation

Calculation of Base Shear

Based on S_s and S_1, $F_a = 1$ (per table 11.4-1) and $F_v = 1.7$ (per table 11.4-2).

Hence:

$$S_{MS} = F_a \times S_s = 1 \times 2.02 = 2.02 \qquad \text{Eq. 11.4-1}$$

$$S_{DS} = 2/3 \times S_{MS} = 1.347 \qquad \text{Eq. 11.4-3}$$

$$S_{M1} = F_v \times S_1 = 1.7 \times 0.795 = 1.382 \qquad \text{Eq. 11.4-3}$$

$$S_{D1} = 2/3 \times S_{M1} = 0.901 \qquad \text{Eq. 11.4-4}$$

$$(C_s)_{\text{initial}} = \frac{S_{DS}}{R / I} = 0.4489$$ Eq. 12.8-2

The time period of the structure is taken as the Rayleigh frequency of the structure (taken from the output file), $T_R = 0.201\ s$.

Height of the structure, $h = 9\ m$.

For concrete moment-resisting frames, $C_t = 0.0466$ and $x = 0.9$ (per Table 12.8-2).

So the approximate time period, $T_a = C_t \times h^x = 0.0466(9\ m)^{0.9} = 0.3367\ s$

From Table 12.8-1, $C_u = 1.4$ (for $S_{D1} > 0.4$).

$$C_u \times T_a = 0.4713\ \text{s}$$

The time period used, T, is the lesser of T_R and $C_u \times T_a$, which is $0.201\ s$ ($< T_L = 12\ s$).

$$(C_s)_{\max} = \frac{S_{D1}}{T \times (R / I)} = 1.4942$$ Eq. 12.8-3

$$(C_S)_{\min1} = 0.044 \times S_{DS} \times I = 0.0592$$ Eq. 12.8-5

Since $S_1 = 0.795 > 0.6g$, equation 12.8-6 also needs to be considered for calculating the lower limit of C_s :

$$(C_s)_{\min2} = \frac{0.5 \times S_1}{R / I} = 0.1325$$ Eq. 12.8-6

Therefore, $(C_s)_{min} = 0.1325 < C_s$.

$$C_s = 0.4489$$

The seismic weight, W is taken as the total seismic weight of the beams: $= 5 \times 3 \times 69 = 1{,}035\ kN$

So the total base shear, $V = C_S \times W = 0.4489 \times 1{,}035 = 464.6\ kN$

Vertical Distribution of Lateral Forces

Since, time period of the structure $T = 0.201\ s < 0.5\ s$, as per clause 12.8.3, $k = 1$.

Hence, from equation 12.8-11 and equation 12.8-12, we can find the lateral forces in different story levels, as follows:

Table 468: Vertical force distribution

Story Level	W_x **(kN)**	h_x **(m)**	$W_x \times h_x^{\ k}$	$\frac{W_x \times h_x^k}{\sum_{i=1}^{3}\left(W_i \times h_i^k\right)}$	**Lateral force at story level (kN)**
Roof	5 × 3 × 23 = 345	9	3,105	0.500	232.3
2nd	345	6	2,070	0.333	154.9
1st	345	3	1,035	0.167	77.40
Σ	1,035	-	6,210	1	464.6

Consideration of Inherent Torsion (Clause 12.8.4.1)

Floor Level – Roof (9 m)

From output file, CRZ = 3. 831 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = 0.017 m

Floor Level – 2nd (6 m)

From output file , CRZ = 3.858 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = -0.01 m

Floor Level – 1st (3 m)

From output file , CRZ = 3.902 m and CMZ = 3.848 m

Hence, static eccentricity esi = CMZ - CRZ = -0.054 m

Consideration of Accidental Torsion (Clause 12.8.4.2)

At all floor levels:

$$0.05 \times L_z = 0.05 \times 9\text{m} = 0.45 \text{ m}$$

where

L_z = the dimension of the structure along the global Z axis

Hence, total eccentricity to inherent and accidental torsion at roof level $e_r = (0.45 + 0.017) = 0.467\ m$

Total eccentricity to inherent and accidental torsion at 2nd floor level $e_2 = (0.45 - 0.01) = 0.44\ m$

Total eccentricity to inherent and accidental torsion at 1st floor level $e_1 = (0.45 - 0.054) = 0.396\ m$

Total torsional moment at roof level = $F_{roof} \times e_r = 232.3 \times 0.467 = 108.48\ kN \cdot m$

Total Torsional moment at 2nd floor level = $F_{2nd} \times e_2$ = 154.867 x 0.44 = 68.141 kN·m

Total torsional moment at 1st floor level = $F_{1st} \times e_1$ = 77.433 x 0.396 = 30.664 kN·m

Results

Table 469: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Base shear, V (kN)	464.6	464.6	none	
Lateral force at roof level (kN)	232.3	232.3	negligible	
Lateral force at 2nd floor (kN)	154.9	154.867	negligible	
Lateral force at 1st floor (kN)	77.4	77.433	negligible	
Torsional moment at roof level (kN·m)	108.48	108.523	negligible	
Torsional moment at 2nd floor (kN·m)	68.141	68.092	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Torsional moment at 1st floor (kN·m)	30.664	30.619	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Static Seismic T Less Than 0.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Mar-19
END JOB INFORMATION
*****************************************************************************
*This problem is created to verify the base shear, distribution of base shear
*And Inherent and Accidental Torsional Moment at different floor levels
*Of the Structure
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 0 3; 6 0 3 3; 7 3 3 3; 8 3 0 3;
9 0 0 6; 10 0 3 6; 11 3 3 6; 12 3 0 6; 13 0 0 9; 14 0 3 9; 15 3 3 9; 16 3 0 9;
17 6 3 0; 18 6 0 0; 19 6 3 3; 20 6 0 3; 21 6 3 6; 22 6 0 6; 25 9 3 3; 26 9 0
3;
27 9 3 6; 28 9 0 6; 33 0 6 0; 34 3 6 0; 35 0 6 3; 36 3 6 3; 37 0 6 6; 38 3 6
6;
39 0 6 9; 40 3 6 9; 41 6 6 0; 42 6 6 3; 43 6 6 6; 45 9 6 3; 46 9 6 6; 49 0 9
0;
50 3 9 0; 51 0 9 3; 52 3 9 3; 53 0 9 6; 54 3 9 6; 55 0 9 9; 56 3 9 9; 57 6 9
0;
58 6 9 3; 59 6 9 6; 70 9 9 3; 71 9 9 6; 72 -3 0 0; 73 -3 3 0; 74 -3 0 3;
75 -3 3 3; 76 -3 0 6; 77 -3 3 6; 80 -3 6 0; 81 -3 6 3; 82 -3 6 6; 84 -3 9 0;
85 -3 9 3; 86 -3 9 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 10 14; 15 11 15; 16 13 14; 17 14 15; 18 15 16;
19 3 17; 20 7 19; 21 11 21; 23 17 18; 24 17 19; 25 19 20; 26 19 21; 27 21 22;
30 19 25; 31 21 27; 32 25 26; 33 25 27; 34 27 28; 40 2 33; 41 3 34; 42 6 35;
43 7 36; 44 10 37; 45 11 38; 46 14 39; 47 15 40; 48 17 41; 49 19 42; 50 21 43;
52 25 45; 53 27 46; 56 33 34; 57 33 35; 58 34 36; 59 35 36; 60 35 37; 61 36
38;
62 37 38; 63 37 39; 64 38 40; 65 39 40; 66 34 41; 67 36 42; 68 38 43; 70 41
42;
71 42 43; 73 42 45; 74 43 46; 75 45 46; 79 33 49; 80 34 50; 81 35 51; 82 36
52;
83 37 53; 84 38 54; 85 39 55; 86 40 56; 87 41 57; 88 42 58; 89 43 59; 95 49
50;
96 49 51; 97 50 52; 98 51 52; 99 51 53; 100 52 54; 101 53 54; 102 53 55;
103 54 56; 104 55 56; 105 50 57; 106 52 58; 107 54 59; 109 57 58; 110 58 59;
128 45 70; 129 46 71; 132 58 70; 133 59 71; 134 70 71; 135 2 73; 136 6 75;
137 10 77; 139 33 80; 140 35 81; 141 37 82; 143 49 84; 144 51 85; 145 53 86;
147 72 73; 148 73 75; 149 74 75; 150 75 77; 151 76 77; 154 73 80; 155 75 81;
156 77 82; 158 80 81; 159 81 82; 161 80 84; 162 81 85; 163 82 86; 165 84 85;
```

```
166 85 86;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 21 23 TO 27 30 TO 34 40 TO 50 52 53 56 TO 68 70 71 73 TO 75 79 TO 89 -
95 TO 107 109 110 128 129 132 TO 137 139 TO 141 143 TO 145 147 TO 151 154 -
155 TO 156 158 159 161 TO 163 165 166 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 9 12 13 16 18 20 22 26 28 72 74 76 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
MEMBER LOAD
2 4 5 7 9 10 12 14 15 17 19 TO 21 24 26 30 31 33 56 TO 68 70 71 73 TO 75 95 -
96 TO 107 109 110 132 TO 137 139 TO 141 143 TO 145 148 150 158 159 165 -
166 UNI GY -5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
DEFINE IBC 2018
SS 2.02 S1 0.795 I 1 RX 3 RZ 3 SCLASS 4 TL 12
LOAD 1 LOADTYPE Seismic  TITLE SL +X
IBC LOAD X 1 DEC 2 ACC 0.05
PERFORM ANALYSIS PRINT LOAD DATA
PRINT DIA CR
FINISH
```

STAAD.Pro Output

```
*****************************************************************************
   * EQUIV. SEISMIC LOADS AS PER IBC
2018                                      *
   * PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD
GENERATION                        *
   *  SS =  2.020 S1 =  0.795 FA =  1.000 FV =
1.700                            *
   *  SDS =  1.347 SD1 =
0.901                                                *

*****************************************************************************
    72. LOAD 1 LOADTYPE SEISMIC  TITLE SL +X
    73. IBC LOAD X 1 DEC 2 ACC 0.05
    74. PERFORM ANALYSIS PRINT LOAD DATA
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
```

```
     NUMBER OF JOINTS         67  NUMBER OF MEMBERS     117
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     162
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
      STAAD SPACE                                              -- PAGE NO.
4
   LOADING     1  LOADTYPE SEISMIC  TITLE SL +X
   -----------
   ************************************************************
   *  IBC 2018 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.047 Cu =  1.400 x =  0.9000                  *
   *  TIME PERIODS :                                          *
   *     Ta =  0.337 T =  0.201 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.201                           *
   *  Cs LIMITS : LOWER =  0.133 UPPER =  1.492               *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.449 X      1035.00   *
   *                       =       464.60  KN                 *
   ************************************************************
   ************************************************************************
   ***NOTE: SEISMIC LOAD IS ACTING AT CENTER OF MASS FOR RIGID DIAPHRAGM.
            TORSION FROM STATIC ECCENTRICITY (esi) IS INCLUDED IN ANALYSIS.
            DYNAMIC ECCENTRICITY APPLIED = DEC - 1
   LOAD NO.:     1  DIRECTION : X   UNIT - METE
   STORY  LEVEL     DYN. ECC. (dec)     ACC. ECC. (aec)     DESIGN     ECC.
   -----  -----     ---------------     ---------------     ---------------
                      X         Z         X         Z         X         Z
                                                          dec + aec dec + aec
     1     3.00     -0.05     -0.05      0.60      0.45      0.00      0.40
     2     6.00      0.01     -0.01      0.60      0.45      0.00      0.44
     3     9.00      0.05      0.02      0.60      0.45      0.00      0.47
   ************************************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
                                      DEC + AEC
      STAAD SPACE                                              -- PAGE NO.
5
       2     FX        5.050    MY        1.997
       3     FX        5.050    MY        1.997
       6     FX        6.733    MY        2.663
       7     FX        6.733    MY        2.663
      10     FX        6.733    MY        2.663
      11     FX        6.733    MY        2.663
      14     FX        3.367    MY        1.331
      15     FX        3.367    MY        1.331
      17     FX        3.367    MY        1.331
      19     FX        6.733    MY        2.663
      21     FX        5.050    MY        1.997
      25     FX        3.367    MY        1.331
      27     FX        3.367    MY        1.331
      73     FX        3.367    MY        1.331
      75     FX        5.050    MY        1.997
      77     FX        3.367    MY        1.331
                   -----------      -----------
```

```
         TOTAL =     77.433             30.619 AT LEVEL     3.000 METE
     33      FX      10.100    MY        4.441
     34      FX      10.100    MY        4.441
     35      FX      13.467    MY        5.921
     36      FX      13.467    MY        5.921
     37      FX      13.467    MY        5.921
     38      FX      13.467    MY        5.921
     39      FX       6.733    MY        2.961
     40      FX       6.733    MY        2.961
     41      FX       6.733    MY        2.961
     42      FX      13.467    MY        5.921
     43      FX      10.100    MY        4.441
     45      FX       6.733    MY        2.961
     46      FX       6.733    MY        2.961
     80      FX       6.733    MY        2.961
     81      FX      10.100    MY        4.441
     82      FX       6.733    MY        2.961
                -----------       -----------
         TOTAL =    154.867             68.092 AT LEVEL     6.000 METE
     49      FX      15.150    MY        7.078
     50      FX      15.150    MY        7.078
     51      FX      20.200    MY        9.437
     52      FX      20.200    MY        9.437
     53      FX      20.200    MY        9.437
     54      FX      20.200    MY        9.437
     55      FX      10.100    MY        4.718
     56      FX      10.100    MY        4.718
     57      FX      10.100    MY        4.718
     58      FX      20.200    MY        9.437
     59      FX      15.150    MY        7.078
     70      FX      10.100    MY        4.718
     71      FX      10.100    MY        4.718
     84      FX      10.100    MY        4.718
     85      FX      15.150    MY        7.078
     86      FX      10.100    MY        4.718
     STAAD SPACE                                                -- PAGE NO.
6
                -----------       -----------
         TOTAL =    232.300            108.523 AT LEVEL     9.000 METE
   ************ END OF DATA FROM INTERNAL STORAGE ************
    75. PRINT DIA CR
  DIA      CR
   ************************************************************
   CENTRE OF RIGIDITY     UNIT - METE
   ------------------     -----------
   DIAPHRAM      FL. LEVEL      X-COORDINATE      Z-COORDINATE
       1           3.000            2.393             3.902
       2           6.000            2.335             3.858
       3           9.000            2.299             3.831
   ************************************************************
```

Related Links

- *TR.31.2.17 IBC 2018 Seismic Load Definition* (on page 2416)

V. IBC 2018 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in IBC 2018.

Details

A three story building is modelled in STAAD.Pro considering the following:

Location: Corona, CA (`ZIP 92887`)
Short period acceleration, $S_s = 1.996$
One second acceleration: $S_1 = 0.7$
Site class A
Response modification factor, $R = 3$
Importance factor, $I = 1.0$
$F_a = 0.8$
$F_v = 0.8$
$TL = 8$

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 470: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 471: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 472: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.5175
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight Mk × g		227.66			16.35			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $\prod \sum Wi \times \phi / \sum Wi \times \phi^2$		1.244			0.333			0.0893		

Horizontal Acceleration Spectrum Value Calculation.

$$S_{ms} = S_s \times F_a = 1.996 \times 0.8 = 1.597$$

$$S_{m1} = S_1 \times F_v = 0.7 \times 0.8 = 0.56$$

$$S_{ds} = 2/3 \times S_{ms} = 2/3 \times 1.597 = 1.065$$

$S_{d1} = 2/3 \times S_{m1} = 2/3 \times 0.56 \times 0.373$

$T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (0.373 / 1.065) = 0.070$

$T_s = S_{d1}/S_{ds} = 0.373 / 1.065 = 0.350$

	Mode 1	Mode 2	Mode 3
Time Period T	0.30014 ($T_0 \leq T \leq T_s$)	0.10986 ($T_0 \leq T \leq T_s$)	0.08042 ($T_0 \leq T \leq T_s$)
Formula	$S_a = S_{ds}$	$S_a = S_{ds}$	$S_a = S_{ds}$
S_a	1.065	1.065	1.065
$C_s = S_a/(R_x/I)$	0.355	0.355	0.355

Table 473: Base Shear Calculation and its Distribution Across Height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	
Roof	49.035	1	21.646	21.646	-1	-5.80	-5.80	1	1.554	1.554	22.463
2nd Floor	98.07	0.866603	37.491	59.137	0	0	-5.80	-0.866603	-2.692	-1.138	59.432
1st Floor	98.07	0.5	21.646	80.783	1	11.60	5.80	0.5	1.554	0.416	80.992
Base Shear, V											80.992

Comparison

Table 474: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Base shear (kN)	80.91	80.992	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2018 Response Spectrum.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
```

```
SPECTRUM SRSS IBC 2018 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS A FA 0.800 FV 0.800 TL 8.000
*SS 1.996 S1 0.7 SITE CLASS A FA 0.8 FV 0.8 TL 8.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                      3.332                   0.30014
         2                      9.103                   0.10986
         3                     12.435                   0.08042
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z               WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
           MODE                    ACCELERATION-G     DAMPING
           ----                    --------------     -------
              1                         1.06453       0.05000
              2                         1.06453       0.05000
              3                         1.06453       0.05000
      STAAD SPACE                                                 -- PAGE NO.
15
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.300       80.70        0.00        0.00        0.00
0.00     -484.21
      2      0.110        5.79        0.00        0.00        0.00
0.00       17.38
      3      0.080        0.42        0.00        0.00        0.00
0.00       -2.50
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KN
           --------------------------------------         ------------------
 MODE    X      Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000     80.70     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      5.79     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.42     0.00
0.00
```

```
---------------------------
                                  TOTAL SRSS  SHEAR     80.91     0.00
0.00
                                  TOTAL 10PCT SHEAR     80.91     0.00
0.00
                                  TOTAL ABS   SHEAR     86.91     0.00
0.00
```

V. IBC 2015 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in IBC 2015.

Details

A three story buiding is modelled in STAAD.Pro considering the following:

Location: Corona, CA (`ZIP 92887`)
Short period acceleration, $S_s = 2.24658$
One second acceleration: $S_1 = 0.817039$
Site class E
Response modification factor, $R = 3$
Importance factor, $I = 1.0$
$F_a = 0.9$
$F_v = 2.4$
$TL = 8$

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 475: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 476: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 477: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.5175
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Modal Weight Mk × g		227.66			16.35			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $\prod \sum Wi \times \phi / \sum Wi \times \phi^2$		1.244			0.333			0.0893		

Horizontal Acceleration Spectrum Value Calculation.

$S_{ms} = S_s \times F_a = 2.24658 \times 0.9 = 2.022$

$S_{m1} = S_1 \times F_v = 0.817039 \times 2.4 = 1.961$

$S_{ds} = 2/3 \times S_{ms} = 2/3 \times 2.022 = 1.348$

$S_{d1} = 2/3 \times S_{m1} = 2/3 \times 1.961 = 1.307$

$T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (1.307 / 1.348) = 0.1940$

$T_s = S_{d1}/S_{ds} = 1.307 / 1.348 = 0.9698$

	Mode 1	Mode 2	Mode 3
Time Period T	0.30014 ($T_0 \leq T \leq T_s$)	0.10986 ($T < T_0$)	0.08042 ($T < T_0$)
Formula	$S_a = S_{ds}$	$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$	$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$
S_a	1.348	0.997	0.875
$C_s = S_a/(R_x/I)$	0.449	0.332	0.292

Table 478: Base Shear Calculation and its Distribution Across Height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	
Roof	49.035	1	27.408	27.408	-1	-5.433	-5.433	1	1.277	1.277	27.971

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	
2nd Floor	98.07	0.866603	47.473	74.881	0	0	-5.433	-0.866603	-2.211	-0.9346	75.084
1st Floor	98.07	0.5	27.408	102.29	1	10.867	5.433	0.5	1.277	0.3421	102.43
Base Shear, V											102.43

Comparison

Table 479: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Base shear (kN)	102.43	102.33	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2015 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
```

```
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IBC 2015 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                      3.332                  0.30014
         2                      9.103                  0.10986
         3                     12.435                  0.08042
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS           MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
           MODE                 ACCELERATION-G     DAMPING
```

```
           ----                 --------------     -------
             1                      1.34795       0.05000
             2                      0.99725       0.05000
             3                      0.87451       0.05000
      STAAD SPACE                                                -- PAGE NO.
15
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE      PERIOD        FX          FY          FZ          MX
MY          MZ
      1       0.300      102.19        0.00        0.00        0.00
0.00     -613.12
      2       0.110        5.43        0.00        0.00        0.00
0.00       16.28
      3       0.080        0.34        0.00        0.00        0.00
0.00       -2.05
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE     X      Y      Z      SUMM-X   SUMM-Y   SUMM-Z       X         Y         Z
   1    92.85  0.00  0.00    92.855    0.000    0.000    102.19      0.00
0.00
   2     6.67  0.00  0.00    99.521    0.000    0.000      5.43      0.00
0.00
   3     0.48  0.00  0.00   100.000    0.000    0.000      0.34      0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR    102.33      0.00
0.00
                                  TOTAL 10PCT SHEAR    102.33      0.00
0.00
                                  TOTAL ABS   SHEAR    107.96      0.00
0.00
```

Related Links

- *TR.32.10.1.12 Response Spectrum Specification per IBC 2015* (on page 2573)

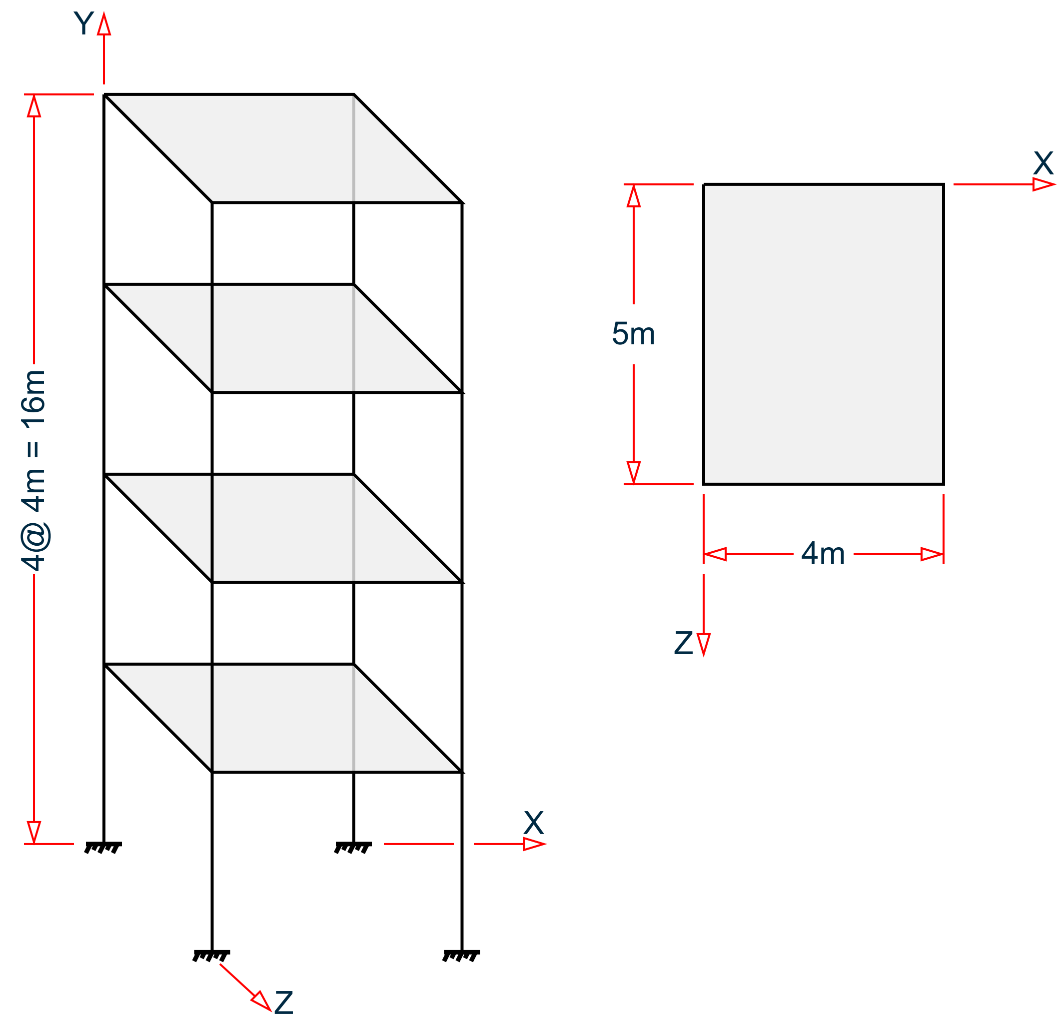
V. IBC 2015 Static Seismic

Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2015.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m



Input parameters:

Location: Corona, CA, USA (`ZIP 92877`)
Site class D (`SCLASS 4`)
Long-period transition time = 12s (`TL 12`)
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)
Response modification factor Z = 4 (`RZ 4`)

The seismic weight for each floor is assumed as 200 KN applied as joint weight of 50KN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2015 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

ZIP = 92887; Ss = 2.247 & S1 = 0.817

For Site Class D, Fa = 1 (Ss > 1.25) and Fv = 1.5 (S1 > 0.5)

SMS = Ss × Fa = 2.247, SM1 = S1 × Fv = 1.2255

SDS = 2/3 × SMS = 1.498, SD1 = 2/3 × SM1 = 0.817

For concrete moment resisting frames Ct = 0.0466 in metric system & x = 0.9

Height = ha = 16m, Ta = Ct × ha^x = 0.0466 × $16^{0.9}$ = 0.565,

Since SD1 = 0.817 > 0.4, Cu = 1.4

Cu × Ta = 1.4 × 0.565 = 0.791

T (from output file) = 1.286, Cu × Ta = 0.791 < 1.286, T_{used} = 0.791

Cs_x = SDS/(Rx/I) = 1.498 / (3/1) = 0.49933

Cs_x lower limit = 0.044 × SDS × I & 0.5 × S1 / (Rx/I),

Cs_x lower limit = 0.044 × 1.498 × 1 & 0.5 × 0.817 / (3/1)

Cs_x lower limit = 0.065912 & 0.136166

Cs_x upper limit = SD1 / (T_{used} × (Rx/I)) as T_{used} < TL

Cs_x upper limit = 0.817 / (0.791 × (3/1)) = 0.34429

Cs_x used = 0.34429

Seismic Base Shear V = Cs × W = 0.34429 × (200 × 4) = 275.432 KN

Lateral seismic force Fx = C_{vx} × V where C_{vx} = W_x × hx^k / ∑ W_i × hi^k

Since T_{used} = 0.791 sec > 0.5 Sec distribution of Base Shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. power = 1.1455.

Story Level	W_i	h_i	$Wi \times hi^{1.1455}$	$(Wi \times hi^{1.1455})/\sum(Wi \times hi^{1.1455})$	F_x
Roof	200	16	4,790.2	0.4209	115.941
3rd	200	12	3,445.3	0.3028	83.391
2nd	200	8	2,165.3	0.1903	52.409
1st	200	4	978.79	0.0860	23.691
Summation	800		11,379.6	1	
V_x	275.432				

Comparison

Table 480: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	23.691	23.697	Negligible	
	2nd	52.409	52.389	Negligible	
	3rd	83.391	83.364	Negligible	
	roof	115.941	115.908	Negligible	
Base shear, V_x (kN)		275.432	275.34	Negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2015 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE IBC 2015
```

```
*DEFINE IBC 2012
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 4 TL 12
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
IBC LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   ************************************************************
   *  IBC 2015 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.047 Cu =  1.400 x =  0.9000                  *
   *  TIME PERIODS :                                          *
   *     Ta =  0.565 T =  1.286 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.791                           *
   *  Cs LIMITS : LOWER =  0.136 UPPER =  0.344               *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.344 X       800.00   *
   *                       =       275.34  KN                 *
   ************************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN   )     MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX        5.920    MY        0.000
       3     FX        5.920    MY        0.000
       8     FX        5.920    MY        0.000
       9     FX        5.920    MY        0.000
                 -----------        -----------
          TOTAL =     23.679              0.000 AT LEVEL     4.000 METE
       5     FX       13.097    MY        0.000
       6     FX       13.097    MY        0.000
      11     FX       13.097    MY        0.000
      12     FX       13.097    MY        0.000
                 -----------        -----------
          TOTAL =     52.389              0.000 AT LEVEL     8.000 METE
      13     FX       20.841    MY        0.000
      14     FX       20.841    MY        0.000
      15     FX       20.841    MY        0.000
      16     FX       20.841    MY        0.000
                 -----------        -----------
          TOTAL =     83.364              0.000 AT LEVEL    12.000 METE
      17     FX       28.977    MY        0.000
      18     FX       28.977    MY        0.000
      19     FX       28.977    MY        0.000
      20     FX       28.977    MY        0.000
                 -----------        -----------
          TOTAL =    115.908              0.000 AT LEVEL    16.000 METE
```

Related Links

- *TR.31.2.16 IBC 2015 Seismic Load Definition* (on page 2413)

V. IBC 2012 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in IBC 2012.

Details

A three story buiding is modelled in STAAD.Pro considering the following:

Location: Corona, CA (`ZIP 92887`)
Short period acceleration, $S_s = 2.24658$
One second acceleration: $S_1 = 0.817039$
Site class E
Response modification factor, $R = 3$
Importance factor, $I = 1.0$
$F_a = 0.9$
$F_v = 2.4$
$TL = 8$

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 481: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 482: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 483: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.5175
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight Mk × g		227.66			16.35			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $\prod \sum Wi \times \phi \;/\; \sum Wi \times \phi^2$		1.244			0.333			0.0893		

Horizontal Acceleration Spectrum Value Calculation.

$S_{ms} = S_s \times F_a = 2.24658 \times 0.9 = 2.022$

$S_{m1} = S_1 \times F_v = 0.817039 \times 2.4 = 1.961$

$S_{ds} = 2/3 \times S_{ms} = 2/3 \times 2.022 = 1.348$

$S_{d1} = 2/3 \times S_{m1} = 2/3 \times 1.961 = 1.307$

$T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (1.307 / 1.348) = 0.1940$

$T_s = S_{d1}/S_{ds} = 1.307 / 1.348 = 0.9698$

	Mode 1	**Mode 2**	**Mode 3**
Time Period T	0.30014 ($T_0 \leq T \leq T_s$)	0.10986 ($T < T_0$)	0.08042 ($T < T_0$)
Formula	$S_a = S_{ds}$	$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$	$S_a = S_{ds} \times (0.4 + 0.6 \times (T/T_0))$
S_a	1.348	0.997	0.875
$C_s = S_a/(R_x/I)$	0.449	0.332	0.292

Table 484: Base Shear Calculation and its Distribution Across Height

Story Level	**Weight Wi (KN)**	**Mode 1**			**Mode 2**			**Mode 3**			**Story Shear due to all modes i.e. SRSS of all modes**
		ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	
Roof	49.035	1	27.408	27.408	-1	-5.433	-5.433	1	1.277	1.277	27.971
2nd Floor	98.07	0.866603	47.473	74.881	0	0	-5.433	-0.866603	-2.211	-0.9346	75.084
1st Floor	98.07	0.5	27.408	102.29	1	10.867	5.433	0.5	1.277	0.3421	102.43
Base Shear, V											102.43

Comparison

Table 485: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Base shear (kN)	102.43	102.33	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2012 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
```

```
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IBC 2012 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS E FA 0.900 FV 2.400 TL 8.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       3.332                  0.30014
         2                       9.103                  0.10986
         3                      12.435                  0.08042
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN          GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
           MODE                    ACCELERATION-G     DAMPING
           ----                    --------------     -------
             1                          1.34795       0.05000
             2                          0.99725       0.05000
             3                          0.87451       0.05000
      STAAD SPACE                                               -- PAGE NO.
15
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.300      102.19        0.00        0.00        0.00
0.00     -613.12
      2      0.110        5.43        0.00        0.00        0.00
0.00       16.28
      3      0.080        0.34        0.00        0.00        0.00
0.00       -2.05
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KN
```

```
            --------------------------------------        ------------------
 MODE     X      Y      Z      SUMM-X    SUMM-Y    SUMM-Z       X         Y         Z
   1    92.85  0.00  0.00    92.855     0.000     0.000    102.19      0.00
0.00
   2     6.67  0.00  0.00    99.521     0.000     0.000      5.43      0.00
0.00
   3     0.48  0.00  0.00   100.000     0.000     0.000      0.34      0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR    102.33      0.00
0.00
                                  TOTAL 10PCT SHEAR    102.33      0.00
0.00
                                  TOTAL ABS   SHEAR    107.96      0.00
0.00
```

Related Links

- *TR.32.10.1.11 Response Spectrum Specification per IBC 2012* (on page 2566)

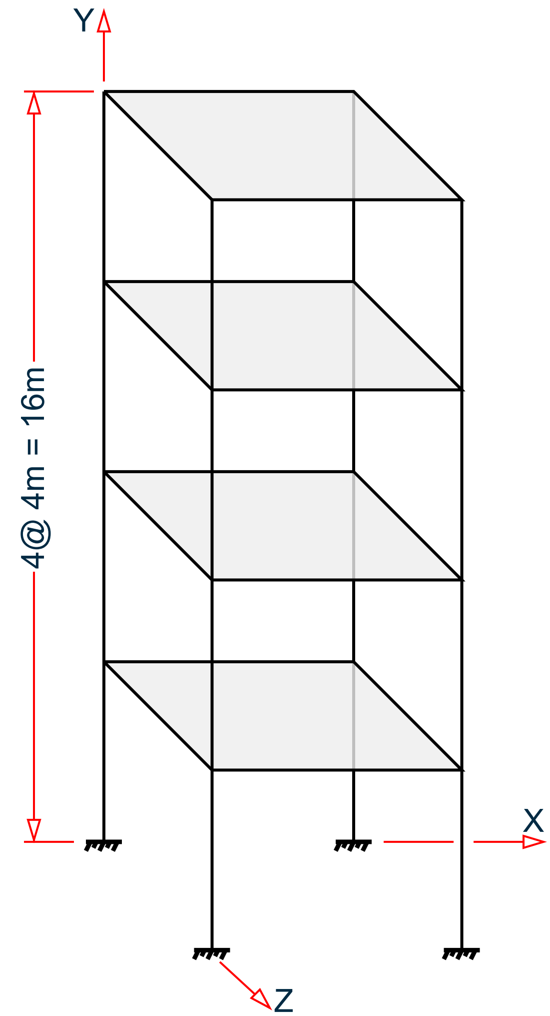
V. IBC 2012 Static Seismic

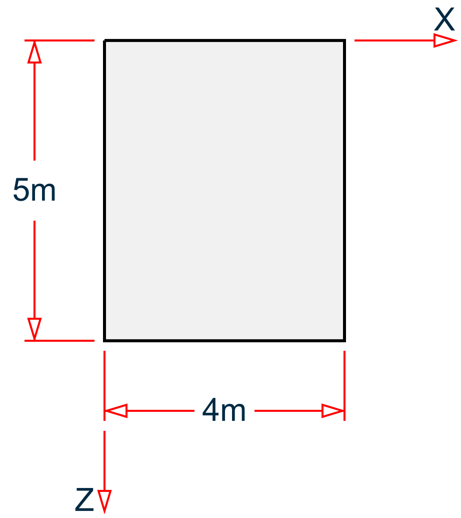
Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2012.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Location: Corona, CA, USA (`ZIP 92877`)
Site class D (`SCLASS 4`)
Long-period transition time = 12s (`TL 12`)
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)
Response modification factor Z = 4 (`RZ 4`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2012 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

ZIP = 92887; $S_s = 2.247$ & $S_1 = 0.817$

For Site Class D, $F_a = 1$ ($S_s > 1.25$) and $F_v = 1.5$ ($S_1 > 0.5$)

SMS = $S_s \times F_a$ = 2.247, SM1 = $S_1 \times F_v$ = 1.2255

SDS = 2/3 × SMS = 1.498, SD1 = 2/3 × SM1 = 0.817

For concrete moment resisting frames Ct = 0.0466 in metric system & x = 0.9

Height = ha = 16m, $Ta = Ct \times ha^x = 0.0466 \times 16^{0.9} = 0.565$,

Since SD1 = 0.817 > 0.4, Cu = 1.4

Cu × Ta = 1.4 × 0.565 = 0.791

T (from output file) = 1.286, Cu × Ta = 0.791 < 1.286, T_{used} = 0.791

Cs_x = SDS/(Rx/I) = 1.498 / (3/1) = 0.49933

Cs_x lower limit = 0.044 × SDS × I & 0.5 × S1 / (Rx/I),

Cs_x lower limit = 0.044 × 1.498 × 1 & 0.5 × 0.817 / (3/1)

Cs_x lower limit = 0.065912 & 0.136166

Cs_x upper limit = SD1 / (T_{used} × (Rx/I)) as T_{used} < TL

Cs_x upper limit = 0.817 / (0.791 × (3/1)) = 0.34429

Cs_x used = 0.34429

Seismic Base Shear V = Cs × W = 0.34429 × (200 × 4) = 275.432 kN

Lateral seismic force Fx = C_{vx} × V where $C_{vx} = W_x \times hx^k / \sum W_i \times hi^k$

Since T_{used} = 0.791 sec > 0.5 Sec distribution of Base Shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. power = 1.1455.

Story Level	W_i	h_i	$Wi \times hi^{1.1455}$	$(Wi \times hi^{1.1455})/\sum(Wi \times hi^{1.1455})$	F_x
Roof	200	16	4,790.2	0.4209	115.941
3rd	200	12	3,445.3	0.3028	83.391
2nd	200	8	2,165.3	0.1903	52.409
1st	200	4	978.79	0.0860	23.691
Summation	800		11,379.6	1	
V_x	275.432				

Comparison

Table 486: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	23.691	23.697	Negligible	
	2nd	52.409	52.389	Negligible	
	3rd	83.391	83.364	Negligible	
	roof	115.941	115.908	Negligible	
Base shear, V_x (kN)		275.432	275.34	Negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2012 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
*DEFINE IBC 2015
```

```
DEFINE IBC 2012
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 4 TL 12
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
IBC LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   ************************************************************
   *  IBC 2012 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.047 Cu =  1.400 x =  0.9000                  *
   *  TIME PERIODS :                                          *
   *     Ta =  0.565 T =  1.286 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.791                           *
   *  Cs LIMITS : LOWER =  0.136 UPPER =  0.344               *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.344 X       800.00   *
   *                       =       275.34  KN                 *
   ************************************************************
    JOINT            LATERAL          TORSIONAL           LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX        5.920    MY        0.000
       3     FX        5.920    MY        0.000
       8     FX        5.920    MY        0.000
       9     FX        5.920    MY        0.000
                ----------          -----------
          TOTAL =     23.679              0.000 AT LEVEL     4.000 METE
       5     FX       13.097    MY        0.000
       6     FX       13.097    MY        0.000
      11     FX       13.097    MY        0.000
      12     FX       13.097    MY        0.000
                ----------          -----------
          TOTAL =     52.389              0.000 AT LEVEL     8.000 METE
      13     FX       20.841    MY        0.000
      14     FX       20.841    MY        0.000
      15     FX       20.841    MY        0.000
      16     FX       20.841    MY        0.000
                ----------          -----------
          TOTAL =     83.364              0.000 AT LEVEL    12.000 METE
      17     FX       28.977    MY        0.000
      18     FX       28.977    MY        0.000
      19     FX       28.977    MY        0.000
      20     FX       28.977    MY        0.000
                ----------          -----------
          TOTAL =    115.908              0.000 AT LEVEL    16.000 METE
```

Related Links

- *TR.31.2.15 IBC 2012 Seismic Load Definition* (on page 2409)

V. IBC 2006 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in IBC 2006.

Details

A three story buiding is modelled in STAAD.Pro considering the following:

Location: Corona, CA (`ZIP 92887`)
Short period acceleration, $S_s = 1.8157$
One second acceleration: $S_1 = 0.67269$
Site class D
Response modification factor, $R = 3$
Importance factor, $I = 1.0$
$F_a = 1.0$
$F_v = 1.5$
$TL = 12$

The generated model is subjected to a Response Spectrum Load along global X direction. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 487: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	0.627	1.59501
2	1.713	0.58381
3	2.340	0.42738

Table 488: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 489: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.5175
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight Mk × g		227.66			16.35			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $\prod \sum Wi \times \phi / \sum Wi \times \phi^2$		1.244			0.333			0.0893		

Horizontal Acceleration Spectrum Value Calculation.

$$S_{ms} = S_s \times F_a = 1.8157 \times 1 = 1.8157$$

$$S_{m1} = S_1 \times F_v = 0.67269 \times 1.5 = 1.0090$$

$$S_{ds} = 2/3 \times S_{ms} = 2/3 \times 1.8157 = 1.2104$$

$$S_{d1} = 2/3 \times S_{m1} = 2/3 \times 1.0090 = 0.6727$$

$$T_0 = 0.2 \times (S_{d1}/S_{ds}) = 0.2 \times (0.6727 / 1.2104) = 0.1111$$

$$T_s = S_{d1}/S_{ds} = 0.6727 / 1.2104 = 0.5557$$

	Mode 1	Mode 2	Mode 3
Time Period T	1.5950 ($T_s \leq T \leq T_l$)	0.5838 ($T_s \leq T \leq T_l$)	0.4274 ($T_0 \leq T \leq T_s$)
Formula	$S_a = S_{d1} / T$	$S_a = S_{d1} / T$	$S_a = S_{ds}$
S_a	0.4217	1.1522	1.2104
$C_s = S_a/(R_x/I)$	0.141	0.384	0.403

Table 490: Base Shear Calculation and its Distribution Across Height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	ϕ	Qi ($Cs \times \phi \times \prod \times Wi$)	Vi ($\sum Qi$)	
Roof	49.035	1	8.575	8.575	-1	-6.278	-6.278	1	1.767	1.767	10.77
2nd Floor	98.07	0.866603	14.85	23.43	0	0	-6.278	-0.866603	-3.061	-1.294	24.29
1st Floor	98.07	0.5	8.575	32.00	1	12.56	6.278	0.5	1.767	0.473	32.62
Base Shear, V											32.62

Comparison

Table 491: Comparison of results

Parameter	Reference	STAAD.Pro	Difference	Comments
Base shear (kN)	32.62	32.59	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2006 Response Spectrum.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.00003 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
```

```
SPECTRUM SRSS IBC 2006 X 0.333 ACC DAMP 0.05 LIN
ZIP 92887 SITE CLASS D FA 1.000 FV 1.500 TL 12.000
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X         Y         Z
   1   92.85  0.00  0.00   92.855    0.000    0.000     31.97      0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      6.27      0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.47      0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR     32.59      0.00
0.00
                                  TOTAL 10PCT SHEAR     32.59      0.00
0.00
                                  TOTAL ABS   SHEAR     38.72      0.00
0.00
```

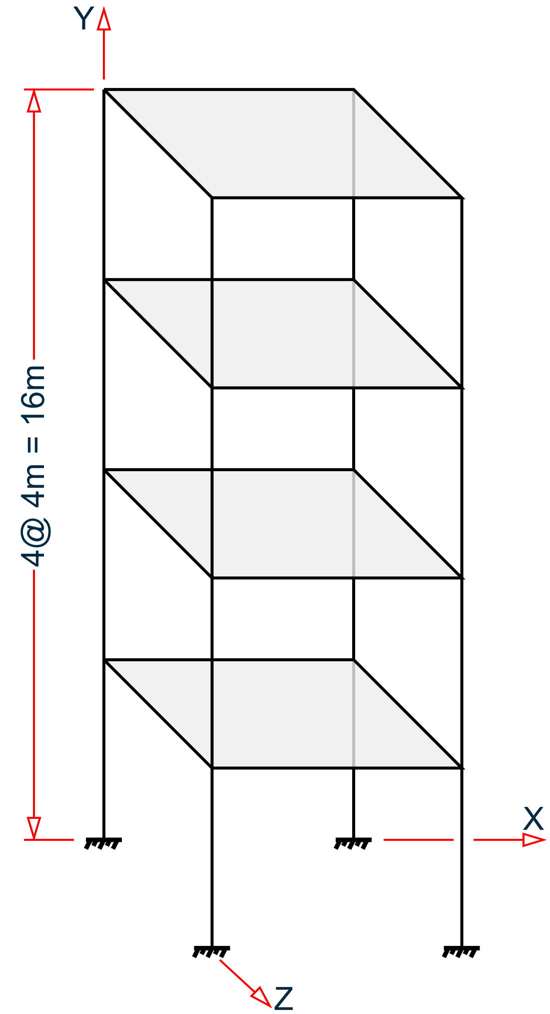
V. IBC 2006 Static Seismic

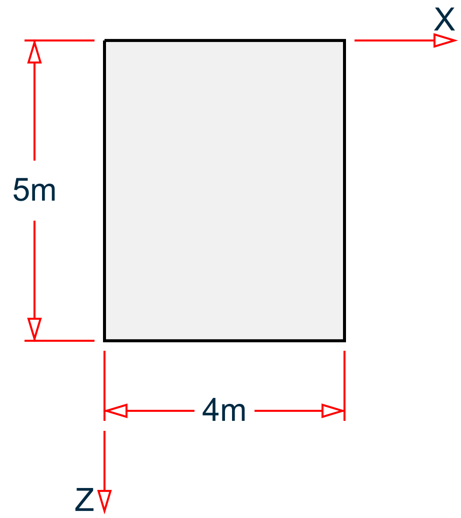
Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2006.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Location: Corona, CA, USA (`ZIP 92877`)
Site class D (`SCLASS 4`)
Long-period transition time = 12s (`TL 12`)
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)
Response modification factor Z = 4 (`RZ 4`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2006 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

ZIP = 92887; Ss = 2.247 & S1 = 0.817

For Site Class D, Fa = 1 (Ss > 1.25) and Fv = 1.5 (S1 > 0.5)

SMS = Ss × Fa = 2.247, SM1 = S1 × Fv = 1.2255

SDS = 2/3 × SMS = 1.498, SD1 = 2/3 × SM1 = 0.817

For concrete moment resisting frames Ct = 0.0466 in metric system & x = 0.9

Height = $hn = 16m = 52.49\ ft$, $Ta = C_T \times hn^x = 0.016 \times 52.49^{0.9} = 0.565$,

Since $S_{D1} = 0.673 > 0.4$, $Cu = 1.4$

$$\mathrm{Cu} \times \mathrm{Ta} = 1.4 \times 0.565 = 0.791$$

T (from output file) = 1.286, Cu × Ta = 0.791 < 1.286, $T_{used} = 0.791$

$$Cs_x = S_{DS}/(R_x/I) = 1.21067 / (3/1) = 0.404$$

$$Cs_x \text{lower limit} = \max \left| \begin{array}{l} 0.01 \\ \dfrac{0.5 \times S_1}{R_x / I} = \dfrac{0.5 \times 0.673}{3/1} = 0.112 \end{array} \right.$$

$$Cs_x \text{ upper limit} = S_{D1} / (T_{used} \times (Rx/I)) = 0.673 / (0.791 \times (3/1)) = 0.2835$$

$$Cs_x \text{ used} = 0.2835$$

Seismic Base Shear $V = Cs \times W = 0.2835 \times (200 \times 4) = 226.8\ kN$

Lateral seismic force $Fx = C_{vx} \times V$

where

$$C_{vx} = W_x \times h_x^k \ / \sum W_i \times h_i^k$$

Since $T_{used} = 0.791\ sec > 0.5\ Sec$ distribution of base shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. $k = 1.1455$.

Story Level	W_i	h_i	$W_i \times h_i^{1.1455}$	$(W_i \times h_i^{1.1455})/\sum(W_i \times h_i^{1.1455})$	F_x
Roof	200	16	4,792.1	0.4210	95.51
3rd	200	12	3,446.6	0.3028	68.69
2nd	200	8	2,166.0	0.1903	43.17
1st	200	4	979.0	0.0860	19.52
Summation	800		11,383.7	1	
V_x	226.8				

Comparison

Table 492: Comparison of results

Parameter		**Hand Calculation**	**STAAD.Pro**	**Difference**	**Comments**
Story shear, F_x (kN)	1st	19.52	19.505	negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	2nd	43.17	43.153	negligible	
	3rd	68.69	68.667	negligible	
	roof	95.51	95.474	negligible	
Base shear, V_x (kN)		226.8	226.69	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2006 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE IBC 2006
ZIP 92887 I 1 RX 3 RZ 4 SCLASS 4 TL 12
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
```

```
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
IBC LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   ************************************************************
   *  IBC 2006 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.047 Cu =  1.400 x =  0.9000                  *
   *  TIME PERIODS :                                          *
   *     Ta =  0.565 T =  1.286 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.791                           *
   *  Cs LIMITS : LOWER =  0.112 UPPER =  0.283               *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.283 X       800.00   *
   *                       =       226.69  KN                 *
   ************************************************************
    JOINT           LATERAL          TORSIONAL            LOAD -    1
                    LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----           -------          ---------
        2     FX        4.874    MY        0.000
        3     FX        4.874    MY        0.000
        8     FX        4.874    MY        0.000
        9     FX        4.874    MY        0.000
                 -----------       -----------
          TOTAL =     19.495               0.000 AT LEVEL     4.000 METE
        5     FX       10.783    MY        0.000
        6     FX       10.783    MY        0.000
       11     FX       10.783    MY        0.000
       12     FX       10.783    MY        0.000
                 -----------       -----------
          TOTAL =     43.133               0.000 AT LEVEL     8.000 METE
       13     FX       17.159    MY        0.000
       14     FX       17.159    MY        0.000
       15     FX       17.159    MY        0.000
       16     FX       17.159    MY        0.000
                 -----------       -----------
          TOTAL =     68.635               0.000 AT LEVEL    12.000 METE
       17     FX       23.857    MY        0.000
       18     FX       23.857    MY        0.000
       19     FX       23.857    MY        0.000
       20     FX       23.857    MY        0.000
                 -----------       -----------
          TOTAL =     95.430               0.000 AT LEVEL    16.000 METE
```

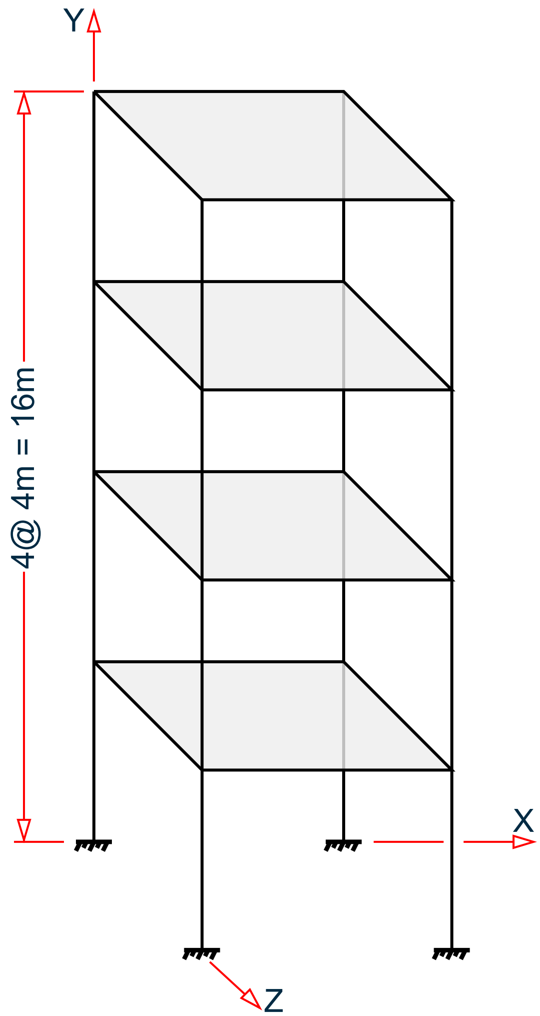
V. IBC 2003 Static Seismic

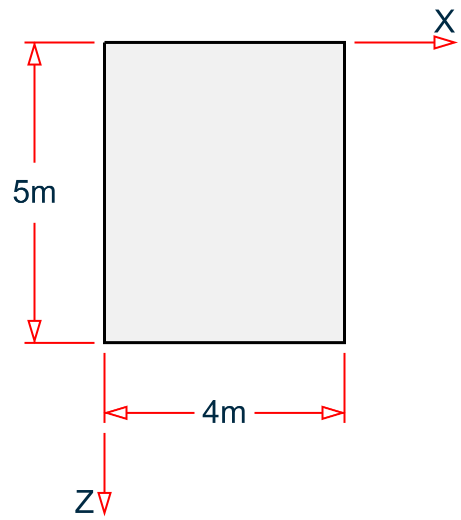
Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2003.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Short period acceleration, S_{DS} = 1.21067 (`SDS 1.21067`)
1-Second period acceleration, S_{D1} = 0.673 (`SD1 0.673`)
Site class D (`SCLASS 4`)
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2003 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

For concrete moment-resisting frames, $C_T = 0.016$ and $x = 0.9$.

Height = $hn = 16m = 52.49\ ft$, $Ta = C_T \times hn^x = 0.016 \times 52.49^{0.9} = 0.565$,

Since $S_{D1} = 0.673 > 0.4$, $Cu = 1.4$

$$Cu \times Ta = 1.4 \times 0.565 = 0.791$$

T (from output file) = 1.286, Cu × Ta = 0.791 < 1.286, $T_{used} = 0.791$

$$Cs_x = S_{DS}/(R_x/I) = 1.21067 / (3/1) = 0.404$$

$$Cs_x \text{lower limit} = \max \begin{cases} 0.044 \times S_{DS} \times I = 0.044 \times 1.21067 \times 1 = 0.0533 \\ \dfrac{0.5 \times S_1}{R_x / I} = \dfrac{0.5 \times 0.673}{3/1} = 0.112 \end{cases}$$

$$Cs_x \text{ upper limit} = S_{D1} / (T_{used} \times (Rx/I)) = 0.673 / (0.791 \times (3/1)) = 0.2835$$

$$Cs_x \text{ used} = 0.2835$$

Seismic Base Shear $V = Cs \times W = 0.2835 \times (200 \times 4) = 226.8\ kN$

Lateral seismic force $Fx = C_{vx} \times V$
where

$$C_{vx} = W_x \times h_x^k / \sum W_i \times h_i^k$$

Since $T_{used} = 0.791\ sec > 0.5\ Sec$ distribution of base shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. $k = 1.1457$.

Story Level	W_i	h_i	$W_i \times h_i^{1.1457}$	$(W_i \times h_i^{1.1457})/\sum(W_i \times h_i^{1.1457})$	F_x
Roof	200	16	4,792.1	0.4210	95.47
3rd	200	12	3,446.6	0.3028	68.67
2nd	200	8	2,166.0	0.1903	43.15
1st	200	4	979.0	0.0860	19.51
Summation	800		11,383.7	1	
V_x	226.8				

Comparison

Table 493: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	19.51	19.505	negligible	
	2nd	43.15	43.153	negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	3rd	68.67	68.667	negligible	
	roof	95.47	95.474	negligible	
Base shear, V_x (kN)		226.8	226.8	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2003 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE IBC 2003
SDS 1.21067 SD1 0.673 S1 0.673 IE 1 RX 3 RZ 4 SCLASS 4
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
IBC LOAD X 1
```

```
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   ************************************************************
   *  IBC 2003 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.016 Cu =  1.400                              *
   *  TIME PERIODS :                                          *
   *     Ta =  0.565 T =  1.286 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.791                           *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.283 X       800.00   *
   *                       =       226.80  KN                 *
   ************************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      2     FX         4.876    MY        0.000
      3     FX         4.876    MY        0.000
      8     FX         4.876    MY        0.000
      9     FX         4.876    MY        0.000
                 -----------        -----------
         TOTAL =      19.505              0.000 AT LEVEL     4.000 METE
      5     FX        10.788    MY        0.000
      6     FX        10.788    MY        0.000
     11     FX        10.788    MY        0.000
     12     FX        10.788    MY        0.000
                 -----------        -----------
         TOTAL =      43.153              0.000 AT LEVEL     8.000 METE
     13     FX        17.167    MY        0.000
     14     FX        17.167    MY        0.000
     15     FX        17.167    MY        0.000
     16     FX        17.167    MY        0.000
                 -----------        -----------
         TOTAL =      68.667              0.000 AT LEVEL    12.000 METE
     17     FX        23.869    MY        0.000
     18     FX        23.869    MY        0.000
     19     FX        23.869    MY        0.000
     20     FX        23.869    MY        0.000
                 -----------        -----------
         TOTAL =      95.474              0.000 AT LEVEL    16.000 METE
```

V. IBC 2000 Static Seismic

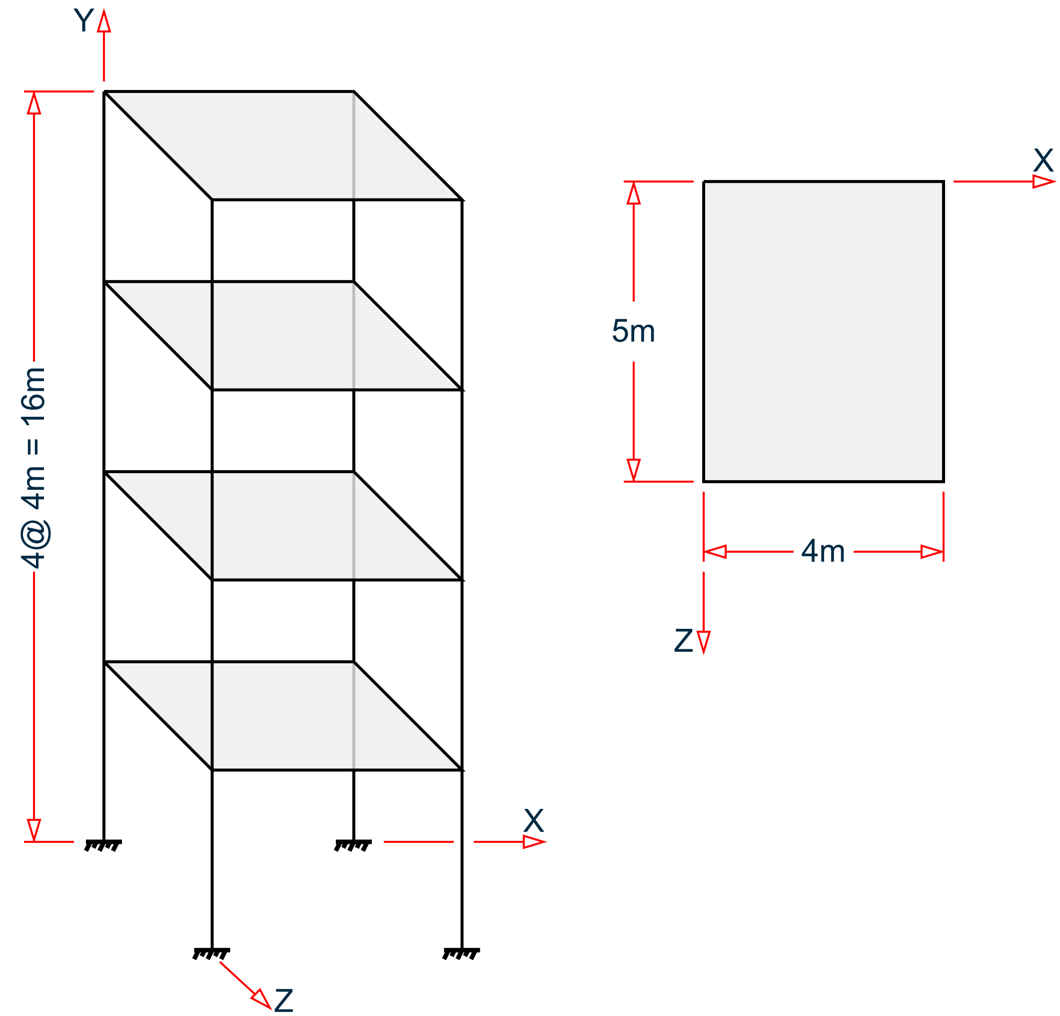
Calculation of base shear and its distribution along the height for equivalent lateral force method in IBC 2000.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m

plan dimension Z = 5m



Input parameters:

Short period acceleration, S_{DS} = 0.06 (`SDS 0.06`)
1-Second period acceleration, S_{D1} = 1.36 (`SD1 1.36`)
Site class D (`SCLASS 4`)
Building period coefficient = 0.073 (`CT 0.073`)
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per IBC 2000 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

Height = $hn = 16m$, $Ta = C_T \times hn^x = 0.073 \times 16^{0.75} = 0.584$,

Since $S_{D1} = 1.36 > 0.4$, $Cu = 1.2$

$$Cu \times Ta = 1.2 \times 0.584 = 0.701$$

T (from output file) = 1.286, Cu × Ta = 0.701 < 1.286, $T_{used} = 0.701$

$$Cs_x = S_{DS}/(R_x/I) = 0.06 / (3/1) = 0.02$$

$$Cs_x \text{lower limit} = \max \begin{vmatrix} 0.044 \times S_{DS} \times I = 0.044 \times 0.06 \times 1 = 0.00264 \\ \frac{0.5 \times S_1}{R_x / I} = \frac{0.5 \times 1.3}{3/1} = 0.2167 \end{vmatrix}$$

$$Cs_x \text{ upper limit} = S_{D1} / (T_{used} \times (Rx/I)) = 1.36 / (0.701 \times (3/1)) = 0.6469$$

$$Cs_x \text{ used} = 0.2167$$

Seismic Base Shear $V = Cs \times W = 0.2167 \times (200 \times 4) = 173.3\ kN$

Lateral seismic force $Fx = C_{vx} \times V$

where

$$C_{vx} = W_x \times h_x^k / \sum W_i \times h_i^k$$

Since $T_{used} = 0.701\ sec > 0.5\ Sec$ distribution of base shear along story levels would have the value of exponent k linearly interpolated between 1 (less than equal to 0.5 sec) and 2 (greater than equal to 2.5 sec) i.e. $k = 1.1004$.

Story Level	W_i	h_i	$W_i \times h_i^{1.1004}$	$(W_i \times h_i^{1.1004})/\sum(W_i \times h_i^{1.1004})$	F_x
Roof	200	16	4,227.1	0.4145	71.85
3rd	200	12	3,080.1	0.3020	52.35
2nd	200	8	1,971.5	0.1933	33.51
1st	200	4	919.5	0.0902	15.63
Summation	800		10,198.2	1	
V_x	173.33				

Comparison

Table 494: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	15.63	15.628	Negligible	
	2nd	33.51	33.508	Negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	3rd	52.35	52.351	Negligible	
	roof	71.85	71.846	Negligible	
Base shear, V_x (kN)		173.33	173.33	Negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IBC\IBC 2000 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE IBC 2000
SDS 0.06 SD1 1.36 S1 1.3 IE 1 RX 3 RZ 4 SCLASS 4 CT 0.073
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
IBC LOAD X 1
```

```
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   ************************************************************
   *  IBC 2000 SEISMIC LOAD ALONG X  :                        *
   *     CT =  0.073 Cu =  1.200                              *
   *  TIME PERIODS :                                          *
   *     Ta =  0.584 T =  1.286 Tuser = 0.000                 *
   *  TIME PERIOD USED (T) =  0.701                           *
   *  LOAD FACTOR          =  1.000                           *
   *  DESIGN BASE SHEAR    =  1.000 X  0.217 X       800.00   *
   *                       =       173.33  KN                 *
   ************************************************************
    JOINT           LATERAL          TORSIONAL            LOAD -    1
                    LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----           -------          ---------
       2     FX        3.907    MY        0.000
       3     FX        3.907    MY        0.000
       8     FX        3.907    MY        0.000
       9     FX        3.907    MY        0.000
                -----------        -----------
          TOTAL =     15.628              0.000 AT LEVEL     4.000 METE
       5     FX        8.377    MY        0.000
       6     FX        8.377    MY        0.000
      11     FX        8.377    MY        0.000
      12     FX        8.377    MY        0.000
                -----------        -----------
          TOTAL =     33.508              0.000 AT LEVEL     8.000 METE
      13     FX       13.088    MY        0.000
      14     FX       13.088    MY        0.000
      15     FX       13.088    MY        0.000
      16     FX       13.088    MY        0.000
                -----------        -----------
          TOTAL =     52.351              0.000 AT LEVEL    12.000 METE
      17     FX       17.962    MY        0.000
      18     FX       17.962    MY        0.000
      19     FX       17.962    MY        0.000
      20     FX       17.962    MY        0.000
                -----------        -----------
          TOTAL =     71.846              0.000 AT LEVEL    16.000 METE
```

V. IS 1893

V. IS 1893 2016 Response Spectrum

Calculation of base shear and its distribution along the height for response spectrum method in IS 1893 (Part 1) : 2016

Problem

A three story building is modelled in STAAD.Pro considering the following:

Seismic zone factor, Z = 0.36
Importance factor, I = 1.2
Response reduction factor, R = 5
Soil site conditions = Hard

The generated model is subjected to a response spectrum load along global X direction. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 495: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 496: Mode Shapes

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	0	0	0	0	0	0
	1st Floor/Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0

Table 497: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight $Mk \times g$		227.66			16.345			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $Pk \sum Wi \times \phi$ / $\sum Wi \times \phi^2$		1.244			0.3333			0.0893		

	Mode 1	Mode 2	Mode 3
S_a/g	2.5 (time period < 0.4 s)	2.5 (time period < 0.4 s)	2.2063 (1 + 15 × 0.08042 as time period < 0.1 s)
Ak (Z/2 × I/R × S_a/g)	0.108 (0.36/2 × 1.2/5 × 2.5)	0.108 (0.36/2 × 1.2/5 × 2.5)	0.09531216 (0.36/2 × 1.2/5 × 2.2063)

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	
Roof	49.035	1	6.588	6.588	-1	-1.765	-1.765	1	0.4174	0.4174	6.83
2nd Floor	98.07	0.86603	11.411	18.0	0	0	-1.765	-0.86603	-0.7230	-0.3056	18.09

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	
1st Floor	98.07	0.5	6.588	24.587	1	3.531	1.765	0.5	0.4174	0.1118	24.65
Base Shear											24.65

Comparison

Table 498: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear (kN)	Roof	6.83	6.83	none	
	2nd	18.09	18.09	none	
	1st	24.65	24.65	none	
Base shear (kN)		24.65	24.65	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Response Spectrum.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
```

```
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IS1893 2016 X 0.0432 DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
 1893 RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     RESPONSE LOAD CASE     1
        MODE    SPECTRAL ACCELERATION       DESIGN   SEISMIC   COEFFICIENT
        ----    ---------------------     ------------------------------
```

```
                                              X           Y           Z
         1          2.50000                0.1080      0.0000      0.0000
         2          2.50000                0.1080      0.0000      0.0000
         3          2.20632                0.0953      0.0000      0.0000
      STAAD SPACE                                                -- PAGE NO.
6
 PEAK STORY SHEAR
       STORY          LEVEL IN METE                    PEAK STORY SHEAR IN
KN
       -----          -------------
--------------------------------------------------
                                                  X
Y                 Z
         3                   9.00            6.83
0.00              0.00
         2                   6.00           18.09
0.00              0.00
         1                   3.00           24.65
0.00              0.00
       BASE                  0.00           24.65
0.00              0.00
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
           DIRECTIONAL VALUES:                SCALE FACTOR =    1.00
    X =  0.04   Y =  0.00   Z =  0.00         DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           0.3001         24.5166
           0.1099         24.5166
           0.0804         21.6366
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
              1                      2.50000       0.05000
              2                      2.50000       0.05000
              3                      2.20632       0.05000
      STAAD SPACE                                                -- PAGE NO.
7
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.300       24.59        0.00        0.00        0.00
0.00     -147.52
      2      0.110        1.77        0.00        0.00        0.00
0.00        5.30
      3      0.080        0.11        0.00        0.00        0.00
0.00       -0.67
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X      Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000     24.59     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      1.77     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.11     0.00
```

```
0.00
---------------------------
                                  TOTAL SRSS  SHEAR     24.65     0.00
0.00
                                  TOTAL 10PCT SHEAR     24.65     0.00
0.00
                                  TOTAL ABS   SHEAR     26.46     0.00
0.00
                                  TOTAL CSM   SHEAR     24.65     0.00
0.00
```

Related Links

- *TR.32.10.1.8 Response Spectrum Specification per IS: 1893 (Part 1)-2016* (on page 2544)

V. IS 1893 2016 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static method in IS 1893 (Part 1) : 2016

Problem

A reinforced concrete frame structure is 4 bays at 4m ea. by 3 bays at 4m ea. in plan. The structure is 3 stories at 3m ea.

The structure is modelled in STAAD.Pro considering the following:

Seismic zone factor, $Z = 0.36$ (`ZONE 0.36`)
Importance factor, $I = 1.2$ (`I 1.2`)
Response reduction factor, $R = 5$ (`RF 5`)
Soil site condition = Hard (`SS 1`)
Structure type is a reinforced concrete frame (`ST 1`)

The seismic weight for each floor is assumed as 100 kN applied as joint weight to nodes 39, 60 & 80. Seismic Loads as per IS 1893 (Part 1) : 2016 specifications are generated along two horizontal directions global X & global Z and also along vertical direction global Y. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Height = 3 × 3 = 9m

$$Ta_x = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}$$

$$Ta_z = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}$$

$$[Sa/g]_x = [Sa/g]_z = 2.5 \text{ (As time period < 0.4 sec)}$$

$$[Sa/g]_y = 2.5 \text{ (constant)}$$

$$Ah_x = (Z/2) \times (I/R) \times [Sa/g]_x = 0.36/2 \times 1.2/5 \times 2.5 = 0.108$$

$$Ah_z = (Z/2) \times (I/R) \times [Sa/g]_z = 0.36/2 \times 1.2/5 \times 2.5 = 0.108$$

$$Av = (2/3) \times (Z/2) \times (I/R) \times [Sa/g]_y = 2/3 \times 0.36/2 \times 1.2/5 \times 2.5 = 0.072$$

$$VB = Ah \times W \text{ or } Av \times W$$

$$VB_x = Ah_x \times W = 0.108 \times (3 \times 100) = 32.4 \text{ kN}$$

$VBmin_x$ = 2.4% of W = 2.4 × (3 × 100) / 100 = 7.2 kN

$VB_x > VBmin_x$

$VB_z = Ah_z \times W = 0.108 \times (3 \times 100) = 32.4$ kN

$VBmin_z$ = 2.4% of W = 2.4 × (3 × 100) / 100 = 7.2 kN

$VB_z > VBmin_z$

$VB_y = Av \times W = 0.072 \times (3 \times 100) = 21.6$ kN

Story Level	Wi	hi	Wi × hi ^{2}	(Wi × hi ^{2})/∑ (Wi × hi ^{2})	Qx	Qz	Qy
Roof	100	9	8,100	0.6429	20.83	20.83	13.886
2nd	100	6	3,600	0.2857	9.257	9.257	6.171
1st	100	3	900	0.0714	2.314	2.314	1.543
Summation	300		12,600				
Vb_h		32.4			32.4		
Vbv		21.6					
$Vb_{h(min)}$		7.2			7.2		

Comparison

Table 499: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear X, Z, Y (kN)	Roof	2.314, 2.314, 1.543	2.314, 2.314, 1.543	none	
	2nd	9.257, 9.257, 6.171	9.257, 9.257, 6.171	none	
	1st	20.829, 20.829, 13.886	20.829, 20.829, 13.886	none	
Base shear (kN)	VB_x	32.4	32.4	none	
	VB_z	32.4	32.4	none	
	VB_y	21.6	21.6	none	
Min base shear (kN)	VB_x	7.2	7.2	none	
	VB_z	7.2	7.2	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Jun-18
END JOB INFORMATION
INPUT WIDTH 79
*SET STAR 0
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 4 3 0; 4 4 0 0; 5 0 0 4; 6 0 3 4; 7 4 3 4; 8 4 0 4;
9 8 3 0; 10 8 0 0; 11 8 3 4; 12 8 0 4; 13 12 3 0; 14 12 0 0; 15 12 3 4;
16 12 0 4; 17 16 3 0; 18 16 0 0; 19 16 3 4; 20 16 0 4; 21 0 0 8; 22 0 3 8;
23 4 3 8; 24 4 0 8; 25 8 3 8; 26 8 0 8; 27 12 3 8; 28 12 0 8; 29 16 3 8;
30 16 0 8; 31 0 0 12; 32 0 3 12; 33 4 3 12; 34 4 0 12; 35 8 3 12; 36 8 0 12;
37 12 3 12; 38 12 0 12; 39 16 3 12; 40 16 0 12; 41 0 6 0; 42 4 6 0; 43 0 6 4;
44 4 6 4; 45 8 6 0; 46 8 6 4; 47 12 6 0; 48 12 6 4; 49 16 6 0; 50 16 6 4;
51 0 6 8; 52 4 6 8; 53 8 6 8; 54 12 6 8; 55 16 6 8; 56 0 6 12; 57 4 6 12;
58 8 6 12; 59 12 6 12; 60 16 6 12; 61 0 9 0; 62 4 9 0; 63 0 9 4; 64 4 9 4;
65 8 9 0; 66 8 9 4; 67 12 9 0; 68 12 9 4; 69 16 9 0; 70 16 9 4;
71 0 9 8; 72 4 9 8; 73 8 9 8; 74 12 9 8; 75 16 9 8; 76 0 9 12;
77 4 9 12; 78 8 9 12; 79 12 9 12; 80 16 9 12;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 3 9; 10 7 11;
11 9 10; 12 9 11; 13 11 12; 14 9 13; 15 11 15; 16 13 14; 17 13 15; 18 15 16;
19 13 17; 20 15 19; 21 17 18; 22 17 19; 23 19 20; 24 6 22; 25 7 23; 26 11 25;
27 15 27; 28 19 29; 29 21 22; 30 22 23; 31 23 24; 32 23 25; 33 25 26; 34 25
27;
35 27 28; 36 27 29; 37 29 30; 38 22 32; 39 23 33; 40 25 35; 41 27 37; 42 29
39;
43 31 32; 44 32 33; 45 33 34; 46 33 35; 47 35 36; 48 35 37; 49 37 38; 50 37
39;
51 39 40; 52 2 41; 53 3 42; 54 6 43; 55 7 44; 56 9 45; 57 11 46; 58 13 47;
59 15 48; 60 17 49; 61 19 50; 62 22 51; 63 23 52; 64 25 53; 65 27 54; 66 29
55;
67 32 56; 68 33 57; 69 35 58; 70 37 59; 71 39 60; 72 41 42; 73 41 43; 74 42
44;
75 43 44; 76 42 45; 77 44 46; 78 45 46; 79 45 47; 80 46 48; 81 47 48; 82 47
49;
83 48 50; 84 49 50; 85 43 51; 86 44 52; 87 46 53; 88 48 54; 89 50 55; 90 51
52;
91 52 53; 92 53 54; 93 54 55; 94 51 56; 95 52 57; 96 53 58; 97 54 59; 98 55
60;
99 56 57; 100 57 58; 101 58 59; 102 59 60; 103 41 61; 104 42 62; 105 43 63;
106 44 64; 107 45 65; 108 46 66; 109 47 67; 110 48 68; 111 49 69; 112 50 70;
113 51 71; 114 52 72; 115 53 73; 116 54 74; 117 55 75; 118 56 76; 119 57 77;
120 58 78; 121 59 79; 122 60 80; 123 61 62; 124 61 63; 125 62 64; 126 63 64;
127 62 65; 128 64 66; 129 65 66; 130 65 67; 131 66 68; 132 67 68; 133 67 69;
134 68 70; 135 69 70; 136 63 71; 137 64 72; 138 66 73; 139 68 74; 140 70 75;
141 71 72; 142 72 73; 143 73 74; 144 74 75; 145 71 76; 146 72 77; 147 73 78;
148 74 79; 149 75 80; 150 76 77; 151 77 78; 152 78 79; 153 79 80;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
```

```
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 153 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 10 12 14 16 18 20 21 24 26 28 30 31 34 36 38 40 FIXED
*DEFINE 1893 LOAD
DEFINE IS1893 2016 LOAD
*ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05 HT 9 DX 16 DZ 12
*ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05 HT 9 DX 16 DZ 12
*ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
 BASE SHEAR AND TIME PERIOD IN X
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR X 1893 LOADING =    0.38971 SEC       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.1080 X      300.00=      32.40 KN   *
   * VB Act Based on Clause 7.2.1 =       32.40 KN         *
   * VB Min based on Clause 7.2.2 =        7.20 KN         *
   *                                                       *
   *********************************************************
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Z
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR Z 1893 LOADING =    0.38971 SEC       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.1080 X      300.00=      32.40 KN   *
   * VB Act Based on Clause 7.2.1 =       32.40 KN         *
   * VB Min based on Clause 7.2.2 =        7.20 KN         *
```

```
   *                                                        *
   **********************************************************
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Y
   **********************************************************
   * UNITS - KN    METE                                     *
   * Calculation of SA/G for Y  based on Clause 6.4.6       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000            *
   * VB PER 1893=    0.0720 X      300.00=      21.60 KN    *
   *                                                        *
   **********************************************************
      STAAD SPACE                                                -- PAGE NO.
5
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      39     FX        2.314    MY        0.000
                 -----------        -----------
          TOTAL =      2.314              0.000 AT LEVEL     3.000 METE
    VB PER 1893 =     32.400 KN
      60     FX        9.257    MY        0.000
                 -----------        -----------
          TOTAL =      9.257              0.000 AT LEVEL     6.000 METE
    VB PER 1893 =     32.400 KN
      80     FX       20.829    MY        0.000
                 -----------        -----------
          TOTAL =     20.829              0.000 AT LEVEL     9.000 METE
    VB PER 1893 =     32.400 KN
    JOINT            LATERAL          TORSIONAL            LOAD -    2
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      39     FZ        2.314    MY        0.000
                 -----------        -----------
          TOTAL =      2.314              0.000 AT LEVEL     3.000 METE
    VB PER 1893 =     32.400 KN
      60     FZ        9.257    MY        0.000
                 -----------        -----------
          TOTAL =      9.257              0.000 AT LEVEL     6.000 METE
    VB PER 1893 =     32.400 KN
      80     FZ       20.829    MY        0.000
                 -----------        -----------
          TOTAL =     20.829              0.000 AT LEVEL     9.000 METE
    VB PER 1893 =     32.400 KN
    JOINT            LATERAL          TORSIONAL            LOAD -    3
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      39     FY        1.543    MY        0.000
                 -----------        -----------
          TOTAL =      1.543              0.000 AT LEVEL     3.000 METE
    VB PER 1893 =     21.600 KN
      60     FY        6.171    MY        0.000
                 -----------        -----------
          TOTAL =      6.171              0.000 AT LEVEL     6.000 METE
    VB PER 1893 =     21.600 KN
      80     FY       13.886    MY        0.000
```

```
      STAAD SPACE                                                   -- PAGE NO.
6
                 -----------       -----------
          TOTAL =     13.886             0.000 AT LEVEL     9.000 METE
    VB PER 1893 =     21.600 KN
```

Related Links

- *TR.31.2.11 IS:1893 (Part 1) 2016 Codes - Lateral Seismic Load* (on page 2393)

V. IS 1893 2016 GL Calculation

Calculation of the base shear and its distribution along the height using the static method for Global X, Global Z, Global Y directions using IS 1893:2016 (Part 1).

Details

The following structure is modelled.

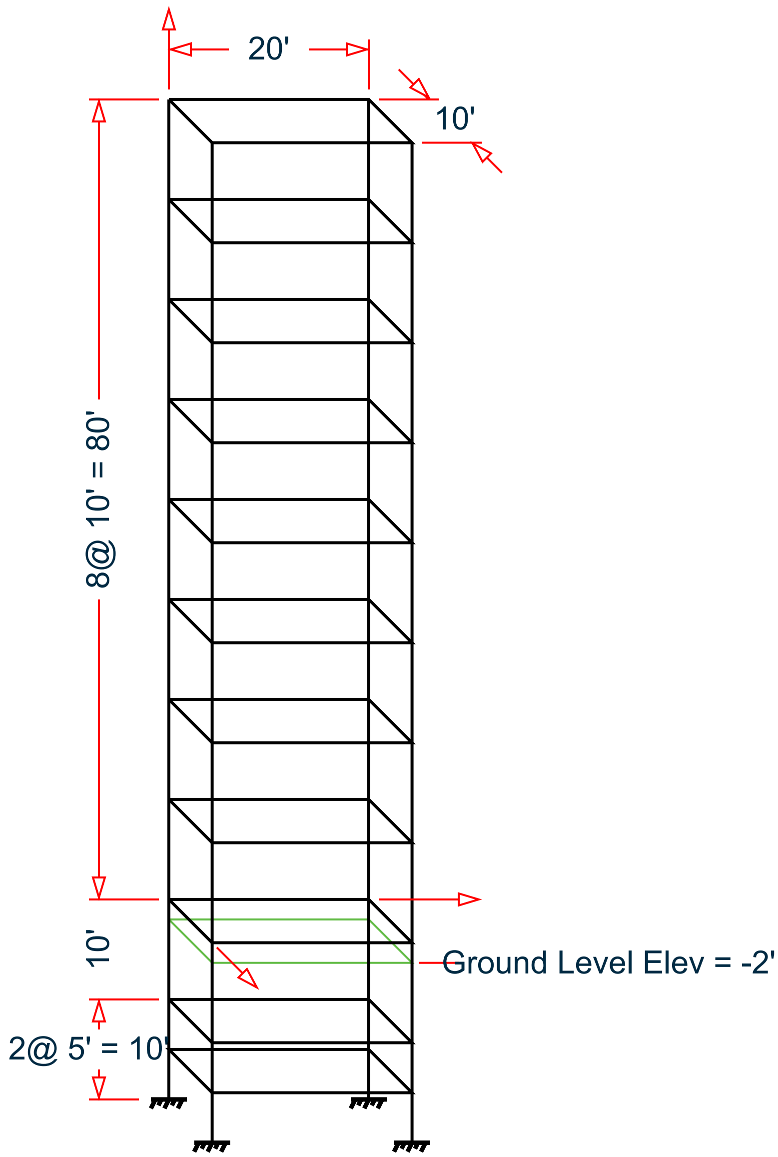


Figure 429:

The STAAD.Pro load definition parameters are also specified:

Parameter	Value	STAAD.Pro Value
Seismic zone factor, Z	0.16	`Z 0.16`
Importance factor, I	1	`I 1`
Response reduce factor, R	3 (OMRF)	`RF 3`
Soil site condition	Hard soil	`SS 1`

Parameter	Value	STAAD.Pro Value
Structure type	Steel moment-resisting frame building	`ST 3`

Total height of the building is 100 ft

Dimension of each floor slab is 20 ft × 10 ft

Floor to floor height for super structure is 10 ft

Floor to floor height for sub structure is 5 ft

GL is located at -2 ft (Y co-ordinate)

Depth of Foundation (DT) is (20-2) = 18 ft = 5.4864 m

All the analytical members are W shaped steel beam (W12X190)

Supports are fixed

The generated model is subjected to joint weight(100kip). The base shear and its distribution along the height is calculated in Global X, Global Z, Global Y direction.

Verification

Design horizontal acceleration coefficient value calculation for above GL and below GL for Global X(Ah), Global Z(Ah), Global Y(Av) direction.

Table 500: Calculated values for hard soil

Value	X	Z	Y
Time Period (s)	1.103	1.103	
Sa/g	0.907 $(1/T,\ 0.4s < t < 4s)$	0.907 $(1/T,\ 0.4s < t < 4s)$	2.5
A_h or A_v	0.02418	0.02418	0.04445
A_h or A_v 30 m	0.01209	0.01209	0.0222
Reduced A_h or A_v	0.02197	0.02197	0.04038

Base shear calculation and its distribution along the height for above GL and below GL for Global X, Global Z, Global Y direction and V_{bmin} calculation

Table 501: Vertical distribution of forces above GL

Story Level in ft from base.	Wi (kip)	hi (m from GL)	Wi× hi^2	(Wi × hi^2)/∑ (Wi × hi$_2$)	Qx (kip)	Qz (kip)	Qy (kip)
100	400	24.9936	2,689,600	0.3074	26.761	26.761	49.179

Story Level in ft from base.	Wi (kip)	hi (m from GL)	Wi× hi^2	(Wi × hi^2)/∑ (Wi × hi_2)	Qx (kip)	Qz (kip)	Qy (kip)
90	400	21.9456	2,073,600	0.2370	20.632	20.632	37.916
80	400	18.8976	1,537,600	0.1757	15.299	15.299	28.115
70	400	15.8496	1,081,600	0.1236	10.762	10.762	19.777
60	400	12.8016	705,600	0.0806	7.021	7.021	12.902
50	400	9.7536	409,600	0.0468	4.075	4.075	7.489
40	400	6.7056	193,600	0.0221	1.926	1.926	3.540
30	400	3.6576	57,600	0.0066	0.573	0.573	1.053
20	400	0.6096	1,600	0.0002	0.016	0.016	0.029
Summation	3,600		8,750,400	1	87.064	87.064	160
Vbh					87.064	87.064	160
Vbmin					39.6	39.6	

Table 502: Vertical distribution of forces below GL

Below ground level (ft)	Wi (kip)	hi (m below GL)	Wi× hi^2	(Wi × hi^2)/∑ (Wi × hi_2)	Qx (kip)	Qz (kip)	Qy (kip)
10	400	3.048	40,000	0.8	14.063	14.063	25.843
5	400	1.524	10,000	0.2	3.516	3.516	6.461
Summation	800		50,000		17.579	17.579	32.304
Vbh					17.579	17.579	32.304
Vbmin					8.8	8.8	

Results

Table 503: Comparison of results

Location	Shear values	Hand Calculations			STAAD.Pro			Difference	Comments
		Qi(X)	Qi(Z)	Qi(Y)	Qi(X)	Qi(Z)	Qi(Y)		
Above GL	Story shear (kip)	26.761	26.761	49.179	26.761	26.761	49.179	none	
		20.632	20.632	37.916	20.632	20.632	37.916	none	
		15.299	15.299	28.115	15.299	15.299	28.115	none	
		10.762	10.762	19.777	10.762	10.762	19.777	none	
		7.021	7.021	12.902	7.021	7.021	12.902	none	
		4.075	4.075	7.489	4.075	4.075	7.489	none	
		1.926	1.926	3.540	1.962	1.962	3.54	none	
		0.573	0.573	1.053	0.573	0.573	1.053	none	
		0.016	0.016	0.029	0.016	0.016	0.029	none	
	Base Shear (kip)	87.064	87.064	160	87.064	87.064	160	none	
Below GL	Story Shear (kip)	14.063	14.063	25.843	14.063	14.063	25.843	none	
		3.516	3.516	6.461	3.516	3.516	6.461	none	
	Base Shear (kip)	17.579	17.579	32.304	17.579	17.579	32.304	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 GL Calculation.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Aug-18
END JOB INFORMATION
UNIT FEET KIP
```

```
JOINT COORDINATES
1 0 -20 0; 2 0 -10 0; 3 20 -20 0; 4 20 -10 0; 5 0 0 0; 6 20 0 0;
7 0 -10 10; 8 20 -10 10; 9 0 0 10; 10 20 0 10; 11 0 -20 10;
12 20 -20 10; 13 0 10 0; 14 20 10 0; 15 0 10 10; 16 20 10 10; 17 0 20 0;
18 20 20 0; 19 0 20 10; 20 20 20 10; 21 0 30 0; 22 20 30 0; 23 0 30 10;
24 20 30 10; 25 0 40 0; 26 20 40 0; 27 0 40 10; 28 20 40 10; 29 0 50 0;
30 20 50 0; 31 0 50 10; 32 20 50 10; 33 0 60 0; 34 20 60 0; 35 0 60 10;
36 20 60 10; 37 0 70 0; 38 20 70 0; 39 0 70 10; 40 20 70 10; 41 0 80 0;
42 20 80 0; 43 0 80 10; 44 20 80 10; 45 0 -15 10; 46 0 -15 0;
47 20 -15 10; 48 20 -15 0;
MEMBER INCIDENCES
1 1 46; 2 3 48; 3 2 5; 4 4 6; 5 2 4; 6 5 6; 7 2 7; 8 4 8; 9 5 9;
10 6 10; 11 11 45; 12 12 47; 13 7 9; 14 8 10; 15 7 8; 16 9 10; 17 5 13;
18 6 14; 19 13 14; 20 13 15; 21 14 16; 22 9 15; 23 10 16; 24 15 16;
25 13 17; 26 14 18; 27 17 18; 28 17 19; 29 18 20; 30 15 19; 31 16 20;
32 19 20; 33 17 21; 34 18 22; 35 21 22; 36 21 23; 37 22 24; 38 19 23;
39 20 24; 40 23 24; 41 21 25; 42 22 26; 43 25 26; 44 25 27; 45 26 28;
46 23 27; 47 24 28; 48 27 28; 49 25 29; 50 26 30; 51 29 30; 52 29 31;
53 30 32; 54 27 31; 55 28 32; 56 31 32; 57 29 33; 58 30 34; 59 33 34;
60 31 35; 61 32 36; 62 35 36; 63 33 37; 64 34 38; 65 37 38; 66 35 39;
67 36 40; 68 39 40; 69 33 35; 70 34 36; 71 37 39; 72 38 40; 73 37 41;
74 38 42; 75 41 42; 76 39 43; 77 40 44; 78 43 44; 79 41 43; 80 42 44;
81 45 7; 82 46 2; 83 47 8; 84 48 4; 85 45 47; 86 47 48; 87 48 46;
88 46 45;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 88 TABLE ST W12X190
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 11 12 FIXED
DEFINE IS1893 2016 LOAD
ZONE 0.16 RF 3 I 1 SS 1 ST 3 GL -2
JOINT WEIGHT
2 4 TO 10 13 TO 48 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
```

```
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN X
 --ABOVE GROUND LEVEL
   ********************************************************
   * UNITS - KIP   FEET                                   *
   * TIME PERIOD FOR X 1893 LOADING =    1.10263 SEC      *
   * SA/G PER 1893=    0.907, LOAD FACTOR= 1.000          *
   * VB PER 1893=    0.0242 X     3600.00=      87.06 KIP  *
   * VB Act Based on Clause 7.2.1 =       87.06 KIP       *
   * VB Min based on Clause 7.2.2 =       39.60 KIP       *
   *                                                      *
   ********************************************************
 --BELOW GROUND LEVEL
   ********************************************************
   * UNITS - KIP   FEET                                   *
   * TIME PERIOD FOR X 1893 LOADING =    1.10263 SEC      *
   * SA/G PER 1893=    0.907, LOAD FACTOR= 1.000          *
   * FACTOR V  PER 1893 AT GL=    0.0242 X    800.00      *
   * FACTOR V  PER 1893 AT 30 M=    0.0121 X    800.00    *
   *      FACTOR V  PER 1893=    0.0220 X    800.00       *
   * VB Min based on Clause 7.2.2 =        8.80 KIP       *
   *                                                      *
   ********************************************************
 --WEIGHT AND BASE SHEAR SUMMERY
   Units:KIP   FEET
|------------------------------------------------------------------------------
-|
| Category |   Weight   |     Ah     | VB Calculated | VB Minimum |  VB
Final  |
|------------------------------------------------------------------------------
-|
| Above GL |  3600.00000|   0.0241846|      87.06450 |    39.60000|
87.06450|
|------------------------------------------------------------------------------
-|
| Below GL |   800.00000|   0.0219731|      17.57852 |     8.80000|
17.57852|
|------------------------------------------------------------------------------
-|
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Z
 --ABOVE GROUND LEVEL
   ********************************************************
   * UNITS - KIP   FEET                                   *
   * TIME PERIOD FOR Z 1893 LOADING =    1.10263 SEC      *
   * SA/G PER 1893=    0.907, LOAD FACTOR= 1.000          *
   * VB PER 1893=    0.0242 X     3600.00=      87.06 KIP  *
   * VB Act Based on Clause 7.2.1 =       87.06 KIP       *
   * VB Min based on Clause 7.2.2 =       39.60 KIP       *
   *                                                      *
   ********************************************************
 --BELOW GROUND LEVEL
   ********************************************************
   * UNITS - KIP   FEET                                   *
```

```
   * TIME PERIOD FOR Z 1893 LOADING =    1.10263 SEC       *
   * SA/G PER 1893=    0.907, LOAD FACTOR= 1.000           *
   * FACTOR V  PER 1893 AT GL=    0.0242 X    800.00       *
   * FACTOR V  PER 1893 AT 30 M=    0.0121 X    800.00     *
   *      FACTOR V  PER 1893=    0.0220 X    800.00        *
   * VB Min based on Clause 7.2.2 =       8.80 KIP         *
   *                                                       *
   *********************************************************
 --WEIGHT AND BASE SHEAR SUMMERY
   Units:KIP   FEET
|------------------------------------------------------------------------------
-|
| Category |   Weight   |     Ah     | VB Calculated | VB Minimum |  VB
Final  |
|------------------------------------------------------------------------------
-|
| Above GL |  3600.00000|   0.0241846|      87.06450 |    39.60000|
87.06450|
|------------------------------------------------------------------------------
-|
| Below GL |   800.00000|   0.0219731|      17.57852 |     8.80000|
17.57852|
|------------------------------------------------------------------------------
-|
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Y
 --ABOVE GROUND LEVEL
   *********************************************************
   * UNITS - KIP   FEET                                    *
   * Calculation of SA/G for Y  based on Clause 6.4.6      *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.0444 X     3600.00=     160.00 KIP  *
   *                                                       *
   *********************************************************
 --BELOW GROUND LEVEL
   *********************************************************
   * UNITS - KIP   FEET                                    *
   * Calculation of SA/G for Y  based on Clause 6.4.6       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * FACTOR V  PER 1893 AT GL=    0.0444 X    800.00       *
   * FACTOR V  PER 1893 AT 30 M=    0.0223 X    800.00     *
   *      FACTOR V  PER 1893=    0.0404 X    800.00        *
   *                                                       *
   *********************************************************
 --WEIGHT AND BASE SHEAR SUMMERY
   Units:KIP   FEET
|------------------------------------------------------------------------------
-|
| Category |   Weight   |     Ah     | VB Calculated | VB Minimum |  VB
Final  |
|------------------------------------------------------------------------------
-|
| Above GL |  3600.00000|   0.0444444|     160.00002 |     0.00000|
160.00002|
|------------------------------------------------------------------------------
```

```
-|
| Below GL |   800.00000|   0.0403805|      32.30436 |     0.00000|
32.30436|
|--------------------------------------------------------------------------------
-|
      STAAD SPACE                                              -- PAGE NO.
4
    JOINT            LATERAL          TORSIONAL           LOAD -    1
                     LOAD (KIP )      MOMENT (KIP -FEET) FACTOR -   1.000
    -----            -------          ---------
      45     FX        0.879    MY        0.000
      46     FX        0.879    MY        0.000
      47     FX        0.879    MY        0.000
      48     FX        0.879    MY        0.000
             -----------        -----------
        TOTAL =      3.516              0.000 AT LEVEL   -15.000 FEET
    VB PER 1893 =     17.579 KIP
       2     FX        3.516    MY        0.000
       4     FX        3.516    MY        0.000
       7     FX        3.516    MY        0.000
       8     FX        3.516    MY        0.000
             -----------        -----------
        TOTAL =     14.063              0.000 AT LEVEL   -10.000 FEET
    VB PER 1893 =     17.579 KIP
       5     FX        0.004    MY        0.000
       6     FX        0.004    MY        0.000
       9     FX        0.004    MY        0.000
      10     FX        0.004    MY        0.000
             -----------        -----------
        TOTAL =      0.016              0.000 AT LEVEL     0.000 FEET
    VB PER 1893 =     87.064 KIP
      13     FX        0.143    MY        0.000
      14     FX        0.143    MY        0.000
      15     FX        0.143    MY        0.000
      16     FX        0.143    MY        0.000
             -----------        -----------
        TOTAL =      0.573              0.000 AT LEVEL    10.000 FEET
    VB PER 1893 =     87.064 KIP
      17     FX        0.482    MY        0.000
      18     FX        0.482    MY        0.000
      19     FX        0.482    MY        0.000
      20     FX        0.482    MY        0.000
             -----------        -----------
        TOTAL =      1.926              0.000 AT LEVEL    20.000 FEET
    VB PER 1893 =     87.064 KIP
      21     FX        1.019    MY        0.000
      22     FX        1.019    MY        0.000
      23     FX        1.019    MY        0.000
      24     FX        1.019    MY        0.000
             -----------        -----------
        TOTAL =      4.075              0.000 AT LEVEL    30.000 FEET
    VB PER 1893 =     87.064 KIP
      25     FX        1.755    MY        0.000
      26     FX        1.755    MY        0.000
      27     FX        1.755    MY        0.000
      28     FX        1.755    MY        0.000
      STAAD SPACE                                              -- PAGE NO.
5
```

```
                 -----------         -----------
        TOTAL =      7.021              0.000 AT LEVEL    40.000 FEET
  VB PER 1893 =     87.064 KIP
    29     FX        2.690    MY        0.000
    30     FX        2.690    MY        0.000
    31     FX        2.690    MY        0.000
    32     FX        2.690    MY        0.000
                 -----------         -----------
        TOTAL =     10.762              0.000 AT LEVEL    50.000 FEET
  VB PER 1893 =     87.064 KIP
    33     FX        3.825    MY        0.000
    34     FX        3.825    MY        0.000
    35     FX        3.825    MY        0.000
    36     FX        3.825    MY        0.000
                 -----------         -----------
        TOTAL =     15.299              0.000 AT LEVEL    60.000 FEET
  VB PER 1893 =     87.064 KIP
    37     FX        5.158    MY        0.000
    38     FX        5.158    MY        0.000
    39     FX        5.158    MY        0.000
    40     FX        5.158    MY        0.000
                 -----------         -----------
        TOTAL =     20.632              0.000 AT LEVEL    70.000 FEET
  VB PER 1893 =     87.064 KIP
    41     FX        6.690    MY        0.000
    42     FX        6.690    MY        0.000
    43     FX        6.690    MY        0.000
    44     FX        6.690    MY        0.000
                 -----------         -----------
        TOTAL =     26.761              0.000 AT LEVEL    80.000 FEET
  VB PER 1893 =     87.064 KIP
  JOINT            LATERAL           TORSIONAL            LOAD -    2
                   LOAD (KIP )       MOMENT (KIP -FEET) FACTOR -   1.000
  -----            -------           ---------
    45     FZ        0.879    MY        0.000
    46     FZ        0.879    MY        0.000
    47     FZ        0.879    MY        0.000
    48     FZ        0.879    MY        0.000
                 -----------         -----------
        TOTAL =      3.516              0.000 AT LEVEL   -15.000 FEET
  VB PER 1893 =     17.579 KIP
     2     FZ        3.516    MY        0.000
     4     FZ        3.516    MY        0.000
     7     FZ        3.516    MY        0.000
     8     FZ        3.516    MY        0.000
    STAAD SPACE                                             -- PAGE NO.
6
                 -----------         -----------
        TOTAL =     14.063              0.000 AT LEVEL   -10.000 FEET
  VB PER 1893 =     17.579 KIP
     5     FZ        0.004    MY        0.000
     6     FZ        0.004    MY        0.000
     9     FZ        0.004    MY        0.000
    10     FZ        0.004    MY        0.000
                 -----------         -----------
        TOTAL =      0.016              0.000 AT LEVEL     0.000 FEET
  VB PER 1893 =     87.064 KIP
    13     FZ        0.143    MY        0.000
```

```
    14     FZ        0.143    MY        0.000
    15     FZ        0.143    MY        0.000
    16     FZ        0.143    MY        0.000
                ----------        -----------
        TOTAL =      0.573            0.000 AT LEVEL    10.000 FEET
  VB PER 1893 =     87.064 KIP
    17     FZ        0.482    MY        0.000
    18     FZ        0.482    MY        0.000
    19     FZ        0.482    MY        0.000
    20     FZ        0.482    MY        0.000
                ----------        -----------
        TOTAL =      1.926            0.000 AT LEVEL    20.000 FEET
  VB PER 1893 =     87.064 KIP
    21     FZ        1.019    MY        0.000
    22     FZ        1.019    MY        0.000
    23     FZ        1.019    MY        0.000
    24     FZ        1.019    MY        0.000
                ----------        -----------
        TOTAL =      4.075            0.000 AT LEVEL    30.000 FEET
  VB PER 1893 =     87.064 KIP
    25     FZ        1.755    MY        0.000
    26     FZ        1.755    MY        0.000
    27     FZ        1.755    MY        0.000
    28     FZ        1.755    MY        0.000
                ----------        -----------
        TOTAL =      7.021            0.000 AT LEVEL    40.000 FEET
  VB PER 1893 =     87.064 KIP
    29     FZ        2.690    MY        0.000
    30     FZ        2.690    MY        0.000
    31     FZ        2.690    MY        0.000
    32     FZ        2.690    MY        0.000
                ----------        -----------
        TOTAL =     10.762            0.000 AT LEVEL    50.000 FEET
  VB PER 1893 =     87.064 KIP
    33     FZ        3.825    MY        0.000
    34     FZ        3.825    MY        0.000
    35     FZ        3.825    MY        0.000
    36     FZ        3.825    MY        0.000
    STAAD SPACE                                              -- PAGE NO.
7
                ----------        -----------
        TOTAL =     15.299            0.000 AT LEVEL    60.000 FEET
  VB PER 1893 =     87.064 KIP
    37     FZ        5.158    MY        0.000
    38     FZ        5.158    MY        0.000
    39     FZ        5.158    MY        0.000
    40     FZ        5.158    MY        0.000
                ----------        -----------
        TOTAL =     20.632            0.000 AT LEVEL    70.000 FEET
  VB PER 1893 =     87.064 KIP
    41     FZ        6.690    MY        0.000
    42     FZ        6.690    MY        0.000
    43     FZ        6.690    MY        0.000
    44     FZ        6.690    MY        0.000
                ----------        -----------
        TOTAL =     26.761            0.000 AT LEVEL    80.000 FEET
  VB PER 1893 =     87.064 KIP
  JOINT            LATERAL         TORSIONAL          LOAD -    3
```

```
                  LOAD (KIP )      MOMENT (KIP -FEET) FACTOR -   1.000
    -----         -------          ---------
      45     FY        1.615    MY        0.000
      46     FY        1.615    MY        0.000
      47     FY        1.615    MY        0.000
      48     FY        1.615    MY        0.000
               -----------      -----------
          TOTAL =      6.461              0.000 AT LEVEL   -15.000 FEET
    VB PER 1893 =     32.304 KIP
       2     FY        6.461    MY        0.000
       4     FY        6.461    MY        0.000
       7     FY        6.461    MY        0.000
       8     FY        6.461    MY        0.000
               -----------      -----------
          TOTAL =     25.843              0.000 AT LEVEL   -10.000 FEET
    VB PER 1893 =     32.304 KIP
       5     FY        0.007    MY        0.000
       6     FY        0.007    MY        0.000
       9     FY        0.007    MY        0.000
      10     FY        0.007    MY        0.000
               -----------      -----------
          TOTAL =      0.029              0.000 AT LEVEL     0.000 FEET
    VB PER 1893 =    160.000 KIP
      13     FY        0.263    MY        0.000
      14     FY        0.263    MY        0.000
      15     FY        0.263    MY        0.000
      16     FY        0.263    MY        0.000
      STAAD SPACE                                          -- PAGE NO.
8
               -----------      -----------
          TOTAL =      1.053              0.000 AT LEVEL    10.000 FEET
    VB PER 1893 =    160.000 KIP
      17     FY        0.885    MY        0.000
      18     FY        0.885    MY        0.000
      19     FY        0.885    MY        0.000
      20     FY        0.885    MY        0.000
               -----------      -----------
          TOTAL =      3.540              0.000 AT LEVEL    20.000 FEET
    VB PER 1893 =    160.000 KIP
      21     FY        1.872    MY        0.000
      22     FY        1.872    MY        0.000
      23     FY        1.872    MY        0.000
      24     FY        1.872    MY        0.000
               -----------      -----------
          TOTAL =      7.489              0.000 AT LEVEL    30.000 FEET
    VB PER 1893 =    160.000 KIP
      25     FY        3.225    MY        0.000
      26     FY        3.225    MY        0.000
      27     FY        3.225    MY        0.000
      28     FY        3.225    MY        0.000
               -----------      -----------
          TOTAL =     12.902              0.000 AT LEVEL    40.000 FEET
    VB PER 1893 =    160.000 KIP
      29     FY        4.944    MY        0.000
      30     FY        4.944    MY        0.000
      31     FY        4.944    MY        0.000
      32     FY        4.944    MY        0.000
               -----------      -----------
```

```
         TOTAL =     19.777             0.000 AT LEVEL    50.000 FEET
   VB PER 1893 =    160.000 KIP
     33     FY        7.029    MY        0.000
     34     FY        7.029    MY        0.000
     35     FY        7.029    MY        0.000
     36     FY        7.029    MY        0.000
               -----------      -----------
         TOTAL =     28.115             0.000 AT LEVEL    60.000 FEET
   VB PER 1893 =    160.000 KIP
     37     FY        9.479    MY        0.000
     38     FY        9.479    MY        0.000
     39     FY        9.479    MY        0.000
     40     FY        9.479    MY        0.000
               -----------      -----------
         TOTAL =     37.916             0.000 AT LEVEL    70.000 FEET
   VB PER 1893 =    160.000 KIP
     41     FY       12.295    MY        0.000
     42     FY       12.295    MY        0.000
     43     FY       12.295    MY        0.000
     44     FY       12.295    MY        0.000
     STAAD SPACE                                       -- PAGE NO.
9
               -----------      -----------
         TOTAL =     49.179             0.000 AT LEVEL    80.000 FEET
   VB PER 1893 =    160.000 KIP
```

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

V. IS 1893 2016 Irregular Modes of Oscillation

Verify irregular modes of oscillation check in accordance to IS 1893 2016 Part 1 specifications.

Details

A three story structure is modelled with floor diaphragms defined at the story levels.

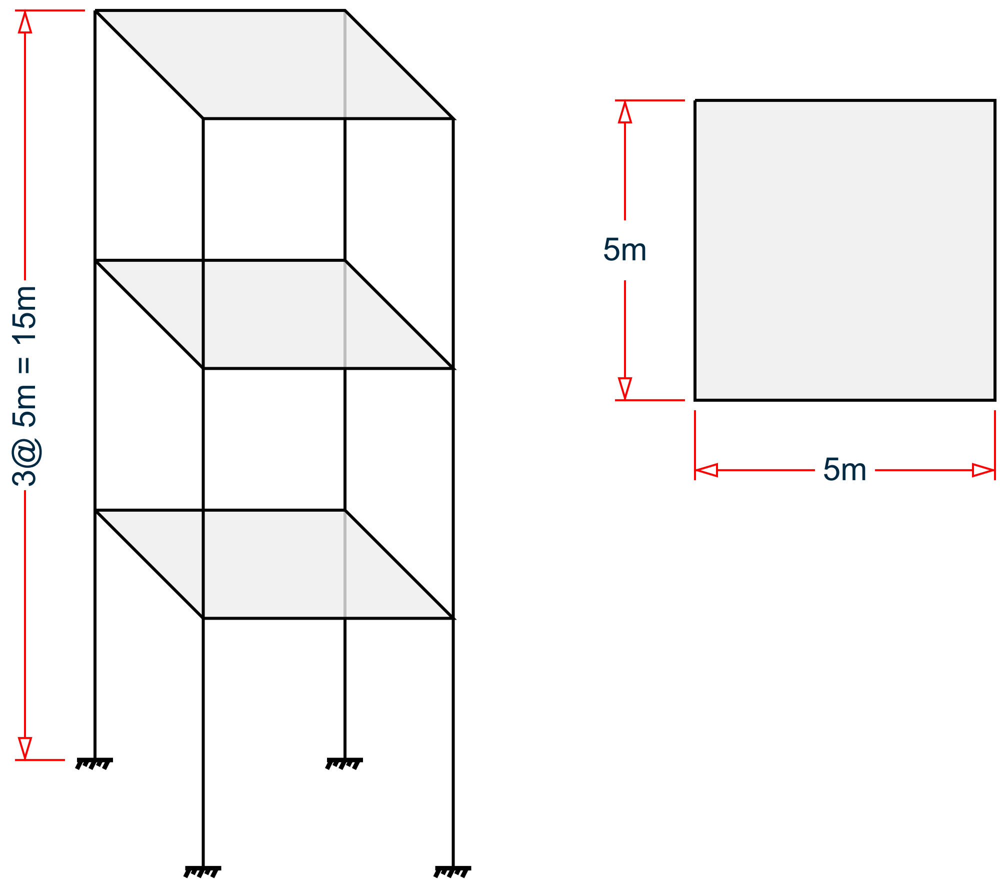


Figure 430: Isometric and Plan View of Structure

Seismic parameters:

Seismic Zone Factor, Z = 0.36
Importance Factor, I = 1.2
Response Reduction Factor, R = 5
Site soil conditions: hard soil (SS 1)
Structure type: other: (ST 5)

Verification

Since the structure is in Zone V the check has two parts. These include:

1. First three lateral translational modes together contribute at least 65% mass participation factor in each principal plan direction.
2. The fundamental lateral natural periods of the building in the two principal plan directions are away from each other by at least 10% of the larger value.

Looking at the mass participation factor table in output file following observations are made:

Mode Number	Mass Participation X	Mass Participation Z	Time Period (s)
1	0	32.66	0.24728
2	25.58	0	0.22941
3	0.09	0	0.17942
4	0	46.31	0.138
5	59.03	0	0.13033
6	0.31	0	0.09761
7	0	21.02	0.05435
8	14.69	0	0.04952
9	0.31	0	0.0419
10	0	0.01	0.02119

First three lateral translational modes for X direction are:

Mode Number	Mass Participation (%)
2	25.58
5	59.03
8	14.69
Sum	99.3
Status (PASS if sum > 65%)	PASS

First three lateral translational modes for Z direction are:

Mode Number	Mass Participation (%)
1	32.66
4	46.31
7	21.02
Sum	99.99
Status (PASS if sum > 65%)	PASS

The sum of first three lateral translational modes for both X and Z directions are more than 65% and hence, the check for irregular modes of oscillation passes for both X and Z directions.

Also, it is observed that the maximum mass participation for X direction occurs in mode 5 and for Z direction occurs in mode 4. Hence for the second part of the check the period of these modes needs to be checked for being away from each other by at least 10% of the larger value.

Mode Number MPx (max)	Mode Number MPz (max)	Time Period, x (max)	Time Period, z (max)	% Separation (of the larger value)	Status (FAIL if % Separation < 10)
5	4	0.1303	0.138	5.557	FAIL

Results

Table 504: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Mass Participation Summation of first three lateral translational modes, X direction (%)	99.3	99.292	negligible	
Mass Participation Summation of first three lateral translational modes, Z direction (%)	99.99	99.981	negligible	
Check status – X direction	PASS	PASS	none	
Check status – Z direction	PASS	PASS	none	
Percent separation	5.557	5.5567	negligible	
Check status	FAIL	FAIL	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Irregular Modes of Oscillation.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
```

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 13 PRIS YD 1.4 ZD 1.4
MEMBER PROPERTY
14 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
MEMBER CRACKED CODE IS1893 2016
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
CUT OFF MODE SHAPE 10
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
CHECK IRREGULARITIES CODE IS1893 2016
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None  TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None  TITLE RS_X
SPECTRUM CQC IS1893 2016 X 0.0432 DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN X
   *********************************************************
   * UNITS - KN    METE                                    *
```

```
   * TIME PERIOD FOR X 1893 LOADING =    0.60374 SEC       *
   * SA/G PER 1893=    1.656, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.0716 X     3325.72=     237.97 KN   *
   * VB Act Based on Clause 7.2.1 =      237.97 KN         *
   * VB Min based on Clause 7.2.2 =       79.82 KN         *
   *                                                       *
   *********************************************************
 EIGEN METHOD   : SUBSPACE
 -------------------------
 NUMBER OF MODES REQUESTED             =    10
 NUMBER OF EXISTING MASSES IN THE MODEL =    21
 NUMBER OF MODES THAT WILL BE USED      =    10
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      STAAD SPACE                                              -- PAGE NO.
4
              CALCULATED FREQUENCIES FOR LOAD CASE       2
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       4.044                  0.24728
         2                       4.359                  0.22941
         3                       5.574                  0.17942
         4                       7.246                  0.13800
         5                       7.673                  0.13033
         6                      10.245                  0.09761
         7                      18.398                  0.05435
         8                      20.195                  0.04952
         9                      23.866                  0.04190
        10                      47.193                  0.02119
 1893 RESPONSE SPECTRUM LOAD     2
     RESPONSE LOAD CASE      2
           MODAL WEIGHT (MODAL MASS TIMES g) IN KN          GENERALIZED
      MODE          X             Y             Z              WEIGHT
         1       9.791364E-32  1.252068E-03  9.352446E+02    2.166890E+02
         2       7.325666E+02  9.786925E-34  1.054791E-30    1.793437E+02
         3       2.439665E+00  3.618119E-29  8.592338E-30    3.041611E+02
         4       1.003010E-29  2.685683E-03  1.326169E+03    9.707596E+02
         5       1.690495E+03  5.373340E-29  1.008425E-29    1.296490E+03
         6       8.750866E+00  7.469693E-28  2.317580E-29    1.755958E+03
         7       5.251789E-31  4.299982E-05  6.019501E+02    2.399612E+03
         8       4.205866E+02  1.446007E-31  1.037225E-30    1.848883E+03
         9       8.785461E+00  2.437257E-26  2.600285E-30    3.684693E+03
        10       3.450383E-24  1.617144E+03  2.302894E-01    4.627161E+02
     CQC           MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.863913E+03  2.863913E+03  2.863913E+03 KN
     MISSING WEIGHT X Y Z  -2.886545E-01 -1.246765E+03 -3.191046E-01 KN
       MODAL WEIGHT X Y Z   2.863624E+03  1.617148E+03  2.863594E+03 KN
     RESPONSE LOAD CASE     2
      STAAD SPACE                                              -- PAGE NO.
5
        MODE    SPECTRAL ACCELERATION      DESIGN   SEISMIC   COEFFICIENT
        ----    ---------------------     ------------------------------
                                              X          Y          Z
          1          2.50000                0.1080     0.0000     0.0000
          2          2.50000                0.1080     0.0000     0.0000
          3          2.50000                0.1080     0.0000     0.0000
          4          2.50000                0.1080     0.0000     0.0000
          5          2.50000                0.1080     0.0000     0.0000
          6          2.46419                0.1065     0.0000     0.0000
          7          1.81530                0.0784     0.0000     0.0000
```

```
          8          1.74275                0.0753      0.0000      0.0000
          9          1.62852                0.0704      0.0000      0.0000
         10          1.31784                0.0569      0.0000      0.0000
 PEAK STORY SHEAR
        STORY          LEVEL IN METE                   PEAK STORY SHEAR IN
KN
        -----          -------------
--------------------------------------------------
                                                  X
Y                  Z
          3                 15.00            40.75
0.00               0.00
          2                 10.00           106.55
0.00               0.00
          1                  5.00           203.67
0.00               0.00
        BASE                 0.00           203.67
0.00               0.00
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     2.50000       0.05000
              2                     2.50000       0.05000
              3                     2.50000       0.05000
              4                     2.50000       0.05000
              5                     2.50000       0.05000
              6                     2.46419       0.05000
              7                     1.81530       0.05000
              8                     1.74275       0.05000
              9                     1.62852       0.05000
             10                     1.31784       0.05000
      STAAD SPACE                                            -- PAGE NO.
6
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.247        0.00       -0.00       -0.00       -0.00
0.00        -0.00
      2      0.229       79.12       -0.00        0.00        0.00
183.25      -888.20
      3      0.179        0.26       -0.00       -0.00       -0.00
0.62        -3.38
      4      0.138        0.00        0.00        0.00        0.00
-0.00        0.00
      5      0.130      182.57        0.00       -0.00       -0.00
395.67    -1244.88
      6      0.098        0.93       -0.00        0.00        0.00
2.27        -3.79
      7      0.054        0.00       -0.00        0.00        0.00
-0.00        -0.00
      8      0.050       31.66        0.00        0.00        0.00
93.32       -41.56
      9      0.042        0.62        0.00       -0.00       -0.00
2.01         1.67
     10      0.021        0.00       -0.00        0.00        0.00
```

```
-0.00       -0.00
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z        X        Y        Z
   1    0.00  0.00 32.66    0.000    0.000   32.656      0.00     0.00
0.00
   2   25.58  0.00  0.00   25.579    0.000   32.656     79.12     0.00
0.00
   3    0.09  0.00  0.00   25.664    0.000   32.656      0.26     0.00
0.00
   4    0.00  0.00 46.31   25.664    0.000   78.962      0.00     0.00
0.00
   5   59.03  0.00  0.00   84.692    0.000   78.962    182.57     0.00
0.00
   6    0.31  0.00  0.00   84.997    0.000   78.962      0.93     0.00
0.00
   7    0.00  0.00 21.02   84.997    0.000   99.981      0.00     0.00
0.00
   8   14.69  0.00  0.00   99.683    0.000   99.981     31.66     0.00
0.00
   9    0.31  0.00  0.00   99.990    0.000   99.981      0.62     0.00
0.00
  10    0.00 56.47  0.01   99.990   56.466   99.989      0.00     0.00
0.00

---------------------------
                                  TOTAL SRSS  SHEAR    201.49     0.00
0.00
                                  TOTAL 10PCT SHEAR    201.49     0.00
0.00
                                  TOTAL ABS   SHEAR    295.17     0.00
0.00
                                  TOTAL CSM   SHEAR    293.36     0.00
0.00
                                  TOTAL CQC   SHEAR    203.67     0.00
0.00
   ********************************************************
   * UNITS - KN    METE                                   *
   * TIME PERIOD FOR X 1893 LOADING =    0.60374 SEC      *
   * SA/G PER 1893=    1.656, LOAD FACTOR= 1.000          *
   *      FACTOR V  PER 1893=    0.0716 X   3325.72       *
   * VB Min based on Clause 7.2.2 =       79.82           *
   ********************************************************
      STAAD SPACE                                              -- PAGE NO.
7
   NOTE : THE BASE SHEAR (VB) FROM RESPONSE SPECTRUM IS LESS THAN THE
   BASE SHEAR (Vb) CALCULATED USING EMPIRICAL FORMULA FOR FUNDAMENTAL
   TIME PERIOD. MULTIPLYING FACTOR (Vb/VB) ACCORDING TO 7.7.3a IS      1.1684
      STAAD SPACE                                              -- PAGE NO.
8
 -IRREGULARITY CHECKS
                    STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                    *********************************************************
                         Including Amendment no. 2 November 2020
                    *********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
```

```
Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
---------------------------------------------------------------------
edi : Design Eccentricity
esi : Static Eccentricity
bi  : Floor/Diaphragm plan dimension perpendicular to force direction
For Details Refer Clause 7.8 IS1893:2016-Part-1
---------------------------------------------------------------------
Using edi = 1.5esi + 0.05bi
---------------------------
Displacement of extreme points of diaphragm(dia.) in X dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      7    0.0047      2    0.0043    0.0045    1.0371     PASS
   2     12    0.0215      9    0.0199    0.0207    1.0371     PASS
   3     16    0.3232     13    0.2882    0.3057    1.0572     PASS
Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in X dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      2    0.0046      7    0.0044    0.0045    1.0298     PASS
   2      9    0.0207     12    0.0200    0.0204    1.0167     PASS
   3     13    0.3092     16    0.3000    0.3046    1.0151     PASS
Using edi = 1.5esi + 0.05bi
---------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      2    0.0048      3    0.0045    0.0047    1.0316     PASS
   2      9    0.0294     10    0.0284    0.0289    1.0177     PASS
   3     13    0.3433     14    0.3238    0.3335    1.0292     PASS
Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      3    0.0048      2    0.0045    0.0047    1.0316     PASS
   2     10    0.0294      9    0.0284    0.0289    1.0177     PASS
   3     14    0.3433     13    0.3238    0.3335    1.0292     PASS
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Table 5 (ii)   - Ratio Limit: 0.15 )
-----------------------------------------
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii)   - Ratio Limit: 1.50
---------------------------------------
```

```
 Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
        ( m)      ( kN)      ( kN)      ( kN)     Above  Below
 ---------------------------------------------------------------------
    1     5.000   1847.229    839.971    Base      2.199   N/A    FAIL
    2    10.000    839.971    176.712   1847.229   4.753  0.455   FAIL
    3    15.000    176.712    Top        839.971    N/A   0.210    OK
 --IRREGULAR MODES OF OSCILLATION
      STAAD SPACE                                              -- PAGE NO.
9
 Irregular Modes of Oscillation Checks based on Table 6 (vii)
 ------------------------------------------------------------
 a) Mass Participation in X direction
 --------------------------------------
  Ratio Limit (%):65.0 Status -   PASS
 --------------------------------------
 Mode  Mass Participation
               (%)
 ------------------------
    2       25.579
    5       59.027
    8       14.686
 ------------------------
  SUM       99.292
 a) Mass Participation in Z direction
 --------------------------------------
  Ratio Limit (%):65.0 Status -   PASS
 --------------------------------------
 Mode  Mass Participation
               (%)
 ------------------------
    1       32.656
    4       46.306
    7       21.018
 ------------------------
  SUM       99.981
 b) Fundamental Period - Ratio Limit (%):10.0 Status -   FAIL
 ------------------------------------------------------------
  X Direction    Z Direction
 Mode  Period   Mode   Period   Pmax-Pmin/Pmax
        (s)             (s)          (%)
 ---------------------------------------------
    5   0.1303     4    0.1380      5.5567
    56. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                              -- PAGE NO.
10
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     0.0370    0.0021   -0.0007    0.0000   -0.0000   -0.0001
           2     0.0325    0.0015    0.0020    0.0000    0.0000    0.0001
      3    1     0.0370   -0.0021    0.0007   -0.0000   -0.0000   -0.0001
           2     0.0325    0.0015    0.0020    0.0000    0.0000    0.0001
      4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      5    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
```

```
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     6    1     0.0355    0.0012   -0.0007    0.0000   -0.0000   -0.0001
          2     0.0298    0.0008    0.0020    0.0000    0.0000    0.0001
     7    1     0.0355   -0.0012    0.0007   -0.0000   -0.0000   -0.0001
          2     0.0298    0.0008    0.0020    0.0000    0.0000    0.0001
     8    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     9    1     0.0972    0.0032    0.0040    0.0000    0.0000   -0.0001
          2     0.0743    0.0022    0.0046    0.0000    0.0000    0.0001
    10    1     0.0972   -0.0032   -0.0040   -0.0000    0.0000   -0.0001
          2     0.0743    0.0022    0.0046    0.0000    0.0000    0.0001
    11    1     0.1053    0.0013    0.0040    0.0000    0.0000   -0.0002
          2     0.0766    0.0009    0.0046    0.0000    0.0000    0.0001
    12    1     0.1053   -0.0013   -0.0040   -0.0000    0.0000   -0.0002
          2     0.0766    0.0009    0.0046    0.0000    0.0000    0.0001
    13    1     0.4596    0.0042    0.0101    0.0000    0.0000   -0.0006
          2     0.3049    0.0026    0.0173    0.0000    0.0001    0.0005
    14    1     0.4596   -0.0042   -0.0101   -0.0000    0.0000   -0.0006
          2     0.3049    0.0026    0.0173    0.0000    0.0001    0.0005
    15    1     0.4798    0.0023    0.0101    0.0000    0.0000   -0.0006
          2     0.3352    0.0015    0.0173    0.0000    0.0001    0.0005
    16    1     0.4798   -0.0023   -0.0101   -0.0000    0.0000   -0.0006
          2     0.3352    0.0015    0.0173    0.0000    0.0001    0.0005
    17    1     0.0362    0.0000    0.0000    0.0000   -0.0000    0.0000
          2     0.0311    0.0000    0.0000    0.0000    0.0000    0.0000
    18    1     0.1003    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0751    0.0000    0.0000    0.0000    0.0000    0.0000
    19    1     0.4697    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.3199    0.0000    0.0000    0.0000    0.0001    0.0000
      STAAD SPACE                                             -- PAGE NO.
11
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1    -59.81   -181.53      5.93     11.97      2.93    236.12
          2     62.25    125.56      6.55     17.52      8.06    221.81
     4    1    -59.81    181.53     -5.93    -11.97      2.93    236.12
          2     62.25    125.56      6.55     17.52      8.06    221.81
     5    1    -59.17    -99.19      5.19     10.83      2.93    229.45
          2     56.97     71.63      6.40     17.43      8.06    203.35
     8    1    -59.17     99.19     -5.19    -10.83      2.93    229.45
          2     56.97     71.63      6.40     17.43      8.06    203.35
      STAAD SPACE                                             -- PAGE NO.
12
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER   LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1   -181.53     59.81     5.93      2.93    -11.97     236.12
                2    181.53    -59.81    -5.93     -2.93    -17.67      62.95
          2     1    125.56     62.25     6.55      8.06     17.52     221.81
                2    125.56     62.25     6.55      8.06     15.86      91.71
     2    1     2      0.00    -87.34     0.00      5.88     -0.00    -218.35
                3      0.00     87.34    -0.00     -5.88     -0.00    -218.35
          2     2      0.00     69.30     0.00      6.83      0.00     173.24
                3      0.00     69.30     0.00      6.83      0.00     173.24
     3    1     3    181.53     59.81     5.93      2.93    -17.67      62.95
                4   -181.53    -59.81    -5.93     -2.93    -11.97     236.12
```

```
          2     3    125.56     62.25      6.55      8.06     15.86     91.71
                4    125.56     62.25      6.55      8.06     17.52    221.81
     4    1     2      0.00     -0.68      0.00     -5.22     -0.00     -2.06
                6      0.00      0.68     -0.00      5.22      0.00     -1.35
          2     2      0.00      3.26      0.00      7.37      0.00      8.31
                6      0.00      3.26      0.00      7.37      0.00      8.02
     5    1     3      0.00      0.68      0.00     -5.22     -0.00      2.06
                7      0.00     -0.68     -0.00      5.22      0.00      1.35
          2     3      0.00      3.26      0.00      7.37      0.00      8.31
                7      0.00      3.26      0.00      7.37      0.00      8.02
     6    1     5    -99.19     59.17      5.19      2.93    -10.83    229.45
                6     99.19    -59.17     -5.19     -2.93    -15.11     66.40
          2     5     71.63     56.97      6.40      8.06     17.43    203.35
                6     71.63     56.97      6.40      8.06     14.99     83.56
     7    1     6      0.00    -85.89      0.00      4.41     -0.00   -214.74
                7      0.00     85.89     -0.00     -4.41     -0.00   -214.74
          2     6      0.00     65.88      0.00      5.89      0.00    164.69
                7      0.00     65.88      0.00      5.89      0.00    164.69
     8    1     7     99.19     59.17      5.19      2.93    -15.11     66.40
                8    -99.19    -59.17     -5.19     -2.93    -10.83    229.45
          2     7     71.63     56.97      6.40      8.06     14.99     83.56
                8     71.63     56.97      6.40      8.06     17.43    203.35
     9    1     2    -93.51     64.31     -8.29    -19.12     25.60    160.62
                9     93.51    -64.31      8.29     19.12     15.83    160.96
     STAAD SPACE                                                -- PAGE NO.
13
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2     2     58.40     44.74      5.94     15.04     18.33    101.92
                9     58.40     44.74      5.94     15.04     11.53    124.32
    10    1     3     93.51     64.31      8.29    -19.12    -25.60    160.62
               10    -93.51    -64.31     -8.29     19.12    -15.83    160.96
          2     3     58.40     44.74      5.94     15.04     18.33    101.92
               10     58.40     44.74      5.94     15.04     11.53    124.32
    11    1     6    -13.98     22.34     -4.29    -19.12     20.87    143.11
               11     13.98    -22.34      4.29     19.12      0.60    -31.44
          2     6     10.14     17.76      2.86     15.04     14.71     91.72
               11     10.14     17.76      2.86     15.04      1.22     35.36
    12    1     7     13.98     22.34      4.29    -19.12    -20.87    143.11
               12    -13.98    -22.34     -4.29     19.12     -0.60    -31.44
          2     7     10.14     17.76      2.86     15.04     14.71     91.72
               12     10.14     17.76      2.86     15.04      1.22     35.36
    13    1     9      0.00    -82.38      0.00     15.96      0.00   -205.94
               10      0.00     82.38     -0.00    -15.96      0.00   -205.94
          2     9      0.00     53.17      0.00     12.52      0.00    132.92
               10      0.00     53.17      0.00     12.52      0.00    132.92
    14    1     9      0.00     -0.12     -0.00      1.38      0.00     -0.27
               11      0.00      0.12      0.00     -1.38     -0.00     -0.34
          2     9      0.00      0.12      0.00      0.90      0.00      0.27
               11      0.00      0.12      0.00      0.90      0.00      0.32
    15    1    10      0.00      0.12     -0.00      1.38      0.00      0.27
               12      0.00     -0.12      0.00     -1.38     -0.00      0.34
          2    10      0.00      0.12      0.00      0.90      0.00      0.27
               12      0.00      0.12      0.00      0.90      0.00      0.32
    16    1    11      0.00     -3.16      0.00      0.41     -0.00     -7.91
               12      0.00      3.16      0.00     -0.41     -0.00     -7.91
```

	2	11	0.00	2.04	0.00	0.32	0.00	5.11
		12	0.00	2.04	0.00	0.32	0.00	5.11
17	1	9	-11.01	14.21	-0.16	-0.40	0.40	43.60

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

V. IS 1893 2016 Mass Irregularity

Verify mass irregularity check in accordance to IS 1893 2016 Part 1 specifications.

Details

A three story structure is modelled with floor diaphragms defined at the story levels.

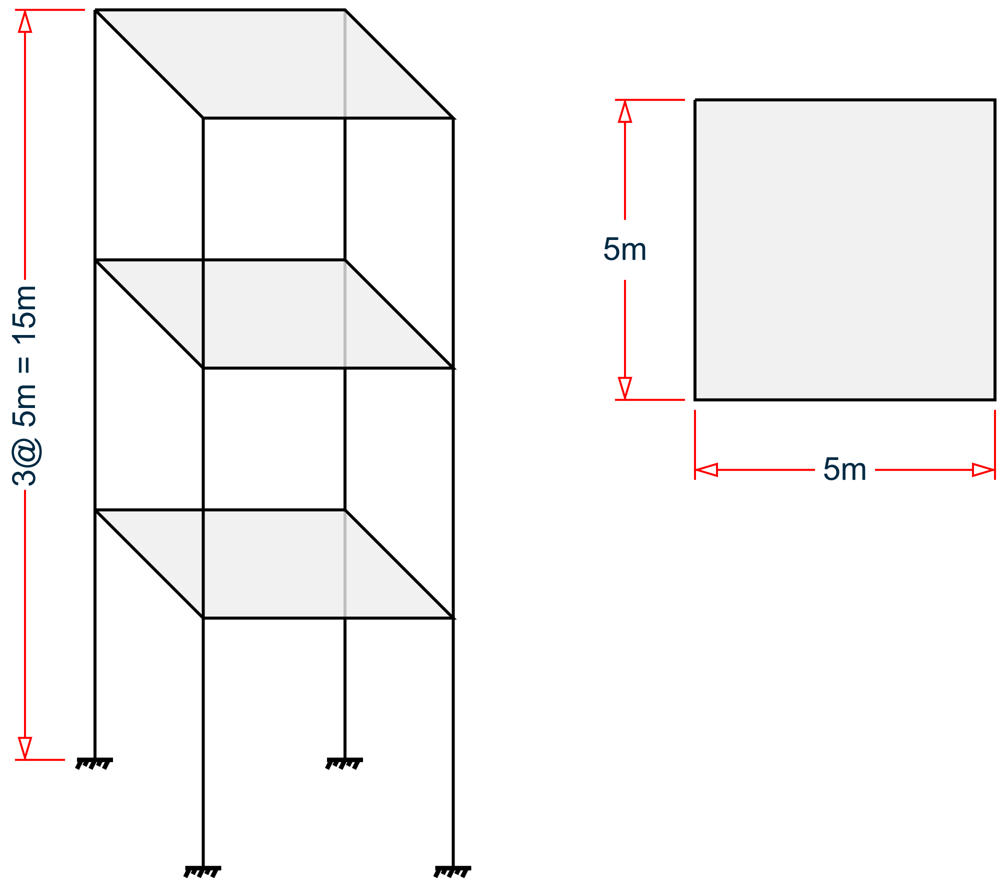


Figure 431: Isometric and Plan View of Structure

Seismic parameters:

Seismic Zone Factor, Z = 0.36
Importance Factor, I = 1.2
Response Reduction Factor, R = 5
Site soil conditions: hard soil (SS 1)

Structure type: other: (ST 5)

Verification

Diaphragm 1 = 1st Floor: Weight = 10 x 4 = 40 kN

Diaphragm 2 = 2nd Floor: Weight = 16 x 4 = 64 kN

Diaphragm 3 = 3rd Floor: Weight = 23 x 4 = 92 kN

Diaphragm	Current Floor Weight (KN)	Above Floor Weight (KN)	Below Floor Weight (KN)	Current Floor Weight/Above Floor Weight	Current Floor Weight/Below Floor Weight	Status (FAIL if ratio > 1.5)
1	40	64	N.A.	0.625	N.A.	OK
2	64	92	40	0.696	1.6	FAIL
3	92	N.A.	64	N.A.	1.4375	OK

Results

Table 505: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Diaphragm 1	Ratio of 1st/2nd floor weights	0.625	0.625	none	
	Check	OK	OK	none	
Diaphragm 2	Ratio of 2nd/3rd floor weights	0.696	0.696	none	
	Check	OK	OK	none	
	Ratio of 2nd/1st floor weights	1.6	1.6	none	
	Check	Fail	Fail	none	
Diaphragm 3	Ratio of 3rd/2nd floor weights	1.438	1.437	negligible	
	Check	OK	OK	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Mass Irregularity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
MEMBER CRACKED CODE IS1893 2016
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 3 6 7 FY -10
9 TO 12 FY -16
13 TO 16 FY -23
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
CHECK IRREGULARITIES CODE IS1893 2016
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None  TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
1893 LOAD Z 1
LOAD 3 LOADTYPE None  TITLE STATIC_Y
```

```
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN X
   *********************************************************
   * UNITS - KN    METE                                     *
   * TIME PERIOD FOR X 1893 LOADING =    0.60374 SEC        *
   * SA/G PER 1893=    1.656, LOAD FACTOR= 1.000            *
   * VB PER 1893=    0.0716 X      196.00=      14.02 KN    *
   * VB Act Based on Clause 7.2.1 =       14.02 KN          *
   * VB Min based on Clause 7.2.2 =        4.70 KN          *
   *                                                        *
   *********************************************************
   **WARNING: BASE DIMENSION IS CALCULATED AT LOWEST SUPPORT
              LEVEL OF THE STRUCTURE. IF THIS LEVEL IS NOT
              PLINTH LEVEL USE PZ PARAMETER FOR TIME PERIOD
              FOR Z 1893 LOADING.
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Z
   *********************************************************
   * UNITS - KN    METE                                     *
   * TIME PERIOD FOR Z 1893 LOADING =    0.60374 SEC        *
   * SA/G PER 1893=    1.656, LOAD FACTOR= 1.000            *
   * VB PER 1893=    0.0716 X      196.00=      14.02 KN    *
   * VB Act Based on Clause 7.2.1 =       14.02 KN          *
   * VB Min based on Clause 7.2.2 =        4.70 KN          *
   *                                                        *
   *********************************************************
      STAAD SPACE                                               -- PAGE NO.
5
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Y
   *********************************************************
   * UNITS - KN    METE                                     *
   * Calculation of SA/G for Y  based on Clause 6.4.6       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000            *
   * VB PER 1893=    0.0720 X      196.00=      14.11 KN    *
   *                                                        *
   *********************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX        0.125    MY        0.000
```

```
      3     FX        0.125    MY        0.000
      6     FX        0.125    MY        0.000
      7     FX        0.125    MY        0.000
                  -----------        -----------
         TOTAL =      0.499              0.000 AT LEVEL     5.000 METE
    VB PER 1893 =    14.025 KN
      9     FX        0.799    MY        0.000
     10     FX        0.799    MY        0.000
     11     FX        0.799    MY        0.000
     12     FX        0.799    MY        0.000
                  -----------        -----------
         TOTAL =      3.194              0.000 AT LEVEL    10.000 METE
    VB PER 1893 =    14.025 KN
     13     FX        2.583    MY        0.000
     14     FX        2.583    MY        0.000
     15     FX        2.583    MY        0.000
     16     FX        2.583    MY        0.000
                  -----------        -----------
         TOTAL =     10.331              0.000 AT LEVEL    15.000 METE
    VB PER 1893 =    14.025 KN
    JOINT            LATERAL          TORSIONAL           LOAD -    2
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      2     FZ        0.125    MY        0.000
      3     FZ        0.125    MY        0.000
      6     FZ        0.125    MY        0.000
      7     FZ        0.125    MY        0.000
                  -----------        -----------
         TOTAL =      0.499              0.000 AT LEVEL     5.000 METE
    VB PER 1893 =    14.025 KN
      9     FZ        0.799    MY        0.000
     10     FZ        0.799    MY        0.000
     STAAD SPACE                                              -- PAGE NO.
6
     11     FZ        0.799    MY        0.000
     12     FZ        0.799    MY        0.000
                  -----------        -----------
         TOTAL =      3.194              0.000 AT LEVEL    10.000 METE
    VB PER 1893 =    14.025 KN
     13     FZ        2.583    MY        0.000
     14     FZ        2.583    MY        0.000
     15     FZ        2.583    MY        0.000
     16     FZ        2.583    MY        0.000
                  -----------        -----------
         TOTAL =     10.331              0.000 AT LEVEL    15.000 METE
    VB PER 1893 =    14.025 KN
    JOINT            LATERAL          TORSIONAL           LOAD -    3
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
      2     FY        0.126    MY        0.000
      3     FY        0.126    MY        0.000
      6     FY        0.126    MY        0.000
      7     FY        0.126    MY        0.000
                  -----------        -----------
         TOTAL =      0.502              0.000 AT LEVEL     5.000 METE
    VB PER 1893 =    14.112 KN
      9     FY        0.804    MY        0.000
     10     FY        0.804    MY        0.000
```

```
     11     FY        0.804    MY        0.000
     12     FY        0.804    MY        0.000
                 -----------       -----------
         TOTAL =      3.214             0.000 AT LEVEL    10.000 METE
   VB PER 1893 =     14.112 KN
     13     FY        2.599    MY        0.000
     14     FY        2.599    MY        0.000
     15     FY        2.599    MY        0.000
     16     FY        2.599    MY        0.000
                 -----------       -----------
         TOTAL =     10.396             0.000 AT LEVEL    15.000 METE
   VB PER 1893 =     14.112 KN
  ************ END OF DATA FROM INTERNAL STORAGE ************
     STAAD SPACE                                           -- PAGE NO.
7
 -IRREGULARITY CHECKS
                    STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                    ********************************************************
                         Including Amendment no. 2 November 2020
                    ********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
 Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
 ---------------------------------------------------------------------
 edi : Design Eccentricity
 esi : Static Eccentricity
 bi  : Floor/Diaphragm plan dimension perpendicular to force direction
 For Details Refer Clause 7.8 IS1893:2016-Part-1
 ---------------------------------------------------------------------
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
    1      7    0.2504      2    0.2336    0.2420    1.0346     PASS
    2     12    0.8737      9    0.8270    0.8503    1.0275     PASS
    3     16    1.6655     13    1.5863    1.6259    1.0244     PASS
 Using edi = esi - 0.05bi
 ------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
    1      2    0.2504      7    0.2336    0.2420    1.0346     PASS
    2      9    0.8737     12    0.8270    0.8503    1.0275     PASS
    3     13    1.6655     16    1.5863    1.6259    1.0244     PASS
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in Z dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
    1      2    0.2504      3    0.2336    0.2420    1.0346     PASS
    2      9    0.8737     10    0.8270    0.8503    1.0275     PASS
```

```
   3     13    1.6655     14    1.5863    1.6259    1.0244     PASS
Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)              (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      3    0.2504      2    0.2336    0.2420    1.0346     PASS
   2     10    0.8737      9    0.8270    0.8503    1.0275     PASS
   3     14    1.6655     13    1.5863    1.6259    1.0244     PASS
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Table 5 (ii)   - Ratio Limit: 0.15 )
------------------------------------------
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii)   - Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass      Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)     ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     5.000     40.000     64.000    Base      0.625   N/A     OK
   2    10.000     64.000     92.000     40.000   0.696  1.600   FAIL
   3    15.000     92.000    Top         64.000    N/A   1.437    OK
  ***NOTE: Linear dynamic analysis needs to carried out for Irregular
  Modes of Oscillation check.
  ***NOTE: Static Seismic Loads for relevant code needs to be defined
  with Zone 4 and 5 for Irregular Modes of Oscillation  check.
   55. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     STAAD SPACE                                               -- PAGE NO.
8
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     2    1     0.1169    0.0013    0.0000    0.0000    0.0000   -0.0003
          2     0.0000    0.0013    0.1169    0.0003    0.0000    0.0000
          3     0.0000    0.0003    0.0000    0.0000    0.0000    0.0000
     3    1     0.1169   -0.0013    0.0000    0.0000    0.0000   -0.0003
          2     0.0000    0.0013    0.1169    0.0003    0.0000    0.0000
          3     0.0000    0.0003    0.0000    0.0000    0.0000    0.0000
     4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     5    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     6    1     0.1169    0.0013    0.0000    0.0000    0.0000   -0.0003
          2     0.0000   -0.0013    0.1169    0.0003    0.0000    0.0000
          3     0.0000    0.0003    0.0000    0.0000    0.0000    0.0000
     7    1     0.1169   -0.0013    0.0000    0.0000    0.0000   -0.0003
```

```
          2     0.0000   -0.0013    0.1169    0.0003    0.0000    0.0000
          3     0.0000    0.0003    0.0000    0.0000    0.0000    0.0000
     8    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     9    1     0.3039    0.0020    0.0000    0.0000    0.0000   -0.0003
          2     0.0000    0.0020    0.3039    0.0003    0.0000    0.0000
          3     0.0000    0.0006    0.0000    0.0000    0.0000    0.0000
    10    1     0.3039   -0.0020    0.0000    0.0000    0.0000   -0.0003
          2     0.0000    0.0020    0.3039    0.0003    0.0000    0.0000
          3     0.0000    0.0006    0.0000    0.0000    0.0000    0.0000
    11    1     0.3039    0.0020    0.0000    0.0000    0.0000   -0.0003
          2     0.0000   -0.0020    0.3039    0.0003    0.0000    0.0000
          3     0.0000    0.0006    0.0000    0.0000    0.0000    0.0000
    12    1     0.3039   -0.0020    0.0000    0.0000    0.0000   -0.0003
          2     0.0000   -0.0020    0.3039    0.0003    0.0000    0.0000
          3     0.0000    0.0006    0.0000    0.0000    0.0000    0.0000
    13    1     0.4558    0.0023    0.0000    0.0000    0.0000   -0.0002
          2     0.0000    0.0023    0.4558    0.0002    0.0000    0.0000
          3     0.0000    0.0009    0.0000    0.0000    0.0000    0.0000
    14    1     0.4558   -0.0023    0.0000    0.0000    0.0000   -0.0002
          2     0.0000    0.0023    0.4558    0.0002    0.0000    0.0000
          3     0.0000    0.0009    0.0000    0.0000    0.0000    0.0000
    15    1     0.4558    0.0023    0.0000    0.0000    0.0000   -0.0002
          2     0.0000   -0.0023    0.4558    0.0002    0.0000    0.0000
          3     0.0000    0.0009    0.0000    0.0000    0.0000    0.0000
    16    1     0.4558   -0.0023    0.0000    0.0000    0.0000   -0.0002
     STAAD SPACE                                             -- PAGE NO.
9
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
          2     0.0000   -0.0023    0.4558    0.0002    0.0000    0.0000
          3     0.0000    0.0009    0.0000    0.0000    0.0000    0.0000
    17    1     0.1169    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.1169    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    18    1     0.3039    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.3039    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    19    1     0.4558    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.4558    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     STAAD SPACE                                             -- PAGE NO.
10
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      -3.51    -13.66      0.00      0.00      0.00     13.19
          2       0.00    -13.66     -3.51    -13.19      0.00      0.00
          3       0.00     -3.53      0.00      0.00      0.00      0.00
     4    1      -3.51     13.66      0.00      0.00      0.00     13.19
          2       0.00    -13.66     -3.51    -13.19      0.00      0.00
          3       0.00     -3.53      0.00      0.00      0.00      0.00
     5    1      -3.51    -13.66      0.00      0.00      0.00     13.19
          2       0.00     13.66     -3.51    -13.19      0.00      0.00
          3       0.00     -3.53      0.00      0.00      0.00      0.00
     8    1      -3.51     13.66      0.00      0.00      0.00     13.19
```

```
           2      0.00     13.66     -3.51    -13.19      0.00      0.00
           3      0.00     -3.53      0.00      0.00      0.00      0.00
      STAAD SPACE                                               -- PAGE NO.
11
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1     1     1    -13.66      3.51     0.00      0.00     -0.00      13.19
                  2     13.66     -3.51    -0.00     -0.00     -0.00       4.34
            2     1    -13.66      0.00    -3.51      0.00     13.19       0.00
                  2     13.66     -0.00     3.51     -0.00      4.34       0.00
            3     1     -3.53     -0.00     0.00     -0.00     -0.00      -0.00
                  2      3.53      0.00    -0.00      0.00     -0.00       0.00
      2     1     2      0.00     -5.17    -0.00     -0.00      0.00     -12.91
                  3      0.00      5.17     0.00      0.00      0.00     -12.91
            2     2      0.00     -0.00     0.00      0.00     -0.00      -0.00
                  3      0.00      0.00    -0.00     -0.00     -0.00      -0.00
            3     2      0.00      0.00    -0.00      0.00     -0.00       0.00
                  3      0.00     -0.00     0.00     -0.00      0.00      -0.00
      3     1     3     13.66      3.51     0.00      0.00     -0.00       4.34
                  4    -13.66     -3.51    -0.00     -0.00     -0.00      13.19
            2     3    -13.66      0.00     3.51      0.00     -4.34       0.00
                  4     13.66     -0.00    -3.51     -0.00    -13.19       0.00
            3     3     -3.53     -0.00    -0.00     -0.00     -0.00       0.00
                  4      3.53      0.00     0.00      0.00      0.00      -0.00
      4     1     2      0.00      0.00     0.00     -0.00     -0.00       0.00
                  6      0.00     -0.00    -0.00      0.00      0.00       0.00
            2     2      0.00     -5.17    -0.00     -0.00      0.00     -12.91
                  6      0.00      5.17     0.00      0.00      0.00     -12.91
            3     2      0.00      0.00    -0.00      0.00      0.00       0.00
                  6      0.00     -0.00     0.00     -0.00      0.00      -0.00
      5     1     3      0.00     -0.00     0.00     -0.00     -0.00      -0.00
                  7      0.00      0.00    -0.00      0.00      0.00      -0.00
            2     3      0.00     -5.17    -0.00     -0.00      0.00     -12.91
                  7      0.00      5.17     0.00      0.00      0.00     -12.91
            3     3      0.00      0.00    -0.00      0.00      0.00       0.00
                  7      0.00     -0.00     0.00     -0.00      0.00       0.00
      6     1     5    -13.66      3.51     0.00      0.00     -0.00      13.19
                  6     13.66     -3.51    -0.00     -0.00     -0.00       4.34
            2     5     13.66     -0.00    -3.51      0.00     13.19      -0.00
                  6    -13.66      0.00     3.51     -0.00      4.34      -0.00
            3     5     -3.53     -0.00    -0.00     -0.00     -0.00      -0.00
                  6      3.53      0.00     0.00      0.00      0.00      -0.00
      7     1     6      0.00     -5.17    -0.00     -0.00      0.00     -12.91
                  7      0.00      5.17     0.00      0.00      0.00     -12.91
      STAAD SPACE                                               -- PAGE NO.
12
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
            2     6      0.00      0.00     0.00     -0.00     -0.00       0.00
                  7      0.00     -0.00    -0.00      0.00     -0.00       0.00
            3     6      0.00      0.00    -0.00     -0.00     -0.00       0.00
                  7      0.00     -0.00     0.00      0.00      0.00       0.00
      8     1     7     13.66      3.51     0.00      0.00     -0.00       4.34
                  8    -13.66     -3.51    -0.00     -0.00     -0.00      13.19
```

```
          2     7     13.66     -0.00      3.51      0.00     -4.34      -0.00
                8    -13.66      0.00     -3.51     -0.00    -13.19      -0.00
          3     7     -3.53     -0.00     -0.00     -0.00     -0.00      -0.00
                8      3.53      0.00      0.00      0.00      0.00      -0.00
     9    1     2     -8.50      3.38     -0.00      0.00      0.00       8.57
                9      8.50     -3.38      0.00     -0.00     -0.00       8.33
          2     2     -8.50     -0.00     -3.38      0.00      8.57       0.00
                9      8.50      0.00      3.38     -0.00      8.33      -0.00
          3     2     -3.40     -0.00     -0.00     -0.00     -0.00      -0.00
                9      3.40      0.00      0.00      0.00      0.00      -0.00
    10    1     3      8.50      3.38      0.00      0.00     -0.00       8.57
               10     -8.50     -3.38     -0.00     -0.00      0.00       8.33
          2     3     -8.50     -0.00     -3.38      0.00      8.57       0.00
               10      8.50      0.00      3.38     -0.00      8.33      -0.00
          3     3     -3.40      0.00      0.00     -0.00     -0.00       0.00
               10      3.40     -0.00     -0.00      0.00      0.00       0.00
    11    1     6     -8.50      3.38     -0.00      0.00      0.00       8.57
               11      8.50     -3.38      0.00     -0.00     -0.00       8.33
          2     6      8.50      0.00     -3.38      0.00      8.57       0.00
               11     -8.50     -0.00      3.38     -0.00      8.33       0.00
          3     6     -3.40     -0.00      0.00     -0.00     -0.00      -0.00
               11      3.40      0.00     -0.00      0.00     -0.00      -0.00
    12    1     7      8.50      3.38      0.00      0.00     -0.00       8.57
               12     -8.50     -3.38     -0.00     -0.00      0.00       8.33
          2     7      8.50      0.00     -3.38      0.00      8.57       0.00
               12     -8.50     -0.00      3.38     -0.00      8.33       0.00
          3     7     -3.40     -0.00     -0.00     -0.00     -0.00      -0.00
               12      3.40      0.00      0.00      0.00      0.00      -0.00
    13    1     9      0.00     -5.25      0.00      0.00      0.00     -13.13
               10      0.00      5.25      0.00     -0.00      0.00     -13.13
          2     9      0.00     -0.00      0.00     -0.00     -0.00      -0.00
               10      0.00      0.00     -0.00      0.00     -0.00      -0.00
          3     9      0.00     -0.00      0.00     -0.00     -0.00      -0.00
               10      0.00      0.00     -0.00      0.00      0.00      -0.00
      STAAD SPACE                                               -- PAGE NO.
13
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    14    1     9      0.00     -0.00      0.00      0.00     -0.00      -0.00
               11      0.00      0.00     -0.00     -0.00     -0.00      -0.00
          2     9      0.00     -5.25     -0.00     -0.00      0.00     -13.13
               11      0.00      5.25      0.00      0.00      0.00     -13.13
          3     9      0.00      0.00     -0.00      0.00      0.00       0.00
               11      0.00     -0.00      0.00     -0.00      0.00       0.00
    15    1    10      0.00      0.00      0.00      0.00     -0.00       0.00
               12      0.00     -0.00     -0.00     -0.00     -0.00       0.00
          2    10      0.00     -5.25     -0.00     -0.00      0.00     -13.13
               12      0.00      5.25      0.00      0.00      0.00     -13.13
          3    10      0.00      0.00     -0.00      0.00      0.00       0.00
               12      0.00     -0.00      0.00     -0.00      0.00       0.00
    16    1    11      0.00     -5.25      0.00      0.00      0.00     -13.13
               12      0.00      5.25      0.00     -0.00      0.00     -13.13
          2    11      0.00      0.00      0.00      0.00     -0.00       0.00
               12      0.00     -0.00     -0.00     -0.00     -0.00       0.00
          3    11      0.00     -0.00      0.00      0.00     -0.00      -0.00
               12      0.00      0.00     -0.00     -0.00      0.00      -0.00
```

```
     17     1      9     -3.25      2.58    -0.00     -0.00      0.00       4.79
                  13      3.25     -2.58     0.00      0.00      0.00       8.12
            2      9     -3.25      0.00    -2.58      0.00      4.79       0.00
                  13      3.25     -0.00     2.58     -0.00      8.12       0.00
            3      9     -2.60      0.00     0.00     -0.00     -0.00      -0.00
                  13      2.60     -0.00    -0.00      0.00     -0.00       0.00
     18     1     10      3.25      2.58     0.00     -0.00     -0.00       4.79
                  14     -3.25     -2.58    -0.00      0.00     -0.00       8.12
            2     10     -3.25      0.00    -2.58      0.00      4.79       0.00
                  14      3.25     -0.00     2.58     -0.00      8.12       0.00
            3     10     -2.60     -0.00     0.00     -0.00     -0.00      -0.00
                  14      2.60      0.00    -0.00      0.00     -0.00       0.00
     19     1     11     -3.25      2.58    -0.00     -0.00      0.00       4.79
                  15      3.25     -2.58     0.00      0.00      0.00       8.12
            2     11      3.25     -0.00    -2.58      0.00      4.79      -0.00
                  15     -3.25      0.00     2.58     -0.00      8.12      -0.00
            3     11     -2.60      0.00     0.00     -0.00     -0.00      -0.00
                  15      2.60     -0.00    -0.00      0.00     -0.00       0.00
     20     1     12      3.25      2.58     0.00     -0.00     -0.00       4.79
                  16     -3.25     -2.58    -0.00      0.00     -0.00       8.12
      STAAD SPACE                                               -- PAGE NO.
14
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE      (LOCAL )
  MEMBER   LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
            2     12      3.25     -0.00    -2.58      0.00      4.79      -0.00
                  16     -3.25      0.00     2.58     -0.00      8.12      -0.00
            3     12     -2.60     -0.00     0.00     -0.00     -0.00      -0.00
                  16      2.60      0.00    -0.00      0.00     -0.00      -0.00
     21     1     13      0.00     -3.25     0.00      0.00      0.00      -8.12
                  14      0.00      3.25     0.00     -0.00      0.00      -8.12
            2     13      0.00     -0.00     0.00     -0.00     -0.00      -0.00
                  14      0.00      0.00    -0.00      0.00     -0.00      -0.00
            3     13      0.00     -0.00     0.00      0.00     -0.00       0.00
                  14      0.00      0.00    -0.00     -0.00     -0.00       0.00
     22     1     13      0.00     -0.00     0.00      0.00     -0.00      -0.00
                  15      0.00      0.00    -0.00     -0.00     -0.00      -0.00
            2     13      0.00     -3.25     0.00     -0.00      0.00      -8.12
                  15      0.00      3.25     0.00      0.00      0.00      -8.12
            3     13      0.00     -0.00     0.00      0.00      0.00      -0.00
                  15      0.00      0.00    -0.00     -0.00      0.00       0.00
```

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

V. IS 1893 2016 Re entrant Corners

Verify reentrant corner (plan irregularity) check in accordance to IS 1893 2016 Part 1 specifications.

Details

A two story structure is modelled with diaphragms at the story levels.

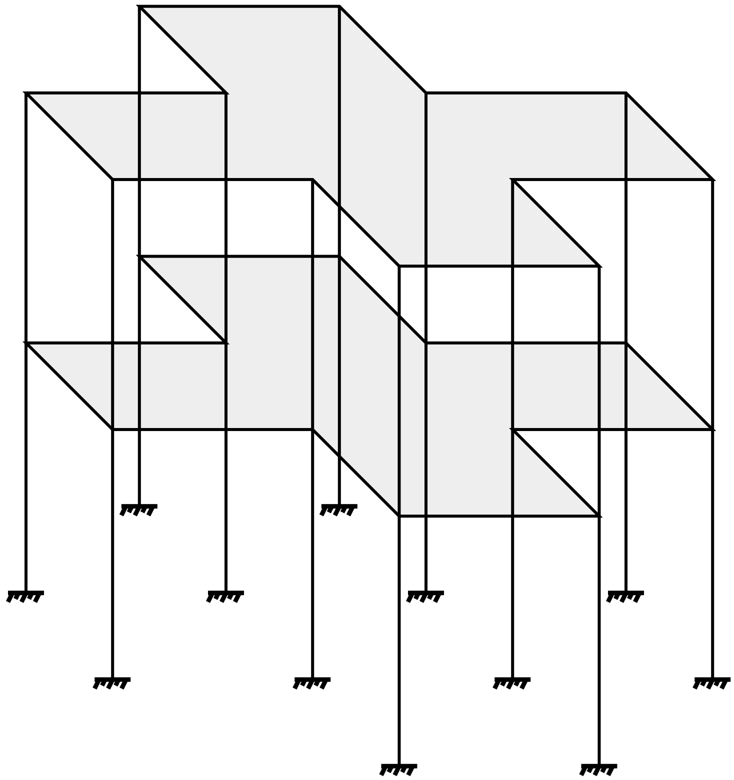


Figure 432: Isometric View

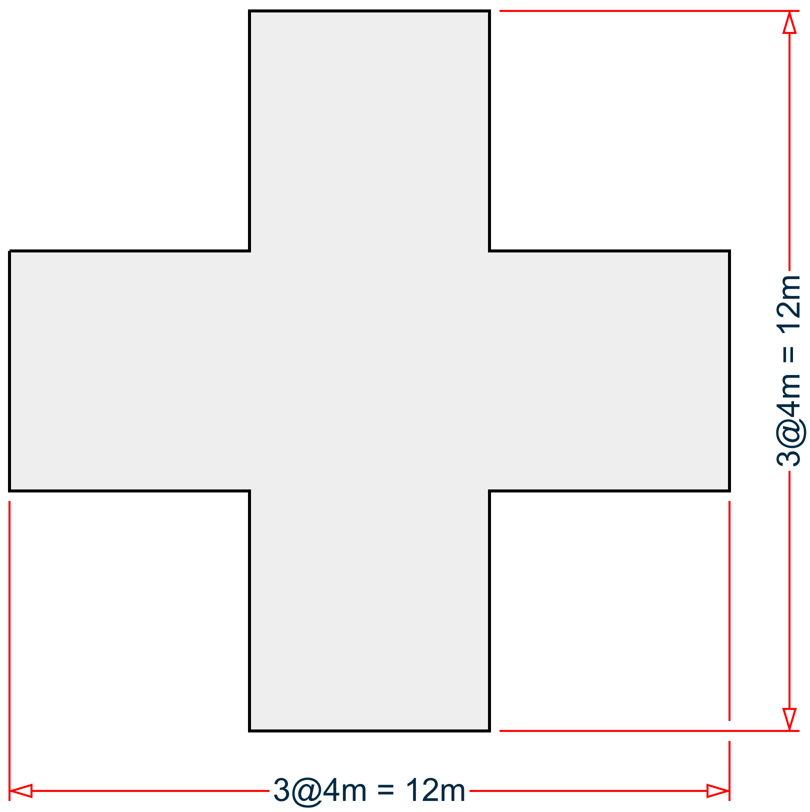


Figure 433: Plan View

Seismic parameters:

Seismic Zone Factor, Z = 0.36
Importance Factor, I = 1.2
Response Reduction Factor, R = 5
Site soil conditions: hard soil (SS 1)
Structure type: other: (ST 5)

Verification

Both diaphragms have the same dimensions and are doubly-symmetric. The overall building width in either direction is 12 m and the potential reentrant corner sides are 4 m. Therefore, the ratio of the $A/L = 4\ m\ /\ 12\ m = 0.333 > 0.15$.

Thus, each of the four corners are reentrant corners per the IS 1893 2016 code.

Results

Table 506: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 1	Reentrant	Reentrant	none	
Node 4	Reentrant	Reentrant	none	
Node 7	Reentrant	Reentrant	none	
Node 10	Reentrant	Reentrant	none	
Node 25	Reentrant	Reentrant	none	
Node 28	Reentrant	Reentrant	none	
Node 31	Reentrant	Reentrant	none	
Node 34	Reentrant	Reentrant	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Re entrant Corners.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 3 0 0; 2 7 0 0; 3 7 0 4; 4 3 0 4; 5 3 0 -4; 6 -1 0 -4; 7 -1 0 0; 8 -5 0 0;
9 -5 0 4; 10 -1 0 4; 11 -1 0 8; 12 3 0 8; 13 3 -5 0; 14 7 -5 0; 15 7 -5 4;
16 3 -5 4; 17 3 -5 -4; 18 -1 -5 -4; 19 -1 -5 0; 20 -5 -5 0; 21 -5 -5 4;
22 -1 -5 4; 23 -1 -5 8; 24 3 -5 8; 25 3 5 0; 26 7 5 0; 27 7 5 4; 28 3 5 4;
29 3 5 -4; 30 -1 5 -4; 31 -1 5 0; 32 -5 5 0; 33 -5 5 4; 34 -1 5 4; 35 -1 5 8;
36 3 5 8;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 1 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10; 10 10 11;
11 11 12; 12 12 4; 13 1 13; 14 2 14; 15 3 15; 16 4 16; 17 5 17; 18 6 18;
19 7 19; 20 8 20; 21 9 21; 22 10 22; 23 11 23; 24 12 24; 25 1 25; 26 2 26;
27 3 27; 28 4 28; 29 5 29; 30 6 30; 31 7 31; 32 8 32; 33 9 33; 34 10 34;
35 11 35; 36 12 36; 37 25 26; 38 26 27; 39 27 28; 40 25 29; 41 29 30; 42 30
31;
43 31 32; 44 32 33; 45 33 34; 46 34 35; 47 35 36; 48 36 28;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
```

```
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 48 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
13 TO 24 FIXED
MEMBER CRACKED CODE IS1893 2016
13 TO 36 REDUCTION RIY 0.7 RIZ 0.7
1 TO 12 37 TO 48 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 0
DIA 2 TYPE RIG HEI 5
CHECK IRREGULARITIES CODE IS1893 2016
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None  TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
1893 LOAD Z 1
LOAD 3 LOADTYPE None  TITLE STATIC_Y
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN X
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR X 1893 LOADING =    0.25981 SEC       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.1080 X     1272.33=     137.41 KN   *
   * VB Act Based on Clause 7.2.1 =      137.41 KN         *
   * VB Min based on Clause 7.2.2 =       30.54 KN         *
   *                                                       *
   *********************************************************
   **WARNING: BASE DIMENSION IS CALCULATED AT LOWEST SUPPORT
              LEVEL OF THE STRUCTURE. IF THIS LEVEL IS NOT
              PLINTH LEVEL USE PZ PARAMETER FOR TIME PERIOD
              FOR Z 1893 LOADING.
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
```

```
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Z
   *******************************************************
   * UNITS - KN    METE                                   *
   * TIME PERIOD FOR Z 1893 LOADING =    0.25981 SEC      *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000          *
   * VB PER 1893=    0.1080 X     1272.33=     137.41 KN   *
   * VB Act Based on Clause 7.2.1 =      137.41 KN        *
   * VB Min based on Clause 7.2.2 =       30.54 KN        *
   *                                                      *
   *******************************************************
      STAAD SPACE                                              -- PAGE NO.
5
   ***NOTE: Equivalent Static Analysis should preferably be performed for
regular
structures with approximate natural time period less than 0.4s and regular
structures with height less than 15m in Seismic Zone II. Ref Cl.6.4.3 and 7.6
 BASE SHEAR AND TIME PERIOD IN Y
   *******************************************************
   * UNITS - KN    METE                                   *
   * Calculation of SA/G for Y  based on Clause 6.4.6     *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000          *
   * VB PER 1893=    0.0720 X     1272.33=      91.61 KN   *
   *                                                      *
   *******************************************************
    JOINT            LATERAL          TORSIONAL           LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       1     FX        2.945    MY        0.000
       2     FX        2.945    MY        0.000
       3     FX        2.945    MY        0.000
       4     FX        2.945    MY        0.000
       5     FX        2.945    MY        0.000
       6     FX        2.945    MY        0.000
       7     FX        2.945    MY        0.000
       8     FX        2.945    MY        0.000
       9     FX        2.945    MY        0.000
      10     FX        2.945    MY        0.000
      11     FX        2.945    MY        0.000
      12     FX        2.945    MY        0.000
                 -----------        -----------
          TOTAL =     35.334              0.000 AT LEVEL     0.000 METE
    VB PER 1893 =    137.411 KN
      25     FX        8.506    MY        0.000
      26     FX        8.506    MY        0.000
      27     FX        8.506    MY        0.000
      28     FX        8.506    MY        0.000
      29     FX        8.506    MY        0.000
      30     FX        8.506    MY        0.000
      31     FX        8.506    MY        0.000
      32     FX        8.506    MY        0.000
      33     FX        8.506    MY        0.000
      34     FX        8.506    MY        0.000
      35     FX        8.506    MY        0.000
      36     FX        8.506    MY        0.000
                 -----------        -----------
          TOTAL =    102.077              0.000 AT LEVEL     5.000 METE
```

```
    VB PER 1893 =    137.411 KN
    JOINT            LATERAL          TORSIONAL            LOAD -    2
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       1     FZ        2.945    MY        0.000
       2     FZ        2.945    MY        0.000
      STAAD SPACE                                             -- PAGE NO.
6
       3     FZ        2.945    MY        0.000
       4     FZ        2.945    MY        0.000
       5     FZ        2.945    MY        0.000
       6     FZ        2.945    MY        0.000
       7     FZ        2.945    MY        0.000
       8     FZ        2.945    MY        0.000
       9     FZ        2.945    MY        0.000
      10     FZ        2.945    MY        0.000
      11     FZ        2.945    MY        0.000
      12     FZ        2.945    MY        0.000
                 -----------        -----------
          TOTAL =     35.334              0.000 AT LEVEL     0.000 METE
    VB PER 1893 =    137.411 KN
      25     FZ        8.506    MY        0.000
      26     FZ        8.506    MY        0.000
      27     FZ        8.506    MY        0.000
      28     FZ        8.506    MY        0.000
      29     FZ        8.506    MY        0.000
      30     FZ        8.506    MY        0.000
      31     FZ        8.506    MY        0.000
      32     FZ        8.506    MY        0.000
      33     FZ        8.506    MY        0.000
      34     FZ        8.506    MY        0.000
      35     FZ        8.506    MY        0.000
      36     FZ        8.506    MY        0.000
                 -----------        -----------
          TOTAL =    102.077              0.000 AT LEVEL     5.000 METE
    VB PER 1893 =    137.411 KN
    JOINT            LATERAL          TORSIONAL            LOAD -    3
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       1     FY        1.963    MY        0.000
       2     FY        1.963    MY        0.000
       3     FY        1.963    MY        0.000
       4     FY        1.963    MY        0.000
       5     FY        1.963    MY        0.000
       6     FY        1.963    MY        0.000
       7     FY        1.963    MY        0.000
       8     FY        1.963    MY        0.000
       9     FY        1.963    MY        0.000
      10     FY        1.963    MY        0.000
      11     FY        1.963    MY        0.000
      12     FY        1.963    MY        0.000
                 -----------        -----------
          TOTAL =     23.556              0.000 AT LEVEL     0.000 METE
    VB PER 1893 =     91.608 KN
      25     FY        5.671    MY        0.000
      26     FY        5.671    MY        0.000
      27     FY        5.671    MY        0.000
      28     FY        5.671    MY        0.000
```

```
      29     FY        5.671    MY        0.000
      STAAD SPACE                                              -- PAGE NO.
7
      30     FY        5.671    MY        0.000
      31     FY        5.671    MY        0.000
      32     FY        5.671    MY        0.000
      33     FY        5.671    MY        0.000
      34     FY        5.671    MY        0.000
      35     FY        5.671    MY        0.000
      36     FY        5.671    MY        0.000
                    -----------        -----------
          TOTAL =     68.051              0.000 AT LEVEL     5.000 METE
    VB PER 1893 =     91.608 KN
   ************ END OF DATA FROM INTERNAL STORAGE ************
      STAAD SPACE                                              -- PAGE NO.
8
 -IRREGULARITY CHECKS
                    STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                    *********************************************************
                         Including Amendment no. 2 November 2020
                    *********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
 Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
 ---------------------------------------------------------------------
 edi : Design Eccentricity
 esi : Static Eccentricity
 bi  : Floor/Diaphragm plan dimension perpendicular to force direction
 For Details Refer Clause 7.8 IS1893:2016-Part-1
 ---------------------------------------------------------------------
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
   1     12    0.0848      6    0.0687    0.0768    1.1049     PASS
   2     36    0.2906     30    0.2403    0.2655    1.0947     PASS
 Using edi = esi - 0.05bi
 ------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
   1      6    0.0848     12    0.0687    0.0768    1.1049     PASS
   2     30    0.2906     36    0.2403    0.2655    1.0947     PASS
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in Z dir.
 ------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
 ------------------------------------------------------------------------
   1      8    0.0848      2    0.0687    0.0768    1.1049     PASS
   2     32    0.2906     26    0.2403    0.2655    1.0947     PASS
 Using edi = esi - 0.05bi
```

```
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
------------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
               (mm)             (mm)      (mm)      Disp.
------------------------------------------------------------------------
   1      2    0.0848      8    0.0687    0.0768    1.1049     PASS
   2     26    0.2906     32    0.2403    0.2655    1.0947     PASS
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Table 5 (ii)   - Ratio Limit: 0.15 )
------------------------------------------
    Node      Re-Entrant X-Proj  X-Proj/Lx Z-Proj  Z-Proj/Lz  Status
 Connectivilty    Node    ( m)             ( m)
----------------------------------------------------------------------
        2->         1      4.0000  0.3333    0.0000  0.0000 Re-Entrant
        5                  0.0000  0.0000    4.0000  0.3333
        6->         7      0.0000  0.0000    4.0000  0.3333 Re-Entrant
        8                  4.0000  0.3333    0.0000  0.0000
        9->        10      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       11                  0.0000  0.0000    4.0000  0.3333
       12->         4      0.0000  0.0000    4.0000  0.3333 Re-Entrant
        3                  4.0000  0.3333    0.0000  0.0000
       26->        25      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       29                  0.0000  0.0000    4.0000  0.3333
       30->        31      0.0000  0.0000    4.0000  0.3333 Re-Entrant
       32                  4.0000  0.3333    0.0000  0.0000
       33->        34      4.0000  0.3333    0.0000  0.0000 Re-Entrant
       35                  0.0000  0.0000    4.0000  0.3333
     STAAD SPACE                                              -- PAGE NO.
9
       36->        28      0.0000  0.0000    4.0000  0.3333 Re-Entrant
       27                  4.0000  0.3333    0.0000  0.0000
                  Diaphragm:    Lx:       Lz:
                               ( m)     ( m)
                  ----------------------------
                       1     12.0000   12.0000
                       2     12.0000   12.0000
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii)   - Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
---------------------------------------------------------------------
   1     0.000    636.163    459.451    Base      1.385   N/A     OK
   2     5.000    459.451    Top        636.163    N/A   0.722    OK
  ***NOTE: Linear dynamic analysis needs to carried out for Irregular
  Modes of Oscillation check.
  ***NOTE: Static Seismic Loads for relevant code needs to be defined
  with Zone 4 and 5 for Irregular Modes of Oscillation  check.
   59. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     STAAD SPACE                                              -- PAGE NO.
10
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
```

```
      1    1     0.3133    0.0027    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000   -0.0027    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      2    1     0.3133   -0.0028    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000    0.0027    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      3    1     0.3133   -0.0028    0.0000    0.0000    0.0000   -0.0006
           2     0.0000   -0.0027    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      4    1     0.3133    0.0027    0.0000    0.0000    0.0000   -0.0006
           2     0.0000    0.0027    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      5    1     0.3133   -0.0027    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000    0.0028    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      6    1     0.3133    0.0027    0.0000    0.0000    0.0000   -0.0006
           2     0.0000    0.0028    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      7    1     0.3133   -0.0027    0.0000    0.0000    0.0000   -0.0006
           2     0.0000   -0.0027    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      8    1     0.3133    0.0028    0.0000    0.0000    0.0000   -0.0006
           2     0.0000    0.0027    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
      9    1     0.3133    0.0028    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000   -0.0027    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
     10    1     0.3133   -0.0027    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000    0.0027    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
     11    1     0.3133    0.0027    0.0000   -0.0000    0.0000   -0.0006
           2     0.0000   -0.0028    0.3133    0.0006    0.0000    0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
     12    1     0.3133   -0.0027    0.0000    0.0000    0.0000   -0.0006
           2     0.0000   -0.0028    0.3133    0.0006    0.0000   -0.0000
           3     0.0000    0.0007    0.0000    0.0000    0.0000    0.0000
     13    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     14    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     15    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     16    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      STAAD SPACE                                                 -- PAGE NO.
11
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     17    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     18    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
```

```
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    19    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    20    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    21    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    22    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    23    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    24    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
    25    1     0.6965    0.0038    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000   -0.0038    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    26    1     0.6965   -0.0039    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000    0.0038    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    27    1     0.6965   -0.0039    0.0000    0.0000    0.0000   -0.0004
          2     0.0000   -0.0038    0.6965    0.0004    0.0000   -0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    28    1     0.6965    0.0038    0.0000    0.0000    0.0000   -0.0004
          2     0.0000    0.0038    0.6965    0.0004    0.0000   -0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    29    1     0.6965   -0.0038    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000    0.0039    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    30    1     0.6965    0.0038    0.0000    0.0000    0.0000   -0.0004
          2     0.0000    0.0039    0.6965    0.0004    0.0000   -0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    31    1     0.6965   -0.0038    0.0000    0.0000    0.0000   -0.0004
          2     0.0000   -0.0038    0.6965    0.0004    0.0000   -0.0000
      STAAD SPACE                                              -- PAGE NO.
12
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    32    1     0.6965    0.0039    0.0000    0.0000    0.0000   -0.0004
          2     0.0000    0.0038    0.6965    0.0004    0.0000   -0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    33    1     0.6965    0.0039    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000   -0.0038    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    34    1     0.6965   -0.0038    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000    0.0038    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    35    1     0.6965    0.0038    0.0000   -0.0000    0.0000   -0.0004
          2     0.0000   -0.0039    0.6965    0.0004    0.0000    0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
    36    1     0.6965   -0.0038    0.0000    0.0000    0.0000   -0.0004
```

```
          2     0.0000   -0.0039    0.6965    0.0004    0.0000   -0.0000
          3     0.0000    0.0012    0.0000    0.0000    0.0000    0.0000
     37   1     0.3133    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.3133    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     38   1     0.6965    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.6965    0.0000    0.0000    0.0000
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      STAAD SPACE                                                 -- PAGE NO.
13
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     13    1    -11.45    -29.84     -0.05     -0.09      0.00     38.75
           2     -0.05     29.84    -11.45    -38.75      0.00      0.09
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     14    1    -11.45     30.52     -0.00     -0.00      0.00     38.75
           2     -0.05    -29.85    -11.45    -38.75      0.00      0.08
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     15    1    -11.45     30.52      0.00      0.00      0.00     38.75
           2      0.05     29.85    -11.45    -38.75      0.00     -0.08
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     16    1    -11.45    -29.84      0.05      0.09      0.00     38.75
           2      0.05    -29.84    -11.45    -38.75      0.00     -0.09
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     17    1    -11.45     29.85     -0.05     -0.08      0.00     38.75
           2     -0.00    -30.52    -11.45    -38.75      0.00      0.00
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     18    1    -11.45    -29.85      0.05      0.08      0.00     38.75
           2      0.00    -30.52    -11.45    -38.75      0.00     -0.00
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     19    1    -11.45     29.84      0.05      0.09      0.00     38.75
           2      0.05     29.84    -11.45    -38.75      0.00     -0.09
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     20    1    -11.45    -30.52      0.00      0.00      0.00     38.75
           2      0.05    -29.85    -11.45    -38.75      0.00     -0.08
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     21    1    -11.45    -30.52     -0.00     -0.00      0.00     38.75
           2     -0.05     29.85    -11.45    -38.75      0.00      0.08
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     22    1    -11.45     29.84     -0.05     -0.09      0.00     38.75
           2     -0.05    -29.84    -11.45    -38.75      0.00      0.09
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     23    1    -11.45    -29.85     -0.05     -0.08      0.00     38.75
           2     -0.00     30.52    -11.45    -38.75      0.00      0.00
           3      0.00     -7.63      0.00      0.00      0.00      0.00
     24    1    -11.45     29.85      0.05      0.08      0.00     38.75
           2      0.00     30.52    -11.45    -38.75      0.00     -0.00
           3      0.00     -7.63      0.00      0.00      0.00      0.00
      STAAD SPACE                                                 -- PAGE NO.
14
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00    -18.29     0.00     -0.06     -0.00     -36.58
                 2      0.00     18.29    -0.00      0.06     -0.00     -36.58
           2     1      0.00     -0.32     0.00      0.00     -0.00      -0.64
```

```
                2      0.00      0.32    -0.00     -0.00     -0.00     -0.64
           3    1      0.00      0.00    -0.00      0.00      0.00      0.00
                2      0.00     -0.00     0.00     -0.00     -0.00      0.00
      2    1    2      0.00      0.00    -0.00      0.00      0.00      0.00
                3      0.00     -0.00     0.00     -0.00      0.00     -0.00
           2    2      0.00    -18.29    -0.00      0.11      0.00    -36.58
                3      0.00     18.29     0.00     -0.11      0.00    -36.58
           3    2      0.00     -0.00     0.00     -0.00      0.00     -0.00
                3      0.00      0.00    -0.00      0.00     -0.00     -0.00
      3    1    3      0.00     18.29     0.00      0.06     -0.00     36.58
                4      0.00    -18.29    -0.00     -0.06     -0.00     36.58
           2    3      0.00     -0.32     0.00      0.00     -0.00     -0.64
                4      0.00      0.32    -0.00     -0.00     -0.00     -0.64
           3    3      0.00      0.00     0.00      0.00      0.00      0.00
                4      0.00     -0.00    -0.00     -0.00     -0.00      0.00
      4    1    1      0.00      0.32    -0.00      0.00      0.00      0.64
                5      0.00     -0.32     0.00     -0.00      0.00      0.64
           2    1      0.00     18.29     0.00     -0.06      0.00     36.58
                5      0.00    -18.29     0.00      0.06      0.00     36.58
           3    1      0.00     -0.00     0.00      0.00     -0.00     -0.00
                5      0.00      0.00    -0.00      0.00     -0.00     -0.00
      5    1    5      0.00     18.29     0.00      0.11     -0.00     36.58
                6      0.00    -18.29    -0.00     -0.11     -0.00     36.58
           2    5      0.00      0.00     0.00     -0.00     -0.00     -0.00
                6      0.00     -0.00    -0.00      0.00     -0.00      0.00
           3    5      0.00      0.00     0.00     -0.00     -0.00      0.00
                6      0.00     -0.00     0.00      0.00     -0.00      0.00
      6    1    6      0.00      0.32    -0.00      0.00      0.00      0.64
                7      0.00     -0.32     0.00     -0.00      0.00      0.64
           2    6      0.00    -18.29    -0.00      0.06      0.00    -36.58
                7      0.00     18.29     0.00     -0.06      0.00    -36.58
           3    6      0.00     -0.00     0.00      0.00     -0.00     -0.00
                7      0.00      0.00    -0.00     -0.00     -0.00     -0.00
      7    1    7      0.00     18.29     0.00     -0.06     -0.00     36.58
                8      0.00    -18.29    -0.00      0.06     -0.00     36.58
      STAAD SPACE                                            -- PAGE NO.
15
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    7      0.00     -0.32     0.00     -0.00     -0.00     -0.64
                8      0.00      0.32    -0.00      0.00     -0.00     -0.64
           3    7      0.00     -0.00     0.00      0.00     -0.00     -0.00
                8      0.00      0.00    -0.00     -0.00     -0.00     -0.00
      8    1    8      0.00     -0.00    -0.00     -0.00      0.00     -0.00
                9      0.00      0.00     0.00      0.00      0.00      0.00
           2    8      0.00    -18.29    -0.00     -0.11      0.00    -36.58
                9      0.00     18.29     0.00      0.11      0.00    -36.58
           3    8      0.00      0.00     0.00     -0.00      0.00      0.00
                9      0.00     -0.00    -0.00      0.00     -0.00      0.00
      9    1    9      0.00    -18.29     0.00      0.06     -0.00    -36.58
               10      0.00     18.29    -0.00     -0.06     -0.00    -36.58
           2    9      0.00     -0.32     0.00     -0.00     -0.00     -0.64
               10      0.00      0.32    -0.00      0.00     -0.00     -0.64
           3    9      0.00      0.00     0.00      0.00     -0.00      0.00
               10      0.00      0.00    -0.00     -0.00     -0.00      0.00
     10    1   10      0.00     -0.32    -0.00     -0.00      0.00     -0.64
```

```
                11       0.00       0.32       0.00       0.00       0.00      -0.64
          2     10       0.00     -18.29      -0.00       0.06      -0.00     -36.58
                11       0.00      18.29       0.00      -0.06       0.00     -36.58
          3     10       0.00      -0.00       0.00       0.00      -0.00      -0.00
                11       0.00       0.00       0.00      -0.00       0.00      -0.00
     11   1     11       0.00     -18.29       0.00      -0.11      -0.00     -36.58
                12       0.00      18.29      -0.00       0.11      -0.00     -36.58
          2     11       0.00       0.00       0.00       0.00      -0.00       0.00
                12       0.00      -0.00      -0.00       0.00      -0.00      -0.00
          3     11       0.00       0.00       0.00      -0.00      -0.00       0.00
                12       0.00      -0.00      -0.00       0.00      -0.00       0.00
     12   1     12       0.00      -0.32      -0.00      -0.00       0.00      -0.64
                 4       0.00       0.32       0.00       0.00       0.00      -0.64
          2     12       0.00      18.29      -0.00      -0.06      -0.00      36.58
                 4       0.00     -18.29       0.00       0.06       0.00      36.58
          3     12       0.00      -0.00       0.00       0.00      -0.00      -0.00
                 4       0.00       0.00       0.00      -0.00      -0.00      -0.00
     13   1      1     -29.84      11.45       0.05      -0.00      -0.17      18.51
                13      29.84     -11.45      -0.05       0.00      -0.09      38.75
          2      1      29.84       0.05      11.45       0.00     -18.51       0.17
                13     -29.84      -0.05     -11.45      -0.00     -38.75       0.09
          3      1      -7.63      -0.00      -0.00      -0.00       0.00       0.00
                13       7.63       0.00       0.00       0.00       0.00      -0.00
      STAAD SPACE                                                 -- PAGE NO.
16
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     14   1      2      30.52      11.45       0.00      -0.00      -0.00      18.51
                14     -30.52     -11.45      -0.00       0.00      -0.00      38.75
          2      2     -29.85       0.05      11.45       0.00     -18.51       0.17
                14      29.85      -0.05     -11.45      -0.00     -38.75       0.08
          3      2      -7.63       0.00       0.00      -0.00      -0.00       0.00
                14       7.63      -0.00      -0.00       0.00      -0.00      -0.00
     15   1      3      30.52      11.45      -0.00      -0.00       0.00      18.51
                15     -30.52     -11.45       0.00       0.00       0.00      38.75
          2      3      29.85      -0.05      11.45       0.00     -18.51      -0.17
                15     -29.85       0.05     -11.45      -0.00     -38.75      -0.08
          3      3      -7.63       0.00       0.00      -0.00      -0.00       0.00
                15       7.63      -0.00      -0.00       0.00       0.00       0.00
     16   1      4     -29.84      11.45      -0.05      -0.00       0.17      18.51
                16      29.84     -11.45       0.05       0.00       0.09      38.75
          2      4     -29.84      -0.05      11.45       0.00     -18.51      -0.17
                16      29.84       0.05     -11.45      -0.00     -38.75      -0.09
          3      4      -7.63       0.00      -0.00      -0.00       0.00       0.00
                16       7.63      -0.00       0.00       0.00       0.00       0.00
     17   1      5      29.85      11.45       0.05      -0.00      -0.17      18.51
                17     -29.85     -11.45      -0.05       0.00      -0.08      38.75
          2      5     -30.52       0.00      11.45       0.00     -18.51       0.00
                17      30.52      -0.00     -11.45      -0.00     -38.75       0.00
          3      5      -7.63      -0.00      -0.00      -0.00       0.00      -0.00
                17       7.63       0.00       0.00       0.00       0.00      -0.00
     18   1      6     -29.85      11.45      -0.05      -0.00       0.17      18.51
                18      29.85     -11.45       0.05       0.00       0.08      38.75
          2      6     -30.52      -0.00      11.45       0.00     -18.51      -0.00
                18      30.52       0.00     -11.45      -0.00     -38.75      -0.00
          3      6      -7.63       0.00       0.00      -0.00      -0.00       0.00
```

```
             18       7.63     -0.00     -0.00      0.00     -0.00     -0.00
    19    1   7      29.84     11.45     -0.05     -0.00      0.17     18.51
             19     -29.84    -11.45      0.05      0.00      0.09     38.75
          2   7      29.84     -0.05     11.45      0.00    -18.51     -0.17
             19     -29.84      0.05    -11.45     -0.00    -38.75     -0.09
          3   7      -7.63     -0.00     -0.00     -0.00      0.00     -0.00
             19       7.63      0.00      0.00      0.00      0.00     -0.00
    20    1   8     -30.52     11.45     -0.00     -0.00      0.00     18.51
             20      30.52    -11.45      0.00      0.00      0.00     38.75
     STAAD SPACE                                              -- PAGE NO.
17
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2   8     -29.85     -0.05     11.45      0.00    -18.51     -0.17
             20      29.85      0.05    -11.45     -0.00    -38.75     -0.08
          3   8      -7.63     -0.00     -0.00     -0.00      0.00     -0.00
             20       7.63      0.00      0.00      0.00      0.00     -0.00
    21    1   9     -30.52     11.45      0.00     -0.00     -0.00     18.51
             21      30.52    -11.45     -0.00      0.00     -0.00     38.75
          2   9      29.85      0.05     11.45      0.00    -18.51      0.17
             21     -29.85     -0.05    -11.45     -0.00    -38.75      0.08
          3   9      -7.63     -0.00     -0.00     -0.00      0.00      0.00
             21       7.63      0.00      0.00      0.00      0.00     -0.00
    22    1  10      29.84     11.45      0.05     -0.00     -0.17     18.51
             22     -29.84    -11.45     -0.05      0.00     -0.09     38.75
          2  10     -29.84      0.05     11.45      0.00    -18.51      0.17
             22      29.84     -0.05    -11.45     -0.00    -38.75      0.09
          3  10      -7.63     -0.00      0.00     -0.00     -0.00     -0.00
             22       7.63      0.00     -0.00      0.00     -0.00     -0.00
    23    1  11     -29.85     11.45      0.05     -0.00     -0.17     18.51
             23      29.85    -11.45     -0.05      0.00     -0.08     38.75
          2  11      30.52      0.00     11.45      0.00    -18.51      0.00
             23     -30.52     -0.00    -11.45     -0.00    -38.75      0.00
          3  11      -7.63     -0.00      0.00     -0.00     -0.00     -0.00
             23       7.63      0.00     -0.00      0.00      0.00     -0.00
    24    1  12      29.85     11.45     -0.05     -0.00      0.17     18.51
             24     -29.85    -11.45      0.05      0.00      0.08     38.75
          2  12      30.52     -0.00     11.45      0.00    -18.51     -0.00
             24     -30.52      0.00    -11.45     -0.00    -38.75     -0.00
          3  12      -7.63     -0.00     -0.00     -0.00      0.00     -0.00
             24       7.63      0.00      0.00      0.00      0.00     -0.00
    25    1   1     -11.87      8.51     -0.20     -0.00      0.41     18.07
             25      11.87     -8.51      0.20      0.00      0.57     24.46
          2   1      11.87      0.20     -8.51      0.00     18.07      0.41
             25     -11.87     -0.20      8.51     -0.00     24.46      0.57
          3   1      -5.67     -0.00      0.00     -0.00     -0.00     -0.00
             25       5.67      0.00     -0.00      0.00     -0.00     -0.00
    26    1   2      12.23      8.51     -0.03     -0.00      0.05     18.07
             26     -12.23     -8.51      0.03      0.00      0.10     24.46
          2   2     -11.88      0.17     -8.51      0.00     18.08      0.36
             26      11.88     -0.17      8.51     -0.00     24.46      0.47
          3   2      -5.67     -0.00      0.00     -0.00      0.00      0.00
             26       5.67      0.00     -0.00      0.00     -0.00     -0.00
     STAAD SPACE                                              -- PAGE NO.
18
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
```

```
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     27    1     3      12.23      8.51     0.03     -0.00     -0.05      18.07
                27     -12.23     -8.51    -0.03      0.00     -0.10      24.46
           2     3      11.88     -0.17    -8.51      0.00     18.08      -0.36
                27     -11.88      0.17     8.51     -0.00     24.46      -0.47
           3     3      -5.67      0.00    -0.00     -0.00      0.00       0.00
                27       5.67     -0.00     0.00      0.00      0.00       0.00
     28    1     4     -11.87      8.51     0.20     -0.00     -0.41      18.07
                28      11.87     -8.51    -0.20      0.00     -0.57      24.46
           2     4     -11.87     -0.20    -8.51      0.00     18.07      -0.41
                28      11.87      0.20     8.51     -0.00     24.46      -0.57
           3     4      -5.67      0.00    -0.00     -0.00      0.00       0.00
                28       5.67     -0.00     0.00      0.00      0.00       0.00
     29    1     5      11.88      8.51    -0.17     -0.00      0.36      18.08
                29     -11.88     -8.51     0.17      0.00      0.47      24.46
           2     5     -12.23      0.03    -8.51      0.00     18.07       0.05
                29      12.23     -0.03     8.51     -0.00     24.46       0.10
           3     5      -5.67      0.00     0.00     -0.00     -0.00       0.00
                29       5.67     -0.00    -0.00      0.00     -0.00       0.00
     30    1     6     -11.88      8.51     0.17     -0.00     -0.36      18.08
                30      11.88     -8.51    -0.17      0.00     -0.47      24.46
           2     6     -12.23     -0.03    -8.51      0.00     18.07      -0.05
                30      12.23      0.03     8.51     -0.00     24.46      -0.10
           3     6      -5.67      0.00    -0.00     -0.00      0.00       0.00
                30       5.67     -0.00     0.00      0.00      0.00       0.00
     31    1     7      11.87      8.51     0.20     -0.00     -0.41      18.07
                31     -11.87     -8.51    -0.20      0.00     -0.57      24.46
           2     7      11.87     -0.20    -8.51      0.00     18.07      -0.41
                31     -11.87      0.20     8.51     -0.00     24.46      -0.57
           3     7      -5.67     -0.00     0.00     -0.00     -0.00      -0.00
                31       5.67      0.00    -0.00      0.00     -0.00      -0.00
     32    1     8     -12.23      8.51     0.03     -0.00     -0.05      18.07
                32      12.23     -8.51    -0.03      0.00     -0.10      24.46
           2     8     -11.88     -0.17    -8.51      0.00     18.08      -0.36
                32      11.88      0.17     8.51     -0.00     24.46      -0.47
           3     8      -5.67     -0.00     0.00     -0.00     -0.00      -0.00
                32       5.67      0.00    -0.00      0.00      0.00      -0.00
     33    1     9     -12.23      8.51    -0.03     -0.00      0.05      18.07
                33      12.23     -8.51     0.03      0.00      0.10      24.46
      STAAD SPACE                                                -- PAGE NO.
19
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2     9      11.88      0.17    -8.51      0.00     18.08       0.36
                33     -11.88     -0.17     8.51     -0.00     24.46       0.47
           3     9      -5.67     -0.00     0.00     -0.00     -0.00       0.00
                33       5.67      0.00    -0.00      0.00     -0.00      -0.00
     34    1    10      11.87      8.51    -0.20     -0.00      0.41      18.07
                34     -11.87     -8.51     0.20      0.00      0.57      24.46
           2    10     -11.87      0.20    -8.51      0.00     18.07       0.41
                34      11.87     -0.20     8.51     -0.00     24.46       0.57
           3    10      -5.67      0.00    -0.00     -0.00      0.00       0.00
                34       5.67     -0.00     0.00      0.00      0.00       0.00
     35    1    11     -11.88      8.51    -0.17     -0.00      0.36      18.08
```

```
              35    11.88     -8.51     0.17      0.00      0.47      24.46
         2    11    12.23      0.03    -8.51      0.00     18.07       0.05
              35   -12.23     -0.03     8.51     -0.00     24.46       0.10
         3    11    -5.67     -0.00    -0.00     -0.00      0.00      -0.00
              35     5.67      0.00     0.00      0.00      0.00      -0.00
    36   1    12    11.88      8.51     0.17     -0.00     -0.36      18.08
              36   -11.88     -8.51    -0.17      0.00     -0.47      24.46
         2    12    12.23     -0.03    -8.51      0.00     18.07      -0.05
              36   -12.23      0.03     8.51     -0.00     24.46      -0.10
         3    12    -5.67     -0.00     0.00     -0.00     -0.00      -0.00
              36     5.67      0.00    -0.00      0.00     -0.00      -0.00
    37   1    25     0.00    -12.23     0.00     -0.13     -0.00     -24.46
              26     0.00     12.23     0.00      0.13     -0.00     -24.46
         2    25     0.00     -0.36     0.00      0.00     -0.00      -0.70
              26     0.00      0.36    -0.00     -0.00     -0.00      -0.73
         3    25     0.00      0.00    -0.00      0.00     -0.00       0.00
              26     0.00     -0.00     0.00     -0.00      0.00       0.00
    38   1    26     0.00      0.00    -0.00     -0.00      0.00       0.03
              27     0.00     -0.00     0.00      0.00      0.00      -0.03
         2    26     0.00    -12.23    -0.00      0.26      0.00     -24.47
              27     0.00     12.23     0.00     -0.26      0.00     -24.47
         3    26     0.00     -0.00     0.00     -0.00     -0.00      -0.00
              27     0.00      0.00     0.00      0.00     -0.00      -0.00
    39   1    27     0.00     12.23     0.00      0.13     -0.00      24.46
              28     0.00    -12.23    -0.00     -0.13     -0.00      24.46
         2    27     0.00     -0.36     0.00      0.00     -0.00      -0.73
              28     0.00      0.36    -0.00     -0.00     -0.00      -0.70
         3    27     0.00      0.00    -0.00      0.00     -0.00       0.00
              28     0.00     -0.00     0.00     -0.00      0.00       0.00
      STAAD SPACE                                            -- PAGE NO.
20
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    40   1    25     0.00      0.36    -0.00      0.00      0.00       0.70
              29     0.00     -0.36     0.00     -0.00      0.00       0.73
         2    25     0.00     12.23    -0.00     -0.13      0.00      24.46
              29     0.00    -12.23     0.00      0.13      0.00      24.46
         3    25     0.00     -0.00    -0.00      0.00     -0.00      -0.00
              29     0.00      0.00     0.00     -0.00      0.00      -0.00
    41   1    29     0.00     12.23     0.00      0.26     -0.00      24.47
              30     0.00    -12.23    -0.00     -0.26     -0.00      24.47
         2    29     0.00      0.00     0.00     -0.00     -0.00      -0.03
              30     0.00     -0.00    -0.00      0.00     -0.00       0.03
         3    29     0.00      0.00    -0.00     -0.00     -0.00       0.00
              30     0.00     -0.00     0.00      0.00     -0.00       0.00
    42   1    30     0.00      0.36    -0.00      0.00      0.00       0.73
              31     0.00     -0.36     0.00     -0.00      0.00       0.70
         2    30     0.00    -12.23    -0.00      0.13      0.00     -24.46
              31     0.00     12.23     0.00     -0.13      0.00     -24.46
         3    30     0.00      0.00    -0.00      0.00     -0.00       0.00
              31     0.00     -0.00     0.00     -0.00      0.00       0.00
    43   1    31     0.00     12.23     0.00     -0.13     -0.00      24.46
              32     0.00    -12.23    -0.00      0.13     -0.00      24.46
         2    31     0.00     -0.36     0.00     -0.00     -0.00      -0.70
              32     0.00      0.36    -0.00      0.00     -0.00      -0.73
         3    31     0.00     -0.00     0.00     -0.00     -0.00      -0.00
```

```
                 32       0.00      0.00      0.00      0.00     -0.00     -0.00
     44     1    32       0.00     -0.00     -0.00     -0.00      0.00     -0.03
                 33       0.00      0.00      0.00      0.00      0.00      0.03
            2    32       0.00    -12.23     -0.00     -0.26      0.00    -24.47
                 33       0.00     12.23      0.00      0.26      0.00    -24.47
            3    32       0.00      0.00      0.00     -0.00     -0.00      0.00
                 33       0.00     -0.00      0.00      0.00     -0.00      0.00
     45     1    33       0.00    -12.23      0.00      0.13     -0.00    -24.46
                 34       0.00     12.23     -0.00     -0.13     -0.00    -24.46
            2    33       0.00     -0.36      0.00     -0.00     -0.00     -0.73
                 34       0.00      0.36     -0.00      0.00     -0.00     -0.70
            3    33       0.00     -0.00      0.00      0.00     -0.00     -0.00
                 34       0.00      0.00      0.00     -0.00     -0.00     -0.00
     46     1    34       0.00     -0.36     -0.00     -0.00      0.00     -0.70
                 35       0.00      0.36      0.00      0.00      0.00     -0.73
      STAAD SPACE                                                -- PAGE NO.
21
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE      (LOCAL )
  MEMBER   LOAD   JT     AXIAL    SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
            2    34       0.00    -12.23      0.00      0.13     -0.00    -24.46
                 35       0.00     12.23     -0.00     -0.13      0.00    -24.46
            3    34       0.00     -0.00      0.00      0.00      0.00     -0.00
```

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

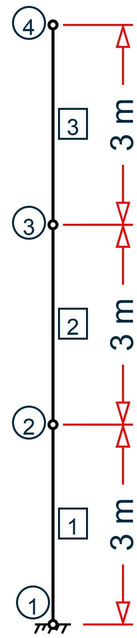
V. IS 1893 2016 Response Spectrum Vertical

Calculation of base shear for vertical motion of oscillation by response spectrum method in IS 1893 : 2016 (Part 1).

Details

The structure is modelled in STAAD.Pro considering the following:

Seismic zone factor, Z = 0.36
Importance factor, I = 1.2
Response reduction factor, R = 5
Soil site conditions = Hard



The dimension of the columns is 0.2m x 0.1 m and material is concrete

The generated model is subjected to a response spectrum load along global Y direction.

Natural frequencies / time period and mode shapes are obtained from the STAAD.Pro output file.

Table 507: Natural frequency / time period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	9.913	0.10088
2	27.083	0.03692
3	36.996	0.02703

Table 508: Mode shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joint 4	0	1	0	0	0	0
	2nd Floor/ Joint 3	0	0.86603	0	0	0	0
	1st Floor/ Joint 2	0	0.5	0	0	0	0
2	Roof/Joint 4	0	-1	0	0	0	0

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
	2nd Floor/ Joint 3	0	0	0	0	0	0
	1st Floor/ Joint 2	0	1	0	0	0	0
3	Roof/Joint 4	0	1	0	0	0	0
	2nd Floor/ Joint 3	0	-0.86603	0	0	0	0
	1st Floor/ Joint 2	0	0.5	0	0	0	0

Validation

Story Level	Weight Wi (kN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.8660	84.932	73.553	0	0	0	-0.8660	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.175	-	183.002	147.106	-	49.035	147.105	-	13.138	147.106
Modal Weight $Mk \times g$		227.7			16.35			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor $Pk \sum Wi\ x\ \phi\ /\ \sum Wi\ x\ \phi 2$		1.244			0.333			0.089		

Table 509: Design vertical acceleration spectrum value calculation

	Mode 1	Mode 2	Mode 3
Time period(sec)	0.1009	0.0369	0.0270
Sa/g	2.5	2.5	2.5
$Ak\ (2/3 \times Z/2 \times I/R \times Sa/g)$	0.072	0.072	0.072

Table 510: Base shear calculation

Story Level	Weight Wi (kN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi	Vi	ϕ	Qi	Vi	ϕQi	Vi	ϕ	
Roof	49.035	1	4.392	4.392	-1	-1.177	-1.177	1	0.315	0.315	4.56
2nd Floor	98.07	0.8660	7.607	12.0	0	0	-1.177	-0.8660	-0.546	-0.231	12.06
1st Floor	98.07	0.5	4.392	16.39	1	2.354	1.177	0.5	0.315	0.084	16.43
										Base shear	16.43

Results

Table 511: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Base shear (kN)	16.43	16.43	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Response Spectrum Vertical.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
```

```
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 3 0 0; 2 3 3 0; 3 3 6 0; 4 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 2 3 PRIS YD 0.2 ZD 0.1
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
4 FY 49.035
2 3 FY 98.07
END DEFINE REFERENCE LOADS
LOAD 1 LOADTYPE NONE  TITLE RS_Y
SPECTRUM SRSS IS1893 2016 Y 0.0432 DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                       9.913                 0.10088
         2                      27.083                 0.03692
         3                      36.996                 0.02703
 1893 RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE            X             Y             Z              WEIGHT
         1       0.000000E+00  2.276565E+02  0.000000E+00    1.471050E+02
         2       0.000000E+00  1.634500E+01  0.000000E+00    1.471050E+02
         3       0.000000E+00  1.173519E+00  0.000000E+00    1.471050E+02
     SRSS           MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   0.000000E+00  2.451750E+02  0.000000E+00 KN
     MISSING WEIGHT X Y Z   0.000000E+00 -3.177129E-06  0.000000E+00 KN
       MODAL WEIGHT X Y Z   0.000000E+00  2.451750E+02  0.000000E+00 KN
   ***NOTE: IT IS RECOMMENDED TO DEFINE FLOOR LEVELS USING "FLOOR HEIGHT"
```

```
INPUT.
     RESPONSE LOAD CASE     1
        MODE    SPECTRAL ACCELERATION       DESIGN   SEISMIC   COEFFICIENT
        ----    ---------------------     ------------------------------
                                              X           Y           Z
          1           2.50000               0.0000      0.0720      0.0000
          2           2.50000               0.0000      0.0720      0.0000
          3           2.50000               0.0000      0.0720      0.0000
      STAAD SPACE                                                -- PAGE NO.
5
 PEAK STORY SHEAR
        STORY           LEVEL IN METE                     PEAK STORY SHEAR IN
KN
        -----           -------------
--------------------------------------------------
                                                  X
Y                  Z
        BASE                  0.00               0.00
16.43              0.00
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
           DIRECTIONAL VALUES:                 SCALE FACTOR =    1.00
    X =  0.00   Y =  0.04   Z =  0.00          DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           0.1009         24.5166
           0.0369         24.5166
           0.0270         24.5166
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     2.50000       0.05000
              2                     2.50000       0.05000
              3                     2.50000       0.05000
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.101        0.00       16.39        0.00        0.00
0.00       49.17
      2      0.037        0.00        1.18        0.00        0.00
0.00        3.53
      3      0.027        0.00        0.08        0.00        0.00
0.00        0.25
      STAAD SPACE                                                -- PAGE NO.
6
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KN
           --------------------------------------         ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1    0.00 92.85  0.00    0.000   92.855    0.000      0.00    16.39
0.00
   2    0.00  6.67  0.00    0.000   99.521    0.000      0.00     1.18
0.00
   3    0.00  0.48  0.00    0.000  100.000    0.000      0.00     0.08
0.00
```

```
---------------------------
                                  TOTAL SRSS  SHEAR      0.00    16.43
0.00
                                  TOTAL 10PCT SHEAR      0.00    16.43
0.00
                                  TOTAL ABS   SHEAR      0.00    17.65
0.00
                                  TOTAL CSM   SHEAR      0.00    16.43
0.00
   ** NOTE: IS-1893:2016 PARAMETERS ARE MISSING.
   DESIGN BASE SHEAR FROM RESPONSE SPECTRUM METHOD CANNOT BE COMPARED WITH
   BASE SHEAR CALCULATED USING EMPIRICAL FORMULA FOR FUNDAMENTAL PERIOD.
   ************ END OF DATA FROM INTERNAL STORAGE ************
    39. PRINT MODE SHAPES
  MODE     SHAPES
      STAAD SPACE                                              -- PAGE NO.
7
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     1    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     1    0.00000   0.50000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     1    0.00000   0.86603   0.00000  0.000E+00  0.000E+00  0.000E+00
     4     1    0.00000   1.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     2    0.00000   1.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     4     2    0.00000  -1.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     3    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     3    0.00000   0.50000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     3    0.00000  -0.86603   0.00000  0.000E+00  0.000E+00  0.000E+00
     4     3    0.00000   1.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
    40. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                              -- PAGE NO.
8
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     0.0000    0.0114    0.0000    0.0000    0.0000    0.0000
      3    1     0.0000    0.0196    0.0000    0.0000    0.0000    0.0000
      4    1     0.0000    0.0227    0.0000    0.0000    0.0000    0.0000
      STAAD SPACE                                              -- PAGE NO.
9
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      0.00     16.43      0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
10
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
```

```
 -----------------
 ALL UNITS ARE -- KN   METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1     16.43     0.00     0.00      0.00      0.00       0.00
               2     16.43     0.00     0.00      0.00      0.00       0.00
    2    1     2     12.06     0.00     0.00      0.00      0.00       0.00
               3     12.06     0.00     0.00      0.00      0.00       0.00
    3    1     3      4.56     0.00     0.00      0.00      0.00       0.00
               4      4.56     0.00     0.00      0.00      0.00       0.00
 ************** END OF LATEST ANALYSIS RESULT **************
```

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

V. IS 1893 2016 Torsion Irregularity

Verify part 1 of the torsional irregularity check per the IS 1893 2016 Part 1 specifications.

Details

A three story building is modelled with floor diaphragms defined at each floor level.

A 1 kip (4.4482 kN) load is applied to each diaphragm in both the global X and Z directions. In STAAD.Pro, this is modelled using an analytical node in each diaphragm. This is used to obtain the displacements in the nodes of the diaphragm.

Nodes 17, 18, and 19 are the control nodes in each diaphragm.

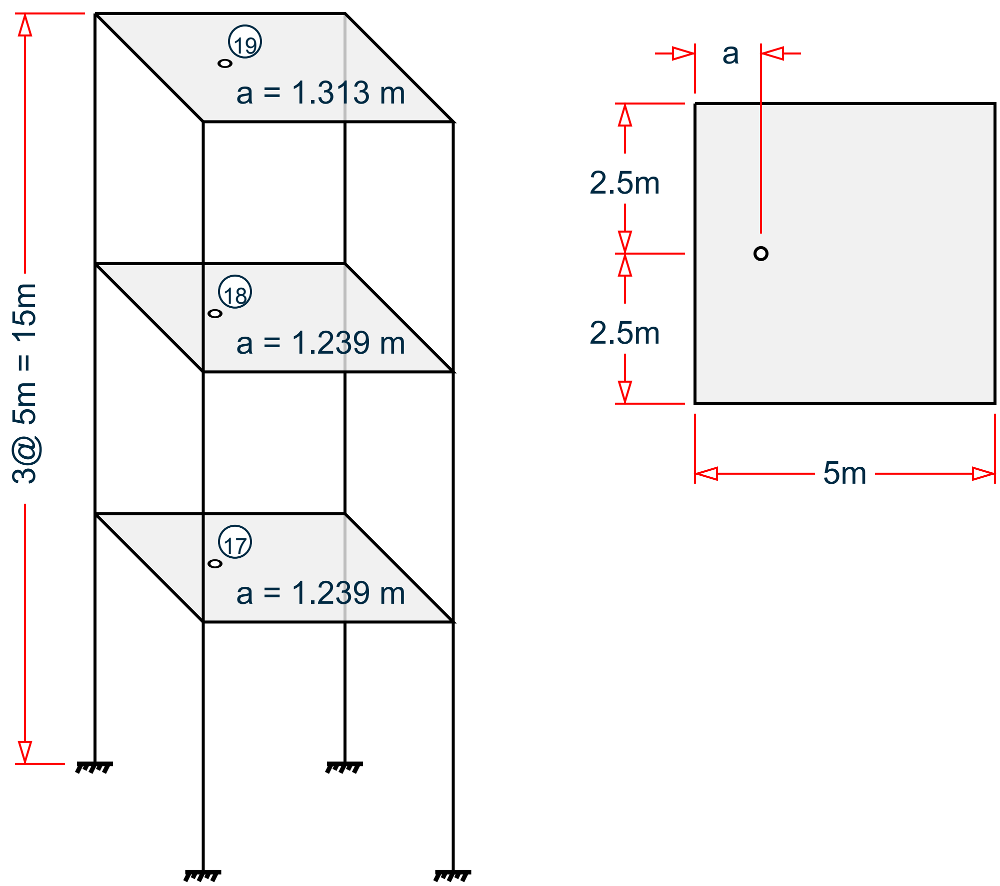


Figure 434: Structure diagram

Validation

As per Cl. 7.8.2, the design eccentricity, e_{di}, is computed as the a constant times the dynamic eccentricity plus or minus the accidental eccentricity:

$$e_{di} = \begin{vmatrix} 1.5e_{si} + 0.05b_i & \text{(Case 1)} \\ 1.0e_{si} - 0.05b_i & \text{(Case 2)} \end{vmatrix}$$

where

e_{si}	=	the static eccentricity at floor i, which is the distance between the center of rigidity and the center of mass at that level
b_i	=	the floor plan dimension at floor i, perpendicular to the direction of force $= 5\ m$ for this example in both direction at all floors

Also, to perform a torsion irregularity check unit load should be applied at edi considering design eccentricity. Hence, in addition to unit load $F = 1\ kip\ (4.45\ kN)$, a moment $M = F \times (lever\ arm)$ is also applied at the center of mass.

The displacements induced because of the applied loads on the control nodes for the nodes situated on the extremities of each diaphragm is taken from the STAAD.Pro output for each lateral direction.

X direction

By inspection, the center of rigidity (as reported by STAAD.Pro) and the center of mass are located at the same point at each floor. Therefore, $e_{si} = 0$.

Table 512: X Direction: Case 1 (ACC = +0.05, DEC = 1.5)

Diaphragm	Control Node	e_{si} (m) (C.M. - C.R.)	Dynamic Eccentricity ($1.5e_{si}$)	Accidental Eccentricity ($0.05b_i$)	Design Eccentricity ($e_{di} = 1.5e_{si} + 0.05b_i$)	F_x Applied at C.M.	M_y Applied at CM ($F_x \times (e_{di} - e_{si})$)
1	17	0	0	0.25	0.25	4.45	1.11
2	18	0	0	0.25	0.25	4.45	1.11
3	19	0	0	0.25	0.25	4.45	1.11

Table 513: X Direction: Case 2 (ACC = -0.05, DEC = 1.0)

Diaphragm	Control Node	e_{si} (m) (C.M. - C.R.)	Dynamic Eccentricity ($1.0e_{si}$)	Accidental Eccentricity ($-0.05b_i$)	Design Eccentricity ($e_{di} = 1.0e_{si} - 0.05 b_i$)	F_x Applied at C.M.	M_y Applied at CM ($F_x \times (e_{di} - e_{si})$)
1	17	0	0	-0.25	-0.25	4.45	-1.11
2	18	0	0	-0.25	-0.25	4.45	-1.11
3	19	0	0	-0.25	-0.25	4.45	-1.11

The displacements induced because of the applied loads centers of mass for the nodes situated on the extremities of each diaphragm are taken from the STAAD.Pro output for each lateral direction.

Table 514: X Direction: Case 1 Checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)			Δmax/ Δavg	Status
			Extreme Node 1	Extreme Node 2	Δx Extreme Node 1	Δx Extreme Node 2	Δx Average		
1	17	1	2 or 3	6 or 7	0.466	0.498	0.482	1.033	PASS
2	18	2	9 or 10	11 or 12	2.399	2.487	2.443	1.018	PASS
3	19	3	13 or 14	15 or 16	6.016	6.164	6.090	1.012	PASS

Table 515: X Direction: Case 2 Checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)			∆max/ ∆avg	Status
			Extreme Node 1	Extreme Node 2	∆x Extreme Node 1	∆x Extreme Node 2	∆x Average		
1	17	4	2 or 3	6 or 7	0.498	0.466	0.482	1.033	PASS
2	18	5	9 or 10	11 or 12	2.487	2.399	2.443	1.018	PASS
3	19	6	13 or 14	15 or 16	6.164	6.016	6.090	1.012	PASS

Z direction checks

Table 516: Z Direction: Case 1 (ACC = +0.05, DEC = 1.5)

Diaphragm	Control Node	C.R. (from output file)	e_{si} (m) (C.M. - C.R.)	Dynamic Eccentricity ($1.5e_{si}$)	Accidental Eccentricity ($0.05b_i$)	Design Eccentricity ($e_{di} = 1.5e_{si} +0.05b_i$)	F_x Applied at C.M.	M_y Applied at CM (F_x×(e_{di}-e_{si})
1	17	0.247	1.239 - 0.247 = 0.992	1.488	0.25	1.738	4.45	3.32
2	18	0.319	1.239 - 0.319 = 0.92	1.380	0.25	1.630	4.45	3.16
3	19	0.372	1.313 - 0.372 = 0.941	1.412	0.25	1.662	4.45	3.20

Table 517: Z Direction: Case 2 (ACC = -0.05, DEC = 1.0)

Diaphragm	Control Node	C.R. (from output file)	e_{si} (m) (C.M. - C.R.)	Dynamic Eccentricity ($1.0e_{si}$)	Accidental Eccentricity ($-0.05b_i$)	Design Eccentricity ($e_{di} = 1.0e_{si}-0.05b_i$)	F_x Applied at C.M.	M_y Applied at CM (F_x×(e_{di}-e_{si})
1	17	0.247	1.239 - 0.247 = 0.992	0.992	-0.25	0.742	4.45	-1.11

Diaphragm	Control Node	C.R. (from output file)	e_{si} (m) (C.M. - C.R.)	Dynamic Eccentricity ($1.0e_{si}$)	Accidental Eccentricity ($-0.05b_i$)	Design Eccentricity ($e_{di} = 1.0e_{si}-0.05b_i$)	F_x Applied at C.M.	M_y Applied at CM ($F_x \times (e_{di}-e_{si})$)
2	18	0.319	1.239 - 0.319 = 0.92	0.920	-0.25	0.670	4.45	-1.11
3	19	0.372	1.313 - 0.372 = 0.941	0.941	-0.25	0.691	4.45	-1.11

Table 518: Z Direction: Case 1 Checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)			∆max/∆avg	Status
			Extreme Node 1	Extreme Node 2	∆z Extreme Node 1	∆z Extreme Node 2	∆z average		
1	17	7	2 or 6	3 or 7	0.238	0.27	0.254	1.063	PASS
2	18	8	9 or 11	10 or 12	0.935	1.009	0.972	1.038	PASS
3	19	9	13 or 15	14 or 16	1.916	2.046	1.981	1.033	PASS

Table 519: Z Direction: Case 2 Checks

Diaphragm	Control Node	Primary LC Number	Diaphragm Extremities		Displacements (mm)			∆max/∆avg	Status
			Extreme Node 1	Extreme Node 2	∆z Extreme Node 1	∆z Extreme Node 2	∆z average		
1	17	10	2 or 6	3 or 7	0.232	0.392	0.312	1.256	WARNING
2	18	11	9 or 11	10 or 12	0.913	1.325	1.119	1.184	PASS
3	19	12	13 or 15	14 or 16	1.873	2.578	2.226	1.158	PASS

Results

Table 520: Comparison of results (max/min displacement ratios and check results)

Result Type	Floor	Reference	STAAD.Pro	Difference	Comments
X Direction; Case 1	1	PASS (1.033)	PASS (1.0333)	negligible	
	2	PASS (1.018)	PASS (1.0180)	negligible	
	3	PASS (1.012)	PASS (1.0122)	negligible	
X Direction; Case 2	1	PASS (1.033)	PASS (1.0333)	negligible	
	2	PASS (1.018)	PASS (1.0180)	negligible	
	3	PASS (1.012)	PASS (1.0122)	negligible	
Z Direction; Case 1	1	PASS (1.063)	PASS (1.0622)	negligible	
	2	PASS (1.038)	PASS (1.0381)	negligible	
	3	PASS (1.033)	PASS (1.0329)	negligible	
Z Direction; Case 2	1	WARNING (1.256)	WARNING (1.2562)	negligible	
	2	PASS (1.184)	PASS (1.1841)	negligible	
	3	PASS (1.158)	PASS (1.1584)	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2016 Torsion Irregularity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5; 17 1.239 5 2.5; 18 1.239 10 2.5; 19 1.313 15 2.5
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
```

```
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
2 3 5 7 8 10 12 13 15 16 18 20 21 23 24 PRIS YD 0.35 ZD 0.25
MEMBER PROPERTY AMERICAN
1 4 6 9 11 14 17 19 22 PRIS YD 0.5 ZD 0.65
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
*MEMBER CRACKED CODE IS1893 2016
*1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
*2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
MEMBER CRACKED
1 3 6 8 TO 12 17 TO 20 REDUCTION RIY 0.7 RIZ 0.7
2 4 5 7 13 TO 16 21 TO 24 REDUCTION RIY 0.35 RIZ 0.35
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 5
DIA 2 TYPE RIG HEI 10
DIA 3 TYPE RIG HEI 15
CHECK IRREGULARITIES CODE IS1893 2016
DEFINE IS1893 2016 LOAD
ZONE 0.36 RF 5 I 1.2 SS 1 ST 5 DM 0.05
LOAD 1 LOADTYPE None  TITLE Diaphragm 1 Case 1 X Dir
JOINT LOAD
17 FX 4.4482 MY 1.11205
LOAD 2 LOADTYPE None  TITLE Diaphragm 2 Case 1 X Dir
JOINT LOAD
18 FX 4.4482 MY 1.11205
LOAD 3 LOADTYPE None  TITLE Diaphragm 3 Case 1 X Dir
JOINT LOAD
19 FX 4.4482 MY 1.11205
LOAD 4 LOADTYPE None  TITLE Diaphragm 1 Case 2 X Dir
JOINT LOAD
17 FX 4.4482 MY -1.11205
LOAD 5 LOADTYPE None  TITLE Diaphragm 2 Case 2 X Dir
JOINT LOAD
18 FX 4.4482 MY -1.11205
LOAD 6 LOADTYPE None  TITLE Diaphragm 3 Case 2 X Dir
JOINT LOAD
19 FX 4.4482 MY -1.11205
LOAD 7 LOADTYPE None  TITLE Diaphragm 1 Case 1 Z Dir
JOINT LOAD
17 FZ 4.4482 MY 3.3183572
LOAD 8 LOADTYPE None  TITLE Diaphragm 2 Case 1 Z Dir
JOINT LOAD
18 FZ 4.4482 MY 3.158222
LOAD 9 LOADTYPE None  TITLE Diaphragm 3 Case 1 Z Dir
```

```
JOINT LOAD
19 FZ 4.4482 MY 3.2049281
LOAD 10 LOADTYPE None  TITLE Diaphragm 1 Case 2 Z Dir
JOINT LOAD
17 FZ 4.4482 MY -1.11205
LOAD 11 LOADTYPE None  TITLE Diaphragm 2 Case 2 Z Dir
JOINT LOAD
18 FZ 4.4482 MY -1.11205
LOAD 12 LOADTYPE None  TITLE Diaphragm 3 Case 2 Z Dir
JOINT LOAD
19 FZ 4.4482 MY -1.11205
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PRINT DIA CR
FINISH
```

STAAD.Pro Output

```
 -IRREGULARITY CHECKS
                      STAAD.PRO IRREGULARITIES CHECK - (  IS1893-2016  )   v1.2
                      *********************************************************
                           Including Amendment no. 2 November 2020
                      *********************************************************
 --TORSION IRREGULARITY CHECKS
 Torsion Irregularity Check
 Ref: Table 5 (i)    - Ratio Limit(s): Lower-1.20 Upper-1.40
 ----------------------------------------------------------------------
 edi : Design Eccentricity
 esi : Static Eccentricity
 bi  : Floor/Diaphragm plan dimension perpendicular to force direction
 For Details Refer Clause 7.8 IS1893:2016-Part-1
 ----------------------------------------------------------------------
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 -------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)              (mm)      (mm)      Disp.
 -------------------------------------------------------------------------
    1      7    0.4984      2    0.4662    0.4823    1.0333     PASS
    2     12    2.4871      9    2.3990    2.4430    1.0180     PASS
    3     16    6.1637     13    6.0157    6.0897    1.0122     PASS
 Using edi = esi - 0.05bi
 ------------------------
 Displacement of extreme points of diaphragm(dia.) in X dir.
 -------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)              (mm)      (mm)      Disp.
 -------------------------------------------------------------------------
    1      2    0.4984      7    0.4662    0.4823    1.0333     PASS
    2      9    2.4871     12    2.3990    2.4430    1.0180     PASS
    3     13    6.1637     16    6.0157    6.0897    1.0122     PASS
 Using edi = 1.5esi + 0.05bi
 ---------------------------
 Displacement of extreme points of diaphragm(dia.) in Z dir.
 -------------------------------------------------------------------------
  Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
```

```
                (mm)             (mm)      (mm)      Disp.
-----------------------------------------------------------------------
   1      3    0.2699      2    0.2383    0.2541    1.0622     PASS
   2     10    1.0087      9    0.9347    0.9717    1.0381     PASS
   3     14    2.0463     13    1.9160    1.9812    1.0329     PASS
Using edi = esi - 0.05bi
------------------------
Displacement of extreme points of diaphragm(dia.) in Z dir.
-----------------------------------------------------------------------
 Dia.  Node Max. Disp.  Node Min. Disp. Avg. Disp. Max./Avg.  Status
                (mm)             (mm)      (mm)      Disp.
-----------------------------------------------------------------------
   1      3    0.3916      2    0.2319    0.3118    1.2562   WARNING*
   2     10    1.3253      9    0.9131    1.1192    1.1841     PASS
   3     14    2.5782     13    1.8732    2.2257    1.1584     PASS
  *** WARNING: The floor is irregular.  Please ensure conformance
   with Cl. 7.1, Table 5, Sl No. (i) sec-i.a or sec-i.b.
--GEOMETRY IRREGULARITY CHECKS
Re-Entrant Corner Check.
(Ref: Table 5 (ii)   - Ratio Limit: 0.15 )
------------------------------------------
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
  ***NOTE: No Irregular Re-Entrant Nodes found in the diaphragm.
--MASS IRREGULARITY CHECKS
Mass Irregularity Check
Ref: Table 6 (ii)   - Ratio Limit: 1.50
---------------------------------------
Dia.   Level     Mass       Above      Below     Ratio  Ratio  Status
       ( m)      ( kN)      ( kN)      ( kN)     Above  Below
--------------------------------------------------------------------
   1     5.000    166.404    166.404    Base      1.000   N/A     OK
   2    10.000    166.404    117.808    166.404   1.412  1.000    OK
   3    15.000    117.808    Top        166.404    N/A   0.708    OK
  ***NOTE: Linear dynamic analysis needs to carried out for Irregular
  Modes of Oscillation check.
  ***NOTE: Static Seismic Loads for relevant code needs to be defined
  with Zone 4 and 5 for Irregular Modes of Oscillation  check.
```

Related Links

- *TR.28.2.2 Check Irregularities* (on page 2333)

V. IS 1893 2015 Response Spectrum

Calculate the base shear and its distribution along the height for the response spectrum method in the IS 1893 (Part4): 2015 specification.

Details

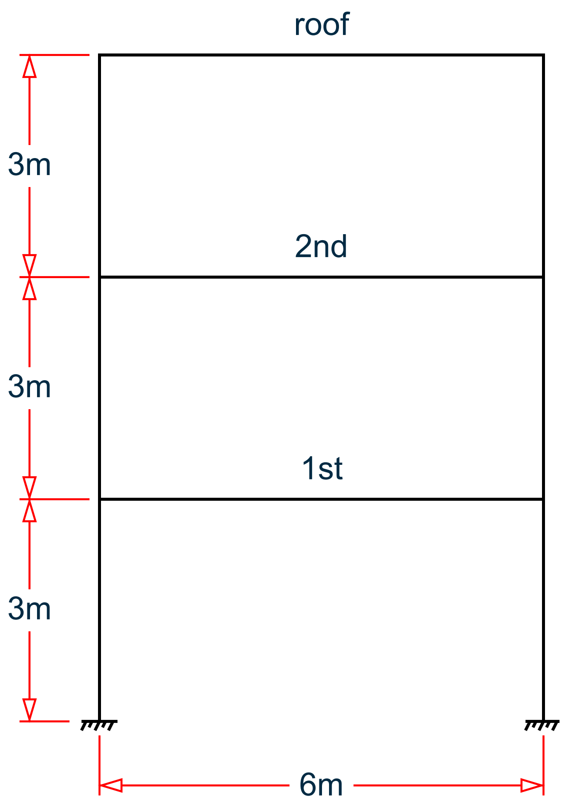


Figure 435: A three story frame

Beam and column dimensions: 500 mm × 500 mm

Material: concrete

The structure is modelled in STAAD.Pro considering the following:

- Seismic zone factor, $Z = 0.16$ (`ZONE 0.16`)
- Importance factor, $I = 1.5$ (`I 1.5`)
- Response reduction factor, $R = 5$ (`RF 5`)
- Soil site condition = Soft (`SOIL TYPE 3`)
- Minimum factor of total weight for base shear (`RSMIN 2.75`)

Validation

Table 521: Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Mode	Frequency (cyc/sec)	Time Period (sec)
1	0.229	4.3681
2	0.625	1.59884

Mode	Frequency (cyc/sec)	Time Period (sec)
3	0.854	1.17043

Table 522: Mode shapes

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
1	2nd Floor/Joints 5 & 6	0.866	0	0	0	0	0
1	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
2	2nd Floor/Joints 5 & 6	0	0	0	0	0	0
2	1st Floor/Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
3	2nd Floor/Joints 5 & 6	-0.866	0	0	0	0	0
3	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0

Story Level	Weight, W_i (kN)	Mode 1			Mode 2			Mode 3		
		ϕ	$W_i \times \phi$	$W_i \times \phi^2$	ϕ	$W_i \times \phi$	$W_i \times \phi^2$	ϕ	$W_i \times \phi$	$W_i \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.866	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.18		183.00	147.11		49.035	147.105		13.138	147.11

Table 523: Mass participation factor and mod participation factor calculations

	Mode 1	Mode 2	Mode 3
Modal Weight, $Mk \times g$	$\frac{183.00^2}{147.11} = 227.7$	$\frac{49.035^2}{147.11} = 16.35$	$\frac{13.138^2}{147.11} = 1.173$
Modal Weight Participation (In %)	$\frac{227.7}{245.18} \times 100 = 92.85$	$\frac{16.35}{245.18} \times 100 = 6.666$	$\frac{1.173}{245.18} \times 100 = 0.479$
Mode Participation Factor, Pk	$\frac{183.00}{147.11} = 1.244$	$\frac{49.035}{147.11} = 0.333$	$\frac{13.138}{147.11} = 0.089$

Direction factor = $\frac{Z}{2} \times \frac{I}{R} = \frac{0.16}{2} \times \frac{1.5}{5} = 0.024$

	Mode 1	Mode 2	Mode 3
Sa/g	$\frac{1.67}{T} = \frac{1.67}{4.368} = 0.382$	$\frac{1.67}{T} = \frac{1.67}{1.599} = 1.045$	$\frac{1.67}{T} = \frac{1.67}{1.170} = 1.427$
$Ak = \frac{Z}{2} \times \frac{I}{R} \times \frac{Sa}{g}$	$0.024 \times 0.382 = 0.00918$	$0.024 \times 1.045 = 0.0251$	$0.024 \times 1.427 = 0.0342$

Story Level	Weight, W_i (kN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes (i.e., SRSS of all modes)
		ϕ	$Qi = (Ak \times \phi \times Pk \times Wi)$	$Vi = (\sum Qi)$	ϕ	$Qi = (Ak \times \phi \times Pk \times Wi)$	$Vi = (\sum Qi)$	ϕ	$Qi = (Ak \times \phi \times Pk \times Wi)$	$Vi = (\sum Qi)$	
Roof	49.035	1	0.560	0.560	-1	-0.410	-0.410	1	0.150	0.150	0.71
2nd Floor	98.07	0.866	0.970	1.530	0	0	-0.410	-0.866	-0.260	-0.110	1.59
1st Floor	98.07	0.5	0.560	2.089	1	0.820	0.410	0.5	0.150	0.040	2.13
Total SRSS Shear											2.13

Minimum base shear per Table 2:

$$2.75 \times 245.18 / 100 = 6.74 \text{ kN}$$

The multiplying factor (ratio of minimum base shear to total SRSS base shear): $6.47 / 2.13 = 3.17$

Results

Table 524:

Result Type	Reference	STAAD.Pro	Difference	Comments
Story shear, roof (kN)	0.71	0.71	none	
Story shear, 2nd (kN)	1.59	1.59	none	
Story shear, 1st (kN)	2.13	2.13	none	
Base shear (kN)	2.13	2.13	none	
Minimum base shear (kN)	6.74	6.7423	negligible	
Multiplying factor	3.17	3.1668	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2015 Response Spectrum.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 4e-06 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
```

```
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS IS1893 2015-P4 X 0.024 DAMP 0.05 RSMIN 2.75
SOIL TYPE 3
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       0.229                  4.36810
         2                       0.625                  1.59884
         3                       0.854                  1.17043
   ***NOTE: Response Spectrum Analysis for IS1893 (Part-4): 2015 should be
carried
   out with P-Delta Analysis as per 10.3
 1893 RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177131E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     RESPONSE LOAD CASE     1
        MODE    SPECTRAL ACCELERATION       DESIGN   SEISMIC   COEFFICIENT
        ----    ---------------------     ------------------------------
                                              X           Y           Z
          1          0.38232                0.0092      0.0000      0.0000
```

```
         2           1.04451                0.0251      0.0000      0.0000
         3           1.42683                0.0342      0.0000      0.0000
      STAAD SPACE                                                -- PAGE NO.
6
 PEAK STORY SHEAR
        STORY           LEVEL IN METE                   PEAK STORY SHEAR IN
KN
        -----           -------------
--------------------------------------------------
                                                  X
Y                  Z
         3                     9.00              0.71
0.00               0.00
         2                     6.00              1.59
0.00               0.00
         1                     3.00              2.13
0.00               0.00
        BASE                   0.00              2.13
0.00               0.00
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
           DIRECTIONAL VALUES:                SCALE FACTOR =    1.00
    X =  0.02   Y =  0.00   Z =  0.00          DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           4.3681          3.7492
           1.5988         10.2431
           1.1704         13.9924
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                    0.38232       0.05000
              2                    1.04451       0.05000
              3                    1.42683       0.05000
      STAAD SPACE                                                -- PAGE NO.
7
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY           MZ
      1      4.368        2.09        0.00        0.00        0.00
0.00      -12.53
      2      1.599        0.41        0.00        0.00        0.00
0.00        1.23
      3      1.170        0.04        0.00        0.00        0.00
0.00       -0.24
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X      Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000      2.09     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      0.41     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.04     0.00
0.00
```

```
---------------------------
                                  TOTAL SRSS   SHEAR       2.13      0.00
0.00
                                  TOTAL 10PCT SHEAR       2.13      0.00
0.00
                                  TOTAL ABS    SHEAR       2.54      0.00
0.00
                                  TOTAL CSM    SHEAR       2.13      0.00
0.00
   Minimum Base Shear Per Table 2 IS1893 2015 Part-4:      6.7423 KN
   NOTE : THE BASE SHEAR (VB) FROM RESPONSE SPECTRUM IS LESS THAN THE
   MINIMUM BASE SHEAR PER TABLE 2. MULTIPLYING FACTOR IS       3.1668
```

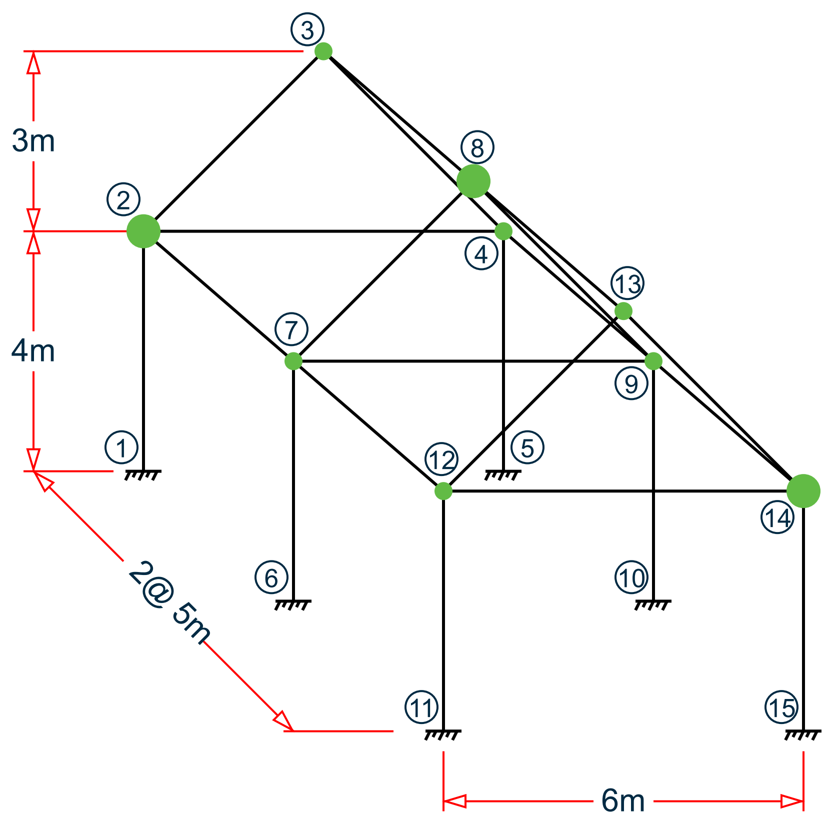
Related Links

- *TR.32.10.1.9 Response Spectrum Specification per IS: 1893 (Part 4)-2015* (on page 2555)

V. IS 1893 2015 Static Seismic

Calculate the design lateral force at each node in each global direction by the equivalent static seismic method per the IS 1893 (Part 4) 2015 specification.

Details



The loads at each node are taken as the same in each global direction (i.e., $W_{xi} = W_{zi} = W_{yi}$). Loads at nodes 2, 8, and 14 are 15 kN ea. Loads at nodes 3, 4, 7, 9, 12, and 13 are 10 kN ea.

Sections used for all members: ISMB 450

Materials used: steel

The structure is modelled in STAAD.Pro considering the following:

- Seismic zone factor, $Z = 0.36$ (`ZONE 0.36`)
- Importance factor, $I = 1.0$ (`I 1.0`)
- Response reduction factor, $R = 5$ (`RF 5`)
- Soil site condition = Hard (`SS 1`)
- Structure type is a category 4 (`ST 4`)
- 2% Damping (`DM 0.02`)

Validation

Seismic Load

Seismic Loads as per IS 1893 (Part 4): 2015 specifications are generated along two horizontal directions, global X & global Z and also along vertical direction, global Y.

Time period in X direction: $T_x = 0.09734$ sec.

Time period in Z direction: $T_z = 0.51406$ sec.

Time Period, T, is calculated by the Rayleigh Method

As per Table 15 of IS 1893 (Part 4): 2015

Damping Factor (DF) for 2% damping = 1.4

From figure 1 of IS 1893 (Part 4): 2015, for Hard Soil sites for (0 second <T<0.1 second)

Spectral acceleration coefficient value (S_a/g) in X, Z, and Y directions for 2% damping:

$$\frac{S_a}{g}_x = DF \times \left(1 + 15T\right) = 1.4 \times \left[1 + \left(15 \times 0.09734\right)\right] = 3.444$$

From figure 1 of IS 1893 (Part 4): 2015, for Hard Soil sites for (0.4 second <T<10 second)

$$\frac{S_a}{g}_z = DF \times \frac{1}{T} = 1.4 \times \frac{1}{0.51406} = 2.723$$

As per clause 10.2 of IS 1893 (Part 4): 2015

$$\frac{S_a}{g}_y = \frac{2}{3} \times Maximum\ of \left(\frac{S_a}{g}_x, \frac{S_a}{g}_z\right) = \frac{2}{3} \times 3.444 = 2.296$$

As per clause 7.3.2 of IS 1893 (Part 4): 2015

The seismic coefficient in X, Z and Y direction

$$A_{h\_x} = \frac{Z}{2} \times \frac{I}{R} \times \frac{S_a}{g}_x = \frac{0.36}{2} \times \frac{1}{5} \times 3.444 = 0.124$$

$$A_{h\_z} = \frac{Z}{2} \times \frac{I}{R} \times \frac{S_a}{g}_z = \frac{0.36}{2} \times \frac{1}{5} \times 2.723 = 0.098$$

$$A_{h\_y} = \frac{Z}{2} \times \frac{I}{R} \times \frac{S_a}{g}_y = \frac{0.36}{2} \times \frac{1}{5} \times 2.296 = 0.083$$

Node	W (kN)	Design lateral force in X direction $Q_x = W_x \times A_{h\_x}$	Design lateral force in Z direction $Q_z = W_z \times A_{h\_z}$	Design force in Y direction $Q_y = W_y \times A_{h\_y}$
2	15	(15 × 0.124) = 1.860	(15 × 0.098) = 1.471	(15 × 0.083) = 1.240
3	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
4	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
7	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
8	15	(15 × 0.124) = 1.860	(15 × 0.098) = 1.471	(15 × 0.083) = 1.240
9	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
12	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
13	10	(10 × 0.124) = 1.240	(10 × 0.098) = 0.980	(10 × 0.083) = 0.827
14	15	(15 × 0.124) = 1.860	(15 × 0.098) = 1.471	(15 × 0.083) = 1.240

Results

Table 525:

Result Type	Calculations	STAAD.Pro	Difference	Comments
S_a/g_x	3.444	3.444	none	
A_{h_x}	0.124	0.124	none	
S_a/g_z	2.723	2.723	none	
A_{h_z}	0.098	0.098	none	
S_a/g_y	2.296	2.296	none	
A_{h_y}	0.083	0.083	none	
Q_x at Nodes 2,8,14 (kN)	1.860	1.860	none	
Q_x at Nodes 3,4,7, 9,12, 13 (kN)	1.240	1.240	none	

Result Type	Calculations	STAAD.Pro	Difference	Comments
Q_z at Nodes 2,8,14 (kN)	1.471	1.471	none	
Q_z at Nodes 3,4,7, 9,12, 13 (kN)	0.980	0.980	none	
Q_y at Nodes 2,8,14 (kN)	1.240	1.240	none	
Q_y at Nodes 3,4,7, 9,12, 13 (kN)	0.827	0.827	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2015 Static Seismic.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Mar-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 3 7 0; 4 6 4 0; 5 6 0 0; 6 0 0 5; 7 0 4 5; 8 3 7 5;
9 6 4 5; 10 6 0 5; 11 0 0 10; 12 0 4 10; 13 3 7 10; 14 6 4 10; 15 6 0 10;
MEMBER INCIDENCES
2 1 2; 3 2 3; 4 3 4; 5 4 5; 7 6 7; 8 7 8; 9 8 9; 10 9 10; 11 2 7; 12 8 3;
13 4 9; 14 2 4; 15 7 9; 16 11 12; 17 12 13; 18 13 14; 19 14 15; 20 12 14;
21 7 12; 22 9 14; 23 8 13;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
2 TO 5 7 TO 23 TABLE ST ISMB450
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 6 10 11 15 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 8 14 FX 15 FY -15 FZ 15
3 4 7 9 12 13 FX 10 FY -10 FZ 10
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
```

```
DIA 1 TYPE RIG HEI 4
DIA 2 TYPE RIG HEI 7
DEFINE IS1893 2015 LOAD PART4
ZONE 0.36 RF 5 I 1 SS 1 ST 4 DM 0.02
LOAD 1 LOADTYPE None  TITLE STATIC_X
1893 LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
1893 LOAD Z 1
LOAD 3 LOADTYPE None  TITLE STATIC_Y
1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE NONE  TITLE STATIC_X
   -----------
   LOADING     2  LOADTYPE NONE  TITLE STATIC_Z
   -----------
   LOADING     3  LOADTYPE NONE  TITLE STATIC_Y
   -----------
  **WARNING: IF THIS UBC/IBC ANALYSIS HAS TENSION/COMPRESSION
  OR REPEAT LOAD OR RE-ANALYSIS OR SELECT OPTIMIZE, THEN EACH
  UBC/IBC CASE SHOULD BE FOLLOWED BY PERFORM ANALYSIS & CHANGE.
 BASE SHEAR AND TIME PERIOD IN X
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR X 1893 LOADING =    0.09734 SEC       *
   * SA/G PER 1893=    3.444, LOAD FACTOR= 1.000           *
   * AH PER 1893=    0.124 WEIGHT=      105.00 KN          *
   *********************************************************
   ***NOTE: For Industrial Structures, Static Seismic Load is applied
   per node. Use command: PRINT LOAD DATA to view the applied load at
   each node. Ref. Cl.10.2
 BASE SHEAR AND TIME PERIOD IN Z
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR Z 1893 LOADING =    0.51406 SEC       *
   * SA/G PER 1893=    2.723, LOAD FACTOR= 1.000           *
   * AH PER 1893=    0.098 WEIGHT=      105.00 KN          *
   *********************************************************
   ***NOTE: For Industrial Structures, Static Seismic Load is applied
   per node. Use command: PRINT LOAD DATA to view the applied load at
   each node. Ref. Cl.10.2
 BASE SHEAR AND TIME PERIOD IN Y
   *********************************************************
   * UNITS - KN    METE                                    *
   * SA/G FOR Y-DIR CALCULATED AS PER CL.10.2              *
   * FOR INDUSTRIAL STRUCTURES                             *
   * SA/G PER 1893=    2.296, LOAD FACTOR= 1.000           *
   * AH PER 1893=    0.083 WEIGHT=      105.00 KN          *
   *********************************************************
   ***NOTE: For Industrial Structures, Static Seismic Load is applied
   per node. Use command: PRINT LOAD DATA to view the applied load at
   each node. Ref. Cl.10.2
      STAAD SPACE                                               -- PAGE NO.
```

```
5
    JOINT             LATERAL           TORSIONAL            LOAD -     1
                      LOAD (KN   )      MOMENT (KN   -METE) FACTOR -    1.000
    -----             -------           ---------
      2      FX         1.860     MY        0.000
      4      FX         1.240     MY        0.000
      7      FX         1.240     MY        0.000
      9      FX         1.240     MY        0.000
     12      FX         1.240     MY        0.000
     14      FX         1.860     MY        0.000
                  -----------         -----------
          TOTAL =       8.679               0.000 AT LEVEL     4.000 METE
      3      FX         1.240     MY        0.000
      8      FX         1.860     MY        0.000
     13      FX         1.240     MY        0.000
                  -----------         -----------
          TOTAL =       4.340               0.000 AT LEVEL     7.000 METE
    JOINT             LATERAL           TORSIONAL            LOAD -     2
                      LOAD (KN   )      MOMENT (KN   -METE) FACTOR -    1.000
    -----             -------           ---------
      2      FZ         1.471     MY        0.000
      4      FZ         0.980     MY        0.000
      7      FZ         0.980     MY        0.000
      9      FZ         0.980     MY        0.000
     12      FZ         0.980     MY        0.000
     14      FZ         1.471     MY        0.000
                  -----------         -----------
          TOTAL =       6.863               0.000 AT LEVEL     4.000 METE
      3      FZ         0.980     MY        0.000
      8      FZ         1.471     MY        0.000
     13      FZ         0.980     MY        0.000
                  -----------         -----------
          TOTAL =       3.431               0.000 AT LEVEL     7.000 METE
    JOINT             LATERAL           TORSIONAL            LOAD -     3
                      LOAD (KN   )      MOMENT (KN   -METE) FACTOR -    1.000
    -----             -------           ---------
      2      FY         1.240     MY        0.000
      4      FY         0.827     MY        0.000
      7      FY         0.827     MY        0.000
      9      FY         0.827     MY        0.000
     12      FY         0.827     MY        0.000
     14      FY         1.240     MY        0.000
     STAAD SPACE                                               -- PAGE NO.
6
                  -----------         -----------
          TOTAL =       5.786               0.000 AT LEVEL     4.000 METE
      3      FY         0.827     MY        0.000
      8      FY         1.240     MY        0.000
     13      FY         0.827     MY        0.000
                  -----------         -----------
          TOTAL =       2.893               0.000 AT LEVEL     7.000 METE
```

Related Links

- *TR.31.2.12 IS:1893 (Part 4) 2015 Codes - Lateral Seismic Load* (on page 2398)

V. IS 1893 2002 Response Spectrum

Calculation of base shear and its distribution along the height for response spectrum method in IS 1893 (Part 1) : 2002.

Problem

A three story building is modelled in STAAD.Pro considering the following:

Seismic zone factor, $Z = 0.36$
Importance factor, $I = 1$
Response reduction factor, $R = 5$
Soil site conditions = Hard

The generated model is subjected to a response spectrum load along global X direction. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 526: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 527: Mode Shapes

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	0	0	0	0	0	0
	1st Floor/Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/Joints 5 & 6	-0.86603	0	0	0	0	0

Mode	Story Level/Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
	1st Floor/Joints 2 & 3	0.5	0	0	0	0	0

Table 528: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.175		183.00	147.11		49.035	147.105		13.138	147.106
Modal Weight $Mk \times g$		227.66			16.345			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.479		
Mode Participation Factor Pk $\sum Wi \times \phi$ / $\sum Wi \times \phi^2$		1.244			0.3333			0.0893		

	Mode 1	Mode 2	Mode 3
S_a/g	2.5 (time period < 0.4 s)	2.5 (time period < 0.4 s)	2.2063 (1 + 15 × 0.08042 as time period < 0.1 s)
Ak $(Z/2 \times I/R \times S_a/g)$	0.09 (0.36/2 × 1/5 × 2.5)	0.09 (0.36/2 × 1/5 × 2.5)	0.0794 (0.36/2 × 1/5 × 2.2063)

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	ϕ	Qi (Ak × ϕ × Pk × Wi)	Vi (∑Qi)	
Roof	49.035	1	5.490	5.490	-1	-1.471	-1.471	1	0.3478	0.3478	5.69
2nd Floor	98.07	0.86603	9.509	15.0	0	0	-1.471	-0.86603	-0.6025	-0.2546	15.073
1st Floor	98.07	0.5	5.490	20.49	1	2.942	1.471	0.5	0.3478	0.0932	20.542
Base Shear											20.54

Comparison

Table 529: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear (kN)	Roof	5.69	5.69	none	
	2nd	15.07	15.07	none	
	1st	20.54	20.54	none	
Base shear (kN)		20.54	20.54	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2002 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
```

```
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS 1893 X 0.036 ACC DAMP 0.05
SOIL TYPE 1
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
 1893 RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS           MODAL COMBINATION METHOD USED.
```

```
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     RESPONSE LOAD CASE     1
        MODE    SPECTRAL ACCELERATION       DESIGN   SEISMIC   COEFFICIENT
        ----    ---------------------     ------------------------------
                                              X           Y           Z
          1           2.50000                0.0900      0.0000      0.0000
          2           2.50000                0.0900      0.0000      0.0000
          3           2.20632                0.0794      0.0000      0.0000
      STAAD SPACE                                                -- PAGE NO.
6
 PEAK STORY SHEAR
        STORY          LEVEL IN METE          PEAK STORY SHEAR IN  KN
        -----          -------------          -------------------------
                                                X                  Z
          3                  9.00              5.69               0.00
          2                  6.00             15.07               0.00
          1                  3.00             20.54               0.00
        BASE                 0.00             20.54               0.00
   RESPONSE SPECTRUM VALUES - UNITS ( METE SECOND )
   ------------------------
           DIRECTIONAL VALUES:                 SCALE FACTOR =    1.00
    X =  0.04   Y =  0.00   Z =  0.00          DAMPING FACTOR = 0.050
           PERIOD  VS.  ACCELERATION
           0.3001         24.5166
           0.1099         24.5166
           0.0804         21.6366
           MODE                ACCELERATION-G     DAMPING
           ----                --------------     -------
              1                     2.50000      0.05000
              2                     2.50000      0.05000
              3                     2.20632      0.05000
      STAAD SPACE                                                -- PAGE NO.
7
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.300      20.49        0.00        0.00        0.00
0.00     -122.93
      2      0.110       1.47        0.00        0.00        0.00
0.00        4.41
      3      0.080       0.09        0.00        0.00        0.00
0.00       -0.56
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000     20.49     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000      1.47     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      0.09     0.00
0.00
```

```
---------------------------
                                  TOTAL SRSS  SHEAR     20.54     0.00
0.00
                                  TOTAL 10PCT SHEAR     20.54     0.00
0.00
                                  TOTAL ABS   SHEAR     22.05     0.00
0.00
                                  TOTAL CSM   SHEAR     20.54     0.00
0.00
```

Related Links

- *TR.32.10.1.7 Response Spectrum Specification per IS: 1893 (Part 1)-2002* (on page 2534)

V. IS 1893 2002 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static method in IS 1893 (Part 1) : 2002

Problem

A reinforced concrete frame structure is 4 bays at 4m ea. by 3 bays at 4m ea. in plan. The structure is 3 stories at 3m ea.

The structure is modelled in STAAD.Pro considering the following:

Seismic zone factor, $Z = 0.36$ (`ZONE 0.36`)
Importance factor, $I = 1$ (`I 1`)
Response reduction factor, $R = 5$ (`RF 5`)
Soil site condition = Hard (`SS 1`)
Structure type is a reinforced concrete frame (`ST 1`)

The seismic weight for each floor is assumed as 100 kN applied as joint weight to nodes 39, 60, & 80. Seismic Loads as per IS 1893 (Part 1) : 2002 specifications are generated along two horizontal directions global X & global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Height = 3 × 3 = 9m

$$Ta_x = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}$$

$$Ta_z = 0.075h^{0.75} = 0.075 \times (9^{0.75}) = 0.39 \text{ sec}$$

$$[Sa/g]_x = [Sa/g]_z = 2.5 \text{ (As time period < 0.4 sec)}$$

$$Ah_x = (Z/2) \times (I/R) \times [Sa/g]_x = 0.36/2 \times 1/5 \times 2.5 = 0.09$$

$$Ah_z = (Z/2) \times (I/R) \times [Sa/g]_z = 0.36/2 \times 1/5 \times 2.5 = 0.09$$

$$VB = Ah \times W$$

$$VB_x = Ah_x \times W = 0.09 \times (3 \times 100) = 27 \text{ kN}$$

$$VB_z = Ah_z \times W = 0.09 \times (3 \times 100) = 27 \text{ kN}$$

Story Level	Wi	hi	$Wi \times hi^2$	$(Wi \times hi^2)/\sum(Wi \times hi^2)$	Qx	Qz
Roof	100	9	8,100	0.643	17.357	17.357
2nd	100	6	3,600	0.286	7.714	7.714
1st	100	3	900	0.0714	1.929	1.929
Summation	300		12,600			
Vb_h		27			27	

Comparison

Table 530: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear X, Z (kN)	roof	1.929, 1.929	1.929, 1.929	none	
	2nd	7.714, 7.714	7.714, 7.714	none	
	1st	17.357, 17.357	17.357, 17.357	none	
Base Shear (kN)	VBx	27	27.000	none	
	VBz	27	27.000	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\IS 1893\IS 1893 2002 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Jun-18
END JOB INFORMATION
INPUT WIDTH 79
*SET STAR 0
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 4 3 0; 4 4 0 0; 5 0 0 4; 6 0 3 4; 7 4 3 4; 8 4 0 4;
9 8 3 0; 10 8 0 0; 11 8 3 4; 12 8 0 4; 13 12 3 0; 14 12 0 0; 15 12 3 4;
16 12 0 4; 17 16 3 0; 18 16 0 0; 19 16 3 4; 20 16 0 4; 21 0 0 8; 22 0 3 8;
23 4 3 8; 24 4 0 8; 25 8 3 8; 26 8 0 8; 27 12 3 8; 28 12 0 8; 29 16 3 8;
30 16 0 8; 31 0 0 12; 32 0 3 12; 33 4 3 12; 34 4 0 12; 35 8 3 12; 36 8 0 12;
37 12 3 12; 38 12 0 12; 39 16 3 12; 40 16 0 12; 41 0 6 0; 42 4 6 0; 43 0 6 4;
44 4 6 4; 45 8 6 0; 46 8 6 4; 47 12 6 0; 48 12 6 4; 49 16 6 0; 50 16 6 4;
51 0 6 8; 52 4 6 8; 53 8 6 8; 54 12 6 8; 55 16 6 8; 56 0 6 12; 57 4 6 12;
58 8 6 12; 59 12 6 12; 60 16 6 12; 61 0 9 0; 62 4 9 0; 63 0 9 4; 64 4 9 4;
65 8 9 0; 66 8 9 4; 67 12 9 0; 68 12 9 4; 69 16 9 0; 70 16 9 4;
71 0 9 8; 72 4 9 8; 73 8 9 8; 74 12 9 8; 75 16 9 8; 76 0 9 12;
```

```
77 4 9 12; 78 8 9 12; 79 12 9 12; 80 16 9 12;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 3 9; 10 7 11;
11 9 10; 12 9 11; 13 11 12; 14 9 13; 15 11 15; 16 13 14; 17 13 15; 18 15 16;
19 13 17; 20 15 19; 21 17 18; 22 17 19; 23 19 20; 24 6 22; 25 7 23; 26 11 25;
27 15 27; 28 19 29; 29 21 22; 30 22 23; 31 23 24; 32 23 25; 33 25 26; 34 25
27;
35 27 28; 36 27 29; 37 29 30; 38 22 32; 39 23 33; 40 25 35; 41 27 37; 42 29
39;
43 31 32; 44 32 33; 45 33 34; 46 33 35; 47 35 36; 48 35 37; 49 37 38; 50 37
39;
51 39 40; 52 2 41; 53 3 42; 54 6 43; 55 7 44; 56 9 45; 57 11 46; 58 13 47;
59 15 48; 60 17 49; 61 19 50; 62 22 51; 63 23 52; 64 25 53; 65 27 54; 66 29
55;
67 32 56; 68 33 57; 69 35 58; 70 37 59; 71 39 60; 72 41 42; 73 41 43; 74 42
44;
75 43 44; 76 42 45; 77 44 46; 78 45 46; 79 45 47; 80 46 48; 81 47 48; 82 47
49;
83 48 50; 84 49 50; 85 43 51; 86 44 52; 87 46 53; 88 48 54; 89 50 55; 90 51
52;
91 52 53; 92 53 54; 93 54 55; 94 51 56; 95 52 57; 96 53 58; 97 54 59; 98 55
60;
99 56 57; 100 57 58; 101 58 59; 102 59 60; 103 41 61; 104 42 62; 105 43 63;
106 44 64; 107 45 65; 108 46 66; 109 47 67; 110 48 68; 111 49 69; 112 50 70;
113 51 71; 114 52 72; 115 53 73; 116 54 74; 117 55 75; 118 56 76; 119 57 77;
120 58 78; 121 59 79; 122 60 80; 123 61 62; 124 61 63; 125 62 64; 126 63 64;
127 62 65; 128 64 66; 129 65 66; 130 65 67; 131 66 68; 132 67 68; 133 67 69;
134 68 70; 135 69 70; 136 63 71; 137 64 72; 138 66 73; 139 68 74; 140 70 75;
141 71 72; 142 72 73; 143 73 74; 144 74 75; 145 71 76; 146 72 77; 147 73 78;
148 74 79; 149 75 80; 150 76 77; 151 77 78; 152 78 79; 153 79 80;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 153 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 10 12 14 16 18 20 21 24 26 28 30 31 34 36 38 40 FIXED
DEFINE 1893 LOAD
*DEFINE IS1893 2016 LOAD
*ZONE 0.36 RF 5 I 1.2 SS 1 ST 1 DM 0.05 HT 9 DX 16 DZ 12
ZONE 0.36 RF 5 I 1 SS 1 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05 HT 9 DX 16 DZ 12
*ZONE 0.36 RF 5 I 1.2 SS 2 ST 1 DM 0.05
*ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05 HT 9 DX 16 DZ 12
*ZONE 0.36 RF 5 I 1.2 SS 3 ST 1 DM 0.05
JOINT WEIGHT
39 60 80 WEIGHT 100
LOAD 1 LOADTYPE Seismic  TITLE SS_(+X)
1893 LOAD X 1
```

```
LOAD 2 LOADTYPE Seismic  TITLE SS_(+Z)
1893 LOAD Z 1
*LOAD 3 LOADTYPE Seismic  TITLE SS_(+Y)
*1893 LOAD Y 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR X 1893 LOADING =    0.38971 SEC       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.0900 X      300.00=      27.00 KN   *
   *                                                       *
   *********************************************************
   *********************************************************
   * UNITS - KN    METE                                    *
   * TIME PERIOD FOR Z 1893 LOADING =    0.38971 SEC       *
   * SA/G PER 1893=    2.500, LOAD FACTOR= 1.000           *
   * VB PER 1893=    0.0900 X      300.00=      27.00 KN   *
   *                                                       *
   *********************************************************
    JOINT             LATERAL          TORSIONAL            LOAD -    1
                      LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----             -------          ---------
      39     FX        1.929    MY        0.000
                 -----------        -----------
          TOTAL =      1.929              0.000 AT LEVEL     3.000 METE
    VB PER 1893 =     27.000 KN
      60     FX        7.714    MY        0.000
                 -----------        -----------
          TOTAL =      7.714              0.000 AT LEVEL     6.000 METE
    VB PER 1893 =     27.000 KN
      80     FX       17.357    MY        0.000
                 -----------        -----------
          TOTAL =     17.357              0.000 AT LEVEL     9.000 METE
    VB PER 1893 =     27.000 KN
      STAAD SPACE                                              -- PAGE NO.
5
    JOINT             LATERAL          TORSIONAL            LOAD -    2
                      LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----             -------          ---------
      39     FZ        1.929    MY        0.000
                 -----------        -----------
          TOTAL =      1.929              0.000 AT LEVEL     3.000 METE
    VB PER 1893 =     27.000 KN
      60     FZ        7.714    MY        0.000
                 -----------        -----------
          TOTAL =      7.714              0.000 AT LEVEL     6.000 METE
    VB PER 1893 =     27.000 KN
      80     FZ       17.357    MY        0.000
                 -----------        -----------
          TOTAL =     17.357              0.000 AT LEVEL     9.000 METE
    VB PER 1893 =     27.000 KN
```

Related Links

- *TR.31.2.10 IS:1893 (Part 1) 2002 & Part 4 (2005) Codes - Lateral Seismic Load* (on page 2387)

V. Moving Load

V. Moving Load Generator

To validate the effect of wheel load assembly on bridge girders simulated by a series of equivalent concentrated loads for a particular wheel position, generated by Moving Load generation option.

Details

A bridge decking system of 10m span is simulated by three longitudinal girder of size 0.3m×0.6m (Concrete) at a spacing of 2m and six transverse girders of size 0.3m×0.45m (Concrete) at a spacing of 2m. Longitudinal girders are pinned support at both the ends.

Vehicle definition – 10 kN and 16 kN per wheel in the Front and Rear Axle respectively. The vehicle width is 1.2m and the axle distance is 1.5m. Starting position is from the end of the deck, aligning the rear axle along the last transverse girder. The direction of movement along the longitudinal girder with an incremental distance of 0.5m.

Validation

Among the generated wheel loads, for the 9th position of the wheel assembly, the structure is analyzed for that generated load. The structure is again analyzed with the static load system equivalent to the generated wheel load and observed the difference in results if any.

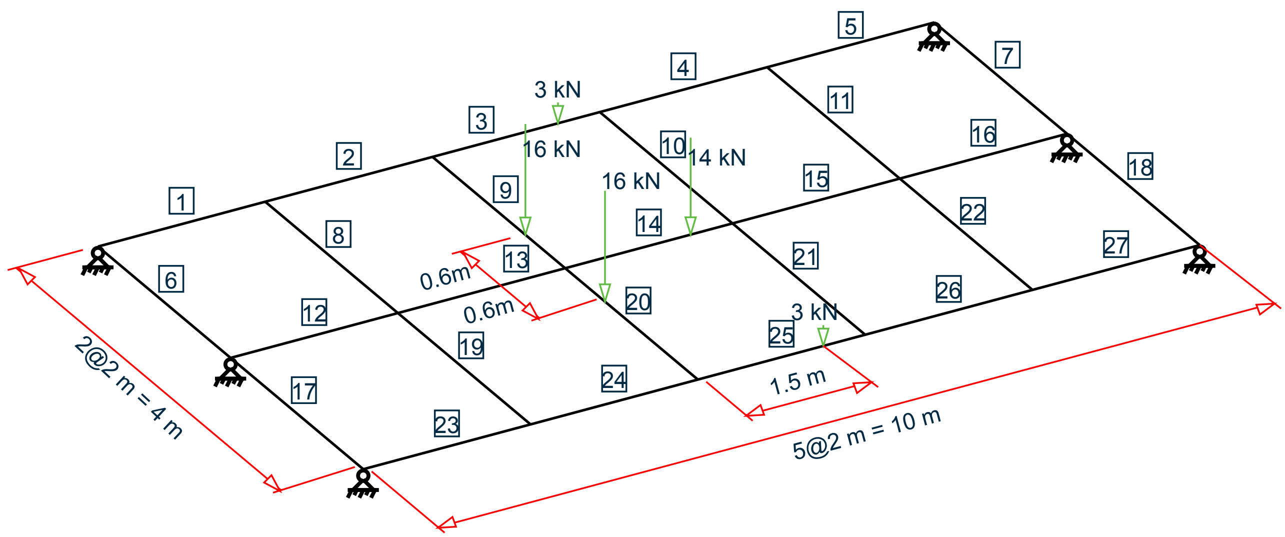


Figure 436: The position of the generated load case 9

Note: The postprocessing diagrams in STAAD.Pro show two overlapping 7 kN axle reactions on member 14.

The model uses the moving load generator facility and a static load equivalent to the generated Load Case 9 (i.e., 9th position of the wheel assembly). Both support reactions and joint displacements are compared.

Comparison

Table 531: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Support reaction at joint 7 (kN)	11.37	11.37	none	
Vertical displacement at joint 10 (cm)	0.3025	0.3025	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Moving Load\Moving Load Generator.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 2 0 0; 4 4 0 0; 5 6 0 0; 6 8 0 0; 7 0 0 2; 8 10 0 2;
9 2 0 2; 10 4 0 2; 11 6 0 2; 12 8 0 2; 13 0 0 4; 14 10 0 4; 15 2 0 4; 16 4 0
4;
17 6 0 4; 18 8 0 4;
MEMBER INCIDENCES
1 1 3; 2 3 4; 3 4 5; 4 5 6; 5 6 2; 6 1 7; 7 2 8; 8 3 9; 9 4 10; 10 5 11;
11 6 12; 12 7 9; 13 9 10; 14 10 11; 15 11 12; 16 12 8; 17 7 13; 18 8 14;
19 9 15; 20 10 16; 21 11 17; 22 12 18; 23 13 15; 24 15 16; 25 16 17; 26 17 18;
27 18 14;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 12 TO 16 23 TO 27 PRIS YD 0.6 ZD 0.3
6 TO 11 17 TO 22 PRIS YD 0.45 ZD 0.3
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 2 7 8 13 14 PINNED
DEFINE MOVING LOAD
TYPE 1 LOAD 16 10
DIST 1.5 WID 1.2
LOAD GENERATION 18
```

```
TYPE 1 0 0 2.6 XINC 0.5
LOAD 19 LOADTYPE Live  TITLE STATIC POS9
MEMBER LOAD
9 CON GY -16 1.4
20 CON GY -16 0.6
14 CON GY -14 1.5
3 25 CON GY -3 1.5
PERFORM ANALYSIS PRINT LOAD DATA
LOAD LIST 9 19
PRINT SUPPORT REACTION ALL
PRINT JOINT DISPLACEMENTS ALL
FINISH
```

STAAD Output

```
   LOADING     1
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     17                           -16.0000 GY   0.60
      6                           -16.0000 GY   1.40
     23                            -3.0000 GY   1.50
     12                            -7.0000 GY   1.50
     12                            -7.0000 GY   1.50
      1                            -3.0000 GY   1.50
   LOADING     2
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     23                            -4.8000 GY   0.50
     12                           -11.2000 GY   0.50
     12                           -11.2000 GY   0.50
      1                            -4.8000 GY   0.50
     19                           -10.0000 GY   0.60
      8                           -10.0000 GY   1.40
   LOADING     3
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     23                            -4.8000 GY   1.00
     12                           -11.2000 GY   1.00
     12                           -11.2000 GY   1.00
      1                            -4.8000 GY   1.00
     24                            -3.0000 GY   0.50
     13                            -7.0000 GY   0.50
     13                            -7.0000 GY   0.50
      2                            -3.0000 GY   0.50
   LOADING     4
   -----------
      STAAD SPACE                                            -- PAGE NO.
4
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     23                            -4.8000 GY   1.50
     12                           -11.2000 GY   1.50
     12                           -11.2000 GY   1.50
      1                            -4.8000 GY   1.50
```

```
     24                            -3.0000 GY   1.00
     13                            -7.0000 GY   1.00
     13                            -7.0000 GY   1.00
      2                            -3.0000 GY   1.00
   LOADING     5
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     19                           -16.0000 GY   0.60
      8                           -16.0000 GY   1.40
     24                            -3.0000 GY   1.50
     13                            -7.0000 GY   1.50
     13                            -7.0000 GY   1.50
      2                            -3.0000 GY   1.50
   LOADING     6
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     24                            -4.8000 GY   0.50
     13                           -11.2000 GY   0.50
     13                           -11.2000 GY   0.50
      2                            -4.8000 GY   0.50
     20                           -10.0000 GY   0.60
      9                           -10.0000 GY   1.40
   LOADING     7
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
      STAAD SPACE                                               -- PAGE NO.
5
     24                            -4.8000 GY   1.00
     13                           -11.2000 GY   1.00
     13                           -11.2000 GY   1.00
      2                            -4.8000 GY   1.00
     25                            -3.0000 GY   0.50
     14                            -7.0000 GY   0.50
     14                            -7.0000 GY   0.50
      3                            -3.0000 GY   0.50
   LOADING     8
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     24                            -4.8000 GY   1.50
     13                           -11.2000 GY   1.50
     13                           -11.2000 GY   1.50
      2                            -4.8000 GY   1.50
     25                            -3.0000 GY   1.00
     14                            -7.0000 GY   1.00
     14                            -7.0000 GY   1.00
      3                            -3.0000 GY   1.00
   LOADING     9
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     20                           -16.0000 GY   0.60
      9                           -16.0000 GY   1.40
     25                            -3.0000 GY   1.50
     14                            -7.0000 GY   1.50
```

```
     14                            -7.0000 GY   1.50
      3                            -3.0000 GY   1.50
   LOADING    10
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     25                            -4.8000 GY   0.50
     14                           -11.2000 GY   0.50
     14                           -11.2000 GY   0.50
      3                            -4.8000 GY   0.50
      STAAD SPACE                                            -- PAGE NO.
6
     21                           -10.0000 GY   0.60
     10                           -10.0000 GY   1.40
   LOADING    11
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     25                            -4.8000 GY   1.00
     14                           -11.2000 GY   1.00
     14                           -11.2000 GY   1.00
      3                            -4.8000 GY   1.00
     26                            -3.0000 GY   0.50
     15                            -7.0000 GY   0.50
     15                            -7.0000 GY   0.50
      4                            -3.0000 GY   0.50
   LOADING    12
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     25                            -4.8000 GY   1.50
     14                           -11.2000 GY   1.50
     14                           -11.2000 GY   1.50
      3                            -4.8000 GY   1.50
     26                            -3.0000 GY   1.00
     15                            -7.0000 GY   1.00
     15                            -7.0000 GY   1.00
      4                            -3.0000 GY   1.00
   LOADING    13
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     21                           -16.0000 GY   0.60
     10                           -16.0000 GY   1.40
     26                            -3.0000 GY   1.50
     15                            -7.0000 GY   1.50
     15                            -7.0000 GY   1.50
      4                            -3.0000 GY   1.50
      STAAD SPACE                                            -- PAGE NO.
7
   LOADING    14
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     26                            -4.8000 GY   0.50
     15                           -11.2000 GY   0.50
     15                           -11.2000 GY   0.50
      4                            -4.8000 GY   0.50
```

```
     22                          -10.0000 GY   0.60
     11                          -10.0000 GY   1.40
   LOADING    15
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     26                           -4.8000 GY   1.00
     15                          -11.2000 GY   1.00
     15                          -11.2000 GY   1.00
      4                           -4.8000 GY   1.00
     27                           -3.0000 GY   0.50
     16                           -7.0000 GY   0.50
     16                           -7.0000 GY   0.50
      5                           -3.0000 GY   0.50
   LOADING    16
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     26                           -4.8000 GY   1.50
     15                          -11.2000 GY   1.50
     15                          -11.2000 GY   1.50
      4                           -4.8000 GY   1.50
     27                           -3.0000 GY   1.00
     16                           -7.0000 GY   1.00
     16                           -7.0000 GY   1.00
      5                           -3.0000 GY   1.00
   LOADING    17
   -----------
      STAAD SPACE                                            -- PAGE NO.
8
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     22                          -16.0000 GY   0.60
     11                          -16.0000 GY   1.40
     27                           -3.0000 GY   1.50
     16                           -7.0000 GY   1.50
     16                           -7.0000 GY   1.50
      5                           -3.0000 GY   1.50
   LOADING    18
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     27                           -4.8000 GY   0.50
     16                          -11.2000 GY   0.50
     16                          -11.2000 GY   0.50
      5                           -4.8000 GY   0.50
     18                          -10.0000 GY   0.60
      7                          -10.0000 GY   1.40
   LOADING    19  LOADTYPE LIVE  TITLE STATIC POS9
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
      9                          -16.0000 GY   1.40
     20                          -16.0000 GY   0.60
     14                          -14.0000 GY   1.50
      3                           -3.0000 GY   1.50
     25                           -3.0000 GY   1.50
   ************ END OF DATA FROM INTERNAL STORAGE ************
```

```
     45. LOAD LIST 9 19
     46. PRINT SUPPORT REACTION ALL
   SUPPORT  REACTION ALL
       STAAD SPACE                                              -- PAGE NO.
9
    SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
    -----------------
  JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
       1    9      0.00      8.41      0.00      0.00      0.00      0.00
           19      0.00      8.41      0.00      0.00      0.00      0.00
       2    9      0.00      7.52      0.00      0.00      0.00      0.00
           19      0.00      7.52      0.00      0.00      0.00      0.00
       7    9      0.00     11.37      0.00      0.00      0.00      0.00
           19      0.00     11.37      0.00      0.00      0.00      0.00
       8    9      0.00      8.76      0.00      0.00      0.00      0.00
           19      0.00      8.76      0.00      0.00      0.00      0.00
      13    9      0.00      8.41      0.00      0.00      0.00      0.00
           19      0.00      8.41      0.00      0.00      0.00      0.00
      14    9      0.00      7.52      0.00      0.00      0.00      0.00
           19      0.00      7.52      0.00      0.00      0.00      0.00
    ************** END OF LATEST ANALYSIS RESULT **************
     47. PRINT JOINT DISPLACEMENTS ALL
   JOINT    DISPLACE ALL
       STAAD SPACE                                              -- PAGE NO.
10
    JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
    ------------------
  JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
       1    9    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0009
           19    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0009
       2    9    0.0000    0.0000    0.0000    0.0000    0.0000    0.0009
           19    0.0000    0.0000    0.0000    0.0000    0.0000    0.0009
       3    9    0.0000   -0.1712    0.0000    0.0001    0.0000   -0.0007
           19    0.0000   -0.1712    0.0000    0.0001    0.0000   -0.0007
       4    9    0.0000   -0.2777    0.0000    0.0002    0.0000   -0.0003
           19    0.0000   -0.2777    0.0000    0.0002    0.0000   -0.0003
       5    9    0.0000   -0.2731    0.0000    0.0001    0.0000    0.0003
           19    0.0000   -0.2731    0.0000    0.0001    0.0000    0.0003
       6    9    0.0000   -0.1645    0.0000    0.0001    0.0000    0.0007
           19    0.0000   -0.1645    0.0000    0.0001    0.0000    0.0007
       7    9    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0010
           19    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0010
       8    9    0.0000    0.0000    0.0000    0.0000    0.0000    0.0009
           19    0.0000    0.0000    0.0000    0.0000    0.0000    0.0009
       9    9    0.0000   -0.1827    0.0000    0.0000    0.0000   -0.0008
           19    0.0000   -0.1827    0.0000    0.0000    0.0000   -0.0008
      10    9    0.0000   -0.3025    0.0000    0.0000    0.0000   -0.0003
           19    0.0000   -0.3025    0.0000    0.0000    0.0000   -0.0003
      11    9    0.0000   -0.2915    0.0000    0.0000    0.0000    0.0004
           19    0.0000   -0.2915    0.0000    0.0000    0.0000    0.0004
      12    9    0.0000   -0.1709    0.0000    0.0000    0.0000    0.0008
           19    0.0000   -0.1709    0.0000    0.0000    0.0000    0.0008
      13    9    0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0009
           19    0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0009
      14    9    0.0000    0.0000    0.0000   -0.0000    0.0000    0.0009
           19    0.0000    0.0000    0.0000   -0.0000    0.0000    0.0009
      15    9    0.0000   -0.1712    0.0000   -0.0001    0.0000   -0.0007
           19    0.0000   -0.1712    0.0000   -0.0001    0.0000   -0.0007
```

```
    16     9     0.0000    -0.2777     0.0000    -0.0002     0.0000    -0.0003
          19     0.0000    -0.2777     0.0000    -0.0002     0.0000    -0.0003
    17     9     0.0000    -0.2731     0.0000    -0.0001     0.0000     0.0003
          19     0.0000    -0.2731     0.0000    -0.0001     0.0000     0.0003
    18     9     0.0000    -0.1645     0.0000    -0.0001     0.0000     0.0007
          19     0.0000    -0.1645     0.0000    -0.0001     0.0000     0.0007
```

Related Links

- *M. Moving Loads* (on page 873)
- *EX. US-12 Moving Load Generation on a Bridge Deck* (on page 6398)
- *EX. UK-12 Moving Load Generation on a Bridge Deck* (on page 6684)
- *TR.31.1 Definition of Moving Load System* (on page 2346)

V. NRC

V. NRC 2010 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in NBCC/NRC 2010.

Details

Example 4.7(1) is modeled in STAAD.Pro.

The generated model is subjected to a Response Spectrum Load along global X direction. Time-Acceleration pairs for the spectrum data can be seen in supporting excel file `NRC_Calculation.xlsx` within worksheet named RS Values. The direction factor along global X direction is assumed as 0.5. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 532: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 533: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 534: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$	ϕ	$Wi \times \phi$	$Wi \times \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.175		183.00	147.11		49.035	147.11		13.138	147.11

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^{2}$	ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^{2}$	ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^{2}$
Modal Weight Mk x g		227.7			16.345			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.4786		
Mode Participation Factor ∏ ∑Wi x ϕ / ∑ Wi x ϕ^{2}		1.244			0.3333			0.0893		

Table 535: Horizontal Acceleration Spectrum Value Calculation

	Mode 1	Mode 2	Mode 3
Time Period T	**0.30014**	**0.10986**	**0.08042**
Spectrum Data	Obtain from `NRC_Calculation.xlsx` within worksheet named **RS Values**		
Formula	Linearly Interpolate acceleration values between 0.3 sec & 0.36 sec to obtain acceleration value for 0.30014 sec.	Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.10986 sec.	Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.08042 sec.
Sa	1.50	1.40	1.10
Horizontal Seismic Coefficient Cs = (Sa x Direction Factor) = Sa x 0.5	**1.** 249998725 1.25	1.20	1.05

Table 536: Base Shear Calculation and its distribution across height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	
Roof	49.035	1	76.25	76.25	-1	-19.60	-19.60	1	4.608	4.608	78.86
2nd Floor	98.07	0.86603	132.1	208.3	0	0	-19.60	-0.86603	-7.981	-3.373	209.27
1st Floor	98.07	0.5	76.25	284.6	1	39.21	19.60	0.5	4.608	1.235	285.25
Base Shear V											285.25

Comparison

Table 537: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Base shear (kN)	285.25	285.25	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\NRC 2010 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
```

```
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS NRC 2010 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3
24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61436; 1.08 9.08023; 1.14 8.60233; 1.2 8.17221;
1.26 7.78306; 1.32 7.42928; 1.38 7.10627; 1.44 6.81018; 1.5 6.53777;
1.56 6.28632; 1.62 6.05349; 1.68 5.83729; 1.74 5.63601; 1.8 5.44814;
1.86 5.2724; 1.92 5.10763; 1.98 4.95286; 2.04 4.80718; 2.1 4.66984;
2.16 4.54012; 2.22 4.41741; 2.28 4.30116; 2.34 4.19088; 2.4 4.08611;
2.46 3.98645; 2.52 3.89153; 2.58 3.80103; 2.64 3.71464; 2.7 3.63209;
2.76 3.55314; 2.82 3.47754; 2.88 3.40509; 2.94 3.3356; 3 3.26889; 3.06
3.20479;
3.12 3.14316; 3.18 3.08385; 3.24 3.02675; 3.3 2.97171; 3.36 2.91865;
3.42 2.86744; 3.48 2.818; 3.54 2.77024; 3.6 2.72407; 3.66 2.67941; 3.72
2.6362;
3.78 2.59435; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643; 4.02 2.45166;
4.08 2.45166; 4.14 2.45166; 4.2 2.45166; 4.26 2.45166; 4.32 2.45166;
4.38 2.45166; 4.44 2.45166; 4.5 2.45166; 4.56 2.45166; 4.62 2.45166;
4.68 2.45166; 4.74 2.45166; 4.8 2.45166; 4.86 2.45166; 4.92 2.45166;
4.98 2.45166; 5.04 2.45166; 5.1 2.45166; 5.16 2.45166; 5.22 2.45166;
5.28 2.45166; 5.34 2.45166; 5.4 2.45166; 5.46 2.45166; 5.52 2.45166;
```

```
5.58 2.45166; 5.64 2.45166; 5.7 2.45166; 5.76 2.45166; 5.82 2.45166;
5.88 2.45166; 5.94 2.45166;
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       3.332                  0.30014
         2                       9.103                  0.10986
         3                      12.435                  0.08042
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN          GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
             1                       2.50000       0.05000
             2                       2.39857       0.05000
             3                       2.10421       0.05000
      STAAD SPACE                                              -- PAGE NO.
8
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                             MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.300      284.57        0.00        0.00        0.00
0.00    -1707.42
      2      0.110       19.60        0.00        0.00        0.00
0.00       58.81
      3      0.080        1.23        0.00        0.00        0.00
0.00       -7.41
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KN
           --------------------------------------         ------------------
 MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z       X         Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000    284.57     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000     19.60     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      1.23     0.00
0.00

---------------------------
```

```
                                   TOTAL SRSS  SHEAR    285.25      0.00
0.00
                                   TOTAL 10PCT SHEAR    285.25      0.00
0.00
                                   TOTAL ABS   SHEAR    305.41      0.00
0.00
```

Related Links

- *TR.32.10.1.3 Response Spectrum Specification per NRC 2010* (on page 2512)
- *Response Spectra tab* (on page 2854)

V. NRC 2010 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static force method in NBCC/NRC 2010.

Problem

Structure to be modelled:

height = 3× 5m = 15m
plan dimension X = 5m
plan dimension Z = 5m

Input parameters:

Seismic acceleration values Sa(0.2) = 0.28, Sa(0.5) = 0.17, Sa(1.0) = 0.11, and Sa(2) = 0.063 (`SA1 0.28`, `SA2 0.17`, `SA3 0.11`, `SA4 0.063`)
Importance factor = 1.3 (`I 1.3`)
Site class C (`SCL 3`)
Higher mode factor X = 1 (`MVX 1`)
Higher mode factor Z = 1 (`MVZ 1`)
Ductility-related force modification factor X = 5 (`RDX 5`)
Ductility-related force modification factor Z = 5 (`RDZ 5`)
Overstrength-related force modification factor X = 5 (`ROX 1.5`)
Overstrength-related force modification factor Z = 5 (`ROZ 1.5`)
Moment resisting frames in both X and Z directions (`STX 1`, `STZ 1`)
Provisions to be checked for member strength (`MD 1`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 6 7 9 To 16. Seismic Load as per NBCC/NRC 2010 specifications are generated along horizontal direction global X and global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

$Ta_x = 0.075\times(hn)^{0.75}$ (for concrete moment frames) $= 0.075 \times 15^{0.75} = 0.57$ sec

$Ta_z = 0.075\times(hn)^{0.75}$ (for concrete moment frames) $= 0.075 \times 15^{0.75} = 0.57$ sec

Tc_x (Rayleigh method from output file) = 0.467, Tc_z (Rayleigh method from output file) = 0.467

$$1.5 \times Ta_x = 0.86 > Tc_x \text{ so } Ta_{used} = Tc_x = 0.467$$

$$1.5 \times Ta_z = 0.86 > Tc_z \text{ so } Ta_{used} = Tc_z = 0.467$$

Fa = 1 (For Site Class = C and Sa(0.2) = 0.28)

Fv = 1 (For Site Class = C and Sa(1.0) = 0.11)

S(0.2) = Fa × Sa(0.2) = 1 × 0.28 = 0.28

S(0.5) = smaller of Fv × Sa(0.5) and Fa × Sa(0.2) = smaller of 1 x 0.17 and 1 × 0.28 = 0.17

$S(Ta_{used})_x = S(Ta_{used})_z$ = 0.17 + ((0.28-0.17)/(0.5-0.2) × (0.5-0.467)) = 0.1821

Vx = $S(Ta_{used})_x$ × MVX × I × W / RDX × ROX = 0.1821 × 1 × 1.3 × (200x3) / 5 × 1.5 = 18.9384 kN

Min Base Shear = S(2.0) × MVX × I × W / (RDX × ROX) = 0.063 × 1 x 1.3 × (200x3) / 5 × 1.5 = 6.552 kN

Max Base Shear = 2/3 × S(0.2) × I × W / (RDX × ROX) = 2/3 × 0.28 x 1.3 × (200x3) / 5 × 1.5 = 19.413 kN

Vx > Min Base Shear & Vx < Max Base Shear, Vx = 18.9384 kN

Vz = $S(Ta_{used})_z$ × MVZ × I × W / RDZ × ROZ = 0.1821 × 1 × 1.3 × (200x3) / 5 × 1.5 = 18.9384 kN

Min Base Shear = S(2.0) × MVZ × I × W / (RDZ × ROZ) = 0.063 × 1 x 1.3 × (200x3) / 5 × 1.5 = 6.552 kN

Max Base Shear = 2/3 × S(0.2) × I × W / (RDZ × ROZ) = 2/3 × 0.28 x 1.3 × (200x3) / 5 × 1.5 = 19.413 kN

Vz > Min Base Shear & Vz < Max Base Shear, Vz = 18.9384 kN

Since fundamental lateral period < 0.7 sec Ft = 0, calculated Base Shear is distributed along the height as under:

Fx = (V - 0) × (Wx × hx / ∑ Wi × hi) = (V) × (Wx × hx / ∑ Wi × hi)

Story Level	Wi	hi	*Wi × hi*	*(Wi × hi)/∑ (Wi × hi)*	Fx	Fz
Roof	200	15	3,000	0.5	9.469	9.469
2nd	200	10	2,000	0. 333	6.313	6.313
1st	200	5	1,000	1.667	3.156	3.156
Summation	600		6,000			
V					18.938	18.938

Comparison

Table 538: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear (kN)	X, 1st	3.156	3.159	Negligible	
	Z, 1st	3.156	3.159	Negligible	
	X, 2nd	6.313	6.318	Negligible	
	Z, 2nd	6.313	6.318	Negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	X, roof	9.469	9.477	Negligible	
	Z, roof	9.469	9.477	Negligible	
Base shear (kN)	Vx	18.938	18.955	Negligible	
	Vz	18.938	18.955	Negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\NRC 2010 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
DEFINE NRC 2010 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1 MVZ 1 RDX 5 RDZ 5 ROX -
1.5 ROZ 1.5 STX 1 STZ 1 MD 1
JOINT WEIGHT
2 3 6 7 9 To 16 WEIGHT 50
LOAD 1 LOADTYPE None  TITLE STATIC_X
NRC LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
NRC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
```

```
PRINT ANALYSIS RESULTS
SECTION 0 0.25 0.5 0.75 1 ALL
PRINT MEMBER SECTION FORCES ALL
FINISH
```

STAAD Output

```
****************************************************************************

*                                                                          *
   * EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2010 ALONG
X *
   *  CT =  0.075 Ta =  0.572 SEC. Tc =  0.467
SEC.                            *
   *  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION =
0.182258            *
   *  EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC
RESPONSE)                      *
   *                    =   0.032 X      600.000 =       18.955
KN            *
   *  DESIGN BASE SHEAR =   1.000 X
18.955                               *
   *                    =       18.955
KN                                     *

*                                                                          *

****************************************************************************

****************************************************************************

*                                                                          *
   * EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2010 ALONG
Z *
   *  CT =  0.075 Ta =  0.572 SEC. Tc =  0.467
SEC.                            *
   *  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION =
0.182258            *
   *  EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC
RESPONSE)                      *
   *                    =   0.032 X      600.000 =       18.955
KN            *
   *  DESIGN BASE SHEAR =   1.000 X
18.955                               *
   *                    =       18.955
KN                                     *

*                                                                          *

****************************************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX        0.790    MY        0.000
       3     FX        0.790    MY        0.000
       6     FX        0.790    MY        0.000
```

```
       7      FX        0.790    MY        0.000
                    -----------       -----------
          TOTAL =       3.159              0.000 AT LEVEL     5.000 METE
       9      FX        1.580    MY        0.000
      10      FX        1.580    MY        0.000
      11      FX        1.580    MY        0.000
      12      FX        1.580    MY        0.000
      STAAD SPACE                                                -- PAGE NO.
4
                    -----------       -----------
          TOTAL =       6.318              0.000 AT LEVEL    10.000 METE
      13      FX        2.369    MY        0.000
      14      FX        2.369    MY        0.000
      15      FX        2.369    MY        0.000
      16      FX        2.369    MY        0.000
                    -----------       -----------
          TOTAL =       9.477              0.000 AT LEVEL    15.000 METE
    JOINT               LATERAL            TORSIONAL              LOAD -     2
                        LOAD (KN   )       MOMENT (KN  -METE) FACTOR -   1.000
    -----               -------            ---------
       2      FZ        0.790    MY        0.000
       3      FZ        0.790    MY        0.000
       6      FZ        0.790    MY        0.000
       7      FZ        0.790    MY        0.000
                    -----------       -----------
          TOTAL =       3.159              0.000 AT LEVEL     5.000 METE
       9      FZ        1.580    MY        0.000
      10      FZ        1.580    MY        0.000
      11      FZ        1.580    MY        0.000
      12      FZ        1.580    MY        0.000
                    -----------       -----------
          TOTAL =       6.318              0.000 AT LEVEL    10.000 METE
      13      FZ        2.369    MY        0.000
      14      FZ        2.369    MY        0.000
      15      FZ        2.369    MY        0.000
      16      FZ        2.369    MY        0.000
                    -----------       -----------
          TOTAL =       9.477              0.000 AT LEVEL    15.000 METE
```

Related Links

- *TR.31.2.4 Canadian Seismic Code (NRC) - 2010* (on page 2362)

V. NRC 2005 Response Spectrum

Calculation of Base Shear for Response Spectrum Method in NBCC/NRC 2005.

Details

Example 4.7(1) is modeled in STAAD.Pro.

The generated model is subjected to a Response Spectrum Load along global X direction. Time-Acceleration pairs for the spectrum data can be seen in supporting excel file `NRC_Calculation.xlsx` within worksheet named RS Values. The direction factor along global X direction is assumed as 0.5. The base shear reported by STAAD.Pro is verified against hand calculation.

Validation

Natural Frequency/Time Period Information & Mode Shapes (Obtained from output file)

Table 539: Natural Frequencies/Time Period

Mode	Frequency (cyc/sec)	Time Period (sec)
1	3.332	0.30014
2	9.103	0.10986
3	12.435	0.08042

Table 540: Mode Shapes

Mode	Story Level/ Joint Numbers	X-Trans	Y-Trans	Z-Trans	X-Rot	Y-Rot	Z-Rot
1	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0
2	Roof/Joints 7 & 8	-1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	0	0	0	0	0	0
	1st Floor/ Joints 2 & 3	1	0	0	0	0	0
3	Roof/Joints 7 & 8	1	0	0	0	0	0
	2nd Floor/ Joints 5 & 6	-0.86603	0	0	0	0	0
	1st Floor/ Joints 2 & 3	0.5	0	0	0	0	0

Table 541: Mode Participation Factor Calculation

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3		
		ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^2$	ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^2$	ϕ	$Wi\ x\ \phi$	$Wi\ x\ \phi^2$
Roof	49.035	1	49.035	49.035	-1	-49.035	49.035	1	49.035	49.035
2nd Floor	98.07	0.86603	84.932	73.553	0	0	0	-0.86603	-84.932	73.553
1st Floor	98.07	0.5	49.035	24.518	1	98.07	98.07	0.5	49.035	24.518
Summation	245.175		183.00	147.11		49.035	147.11		13.138	147.11
Modal Weight Mk x g		227.7			16.345			1.173		
Modal Weight Participation (In %)		92.85			6.667			0.4786		
Mode Participation Factor ∏ ∑Wi x ϕ / ∑ Wi x ϕ ^{2}		1.244			0.3333			0.0893		

Table 542: Horizontal Acceleration Spectrum Value Calculation

	Mode 1	Mode 2	Mode 3
Time Period T	**0.30014**	**0.10986**	**0.08042**
Spectrum Data	Obtain from `NRC_Calculation.xlsx` within worksheet named **RS Values**		
Formula	Linearly Interpolate acceleration values between 0.3 sec & 0.36 sec to obtain acceleration value for 0.30014 sec.	Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.10986 sec.	Linearly Interpolate acceleration values between 0.06 sec & 0.12 sec to obtain acceleration value for 0.08042 sec.
Sa	1.50	1.40	1.10
Horizontal Seismic Coefficient Cs = (Sa x Direction Factor) = Sa x 0.5	**1.** 249998725 1.25	1.20	1.05

Table 543: Base Shear Calculation and its distribution across height

Story Level	Weight Wi (KN)	Mode 1			Mode 2			Mode 3			Story Shear due to all modes i.e. SRSS of all modes
		ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	ϕ	Qi (Cs x ϕ x $\prod$ x Wi)	Vi ($\sum Qi$)	
Roof	49.035	1	76.25	76.25	-1	-19.60	-19.60	1	4.608	4.608	78.86
2nd Floor	98.07	0.86603	132.1	208.3	0	0	-19.60	-0.86603	-7.981	-3.373	209.27
1st Floor	98.07	0.5	76.25	284.6	1	39.21	19.60	0.5	4.608	1.235	285.25
Base Shear V											285.25

Comparison

Table 544: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Base shear (kN)	285.25	285.25	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\NRC 2005 Response Spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 3 3 0; 4 3 0 0; 5 0 6 0; 6 3 6 0; 7 0 9 0; 8 3 9 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6; 7 5 7; 8 6 8; 9 7 8;
START USER TABLE
TABLE 1
UNIT METER KN
PRISMATIC
COLLUMN
```

```
1e+09 0.000847246 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT METER KN
PRISMATIC
BEAM
0.001 1e+09 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
G 9.28139e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TO 5 7 8 UPTABLE 1 COLLUMN
2 6 9 UPTABLE 2 BEAM
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
8 FX 49.035
3 6 FX 98.07
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 3
DIA 2 TYPE RIG HEI 6
DIA 3 TYPE RIG HEI 9
LOAD 1 LOADTYPE None  TITLE RS_X
SPECTRUM SRSS NRC 2005 X 0.5 ACC DAMP 0.05 LIN
0 9.80665; 0.06 18.6326; 0.12 24.5166; 0.18 24.5166; 0.24 24.5166; 0.3
24.5166;
0.36 24.5166; 0.42 23.3492; 0.48 20.4305; 0.54 18.1605; 0.6 16.3444;
0.66 14.8586; 0.72 13.6203; 0.78 12.5726; 0.84 11.6746; 0.9 10.8963;
0.96 10.2153; 1.02 9.61436; 1.08 9.08023; 1.14 8.60233; 1.2 8.17221;
1.26 7.78306; 1.32 7.42928; 1.38 7.10627; 1.44 6.81018; 1.5 6.53777;
1.56 6.28632; 1.62 6.05349; 1.68 5.83729; 1.74 5.63601; 1.8 5.44814;
1.86 5.2724; 1.92 5.10763; 1.98 4.95286; 2.04 4.80718; 2.1 4.66984;
2.16 4.54012; 2.22 4.41741; 2.28 4.30116; 2.34 4.19088; 2.4 4.08611;
2.46 3.98645; 2.52 3.89153; 2.58 3.80103; 2.64 3.71464; 2.7 3.63209;
2.76 3.55314; 2.82 3.47754; 2.88 3.40509; 2.94 3.3356; 3 3.26889; 3.06
3.20479;
3.12 3.14316; 3.18 3.08385; 3.24 3.02675; 3.3 2.97171; 3.36 2.91865;
3.42 2.86744; 3.48 2.818; 3.54 2.77024; 3.6 2.72407; 3.66 2.67941; 3.72
2.6362;
3.78 2.59435; 3.84 2.55382; 3.9 2.51453; 3.96 2.47643; 4.02 2.45166;
4.08 2.45166; 4.14 2.45166; 4.2 2.45166; 4.26 2.45166; 4.32 2.45166;
4.38 2.45166; 4.44 2.45166; 4.5 2.45166; 4.56 2.45166; 4.62 2.45166;
4.68 2.45166; 4.74 2.45166; 4.8 2.45166; 4.86 2.45166; 4.92 2.45166;
4.98 2.45166; 5.04 2.45166; 5.1 2.45166; 5.16 2.45166; 5.22 2.45166;
5.28 2.45166; 5.34 2.45166; 5.4 2.45166; 5.46 2.45166; 5.52 2.45166;
```

```
5.58 2.45166; 5.64 2.45166; 5.7 2.45166; 5.76 2.45166; 5.82 2.45166;
5.88 2.45166; 5.94 2.45166;
PERFORM ANALYSIS PRINT ALL
PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       3.332                  0.30014
         2                       9.103                  0.10986
         3                      12.435                  0.08042
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN KN           GENERALIZED
      MODE           X             Y             Z               WEIGHT
         1       2.276565E+02  0.000000E+00  0.000000E+00    1.471050E+02
         2       1.634500E+01  0.000000E+00  0.000000E+00    1.471050E+02
         3       1.173519E+00  0.000000E+00  0.000000E+00    1.471051E+02
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
     MISSING WEIGHT X Y Z  -3.177132E-06  0.000000E+00  0.000000E+00 KN
       MODAL WEIGHT X Y Z   2.451750E+02  0.000000E+00  0.000000E+00 KN
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
              1                      2.50000       0.05000
              2                      2.39857       0.05000
              3                      2.10421       0.05000
      STAAD SPACE                                              -- PAGE NO.
8
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KN   LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX           FY           FZ           MX
MY           MZ
      1      0.300      284.57         0.00         0.00         0.00
0.00    -1707.42
      2      0.110       19.60         0.00         0.00         0.00
0.00       58.81
      3      0.080        1.23         0.00         0.00         0.00
0.00       -7.41
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KN
           --------------------------------------        ------------------
 MODE    X      Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   92.85  0.00  0.00   92.855    0.000    0.000    284.57     0.00
0.00
   2    6.67  0.00  0.00   99.521    0.000    0.000     19.60     0.00
0.00
   3    0.48  0.00  0.00  100.000    0.000    0.000      1.23     0.00
0.00

---------------------------
```

```
                                    TOTAL SRSS  SHEAR    285.25      0.00
0.00
                                    TOTAL 10PCT SHEAR    285.25      0.00
0.00
                                    TOTAL ABS   SHEAR    305.41      0.00
0.00
```

Related Links

- *TR.32.10.1.2 Response Spectrum Specification per NRC 2005* (on page 2506)
- *Response Spectra tab* (on page 2854)

V. NRC 2005 Static Seismic

Calculation of base shear and its distribution along the height for equivalent static force method in NBCC/NRC 2005.

Problem

Structure to be modelled:

height = 3× 5m = 15m
plan dimension X = 5m
plan dimension Z = 5m

Input parameters:

Seismic acceleration values Sa(0.2) = 0.28, Sa(0.5) = 0.17, Sa(1.0) = 0.11, and Sa(2) = 0.063 (`SA1 0.28`, `SA2 0.17`, `SA3 0.11`, `SA4 0.063`)
Importance factor = 1.3 (`IE 1.3`)
Site class C (`SCL 3`)
Higher mode factor X = 1 (`MVX 1`)
Higher mode factor Z = 1 (`MVZ 1`)
Numerical reduction coefficient for base overturning X = 1 (`JX 1`)
Numerical reduction coefficient for base overturning Z = 1 (`JZ 1`)
Ductility-related force modification factor X = 5 (`RDX 5`)
Ductility-related force modification factor Z = 5 (`RDZ 5`)
Overstrength-related force modification factor X = 5 (`ROX 1.5`)
Overstrength-related force modification factor Z = 5 (`ROZ 1.5`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 6 7 9 To 16. Seismic Load as per NBCC/NRC 2005 specifications are generated along horizontal direction global X and global Z. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Ta_x = 0.075*(hn)^0.75 (for concrete moment frames) = $0.075 \times 15^{0.75}$ = 0.571649342 sec

$Ta_z = 0.075*(hn)^{0.75}$ (for concrete moment frames) = $0.075 \times 15^{0.75}$ = 0.571649342 sec

Tc_x (Rayleigh method from output file) = 0.467, Tc_z (Rayleigh method from output file) = 0.467

$$1.5 \times Ta_x = 0.857474013 > Tc_x \text{ so } Ta_{used} = Tc_x = 0.467$$

$$1.5 \times Ta_z = 0.857474013 > Tc_z \text{ so } Ta_{used} = Tc_z = 0.467$$

Fa = 1 (For Site Class = C and Sa(0.2) = 0.28)

Fv = 1 (For Site Class = C and Sa(1.0) = 0.11)

$$S(0.2) = Fa \times Sa(0.2) = 1 \times 0.28 = 0.28$$

S(0.5) = smaller of Fv × Sa(0.5) and Fa × Sa(0.2) = smaller of 1 × 0.17 and 1 × 0.28 = 0.17

$S(Ta_{used})_x = S(Ta_{used})_z$ = 0.17 + ((0.28-0.17)/(0.5-0.2) × (0.5-0.467)) = 0.1821

Vx = $S(Ta_{used})_x$ × MVX × I × W / RDX × ROX = 0.1821 × 1 × 1.3 × (200×3) / 5 × 1.5 = 18.9384 kN

Min Base Shear = S(2.0) × MVX × I × W / (RDX × ROX) = 0.063 × 1 × 1.3 × (200×3) / 5 × 1.5 = 6.552 kN

Max Base Shear = 2/3 × S(0.2) × I × W / (RDX × ROX) = 2/3 × 0.28 × 1.3 × (200×3) / 5 × 1.5 = 19.413 kN

Vx > Min Base Shear & Vx < Max Base Shear, Vx = 18.9384 kN

Vz = $S(Ta_{used})_z$ × MVZ × I × W / RDZ × ROZ = 0.1821 × 1 × 1.3 × (200×3) / 5 × 1.5 = 18.9384 kN

Min Base Shear = S(2.0) × MVZ × I × W / (RDZ × ROZ) = 0.063 × 1 × 1.3 × (200×3) / 5 × 1.5 = 6.552 kN

Max Base Shear = 2/3 × S(0.2) × I × W / (RDZ × ROZ) = 2/3 × 0.28 × 1.3 × (200×3) / 5 × 1.5 = 19.413 kN

Vz > Min Base Shear & Vz < Max Base Shear, Vz = 18.9384 kN

Since fundamental lateral period < 0.7 sec Ft = 0, calculated Base Shear is distributed along the height as under:

$$Fx = (V - 0) \times (Wx \times hx / \sum Wi \times hi) = (V) \times (Wx \times hx / \sum Wi \times hi)$$

Story Level	Wi	hi	$Wi \times hi$	$(Wi \times hi)/\sum(Wi \times hi)$	Fx	Fz
Roof	200	15	3,000	0.5	9.469	9.469
2nd	200	10	2,000	0.333	6.313	6.313
1st	200	5	1,000	1.667	3.156	3.156
Summation	600		6,000			
V					18.938	18.938

Comparison

Table 545: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear (kN)	X, 1st	3.156	3.159	Negligible	
	Z, 1st	3.156	3.159	Negligible	
	X, 2nd	6.313	6.318	Negligible	
	Z, 2nd	6.313	6.318	Negligible	
	X, roof	9.469	9.477	Negligible	
	Z, roof	9.469	9.477	Negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Base shear (kN)	Vx	18.938	18.955	Negligible	
	Vz	18.938	18.955	Negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\NRC\NRC 2005 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 0 0 5; 6 0 5 5; 7 5 5 5; 8 5 0 5;
9 0 10 0; 10 5 10 0; 11 0 10 5; 12 5 10 5; 13 0 15 0; 14 5 15 0; 15 0 15 5;
16 5 15 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 9 13; 18 10 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 24 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
*DEFINE NRC 2010 LOAD
*SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 I 1.3 SCL 3 MVX 1 MVZ 1 RDX 5 RDZ 5 -
*ROX 1.5 ROZ 1.5 STX 1 STZ 1 MD 1
DEFINE NRC 2005 LOAD
SA1 0.28 SA2 0.17 SA3 0.11 SA4 0.063 IE 1.3 SCLASS 3 MVX 1 MVZ 1 JX 1 JZ 1 -
RDX 5 RDZ 5 ROX 1.5 ROZ 1.5
JOINT WEIGHT
2 3 6 7 9 TO 16 WEIGHT 50
LOAD 1 LOADTYPE None  TITLE STATIC_X
NRC LOAD X 1
LOAD 2 LOADTYPE None  TITLE STATIC_Z
NRC LOAD Z 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
SECTION 0 0.25 0.5 0.75 1 ALL
```

```
PRINT MEMBER SECTION FORCES ALL
FINISH
```

STAAD Output

```
*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2005 ALONG
X *
   *  CT =  0.073 Ta =  0.557 SEC. Tc =  0.467
SEC.                            *
   *  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION =
0.182258          *
   *  EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC
RESPONSE)                      *
   *                   =   0.032 X      600.000 =       18.955
KN            *
   *  DESIGN BASE SHEAR =   1.000 X
18.955                             *
   *                   =       18.955
KN                                  *

*                                                                           *

*****************************************************************************

*****************************************************************************

*                                                                           *
   * EQUIV. SEISMIC LOADS AS PER NATIONAL BUILDING CODE OF CANADA 2005 ALONG
Z *
   *  CT =  0.073 Ta =  0.557 SEC. Tc =  0.467
SEC.                            *
   *  T USED =  0.467 SEC. DESIGN SPECTRAL ACCELERATION =
0.182258          *
   *  EQUIVALENT LATERAL SEISMIC FORCE (ELASTIC
RESPONSE)                      *
   *                   =   0.032 X      600.000 =       18.955
KN            *
   *  DESIGN BASE SHEAR =   1.000 X
18.955                             *
   *                   =       18.955
KN                                  *

*                                                                           *

*****************************************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX        0.790    MY        0.000
       3     FX        0.790    MY        0.000
       6     FX        0.790    MY        0.000
       7     FX        0.790    MY        0.000
                 -----------        -----------
```

```
        TOTAL =      3.159              0.000 AT LEVEL     5.000 METE
    9     FX         1.580    MY        0.000
   10     FX         1.580    MY        0.000
   11     FX         1.580    MY        0.000
   12     FX         1.580    MY        0.000
    STAAD SPACE                                              -- PAGE NO.
4
                  -----------        -----------
        TOTAL =      6.318              0.000 AT LEVEL    10.000 METE
   13     FX         2.369    MY        0.000
   14     FX         2.369    MY        0.000
   15     FX         2.369    MY        0.000
   16     FX         2.369    MY        0.000
                  -----------        -----------
        TOTAL =      9.477              0.000 AT LEVEL    15.000 METE
  JOINT              LATERAL            TORSIONAL           LOAD -    2
                     LOAD (KN   )       MOMENT (KN  -METE) FACTOR -   1.000
  -----              -------            ---------
    2     FZ         0.790    MY        0.000
    3     FZ         0.790    MY        0.000
    6     FZ         0.790    MY        0.000
    7     FZ         0.790    MY        0.000
                  -----------        -----------
        TOTAL =      3.159              0.000 AT LEVEL     5.000 METE
    9     FZ         1.580    MY        0.000
   10     FZ         1.580    MY        0.000
   11     FZ         1.580    MY        0.000
   12     FZ         1.580    MY        0.000
                  -----------        -----------
        TOTAL =      6.318              0.000 AT LEVEL    10.000 METE
   13     FZ         2.369    MY        0.000
   14     FZ         2.369    MY        0.000
   15     FZ         2.369    MY        0.000
   16     FZ         2.369    MY        0.000
                  -----------        -----------
        TOTAL =      9.477              0.000 AT LEVEL    15.000 METE
```

Related Links

- *TR.31.2.3 Canadian Seismic Code (NRC) – 2005 Volume 1* (on page 2358)

V. UBC

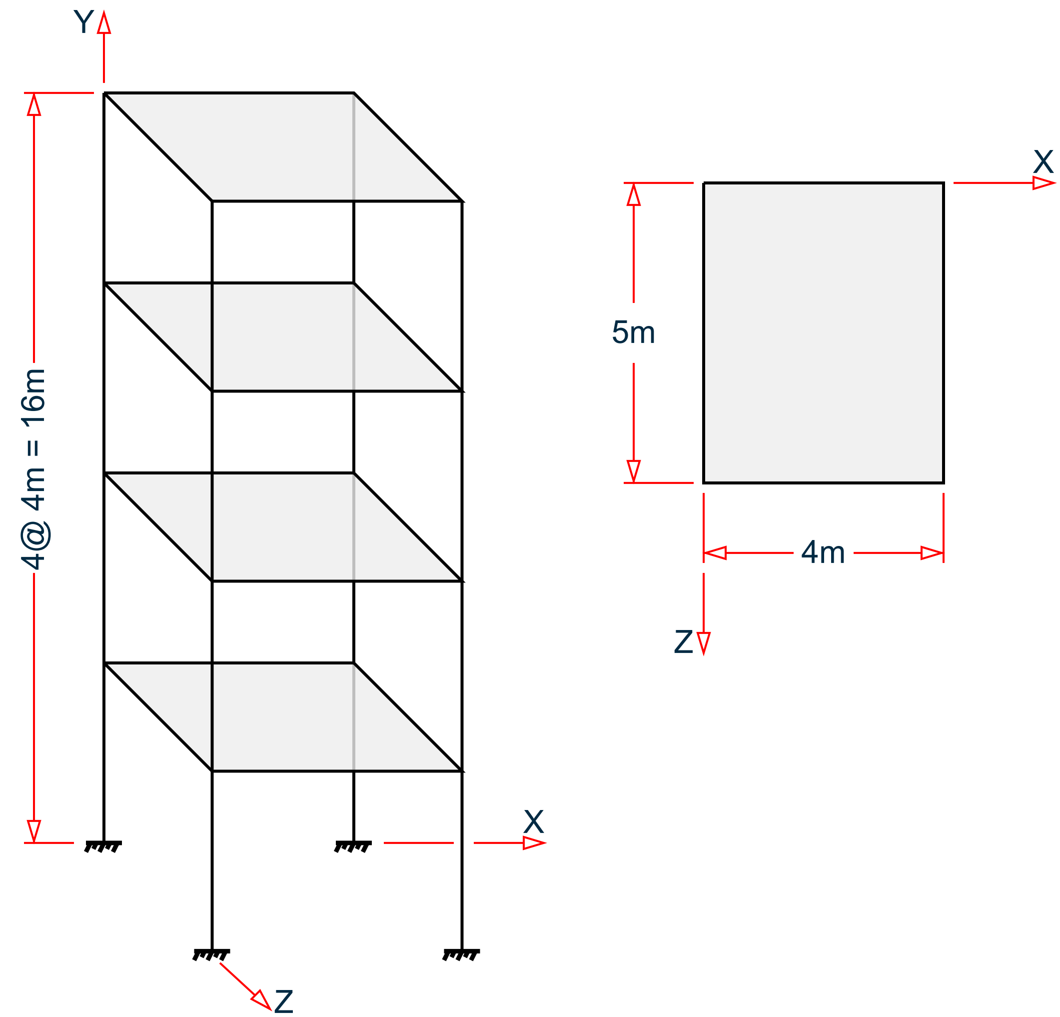
V. UBC 1997 Static Seismic

Calculation of base shear and its distribution along the height for equivalent lateral force method in UBC 1997.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m



Input parameters:

Zone = 0.4
Importance factor = 1.0
Response modification factor X = 3.5
Response modification factor Z = 3.5
Site type = 3
Building period coefficient, C_T = 0.0731
Near source factor, N_a = 1.5
Near source factor, N_v = 2

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per UBC 1997 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

Structure period calculation:

A. Height = $hn = 16m$, $Ta = C_T \times hn^x = 0.0731 \times 16^{0.75} = 0.585$,

B. Tb (from output file) = 1.286

Since Zone = 0.4 & Tb > 1.3 x Ta; T = Ta = 0.585

For STYP = 3 & Zone = 0.4; Ca = 0.4 x NA = 0.4 x 1.5 = 0.6

For STYP = 3 & Zone = 0.4; Cv = 0.56 x NV = 0.56 x 2 = 1.12

Design base shear calculation:

$$V = Cv/R \times I/T \times W = 1.12/3.5 \times 1/0.585 \times (200 \times 4) = 0.547 \times 800 = 437.6 \text{ kN}$$

$$V_{\text{Upper Limit}} = (2.5 \times Ca \times I)/R \times W = (2.5 \times 0.6 \times 1)/3.5 \times (200 \times 4) = 0.4286 \times 800 = 342.88 \text{ kN}$$

$$V_{\text{lower limit}} = \max \left| \begin{array}{l} 0.11 \times C_a \times I \times W = 0.11 \times 0.6 \times 1 \times (200 \times 4) = 0.066 \times 800 = 52.8 \text{ kN} \\ 0.8 \times Z \times N_v \times I / R \times W = 0.8 \times 0.4 \times 2 \times 1 / 3.5 \times (200 \times 4) \\ \quad = 0.1829 \times 800 = 146.3 \text{ kN} \end{array} \right.$$

Hence Design Base Shear V = 342.88 kN

Lateral seismic force $Fx = (W_x \times h_x \,/\, \sum W_i \times h_i) \times (V - F_t)$

Since $T = 0.585 < 0.7$ sec, $F_t = 0$

Story Level	W_i	h_i	$W_i \times h_i$	$(W_i \times h_i)/\sum(W_i \times h_i)$	F_x
Roof	200	16	3,200	0.4	137.15
3rd	200	12	2,400	0.3	102.86
2nd	200	8	1,600	0.2	68.58
1st	200	4	800	0.1	34.29
Summation	800		8,000	1	
V_x	342.88				

Comparison

Table 546: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	34.29	34.286	negligible	
	2nd	68.58	68.571	negligible	
	3rd	102.86	102.857	negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	roof	137.15	137.143	negligible	
Base shear, V_x (kN)		342.88	342.86	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\UBC\UBC 1997 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE UBC LOAD
ZONE 0.4 I 1 RWX 3.5 RWZ 3.5 STYP 3 CT 0.0731 NA 1.5 NV 2
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
UBC LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
```

```
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------
   **********************************************************
   *                                                        *
   *  X  DIRECTION : Ta =  0.585 Tb =  1.286 Tuser =  0.000 *
   *  T =  0.585, LOAD FACTOR =  1.000                      *
   * UBC TYPE = 97                                          *
   * UBC FACTOR V = 0.4286 x        800.00 =     342.86 KN  *
   *                                                        *
   **********************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
    -----            -------          ---------
       2     FX         8.571    MY        0.000
       3     FX         8.571    MY        0.000
       8     FX         8.571    MY        0.000
       9     FX         8.571    MY        0.000
                 -----------        -----------
          TOTAL =     34.286               0.000 AT LEVEL     4.000 METE
       5     FX        17.143    MY        0.000
       6     FX        17.143    MY        0.000
      11     FX        17.143    MY        0.000
      12     FX        17.143    MY        0.000
                 -----------        -----------
          TOTAL =     68.571               0.000 AT LEVEL     8.000 METE
      13     FX        25.714    MY        0.000
      14     FX        25.714    MY        0.000
      15     FX        25.714    MY        0.000
      16     FX        25.714    MY        0.000
                 -----------        -----------
          TOTAL =    102.857               0.000 AT LEVEL    12.000 METE
      17     FX        34.286    MY        0.000
      18     FX        34.286    MY        0.000
      19     FX        34.286    MY        0.000
      20     FX        34.286    MY        0.000
                 -----------        -----------
          TOTAL =    137.143               0.000 AT LEVEL    16.000 METE
```

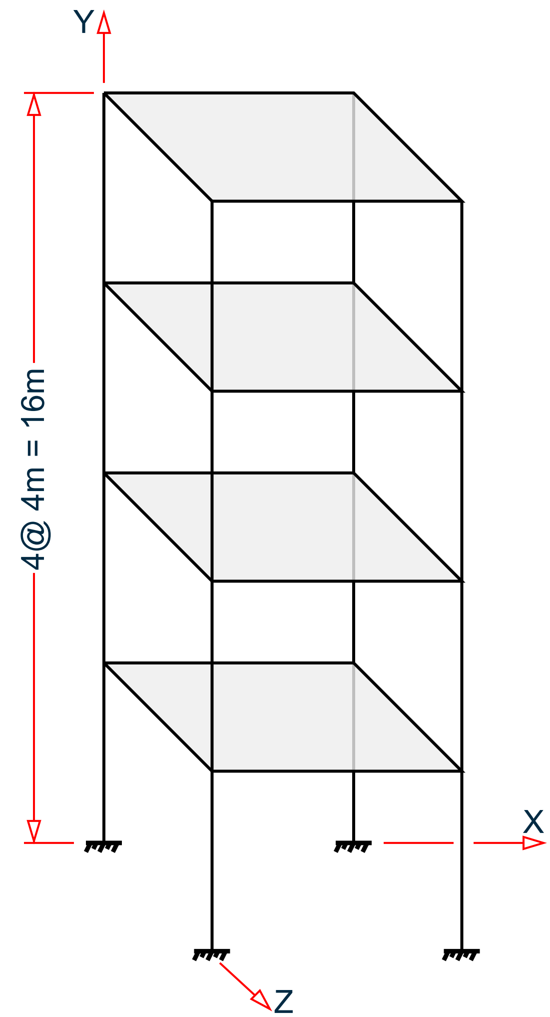
V. UBC 1994 Static Seismic

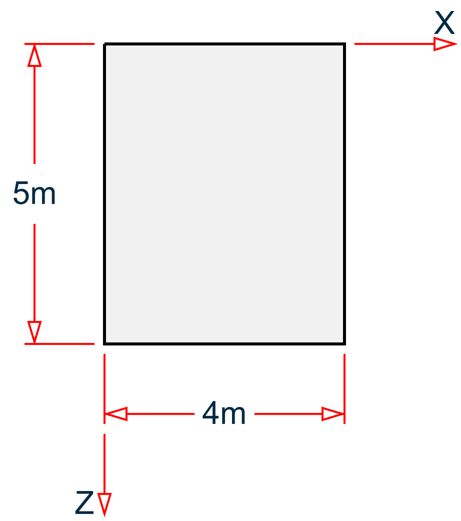
Calculation of base shear and its distribution along the height for equivalent lateral force method in UBC 1994.

Problem

Structure to be modelled:

height = 4× 4m = 16m
plan dimension X = 4m
plan dimension Z = 5m





Input parameters:

Zone = 0.4
Site coefficient, S = 1
Building period coefficient, C_T = 0.0731
Importance factor = 1.0 (`I 1.0`)
Response modification factor X = 3 (`RX 3`)
Response modification factor Z = 3 (`RX 3`)

The seismic weight for each floor is assumed as 200 kN applied as joint weight of 50kN to nodes 2 3 5 6 8 9 11 To 20. Seismic Load as per UBC 1994 specifications are generated along horizontal direction global X. The base shear and its distribution along height reported by STAAD.Pro is verified against hand calculation.

Calculations

Structure period calculation:

A. Height = $hn = 16m, Ta = C_T \times hn^x = 0.0731 \times 16^{0.75} = 0.585,$
B. Tb (from output file) = 1.286

T_{used} = 0.585

$$C = 1.25 \times S / (T^{2/3}) = 1.25 \times 1 / (0.585^{2/3}) = 1.787$$

Design base shear calculation:

$$V = (Z \times I \times C / R_{wx}) \times W = (0.4 \times 1 \times 1.787 / 3) \times (200 \times 4) = 0.2383 \times 800 = 190.62 \text{ kN}$$

Lateral seismic force $Fx = (W_x \times h_x \ / \ \sum W_i \times h_i) \times (V - F_t)$
Since $T = 0.585 < 0.7\ sec, F_t = 0$

Story Level	W_i	h_i	$W_i \times h_i$	$(W_i \times h_i)/\sum(W_i \times h_i)$	F_x
Roof	200	16	3,200	0.4	76.25
3rd	200	12	2,400	0.3	57.19
2nd	200	8	1,600	0.2	38.12
1st	200	4	800	0.1	19.06
Summation	800		8,000	1	
V_x	190.62				

Comparison

Table 547: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Story shear, F_x (kN)	1st	19.06	19.062	negligible	
	2nd	38.12	38.124	negligible	
	3rd	57.19	57.200	negligible	
	roof	76.25	76.267	negligible	
Base shear, V_x (kN)		190.62	190.67	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\UBC\UBC 1994 Static Seismic.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-May-15
```

```
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 4 4 0; 4 4 0 0; 5 0 8 0; 6 4 8 0; 7 0 0 5; 8 0 4 5;
9 4 4 5; 10 4 0 5; 11 0 8 5; 12 4 8 5; 13 0 12 0; 14 4 12 0; 15 0 12 5;
16 4 12 5; 17 0 16 0; 18 4 16 0; 19 0 16 5; 20 4 16 5;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 2 8; 8 3 9; 9 5 11; 10 6 12;
11 7 8; 12 8 9; 13 9 10; 14 8 11; 15 11 12; 16 12 9; 17 5 13; 18 6 14;
19 11 15; 20 12 16; 21 13 14; 22 13 15; 23 14 16; 24 15 16; 25 13 17; 26 14
18;
27 15 19; 28 16 20; 29 17 18; 30 17 19; 31 18 20; 32 19 20;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 4 6 11 13 14 16 TO 20 25 TO 28 PRIS YD 0.3 ZD 0.3
2 5 7 TO 10 12 15 21 TO 24 29 TO 32 PRIS YD 0.3 ZD 0.25
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 7 10 FIXED
DEFINE UBC LOAD
ZONE 0.4 I 1 RWX 3 RWZ 3 CT 0.0731 S 1
JOINT WEIGHT
2 3 5 6 8 9 11 TO 20 WEIGHT 50
*SELFWEIGHT 1
*MEMBER WEIGHT
*2 5 7 TO 10 12 15 UNI 10
LOAD 1 LOADTYPE Seismic  TITLE EL-X DIR
UBC LOAD X 1
PERFORM ANALYSIS PRINT LOAD DATA
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
   LOADING     1  LOADTYPE SEISMIC  TITLE EL-X DIR
   -----------
   ***********************************************************
   *                                                         *
   *  X  DIRECTION : Ta =  0.585 Tb =  1.286 Tuser =  0.000  *
   *  C =   1.7875, LOAD FACTOR =  1.000                     *
   * UBC TYPE = 94                                           *
   * UBC FACTOR V = 0.2383 x        800.00 =     190.67 KN   *
   *                                                         *
   ***********************************************************
    JOINT            LATERAL          TORSIONAL            LOAD -    1
                     LOAD (KN  )      MOMENT (KN  -METE) FACTOR -   1.000
```

```
    -----             -------          ---------
       2     FX         4.767    MY        0.000
       3     FX         4.767    MY        0.000
       8     FX         4.767    MY        0.000
       9     FX         4.767    MY        0.000
                  -----------        -----------
          TOTAL =      19.067              0.000 AT LEVEL     4.000 METE
       5     FX         9.533    MY        0.000
       6     FX         9.533    MY        0.000
      11     FX         9.533    MY        0.000
      12     FX         9.533    MY        0.000
                  -----------        -----------
          TOTAL =      38.133              0.000 AT LEVEL     8.000 METE
      13     FX        14.300    MY        0.000
      14     FX        14.300    MY        0.000
      15     FX        14.300    MY        0.000
      16     FX        14.300    MY        0.000
                  -----------        -----------
          TOTAL =      57.200              0.000 AT LEVEL    12.000 METE
      17     FX        19.067    MY        0.000
      18     FX        19.067    MY        0.000
      19     FX        19.067    MY        0.000
      20     FX        19.067    MY        0.000
                  -----------        -----------
          TOTAL =      76.267              0.000 AT LEVEL    16.000 METE
```

V. Wind Load

V. ASCE 7-16 Wind Load Generation on Building

Verify the windward loads due to wind on a portion of a building structure calculated per the ASCE 7-16 specification.

Details

The wind load is calculated for a building structure using the direction procedure for MWFRS of enclosed buildings (ASCE 7-16, Chapter 27). The following assumptions and parameters apply:

Flat roof type
Category II building (Table 1.5-1)
Basic wind speed = 108 mph
Ground height above sea level = 230 ft
Structure type of “building”
Exposure Category B
Do not consider wind speed-up over hills or escarpments
Building height = 40 ft
Building length along the wind direction, L = 30 ft
Building width normal to wind direction, B = 25 ft
Natural frequency of structure = 2 Hz (i.e., a “rigid” building per Cl. 26.2)
Damping ratio = 0.01

Enclosed building

The portion of the building below 20' in height is blocked by an adjacent structure or other obstruction. Calculate the resulting joint loads on the *windward* side due to wind along the X axis at the upper wall of one segment of the building.

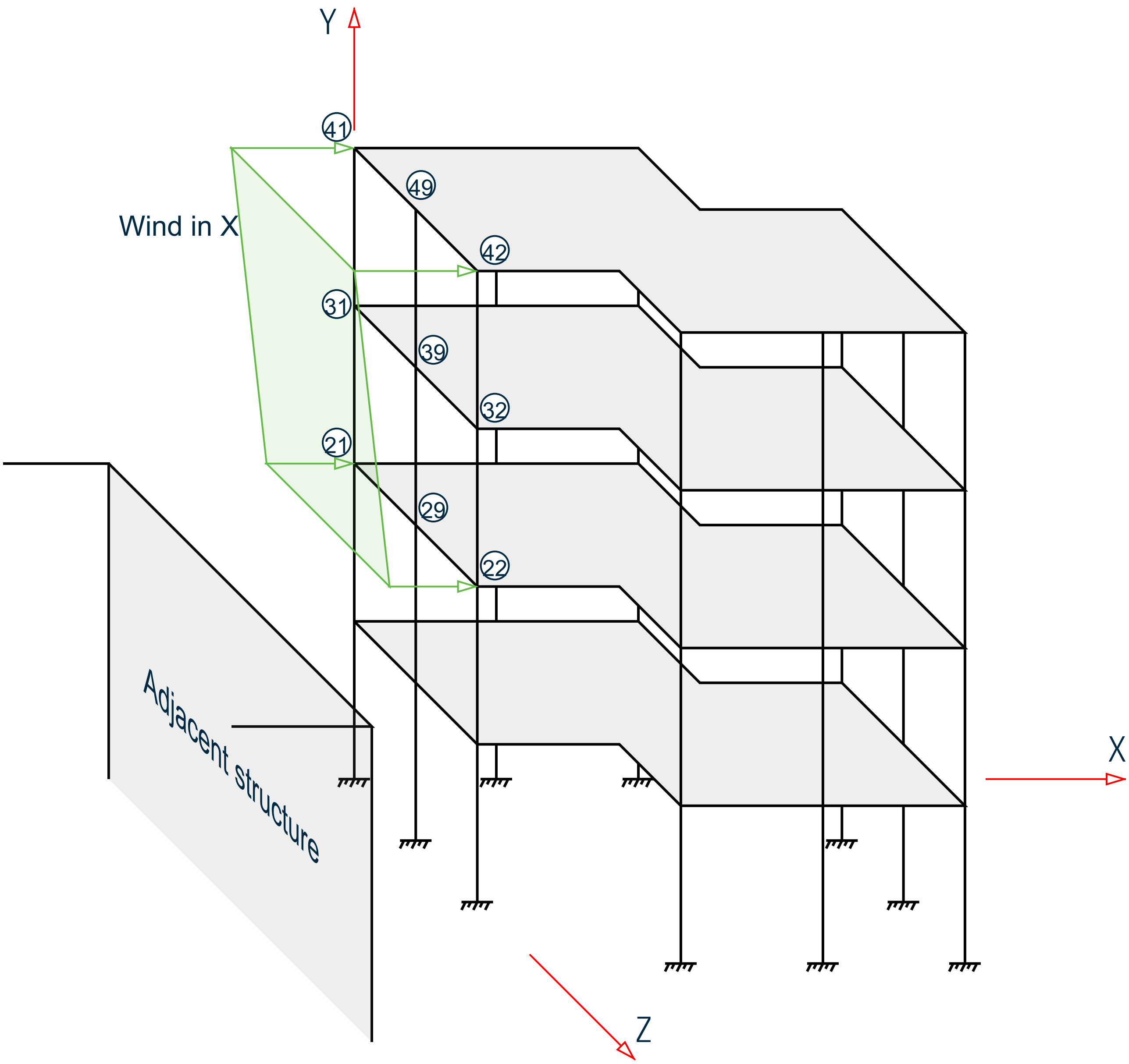


Figure 437: Building structure with portion of wind load

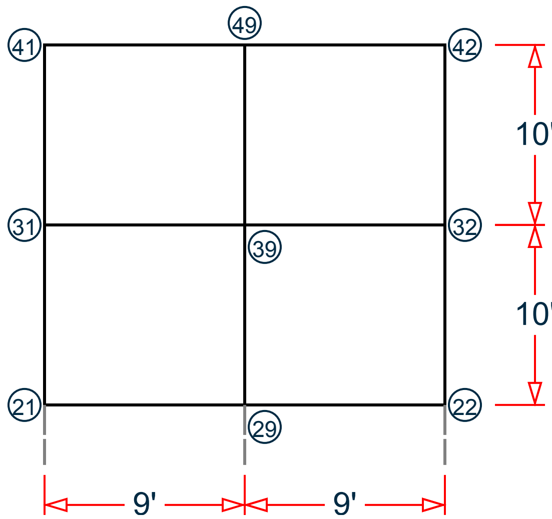


Figure 438: Elevation of loaded area investigated

Validation

Directionality factor, $K_d = 0.85$ (Table 26.6-1)

Topographic factor, $K_{zt} = 1.0$

Ground elevation factor, $K_e = e^{-0.0000362 \times 230} = 0.9917$ (Table 26.9-1)

The velocity pressure exposure coefficients (K^z) are calculated per the formulae given in Table 26.10-1:

$$K_z = \begin{cases} 2.01(15/z_g)^{2/a} & \text{for } z < 15 \text{ ft} \\ 2.01(z/z_g)^{2/a} & \text{for } 15 \text{ ft } \leq z \leq zg \end{cases} \qquad \text{(Table 26.10.1)}$$

where

a	=	7
z_g	=	1,200 ft

See the table below for the values of K^z *at discrete values of z.*

Velocity pressure, q_z

$$q_z = 0.00256 \times K_z K_{zt} K_d K_e V^2 \qquad \text{(Cl. 26.10-1)}$$

Mean roof height = height of the building = 40 ft. Refer to table below for calculation of q_z at the mean roof height.

Gust effect factor, G

G is calculated per Cl. 26.11.4 for rigid structures:

$$G = 0.925\left(\frac{1 + 1.7 g_q I_{\bar{z}} Q}{1 + 1.7 g_v I_{\bar{z}}}\right) \qquad \text{(Cl. 26.11.4)}$$

where

$g_q = g_v$	=	3.4
$\bar{z}$	=	$max \begin{vmatrix} 30 \text{ ft} \\ 0.6 \times h \end{vmatrix} = 30 \text{ ft}$

ε	=	1/3
ℓ	=	320 ft
c	=	0.3
$L_{\bar{z}}$	=	$\ell(\bar{z} / 33)^{\varepsilon} = 309.99\ ft$
Q	=	$\sqrt{\dfrac{1}{1 + 0.63\left(\dfrac{B+h}{L_{\bar{z}}}\right)^{0.63}}} = \sqrt{\dfrac{1}{1 + 0.63\left(\dfrac{25+40}{309.99}\right)^{0.63}}} = 0.8997$
$I_{\bar{z}}$	=	$c(33 / \bar{z})^{1/6} = 0.3048$

Therefore, $G = 0.925\left(\dfrac{1 + 1.7 \times 3.4 \times 0.3048 \times 0.8997}{1 + 1.7 \times 3.4 \times 0.3048}\right) = 0.8658$

Wall pressure coefficient, $C_p = 0.8$ for windward per Fig. 27.3-1.

The external design pressure, $p = q_h G C_p$.

The internal pressure coefficient, $GC_{pi} = -0.18$ for partially enclosed buildings per Table 26.13-1.

The internal design pressure, $p_i = q_i(GC_{pi}) = 19.14 \times -0.18 = -3.45\ lb/ft^2$

where

q_i	=	the velocity pressure evaluated at the building height, h, = q_h

The design wind pressure, p, is then calculated as:

$$p = q_z G C_p - q_i(GC_{pi}) \quad \text{(Eq. 29.4-1)}$$

Table 548: Wind intensity versus height for building structure

h (ft)	K_z	q_z (lb/ft^2)	p (lb/ft^2)
0	0.575	14.47	13.47
15	0.575	14.47	13.47
20	0.624	15.71	14.32
25	0.665	16.74	15.04
30	0.701	17.63	15.66
35	0.732	18.43	16.21
40	0.761	19.14	16.71

Note: To compare different wind load calculations with STAAD.Pro, the tributary areas below are selected based on the applied wind area defined in the model. These do *not* include any surrounding wind areas outside of the indicated area, which would have wind applied in a typical situation.

Load on Nodes 21 and 22

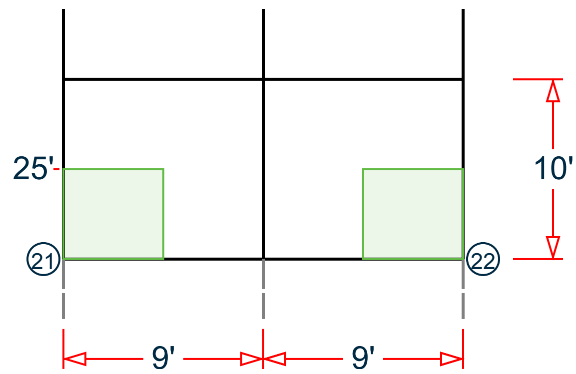


Figure 439: Tributary area for nodes 21 and 22

Wind pressure at half-way between this node and level above (25') = 15.04 lb/ft^2.

$$F = (9\text{ ft} / 2) \times (10\text{ ft} / 2) \times 15.04\text{ lb/ft}^2\ (10)^{-3} = 0.338\text{ kips}$$

<u>Load on Nodes 31 and 32</u>

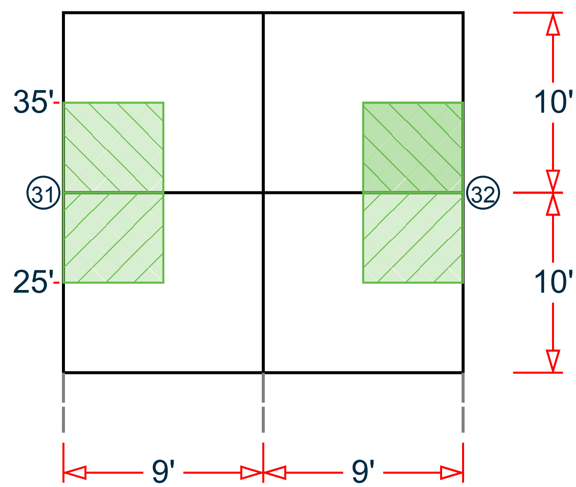


Figure 440: Tributary area for nodes 31 and 32

Wind pressure at half-way between this node and level below (25') = 15.04 lb/ft^2.

Wind pressure at half-way between this node and level above (35') = 16.21 lb/ft^2.

$$F = (9\text{ ft} / 2) \times (10\text{ ft} / 2) \times (15.04\text{ lb/ft}^2 + 16.21\text{ lb/ft}^2)\ (10)^{-3} = 0.703\text{ kips}$$

<u>Load on Nodes 41 and 42</u>

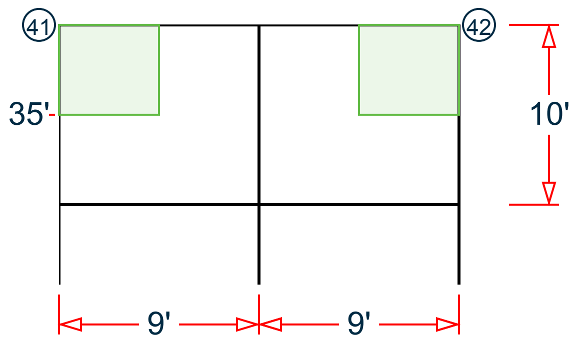


Figure 441: Tributary area for nodes 41 and 42

Wind pressure at roof level (40') = 16.71 lb/ft^2.

$$F = (9\ \text{ft} / 2) \times (10\ \text{ft} / 2) \times (16.71\ \text{lb/ft}^2)\ (10)^{-3} = 0.376\ \text{kips}$$

Load on Node 29

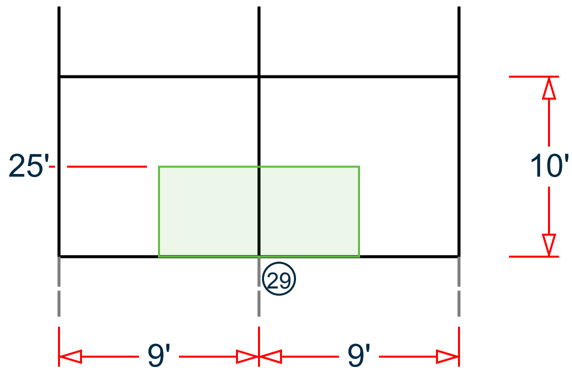


Figure 442: Tributary area for node 29

Wind pressure at half-way between this node and level above (25') = 15.04 lb/ft^2.

$$F = (9\ \text{ft}) \times (10\ \text{ft} / 2) \times 15.04\ \text{lb/ft}^2\ (10)^{-3} = 0.677\ \text{kips}$$

Load on Node 39

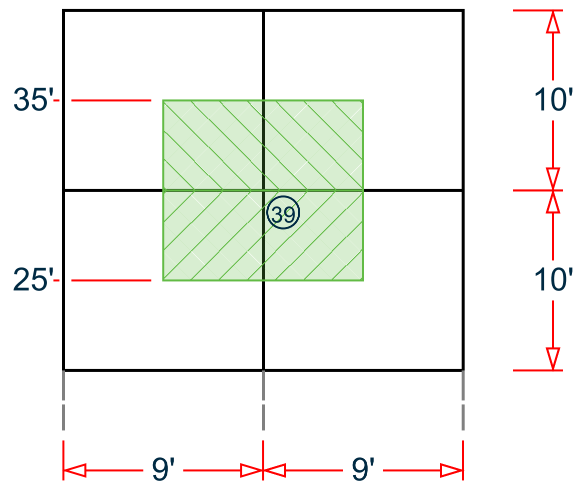


Figure 443: Tributary area for node 39

Wind pressure at half-way between this node and level below (25') = 15.04 lb/ft^2.

Wind pressure at half-way between this node and level above (35') = 16.21 lb/ft^2.

$$F = (9 \text{ ft}) \times (10 \text{ ft} / 2) \times (15.04 \text{ lb/ft}^2 + 16.21 \text{ lb/ft}^2)\ (10)^{-3} = 1.406 \text{ kips}$$

<u>Load on Node 49</u>

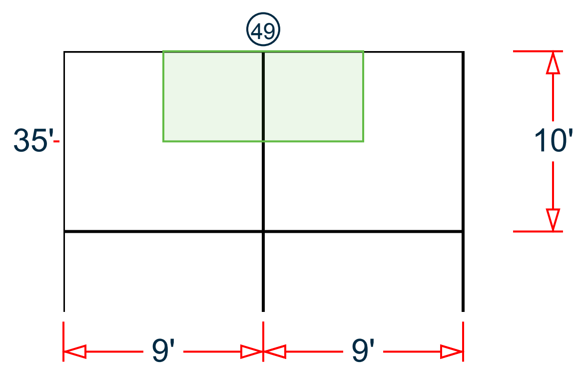


Figure 444: Tributary area for node 49

Wind pressure at roof level (40') = 16.71 lb/ft^2.

$$F = (9 \text{ ft}) \times (10 \text{ ft} / 2) \times (16.71 \text{ lb/ft}^2)\ (10)^{-3} = 0.752 \text{ kips}$$

Results

Table 549: Joint loads due to wind (kips)

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 21	0.338	0.33	negligible	
Node 31	0.703	0.71	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 41	0.376	0.37	negligible	
Node 29	0.677	0.67	negligible	
Node 39	1.406	1.42	negligible	
Node 49	0.752	0.74	negligible	
Node 22	0.338	0.33	negligible	
Node 32	0.703	0.71	negligible	
Node 42	0.376	0.37	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\ASCE 7-16 Wind Load Generation on Building.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jul-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 0 216; 3 324 0 216; 4 216 0 0; 5 108 0 0; 6 108 0 324;
7 324 0 324; 8 324 0 108; 9 0 0 108; 10 216 0 324; 11 0 120 0; 12 0 120 216;
13 324 120 216; 14 216 120 0; 15 108 120 0; 16 108 120 324; 17 324 120 324;
18 324 120 108; 19 0 120 108; 20 216 120 324; 21 0 240 0; 22 0 240 216;
23 324 240 216; 24 216 240 0; 25 108 240 0; 26 108 240 324; 27 324 240 324;
28 324 240 108; 29 0 240 108; 30 216 240 324; 31 0 360 0; 32 0 360 216;
33 324 360 216; 34 216 360 0; 35 108 360 0; 36 108 360 324; 37 324 360 324;
38 324 360 108; 39 0 360 108; 40 216 360 324; 41 0 480 0; 42 0 480 216;
43 324 480 216; 44 216 480 0; 45 108 480 0; 46 108 480 324; 47 324 480 324;
48 324 480 108; 49 0 480 108; 50 216 480 324; 51 108 120 216; 52 216 120 216;
53 108 120 108; 54 216 120 108; 55 108 240 216; 56 216 240 216; 57 108 240
108;
58 216 240 108; 59 108 360 216; 60 216 360 216; 61 108 360 108; 62 216 360
108;
63 108 480 216; 64 216 480 216; 65 108 480 108; 66 216 480 108;
MEMBER INCIDENCES
1 11 19; 2 12 51; 3 11 15; 4 15 53; 5 16 20; 6 17 13; 7 18 54; 8 14 54; 9 1
11;
10 2 12; 11 3 13; 12 4 14; 13 5 15; 14 6 16; 15 7 17; 16 8 18; 17 9 19;
18 10 20; 19 21 29; 20 22 55; 21 21 25; 22 25 57; 23 26 30; 24 27 23; 25 28
58;
26 24 58; 27 11 21; 28 12 22; 29 13 23; 30 14 24; 31 15 25; 32 16 26; 33 17
27;
34 18 28; 35 19 29; 36 20 30; 37 31 39; 38 32 59; 39 31 35; 40 35 61; 41 36
40;
42 37 33; 43 38 62; 44 34 62; 45 21 31; 46 22 32; 47 23 33; 48 24 34; 49 25
35;
```

```
50 26 36; 51 27 37; 52 28 38; 53 29 39; 54 30 40; 55 41 49; 56 42 63; 57 41
45;
58 45 65; 59 46 50; 60 47 43; 61 48 66; 62 44 66; 63 31 41; 64 32 42; 65 33
43;
66 34 44; 67 35 45; 68 36 46; 69 37 47; 70 38 48; 71 39 49; 72 40 50; 73 19
12;
74 51 52; 75 52 13; 76 15 14; 77 53 51; 78 51 16; 79 20 17; 80 13 18; 81 54
53;
82 53 19; 83 54 52; 84 52 20; 85 29 22; 86 55 56; 87 56 23; 88 25 24; 89 57
55;
90 55 26; 91 30 27; 92 23 28; 93 58 57; 94 57 29; 95 58 56; 96 56 30; 97 39
32;
98 59 60; 99 60 33; 100 35 34; 101 61 59; 102 59 36; 103 40 37; 104 33 38;
105 62 61; 106 61 39; 107 62 60; 108 60 40; 109 49 42; 110 63 64; 111 64 43;
112 45 44; 113 65 63; 114 63 46; 115 50 47; 116 43 48; 117 66 65; 118 65 49;
119 66 64; 120 64 50;
DEFINE PMEMBER
1 73 PMEMBER 9
2 74 75 PMEMBER 10
3 76 PMEMBER 11
4 77 78 PMEMBER 12
5 79 PMEMBER 13
6 80 PMEMBER 14
7 81 82 PMEMBER 15
8 83 84 PMEMBER 16
9 PMEMBER 17
10 PMEMBER 18
11 PMEMBER 19
12 PMEMBER 20
13 PMEMBER 21
14 PMEMBER 22
15 PMEMBER 23
16 PMEMBER 24
17 PMEMBER 25
18 PMEMBER 26
19 85 PMEMBER 27
20 86 87 PMEMBER 28
21 88 PMEMBER 29
22 89 90 PMEMBER 30
23 91 PMEMBER 31
24 92 PMEMBER 32
25 93 94 PMEMBER 33
26 95 96 PMEMBER 34
27 PMEMBER 35
28 PMEMBER 36
29 PMEMBER 37
30 PMEMBER 38
31 PMEMBER 39
32 PMEMBER 40
33 PMEMBER 41
34 PMEMBER 42
35 PMEMBER 43
36 PMEMBER 44
37 97 PMEMBER 45
38 98 99 PMEMBER 46
39 100 PMEMBER 47
40 101 102 PMEMBER 48
41 103 PMEMBER 49
```

```
42 104 PMEMBER 50
43 105 106 PMEMBER 51
44 107 108 PMEMBER 52
45 PMEMBER 53
46 PMEMBER 54
47 PMEMBER 55
48 PMEMBER 56
49 PMEMBER 57
50 PMEMBER 58
51 PMEMBER 59
52 PMEMBER 60
53 PMEMBER 61
54 PMEMBER 62
55 109 PMEMBER 63
56 110 111 PMEMBER 64
57 112 PMEMBER 65
58 113 114 PMEMBER 66
59 115 PMEMBER 67
60 116 PMEMBER 68
61 117 118 PMEMBER 69
62 119 120 PMEMBER 70
63 PMEMBER 71
64 PMEMBER 72
65 PMEMBER 73
66 PMEMBER 74
67 PMEMBER 75
68 PMEMBER 76
69 PMEMBER 77
70 PMEMBER 78
71 PMEMBER 79
72 PMEMBER 80
DEFINE MATERIAL START
ISOTROPIC 3000PSI
E 3320.56
POISSON 0.2
DENSITY 8.68056e-05
ALPHA 5.55556e-06
DAMP 0.05
G 1383.57
TYPE CONCRETE
STRENGTH FCU 3
END DEFINE MATERIAL
PMEMBER PROPERTY
9 TO 16 27 TO 34 45 TO 52 63 TO 70 PRIS YD 18 ZD 12
17 TO 26 35 TO 44 53 TO 62 71 TO 80 PRIS YD 18 ZD 18
PMEMBER CONSTANTS
MATERIAL 3000PSI ALL
SUPPORTS
1 TO 10 FIXED
DEFINE WIND LOAD
TYPE 1 ASCE7:16[+X]
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2016:PARAMS 108.000 MPH 230.000 FT 0 1 1 0 0.000 FT 0.000 FT 0.000 FT -
0 3 40.000 FT 30.000 FT 25.000 FT 2.000 0.010 0 0 0 0 0 0 0.761 1.000 1.000 -
0.850 0.992 0 0 0 0 0.866 0.800 -0.180
!> END GENERATED DATA BLOCK
INT 9.35123e-05 9.35123e-05 9.59522e-05 9.82014e-05 0.000100292 0.000102249 -
0.00010409 0.000105832 0.000107485 0.000109061 0.000110567 0.00011201 -
```

```
0.000113396 0.000114731 0.000116018 0.000116018 HEIG 0 180 203.077 226.154 -
249.231 272.308 295.385 318.461 341.539 364.615 387.692 410.769 433.846 -
456.923 480 480
EXP 1 JOINT 11 12 19 21 22 29 31 32 39 41 42 49
LOAD 1 LOADTYPE Wind  TITLE WL
WIND LOAD X 1 TYPE 1 YR 240 480 ZR 0 288
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE WL
   -----------
   JOINT LOAD - UNIT KIP  INCH
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      21      0.33      0.00        0.00      0.00      0.00      0.00
      22      0.33      0.00        0.00      0.00      0.00      0.00
      29      0.67      0.00        0.00      0.00      0.00      0.00
      31      0.71      0.00        0.00      0.00      0.00      0.00
      32      0.71      0.00        0.00      0.00      0.00      0.00
      39      1.42      0.00        0.00      0.00      0.00      0.00
      41      0.37      0.00        0.00      0.00      0.00      0.00
      42      0.37      0.00        0.00      0.00      0.00      0.00
      49      0.74      0.00        0.00      0.00      0.00      0.00
```

V. ASCE 7-16 Wind Load Generation on Tank

Verify the windward loads due to wind on a portion of a tank structure calculated per the ASCE 7-16 specification.

Details

The wind load is calculated for a tank structure using the direction procedure for MWFRS of enclosed buildings (ASCE 7-16, Chapter 27). The following assumptions and parameters apply:

Category II building (Table 1.5-1)
Basic wind speed = 108 mph
Ground height above sea level = 520 ft
Structure type of “chimney, tank, or similar structure”
Exposure Category B
Wind speed-up over hills or escarpments is considered: a two-dimension ridge of height, $H = 300\ ft$. The distance upwind of crest, $L_h = 650\ ft$ and the distance from crest to the building, $x = 750\ ft$
Height of tank = 40 ft
The least horizontal dimension (W or D), D = 25 ft
Round axisymmetric horizontal cross-section
Depth of protruding elements, $D' = 0.5\ ft$
Natural frequency of structure = 2 Hz (i.e., a “rigid” building per Cl. 26.2)
Damping ratio = 0.01

Determine the wind loads acting on a vertical portion of the upper tank structure at a two-panel segment tangent to the YZ plane. For the purpose of this example, the contributions of the surrounding tank portions are ignored (i.e., only the portions of the wind area in this segment are calculated for brevity).

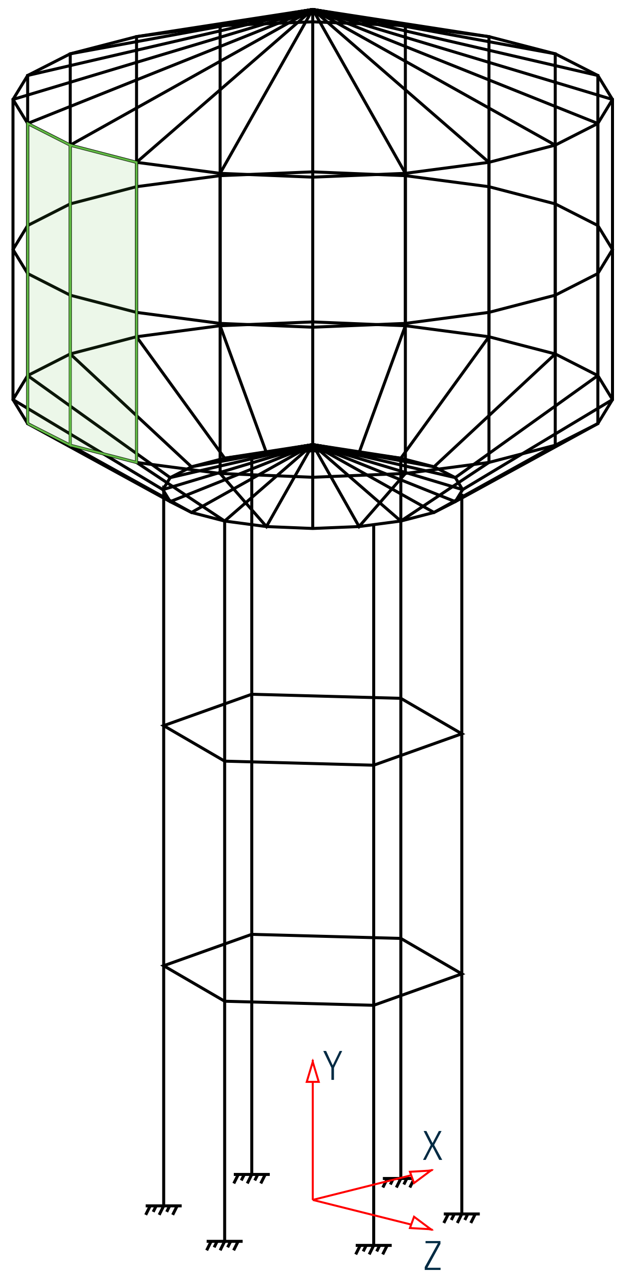


Figure 445: Wind load portion considered on tank structure

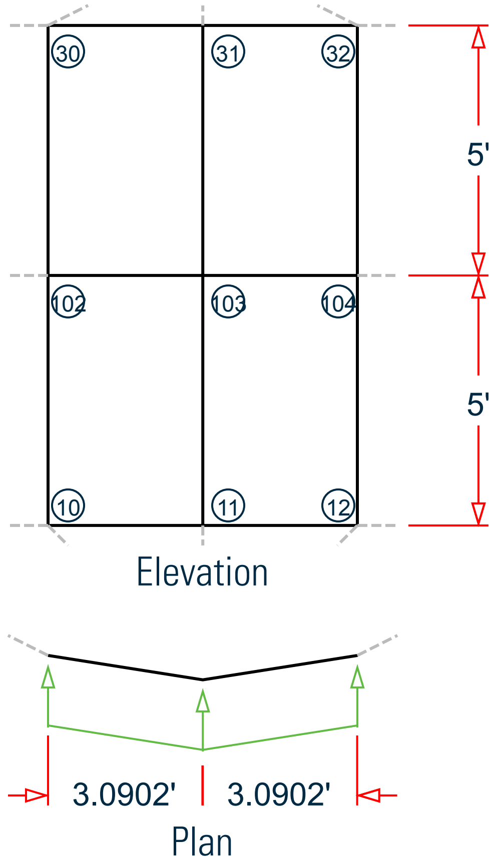


Figure 446: Loaded area plan and elevation

Validation

Directionality factor, $K_d = 1.0$ (Table 26.6-1)

Topographic factor, K_{zt}

For a 2D ridge, $K_1/(H/L_h) = 1.30, \gamma = 3, \mu = 1.5$ (Fig 26.8-1):

$$K_1 = 1.30\frac{H}{L_h} = 1.30 \times \frac{300}{650} = 0.6$$

$$K_2 = 1 - \frac{|x|}{\mu L_h} = 1 - \frac{|750|}{1.5 \times 650} = 0.231$$

$$K_3 = e^{-\gamma z/L_h} = e^{-3\times 40/650}$$

$$K_{zt} = (1 + K_1 K_2 K_3)^2 = \left(1 + 0.6 \times 0.231 \times 0.995^z\right)^2$$

As K_{zt} is a function of height, it must be calculated at each elevation. Refer to the table below.

Ground elevation factor, $K_e = e\text{-}0.0000362 \times 520 = 0.9814$ (Table 26.9-1)

The velocity pressure exposure coefficients (K^z) are calculated per the formulae given in Table 26.10-1:

$$K_z = \begin{cases} 2.01(15/z_g)^{2/a} & \text{for } z < 15 \text{ ft} \\ 2.01(z/z_g)^{2/a} & \text{for } 15 \text{ ft } \leq z \leq zg \end{cases}$$ (Table 26.10.1)

where

a	=	7
z_g	=	1,200 ft

See the table below for the values of K^z *at discrete values of z.*

Velocity pressure, q_z

$$q_z = 0.00256 \times K_z K_{zt} K_d K_e V^2$$ (Cl. 26.10-1)

Mean roof height = height of the building = 40 ft. Refer to table below for calculation of q_z at the mean roof height.

Gust effect factor, G

G is calculated per Cl. 26.11.4 for rigid structures:

$$G = 0.925\left(\frac{1 + 1.7 g_q I_{\bar{z}} Q}{1 + 1.7 g_v I_{\bar{z}}}\right)$$ (Cl. 26.11.4)

where

$g_q = g_v$	=	3.4
$\bar{z}$	=	$max \begin{vmatrix} 30 \ ft \\ 0.6 \times h \end{vmatrix} = 30 \ ft$
ε	=	1/3
ℓ	=	320 ft
c	=	0.3
$L \bar{z}$	=	$\ell(\bar{z} / 33)^{\varepsilon} = 309.99 \ ft$
Q	=	$\sqrt{\frac{1}{1 + 0.63\left(\frac{D + h}{L_{\bar{z}}}\right)^{0.63}}} = \sqrt{\frac{1}{1 + 0.63\left(\frac{25 + 40}{309.99}\right)^{0.63}}} = 0.8997$
$I\bar{z}$	=	$c(33 / \bar{z})^{1/6} = 0.3048$

Therefore, $G = 0.925\left(\frac{1 + 1.7 \times 3.4 \times 0.3048 \times 0.8997}{1 + 1.7 \times 3.4 \times 0.3048}\right) = 0.8658$

Force Coefficient, C_f

C_f is calculated as per Table 29.4-1.

$h/D = 40/25 = 1.6$, so must linearly interpolate between 1 and 7.

$D'/D = 0.5/25 = 0.02$

, so the type of surface is "rough"

The minimum value of $D / \sqrt{q_z}$ corresponds to the maximum value of q_z, which is taken from the tabulated values below (at height, $z = 40 \ ft$).

The minimum value of $D / \sqrt{q_z} = 25 \ ft\sqrt{27.72 \ lb/ft^2} = 4.75 > 2.5$, *so the force coefficient used is for round cross-sections with a relatively wide diameter and rough surface.*

Linearly interpolate between values: $C_f = \frac{(1.6 - 1) \times (0.8 - 0.7)}{(7 - 1)} + 0.7 = 0.71$

The design wind pressure, *p*, is then calculated as:

$$p = q_z \times G \times C_f \qquad \text{(Eq. 29.4-1)}$$

Table 550: Wind intensity versus height for tank structure

h (ft)	K_z	z/L_h	K_3	K_{zt}	q_z (lb/ft^2)	p (lb/ft^2)
0	0.575	0	1	1.296	21.83	13.42
8	0.575	0.012	0.964	1.285	21.64	13.30
16	0.585	0.025	0.929	1.274	21.85	13.43
24	0.657	0.037	0.895	1.263	24.33	14.96
27	0.680	0.042	0.883	1.259	25.09	15.42
29.5	0.697	0.045	0.873	1.256	25.67	15.78
32	0.714	0.049	0.863	1.253	26.21	16.11
34.5	0.729	0.053	0.853	1.250	26.71	16.42
37	0.744	0.057	0.843	1.247	27.18	16.71
40	0.761	0.062	0.831	1.243	27.72	17.04

Note: To compare different wind load calculations with STAAD.Pro, the tributary areas below are selected based on the applied wind area defined in the model. These do *not* include any surrounding wind areas outside of the indicated area, which would have wind applied in a typical situation.

The projected distance along the Z axis perpendicular to the wind direction (along the X axis) is

$\sqrt{(3.1287')^2 - (0.4894')^2} = 3.09\ ft$

Load on Nodes 10 and 12

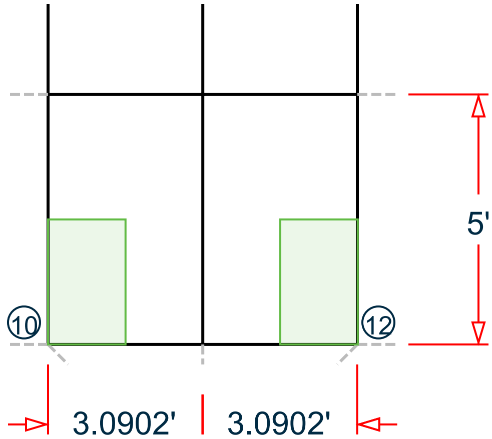


Figure 447: Tributary area for nodes 10 and 12

Wind pressure at half-way between this node and level above (29.5') = 15.78 lb/ft^2.

$$F = (3.09 \text{ ft} / 2) \times (5 \text{ ft} / 2) \times 15.78 \text{ lb/ft}^2 (10)^{-3} = 0.060 \text{ kips}$$

<u>Load on Node 11</u>

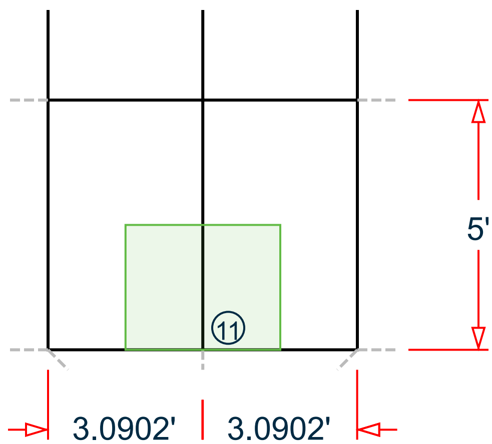


Figure 448: Tributary area for node 11

Wind pressure at half-way between this node and level above (29.5') = 15.78 lb/ft^2.

$$F = (3.09 \text{ ft}) \times (5 \text{ ft} / 2) \times 15.78 \text{ lb/ft}^2 (10)^{-3} = 0.120 \text{ kips}$$

<u>Load on Nodes 102 and 104</u>

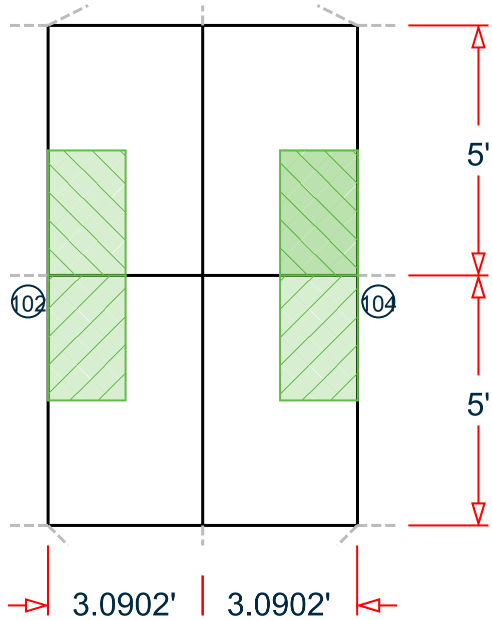


Figure 449: Tributary area for nodes 102 and 104

Wind pressure at half-way between this node and level below (29.5') = 16.11 lb/ft^2.

Wind pressure at half-way between this node and level above (34.5') = 16.42 lb/ft^2.

$$F = (3.09 \text{ ft} / 2) \times (5 \text{ ft} / 2) \times (16.11 \text{ lb/ft}^2 + 16.42 \text{ lb/ft}^2) (10)^{-3} = 0.126 \text{ kips}$$

Load on Node 103

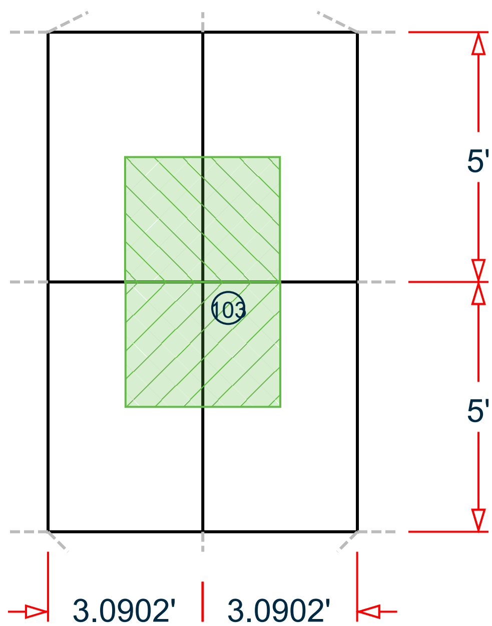


Figure 450: Tributary area for node 103

Wind pressure at half-way between this node and level below (34.5') = 16.11 lb/ft^2.

Wind pressure at half-way between this node and level above (29.5') = 16.42lb/ft^2.

$$F = (3.09 \text{ ft}) \times (5 \text{ ft} / 2) \times (16.11 \text{ lb/ft}^2 + 16.42 \text{ lb/ft}^2)(10)^{-3} = 0.251 \text{ kips}$$

<u>Load on Nodes 30 and 32</u>

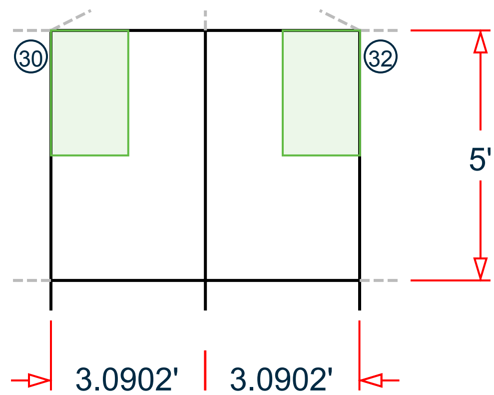


Figure 451: Tributary area for nodes 30 and 32

Wind pressure at top of tributary area (37') = 16.71 lb/ft^2.

$$F = (3.09 \text{ ft} / 2) \times (5 \text{ ft} / 2) \times 16.71 \text{ lb/ft}^2 (10)^{-3} = 0.065 \text{ kips}$$

<u>Load on Node 31</u>

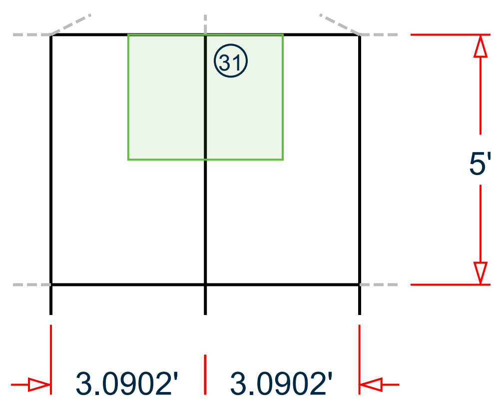


Figure 452: Tributary area for node 31

Wind pressure at top of tributary area (37') = 16.71 lb/ft^2.

$$F = (3.09 \text{ ft}) \times (5 \text{ ft} / 2) \times 16.71 \text{ lb/ft}^2 (10)^{-3} = 0.129 \text{ kips}$$

Results

Table 551: Joint loads due to wind (kips)

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 10	0.060	0.06	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 11	0.120	0.12	none	
Node 12	0.060	0.06	none	
Node 102	0.126	0.13	negligible	
Node 103	0.251	0.25	negligible	
Node 104	0.126	0.13	negligible	
Node 30	0.065	0.06	negligible	
Node 31	0.129	0.13	negligible	
Node 32	0.065	0.06	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\ASCE 7-16 Wind Load Generation on Tank.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Sep-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 10 27 0; 2 9.51057 27 -3.09017; 3 8.09017 27 -5.87785; 4 5.87785 27
-8.09017;
5 3.09017 27 -9.51057; 6 6.12323e-16 27 -10; 7 -3.09017 27 -9.51057;
8 -5.87785 27 -8.09017; 9 -8.09017 27 -5.87785; 10 -9.51057 27 -3.09017;
11 -10 27 -1.22465e-15; 12 -9.51057 27 3.09017; 13 -8.09017 27 5.87785;
14 -5.87785 27 8.09017; 15 -3.09017 27 9.51057; 16 -1.83697e-15 27 10;
17 3.09017 27 9.51057; 18 5.87785 27 8.09017; 19 8.09017 27 5.87785;
20 9.51057 27 3.09017; 21 10 37 0; 22 9.51057 37 -3.09017;
23 8.09017 37 -5.87785; 24 5.87785 37 -8.09017; 25 3.09017 37 -9.51057;
26 6.12323e-16 37 -10; 27 -3.09017 37 -9.51057; 28 -5.87785 37 -8.09017;
29 -8.09017 37 -5.87785; 30 -9.51057 37 -3.09017; 31 -10 37 -1.22465e-15;
32 -9.51057 37 3.09017; 33 -8.09017 37 5.87785; 34 -5.87785 37 8.09017;
35 -3.09017 37 9.51057; 36 -1.83697e-15 37 10; 37 3.09017 37 9.51057;
38 5.87785 37 8.09017; 39 8.09017 37 5.87785; 40 9.51057 37 3.09017; 41 0 40
0;
42 5 24 0; 43 4.75528 24 -1.54508; 44 4.04508 24 -2.93893;
45 2.93893 24 -4.04508; 46 1.54508 24 -4.75528; 47 3.06162e-16 24 -5;
48 -1.54508 24 -4.75528; 49 -2.93893 24 -4.04508; 50 -4.04508 24 -2.93893;
51 -4.75528 24 -1.54508; 52 -5 24 -6.12323e-16; 53 -4.75528 24 1.54508;
54 -4.04508 24 2.93893; 55 -2.93893 24 4.04508; 56 -1.54508 24 4.75528;
57 -9.18485e-16 24 5; 58 1.54508 24 4.75528; 59 2.93893 24 4.04508;
60 4.04508 24 2.93893; 61 4.75528 24 1.54508; 62 0 25.5 0;
63 -2.93893 16 4.04508; 64 -2.93893 8 4.04508; 65 -2.93893 0 4.04508;
66 2.03368 24 4.56773; 67 2.03368 16 4.56773; 68 2.03368 8 4.56773;
69 2.03368 0 4.56773; 70 4.97261 24 0.522642; 71 4.97261 16 0.522642;
```

```
72 4.97261 8 0.522642; 73 4.97261 0 0.522642; 74 2.93893 16 -4.04508;
75 2.93893 8 -4.04508; 76 2.93893 0 -4.04508; 77 -2.03368 24 -4.56773;
78 -2.03368 16 -4.56773; 79 -2.03368 8 -4.56773; 80 -2.03368 0 -4.56773;
81 -4.97261 24 -0.522642; 82 -4.97261 16 -0.522642; 83 -4.97261 8 -0.522642;
84 -4.97261 0 -0.522642; 85 -8.09017 32 5.87785; 86 -5.87785 32 8.09017;
87 -3.09017 32 9.51057; 88 -1.83697e-15 32 10; 89 3.09017 32 9.51057;
90 5.87785 32 8.09017; 91 8.09017 32 5.87785; 92 9.51057 32 3.09017;
93 10 32 0; 94 9.51057 32 -3.09017; 95 8.09017 32 -5.87785;
96 5.87785 32 -8.09017; 97 3.09017 32 -9.51057; 98 6.12323e-16 32 -10;
99 -3.09017 32 -9.51057; 100 -5.87785 32 -8.09017; 101 -8.09017 32 -5.87785;
102 -9.51057 32 -3.09017; 103 -10 32 -1.22465e-15; 104 -9.51057 32 3.09017;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10; 10 10 11;
11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17; 17 17 18; 18 18
19;
19 19 20; 20 20 1; 21 21 22; 22 22 23; 23 23 24; 24 24 25; 25 25 26; 26 26 27;
27 27 28; 28 28 29; 29 29 30; 30 30 31; 31 31 32; 32 32 33; 33 33 34; 34 34
35;
35 35 36; 36 36 37; 37 37 38; 38 38 39; 39 39 40; 40 40 21; 41 1 93; 42 2 94;
43 3 95; 44 4 96; 45 5 97; 46 6 98; 47 7 99; 48 8 100; 49 9 101; 50 10 102;
51 11 103; 52 12 104; 53 13 85; 54 14 86; 55 15 87; 56 16 88; 57 17 89;
58 18 90; 59 19 91; 60 20 92; 61 41 34; 62 41 35; 63 41 36; 64 41 37; 65 41
38;
66 41 39; 67 41 40; 68 41 21; 69 41 22; 70 41 23; 71 41 24; 72 41 25; 73 41
26;
74 41 27; 75 41 28; 76 41 29; 77 41 30; 78 41 31; 79 41 32; 80 41 33; 81 42
43;
82 43 44; 83 44 45; 84 45 46; 85 46 47; 86 47 48; 87 48 49; 88 49 50; 89 50
51;
90 51 52; 91 52 53; 92 53 54; 93 54 55; 94 55 56; 95 56 57; 96 57 58; 97 58
59;
98 59 60; 99 60 61; 100 61 42; 101 14 55; 102 15 56; 103 16 57; 104 17 58;
105 18 59; 106 19 60; 107 20 61; 108 1 42; 109 2 43; 110 3 44; 111 4 45;
112 5 46; 113 6 47; 114 7 48; 115 8 49; 116 9 50; 117 10 51; 118 11 52;
119 12 53; 120 13 54; 121 62 55; 122 62 56; 123 62 57; 124 62 58; 125 62 59;
126 62 60; 127 62 61; 128 62 42; 129 62 43; 130 62 44; 131 62 45; 132 62 46;
133 62 47; 134 62 48; 135 62 49; 136 62 50; 137 62 51; 138 62 52; 139 62 53;
140 62 54; 141 55 63; 142 63 64; 143 64 65; 144 55 66; 145 63 67; 146 64 68;
147 66 67; 148 67 68; 149 68 69; 150 66 70; 151 67 71; 152 68 72; 153 70 71;
154 71 72; 155 72 73; 156 70 45; 157 71 74; 158 72 75; 159 45 74; 160 74 75;
161 75 76; 162 45 77; 163 74 78; 164 75 79; 165 77 78; 166 78 79; 167 79 80;
168 77 81; 169 78 82; 170 79 83; 171 81 82; 172 82 83; 173 83 84; 174 81 55;
175 82 63; 176 83 64; 257 85 33; 258 86 34; 259 85 86; 260 87 35; 261 86 87;
262 88 36; 263 87 88; 264 89 37; 265 88 89; 266 90 38; 267 89 90; 268 91 39;
269 90 91; 270 92 40; 271 91 92; 272 93 21; 273 92 93; 274 94 22; 275 93 94;
276 95 23; 277 94 95; 278 96 24; 279 95 96; 280 97 25; 281 96 97; 282 98 26;
283 97 98; 284 99 27; 285 98 99; 286 100 28; 287 99 100; 288 101 29;
289 100 101; 290 102 30; 291 101 102; 292 103 31; 293 102 103; 294 104 32;
295 103 104; 296 104 85;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
```

```
END DEFINE MATERIAL
MEMBER PROPERTY
141 TO 143 147 TO 149 153 TO 155 159 TO 161 165 TO 167 171 TO 173 PRIS YD 2.5
145 146 151 152 157 158 163 164 169 170 175 176 PRIS YD 3 ZD 2
1 TO 140 144 150 156 162 168 174 257 TO 296 PRIS YD 1 ZD 1
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
65 69 73 76 80 84 FIXED
DEFINE WIND LOAD
TYPE 1 ASCE 7:2016
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2016:PARAMS 108.000 MPH 520.000 FT 1 1 1 1 750.000 FT 650.000 FT -
300.000 FT 0 6 40.000 FT 25.000 FT 0.500 FT 2.000 0.010 0 0 0 0 0 0.761 -
1.243 1.000 1.000 0.981 0 0 0 0 0.866 0.710 1.000
!> END GENERATED DATA BLOCK
INT 0.0132004 0.0132004 0.0136356 0.0140333 0.0143996 0.0147394 0.0150565 -
0.0153537 0.0156335 0.0158978 0.0161483 0.0163864 0.0166132 0.0168298 -
0.017037 HEIG 0 15 16.9231 18.8461 20.7692 22.6923 24.6154 26.5385 28.4615 -
30.3846 32.3077 34.2308 36.1538 38.0769 40
EXP 1 JOINT 1 TO 84
LOAD 1 LOADTYPE Wind  TITLE WL
WIND LOAD X 1 TYPE 1 YR 27 37 ZR -5 5
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE WL
   -----------
   JOINT LOAD - UNIT KIP  FEET
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      10      0.06      0.00        0.00      0.00      0.00      0.00
      11      0.12      0.00        0.00      0.00      0.00      0.00
      12      0.06      0.00        0.00      0.00      0.00      0.00
      30      0.06      0.00        0.00      0.00      0.00      0.00
      31      0.13      0.00        0.00      0.00      0.00      0.00
      32      0.06      0.00        0.00      0.00      0.00      0.00
     102      0.13      0.00        0.00      0.00      0.00      0.00
     103      0.25      0.00        0.00      0.00      0.00      0.00
     104      0.13      0.00        0.00      0.00      0.00      0.00
```

V. ASCE 7-10 Wind Load Generation

To validate STAAD.Pro calculated equivalent joint loads for a closed structure subjected to Wind Loading. Wind Intensity is calculated as per ASCE 07 – 2010.

Details

When closed type structures are subjected to wind load, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. Wind load, in such way, is converted to equivalent joint loads and applied on the nodes of that closed surface. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is calculated. Wind intensities have been calculated as per ASCE 07 – 2010. The detailed calculation of determination of equivalent joint loads is performed here in this document.

A two-story, single-bay frame is subjected to wind loading from +X direction. Assumptions made while calculating the wind intensities are tabulated in the Validation section of this document.

Verification

Calculation of Wind Intensity

Assumptions made in determination of wind intensities as per ASCE 07 – 2010

- **i.** Analysis Procedure – Directional Procedure for MWFRS of enclosed buildings, Chapter 27
- **ii.** Type of roof – flat
- **iii.** Building Classification Category = Category II (Table 1.5-1)
- **iv.** Basic Wind Speed = 110 mph
- **v.** Exposure Category = Exposure B (Clause 26.7.3)
- **vi.** Wind speed-up over Hills or Escarpment is not considered
- **vii.** Building Height = 20 ft
- **viii.** Building length along the direction of Wind (L) = 10 ft
- **ix.** Building length normal the direction of Wind (B) = 20 ft
- **x.** Natural Frequency of the building = 2 Hz (Hence, as per clause 26.2, this is a rigid building)
- **xi.** Building Damping Ratio = 0.05
- **xii.** Enclosure Classification = Enclosed Building (Clause 26.10)
- **xiii.** Directionality Factor Kd = 0.85 (Table 26.6-1, for MWFRS)
- **xiv.** Topographic Factor, Kzt = 1 (Clause 26.8.2)
- **xv.** Building Wall to generate Wind Load on – Windward side
- **xvi.** Gust Effect Factor, G = 0.85 (Clause 26.9.1)
- **xvii.** External pressure Coefficient Cp = 0.85 (Windward wall pressure coefficient, as per Table 27.4-1)
- **xviii.** Internal pressure coefficient GCpi = -0.18 (For Enclosed Buildings, Table 26.11-1)

Hence, the velocity pressure exposure coefficients (Kz) are calculated as per the formulae given in Table 27.3-1:

From Table 26.9.1, For Exposure B, $\alpha = 7$ and $zg = 1{,}200\ ft$

Height (ft)	Velocity Pressure Exposure Coefficient (Kz)
0 – 15	0.5747196698
16	0.5854155685
17	0.5956440691
18	0.6054513812
19	0.6148768735
20	0.6239543877

As per clause 27.3.2, velocity pressure $q_z = 0.00256x\ K_z \times K_{zt} \times K_d \times V^2\ (lb/ft^2)$

Since Mean roof height h = Height of building = 20 ft

$$q_h = 0.00256\times (K_z)_{h=20\ ft} \times K_{zt} \times K_d \times V^2 = 16.42847\ lb/ft^2$$

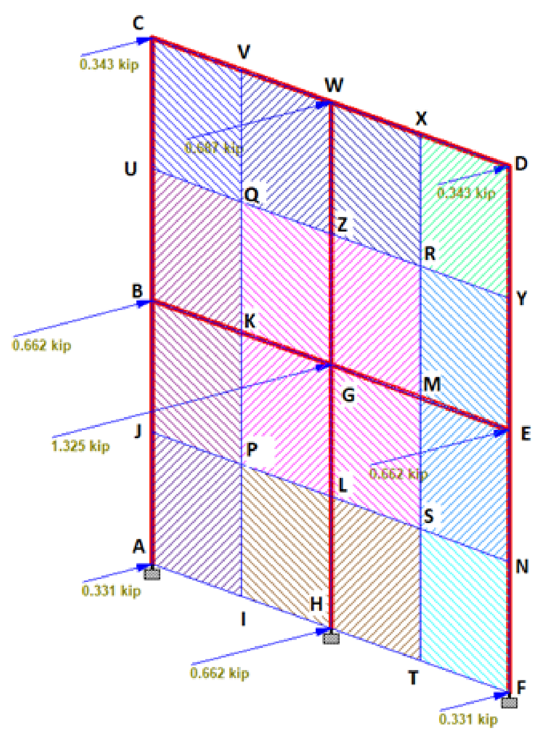
Design wind pressure, p is calculated as per clause 27.4.1 as -

$$p = q \times G \times Cp - q_i \times (GCpi)$$

Values of q_z and p for different heights are calculated using the above formulae and tabulated below.

Height (ft)	Kz	q_z (lb/ft^2)	p (lb/ft2)
15	0.575	15.13	13.25
16	0.585	15.41	13.44
17	0.596	15.68	13.62
18	0.605	15.94	13.80
19	0.615	16.19	13.97
20	0.624	16.43	14.13

Calculation of Equivalent Joint Loads



Evidently, members AB, BG and GH along with the ground surface, forms the closed panel ABGH. P is the CG of the panel, located at 5 ft away from line AH and 5 ft away from line AB. So, influence area for joint A is AJPI. Area of AJPI = 5 × 5 = 25 ft^2.

Average intensity of wind pressure in this area $AJPI = 13.25\ lb/ft^2$

Exposure factor = 1 (for all the nodes in the model)

So, equivalent joint load in joint A = Influence area × Avg. Intensity of wind pressure × Exposure

$$= 25 \times 13.25 \times 1 = 331.2 \text{ lb} = 0.33 \text{ kip}$$

Joint B will be under influence of influence area BKQU and BKPJ. Area of each of those influence areas = 5 × 5 = 25 ft^2. Intensity of wind pressure in each of those influence areas = 13.25 lb/ft^2. Hence, equivalent joint load for each of those areas = 0.33 kip

So, total joint load in joint B = 0.331 × 2 = 0.662 kip

Joint C will be under influence area CVQU. Average intensity of wind pressure in this influence area will be

$$= (13.25 + 14.1284837) / 2 = 13.69 \text{ lb/ft}^2$$

Exposure Factor = 1

Area of CVQU = $5 \times 5 = 25\ ft^2$

Hence, equivalent joint load at joint $C = 342.2\ lb \approx 0.343\ kip$

Joint W will be under influence of area WXRZ and WVQZ. Area of both of these = 25 ft^2. Average intensity of wind pressure in both of the influence areas = Average intensity of wind in the influence area CVQU = 13.68 lb/ft^2. Hence, equivalent joint load on joint $W = 2 \times 0.343\ kip = 0.686\ kip$

Joint G will be under the influence of area GZRM, GZQK, GMSL and GKPL. Area of each of these influence areas = 5 × 5 = 25 ft^2. Average intensity of wind pressure in each of these areas = average intensity of wind pressure in area BKQU = 13.25 lb/ft^2. Hence, equivalent joint load on joint $G = 4 \times 0.33 = 1.32\ kip$

Joint H will be under the influence of areas HLPI and HTSL. Area of each of those influence areas = 5 × 5 = 25 ft^2. Intensity of wind pressure in each of those influence areas = 13.25 lb/ft^2. Hence, equivalent joint load on joint H $= 2 \times 0.33 = 0.66\ kip$

Since the structure is symmetric about the line WH, the equivalent joint load at D = equivalent joint load at C = 0.34 kip, equivalent joint load at B = equivalent joint load at E = 0.66 kip and equivalent joint load at A = equivalent joint load at F = 0.33 kip.

Comparison

Table 552: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Joint load at node (kips)	1	0.33	0.33	none	
	2	0.66	0.66	none	
	5	0.66	0.66	none	
	6	1.32	1.32	none	
	9	0.33	0.33	none	
	10	0.66	0.66	none	
	13	0.34	0.34	none	
	15	0.69	0.69	none	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	17	0.34	0.34	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\ASCE 7-10 Wind Load Generation.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Sep-18
END JOB INFORMATION
*****************************************************************
*This problem has been created to verify the program calculated
*intensities of wind pressure as per ASCE 07-2010 and
*verify equivalent joint loads in the joints of a
*closed structure subjected to wind
*****************************************************************
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 0 20; 10 0 10 20; 11 10 10 20; 12 10 0 20; 13 0 20 0;
14 10 20 0; 15 0 20 10; 16 10 20 10; 17 0 20 20; 18 10 20 20;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 2 13; 15 3 14; 16 6 15; 17 7 16; 18 10 17;
19 11 18; 20 13 14; 21 13 15; 22 14 16; 23 15 16; 24 15 17; 25 16 18; 26 17
18;
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 26 TABLE ST W14X873
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 4 5 8 9 12 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2010:PARAMS 110.000 MPH 0 1 1 0 0.000 FT 30.000 FT 10.000 FT 1 2 -
20.000 FT 10.000 FT 10.000 FT 2.000 0.050 0 0 0 0 1 0.624 1.000 1.000 0.850 -
0 1 1 1 0.850 0.800 -0.180
!> END GENERATED DATA BLOCK
INT 0.013247 0.013247 0.0133217 0.0133951 0.0134672 0.0135381 0.0136078 -
0.0136764 0.0137439 0.0138104 0.0138759 0.0139404 0.014004 0.0140667 -
0.0141285 0.0169213 HEIG 0 15 15.3846 15.7692 16.1539 16.5385 16.9231 -
```

```
17.3077 17.6923 18.0769 18.4615 18.8461 19.2308 19.6154 20 20
EXP 1 JOINT 1 TO 12
LOAD 1 LOADTYPE Wind  TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 YR 0 40
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD Output

```
   LOADING     1  LOADTYPE WIND  TITLE LOAD CASE 1
   -----------
   JOINT LOAD - UNIT KIP  FEET
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.33      0.00        0.00      0.00      0.00      0.00
       2      0.66      0.00        0.00      0.00      0.00      0.00
       5      0.66      0.00        0.00      0.00      0.00      0.00
       6      1.32      0.00        0.00      0.00      0.00      0.00
       9      0.33      0.00        0.00      0.00      0.00      0.00
      10      0.66      0.00        0.00      0.00      0.00      0.00
      13      0.34      0.00        0.00      0.00      0.00      0.00
      15      0.69      0.00        0.00      0.00      0.00      0.00
      17      0.34      0.00        0.00      0.00      0.00      0.00
   ************ END OF DATA FROM INTERNAL STORAGE ************
```

V. ASCE 7-02 Wind Load Generation on Building

Verify the windward loads due to wind on a portion of a building structure calculated per the ASCE 7-02 specification.

Details

The wind load is calculated for a rigid building structure using the procedure for MWFRS of enclosed buildings (ASCE 7-02, Section 6.5.12.2.1). The following assumptions and parameters apply:

Flat roof type
Category II building (Table 1-1)
Basic wind speed = 108 mph
Structure type of “building”
Exposure Category B
Do not consider wind speed-up over hills or escarpments
Building height = 40 ft
Building length along the wind direction, L = 10 ft
Building width normal to wind direction, B = 20 ft
Natural frequency of structure = 2 Hz (i.e., a “rigid” building per Section 6.2)
Damping ratio = 0.05
Enclosed building (per Section 6.2)

Calculate the resulting joint loads on the *windward* side due to wind along the X axis at the upper wall of one segment of the building.

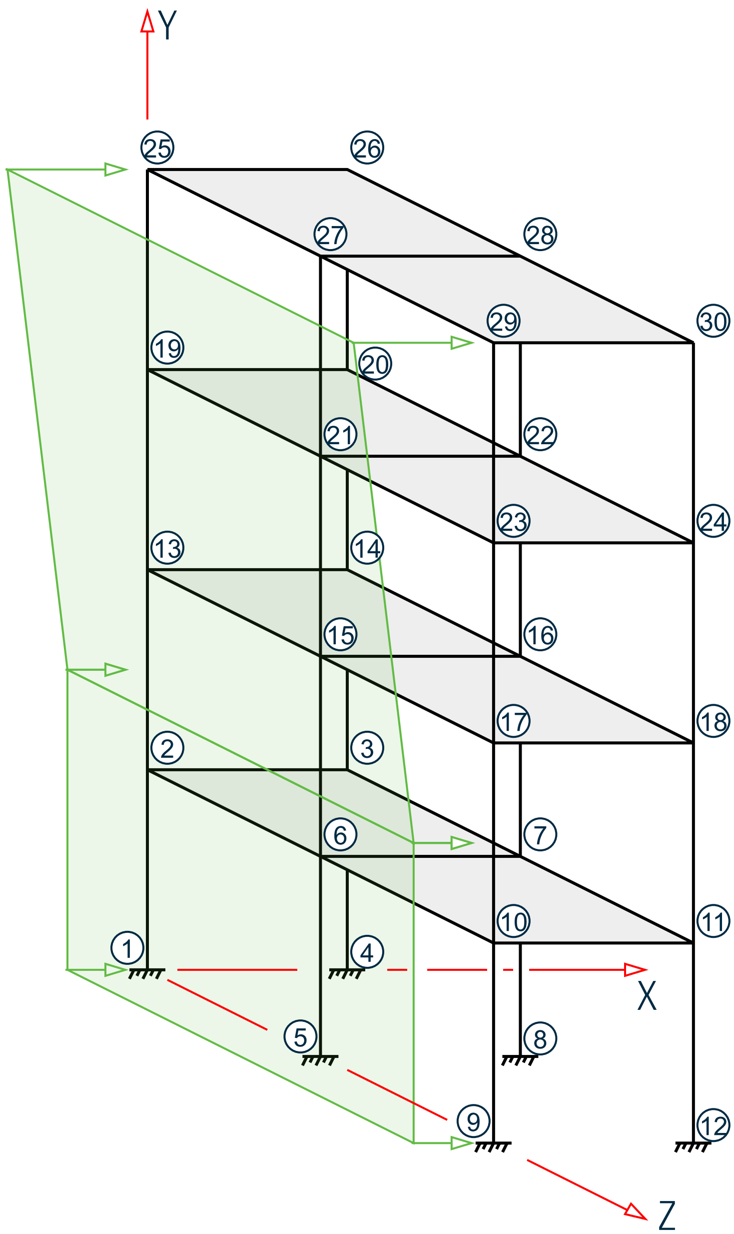


Figure 453: Building structure with wind load in X direction

Validation

Importance factor, $I = 1.0$ (Table 6-1)

The velocity pressure exposure coefficients (K^z) are calculated per the formulae given in Table 6-3:

$$K_z = \begin{vmatrix} 2.01(15/z_g)^{2/a} & \text{for } z < 15 \text{ ft} \\ 2.01(z/z_g)^{2/a} & \text{for } 15 \text{ ft } \le z \le zg \end{vmatrix} \qquad \text{(Table 6-3)}$$

where

a = 7 (Table 6-2)

z_g = 1,200 ft (Table 6-2)

See the table below for the values of K^z *at discrete values of z.*

Directionality factor, $K_d = 0.85$ (Table 6-4)

Topographic factor, $K_{zt} = 1.0$ (i.e., not considered)

Velocity pressure, q_z

$$q_z = 0.00256 K_z K_{zt} K_d V^2 I \quad \text{(Eq. 6-15)}$$

Mean roof height = height of the building = 30 ft. Refer to table below for calculation of q_z at the mean roof height.

Gust effect factor, G

G is calculated per Cl. 26.11.4 for rigid structures:

$$G = 0.925\left(\frac{1 + 1.7 g_q I_{\bar{z}} Q}{1 + 1.7 g_v I_{\bar{z}}}\right) \quad \text{(Eq. 6-4)}$$

where

$g_q = g_v$ = 3.4

$\bar{z}$ = $max \left| \begin{matrix} 30\ ft \\ 0.6 \times h \end{matrix} \right. = 30\ ft$

ε = 1/3

ℓ = 320 ft

c = 0.3

$L_{\bar{z}}$ = $\ell\left(\bar{z} / 33\right)^{\varepsilon} = 309.99\ ft$

Q = $\sqrt{\frac{1}{1 + 0.63\left(\frac{B + h}{L_{\bar{z}}}\right)^{0.63}}} = \sqrt{\frac{1}{1 + 0.63\left(\frac{20 + 40}{309.99}\right)^{0.63}}} = 0.904$

$I_{\bar{z}}$ = $c\left(33 / \bar{z}\right)^{1/6} = 0.305$

Therefore, $G = 0.925\left(\frac{1 + 1.7 \times 3.4 \times 0.305 \times 0.904}{1 + 1.7 \times 3.4 \times 0.305}\right) = 0.8683$

External pressure coefficient, $C_p = 0.8$ for windward per Fig. 6-6.

The external design pressure, $p = q_h G C_p$.

The internal pressure coefficient, $GC_{pi} = -0.18$ for partially enclosed buildings per Figure 6-5.

The internal design pressure, $p_i = q_i(GC_{pi}) = 19.30 \times -0.18 = -3.47\ lb/ft^2$

where

q_i = the velocity pressure evaluated at the building height, h,

$q_h = 0.00256 K_{z=h} K_{zt} K_d V^2 I = 0.00256\left(0.761\right)\left(1.0\right)\left(0.85\right)(108)^2\left(1.0\right) = 19.30\ lb/f$

The design wind pressure, p, is then calculated as:

$$p = q_z G C_p - q_i(GC_{pi}) \quad \text{(Eq. 6-17)}$$

Table 553: Wind intensity versus height for building structure

h (ft)	K_z	q_z (lb/ft^2)	p (lb/ft^2)
0	0.575	14.59	13.61
15	0.575	14.59	13.61
20	0.624	15.84	14.48
25	0.665	16.88	15.20
30	0.701	17.78	15.83
35	0.732	18.58	16.38
40	0.761	19.30	16.89

Load on Nodes 1 and 9

Assume that the ground (Y = 0') acts as a bounded edge for the wind load acting on the exterior of the building. This edge transfers load to the columns.

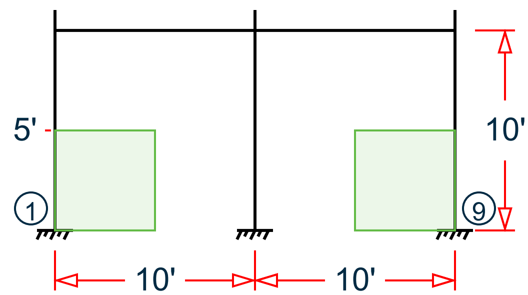


Figure 454: Tributary area for nodes 1 and 9

Wind pressure at half-way between this node and level above (5') = 13.61 lb/ft^2.

$$F = (10\ ft / 2) \times (10\ ft / 2) \times 13.61\ lb/ft^2\ (10)^{-3} = 0.34\ kips$$

Load on Node 5

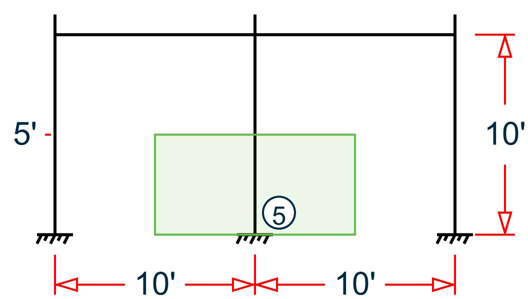


Figure 455: Tributary area for node 5

Wind pressure at half-way between this node and level above (5') = 13.61 lb/ft^2.

$$F = (10 \text{ ft} / 2) \times (10 \text{ ft}) \times 13.60 \text{ lb/ft}^2 (10)^{-3} = 0.68 \text{ kips}$$

Load on Nodes 2 and 10

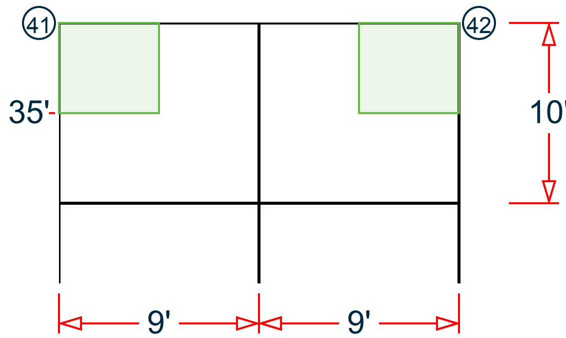


Figure 456: Tributary area for nodes 2 and 10

Wind pressure at half-way between this node and level above (5') = 13.61 lb/ft^2. This pressure is constant below this height.

$$F = (10 \text{ ft} / 2) \times (10 \text{ ft}) \times 13.61 \text{ lb/ft}^2 (10)^{-3} = 0.68 \text{ kips}$$

Load on Node 6

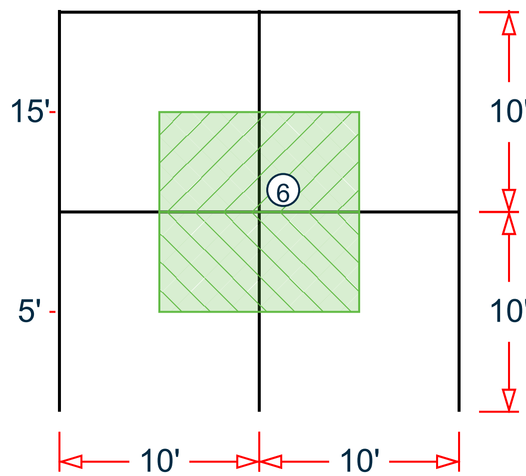


Figure 457: Tributary area for node 6

Wind pressure at half-way between this node and level above (5') = 13.61 lb/ft^2. This pressure is constant below this height.

$$F = (10 \text{ ft}) \times (10 \text{ ft}) \times 13.61 \text{ lb/ft}^2 (10)^{-3} = 1.36 \text{ kips}$$

Load on Nodes 13 and 17

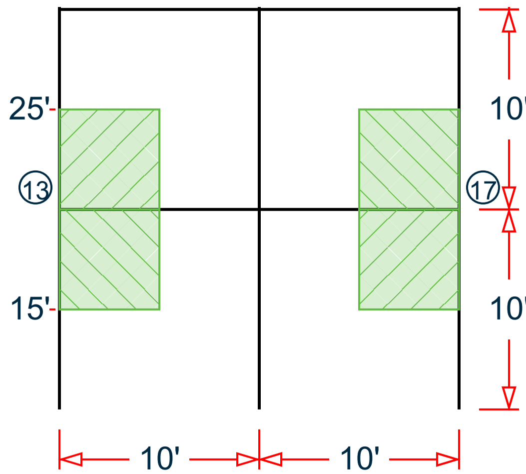


Figure 458: Tributary area for nodes 13 and 17

Wind pressure at half-way between this node and level below (15') = 13.61 lb/ft^2.

Wind pressure at half-way between this node and level above (25') = 15.20 lb/ft^2.

$$F = (10 \text{ ft} /2) \times (10 \text{ ft} / 2) \times (13.61 \text{ lb/ft}^2 + 15.20 \text{ lb/ft}^2) (10)^{-3} = 0.72 \text{ kips}$$

<u>Load on Node 15</u>

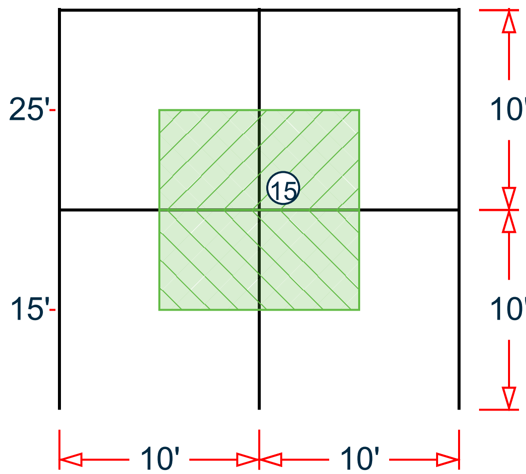


Figure 459: Tributary area for node 15

Wind pressure at half-way between this node and level below (15') = 13.61 lb/ft^2.

Wind pressure at half-way between this node and level above (25') = 15.20 lb/ft^2.

$$F = (10 \text{ ft} /2) \times (10 \text{ ft}) \times (13.61 \text{ lb/ft}^2 + 15.20 \text{ lb/ft}^2) (10)^{-3} = 1.44 \text{ kips}$$

<u>Load on Node 19</u>

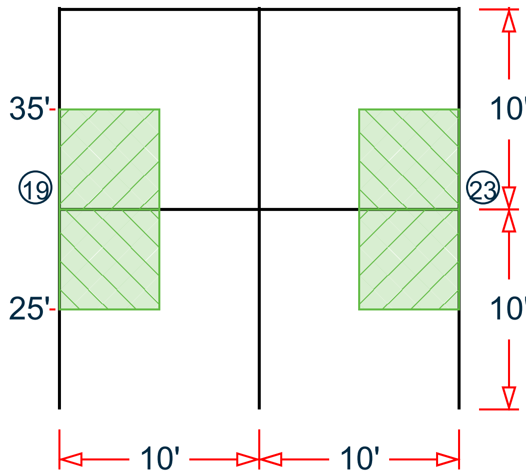


Figure 460: Tributary area for nodes 19 and 23

Wind pressure at half-way between this node and level below (25') = 15.20 lb/ft^2.

Wind pressure at half-way between this node and level above (35') = 16.38 lb/ft^2.

$$F = (10 \text{ ft} /2) \times (10 \text{ ft} / 2) \times (15.20 \text{ lb/ft}^2 + 16.38 \text{ lb/ft}^2) (10)^{-3} = 0.79 \text{ kips}$$

Load on Node 21

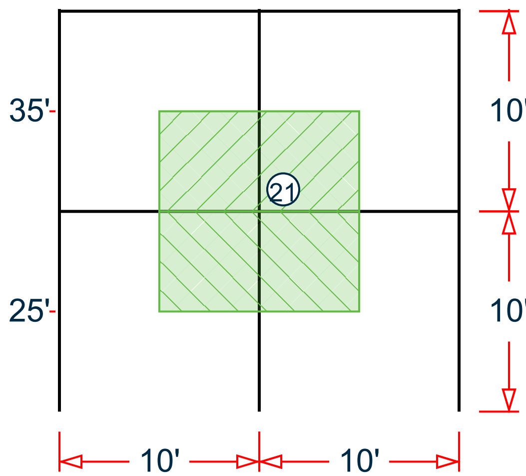


Figure 461: Tributary area for node 21

Wind pressure at half-way between this node and level below (25') = 15.20 lb/ft^2.

Wind pressure at half-way between this node and level above (35') = 16.38 lb/ft^2.

$$F = (10 \text{ ft}) \times (10 \text{ ft} / 2) \times (15.20 \text{ lb/ft}^2 + 16.38 \text{ lb/ft}^2) (10)^{-3} = 1.58 \text{ kips}$$

Load on Nodes 25 and 29

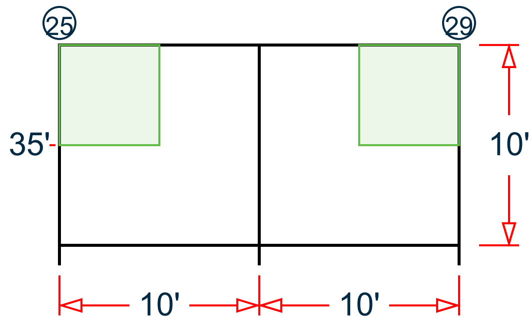


Figure 462: Tributary area for nodes 25 and 29

Wind pressure at roof level (40') = 16.89 lb/ft^2.

$$F = (10\text{ ft} / 2) \times (10\text{ ft} / 2) \times 16.89\text{ lb/ft}^2\ (10)^{-3} = 0.42\text{ kips}$$

Load on Node 27

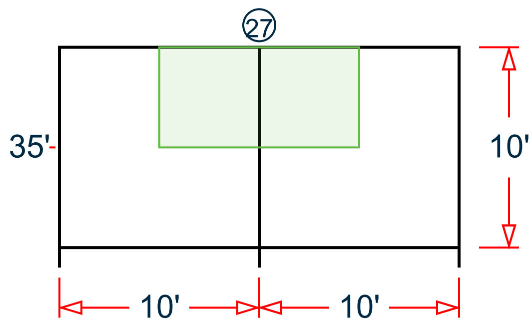


Figure 463: Tributary area for node 27

Wind pressure at roof level (40') = 16.89 lb/ft^2.

$$F = (10\text{ ft}) \times (10\text{ ft} / 2) \times 15.04\text{ lb/ft}^2\ (10)^{-3} = 0.84\text{ kips}$$

Results

Table 554: Joint loads due to wind (kips)

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 1	0.34	0.34	none	
Node 5	0.68	0.68	none	
Node 2	0.68	0.68	none	
Node 6	1.36	1.36	none	
Node 13	0.72	0.73	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Node 15	1.44	1.46	negligible	
Node 19	0.79	0.80	negligible	
Node 21	1.58	1.59	negligible	
Node 25	0.42	0.42	none	
Node 27	0.84	0.84	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\ASCE7-02 Wind Load Generation on Building.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Sep-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 0 20; 10 0 10 20; 11 10 10 20; 12 10 0 20; 13 0 20 0;
14 10 20 0; 15 0 20 10; 16 10 20 10; 17 0 20 20; 18 10 20 20; 19 0 30 0;
20 10 30 0; 21 0 30 10; 22 10 30 10; 23 0 30 20; 24 10 30 20; 25 0 40 0;
26 10 40 0; 27 0 40 10; 28 10 40 10; 29 0 40 20; 30 10 40 20;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 6 10; 10 7 11;
11 9 10; 12 10 11; 13 11 12; 14 2 13; 15 3 14; 16 6 15; 17 7 16; 18 10 17;
19 11 18; 20 13 14; 21 13 15; 22 14 16; 23 15 16; 24 15 17; 25 16 18; 26 17
18;
27 13 19; 28 14 20; 29 15 21; 30 16 22; 31 17 23; 32 18 24; 33 19 20; 34 19
21;
35 20 22; 36 21 22; 37 21 23; 38 22 24; 39 23 24; 40 19 25; 41 20 26; 42 21
27;
43 22 28; 44 23 29; 45 24 30; 46 25 26; 47 25 27; 48 26 28; 49 27 28; 50 27
29;
51 28 30; 52 29 30;
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 52 TABLE ST W14X873
CONSTANTS
MATERIAL STEEL_50_KSI ALL
```

```
SUPPORTS
1 4 5 8 9 12 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2002:PARAMS 108.000 mph 0 1 1 0 0.000 ft 30.000 ft 10.000 ft 1 2 -
40.000 ft 10.000 ft 20.000 ft 2.000 0.050 0 0 0 0 0 0.761 1.000 1.000 0.850 -
0 0 0 0 0.868 0.800 -0.180
!> END GENERATED DATA BLOCK
INT 0.0136076 0.0136076 0.0139629 0.0142904 0.0145949 0.0148798 0.015148 -
0.0154016 0.0156424 0.0158718 0.0160911 0.0163013 0.0165031 0.0166975 -
0.0168849 HEIG 0 15 16.9231 18.8461 20.7692 22.6923 24.6154 26.5385 28.4615 -
30.3846 32.3077 34.2308 36.1538 38.0769 40
EXP 1 JOINT 1 TO 12
LOAD 1 LOADTYPE Wind  TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 YR 0 40
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE LOAD CASE 1
   -----------
   JOINT LOAD - UNIT KIP  FEET
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.34      0.00        0.00      0.00      0.00      0.00
       2      0.68      0.00        0.00      0.00      0.00      0.00
       5      0.68      0.00        0.00      0.00      0.00      0.00
       6      1.36      0.00        0.00      0.00      0.00      0.00
       9      0.34      0.00        0.00      0.00      0.00      0.00
      10      0.68      0.00        0.00      0.00      0.00      0.00
      13      0.73      0.00        0.00      0.00      0.00      0.00
      15      1.46      0.00        0.00      0.00      0.00      0.00
      17      0.73      0.00        0.00      0.00      0.00      0.00
      19      0.80      0.00        0.00      0.00      0.00      0.00
      21      1.59      0.00        0.00      0.00      0.00      0.00
      23      0.80      0.00        0.00      0.00      0.00      0.00
      25      0.42      0.00        0.00      0.00      0.00      0.00
      27      0.84      0.00        0.00      0.00      0.00      0.00
      29      0.42      0.00        0.00      0.00      0.00      0.00
```

V. GB 50009-2012 Wind Load generation on Rectangular Building

Verify the calculated wind pressure at the mid-height (4.5 m) on a structure per the Chinese GB 50009-2012 code.

Problem

This problem determines the pressures applied to a rectangular building at various significant heights, from a wind force that is flowing in the positive X direction (i.e. from X= -infinity to X= + infinity).

Note: STAAD.Pro generates loading on the windward face, sidewalls, and also leeward face of the buildings.

A 9 m tall, two story structure is subject to a wind load along the along the global X direction. The building width (along the Z direction, perpendicular to wind) is 6 m. The building length (along the X direction, parallel to wind) is 10 m. The structure is a closed structure. The building is located in Beijing City in the Beijing province.

> The shape factor, $\mu_s = 0.8$. This is for the left face of the structure and the direction of wind is from left (+X). (+X) means that the wind is originating along the X axis pointing in the +X direction. The plan section type of the structure is rectangle. Refer to table 8.3.1 of the Chinese load code GB 50009-2012.
> The reference wind pressure, $w_0 = 0.45\ kN/m^2$. Refer to table E.5 in the Chinese load code GB 50009-2012.
> The roughness type is “C”

Wind-induced vibration along-wind is not considered ($\beta_z = 1$).

Calculations

Calculate the modified wind shape factor:

$$\mu_s' = \mu_s \times \beta_z = 0.8 \times 1.0 = 0.8$$

Calculate the modified wind height factor:

Per Table 8.2.1 of GB 50009-2012, at a height of z = 4.5 m for a roughness type of “A”, $\mu_z = 1.09$

$$\mu_z' = \mu_z \times \beta_z = 1.09 \times 1.0 = 1.09$$

There is no modification for the reference wind pressure, so $w_0 = 0.45\ kN/m^2$

The origin value of reference wind pressure is taken from table E.5 in the Chinese load code GB 50009-2012, $w_{o,origin}\ b= 0.45$

Calculate the wind pressure characteristic value, w_k (wind intensity at height, z = 4.5 m):

$$w_k = \beta_z \mu_z w_0 = 1 \times 1.09 \times 0.45 = 0.491 \text{ kN/m}^2$$

The height factor for z = 9 is determined by linear interpolation between z = 5 ($\mu_z = 1.09$) and z = 10 ($\mu_z = 1.28$): $\mu_z = 1.24$.

Calculate the wind pressure characteristic value, w_k (wind intensity at height, z = 9 m):

$$w_k = \beta_z \mu_z w_0 = 1 \times 1.24 \times 0.45 = 0.558 \text{ kN/m}^2$$

Comparison

Result Type	Reference	STAAD.Pro	Difference	Comment
Wind pressure characteristic value, w_k at z = 4.5 m (kN/m^2)	0.491	0.490	negligible	
Wind pressure characteristic value, w_k at z = 9 m (kN/m^2)	0.558	0.559	negligible	

These values are compared with the generated intensity values that the STAAD.Pro Analytical workflow adds into the STAAD input file.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\GB 50009-2012 Wind Load Generation on Rectangular Building.std` is typically installed with the program.

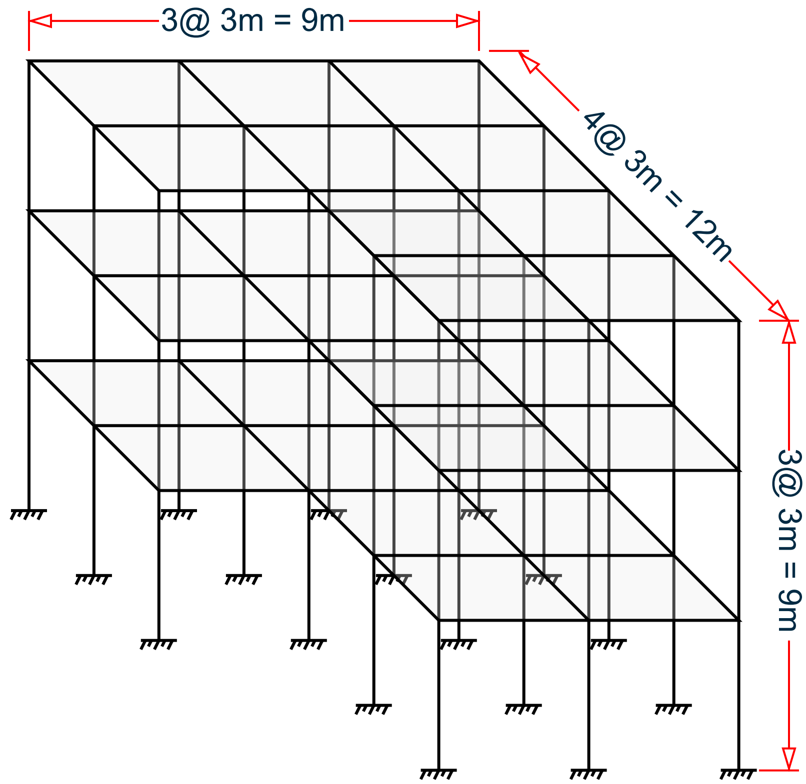
```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4.5 0; 3 0 9 0; 5 10 0 0; 6 10 4.5 0; 7 10 9 0; 17 0 0 6;
18 0 4.5 6; 19 0 9 6; 20 10 0 6; 21 10 4.5 6; 22 10 9 6; 23 2.5 4.5 6;
24 5 4.5 6; 25 7.5 4.5 6; 26 2.5 4.5 0; 27 5 4.5 0; 28 7.5 4.5 0; 29 2.5 9 0;
30 5 9 0; 31 7.5 9 0; 32 2.5 9 6; 33 5 9 6; 34 7.5 9 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 4 2 26; 5 3 29; 7 5 6; 8 6 7; 23 3 19; 24 6 21; 25 7 22;
30 17 18; 31 18 19; 32 18 23; 33 19 32; 34 20 21; 35 21 22; 36 23 24; 37 24
25;
38 25 21; 39 26 27; 40 27 28; 41 28 6; 42 29 30; 43 30 31; 44 31 7; 45 32 33;
46 33 34; 47 34 22; 48 23 26; 49 24 27; 50 25 28; 51 32 29; 52 33 30; 53 34
31;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 2 7 8 30 31 34 35 TABLE ST HW400X408
4 5 32 33 36 TO 47 TABLE ST HN500X200
23 TO 25 48 TO 53 TABLE ST HN300X150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 17 20 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
GB50009-2012:PARAMS Whole Wind_Group_ID 20210621162429414 Shape_Item "30 : -
CLOSED POLYGON BULDING" Second_Shape_UI "Rectangle" Interference_Factor 1 -
Shape_Factor Shape_Factor_Count 16 "LEFT Wind" "LEFT Face" 0.8 "LEFT Wind" -
"BACK Face" -0.7 "LEFT Wind" "RIGHT Face" -0.5 "LEFT Wind" "FRONT Face" -0.7 -
"RIGHT Wind" "LEFT Face" -0.5 "RIGHT Wind" "BACK Face" -0.7 "RIGHT Wind" -
"RIGHT Face" 0.8 "RIGHT Wind" "FRONT Face" -0.7 "FRONT Wind" "LEFT Face" -
-0.7 "FRONT Wind" "BACK Face" -0.5 "FRONT Wind" "RIGHT Face" -0.7 "FRONT -
Wind" "FRONT Face" 0.8 "BACK Wind" "LEFT Face" -0.7 "BACK Wind" "BACK Face" -
0.8 "BACK Wind" "RIGHT Face" -0.7 "BACK Wind" "FRONT Face" -0.5 -
Building_Height_H 9 Bottom_Elevation 0 z_Provide_Mothod 2 Segment_Count 5 -
Segment_Length 3 z_Special_List "4.5,9" Rough A -
Modify_Factor_of_Height_Factor 1 Province "" City_UI "" Refer_Wind_Press -
0.45 Modify_Factor_of_Press 1 Is_Calc_Vibration_Factor 0 Damp_Ratio 0.01 -
```

```
Basic_Period 1 Structure_Type 1 Windward_Width Width_Count 4 "LEFT Wind" -
Bottom_Width 0 Top_Width 0 "RIGHT Wind" Bottom_Width 0 Top_Width 0 "FRONT -
Wind" Bottom_Width 0 Top_Width 0 "BACK Wind" Bottom_Width 0 Top_Width 0 -
Generate_Load_Case 1 Select_Method 0 Face_Info Face_Count 4 "LEFT Face" -
Group "" Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 -
Is_Open 0 "BACK Face" Group "" Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max -
0 Z_Min 0 Z_Max 0 Is_Open 0 "RIGHT Face" Group "" Member_List "" X_Min 0 -
X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 Is_Open 0 "FRONT Face" Group "" -
Member_List "" X_Min 0 X_Max 0 Y_Min 0 Y_Max 0 Z_Min 0 Z_Max 0 Is_Open 0 -
Build_Rotation_In_Model 0 Each_Type "LEFT Face" "LEFT Wind"
!> END GENERATED DATA BLOCK
INT 0.49 0.559 HEIG  4.5  9
LOAD 1 LOADTYPE Wind  TITLE WIND FROM LEFT (+X) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "LEFT Wind"
WIND LOAD X 0.8 TYPE 1
WIND LOAD -Z -0.7 TYPE 1
WIND LOAD -X 0.5 TYPE 1
WIND LOAD -Z 0.7 TYPE 1
LOAD 2 LOADTYPE Wind  TITLE WIND FROM RIGHT (-X) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "RIGHT Wind"
WIND LOAD -X -0.5 TYPE 1
WIND LOAD -Z -0.7 TYPE 1
WIND LOAD X -0.8 TYPE 1
WIND LOAD -Z 0.7 TYPE 1
LOAD 3 LOADTYPE Wind  TITLE WIND FROM FRONT (-Z) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "FRONT Wind"
WIND LOAD -X -0.7 TYPE 1
WIND LOAD -Z -0.5 TYPE 1
WIND LOAD -X 0.7 TYPE 1
WIND LOAD Z -0.8 TYPE 1
LOAD 4 LOADTYPE Wind  TITLE WIND FROM BACK (+Z) LOAD CASE
* GB50009-2012:PARAMS Each_Load Wind_Group_ID 20210621162429414 "BACK Wind"
WIND LOAD -X -0.7 TYPE 1
WIND LOAD Z 0.8 TYPE 1
WIND LOAD -X 0.7 TYPE 1
WIND LOAD -Z 0.5 TYPE 1
PERFORM ANALYSIS
FINISH
```

V. IS 875 (Pt 3) 2015 WL Generation on Rectangular Clad Bldg 1

Determine the lateral wind loads on a three-story clad building per the IS 875 (Part 3): 2015 code.

Details



- Building length along wind when wind blow +X or -X direction, L_x = 9 m
- Building length across wind when wind blow +X or -X direction, L_z = 12 m
- Building length along wind when wind blow +Z or -Z direction, L_z = 12 m
- Building length across wind when wind blow +Z or -Z direction, L_x = 9 m
- Building Height, $h = 9\ m$
- Basic Wind Speed, $V_b = 47\ m/s$ (Location Agra as per ANNEX A of IS 875 3:2015)
- Terrain Category: Terrain Category 1 (Cl 6.3.2.1 as per IS 875 3:2015)
- Risk Coefficient Factors, $k_1 = 1.0$ (All general buildings from Table 1 of IS 875 3:2015)
- Topography Factor, $k_3 = 1.0$ (Up wind slope (Θ) below 3 degree from Cl 6.3.3.1 of IS 875 3:2015)
- Importance Factor for Cyclonic Region, $k_4 = 1.0$ (All other Structures from Cl 6.3.4 of IS 875 3:2015)
- Wind Directionality Factor for Buildings, $K_d = 0.9$ (From Cl 7.2.1 of IS 875 3:2015)
- Area Averaging Factor, $K_a = 0.9$ (From Table 4 of IS 875 3:2015)
- Combination Factor, $K_c = 0.9$ (From Cl 7.3.3.13 of IS 875 3:2015)

Validation

Calculate the design wind speed and pressure:

There is no variation in either for the first 10 m in height, thus the values are constant for the height of the building.

- Terrain & height factor, k_2 (from Table 2 of IS 875 (Part 3): 2015) = 1.05

The design wind speed:

$$V_z = V_b \times k_1 \times k_2 \times k_3 \times k_4 = 47\ (1.0)\ (1.05)\ (1.0)\ (1.0) = 49.35 \qquad \text{(Cl 6.3)}$$

Wind pressure at height, z = 10 m:

$$P_z = 0.6 \times V_z^2 = 0.6\ (49.35)^2 = 1.461\ \text{kN/m}^2 \qquad \text{(Cl. 7.2)}$$

Design wind pressure:

$$P_d = K_d \times K_a \times K_c \times P_z = (0.9)\ (0.9)\ (0.9)\ (1.461) = 1.065\ \text{kN/m}^2 \qquad \text{(Cl 7.2)}$$

Check the minimum wind pressure: $0.7 \times P_z = 1.023 < P_d$, OK.

Internal pressure coefficient:

- $C_{pi} = 0.2$ (Positive internal Coefficient as the building opening not more than 5 % From Cl 7. 3.2.1 of IS 875 3:2015)
- $C_{pi} = -0.2$ (Negative internal Coefficient as the building opening not more than 5 % From Cl 7. 3.2.1 of IS 875 3:2015)

External Pressure Coefficient when wind Positive (+X) Direction (From Table 5 of IS 875 3:2015):

- External Pressure Coefficient wind along +X on face A, $C_{pe} = 0.7$
- External Pressure Coefficient wind along +X on face B, $C_{pe} = -0.25$
- External Pressure Coefficient wind along +X on face C, $C_{pe} = -0.6$
- External Pressure Coefficient wind along +X on face D, $C_{pe} = -0.6$

Calculate the equivalent joint loads:

equiv joint load = influence area × avg. intensity of wind pressure × net pressure coefficient

<u>Face A (From +X Direction)</u>

The positive wind load is acting towards face A as per clause 7.3.1 of IS 875 (Part 3) : 2015 and the equivalent joint loads are calculated as follows

- At joint 1 (joints 13 , 33 , 34 , 45 , 46 , 66 , 78 similar)
  - For Positive Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [0.7 – (+0.2)] = 1.198 kN
  - For Negative Cpi , Equivalent joint load= 2.25 m^2 × 1.065 kN/m^2 × [0.7 – (-0.2)] = 2.157 kN
- At joint 5 (joints 9 , 17 , 29 , 37 , 38 , 41 , 42 , 50 , 62 , 70 , 74 similar)
  - For Positive Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [0.7 – (+0.2)] = 2.397 kN
  - For Negative Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [0.7 – (-0.2)] = 4.314 kN
- At joint 21 (joints 25 , 54 , 58 similar)
  - For Positive Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [0.7 – (+0.2)] = 4.79 kN
  - For Negative Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [0.7 – (-0.2)] = 8.62 kN

<u>Face B (From +X Direction)</u>

The negative wind load is acting away from face B as per clause 7.3.1 of IS 875 (Part 3) : 2015 and the equivalent joint loads are calculated as follows

- At joint 4 (joints 16 , 68 , 80 similar)
  - For Positive Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.25) – (+0.2)] = -1.079 kN
  - For Negative Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.25) – (-0.2)] = -0.12 kN

- At joint 8 (joints 12 , 20 , 32 , 36 , 48 , 52 , 64 , 72 , 76 similar)
  - For Positive Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.25) – (+0.2)] = -2.157 kN
  - For Negative Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.25) – (-0.2)] = -0.24 kN
- At joint 24 (joints 28 , 40 , 44 , 56 , 60 similar)
  - For Positive Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.25) – (+0.2)] = -4.314 kN
  - For Negative Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.25) – (-0.2)] = -0.479 kN

Face C (Wind From +X Direction)

The negative wind load is acting away from face C as per clause 7.3.1 of IS 875 (Part 3) : 2015 and the equivalent joint loads are calculated as follows

- At joint 1 (joints 4 , 13 , 16 similar)
  - For Positive Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -1.917 kN
  - For Negative Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -0.959 kN
- At joint 2 (joints 3 , 5 , 8 , 9 , 12 , 14 , 15 similar)
  - For Positive Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -3.835 kN
  - For Negative Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -1.917 kN
- At joint 6 (joints 7 , 10 , 11 similar)
  - For Positive Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -7.668 kN
  - For Negative Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -3.834 kN

Face D (From +X Direction)

The negative wind load is acting away from face D as per clause 7.3.1 of IS 875 (Part 3) : 2015 and the equivalent joint loads are calculated as follows

- At joint 33 (joints 34 , 45 , 46 , 66 , 68 , 78 , 80 similar)
  - For Positive Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -1.917 kN
  - For Negative Cpi , Equivalent joint load = 2.25 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -0.959 kN
- At joint 37 (joints 38 , 41 , 42 , 67 , 70 , 72 , 74 , 76 , 79 similar)
  - For Positive Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -3.835 kN
  - For Negative Cpi , Equivalent joint load = 4.5 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -1.917 kN
- At joint 71 (joint 75 similar)
  - For Positive Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.6) – (+0.2)] = -7.668 kN
  - For Negative Cpi , Equivalent joint load = 9 m^2 × 1.065 kN/m^2 × [(-0.6) – (-0.2)] = -3.834 kN

Results

Table 555: Representative joint loads due to wind in +X direction for positive internal presssure (+Cpi) (kN)

Result Type		Reference	STAAD.Pro	Difference	Comments
Face A	Joint 1	1.198	1.198	none	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Joint 5	2.397	2.397	none	
	Joint 21	4.790	4.790	none	
Face B	Joint 4	-1.079	-1.079	none	
	Joint 8	-2.157	-2.157	none	
	Joint 24	-4.134	-4.134	none	
Face C	Joint 1	-1.971	-1.971	none	
	Joint 2	-3.835	-3.835	none	
	Joint 6	-7.668	-7.668	none	
Face D	Joint 33	-1.917	-1.917	none	
	Joint 37	-3.835	-3.835	none	
	Joint 71	-7.668	-7.668	none	

Table 556: Representative joint loads due to wind in +X direction for negative internal pressure (-Cpi) (kN)

Result Type		Reference	STAAD.Pro	Difference	Comments
Face A	Joint 1	2.157	2.157	none	
	Joint 5	4.314	4.314	none	
	Joint 21	8.620	8.620	none	
Face B	Joint 4	-0.120	-0.120	none	
	Joint 8	-0.240	-0.240	none	
	Joint 24	-0.479	-0.479	none	
Face C	Joint 1	-0.959	-0.959	none	
	Joint 2	-1.917	-1.917	none	
	Joint 6	-3.834	-3.834	none	
Face D	Joint 33	-0.959	-0.959	none	
	Joint 37	-1.917	-1.917	none	
	Joint 71	-3.834	-3.834	none	

Note: These values are representative of the joint loads due to wind in the +X direction only. Similar calculations would be performed for the -X, +Z, and -Z directions.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\IS 875 (Pt 3) 2015 WL Generation on Rectangular Clad Bldg 1.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Oct-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0; 5 0 3 0; 6 3 3 0; 7 6 3 0; 8 9 3 0;
9 0 6 0; 10 3 6 0; 11 6 6 0; 12 9 6 0; 13 0 9 0; 14 3 9 0; 15 6 9 0; 16 9 9 0;
17 0 0 3; 18 3 0 3; 19 6 0 3; 20 9 0 3; 21 0 3 3; 22 3 3 3; 23 6 3 3; 24 9 3
3;
25 0 6 3; 26 3 6 3; 27 6 6 3; 28 9 6 3; 29 0 9 3; 30 3 9 3; 31 6 9 3; 32 9 9
3;
33 0 0 6; 34 3 0 6; 35 6 0 6; 36 9 0 6; 37 0 3 6; 38 3 3 6; 39 6 3 6; 40 9 3
6;
41 0 6 6; 42 3 6 6; 43 6 6 6; 44 9 6 6; 45 0 9 6; 46 3 9 6; 47 6 9 6; 48 9 9
6;
50 3 0 9; 51 6 0 9; 52 9 0 9; 54 3 3 9; 55 6 3 9; 56 9 3 9; 58 3 6 9; 59 6 6
9;
60 9 6 9; 62 3 9 9; 63 6 9 9; 64 9 9 9; 66 3 0 12; 67 6 0 12; 68 9 0 12;
70 3 3 12; 71 6 3 12; 72 9 3 12; 74 3 6 12; 75 6 6 12; 76 9 6 12; 78 3 9 12;
79 6 9 12; 80 9 9 12;
MEMBER INCIDENCES
1 5 6; 2 6 7; 3 7 8; 4 9 10; 5 10 11; 6 11 12; 7 13 14; 8 14 15; 9 15 16;
10 1 5; 11 2 6; 12 3 7; 13 4 8; 14 5 9; 15 6 10; 16 7 11; 17 8 12; 18 9 13;
19 10 14; 20 11 15; 21 12 16; 22 21 22; 23 22 23; 24 23 24; 25 25 26; 26 26
27;
27 27 28; 28 29 30; 29 30 31; 30 31 32; 31 17 21; 32 18 22; 33 19 23; 34 20
24;
35 21 25; 36 22 26; 37 23 27; 38 24 28; 39 25 29; 40 26 30; 41 27 31; 42 28
32;
43 37 38; 44 38 39; 45 39 40; 46 41 42; 47 42 43; 48 43 44; 49 45 46; 50 46
47;
51 47 48; 52 33 37; 53 34 38; 54 35 39; 55 36 40; 56 37 41; 57 38 42; 58 39
43;
59 40 44; 60 41 45; 61 42 46; 62 43 47; 63 44 48; 65 54 55; 66 55 56; 68 58
59;
69 59 60; 71 62 63; 72 63 64; 74 50 54; 75 51 55; 76 52 56; 78 54 58; 79 55
59;
80 56 60; 82 58 62; 83 59 63; 84 60 64; 86 70 71; 87 71 72; 89 74 75; 90 75
76;
92 78 79; 93 79 80; 95 66 70; 96 67 71; 97 68 72; 99 70 74; 100 71 75;
101 72 76; 103 74 78; 104 75 79; 105 76 80; 106 5 21; 107 6 22; 108 7 23;
109 8 24; 110 9 25; 111 10 26; 112 11 27; 113 12 28; 114 13 29; 115 14 30;
116 15 31; 117 16 32; 118 21 37; 119 22 38; 120 23 39; 121 24 40; 122 25 41;
123 26 42; 124 27 43; 125 28 44; 126 29 45; 127 30 46; 128 31 47; 129 32 48;
131 38 54; 132 39 55; 133 40 56; 135 42 58; 136 43 59; 137 44 60; 139 46 62;
140 47 63; 141 48 64; 143 54 70; 144 55 71; 145 56 72; 147 58 74; 148 59 75;
```

```
149 60 76; 151 62 78; 152 63 79; 153 64 80;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 63 65 66 68 69 71 72 74 TO 76 78 TO 80 82 TO 84 86 87 89 90 92 93 95 -
96 TO 97 99 TO 101 103 TO 129 131 TO 133 135 TO 137 139 TO 141 143 TO 145 -
147 TO 149 151 TO 153 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 TO 4 17 TO 20 33 TO 36 50 TO 52 66 TO 68 FIXED
DEFINE WIND LOAD
TYPE 1 IS875 RECT CLAD (+X)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1 0 0 9.000 0.000 3.000 0 Agra 47.000 0 1.000 0 4 1.050 -
1.050 1.050 1.050 1.000 2 1.000 0.002 0 9.000 12.000 9.000 0 1 1 0.200 0 0 4 -
0 4 0.700 -0.250 -0.600 -0.600 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 -
1 4 -0.250 0.700 -0.600 -0.600 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 -
2 4 -0.600 -0.600 0.700 -0.250 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 -
3 4 -0.600 -0.600 -0.250 0.700 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 -
0.900 0.900 0.900 TYPES 4 1 2 3 4 LOADS 8 1 2 3 4 5 6 7 8
!> END GENERATED DATA BLOCK
INT 1.06525 1.06525 1.06525 1.06525 HEIG  0  3  6  9
TYPE 2 IS875 RECT CLAD (-X)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 1.06525 1.06525 1.06525 1.06525 HEIG  0  3  6  9
TYPE 3 IS875 RECT CLAD (+Z)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 1.06525 1.06525 1.06525 1.06525 HEIG  0  3  6  9
TYPE 4 IS875 RECT CLAD (-Z)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 1.06525 1.06525 1.06525 1.06525 HEIG  0  3  6  9
LOAD 1 LOADTYPE Wind  TITLE Wind (+X) IS875 RECT CLAD [+Cpi]
WIND LOAD X 0.5 TYPE 1
WIND LOAD -X 0.45 TYPE 1
WIND LOAD -Z -0.8 TYPE 1
WIND LOAD -Z 0.8 TYPE 1
LOAD 2 LOADTYPE Wind  TITLE Wind (+X) IS875 RECT CLAD [-Cpi]
WIND LOAD X 0.9 TYPE 1
WIND LOAD -X 0.05 TYPE 1
WIND LOAD -Z -0.4 TYPE 1
WIND LOAD -Z 0.4 TYPE 1
LOAD 3 LOADTYPE Wind  TITLE Wind (-X) IS875 RECT CLAD [+Cpi]
```

```
WIND LOAD -X -0.45 TYPE 2
WIND LOAD X -0.5 TYPE 2
WIND LOAD -Z -0.8 TYPE 2
WIND LOAD -Z 0.8 TYPE 2
LOAD 4 LOADTYPE Wind  TITLE Wind (-X) IS875 RECT CLAD [-Cpi]
WIND LOAD -X -0.05 TYPE 2
WIND LOAD X -0.9 TYPE 2
WIND LOAD -Z -0.4 TYPE 2
WIND LOAD -Z 0.4 TYPE 2
LOAD 5 LOADTYPE Wind  TITLE Wind (+Z) IS875 RECT CLAD [+Cpi]
WIND LOAD -X -0.8 TYPE 3
WIND LOAD -X 0.8 TYPE 3
WIND LOAD Z 0.5 TYPE 3
WIND LOAD -Z 0.45 TYPE 3
LOAD 6 LOADTYPE Wind  TITLE Wind (+Z) IS875 RECT CLAD [-Cpi]
WIND LOAD -X -0.4 TYPE 3
WIND LOAD -X 0.4 TYPE 3
WIND LOAD Z 0.9 TYPE 3
WIND LOAD -Z 0.05 TYPE 3
LOAD 7 LOADTYPE Wind  TITLE Wind (-Z) IS875 RECT CLAD [+Cpi]
WIND LOAD -X -0.8 TYPE 4
WIND LOAD -X 0.8 TYPE 4
WIND LOAD -Z -0.45 TYPE 4
WIND LOAD Z -0.5 TYPE 4
LOAD 8 LOADTYPE Wind  TITLE Wind (-Z) IS875 RECT CLAD [-Cpi]
WIND LOAD -X -0.4 TYPE 4
WIND LOAD -X 0.4 TYPE 4
WIND LOAD -Z -0.05 TYPE 4
WIND LOAD Z -0.9 TYPE 4
PERFORM ANALYSIS PRINT ALL
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE WIND (+X) IS875 RECT CLAD [+CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      1.20      0.00        0.00      0.00      0.00      0.00
       5      2.40      0.00        0.00      0.00      0.00      0.00
       9      2.40      0.00        0.00      0.00      0.00      0.00
      13      1.20      0.00        0.00      0.00      0.00      0.00
      17      2.40      0.00        0.00      0.00      0.00      0.00
      21      4.79      0.00        0.00      0.00      0.00      0.00
      25      4.79      0.00        0.00      0.00      0.00      0.00
      29      2.40      0.00        0.00      0.00      0.00      0.00
      33      1.20      0.00        0.00      0.00      0.00      0.00
      34      1.20      0.00        0.00      0.00      0.00      0.00
      37      2.40      0.00        0.00      0.00      0.00      0.00
      38      2.40      0.00        0.00      0.00      0.00      0.00
      41      2.40      0.00        0.00      0.00      0.00      0.00
      42      2.40      0.00        0.00      0.00      0.00      0.00
      45      1.20      0.00        0.00      0.00      0.00      0.00
      46      1.20      0.00        0.00      0.00      0.00      0.00
      50      2.40      0.00        0.00      0.00      0.00      0.00
      54      4.79      0.00        0.00      0.00      0.00      0.00
      58      4.79      0.00        0.00      0.00      0.00      0.00
```

```
      62      2.40      0.00        0.00      0.00      0.00      0.00
      66      1.20      0.00        0.00      0.00      0.00      0.00
      70      2.40      0.00        0.00      0.00      0.00      0.00
      74      2.40      0.00        0.00      0.00      0.00      0.00
      78      1.20      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       4      1.08      0.00        0.00      0.00      0.00      0.00
       8      2.16      0.00        0.00      0.00      0.00      0.00
      12      2.16      0.00        0.00      0.00      0.00      0.00
      16      1.08      0.00        0.00      0.00      0.00      0.00
      20      2.16      0.00        0.00      0.00      0.00      0.00
      24      4.31      0.00        0.00      0.00      0.00      0.00
      28      4.31      0.00        0.00      0.00      0.00      0.00
      32      2.16      0.00        0.00      0.00      0.00      0.00
      36      2.16      0.00        0.00      0.00      0.00      0.00
      40      4.31      0.00        0.00      0.00      0.00      0.00
      44      4.31      0.00        0.00      0.00      0.00      0.00
      48      2.16      0.00        0.00      0.00      0.00      0.00
      52      2.16      0.00        0.00      0.00      0.00      0.00
      56      4.31      0.00        0.00      0.00      0.00      0.00
      60      4.31      0.00        0.00      0.00      0.00      0.00
      64      2.16      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
7
      68      1.08      0.00        0.00      0.00      0.00      0.00
      72      2.16      0.00        0.00      0.00      0.00      0.00
      76      2.16      0.00        0.00      0.00      0.00      0.00
      80      1.08      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00       -1.92      0.00      0.00      0.00
       2      0.00      0.00       -3.83      0.00      0.00      0.00
       3      0.00      0.00       -3.83      0.00      0.00      0.00
       4      0.00      0.00       -1.92      0.00      0.00      0.00
       5      0.00      0.00       -3.83      0.00      0.00      0.00
       6      0.00      0.00       -7.67      0.00      0.00      0.00
       7      0.00      0.00       -7.67      0.00      0.00      0.00
       8      0.00      0.00       -3.83      0.00      0.00      0.00
       9      0.00      0.00       -3.83      0.00      0.00      0.00
      10      0.00      0.00       -7.67      0.00      0.00      0.00
      11      0.00      0.00       -7.67      0.00      0.00      0.00
      12      0.00      0.00       -3.83      0.00      0.00      0.00
      13      0.00      0.00       -1.92      0.00      0.00      0.00
      14      0.00      0.00       -3.83      0.00      0.00      0.00
      15      0.00      0.00       -3.83      0.00      0.00      0.00
      16      0.00      0.00       -1.92      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      33      0.00      0.00        1.92      0.00      0.00      0.00
      34      0.00      0.00        1.92      0.00      0.00      0.00
      37      0.00      0.00        3.83      0.00      0.00      0.00
      38      0.00      0.00        3.83      0.00      0.00      0.00
      41      0.00      0.00        3.83      0.00      0.00      0.00
      42      0.00      0.00        3.83      0.00      0.00      0.00
      45      0.00      0.00        1.92      0.00      0.00      0.00
      46      0.00      0.00        1.92      0.00      0.00      0.00
      66      0.00      0.00        1.92      0.00      0.00      0.00
```

```
     67      0.00      0.00        3.83      0.00      0.00      0.00
     68      0.00      0.00        1.92      0.00      0.00      0.00
     70      0.00      0.00        3.83      0.00      0.00      0.00
     71      0.00      0.00        7.67      0.00      0.00      0.00
     72      0.00      0.00        3.83      0.00      0.00      0.00
     74      0.00      0.00        3.83      0.00      0.00      0.00
     75      0.00      0.00        7.67      0.00      0.00      0.00
     76      0.00      0.00        3.83      0.00      0.00      0.00
     78      0.00      0.00        1.92      0.00      0.00      0.00
     79      0.00      0.00        3.83      0.00      0.00      0.00
     80      0.00      0.00        1.92      0.00      0.00      0.00
   LOADING     2  LOADTYPE WIND  TITLE WIND (+X) IS875 RECT CLAD [-CPI]
   -----------
      STAAD SPACE                                             -- PAGE NO.
8
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      1      2.16      0.00        0.00      0.00      0.00      0.00
      5      4.31      0.00        0.00      0.00      0.00      0.00
      9      4.31      0.00        0.00      0.00      0.00      0.00
     13      2.16      0.00        0.00      0.00      0.00      0.00
     17      4.31      0.00        0.00      0.00      0.00      0.00
     21      8.63      0.00        0.00      0.00      0.00      0.00
     25      8.63      0.00        0.00      0.00      0.00      0.00
     29      4.31      0.00        0.00      0.00      0.00      0.00
     33      2.16      0.00        0.00      0.00      0.00      0.00
     34      2.16      0.00        0.00      0.00      0.00      0.00
     37      4.31      0.00        0.00      0.00      0.00      0.00
     38      4.31      0.00        0.00      0.00      0.00      0.00
     41      4.31      0.00        0.00      0.00      0.00      0.00
     42      4.31      0.00        0.00      0.00      0.00      0.00
     45      2.16      0.00        0.00      0.00      0.00      0.00
     46      2.16      0.00        0.00      0.00      0.00      0.00
     50      4.31      0.00        0.00      0.00      0.00      0.00
     54      8.63      0.00        0.00      0.00      0.00      0.00
     58      8.63      0.00        0.00      0.00      0.00      0.00
     62      4.31      0.00        0.00      0.00      0.00      0.00
     66      2.16      0.00        0.00      0.00      0.00      0.00
     70      4.31      0.00        0.00      0.00      0.00      0.00
     74      4.31      0.00        0.00      0.00      0.00      0.00
     78      2.16      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      4      0.12      0.00        0.00      0.00      0.00      0.00
      8      0.24      0.00        0.00      0.00      0.00      0.00
     12      0.24      0.00        0.00      0.00      0.00      0.00
     16      0.12      0.00        0.00      0.00      0.00      0.00
     20      0.24      0.00        0.00      0.00      0.00      0.00
     24      0.48      0.00        0.00      0.00      0.00      0.00
     28      0.48      0.00        0.00      0.00      0.00      0.00
     32      0.24      0.00        0.00      0.00      0.00      0.00
     36      0.24      0.00        0.00      0.00      0.00      0.00
     40      0.48      0.00        0.00      0.00      0.00      0.00
     44      0.48      0.00        0.00      0.00      0.00      0.00
     48      0.24      0.00        0.00      0.00      0.00      0.00
     52      0.24      0.00        0.00      0.00      0.00      0.00
     56      0.48      0.00        0.00      0.00      0.00      0.00
     60      0.48      0.00        0.00      0.00      0.00      0.00
```

```
     64      0.24      0.00        0.00      0.00      0.00      0.00
     68      0.12      0.00        0.00      0.00      0.00      0.00
     72      0.24      0.00        0.00      0.00      0.00      0.00
     76      0.24      0.00        0.00      0.00      0.00      0.00
     80      0.12      0.00        0.00      0.00      0.00      0.00
     STAAD SPACE                                          -- PAGE NO.
9
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      1      0.00      0.00       -0.96      0.00      0.00      0.00
      2      0.00      0.00       -1.92      0.00      0.00      0.00
      3      0.00      0.00       -1.92      0.00      0.00      0.00
      4      0.00      0.00       -0.96      0.00      0.00      0.00
      5      0.00      0.00       -1.92      0.00      0.00      0.00
      6      0.00      0.00       -3.83      0.00      0.00      0.00
      7      0.00      0.00       -3.83      0.00      0.00      0.00
      8      0.00      0.00       -1.92      0.00      0.00      0.00
      9      0.00      0.00       -1.92      0.00      0.00      0.00
     10      0.00      0.00       -3.83      0.00      0.00      0.00
     11      0.00      0.00       -3.83      0.00      0.00      0.00
     12      0.00      0.00       -1.92      0.00      0.00      0.00
     13      0.00      0.00       -0.96      0.00      0.00      0.00
     14      0.00      0.00       -1.92      0.00      0.00      0.00
     15      0.00      0.00       -1.92      0.00      0.00      0.00
     16      0.00      0.00       -0.96      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
     33      0.00      0.00        0.96      0.00      0.00      0.00
     34      0.00      0.00        0.96      0.00      0.00      0.00
     37      0.00      0.00        1.92      0.00      0.00      0.00
     38      0.00      0.00        1.92      0.00      0.00      0.00
     41      0.00      0.00        1.92      0.00      0.00      0.00
     42      0.00      0.00        1.92      0.00      0.00      0.00
     45      0.00      0.00        0.96      0.00      0.00      0.00
     46      0.00      0.00        0.96      0.00      0.00      0.00
     66      0.00      0.00        0.96      0.00      0.00      0.00
     67      0.00      0.00        1.92      0.00      0.00      0.00
     68      0.00      0.00        0.96      0.00      0.00      0.00
     70      0.00      0.00        1.92      0.00      0.00      0.00
     71      0.00      0.00        3.83      0.00      0.00      0.00
     72      0.00      0.00        1.92      0.00      0.00      0.00
     74      0.00      0.00        1.92      0.00      0.00      0.00
     75      0.00      0.00        3.83      0.00      0.00      0.00
     76      0.00      0.00        1.92      0.00      0.00      0.00
     78      0.00      0.00        0.96      0.00      0.00      0.00
     79      0.00      0.00        1.92      0.00      0.00      0.00
     80      0.00      0.00        0.96      0.00      0.00      0.00
```

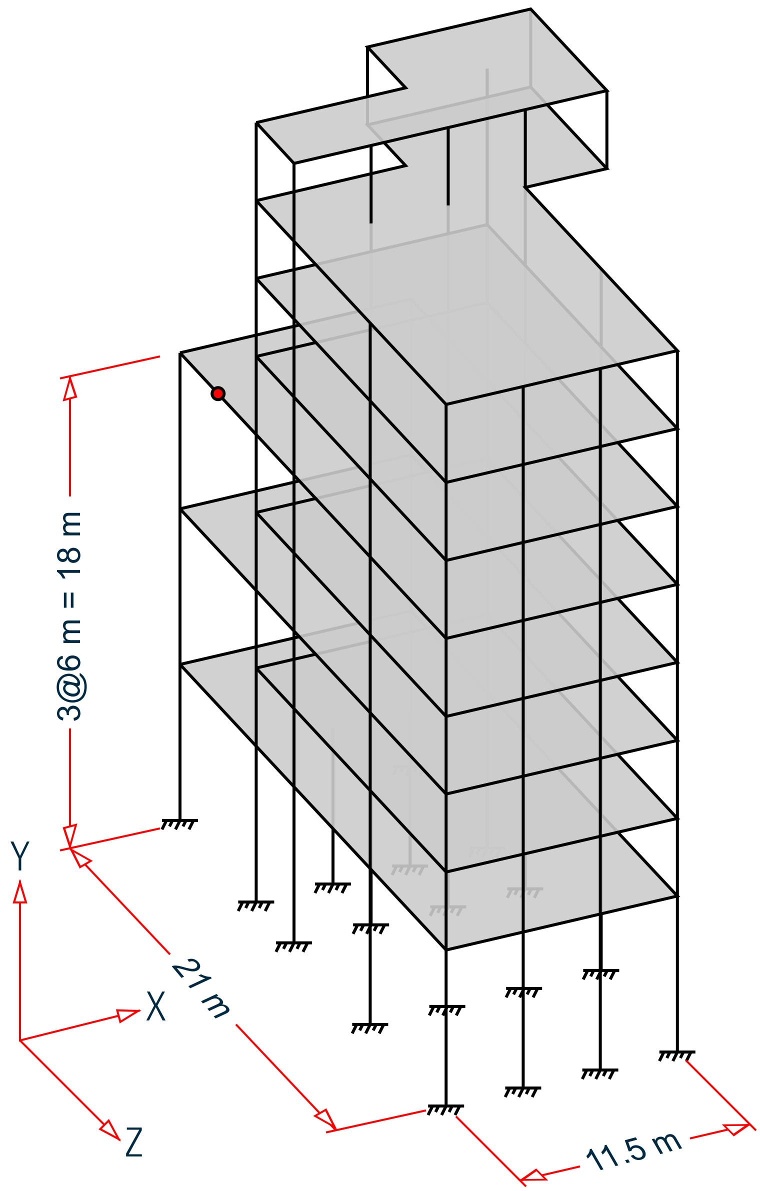
V. IS 875 (Pt 3) 2015 WL Generation on Rectangular Clad Bldg 2

Verify the calculated wind pressure on an edge node on a clad building per the IS 875 (Part 3): 2015 code.

Details

The following structure:

Not all members are displayed here for clarity.



- Building length along wind when wind blow +X or -X direction, $L_x = 11.5\ m$
- Building length across wind when wind blow +X or -X direction, $L_z = 21\ m$
- Building length along wind when wind blow +Z or -Z direction, $L_z = 21\ m$
- Building length across wind when wind blow +Z or -Z direction, $L_x = 11.5\ m$
- Building Height, $h = 30\ m$
- Basic Wind Speed, $V_b = 33\ m/s$ (Location: Bengaluru; as per Annex A of IS 875 3:2015)
- Terrain Category: Terrain Category 2 (Cl 6.3.2.1 as per IS 875 3:2015)
- Risk Coefficient Factors, $k_1 = 1.05$ (All general buildings from Table 1 of IS 875 3:2015)

- Topography Factor, k_3 = *1.15* (Up wind slope (Θ) below 3 degree from Cl 6.3.3.1 of IS 875 3:2015)
- Importance Factor for Cyclonic Region, k_4 = *1.0* (All other Structures from Cl 6.3.4 of IS 875 3:2015)
- Wind Directionality Factor for Buildings, K_d = *1* (From Cl 7.2.1 of IS 875 3:2015)
- Area Averaging Factor, K_a = *0.95* (From Table 4 of IS 875 3:2015)
- Combination Factor, K_c = *0.9* (From Cl 7.3.3.13 of IS 875 3:2015)

Validation

Table 557: Calculation of Variation in Design Wind Speed & Pressure with Height

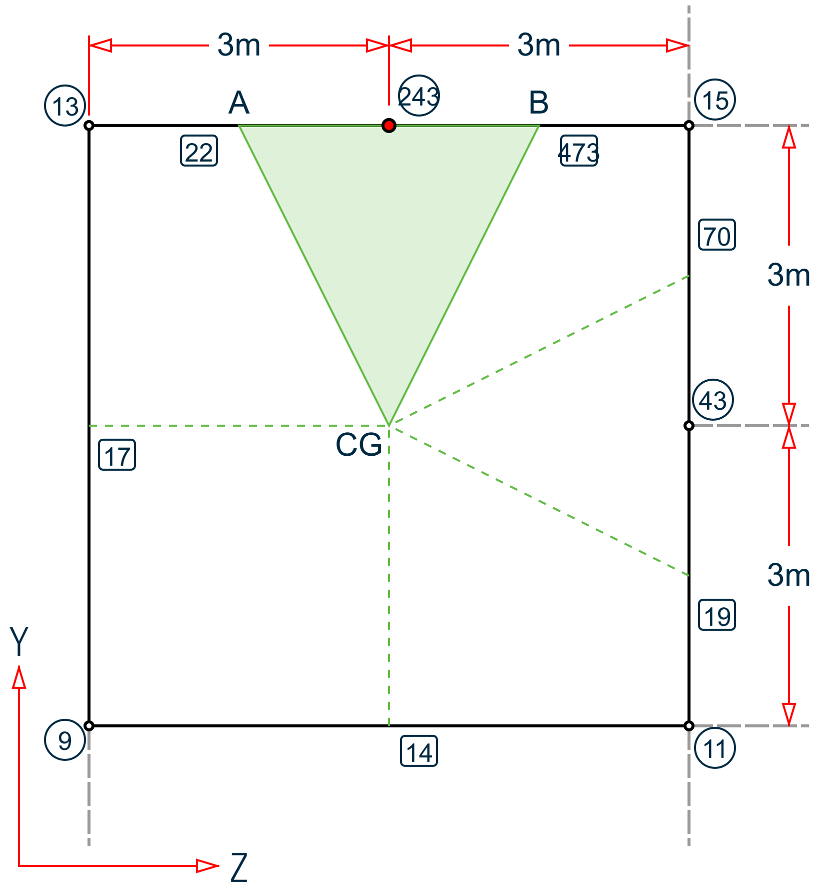
Height from Ground, m	Terrain & Height Factor, k_2 (from TABLE 2 of IS 875 Part 3:2015)	Design Wind Speed, $V_z = V_b \times k_1 \times k_2 \times k_3 \times k_4$ m/s (from Cl 6.3 of IS 875 Part 3:2015)	Wind Pressure at height z, $P_z = 0.6 \times V_z^2$ N/m² (from Cl 7.2 of IS 875 Part 3:2015)	Design Wind Pressure, $P_d = K_d \times K_a \times K_c \times P_z$ kN/m² (from Cl 7.2 of IS 875 Part 3:2015)	The Value of $0.7 \times P_z$ kN/m² (from Cl 7.2 of IS 875 Part 3:2015)	Status of the Value of $P_d \geq 0.7 \times P_z$ kN/m² (from Cl 7.2 of IS 875 Part 3:2015)
≤ 10	1	39.85	952.7	0.815	0.667	OK
12	1.02	40.64	991.2	0.847	0.694	OK
15	1.05	41.84	1050.3	0.898	0.735	OK
18	1.062	42.32	1074.5	0.919	0.752	OK
21	1.075	42.84	1101.0	0.941	0.771	OK
24	1.09	43.43	1131.9	0.968	0.792	OK
27	1.105	44.03	1163.3	0.995	0.814	OK
30	1.12	44.63	1195.1	1.022	0.837	OK

Internal Pressure Coefficient per Cl 7. 3.2.2 of IS 875 3:2015 for buildings with openings between 5 and 20%:

- C_{pi} = *0.5* (positive internal coefficient)
- C_{pi} = *-0.5* (negative internal coefficient)

Node 243 is at an elevation of 18 m, thus the average intensity of wind pressure in this area, $P_{d,\ z=18} = 0.919\ kN/m^2$.

<u>Tributary Area of Node 243</u>



For a given panel, draw straight lines from the center of gravity (CG) to the midpoint of the members. So, the panel region will now contain several quadrilaterals whose two sides are made of portions of the respective members (or the ground) and the other two sides are lines going from the CG to the midpoint of the corresponding members.

Points A and B are the midpoints of members 473 and 22, respectively. Points A and B for a closed panel with the CG. Further, point 243 is located along segment A-B, so in this case the quadrilateral reduced to a triangle. So, the influence area for joint 243 is A-243-B-CG, which has an area of:

$$A_{243} = (3 \times 3) / 2 = 4.5 \text{ m}^2$$

<u>Positive X Direction</u>

The positive wind load is acting towards face A as per clause 7.3.1 of IS 875 (Part 3) : 2015 and the equivalent joint loads are calculated as follows:

External pressure coefficient for wind in the +X direction (from Table 5 of IS 875 3:2015):

$$3/2 < h/w = 30 / 11.5 = 2.6 < 6$$

$$3/2 < l /w = 21 / 11.5 = 1.8 < 4$$

- External Pressure Coefficient wind along +X on face A, $C_{pe} = 0.7$
- External Pressure Coefficient wind along +X on face B, $C_{pe} = -0.4$
- External Pressure Coefficient wind along +X on face C, $C_{pe} = -0.7$

- External Pressure Coefficient wind along +X on face D, C_{pe} = *-0.7*

The equivalent joint loads are:

- For Positive Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [0.7 – (+0.5)] = 0.83\ kN$
- For Negative Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [0.7 – (-0.5)] = 4.96\ kN$

Negative X Direction

- For Positive Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.4 – (+0.5)] = 3.72\ kN$
- For Negative Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.4 – (-0.5)] = 0.41\ kN$

Positive Z Direction

- For Positive Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.5 - (+0.5)] = 4.13\ kN$
- For Negative Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.5 - (-0.5)] = 0\ kN$

Negative Z Direction

- For Positive Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.5 – (+0.5)] = 4.13\ kN$
- For Negative Cpi , equivalent joint load = $4.5\ m^2 \times 0.919\ kN/m^2 \times [-0.5 – (-0.5)] = 0\ kN$

Results

Table 558: Wind loads on Face A at Joint 243

Result Type	Reference	STAAD.Pro	Difference	Comments
Wind direction +X, positive internal pressure	0.83	0.83	none	
Wind direction +X, negative internal pressure	4.96	4.96	none	
Wind direction -X, positive internal pressure	3.72	3.72	none	
Wind direction -X, negative internal pressure	0.41	0.41	none	
Wind direction +Z, positive internal pressure	4.13	4.13	none	
Wind direction -Z, positive internal pressure	4.13	4.13	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\IS 875 (Pt 3) 2015 WL Generation on Rectangular Clad Bldg 2.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Nov-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0; 3 8.5 6 0; 4 8.5 0 0; 5 0 0 6; 6 0 6 6; 7 8.5 6 6; 8 8.5 0
6;
9 0 12 0; 10 8.5 12 0; 11 0 12 6; 12 8.5 12 6; 13 0 18 0; 14 8.5 18 0;
15 0 18 6; 16 8.5 18 6; 17 0 0 9; 18 0 6 9; 19 8.5 6 9; 20 8.5 0 9; 29 0 9 6;
30 8.5 9 6; 31 0 9 9; 32 8.5 9 9; 37 0 12 9; 38 8.5 12 9; 43 0 15 6;
44 8.5 15 6; 45 0 15 9; 46 8.5 15 9; 51 0 18 9; 52 8.5 18 9; 57 0 21 6;
58 8.5 21 6; 59 0 21 9; 60 8.5 21 9; 65 0 24 6; 66 8.5 24 6; 67 0 24 9;
68 8.5 24 9; 73 0 27 6; 74 8.5 27 6; 75 0 27 9; 76 8.5 27 9; 81 0 30 6;
82 8.5 30 6; 83 0 30 9; 84 8.5 30 9; 85 2.83333 6 6; 86 5.66667 6 6;
87 2.83333 12 6; 88 5.66667 12 6; 89 2.83333 18 6; 90 5.66667 18 6;
91 2.83333 6 9; 92 5.66667 6 9; 125 2.83333 30 6; 126 5.66667 30 6;
127 2.83333 21 6; 128 5.66667 21 6; 129 5.66667 24 6; 130 2.83333 24 6;
131 2.83333 27 6; 132 5.66667 27 6; 133 2.83333 30 9; 134 5.66667 30 9;
135 2.83333 27 9; 136 5.66667 27 9; 137 2.833 0 6; 138 2.833 0 9;
139 5.666 0 6; 140 5.666 0 9; 141 0 0 15; 142 0 6 15; 143 8.5 6 15;
144 8.5 0 15; 145 2.83333 6 15; 146 5.66667 6 15; 147 2.833 0 15;
148 5.666 0 15; 149 0 0 21; 150 0 6 21; 151 8.5 6 21; 152 8.5 0 21;
153 2.83333 6 21; 154 5.66667 6 21; 155 2.833 0 21; 156 5.666 0 21;
157 2.83333 9 9; 158 5.66667 9 9; 159 0 9 15; 160 8.5 9 15; 161 2.83333 9 15;
162 5.66667 9 15; 163 0 9 21; 164 8.5 9 21; 165 2.83333 9 21; 166 5.66667 9
21;
167 2.83333 12 9; 168 5.66667 12 9; 169 0 12 15; 170 8.5 12 15;
171 2.83333 12 15; 172 5.66667 12 15; 173 0 12 21; 174 8.5 12 21;
175 2.83333 12 21; 176 5.66667 12 21; 177 2.83333 15 9; 178 5.66667 15 9;
179 0 15 15; 180 8.5 15 15; 181 2.83333 15 15; 182 5.66667 15 15; 183 0 15 21;
184 8.5 15 21; 185 2.83333 15 21; 186 5.66667 15 21; 187 2.83333 18 9;
188 5.66667 18 9; 189 0 18 15; 190 8.5 18 15; 191 2.83333 18 15;
192 5.66667 18 15; 193 0 18 21; 194 8.5 18 21; 195 2.83333 18 21;
196 5.66667 18 21; 197 2.83333 21 9; 198 5.66667 21 9; 199 0 21 15;
200 8.5 21 15; 201 2.83333 21 15; 202 5.66667 21 15; 203 0 21 21;
204 8.5 21 21; 205 2.83333 21 21; 206 5.66667 21 21; 207 2.83333 24 9;
208 5.66667 24 9; 209 0 24 15; 210 8.5 24 15; 211 2.83333 24 15;
212 5.66667 24 15; 213 0 24 21; 214 8.5 24 21; 215 2.83333 24 21;
216 5.66667 24 21; 217 0 27 15; 218 8.5 27 15; 219 2.83333 27 15;
220 5.66667 27 15; 221 0 27 21; 222 8.5 27 21; 223 2.83333 27 21;
224 5.66667 27 21; 225 11.5 27 6; 226 11.5 27 9; 227 11.5 30 6; 228 11.5 30 9;
229 8.5 27 3; 230 8.5 30 3; 231 11.5 27 3; 232 11.5 30 3; 233 5.5 27 6;
234 5.5 30 6; 235 5.5 27 3; 236 5.5 30 3; 237 2.83333 18 0; 238 5.66667 18 0;
239 2.83333 9 6; 240 5.66667 9 6; 241 2.83333 15 6; 242 5.66667 15 6;
243 0 18 3; 244 8.5 18 3; 245 5.66667 18 3; 246 2.83333 18 3;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 7 8; 8 6 85; 9 2 9; 10 3 10;
11 6 29; 12 7 30; 13 9 10; 14 9 11; 15 10 12; 16 11 87; 17 9 13; 18 10 14;
19 11 43; 20 12 44; 21 13 237; 22 13 243; 23 14 244; 24 15 89; 25 6 18;
26 7 19; 27 17 18; 28 19 20; 29 18 91; 40 29 11; 41 30 12; 42 18 31; 43 19 32;
```

```
48 29 31; 49 30 32; 56 31 37; 57 32 38; 62 11 37; 63 12 38; 70 43 15; 71 44
16;
72 37 45; 73 38 46; 78 43 45; 79 44 46; 86 45 51; 87 46 52; 92 15 51; 93 16
52;
100 15 57; 101 16 58; 102 51 59; 103 52 60; 108 57 59; 109 58 60; 116 57 65;
117 58 66; 118 59 67; 119 60 68; 124 65 67; 125 66 68; 132 65 73; 133 66 74;
134 67 75; 135 68 76; 140 73 75; 141 74 76; 148 73 81; 149 74 82; 150 75 83;
151 76 84; 156 81 83; 157 82 84; 164 81 125; 165 57 127; 166 66 129;
167 73 131; 168 83 133; 169 75 135; 170 85 86; 171 86 7; 172 87 88; 173 88 12;
174 89 90; 175 90 16; 176 91 92; 177 92 19; 210 125 234; 211 126 82;
212 127 128; 213 128 58; 214 129 130; 215 130 65; 216 131 233; 217 132 74;
218 133 134; 219 134 84; 220 135 136; 221 136 76; 222 85 91; 223 86 92;
224 137 85; 225 138 91; 226 139 86; 227 140 92; 228 18 142; 229 19 143;
230 91 145; 231 92 146; 232 141 142; 233 143 144; 234 142 145; 235 145 146;
236 146 143; 237 147 145; 238 148 146; 239 142 150; 240 143 151; 241 145 153;
242 146 154; 243 149 150; 244 151 152; 245 150 153; 246 153 154; 247 154 151;
248 155 153; 249 156 154; 250 91 157; 251 92 158; 252 142 159; 253 143 160;
254 145 161; 255 146 162; 256 150 163; 257 151 164; 258 153 165; 259 154 166;
260 31 159; 261 32 160; 262 157 161; 263 158 162; 264 159 161; 265 161 162;
266 162 160; 267 159 163; 268 160 164; 269 161 165; 270 162 166; 271 163 165;
272 165 166; 273 166 164; 274 157 167; 275 158 168; 276 159 169; 277 160 170;
278 161 171; 279 162 172; 280 163 173; 281 164 174; 282 165 175; 283 166 176;
284 37 169; 285 38 170; 286 167 171; 287 168 172; 288 169 171; 289 171 172;
290 172 170; 291 169 173; 292 170 174; 293 171 175; 294 172 176; 295 173 175;
296 175 176; 297 176 174; 298 167 177; 299 168 178; 300 169 179; 301 170 180;
302 171 181; 303 172 182; 304 173 183; 305 174 184; 306 175 185; 307 176 186;
308 45 179; 309 46 180; 310 177 181; 311 178 182; 312 179 181; 313 181 182;
314 182 180; 315 179 183; 316 180 184; 317 181 185; 318 182 186; 319 183 185;
320 185 186; 321 186 184; 322 177 187; 323 178 188; 324 179 189; 325 180 190;
326 181 191; 327 182 192; 328 183 193; 329 184 194; 330 185 195; 331 186 196;
332 51 189; 333 52 190; 334 187 191; 335 188 192; 336 189 191; 337 191 192;
338 192 190; 339 189 193; 340 190 194; 341 191 195; 342 192 196; 343 193 195;
344 195 196; 345 196 194; 346 187 197; 347 188 198; 348 189 199; 349 190 200;
350 191 201; 351 192 202; 352 193 203; 353 194 204; 354 195 205; 355 196 206;
356 59 199; 357 60 200; 358 197 201; 359 198 202; 360 199 201; 361 201 202;
362 202 200; 363 199 203; 364 200 204; 365 201 205; 366 202 206; 367 203 205;
368 205 206; 369 206 204; 370 197 207; 371 198 208; 372 199 209; 373 200 210;
374 201 211; 375 202 212; 376 203 213; 377 204 214; 378 205 215; 379 206 216;
380 67 209; 381 68 210; 382 207 211; 383 208 212; 384 209 211; 385 211 212;
386 212 210; 387 209 213; 388 210 214; 389 211 215; 390 212 216; 391 213 215;
392 215 216; 393 216 214; 394 207 135; 395 208 136; 396 209 217; 397 210 218;
398 211 219; 399 212 220; 400 213 221; 401 214 222; 402 215 223; 403 216 224;
404 75 217; 405 76 218; 406 135 219; 407 136 220; 408 217 219; 409 219 220;
410 220 218; 411 217 221; 412 218 222; 413 219 223; 414 220 224; 415 221 223;
416 223 224; 417 224 222; 418 125 133; 419 126 134; 420 131 125; 421 132 126;
422 135 133; 423 136 134; 424 74 225; 425 76 226; 426 82 227; 427 84 228;
428 225 226; 429 225 227; 430 226 228; 431 227 228; 432 131 135; 433 132 136;
434 74 229; 435 82 230; 436 225 231; 437 227 232; 438 229 230; 439 229 231;
440 230 232; 441 231 232; 442 233 132; 443 74 233; 444 234 126; 445 82 234;
446 229 235; 447 230 236; 448 233 235; 449 234 236; 450 235 236; 451 237 238;
452 238 14; 453 238 245; 454 237 246; 455 29 239; 456 167 87; 457 168 88;
458 43 241; 459 187 89; 460 188 90; 461 197 127; 462 207 130; 463 198 128;
464 208 129; 465 239 240; 466 240 30; 467 241 242; 468 242 44; 469 157 239;
470 158 240; 471 177 241; 472 178 242; 473 243 15; 474 244 16; 475 245 90;
476 246 89; 477 244 245; 478 245 246; 479 246 243;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
```

```
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
G 9.28139e+06
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 29 40 TO 43 48 49 56 57 62 63 70 TO 73 78 79 86 87 92 93 100 TO 103 108 -
109 116 TO 119 124 125 132 TO 135 140 141 148 TO 151 156 157 164 TO 177 210 -
211 TO 479 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 17 20 137 TO 141 144 147 TO 149 152 155 156 FIXED
DEFINE WIND LOAD
TYPE 1 IS875 RECT CLAD (+X)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1 0 0 30.000 0.000 3.000 0 Bengaluru 33.000 3 1.050 1 11 -
1.000 1.000 1.000 1.000 1.020 1.050 1.062 1.075 1.090 1.105 1.120 1.150 2 -
1.000 0.020 1 11.500 21.000 30.000 0 1 1 0.500 0 0 4 0 4 0.700 -0.400 -0.500 -
-0.500 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 1 4 -0.400 0.700 -0.500 -
-0.500 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 2 4 -0.700 -0.700 0.800 -
-0.100 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 3 4 -0.700 -0.700 -
-0.100 0.800 NA NA 6 0.000 0.000 0.000 0.000 0.000 0.000 1 1.000 0.950 0.900 -
TYPES 4 1 2 3 4 LOADS 8 3 4 5 6 7 8 9 10
!> END GENERATED DATA BLOCK
INT 0.814553 0.814553 0.814553 0.814553 0.847461 0.898045 0.918689 0.941318 -
0.967771 0.99459 1.02178 HEIG 0 3 6 9 12 15 18 21 24 27 30
TYPE 2 IS875 RECT CLAD (-X)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 0.814553 0.814553 0.814553 0.814553 0.847461 0.898045 0.918689 0.941318 -
0.967771 0.99459 1.02178 HEIG 0 3 6 9 12 15 18 21 24 27 30
TYPE 3 IS875 RECT CLAD (+Z)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 0.814553 0.814553 0.814553 0.814553 0.847461 0.898045 0.918689 0.941318 -
0.967771 0.99459 1.02178 HEIG 0 3 6 9 12 15 18 21 24 27 30
TYPE 4 IS875 RECT CLAD (-Z)
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
IS875-2015:PARAMS 1  1
!> END GENERATED DATA BLOCK
INT 0.814553 0.814553 0.814553 0.814553 0.847461 0.898045 0.918689 0.941318 -
0.967771 0.99459 1.02178 HEIG 0 3 6 9 12 15 18 21 24 27 30
LOAD 1 LOADTYPE Dead  TITLE DL
SELFWEIGHT Y -1.2
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
2 4 5 8 13 TO 16 21 TO 26 29 48 49 62 63 78 79 92 93 108 109 124 125 140 141 -
156 157 164 TO 177 210 TO 223 228 TO 231 234 TO 236 239 245 TO 247 -
260 TO 273 284 TO 297 308 TO 321 332 TO 345 356 TO 369 380 TO 393 -
404 TO 419 424 TO 428 431 TO 437 439 440 442 TO 449 451 TO 479 UNI GY -12
LOAD 3 LOADTYPE Wind  TITLE Wind (+X) IS875 RECT CLAD [+Cpi]
WIND LOAD X 0.2 TYPE 1
```

```
WIND LOAD -X 0.9 TYPE 1
WIND LOAD -Z -1.2 TYPE 1
WIND LOAD -Z 1.2 TYPE 1
LOAD 4 LOADTYPE Wind  TITLE Wind (+X) IS875 RECT CLAD [-Cpi]
WIND LOAD X 1.2 TYPE 1
WIND LOAD X -0.1 TYPE 1
WIND LOAD -Z -0.2 TYPE 1
WIND LOAD -Z 0.2 TYPE 1
LOAD 5 LOADTYPE Wind  TITLE Wind (-X) IS875 RECT CLAD [+Cpi]
WIND LOAD -X -0.9 TYPE 2
WIND LOAD X -0.2 TYPE 2
WIND LOAD -Z -1.2 TYPE 2
WIND LOAD -Z 1.2 TYPE 2
LOAD 6 LOADTYPE Wind  TITLE Wind (-X) IS875 RECT CLAD [-Cpi]
WIND LOAD X 0.1 TYPE 2
WIND LOAD X -1.2 TYPE 2
WIND LOAD -Z -0.2 TYPE 2
WIND LOAD -Z 0.2 TYPE 2
LOAD 7 LOADTYPE Wind  TITLE Wind (+Z) IS875 RECT CLAD [+Cpi]
WIND LOAD -X -1 TYPE 3
WIND LOAD -X 1 TYPE 3
WIND LOAD Z 0.3 TYPE 3
WIND LOAD -Z 0.6 TYPE 3
LOAD 8 LOADTYPE Wind  TITLE Wind (+Z) IS875 RECT CLAD [-Cpi]
WIND LOAD Z 1.3 TYPE 3
WIND LOAD Z -0.4 TYPE 3
LOAD 9 LOADTYPE Wind  TITLE Wind (-Z) IS875 RECT CLAD [+Cpi]
WIND LOAD -X -1 TYPE 4
WIND LOAD -X 1 TYPE 4
WIND LOAD -Z -0.6 TYPE 4
WIND LOAD Z -0.3 TYPE 4
LOAD 10 LOADTYPE Wind  TITLE Wind (-Z) IS875 RECT CLAD [-Cpi]
WIND LOAD Z 0.4 TYPE 4
WIND LOAD Z -1.3 TYPE 4
LOAD COMB 100 COMB - 1.5 Dead + 1.5 Live
1 1.5 2 1.5
LOAD COMB 101 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (1)
1 1.2 2 1.2 3 1.2
LOAD COMB 102 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (2)
1 1.2 2 1.2 4 1.2
LOAD COMB 103 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (3)
1 1.2 2 1.2 5 1.2
LOAD COMB 104 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (4)
1 1.2 2 1.2 6 1.2
LOAD COMB 105 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (5)
1 1.2 2 1.2 7 1.2
LOAD COMB 106 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (6)
1 1.2 2 1.2 8 1.2
LOAD COMB 107 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (7)
1 1.2 2 1.2 9 1.2
LOAD COMB 108 COMB - 1.2 Dead + 1.2 Live + 1.2 Wind (8)
1 1.2 2 1.2 10 1.2
LOAD COMB 109 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (1)
1 1.2 2 1.2 3 -1.2
LOAD COMB 110 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (2)
1 1.2 2 1.2 4 -1.2
LOAD COMB 111 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (3)
1 1.2 2 1.2 5 -1.2
```

```
LOAD COMB 112 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (4)
1 1.2 2 1.2 6 -1.2
LOAD COMB 113 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (5)
1 1.2 2 1.2 7 -1.2
LOAD COMB 114 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (6)
1 1.2 2 1.2 8 -1.2
LOAD COMB 115 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (7)
1 1.2 2 1.2 9 -1.2
LOAD COMB 116 COMB - 1.2 Dead + 1.2 Live + -1.2 Wind (8)
1 1.2 2 1.2 10 -1.2
LOAD COMB 117 COMB - 1.2 Dead + 1.2 Live
1 1.2 2 1.2
LOAD COMB 118 COMB - 1.5 Dead + 1.5 Wind (1)
1 1.5 3 1.5
LOAD COMB 119 COMB - 1.5 Dead + 1.5 Wind (2)
1 1.5 4 1.5
LOAD COMB 120 COMB - 1.5 Dead + 1.5 Wind (3)
1 1.5 5 1.5
LOAD COMB 121 COMB - 1.5 Dead + 1.5 Wind (4)
1 1.5 6 1.5
LOAD COMB 122 COMB - 1.5 Dead + 1.5 Wind (5)
1 1.5 7 1.5
LOAD COMB 123 COMB - 1.5 Dead + 1.5 Wind (6)
1 1.5 8 1.5
LOAD COMB 124 COMB - 1.5 Dead + 1.5 Wind (7)
1 1.5 9 1.5
LOAD COMB 125 COMB - 1.5 Dead + 1.5 Wind (8)
1 1.5 10 1.5
LOAD COMB 126 COMB - 1.5 Dead + -1.5 Wind (1)
1 1.5 3 -1.5
LOAD COMB 127 COMB - 1.5 Dead + -1.5 Wind (2)
1 1.5 4 -1.5
LOAD COMB 128 COMB - 1.5 Dead + -1.5 Wind (3)
1 1.5 5 -1.5
LOAD COMB 129 COMB - 1.5 Dead + -1.5 Wind (4)
1 1.5 6 -1.5
LOAD COMB 130 COMB - 1.5 Dead + -1.5 Wind (5)
1 1.5 7 -1.5
LOAD COMB 131 COMB - 1.5 Dead + -1.5 Wind (6)
1 1.5 8 -1.5
LOAD COMB 132 COMB - 1.5 Dead + -1.5 Wind (7)
1 1.5 9 -1.5
LOAD COMB 133 COMB - 1.5 Dead + -1.5 Wind (8)
1 1.5 10 -1.5
LOAD COMB 134 COMB - 1.5 Dead
1 1.5
LOAD COMB 135 COMB - 0.9 Dead + 1.5 Wind (1)
1 0.9 3 1.5
LOAD COMB 136 COMB - 0.9 Dead + 1.5 Wind (2)
1 0.9 4 1.5
LOAD COMB 137 COMB - 0.9 Dead + 1.5 Wind (3)
1 0.9 5 1.5
LOAD COMB 138 COMB - 0.9 Dead + 1.5 Wind (4)
1 0.9 6 1.5
LOAD COMB 139 COMB - 0.9 Dead + 1.5 Wind (5)
1 0.9 7 1.5
LOAD COMB 140 COMB - 0.9 Dead + 1.5 Wind (6)
1 0.9 8 1.5
```

```
LOAD COMB 141 COMB - 0.9 Dead + 1.5 Wind (7)
1 0.9 9 1.5
LOAD COMB 142 COMB - 0.9 Dead + 1.5 Wind (8)
1 0.9 10 1.5
LOAD COMB 143 COMB - 0.9 Dead + -1.5 Wind (1)
1 0.9 3 -1.5
LOAD COMB 144 COMB - 0.9 Dead + -1.5 Wind (2)
1 0.9 4 -1.5
LOAD COMB 145 COMB - 0.9 Dead + -1.5 Wind (3)
1 0.9 5 -1.5
LOAD COMB 146 COMB - 0.9 Dead + -1.5 Wind (4)
1 0.9 6 -1.5
LOAD COMB 147 COMB - 0.9 Dead + -1.5 Wind (5)
1 0.9 7 -1.5
LOAD COMB 148 COMB - 0.9 Dead + -1.5 Wind (6)
1 0.9 8 -1.5
LOAD COMB 149 COMB - 0.9 Dead + -1.5 Wind (7)
1 0.9 9 -1.5
LOAD COMB 150 COMB - 0.9 Dead + -1.5 Wind (8)
1 0.9 10 -1.5
LOAD COMB 151 COMB - 0.9 Dead
1 0.9
PERFORM ANALYSIS PRINT ALL
FINISH
```

STAAD.Pro Output

```
   LOADING     3  LOADTYPE WIND  TITLE WIND (+X) IS875 RECT CLAD [+CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      1.47      0.00        0.00      0.00      0.00      0.00
       2      2.93      0.00        0.00      0.00      0.00      0.00
       5      2.20      0.00        0.00      0.00      0.00      0.00
       6      3.67      0.00        0.00      0.00      0.00      0.00
       9      3.14      0.00        0.00      0.00      0.00      0.00
      11      3.14      0.00        0.00      0.00      0.00      0.00
      13      1.24      0.00        0.00      0.00      0.00      0.00
      15      1.66      0.00        0.00      0.00      0.00      0.00
      17      2.20      0.00        0.00      0.00      0.00      0.00
      18      3.30      0.00        0.00      0.00      0.00      0.00
      29      1.50      0.00        0.00      0.00      0.00      0.00
      31      2.24      0.00        0.00      0.00      0.00      0.00
      37      2.36      0.00        0.00      0.00      0.00      0.00
      43      1.64      0.00        0.00      0.00      0.00      0.00
      45      2.45      0.00        0.00      0.00      0.00      0.00
      51      2.51      0.00        0.00      0.00      0.00      0.00
      57      0.86      0.00        0.00      0.00      0.00      0.00
      59      2.58      0.00        0.00      0.00      0.00      0.00
      65      0.88      0.00        0.00      0.00      0.00      0.00
      67      2.65      0.00        0.00      0.00      0.00      0.00
      73      0.91      0.00        0.00      0.00      0.00      0.00
      75      1.80      0.00        0.00      0.00      0.00      0.00
      81      0.46      0.00        0.00      0.00      0.00      0.00
      83      0.46      0.00        0.00      0.00      0.00      0.00
     141      2.93      0.00        0.00      0.00      0.00      0.00
     142      4.40      0.00        0.00      0.00      0.00      0.00
```

```
     149       1.47      0.00        0.00      0.00      0.00      0.00
     150       2.20      0.00        0.00      0.00      0.00      0.00
     159       2.99      0.00        0.00      0.00      0.00      0.00
     163       1.50      0.00        0.00      0.00      0.00      0.00
     169       3.14      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                             -- PAGE NO.
14
     173       1.57      0.00        0.00      0.00      0.00      0.00
     179       3.27      0.00        0.00      0.00      0.00      0.00
     183       1.64      0.00        0.00      0.00      0.00      0.00
     189       3.35      0.00        0.00      0.00      0.00      0.00
     193       1.67      0.00        0.00      0.00      0.00      0.00
     199       3.44      0.00        0.00      0.00      0.00      0.00
     203       1.72      0.00        0.00      0.00      0.00      0.00
     209       3.53      0.00        0.00      0.00      0.00      0.00
     213       1.77      0.00        0.00      0.00      0.00      0.00
     217       1.79      0.00        0.00      0.00      0.00      0.00
     221       0.90      0.00        0.00      0.00      0.00      0.00
     243       0.83      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       3      13.20      0.00        0.00      0.00      0.00      0.00
       4       6.60      0.00        0.00      0.00      0.00      0.00
       7      16.49      0.00        0.00      0.00      0.00      0.00
       8       9.90      0.00        0.00      0.00      0.00      0.00
      10      14.14      0.00        0.00      0.00      0.00      0.00
      12      14.14      0.00        0.00      0.00      0.00      0.00
      14       5.58      0.00        0.00      0.00      0.00      0.00
      16       7.49      0.00        0.00      0.00      0.00      0.00
      19      14.85      0.00        0.00      0.00      0.00      0.00
      20       9.90      0.00        0.00      0.00      0.00      0.00
      30       6.73      0.00        0.00      0.00      0.00      0.00
      32      10.10      0.00        0.00      0.00      0.00      0.00
      38      10.60      0.00        0.00      0.00      0.00      0.00
      44       7.36      0.00        0.00      0.00      0.00      0.00
      46      11.04      0.00        0.00      0.00      0.00      0.00
      52      11.30      0.00        0.00      0.00      0.00      0.00
      58       3.87      0.00        0.00      0.00      0.00      0.00
      60      11.60      0.00        0.00      0.00      0.00      0.00
      66       3.97      0.00        0.00      0.00      0.00      0.00
      68      11.92      0.00        0.00      0.00      0.00      0.00
      74       2.01      0.00        0.00      0.00      0.00      0.00
      76       6.04      0.00        0.00      0.00      0.00      0.00
     143      19.79      0.00        0.00      0.00      0.00      0.00
     144      13.20      0.00        0.00      0.00      0.00      0.00
     151       9.90      0.00        0.00      0.00      0.00      0.00
     152       6.60      0.00        0.00      0.00      0.00      0.00
     160      13.46      0.00        0.00      0.00      0.00      0.00
     164       6.73      0.00        0.00      0.00      0.00      0.00
     170      14.14      0.00        0.00      0.00      0.00      0.00
     174       7.07      0.00        0.00      0.00      0.00      0.00
     180      14.72      0.00        0.00      0.00      0.00      0.00
     184       7.36      0.00        0.00      0.00      0.00      0.00
     190      15.07      0.00        0.00      0.00      0.00      0.00
     194       7.53      0.00        0.00      0.00      0.00      0.00
     200      15.46      0.00        0.00      0.00      0.00      0.00
     204       7.73      0.00        0.00      0.00      0.00      0.00
     210      15.90      0.00        0.00      0.00      0.00      0.00
```

```
     214      7.95      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
15
     218      8.06      0.00        0.00      0.00      0.00      0.00
     222      4.03      0.00        0.00      0.00      0.00      0.00
     225      4.14      0.00        0.00      0.00      0.00      0.00
     226      2.07      0.00        0.00      0.00      0.00      0.00
     227      4.14      0.00        0.00      0.00      0.00      0.00
     228      2.07      0.00        0.00      0.00      0.00      0.00
     231      2.07      0.00        0.00      0.00      0.00      0.00
     232      2.07      0.00        0.00      0.00      0.00      0.00
     244      3.72      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00      -12.46      0.00      0.00      0.00
       2      0.00      0.00      -24.93      0.00      0.00      0.00
       3      0.00      0.00      -24.93      0.00      0.00      0.00
       4      0.00      0.00      -12.46      0.00      0.00      0.00
       9      0.00      0.00      -26.71      0.00      0.00      0.00
      10      0.00      0.00      -26.71      0.00      0.00      0.00
      13      0.00      0.00       -9.37      0.00      0.00      0.00
      14      0.00      0.00       -9.37      0.00      0.00      0.00
      15      0.00      0.00       -4.80      0.00      0.00      0.00
      16      0.00      0.00       -4.80      0.00      0.00      0.00
      57      0.00      0.00       -9.74      0.00      0.00      0.00
      58      0.00      0.00       -9.74      0.00      0.00      0.00
      65      0.00      0.00       -4.94      0.00      0.00      0.00
      66      0.00      0.00       -4.94      0.00      0.00      0.00
      67      0.00      0.00       -2.54      0.00      0.00      0.00
      68      0.00      0.00       -2.54      0.00      0.00      0.00
      73      0.00      0.00       -2.61      0.00      0.00      0.00
      75      0.00      0.00       -2.54      0.00      0.00      0.00
      76      0.00      0.00       -2.54      0.00      0.00      0.00
      81      0.00      0.00       -2.61      0.00      0.00      0.00
      89      0.00      0.00       -2.40      0.00      0.00      0.00
      90      0.00      0.00       -2.40      0.00      0.00      0.00
     125      0.00      0.00       -2.61      0.00      0.00      0.00
     127      0.00      0.00       -4.87      0.00      0.00      0.00
     128      0.00      0.00       -4.87      0.00      0.00      0.00
     129      0.00      0.00       -2.47      0.00      0.00      0.00
     130      0.00      0.00       -2.47      0.00      0.00      0.00
     131      0.00      0.00       -2.61      0.00      0.00      0.00
     133      0.00      0.00       -2.45      0.00      0.00      0.00
     135      0.00      0.00       -4.99      0.00      0.00      0.00
     136      0.00      0.00       -2.54      0.00      0.00      0.00
     207      0.00      0.00       -2.54      0.00      0.00      0.00
     208      0.00      0.00       -2.54      0.00      0.00      0.00
     229      0.00      0.00       -5.52      0.00      0.00      0.00
     230      0.00      0.00       -5.52      0.00      0.00      0.00
     231      0.00      0.00       -2.76      0.00      0.00      0.00
     232      0.00      0.00       -2.76      0.00      0.00      0.00
     235      0.00      0.00       -2.76      0.00      0.00      0.00
     236      0.00      0.00       -2.76      0.00      0.00      0.00
     237      0.00      0.00       -4.69      0.00      0.00      0.00
     238      0.00      0.00       -4.69      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
16
     189      0.00      0.00       -2.45      0.00      0.00      0.00
```

```
    192      0.00      0.00       -2.45      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
     75      0.00      0.00        2.61      0.00      0.00      0.00
     76      0.00      0.00        5.36      0.00      0.00      0.00
     83      0.00      0.00        2.61      0.00      0.00      0.00
     84      0.00      0.00        5.36      0.00      0.00      0.00
    133      0.00      0.00        5.21      0.00      0.00      0.00
    134      0.00      0.00        5.21      0.00      0.00      0.00
    135      0.00      0.00        5.21      0.00      0.00      0.00
    136      0.00      0.00        5.21      0.00      0.00      0.00
    149      0.00      0.00        4.15      0.00      0.00      0.00
    150      0.00      0.00        6.23      0.00      0.00      0.00
    151      0.00      0.00        6.23      0.00      0.00      0.00
    152      0.00      0.00        4.15      0.00      0.00      0.00
    153      0.00      0.00       12.46      0.00      0.00      0.00
    154      0.00      0.00       12.46      0.00      0.00      0.00
    155      0.00      0.00        8.31      0.00      0.00      0.00
    156      0.00      0.00        8.31      0.00      0.00      0.00
    163      0.00      0.00        4.24      0.00      0.00      0.00
    164      0.00      0.00        4.24      0.00      0.00      0.00
    165      0.00      0.00        8.48      0.00      0.00      0.00
    166      0.00      0.00        8.48      0.00      0.00      0.00
    173      0.00      0.00        4.45      0.00      0.00      0.00
    174      0.00      0.00        4.45      0.00      0.00      0.00
    175      0.00      0.00        8.90      0.00      0.00      0.00
    176      0.00      0.00        8.90      0.00      0.00      0.00
    183      0.00      0.00        4.63      0.00      0.00      0.00
    184      0.00      0.00        4.63      0.00      0.00      0.00
    185      0.00      0.00        9.27      0.00      0.00      0.00
    186      0.00      0.00        9.27      0.00      0.00      0.00
    193      0.00      0.00        4.74      0.00      0.00      0.00
    194      0.00      0.00        4.74      0.00      0.00      0.00
    195      0.00      0.00        9.49      0.00      0.00      0.00
    196      0.00      0.00        9.49      0.00      0.00      0.00
    203      0.00      0.00        4.87      0.00      0.00      0.00
    204      0.00      0.00        4.87      0.00      0.00      0.00
    205      0.00      0.00        9.74      0.00      0.00      0.00
    206      0.00      0.00        9.74      0.00      0.00      0.00
    213      0.00      0.00        5.00      0.00      0.00      0.00
    214      0.00      0.00        5.00      0.00      0.00      0.00
    215      0.00      0.00       10.01      0.00      0.00      0.00
    216      0.00      0.00       10.01      0.00      0.00      0.00
    221      0.00      0.00        2.54      0.00      0.00      0.00
    222      0.00      0.00        2.54      0.00      0.00      0.00
    223      0.00      0.00        5.07      0.00      0.00      0.00
    224      0.00      0.00        5.07      0.00      0.00      0.00
    226      0.00      0.00        2.76      0.00      0.00      0.00
    228      0.00      0.00        2.76      0.00      0.00      0.00
      STAAD SPACE                                                -- PAGE NO.
17
   LOADING     4  LOADTYPE WIND  TITLE WIND (+X) IS875 RECT CLAD [-CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      1      8.80      0.00        0.00      0.00      0.00      0.00
      2     17.59      0.00        0.00      0.00      0.00      0.00
      5     13.20      0.00        0.00      0.00      0.00      0.00
```

```
      6      21.99      0.00        0.00      0.00      0.00      0.00
      9      18.85      0.00        0.00      0.00      0.00      0.00
     11      18.85      0.00        0.00      0.00      0.00      0.00
     13       7.44      0.00        0.00      0.00      0.00      0.00
     15       9.98      0.00        0.00      0.00      0.00      0.00
     17      13.20      0.00        0.00      0.00      0.00      0.00
     18      19.79      0.00        0.00      0.00      0.00      0.00
     29       8.97      0.00        0.00      0.00      0.00      0.00
     31      13.46      0.00        0.00      0.00      0.00      0.00
     37      14.14      0.00        0.00      0.00      0.00      0.00
     43       9.81      0.00        0.00      0.00      0.00      0.00
     45      14.72      0.00        0.00      0.00      0.00      0.00
     51      15.07      0.00        0.00      0.00      0.00      0.00
     57       5.15      0.00        0.00      0.00      0.00      0.00
     59      15.46      0.00        0.00      0.00      0.00      0.00
     65       5.30      0.00        0.00      0.00      0.00      0.00
     67      15.90      0.00        0.00      0.00      0.00      0.00
     73       5.44      0.00        0.00      0.00      0.00      0.00
     75      10.82      0.00        0.00      0.00      0.00      0.00
     81       2.76      0.00        0.00      0.00      0.00      0.00
     83       2.76      0.00        0.00      0.00      0.00      0.00
    141      17.59      0.00        0.00      0.00      0.00      0.00
    142      26.39      0.00        0.00      0.00      0.00      0.00
    149       8.80      0.00        0.00      0.00      0.00      0.00
    150      13.20      0.00        0.00      0.00      0.00      0.00
    159      17.95      0.00        0.00      0.00      0.00      0.00
    163       8.97      0.00        0.00      0.00      0.00      0.00
    169      18.85      0.00        0.00      0.00      0.00      0.00
    173       9.43      0.00        0.00      0.00      0.00      0.00
    179      19.62      0.00        0.00      0.00      0.00      0.00
    183       9.81      0.00        0.00      0.00      0.00      0.00
    189      20.09      0.00        0.00      0.00      0.00      0.00
    193      10.04      0.00        0.00      0.00      0.00      0.00
    199      20.62      0.00        0.00      0.00      0.00      0.00
    203      10.31      0.00        0.00      0.00      0.00      0.00
    209      21.19      0.00        0.00      0.00      0.00      0.00
    213      10.60      0.00        0.00      0.00      0.00      0.00
    217      10.74      0.00        0.00      0.00      0.00      0.00
    221       5.37      0.00        0.00      0.00      0.00      0.00
    243       4.96      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                               -- PAGE NO.
18
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      3      -1.47      0.00        0.00      0.00      0.00      0.00
      4      -0.73      0.00        0.00      0.00      0.00      0.00
      7      -1.83      0.00        0.00      0.00      0.00      0.00
      8      -1.10      0.00        0.00      0.00      0.00      0.00
     10      -1.57      0.00        0.00      0.00      0.00      0.00
     12      -1.57      0.00        0.00      0.00      0.00      0.00
     14      -0.62      0.00        0.00      0.00      0.00      0.00
     16      -0.83      0.00        0.00      0.00      0.00      0.00
     19      -1.65      0.00        0.00      0.00      0.00      0.00
     20      -1.10      0.00        0.00      0.00      0.00      0.00
     30      -0.75      0.00        0.00      0.00      0.00      0.00
     32      -1.12      0.00        0.00      0.00      0.00      0.00
     38      -1.18      0.00        0.00      0.00      0.00      0.00
     44      -0.82      0.00        0.00      0.00      0.00      0.00
```

```
      46     -1.23      0.00        0.00      0.00      0.00      0.00
      52     -1.26      0.00        0.00      0.00      0.00      0.00
      58     -0.43      0.00        0.00      0.00      0.00      0.00
      60     -1.29      0.00        0.00      0.00      0.00      0.00
      66     -0.44      0.00        0.00      0.00      0.00      0.00
      68     -1.32      0.00        0.00      0.00      0.00      0.00
      74     -0.22      0.00        0.00      0.00      0.00      0.00
      76     -0.67      0.00        0.00      0.00      0.00      0.00
     143     -2.20      0.00        0.00      0.00      0.00      0.00
     144     -1.47      0.00        0.00      0.00      0.00      0.00
     151     -1.10      0.00        0.00      0.00      0.00      0.00
     152     -0.73      0.00        0.00      0.00      0.00      0.00
     160     -1.50      0.00        0.00      0.00      0.00      0.00
     164     -0.75      0.00        0.00      0.00      0.00      0.00
     170     -1.57      0.00        0.00      0.00      0.00      0.00
     174     -0.79      0.00        0.00      0.00      0.00      0.00
     180     -1.64      0.00        0.00      0.00      0.00      0.00
     184     -0.82      0.00        0.00      0.00      0.00      0.00
     190     -1.67      0.00        0.00      0.00      0.00      0.00
     194     -0.84      0.00        0.00      0.00      0.00      0.00
     200     -1.72      0.00        0.00      0.00      0.00      0.00
     204     -0.86      0.00        0.00      0.00      0.00      0.00
     210     -1.77      0.00        0.00      0.00      0.00      0.00
     214     -0.88      0.00        0.00      0.00      0.00      0.00
     218     -0.90      0.00        0.00      0.00      0.00      0.00
     222     -0.45      0.00        0.00      0.00      0.00      0.00
     225     -0.46      0.00        0.00      0.00      0.00      0.00
     226     -0.23      0.00        0.00      0.00      0.00      0.00
     227     -0.46      0.00        0.00      0.00      0.00      0.00
     228     -0.23      0.00        0.00      0.00      0.00      0.00
     231     -0.23      0.00        0.00      0.00      0.00      0.00
     232     -0.23      0.00        0.00      0.00      0.00      0.00
     244     -0.41      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
19
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00       -2.08      0.00      0.00      0.00
       2      0.00      0.00       -4.15      0.00      0.00      0.00
       3      0.00      0.00       -4.15      0.00      0.00      0.00
       4      0.00      0.00       -2.08      0.00      0.00      0.00
       9      0.00      0.00       -4.45      0.00      0.00      0.00
      10      0.00      0.00       -4.45      0.00      0.00      0.00
      13      0.00      0.00       -1.56      0.00      0.00      0.00
      14      0.00      0.00       -1.56      0.00      0.00      0.00
      15      0.00      0.00       -0.80      0.00      0.00      0.00
      16      0.00      0.00       -0.80      0.00      0.00      0.00
      57      0.00      0.00       -1.62      0.00      0.00      0.00
      58      0.00      0.00       -1.62      0.00      0.00      0.00
      65      0.00      0.00       -0.82      0.00      0.00      0.00
      66      0.00      0.00       -0.82      0.00      0.00      0.00
      67      0.00      0.00       -0.42      0.00      0.00      0.00
      68      0.00      0.00       -0.42      0.00      0.00      0.00
      73      0.00      0.00       -0.43      0.00      0.00      0.00
      75      0.00      0.00       -0.42      0.00      0.00      0.00
      76      0.00      0.00       -0.42      0.00      0.00      0.00
      81      0.00      0.00       -0.43      0.00      0.00      0.00
      89      0.00      0.00       -0.40      0.00      0.00      0.00
```

```
      90      0.00      0.00       -0.40      0.00      0.00      0.00
     125      0.00      0.00       -0.43      0.00      0.00      0.00
     127      0.00      0.00       -0.81      0.00      0.00      0.00
     128      0.00      0.00       -0.81      0.00      0.00      0.00
     129      0.00      0.00       -0.41      0.00      0.00      0.00
     130      0.00      0.00       -0.41      0.00      0.00      0.00
     131      0.00      0.00       -0.43      0.00      0.00      0.00
     133      0.00      0.00       -0.41      0.00      0.00      0.00
     135      0.00      0.00       -0.83      0.00      0.00      0.00
     136      0.00      0.00       -0.42      0.00      0.00      0.00
     207      0.00      0.00       -0.42      0.00      0.00      0.00
     208      0.00      0.00       -0.42      0.00      0.00      0.00
     229      0.00      0.00       -0.92      0.00      0.00      0.00
     230      0.00      0.00       -0.92      0.00      0.00      0.00
     231      0.00      0.00       -0.46      0.00      0.00      0.00
     232      0.00      0.00       -0.46      0.00      0.00      0.00
     235      0.00      0.00       -0.46      0.00      0.00      0.00
     236      0.00      0.00       -0.46      0.00      0.00      0.00
     237      0.00      0.00       -0.78      0.00      0.00      0.00
     238      0.00      0.00       -0.78      0.00      0.00      0.00
     189      0.00      0.00       -0.41      0.00      0.00      0.00
     192      0.00      0.00       -0.41      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      75      0.00      0.00        0.43      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
20
      76      0.00      0.00        0.89      0.00      0.00      0.00
      83      0.00      0.00        0.43      0.00      0.00      0.00
      84      0.00      0.00        0.89      0.00      0.00      0.00
     133      0.00      0.00        0.87      0.00      0.00      0.00
     134      0.00      0.00        0.87      0.00      0.00      0.00
     135      0.00      0.00        0.87      0.00      0.00      0.00
     136      0.00      0.00        0.87      0.00      0.00      0.00
     149      0.00      0.00        0.69      0.00      0.00      0.00
     150      0.00      0.00        1.04      0.00      0.00      0.00
     151      0.00      0.00        1.04      0.00      0.00      0.00
     152      0.00      0.00        0.69      0.00      0.00      0.00
     153      0.00      0.00        2.08      0.00      0.00      0.00
     154      0.00      0.00        2.08      0.00      0.00      0.00
     155      0.00      0.00        1.38      0.00      0.00      0.00
     156      0.00      0.00        1.38      0.00      0.00      0.00
     163      0.00      0.00        0.71      0.00      0.00      0.00
     164      0.00      0.00        0.71      0.00      0.00      0.00
     165      0.00      0.00        1.41      0.00      0.00      0.00
     166      0.00      0.00        1.41      0.00      0.00      0.00
     173      0.00      0.00        0.74      0.00      0.00      0.00
     174      0.00      0.00        0.74      0.00      0.00      0.00
     175      0.00      0.00        1.48      0.00      0.00      0.00
     176      0.00      0.00        1.48      0.00      0.00      0.00
     183      0.00      0.00        0.77      0.00      0.00      0.00
     184      0.00      0.00        0.77      0.00      0.00      0.00
     185      0.00      0.00        1.54      0.00      0.00      0.00
     186      0.00      0.00        1.54      0.00      0.00      0.00
     193      0.00      0.00        0.79      0.00      0.00      0.00
     194      0.00      0.00        0.79      0.00      0.00      0.00
     195      0.00      0.00        1.58      0.00      0.00      0.00
     196      0.00      0.00        1.58      0.00      0.00      0.00
```

```
     203      0.00      0.00        0.81      0.00      0.00      0.00
     204      0.00      0.00        0.81      0.00      0.00      0.00
     205      0.00      0.00        1.62      0.00      0.00      0.00
     206      0.00      0.00        1.62      0.00      0.00      0.00
     213      0.00      0.00        0.83      0.00      0.00      0.00
     214      0.00      0.00        0.83      0.00      0.00      0.00
     215      0.00      0.00        1.67      0.00      0.00      0.00
     216      0.00      0.00        1.67      0.00      0.00      0.00
     221      0.00      0.00        0.42      0.00      0.00      0.00
     222      0.00      0.00        0.42      0.00      0.00      0.00
     223      0.00      0.00        0.85      0.00      0.00      0.00
     224      0.00      0.00        0.85      0.00      0.00      0.00
     226      0.00      0.00        0.46      0.00      0.00      0.00
     228      0.00      0.00        0.46      0.00      0.00      0.00
   LOADING     5  LOADTYPE WIND  TITLE WIND (-X) IS875 RECT CLAD [+CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1     -6.60      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                            -- PAGE NO.
21
       2    -13.20      0.00        0.00      0.00      0.00      0.00
       5     -9.90      0.00        0.00      0.00      0.00      0.00
       6    -16.49      0.00        0.00      0.00      0.00      0.00
       9    -14.14      0.00        0.00      0.00      0.00      0.00
      11    -14.14      0.00        0.00      0.00      0.00      0.00
      13     -5.58      0.00        0.00      0.00      0.00      0.00
      15     -7.49      0.00        0.00      0.00      0.00      0.00
      17     -9.90      0.00        0.00      0.00      0.00      0.00
      18    -14.85      0.00        0.00      0.00      0.00      0.00
      29     -6.73      0.00        0.00      0.00      0.00      0.00
      31    -10.10      0.00        0.00      0.00      0.00      0.00
      37    -10.60      0.00        0.00      0.00      0.00      0.00
      43     -7.36      0.00        0.00      0.00      0.00      0.00
      45    -11.04      0.00        0.00      0.00      0.00      0.00
      51    -11.30      0.00        0.00      0.00      0.00      0.00
      57     -3.87      0.00        0.00      0.00      0.00      0.00
      59    -11.60      0.00        0.00      0.00      0.00      0.00
      65     -3.97      0.00        0.00      0.00      0.00      0.00
      67    -11.92      0.00        0.00      0.00      0.00      0.00
      73     -4.08      0.00        0.00      0.00      0.00      0.00
      75     -8.11      0.00        0.00      0.00      0.00      0.00
      81     -2.07      0.00        0.00      0.00      0.00      0.00
      83     -2.07      0.00        0.00      0.00      0.00      0.00
     141    -13.20      0.00        0.00      0.00      0.00      0.00
     142    -19.79      0.00        0.00      0.00      0.00      0.00
     149     -6.60      0.00        0.00      0.00      0.00      0.00
     150     -9.90      0.00        0.00      0.00      0.00      0.00
     159    -13.46      0.00        0.00      0.00      0.00      0.00
     163     -6.73      0.00        0.00      0.00      0.00      0.00
     169    -14.14      0.00        0.00      0.00      0.00      0.00
     173     -7.07      0.00        0.00      0.00      0.00      0.00
     179    -14.72      0.00        0.00      0.00      0.00      0.00
     183     -7.36      0.00        0.00      0.00      0.00      0.00
     189    -15.07      0.00        0.00      0.00      0.00      0.00
     193     -7.53      0.00        0.00      0.00      0.00      0.00
     199    -15.46      0.00        0.00      0.00      0.00      0.00
     203     -7.73      0.00        0.00      0.00      0.00      0.00
```

```
     209    -15.90      0.00        0.00       0.00       0.00       0.00
     213     -7.95      0.00        0.00       0.00       0.00       0.00
     217     -8.06      0.00        0.00       0.00       0.00       0.00
     221     -4.03      0.00        0.00       0.00       0.00       0.00
     243     -3.72      0.00        0.00       0.00       0.00       0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X      MOM-Y      MOM-Z
       3     -2.93      0.00        0.00       0.00       0.00       0.00
       4     -1.47      0.00        0.00       0.00       0.00       0.00
       7     -3.67      0.00        0.00       0.00       0.00       0.00
       8     -2.20      0.00        0.00       0.00       0.00       0.00
      10     -3.14      0.00        0.00       0.00       0.00       0.00
      12     -3.14      0.00        0.00       0.00       0.00       0.00
      14     -1.24      0.00        0.00       0.00       0.00       0.00
      16     -1.66      0.00        0.00       0.00       0.00       0.00
      STAAD SPACE                                                -- PAGE NO.
22
      19     -3.30      0.00        0.00       0.00       0.00       0.00
      20     -2.20      0.00        0.00       0.00       0.00       0.00
      30     -1.50      0.00        0.00       0.00       0.00       0.00
      32     -2.24      0.00        0.00       0.00       0.00       0.00
      38     -2.36      0.00        0.00       0.00       0.00       0.00
      44     -1.64      0.00        0.00       0.00       0.00       0.00
      46     -2.45      0.00        0.00       0.00       0.00       0.00
      52     -2.51      0.00        0.00       0.00       0.00       0.00
      58     -0.86      0.00        0.00       0.00       0.00       0.00
      60     -2.58      0.00        0.00       0.00       0.00       0.00
      66     -0.88      0.00        0.00       0.00       0.00       0.00
      68     -2.65      0.00        0.00       0.00       0.00       0.00
      74     -0.45      0.00        0.00       0.00       0.00       0.00
      76     -1.34      0.00        0.00       0.00       0.00       0.00
     143     -4.40      0.00        0.00       0.00       0.00       0.00
     144     -2.93      0.00        0.00       0.00       0.00       0.00
     151     -2.20      0.00        0.00       0.00       0.00       0.00
     152     -1.47      0.00        0.00       0.00       0.00       0.00
     160     -2.99      0.00        0.00       0.00       0.00       0.00
     164     -1.50      0.00        0.00       0.00       0.00       0.00
     170     -3.14      0.00        0.00       0.00       0.00       0.00
     174     -1.57      0.00        0.00       0.00       0.00       0.00
     180     -3.27      0.00        0.00       0.00       0.00       0.00
     184     -1.64      0.00        0.00       0.00       0.00       0.00
     190     -3.35      0.00        0.00       0.00       0.00       0.00
     194     -1.67      0.00        0.00       0.00       0.00       0.00
     200     -3.44      0.00        0.00       0.00       0.00       0.00
     204     -1.72      0.00        0.00       0.00       0.00       0.00
     210     -3.53      0.00        0.00       0.00       0.00       0.00
     214     -1.77      0.00        0.00       0.00       0.00       0.00
     218     -1.79      0.00        0.00       0.00       0.00       0.00
     222     -0.90      0.00        0.00       0.00       0.00       0.00
     225     -0.92      0.00        0.00       0.00       0.00       0.00
     226     -0.46      0.00        0.00       0.00       0.00       0.00
     227     -0.92      0.00        0.00       0.00       0.00       0.00
     228     -0.46      0.00        0.00       0.00       0.00       0.00
     231     -0.46      0.00        0.00       0.00       0.00       0.00
     232     -0.46      0.00        0.00       0.00       0.00       0.00
     244     -0.83      0.00        0.00       0.00       0.00       0.00
   JOINT LOAD - UNIT KN   METE
   JOINT    FORCE-X   FORCE-Y     FORCE-Z     MOM-X      MOM-Y      MOM-Z
```

```
      1      0.00      0.00      -12.46      0.00      0.00      0.00
      2      0.00      0.00      -24.93      0.00      0.00      0.00
      3      0.00      0.00      -24.93      0.00      0.00      0.00
      4      0.00      0.00      -12.46      0.00      0.00      0.00
      9      0.00      0.00      -26.71      0.00      0.00      0.00
     10      0.00      0.00      -26.71      0.00      0.00      0.00
     13      0.00      0.00       -9.37      0.00      0.00      0.00
     14      0.00      0.00       -9.37      0.00      0.00      0.00
     15      0.00      0.00       -4.80      0.00      0.00      0.00
     16      0.00      0.00       -4.80      0.00      0.00      0.00
     57      0.00      0.00       -9.74      0.00      0.00      0.00
     STAAD SPACE                                           -- PAGE NO.
23
     58      0.00      0.00       -9.74      0.00      0.00      0.00
     65      0.00      0.00       -4.94      0.00      0.00      0.00
     66      0.00      0.00       -4.94      0.00      0.00      0.00
     67      0.00      0.00       -2.54      0.00      0.00      0.00
     68      0.00      0.00       -2.54      0.00      0.00      0.00
     73      0.00      0.00       -2.61      0.00      0.00      0.00
     75      0.00      0.00       -2.54      0.00      0.00      0.00
     76      0.00      0.00       -2.54      0.00      0.00      0.00
     81      0.00      0.00       -2.61      0.00      0.00      0.00
     89      0.00      0.00       -2.40      0.00      0.00      0.00
     90      0.00      0.00       -2.40      0.00      0.00      0.00
    125      0.00      0.00       -2.61      0.00      0.00      0.00
    127      0.00      0.00       -4.87      0.00      0.00      0.00
    128      0.00      0.00       -4.87      0.00      0.00      0.00
    129      0.00      0.00       -2.47      0.00      0.00      0.00
    130      0.00      0.00       -2.47      0.00      0.00      0.00
    131      0.00      0.00       -2.61      0.00      0.00      0.00
    133      0.00      0.00       -2.45      0.00      0.00      0.00
    135      0.00      0.00       -4.99      0.00      0.00      0.00
    136      0.00      0.00       -2.54      0.00      0.00      0.00
    207      0.00      0.00       -2.54      0.00      0.00      0.00
    208      0.00      0.00       -2.54      0.00      0.00      0.00
    229      0.00      0.00       -5.52      0.00      0.00      0.00
    230      0.00      0.00       -5.52      0.00      0.00      0.00
    231      0.00      0.00       -2.76      0.00      0.00      0.00
    232      0.00      0.00       -2.76      0.00      0.00      0.00
    235      0.00      0.00       -2.76      0.00      0.00      0.00
    236      0.00      0.00       -2.76      0.00      0.00      0.00
    237      0.00      0.00       -4.69      0.00      0.00      0.00
    238      0.00      0.00       -4.69      0.00      0.00      0.00
    189      0.00      0.00       -2.45      0.00      0.00      0.00
    192      0.00      0.00       -2.45      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
     75      0.00      0.00        2.61      0.00      0.00      0.00
     76      0.00      0.00        5.36      0.00      0.00      0.00
     83      0.00      0.00        2.61      0.00      0.00      0.00
     84      0.00      0.00        5.36      0.00      0.00      0.00
    133      0.00      0.00        5.21      0.00      0.00      0.00
    134      0.00      0.00        5.21      0.00      0.00      0.00
    135      0.00      0.00        5.21      0.00      0.00      0.00
    136      0.00      0.00        5.21      0.00      0.00      0.00
    149      0.00      0.00        4.15      0.00      0.00      0.00
    150      0.00      0.00        6.23      0.00      0.00      0.00
    151      0.00      0.00        6.23      0.00      0.00      0.00
```

```
     152      0.00      0.00       4.15      0.00      0.00      0.00
     153      0.00      0.00      12.46      0.00      0.00      0.00
     154      0.00      0.00      12.46      0.00      0.00      0.00
     155      0.00      0.00       8.31      0.00      0.00      0.00
     156      0.00      0.00       8.31      0.00      0.00      0.00
     163      0.00      0.00       4.24      0.00      0.00      0.00
     164      0.00      0.00       4.24      0.00      0.00      0.00
      STAAD SPACE                                          -- PAGE NO.
24
     165      0.00      0.00       8.48      0.00      0.00      0.00
     166      0.00      0.00       8.48      0.00      0.00      0.00
     173      0.00      0.00       4.45      0.00      0.00      0.00
     174      0.00      0.00       4.45      0.00      0.00      0.00
     175      0.00      0.00       8.90      0.00      0.00      0.00
     176      0.00      0.00       8.90      0.00      0.00      0.00
     183      0.00      0.00       4.63      0.00      0.00      0.00
     184      0.00      0.00       4.63      0.00      0.00      0.00
     185      0.00      0.00       9.27      0.00      0.00      0.00
     186      0.00      0.00       9.27      0.00      0.00      0.00
     193      0.00      0.00       4.74      0.00      0.00      0.00
     194      0.00      0.00       4.74      0.00      0.00      0.00
     195      0.00      0.00       9.49      0.00      0.00      0.00
     196      0.00      0.00       9.49      0.00      0.00      0.00
     203      0.00      0.00       4.87      0.00      0.00      0.00
     204      0.00      0.00       4.87      0.00      0.00      0.00
     205      0.00      0.00       9.74      0.00      0.00      0.00
     206      0.00      0.00       9.74      0.00      0.00      0.00
     213      0.00      0.00       5.00      0.00      0.00      0.00
     214      0.00      0.00       5.00      0.00      0.00      0.00
     215      0.00      0.00      10.01      0.00      0.00      0.00
     216      0.00      0.00      10.01      0.00      0.00      0.00
     221      0.00      0.00       2.54      0.00      0.00      0.00
     222      0.00      0.00       2.54      0.00      0.00      0.00
     223      0.00      0.00       5.07      0.00      0.00      0.00
     224      0.00      0.00       5.07      0.00      0.00      0.00
     226      0.00      0.00       2.76      0.00      0.00      0.00
     228      0.00      0.00       2.76      0.00      0.00      0.00
   LOADING     6  LOADTYPE WIND  TITLE WIND (-X) IS875 RECT CLAD [-CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.73      0.00       0.00      0.00      0.00      0.00
       2      1.47      0.00       0.00      0.00      0.00      0.00
       5      1.10      0.00       0.00      0.00      0.00      0.00
       6      1.83      0.00       0.00      0.00      0.00      0.00
       9      1.57      0.00       0.00      0.00      0.00      0.00
      11      1.57      0.00       0.00      0.00      0.00      0.00
      13      0.62      0.00       0.00      0.00      0.00      0.00
      15      0.83      0.00       0.00      0.00      0.00      0.00
      17      1.10      0.00       0.00      0.00      0.00      0.00
      18      1.65      0.00       0.00      0.00      0.00      0.00
      29      0.75      0.00       0.00      0.00      0.00      0.00
      31      1.12      0.00       0.00      0.00      0.00      0.00
      37      1.18      0.00       0.00      0.00      0.00      0.00
      43      0.82      0.00       0.00      0.00      0.00      0.00
      45      1.23      0.00       0.00      0.00      0.00      0.00
      51      1.26      0.00       0.00      0.00      0.00      0.00
      57      0.43      0.00       0.00      0.00      0.00      0.00
```

```
      59      1.29      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
25
      65      0.44      0.00        0.00      0.00      0.00      0.00
      67      1.32      0.00        0.00      0.00      0.00      0.00
      73      0.45      0.00        0.00      0.00      0.00      0.00
      75      0.90      0.00        0.00      0.00      0.00      0.00
      81      0.23      0.00        0.00      0.00      0.00      0.00
      83      0.23      0.00        0.00      0.00      0.00      0.00
     141      1.47      0.00        0.00      0.00      0.00      0.00
     142      2.20      0.00        0.00      0.00      0.00      0.00
     149      0.73      0.00        0.00      0.00      0.00      0.00
     150      1.10      0.00        0.00      0.00      0.00      0.00
     159      1.50      0.00        0.00      0.00      0.00      0.00
     163      0.75      0.00        0.00      0.00      0.00      0.00
     169      1.57      0.00        0.00      0.00      0.00      0.00
     173      0.79      0.00        0.00      0.00      0.00      0.00
     179      1.64      0.00        0.00      0.00      0.00      0.00
     183      0.82      0.00        0.00      0.00      0.00      0.00
     189      1.67      0.00        0.00      0.00      0.00      0.00
     193      0.84      0.00        0.00      0.00      0.00      0.00
     199      1.72      0.00        0.00      0.00      0.00      0.00
     203      0.86      0.00        0.00      0.00      0.00      0.00
     209      1.77      0.00        0.00      0.00      0.00      0.00
     213      0.88      0.00        0.00      0.00      0.00      0.00
     217      0.90      0.00        0.00      0.00      0.00      0.00
     221      0.45      0.00        0.00      0.00      0.00      0.00
     243      0.41      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       3    -17.59      0.00        0.00      0.00      0.00      0.00
       4     -8.80      0.00        0.00      0.00      0.00      0.00
       7    -21.99      0.00        0.00      0.00      0.00      0.00
       8    -13.20      0.00        0.00      0.00      0.00      0.00
      10    -18.85      0.00        0.00      0.00      0.00      0.00
      12    -18.85      0.00        0.00      0.00      0.00      0.00
      14     -7.44      0.00        0.00      0.00      0.00      0.00
      16     -9.98      0.00        0.00      0.00      0.00      0.00
      19    -19.79      0.00        0.00      0.00      0.00      0.00
      20    -13.20      0.00        0.00      0.00      0.00      0.00
      30     -8.97      0.00        0.00      0.00      0.00      0.00
      32    -13.46      0.00        0.00      0.00      0.00      0.00
      38    -14.14      0.00        0.00      0.00      0.00      0.00
      44     -9.81      0.00        0.00      0.00      0.00      0.00
      46    -14.72      0.00        0.00      0.00      0.00      0.00
      52    -15.07      0.00        0.00      0.00      0.00      0.00
      58     -5.15      0.00        0.00      0.00      0.00      0.00
      60    -15.46      0.00        0.00      0.00      0.00      0.00
      66     -5.30      0.00        0.00      0.00      0.00      0.00
      68    -15.90      0.00        0.00      0.00      0.00      0.00
      74     -2.69      0.00        0.00      0.00      0.00      0.00
      76     -8.06      0.00        0.00      0.00      0.00      0.00
     143    -26.39      0.00        0.00      0.00      0.00      0.00
     144    -17.59      0.00        0.00      0.00      0.00      0.00
     151    -13.20      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
26
     152     -8.80      0.00        0.00      0.00      0.00      0.00
```

```
     160     -17.95      0.00        0.00      0.00      0.00      0.00
     164      -8.97      0.00        0.00      0.00      0.00      0.00
     170     -18.85      0.00        0.00      0.00      0.00      0.00
     174      -9.43      0.00        0.00      0.00      0.00      0.00
     180     -19.62      0.00        0.00      0.00      0.00      0.00
     184      -9.81      0.00        0.00      0.00      0.00      0.00
     190     -20.09      0.00        0.00      0.00      0.00      0.00
     194     -10.04      0.00        0.00      0.00      0.00      0.00
     200     -20.62      0.00        0.00      0.00      0.00      0.00
     204     -10.31      0.00        0.00      0.00      0.00      0.00
     210     -21.19      0.00        0.00      0.00      0.00      0.00
     214     -10.60      0.00        0.00      0.00      0.00      0.00
     218     -10.74      0.00        0.00      0.00      0.00      0.00
     222      -5.37      0.00        0.00      0.00      0.00      0.00
     225      -5.52      0.00        0.00      0.00      0.00      0.00
     226      -2.76      0.00        0.00      0.00      0.00      0.00
     227      -5.52      0.00        0.00      0.00      0.00      0.00
     228      -2.76      0.00        0.00      0.00      0.00      0.00
     231      -2.76      0.00        0.00      0.00      0.00      0.00
     232      -2.76      0.00        0.00      0.00      0.00      0.00
     244      -4.96      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00       -2.08      0.00      0.00      0.00
       2      0.00      0.00       -4.15      0.00      0.00      0.00
       3      0.00      0.00       -4.15      0.00      0.00      0.00
       4      0.00      0.00       -2.08      0.00      0.00      0.00
       9      0.00      0.00       -4.45      0.00      0.00      0.00
      10      0.00      0.00       -4.45      0.00      0.00      0.00
      13      0.00      0.00       -1.56      0.00      0.00      0.00
      14      0.00      0.00       -1.56      0.00      0.00      0.00
      15      0.00      0.00       -0.80      0.00      0.00      0.00
      16      0.00      0.00       -0.80      0.00      0.00      0.00
      57      0.00      0.00       -1.62      0.00      0.00      0.00
      58      0.00      0.00       -1.62      0.00      0.00      0.00
      65      0.00      0.00       -0.82      0.00      0.00      0.00
      66      0.00      0.00       -0.82      0.00      0.00      0.00
      67      0.00      0.00       -0.42      0.00      0.00      0.00
      68      0.00      0.00       -0.42      0.00      0.00      0.00
      73      0.00      0.00       -0.43      0.00      0.00      0.00
      75      0.00      0.00       -0.42      0.00      0.00      0.00
      76      0.00      0.00       -0.42      0.00      0.00      0.00
      81      0.00      0.00       -0.43      0.00      0.00      0.00
      89      0.00      0.00       -0.40      0.00      0.00      0.00
      90      0.00      0.00       -0.40      0.00      0.00      0.00
     125      0.00      0.00       -0.43      0.00      0.00      0.00
     127      0.00      0.00       -0.81      0.00      0.00      0.00
     128      0.00      0.00       -0.81      0.00      0.00      0.00
     129      0.00      0.00       -0.41      0.00      0.00      0.00
     130      0.00      0.00       -0.41      0.00      0.00      0.00
     131      0.00      0.00       -0.43      0.00      0.00      0.00
      STAAD SPACE                                               -- PAGE NO.
27
     133      0.00      0.00       -0.41      0.00      0.00      0.00
     135      0.00      0.00       -0.83      0.00      0.00      0.00
     136      0.00      0.00       -0.42      0.00      0.00      0.00
     207      0.00      0.00       -0.42      0.00      0.00      0.00
     208      0.00      0.00       -0.42      0.00      0.00      0.00
```

```
     229      0.00      0.00       -0.92      0.00      0.00      0.00
     230      0.00      0.00       -0.92      0.00      0.00      0.00
     231      0.00      0.00       -0.46      0.00      0.00      0.00
     232      0.00      0.00       -0.46      0.00      0.00      0.00
     235      0.00      0.00       -0.46      0.00      0.00      0.00
     236      0.00      0.00       -0.46      0.00      0.00      0.00
     237      0.00      0.00       -0.78      0.00      0.00      0.00
     238      0.00      0.00       -0.78      0.00      0.00      0.00
     189      0.00      0.00       -0.41      0.00      0.00      0.00
     192      0.00      0.00       -0.41      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      75      0.00      0.00        0.43      0.00      0.00      0.00
      76      0.00      0.00        0.89      0.00      0.00      0.00
      83      0.00      0.00        0.43      0.00      0.00      0.00
      84      0.00      0.00        0.89      0.00      0.00      0.00
     133      0.00      0.00        0.87      0.00      0.00      0.00
     134      0.00      0.00        0.87      0.00      0.00      0.00
     135      0.00      0.00        0.87      0.00      0.00      0.00
     136      0.00      0.00        0.87      0.00      0.00      0.00
     149      0.00      0.00        0.69      0.00      0.00      0.00
     150      0.00      0.00        1.04      0.00      0.00      0.00
     151      0.00      0.00        1.04      0.00      0.00      0.00
     152      0.00      0.00        0.69      0.00      0.00      0.00
     153      0.00      0.00        2.08      0.00      0.00      0.00
     154      0.00      0.00        2.08      0.00      0.00      0.00
     155      0.00      0.00        1.38      0.00      0.00      0.00
     156      0.00      0.00        1.38      0.00      0.00      0.00
     163      0.00      0.00        0.71      0.00      0.00      0.00
     164      0.00      0.00        0.71      0.00      0.00      0.00
     165      0.00      0.00        1.41      0.00      0.00      0.00
     166      0.00      0.00        1.41      0.00      0.00      0.00
     173      0.00      0.00        0.74      0.00      0.00      0.00
     174      0.00      0.00        0.74      0.00      0.00      0.00
     175      0.00      0.00        1.48      0.00      0.00      0.00
     176      0.00      0.00        1.48      0.00      0.00      0.00
     183      0.00      0.00        0.77      0.00      0.00      0.00
     184      0.00      0.00        0.77      0.00      0.00      0.00
     185      0.00      0.00        1.54      0.00      0.00      0.00
     186      0.00      0.00        1.54      0.00      0.00      0.00
     193      0.00      0.00        0.79      0.00      0.00      0.00
     194      0.00      0.00        0.79      0.00      0.00      0.00
     195      0.00      0.00        1.58      0.00      0.00      0.00
     196      0.00      0.00        1.58      0.00      0.00      0.00
     203      0.00      0.00        0.81      0.00      0.00      0.00
     204      0.00      0.00        0.81      0.00      0.00      0.00
     205      0.00      0.00        1.62      0.00      0.00      0.00
      STAAD SPACE                                               -- PAGE NO.
28
     206      0.00      0.00        1.62      0.00      0.00      0.00
     213      0.00      0.00        0.83      0.00      0.00      0.00
     214      0.00      0.00        0.83      0.00      0.00      0.00
     215      0.00      0.00        1.67      0.00      0.00      0.00
     216      0.00      0.00        1.67      0.00      0.00      0.00
     221      0.00      0.00        0.42      0.00      0.00      0.00
     222      0.00      0.00        0.42      0.00      0.00      0.00
     223      0.00      0.00        0.85      0.00      0.00      0.00
     224      0.00      0.00        0.85      0.00      0.00      0.00
```

```
    226      0.00      0.00        0.46      0.00      0.00      0.00
    228      0.00      0.00        0.46      0.00      0.00      0.00
   LOADING     7  LOADTYPE WIND  TITLE WIND (+Z) IS875 RECT CLAD [+CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1     -7.33      0.00        0.00      0.00      0.00      0.00
       2    -14.66      0.00        0.00      0.00      0.00      0.00
       5    -11.00      0.00        0.00      0.00      0.00      0.00
       6    -18.33      0.00        0.00      0.00      0.00      0.00
       9    -15.71      0.00        0.00      0.00      0.00      0.00
      11    -15.71      0.00        0.00      0.00      0.00      0.00
      13     -6.20      0.00        0.00      0.00      0.00      0.00
      15     -8.32      0.00        0.00      0.00      0.00      0.00
      17    -11.00      0.00        0.00      0.00      0.00      0.00
      18    -16.49      0.00        0.00      0.00      0.00      0.00
      29     -7.48      0.00        0.00      0.00      0.00      0.00
      31    -11.22      0.00        0.00      0.00      0.00      0.00
      37    -11.78      0.00        0.00      0.00      0.00      0.00
      43     -8.18      0.00        0.00      0.00      0.00      0.00
      45    -12.26      0.00        0.00      0.00      0.00      0.00
      51    -12.56      0.00        0.00      0.00      0.00      0.00
      57     -4.30      0.00        0.00      0.00      0.00      0.00
      59    -12.89      0.00        0.00      0.00      0.00      0.00
      65     -4.42      0.00        0.00      0.00      0.00      0.00
      67    -13.25      0.00        0.00      0.00      0.00      0.00
      73     -4.54      0.00        0.00      0.00      0.00      0.00
      75     -9.01      0.00        0.00      0.00      0.00      0.00
      81     -2.30      0.00        0.00      0.00      0.00      0.00
      83     -2.30      0.00        0.00      0.00      0.00      0.00
     141    -14.66      0.00        0.00      0.00      0.00      0.00
     142    -21.99      0.00        0.00      0.00      0.00      0.00
     149     -7.33      0.00        0.00      0.00      0.00      0.00
     150    -11.00      0.00        0.00      0.00      0.00      0.00
     159    -14.96      0.00        0.00      0.00      0.00      0.00
     163     -7.48      0.00        0.00      0.00      0.00      0.00
     169    -15.71      0.00        0.00      0.00      0.00      0.00
     173     -7.85      0.00        0.00      0.00      0.00      0.00
     179    -16.35      0.00        0.00      0.00      0.00      0.00
     183     -8.18      0.00        0.00      0.00      0.00      0.00
     189    -16.74      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                          -- PAGE NO.
29
     193     -8.37      0.00        0.00      0.00      0.00      0.00
     199    -17.18      0.00        0.00      0.00      0.00      0.00
     203     -8.59      0.00        0.00      0.00      0.00      0.00
     209    -17.66      0.00        0.00      0.00      0.00      0.00
     213     -8.83      0.00        0.00      0.00      0.00      0.00
     217     -8.95      0.00        0.00      0.00      0.00      0.00
     221     -4.48      0.00        0.00      0.00      0.00      0.00
     243     -4.13      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       3     14.66      0.00        0.00      0.00      0.00      0.00
       4      7.33      0.00        0.00      0.00      0.00      0.00
       7     18.33      0.00        0.00      0.00      0.00      0.00
       8     11.00      0.00        0.00      0.00      0.00      0.00
      10     15.71      0.00        0.00      0.00      0.00      0.00
```

```
      12     15.71      0.00        0.00       0.00       0.00       0.00
      14      6.20      0.00        0.00       0.00       0.00       0.00
      16      8.32      0.00        0.00       0.00       0.00       0.00
      19     16.49      0.00        0.00       0.00       0.00       0.00
      20     11.00      0.00        0.00       0.00       0.00       0.00
      30      7.48      0.00        0.00       0.00       0.00       0.00
      32     11.22      0.00        0.00       0.00       0.00       0.00
      38     11.78      0.00        0.00       0.00       0.00       0.00
      44      8.18      0.00        0.00       0.00       0.00       0.00
      46     12.26      0.00        0.00       0.00       0.00       0.00
      52     12.56      0.00        0.00       0.00       0.00       0.00
      58      4.30      0.00        0.00       0.00       0.00       0.00
      60     12.89      0.00        0.00       0.00       0.00       0.00
      66      4.42      0.00        0.00       0.00       0.00       0.00
      68     13.25      0.00        0.00       0.00       0.00       0.00
      74      2.24      0.00        0.00       0.00       0.00       0.00
      76      6.71      0.00        0.00       0.00       0.00       0.00
     143     21.99      0.00        0.00       0.00       0.00       0.00
     144     14.66      0.00        0.00       0.00       0.00       0.00
     151     11.00      0.00        0.00       0.00       0.00       0.00
     152      7.33      0.00        0.00       0.00       0.00       0.00
     160     14.96      0.00        0.00       0.00       0.00       0.00
     164      7.48      0.00        0.00       0.00       0.00       0.00
     170     15.71      0.00        0.00       0.00       0.00       0.00
     174      7.85      0.00        0.00       0.00       0.00       0.00
     180     16.35      0.00        0.00       0.00       0.00       0.00
     184      8.18      0.00        0.00       0.00       0.00       0.00
     190     16.74      0.00        0.00       0.00       0.00       0.00
     194      8.37      0.00        0.00       0.00       0.00       0.00
     200     17.18      0.00        0.00       0.00       0.00       0.00
     204      8.59      0.00        0.00       0.00       0.00       0.00
     210     17.66      0.00        0.00       0.00       0.00       0.00
     214      8.83      0.00        0.00       0.00       0.00       0.00
     218      8.95      0.00        0.00       0.00       0.00       0.00
     222      4.48      0.00        0.00       0.00       0.00       0.00
     225      4.60      0.00        0.00       0.00       0.00       0.00
     226      2.30      0.00        0.00       0.00       0.00       0.00
      STAAD SPACE                                             -- PAGE NO.
30
     227      4.60      0.00        0.00       0.00       0.00       0.00
     228      2.30      0.00        0.00       0.00       0.00       0.00
     231      2.30      0.00        0.00       0.00       0.00       0.00
     232      2.30      0.00        0.00       0.00       0.00       0.00
     244      4.13      0.00        0.00       0.00       0.00       0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00        3.12       0.00       0.00       0.00
       2      0.00      0.00        6.23       0.00       0.00       0.00
       3      0.00      0.00        6.23       0.00       0.00       0.00
       4      0.00      0.00        3.12       0.00       0.00       0.00
       9      0.00      0.00        6.68       0.00       0.00       0.00
      10      0.00      0.00        6.68       0.00       0.00       0.00
      13      0.00      0.00        2.34       0.00       0.00       0.00
      14      0.00      0.00        2.34       0.00       0.00       0.00
      15      0.00      0.00        1.20       0.00       0.00       0.00
      16      0.00      0.00        1.20       0.00       0.00       0.00
      57      0.00      0.00        2.43       0.00       0.00       0.00
      58      0.00      0.00        2.43       0.00       0.00       0.00
```

```
      65       0.00      0.00         1.23      0.00      0.00      0.00
      66       0.00      0.00         1.23      0.00      0.00      0.00
      67       0.00      0.00         0.63      0.00      0.00      0.00
      68       0.00      0.00         0.63      0.00      0.00      0.00
      73       0.00      0.00         0.65      0.00      0.00      0.00
      75       0.00      0.00         0.63      0.00      0.00      0.00
      76       0.00      0.00         0.63      0.00      0.00      0.00
      81       0.00      0.00         0.65      0.00      0.00      0.00
      89       0.00      0.00         0.60      0.00      0.00      0.00
      90       0.00      0.00         0.60      0.00      0.00      0.00
     125       0.00      0.00         0.65      0.00      0.00      0.00
     127       0.00      0.00         1.22      0.00      0.00      0.00
     128       0.00      0.00         1.22      0.00      0.00      0.00
     129       0.00      0.00         0.62      0.00      0.00      0.00
     130       0.00      0.00         0.62      0.00      0.00      0.00
     131       0.00      0.00         0.65      0.00      0.00      0.00
     133       0.00      0.00         0.61      0.00      0.00      0.00
     135       0.00      0.00         1.25      0.00      0.00      0.00
     136       0.00      0.00         0.63      0.00      0.00      0.00
     207       0.00      0.00         0.63      0.00      0.00      0.00
     208       0.00      0.00         0.63      0.00      0.00      0.00
     229       0.00      0.00         1.38      0.00      0.00      0.00
     230       0.00      0.00         1.38      0.00      0.00      0.00
     231       0.00      0.00         0.69      0.00      0.00      0.00
     232       0.00      0.00         0.69      0.00      0.00      0.00
     235       0.00      0.00         0.69      0.00      0.00      0.00
     236       0.00      0.00         0.69      0.00      0.00      0.00
     237       0.00      0.00         1.17      0.00      0.00      0.00
     238       0.00      0.00         1.17      0.00      0.00      0.00
     189       0.00      0.00         0.61      0.00      0.00      0.00
     192       0.00      0.00         0.61      0.00      0.00      0.00
      STAAD SPACE                                                -- PAGE NO.
31
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      75       0.00      0.00         1.30      0.00      0.00      0.00
      76       0.00      0.00         2.68      0.00      0.00      0.00
      83       0.00      0.00         1.30      0.00      0.00      0.00
      84       0.00      0.00         2.68      0.00      0.00      0.00
     133       0.00      0.00         2.61      0.00      0.00      0.00
     134       0.00      0.00         2.61      0.00      0.00      0.00
     135       0.00      0.00         2.61      0.00      0.00      0.00
     136       0.00      0.00         2.61      0.00      0.00      0.00
     149       0.00      0.00         2.08      0.00      0.00      0.00
     150       0.00      0.00         3.12      0.00      0.00      0.00
     151       0.00      0.00         3.12      0.00      0.00      0.00
     152       0.00      0.00         2.08      0.00      0.00      0.00
     153       0.00      0.00         6.23      0.00      0.00      0.00
     154       0.00      0.00         6.23      0.00      0.00      0.00
     155       0.00      0.00         4.15      0.00      0.00      0.00
     156       0.00      0.00         4.15      0.00      0.00      0.00
     163       0.00      0.00         2.12      0.00      0.00      0.00
     164       0.00      0.00         2.12      0.00      0.00      0.00
     165       0.00      0.00         4.24      0.00      0.00      0.00
     166       0.00      0.00         4.24      0.00      0.00      0.00
     173       0.00      0.00         2.23      0.00      0.00      0.00
     174       0.00      0.00         2.23      0.00      0.00      0.00
     175       0.00      0.00         4.45      0.00      0.00      0.00
```

```
     176      0.00      0.00        4.45      0.00      0.00      0.00
     183      0.00      0.00        2.32      0.00      0.00      0.00
     184      0.00      0.00        2.32      0.00      0.00      0.00
     185      0.00      0.00        4.63      0.00      0.00      0.00
     186      0.00      0.00        4.63      0.00      0.00      0.00
     193      0.00      0.00        2.37      0.00      0.00      0.00
     194      0.00      0.00        2.37      0.00      0.00      0.00
     195      0.00      0.00        4.74      0.00      0.00      0.00
     196      0.00      0.00        4.74      0.00      0.00      0.00
     203      0.00      0.00        2.43      0.00      0.00      0.00
     204      0.00      0.00        2.43      0.00      0.00      0.00
     205      0.00      0.00        4.87      0.00      0.00      0.00
     206      0.00      0.00        4.87      0.00      0.00      0.00
     213      0.00      0.00        2.50      0.00      0.00      0.00
     214      0.00      0.00        2.50      0.00      0.00      0.00
     215      0.00      0.00        5.00      0.00      0.00      0.00
     216      0.00      0.00        5.00      0.00      0.00      0.00
     221      0.00      0.00        1.27      0.00      0.00      0.00
     222      0.00      0.00        1.27      0.00      0.00      0.00
     223      0.00      0.00        2.54      0.00      0.00      0.00
     224      0.00      0.00        2.54      0.00      0.00      0.00
     226      0.00      0.00        1.38      0.00      0.00      0.00
     228      0.00      0.00        1.38      0.00      0.00      0.00
   LOADING     8  LOADTYPE WIND  TITLE WIND (+Z) IS875 RECT CLAD [-CPI]
   -----------
      STAAD SPACE                                                -- PAGE NO.
32
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      0.00      0.00       13.50      0.00      0.00      0.00
       2      0.00      0.00       27.00      0.00      0.00      0.00
       3      0.00      0.00       27.00      0.00      0.00      0.00
       4      0.00      0.00       13.50      0.00      0.00      0.00
       9      0.00      0.00       28.93      0.00      0.00      0.00
      10      0.00      0.00       28.93      0.00      0.00      0.00
      13      0.00      0.00       10.15      0.00      0.00      0.00
      14      0.00      0.00       10.15      0.00      0.00      0.00
      15      0.00      0.00        5.20      0.00      0.00      0.00
      16      0.00      0.00        5.20      0.00      0.00      0.00
      57      0.00      0.00       10.55      0.00      0.00      0.00
      58      0.00      0.00       10.55      0.00      0.00      0.00
      65      0.00      0.00        5.35      0.00      0.00      0.00
      66      0.00      0.00        5.35      0.00      0.00      0.00
      67      0.00      0.00        2.75      0.00      0.00      0.00
      68      0.00      0.00        2.75      0.00      0.00      0.00
      73      0.00      0.00        2.82      0.00      0.00      0.00
      75      0.00      0.00        2.75      0.00      0.00      0.00
      76      0.00      0.00        2.75      0.00      0.00      0.00
      81      0.00      0.00        2.82      0.00      0.00      0.00
      89      0.00      0.00        2.60      0.00      0.00      0.00
      90      0.00      0.00        2.60      0.00      0.00      0.00
     125      0.00      0.00        2.82      0.00      0.00      0.00
     127      0.00      0.00        5.27      0.00      0.00      0.00
     128      0.00      0.00        5.27      0.00      0.00      0.00
     129      0.00      0.00        2.67      0.00      0.00      0.00
     130      0.00      0.00        2.67      0.00      0.00      0.00
     131      0.00      0.00        2.82      0.00      0.00      0.00
     133      0.00      0.00        2.66      0.00      0.00      0.00
```

```
     135       0.00       0.00         5.40       0.00       0.00       0.00
     136       0.00       0.00         2.75       0.00       0.00       0.00
     207       0.00       0.00         2.75       0.00       0.00       0.00
     208       0.00       0.00         2.75       0.00       0.00       0.00
     229       0.00       0.00         5.98       0.00       0.00       0.00
     230       0.00       0.00         5.98       0.00       0.00       0.00
     231       0.00       0.00         2.99       0.00       0.00       0.00
     232       0.00       0.00         2.99       0.00       0.00       0.00
     235       0.00       0.00         2.99       0.00       0.00       0.00
     236       0.00       0.00         2.99       0.00       0.00       0.00
     237       0.00       0.00         5.08       0.00       0.00       0.00
     238       0.00       0.00         5.08       0.00       0.00       0.00
     189       0.00       0.00         2.66       0.00       0.00       0.00
     192       0.00       0.00         2.66       0.00       0.00       0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      75       0.00       0.00        -0.87       0.00       0.00       0.00
      STAAD SPACE                                               -- PAGE NO.
33
      76       0.00       0.00        -1.79       0.00       0.00       0.00
      83       0.00       0.00        -0.87       0.00       0.00       0.00
      84       0.00       0.00        -1.79       0.00       0.00       0.00
     133       0.00       0.00        -1.74       0.00       0.00       0.00
     134       0.00       0.00        -1.74       0.00       0.00       0.00
     135       0.00       0.00        -1.74       0.00       0.00       0.00
     136       0.00       0.00        -1.74       0.00       0.00       0.00
     149       0.00       0.00        -1.38       0.00       0.00       0.00
     150       0.00       0.00        -2.08       0.00       0.00       0.00
     151       0.00       0.00        -2.08       0.00       0.00       0.00
     152       0.00       0.00        -1.38       0.00       0.00       0.00
     153       0.00       0.00        -4.15       0.00       0.00       0.00
     154       0.00       0.00        -4.15       0.00       0.00       0.00
     155       0.00       0.00        -2.77       0.00       0.00       0.00
     156       0.00       0.00        -2.77       0.00       0.00       0.00
     163       0.00       0.00        -1.41       0.00       0.00       0.00
     164       0.00       0.00        -1.41       0.00       0.00       0.00
     165       0.00       0.00        -2.83       0.00       0.00       0.00
     166       0.00       0.00        -2.83       0.00       0.00       0.00
     173       0.00       0.00        -1.48       0.00       0.00       0.00
     174       0.00       0.00        -1.48       0.00       0.00       0.00
     175       0.00       0.00        -2.97       0.00       0.00       0.00
     176       0.00       0.00        -2.97       0.00       0.00       0.00
     183       0.00       0.00        -1.54       0.00       0.00       0.00
     184       0.00       0.00        -1.54       0.00       0.00       0.00
     185       0.00       0.00        -3.09       0.00       0.00       0.00
     186       0.00       0.00        -3.09       0.00       0.00       0.00
     193       0.00       0.00        -1.58       0.00       0.00       0.00
     194       0.00       0.00        -1.58       0.00       0.00       0.00
     195       0.00       0.00        -3.16       0.00       0.00       0.00
     196       0.00       0.00        -3.16       0.00       0.00       0.00
     203       0.00       0.00        -1.62       0.00       0.00       0.00
     204       0.00       0.00        -1.62       0.00       0.00       0.00
     205       0.00       0.00        -3.25       0.00       0.00       0.00
     206       0.00       0.00        -3.25       0.00       0.00       0.00
     213       0.00       0.00        -1.67       0.00       0.00       0.00
     214       0.00       0.00        -1.67       0.00       0.00       0.00
     215       0.00       0.00        -3.34       0.00       0.00       0.00
     216       0.00       0.00        -3.34       0.00       0.00       0.00
```

```
     221      0.00      0.00        -0.85      0.00      0.00      0.00
     222      0.00      0.00        -0.85      0.00      0.00      0.00
     223      0.00      0.00        -1.69      0.00      0.00      0.00
     224      0.00      0.00        -1.69      0.00      0.00      0.00
     226      0.00      0.00        -0.92      0.00      0.00      0.00
     228      0.00      0.00        -0.92      0.00      0.00      0.00
   LOADING     9  LOADTYPE WIND  TITLE WIND (-Z) IS875 RECT CLAD [+CPI]
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1     -7.33      0.00        0.00      0.00      0.00      0.00
      STAAD SPACE                                             -- PAGE NO.
34
       2    -14.66      0.00        0.00      0.00      0.00      0.00
       5    -11.00      0.00        0.00      0.00      0.00      0.00
       6    -18.33      0.00        0.00      0.00      0.00      0.00
       9    -15.71      0.00        0.00      0.00      0.00      0.00
      11    -15.71      0.00        0.00      0.00      0.00      0.00
      13     -6.20      0.00        0.00      0.00      0.00      0.00
      15     -8.32      0.00        0.00      0.00      0.00      0.00
      17    -11.00      0.00        0.00      0.00      0.00      0.00
      18    -16.49      0.00        0.00      0.00      0.00      0.00
      29     -7.48      0.00        0.00      0.00      0.00      0.00
      31    -11.22      0.00        0.00      0.00      0.00      0.00
      37    -11.78      0.00        0.00      0.00      0.00      0.00
      43     -8.18      0.00        0.00      0.00      0.00      0.00
      45    -12.26      0.00        0.00      0.00      0.00      0.00
      51    -12.56      0.00        0.00      0.00      0.00      0.00
      57     -4.30      0.00        0.00      0.00      0.00      0.00
      59    -12.89      0.00        0.00      0.00      0.00      0.00
      65     -4.42      0.00        0.00      0.00      0.00      0.00
      67    -13.25      0.00        0.00      0.00      0.00      0.00
      73     -4.54      0.00        0.00      0.00      0.00      0.00
      75     -9.01      0.00        0.00      0.00      0.00      0.00
      81     -2.30      0.00        0.00      0.00      0.00      0.00
      83     -2.30      0.00        0.00      0.00      0.00      0.00
     141    -14.66      0.00        0.00      0.00      0.00      0.00
     142    -21.99      0.00        0.00      0.00      0.00      0.00
     149     -7.33      0.00        0.00      0.00      0.00      0.00
     150    -11.00      0.00        0.00      0.00      0.00      0.00
     159    -14.96      0.00        0.00      0.00      0.00      0.00
     163     -7.48      0.00        0.00      0.00      0.00      0.00
     169    -15.71      0.00        0.00      0.00      0.00      0.00
     173     -7.85      0.00        0.00      0.00      0.00      0.00
     179    -16.35      0.00        0.00      0.00      0.00      0.00
     183     -8.18      0.00        0.00      0.00      0.00      0.00
     189    -16.74      0.00        0.00      0.00      0.00      0.00
     193     -8.37      0.00        0.00      0.00      0.00      0.00
     199    -17.18      0.00        0.00      0.00      0.00      0.00
     203     -8.59      0.00        0.00      0.00      0.00      0.00
     209    -17.66      0.00        0.00      0.00      0.00      0.00
     213     -8.83      0.00        0.00      0.00      0.00      0.00
     217     -8.95      0.00        0.00      0.00      0.00      0.00
     221     -4.48      0.00        0.00      0.00      0.00      0.00
     243     -4.13      0.00        0.00      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       3     14.66      0.00        0.00      0.00      0.00      0.00
```

```
      4     7.33     0.00       0.00     0.00     0.00     0.00
      7    18.33     0.00       0.00     0.00     0.00     0.00
      8    11.00     0.00       0.00     0.00     0.00     0.00
     10    15.71     0.00       0.00     0.00     0.00     0.00
     12    15.71     0.00       0.00     0.00     0.00     0.00
     14     6.20     0.00       0.00     0.00     0.00     0.00
     16     8.32     0.00       0.00     0.00     0.00     0.00
     STAAD SPACE                                      -- PAGE NO.
35
     19    16.49     0.00       0.00     0.00     0.00     0.00
     20    11.00     0.00       0.00     0.00     0.00     0.00
     30     7.48     0.00       0.00     0.00     0.00     0.00
     32    11.22     0.00       0.00     0.00     0.00     0.00
     38    11.78     0.00       0.00     0.00     0.00     0.00
     44     8.18     0.00       0.00     0.00     0.00     0.00
     46    12.26     0.00       0.00     0.00     0.00     0.00
     52    12.56     0.00       0.00     0.00     0.00     0.00
     58     4.30     0.00       0.00     0.00     0.00     0.00
     60    12.89     0.00       0.00     0.00     0.00     0.00
     66     4.42     0.00       0.00     0.00     0.00     0.00
     68    13.25     0.00       0.00     0.00     0.00     0.00
     74     2.24     0.00       0.00     0.00     0.00     0.00
     76     6.71     0.00       0.00     0.00     0.00     0.00
    143    21.99     0.00       0.00     0.00     0.00     0.00
    144    14.66     0.00       0.00     0.00     0.00     0.00
    151    11.00     0.00       0.00     0.00     0.00     0.00
    152     7.33     0.00       0.00     0.00     0.00     0.00
    160    14.96     0.00       0.00     0.00     0.00     0.00
    164     7.48     0.00       0.00     0.00     0.00     0.00
    170    15.71     0.00       0.00     0.00     0.00     0.00
    174     7.85     0.00       0.00     0.00     0.00     0.00
    180    16.35     0.00       0.00     0.00     0.00     0.00
    184     8.18     0.00       0.00     0.00     0.00     0.00
    190    16.74     0.00       0.00     0.00     0.00     0.00
    194     8.37     0.00       0.00     0.00     0.00     0.00
    200    17.18     0.00       0.00     0.00     0.00     0.00
    204     8.59     0.00       0.00     0.00     0.00     0.00
    210    17.66     0.00       0.00     0.00     0.00     0.00
    214     8.83     0.00       0.00     0.00     0.00     0.00
    218     8.95     0.00       0.00     0.00     0.00     0.00
    222     4.48     0.00       0.00     0.00     0.00     0.00
    225     4.60     0.00       0.00     0.00     0.00     0.00
    226     2.30     0.00       0.00     0.00     0.00     0.00
    227     4.60     0.00       0.00     0.00     0.00     0.00
    228     2.30     0.00       0.00     0.00     0.00     0.00
    231     2.30     0.00       0.00     0.00     0.00     0.00
    232     2.30     0.00       0.00     0.00     0.00     0.00
    244     4.13     0.00       0.00     0.00     0.00     0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y    FORCE-Z    MOM-X    MOM-Y    MOM-Z
      1     0.00     0.00      -6.23     0.00     0.00     0.00
      2     0.00     0.00     -12.46     0.00     0.00     0.00
      3     0.00     0.00     -12.46     0.00     0.00     0.00
      4     0.00     0.00      -6.23     0.00     0.00     0.00
      9     0.00     0.00     -13.35     0.00     0.00     0.00
     10     0.00     0.00     -13.35     0.00     0.00     0.00
     13     0.00     0.00      -4.69     0.00     0.00     0.00
     14     0.00     0.00      -4.69     0.00     0.00     0.00
```

```
      15      0.00      0.00      -2.40      0.00      0.00      0.00
      16      0.00      0.00      -2.40      0.00      0.00      0.00
      57      0.00      0.00      -4.87      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
36
      58      0.00      0.00      -4.87      0.00      0.00      0.00
      65      0.00      0.00      -2.47      0.00      0.00      0.00
      66      0.00      0.00      -2.47      0.00      0.00      0.00
      67      0.00      0.00      -1.27      0.00      0.00      0.00
      68      0.00      0.00      -1.27      0.00      0.00      0.00
      73      0.00      0.00      -1.30      0.00      0.00      0.00
      75      0.00      0.00      -1.27      0.00      0.00      0.00
      76      0.00      0.00      -1.27      0.00      0.00      0.00
      81      0.00      0.00      -1.30      0.00      0.00      0.00
      89      0.00      0.00      -1.20      0.00      0.00      0.00
      90      0.00      0.00      -1.20      0.00      0.00      0.00
     125      0.00      0.00      -1.30      0.00      0.00      0.00
     127      0.00      0.00      -2.43      0.00      0.00      0.00
     128      0.00      0.00      -2.43      0.00      0.00      0.00
     129      0.00      0.00      -1.23      0.00      0.00      0.00
     130      0.00      0.00      -1.23      0.00      0.00      0.00
     131      0.00      0.00      -1.30      0.00      0.00      0.00
     133      0.00      0.00      -1.23      0.00      0.00      0.00
     135      0.00      0.00      -2.49      0.00      0.00      0.00
     136      0.00      0.00      -1.27      0.00      0.00      0.00
     207      0.00      0.00      -1.27      0.00      0.00      0.00
     208      0.00      0.00      -1.27      0.00      0.00      0.00
     229      0.00      0.00      -2.76      0.00      0.00      0.00
     230      0.00      0.00      -2.76      0.00      0.00      0.00
     231      0.00      0.00      -1.38      0.00      0.00      0.00
     232      0.00      0.00      -1.38      0.00      0.00      0.00
     235      0.00      0.00      -1.38      0.00      0.00      0.00
     236      0.00      0.00      -1.38      0.00      0.00      0.00
     237      0.00      0.00      -2.34      0.00      0.00      0.00
     238      0.00      0.00      -2.34      0.00      0.00      0.00
     189      0.00      0.00      -1.23      0.00      0.00      0.00
     192      0.00      0.00      -1.23      0.00      0.00      0.00
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      75      0.00      0.00      -0.65      0.00      0.00      0.00
      76      0.00      0.00      -1.34      0.00      0.00      0.00
      83      0.00      0.00      -0.65      0.00      0.00      0.00
      84      0.00      0.00      -1.34      0.00      0.00      0.00
     133      0.00      0.00      -1.30      0.00      0.00      0.00
     134      0.00      0.00      -1.30      0.00      0.00      0.00
     135      0.00      0.00      -1.30      0.00      0.00      0.00
     136      0.00      0.00      -1.30      0.00      0.00      0.00
     149      0.00      0.00      -1.04      0.00      0.00      0.00
     150      0.00      0.00      -1.56      0.00      0.00      0.00
     151      0.00      0.00      -1.56      0.00      0.00      0.00
     152      0.00      0.00      -1.04      0.00      0.00      0.00
     153      0.00      0.00      -3.12      0.00      0.00      0.00
     154      0.00      0.00      -3.12      0.00      0.00      0.00
     155      0.00      0.00      -2.08      0.00      0.00      0.00
     156      0.00      0.00      -2.08      0.00      0.00      0.00
     163      0.00      0.00      -1.06      0.00      0.00      0.00
     164      0.00      0.00      -1.06      0.00      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
```

```
37
     165     0.00     0.00      -2.12     0.00     0.00     0.00
     166     0.00     0.00      -2.12     0.00     0.00     0.00
     173     0.00     0.00      -1.11     0.00     0.00     0.00
     174     0.00     0.00      -1.11     0.00     0.00     0.00
     175     0.00     0.00      -2.23     0.00     0.00     0.00
     176     0.00     0.00      -2.23     0.00     0.00     0.00
     183     0.00     0.00      -1.16     0.00     0.00     0.00
     184     0.00     0.00      -1.16     0.00     0.00     0.00
     185     0.00     0.00      -2.32     0.00     0.00     0.00
     186     0.00     0.00      -2.32     0.00     0.00     0.00
     193     0.00     0.00      -1.19     0.00     0.00     0.00
     194     0.00     0.00      -1.19     0.00     0.00     0.00
     195     0.00     0.00      -2.37     0.00     0.00     0.00
     196     0.00     0.00      -2.37     0.00     0.00     0.00
     203     0.00     0.00      -1.22     0.00     0.00     0.00
     204     0.00     0.00      -1.22     0.00     0.00     0.00
     205     0.00     0.00      -2.43     0.00     0.00     0.00
     206     0.00     0.00      -2.43     0.00     0.00     0.00
     213     0.00     0.00      -1.25     0.00     0.00     0.00
     214     0.00     0.00      -1.25     0.00     0.00     0.00
     215     0.00     0.00      -2.50     0.00     0.00     0.00
     216     0.00     0.00      -2.50     0.00     0.00     0.00
     221     0.00     0.00      -0.63     0.00     0.00     0.00
     222     0.00     0.00      -0.63     0.00     0.00     0.00
     223     0.00     0.00      -1.27     0.00     0.00     0.00
     224     0.00     0.00      -1.27     0.00     0.00     0.00
     226     0.00     0.00      -0.69     0.00     0.00     0.00
     228     0.00     0.00      -0.69     0.00     0.00     0.00
```

V.Wind On Closed Structure 1

Evaluate the equivalent joint loads for a closed structure subjected to wind loads.

Details

A structure lying in YZ plane is composed of three members and is subjected to wind load from (+X) direction (i.e., out-of-plane). Those three members, along with the ground surface, form a closed panel. The wind load incident on that panel is converted to equivalent joint loads incident on the nodes by the program.

The exposure factor, e, for the structure is 0.85.

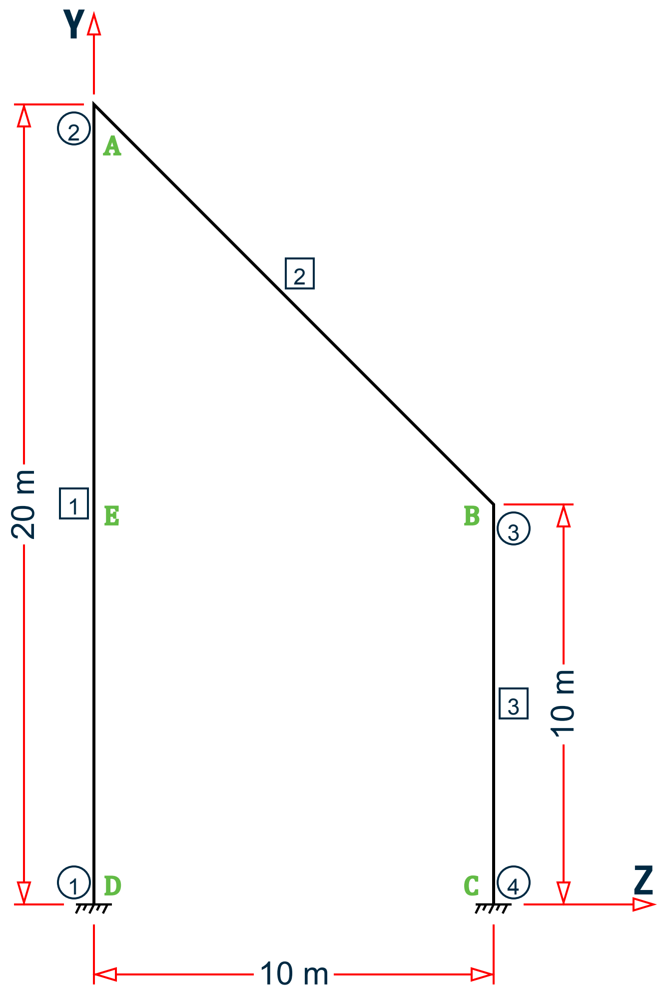


Table 559: Wind intensity variation with height above ground

Height (m)	Intensity of wind load (kg/m^2)
≤ 5m	1.5
5m - 10 m	2
10 m - 20 m	3

Validation

Determine CG of the Wind Area

Naming the nodal points A, B, C, and D, divide the area into two sub-areas: a rectangle and a triangle. The division is taken at the mid-point of member 1, which is at the same Y ordinate as node 3 (point B).

Area of the triangle ABE = $1/2 \times base \times height = 1/2 \times (10m) \times (10m) = 50\ m^2$

CG of triangle ABE: $\overline{X}_{ABE} = \frac{1}{3} 10 = 3.33\ m, \overline{Y}_{ABE} = 10 + \frac{1}{3} 10 = 13.33\ m$

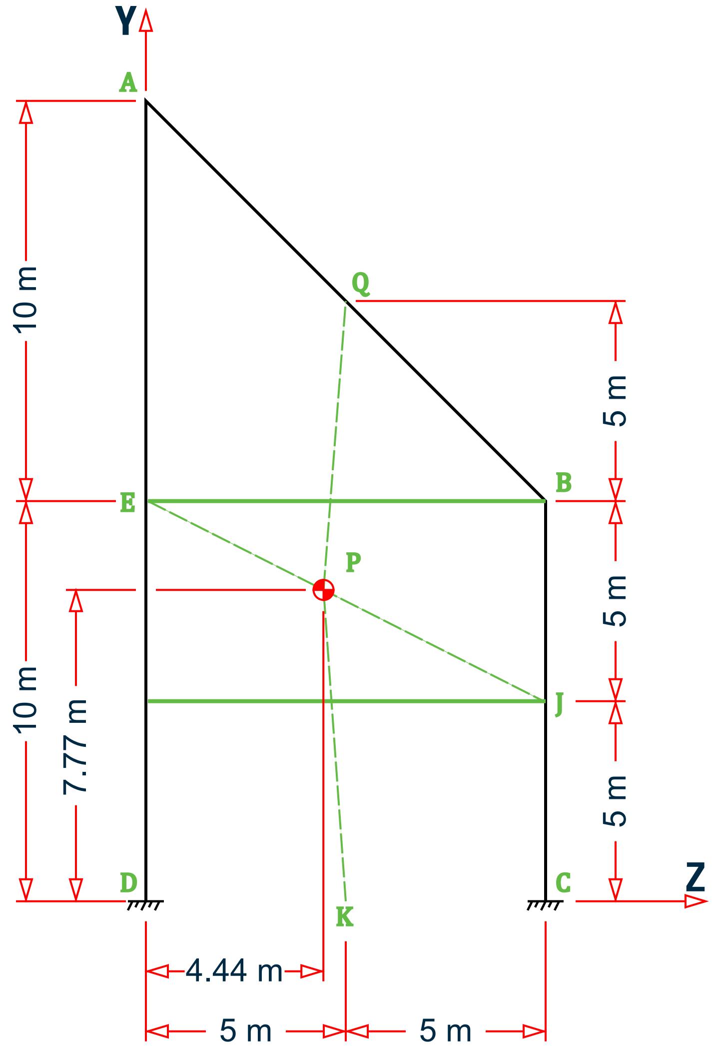
Area of rectangle BCDE = $10\ m \times 10\ m = 100\ m^2$

CG of rectangle BCDE: $\overline{X}_{BCDE} = \frac{1}{2}10 = 5\ m$, $\overline{Y}_{BCDE} = \frac{1}{2}10 = 5\ m$

CG of total area ABCD: $\overline{X} = \frac{50(3.33) + 100(5)}{50 + 100} = 4.44\ m$, $\overline{Y} = \frac{50(13.33) + 100(5)}{50 + 100} = 7.77\ m$

The CG is labeled point P.

The influence area of each joint is then take as the midpoint of the connecting members to this CG. Label the midpoints of each member E, Q, and J. The point at support level between nodes 1 and 2 is labeled K.



<u>Equivalent Joint Load on Node 2 (A)</u>

The influence area for node 2 is the shape formed by points A, Q, P, and E. Notice that this area can be decomposed into a trapezoid and two triangles. Also notice that this area has *two* different wind pressures, above and below y = 10m (which coincides with line EB). Label a point on line AE with the same Y coordinate as Q as N.

Determine the Z coordinate of where line EB intersects line PQ and label this point M.

slope of PQ = (15 - 7.77)/(5 - 4.44) = 13.0

z = 4.44 + (10 - 7.77)/13 = 4.615 m

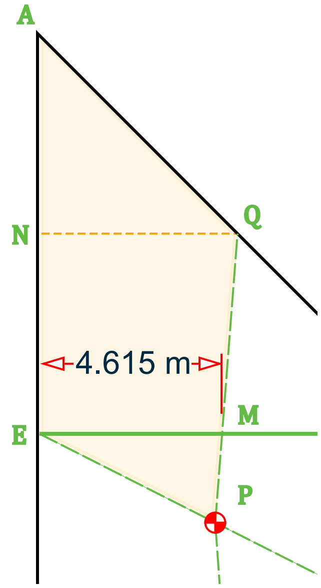


Figure 464: Influence area of point A

The sub-areas of AQPE:

$A_{EMP} = 0.5 \times (10 - 7.77) \times 4.615 = 5.13\ m^2$

$A_{NQME} = 0.5 \times (5+4.615) \times 5 = 24.04\ m^2$

$A_{ANQ} = 0.5 \times 5 \times 5 = 12.5\ m^2$

The equivalent joint load, F, is calculated as $Influence\ area,\ A,$ × $avg.\ wind\ pressure,\ p$ × $exposure\ factor,\ e$

$F = 0.85 \times [2 \times (5.13) + 3 \times (24.04 + 12.5)] = 101.90\ kN$

Equivalent Joint Load on Node 3 (B)

The influence area for node 3 is the shape formed by points B, J, P and Q. Label a point on line BJ with the same Y coordinate as P as point V.

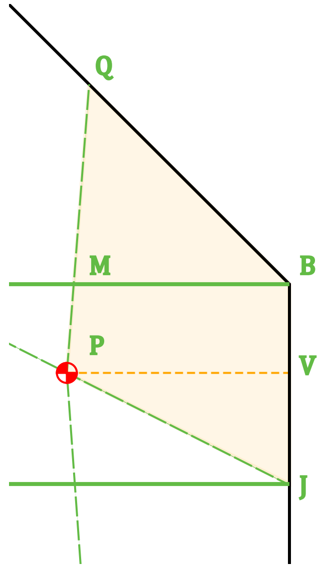


Figure 465: Influence area of point B

The sub-areas of BJPQ:

$$A_{BMQ} = 0.5\times(10 - 4.615)\times5 = 13.46 \text{ m}^2$$

$$A_{BVPM} = 0.5\times[(10 - 4.615) + (10 - 4.44)]\times(10 - 7.77) = 12.2 \text{ m}^2$$

$$A_{VJP} = 0.5\times(10 - 4.44)\times(7.77 - 5) = 7.72 \text{ m}^2$$

The equivalent joint load,

$$F = 0.85\times[2\times(12.2 + 7.72) + 3\times(13.46)] = 67.85 \text{ kN}$$

<u>Equivalent Joint Load on Node 4 (C)</u>

The influence area for node 4 is the shape formed by points C, K, P, and J. The change in pressure magnitude is at the same Y coordinate as point J. Label the point on line ED with this same Y coordinate as point R. Determine the Z coordinate of where line RJ intersects line PK and label this point W.

$$\text{slope of PK} = (7.77)/(5 - 4.44) = 14.0$$

$$z = 4.44 + (7.77 - 5)/14 = 4.643 \text{ m}$$

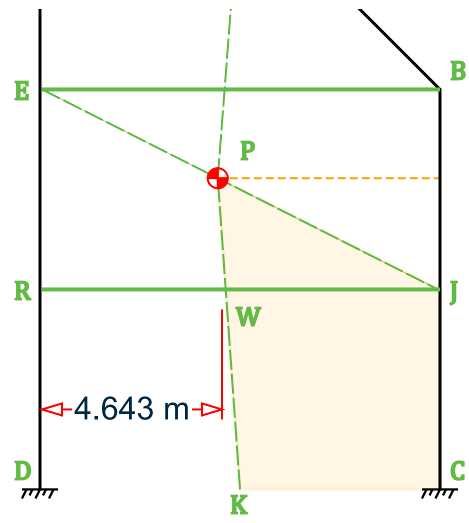


Figure 466: Influence area of point C

The sub-areas of CKPJ:

$$A_{JWP} = 0.5\times(7.77 - 5)\times(10 - 4.643) = 7.44 \text{ m}^2$$

$$A_{BVPM} = 0.5\times[(10 - 4.643) + 5]\times 5 = 25.89 \text{ m}^2$$

The equivalent joint load,

$$F = 0.85\times[1.5\times(25.89) + 2\times(7.44)] = 45.66 \text{ kN}$$

<u>Equivalent Joint Load on Node 1 (D)</u>

The influence area for node 1 is the shape formed by points D, E, P, and K. Label a point on line DE with the same Y coordinate as P as point L.

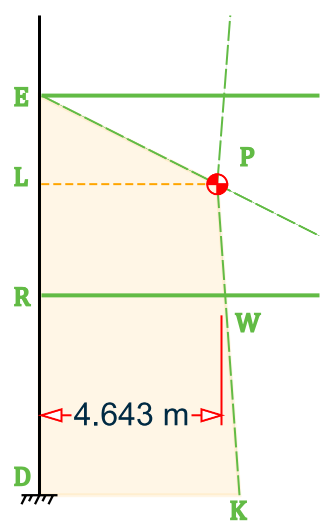


Figure 467: Influence area of point D

The sub-areas of DEPK:

$$A_{LEP} = 0.5\times(10 - 7.77)\times4.44 = 4.94 \text{ m}^2$$

$$A_{RLPW} = 0.5\times[4.44 + 4.643]\times(7.77 - 5) = 12.62 \text{ m}^2$$

$$A_{DRWK} = 0.5\times[4.643 + 5]\times5 = 24.11 \text{ m}^2$$

The equivalent joint load,

$$F = 0.85\times[1.5\times(24.11) + 2\times(4.94 + 12.62)] = 60.59 \text{ kN}$$

Results

Table 560: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Joint load at node 1 (kN)	60.59	60.59	none	
Joint load at node 2 (kN)	101.90	101.89	negligible	
Joint load at node 3 (kN)	67.85	68.11	negligible	
Joint load at node 4 (kN)	45.66	45.66	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\Wind On Closed Structure 1.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Aug-18
END JOB INFORMATION
******************************************************************************
*
* This problem is created to verify the calculations of equivalent -
* joint load on the joints of a closed structure subjected to wind load
******************************************************************************
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 0 10 10; 4 0 0 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
```

```
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
INT 1.5 2 3 HEIG  5  10  20
EXP 0.85 JOINT 1 TO 4
LOAD 1 LOADTYPE Wind  TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 YR 0 20
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE LOAD CASE 1
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1     60.59      0.00        0.00      0.00      0.00      0.00
       2    101.89      0.00        0.00      0.00      0.00      0.00
       3     68.11      0.00        0.00      0.00      0.00      0.00
       4     45.66      0.00        0.00      0.00      0.00      0.00
   ************ END OF DATA FROM INTERNAL STORAGE ************
```

Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)

V.Wind On Closed Structure 2

Evaluate the equivalent joint loads for a closed structure subjected to wind loads.

Details

A structure lying in YZ plane is composed of three members and is subjected to wind load from (+X) direction (i.e., out-of-plane). Those three members, along with the ground surface, form a closed panel. The wind load incident on that panel is converted to equivalent joint loads incident on the nodes by the program.

The exposure factor, e, for the structure is 0.85.

Note: This structure is similar to that in V.Wind On Closed Structure 1 (on page 3711) with an added node at the mid-point of the left column (Point E).

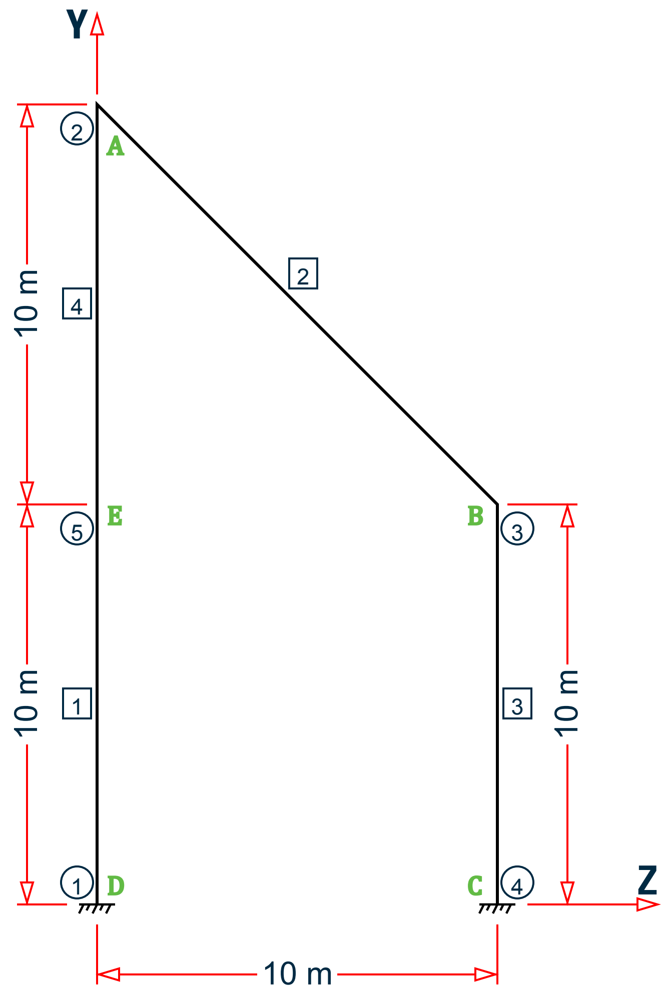


Table 561: Wind intensity variation with height above ground

Height (m)	Intensity of wind load (kg/m^2)
≤ 5m	1.5
5m - 10 m	2
10 m - 20 m	3

Validation

Determine CG of the Wind Area

Naming the nodal points A, B, C, D, and E, divide the area into two sub-areas: a rectangle and a triangle. The division is taken at nodes 5 and 3 (points E and B).

Area of the triangle ABE = $1/2 \times base \times height = 1/2 \times (10m) \times (10m) = 50\ m^2$

CG of triangle ABE: $\overline{X}_{ABE} = \frac{1}{3}10 = 3.33\ m, \overline{Y}_{ABE} = 10 + \frac{1}{3}10 = 13.33\ m$

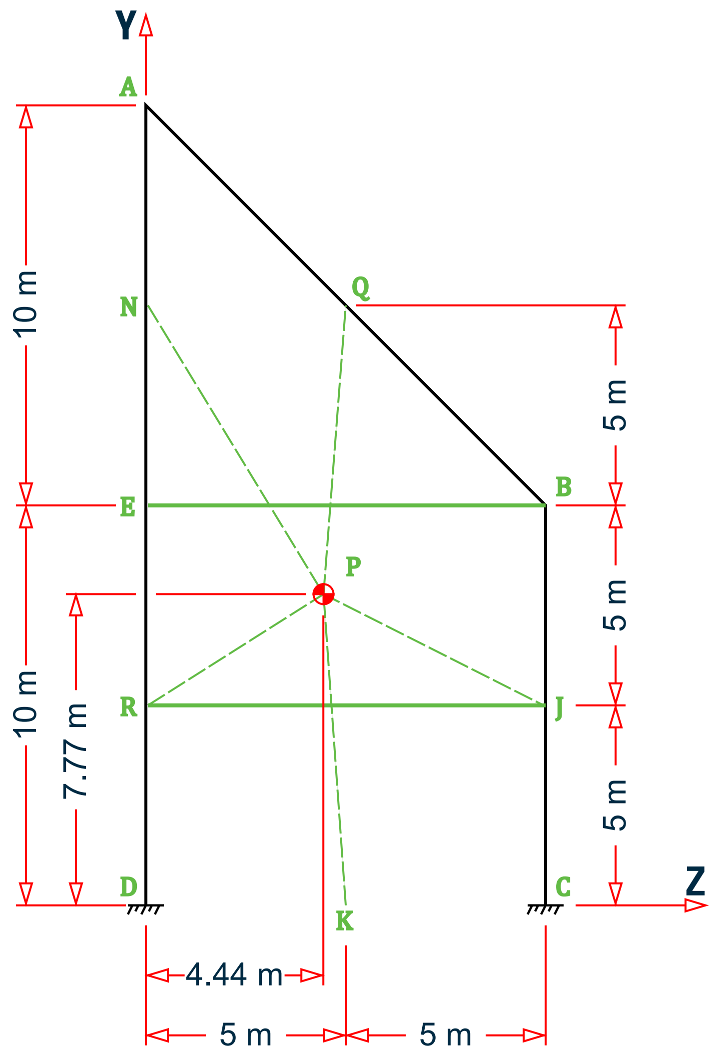
Area of rectangle BCDE = $10\ m \times 10\ m = 100\ m^2$

CG of rectangle BCDE: $\overline{X}_{BCDE} = \frac{1}{2} 10 = 5\ m$, $\overline{Y}_{BCDE} = \frac{1}{2} 10 = 5\ m$

CG of total area ABCD: $\overline{X} = \frac{50(3.33) + 100(5)}{50 + 100} = 4.44\ m$, $\overline{Y} = \frac{50(13.33) + 100(5)}{50 + 100} = 7.77\ m$

The CG is labeled point P.

The influence area of each joint is then take as the midpoint of the connecting members to this CG. Label the midpoints of each member R, N, Q, and J. The point at support level between nodes 1 and 2 is labeled K.



Equivalent Joint Load on Node 2 (A)

The influence area for node 2 is the shape formed by points A, Q, P, and N. Notice that this area can be decomposed into a trapezoid and two triangles. Also notice that this area has *two* different wind pressures, above and below y = 10m (which coincides with line EB).

Determine the Z coordinate of where line EB intersects line NP and label this point S.

slope of NP = (15 - 7.77)/(4.44) = 1.625

$z = 4.44 - (10 - 7.77)/1.625 = 4.615$ m

Determine the Z coordinate of where line EB intersects line PQ and label this point M.

slope of PQ = $(15 - 7.77)/(5 - 4.44) = 13.0$

$z = 4.44 + (10 - 7.77)/13 = 3.077$ m

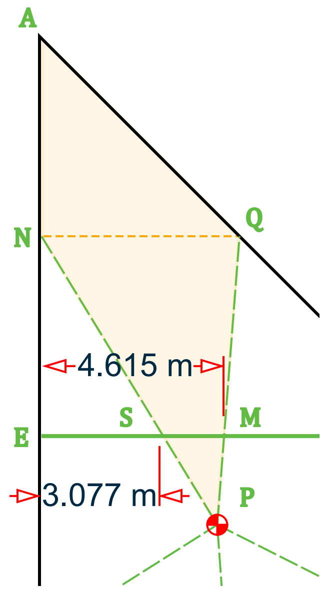


Figure 468: Influence area of point A

The sub-areas of AQPN:

$A_{SMP} = 0.5\times(10 - 7.77)\times(4.615 - 3.077) = 1.71\ m^2$

$A_{NQMS} = 0.5\times(5 + 4.615 - 3.077)\times5 = 16.35\ m^2$

$A_{ANQ} = 0.5\times5\times5 = 12.5\ m^2$

The equivalent joint load, F, is calculated as $Influence\ area,\ A,\ \times avg.\ wind\ pressure,\ p \times exposure\ factor,\ e$

$F = 0.85\times[2\times(1.71) + 3\times(16.35 + 12.5)] = 76.47$ kN

Equivalent Joint Load on Node 3 (B)

The influence area for node 3 is the shape formed by points B, J, P and Q. Label a point on line BJ with the same Y coordinate as P as point V.

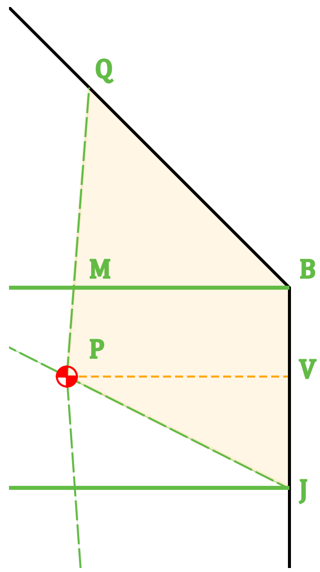


Figure 469: Influence area of point B

The sub-areas of BJPQ:

$$A_{BMQ} = 0.5\times(10 - 4.615)\times5 = 13.46\ m^2$$

$$A_{BVPM} = 0.5\times[(10 - 4.615) + (10 - 4.44)]\times(10 - 7.77) = 12.2\ m^2$$

$$A_{VJP} = 0.5\times(10 - 4.44)\times(7.77 - 5) = 7.72\ m^2$$

The equivalent joint load,

$$F = 0.85\times[2\times(12.2 + 7.72) + 3\times(13.46)] = 67.85\ kN$$

Equivalent Joint Load on Node 4 (C)

The influence area for node 4 is the shape formed by points C, K, P, and J. The change in pressure magnitude is at the same Y coordinate as point J. Label the point on line ED with this same Y coordinate as point R. Determine the Z coordinate of where line RJ intersects line PK and label this point W.

$$\text{slope of PK} = (7.77)/(5 - 4.44) = 14.0$$

$$z = 4.44 + (7.77 - 5)/14 = 4.643\ m$$

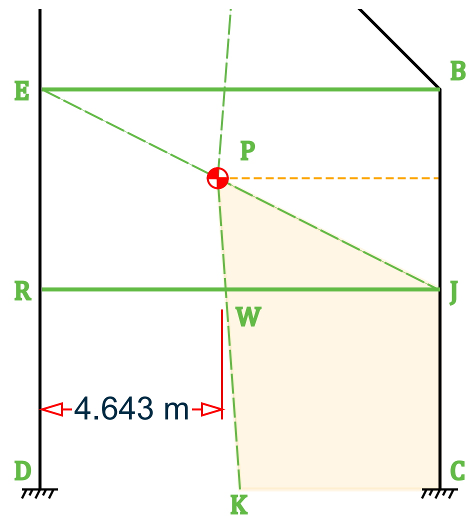


Figure 470: Influence area of point C

The sub-areas of CKPJ:

$$A_{JWP} = 0.5\times(7.77 - 5)\times(10 - 4.643) = 7.44 \text{ m}^2$$

$$A_{BVPM} = 0.5\times[(10 - 4.643) + 5]\times5 = 25.89 \text{ m}^2$$

The equivalent joint load,

$$F = 0.85\times[1.5\times(25.89) + 2\times(7.44)] = 45.66 \text{ kN}$$

<u>Equivalent Joint Load on Node 1 (D)</u>

The influence area for node 1 is the shape formed by points D, R, P, and K.

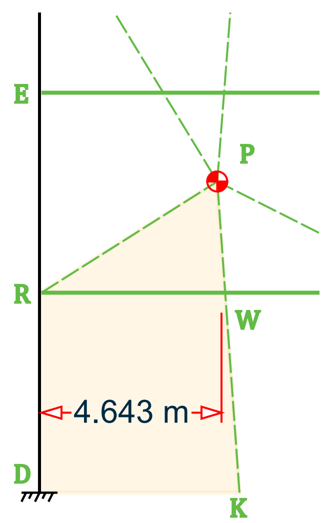


Figure 471: Influence area of point D

The sub-areas of DRPK:

$$A_{RPW} = 0.5\times(7.77 - 5)\times4.643 = 6.45 \text{ m}^2$$

$$A_{DRWK} = 0.5\times[4.643 + 5]\times5 = 24.11\ m^2$$

The equivalent joint load,

$$F = 0.85\times[1.5\times(24.11) + 2\times(6.17)] = 41.71\ kN$$

<u>Equivalent Joint Load on Node 5 (E)</u>

The influence area for node 5 is the shape formed by points N, P, and R. Label a point on line RE with the same Y coordinate as P as point L.

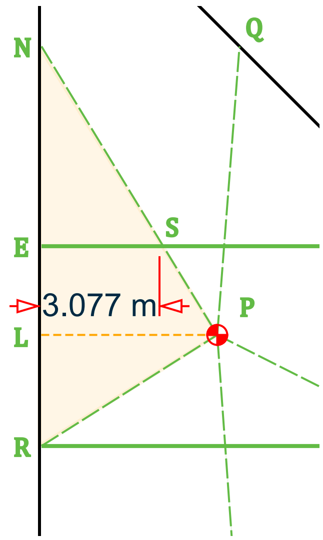


Figure 472: Influence area of point E

The sub-areas of NPR:

$$A_{NSE} = 0.5\times(5)\times3.077 = 7.69\ m^2$$

$$A_{ESPL} = 0.5\times[3.077 + 4.44]\times(10 - 7.77) = 8.36\ m^2$$

$$A_{LPR} = 0.5\times(4.44)\times(7.77 - 5) = 6.17\ m^2$$

The equivalent joint load,

$$F = 0.85\times[2\times(8.36 + 6.17) + 3\times(7.69)] = 44.31\ kN$$

Results

Table 562: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Joint load at node 1 (kN)	41.71	41.70	negligible	
Joint load at node 2 (kN)	76.47	76.46	negligible	
Joint load at node 3 (kN)	67.85	68.11	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Joint load at node 4 (kN)	45.66	45.66	none	
Joint load at node 5 (kN)	44.31	44.32	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\Wind On Closed Structure 2.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Aug-18
END JOB INFORMATION
******************************************************************************
*
* This problem is created to verify the calculations of equivalent -
* joint load on the joints of a closed structure subjected to wind load
******************************************************************************
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 20 0; 3 0 10 10; 4 0 0 10; 5 0 10 0;
MEMBER INCIDENCES
1 1 5; 2 2 3; 3 3 4; 4 5 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
INT 1.5 2 3 HEIG  5  10  20
EXP 0.85 JOINT 1 TO 5
LOAD 1 LOADTYPE Wind  TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 YR 0 20
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE LOAD CASE 1
   -----------
   JOINT LOAD - UNIT KN   METE
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1     41.70      0.00        0.00      0.00      0.00      0.00
       2     76.46      0.00        0.00      0.00      0.00      0.00
       3     68.11      0.00        0.00      0.00      0.00      0.00
       4     45.66      0.00        0.00      0.00      0.00      0.00
       5     44.32      0.00        0.00      0.00      0.00      0.00
```

Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)

V. Wind On Open Structure

Validate the calculated member loads for an open structure subject to wind loading.

Details

A structure is subjected to wind loading from +X direction. The members are rectangular members with 500 mm × 500 mm in dimension. The diagonal member is also of same dimension but rotated at a beta angle value of 45°. Thus, the wind facing area of that member will be different than that of the other members.

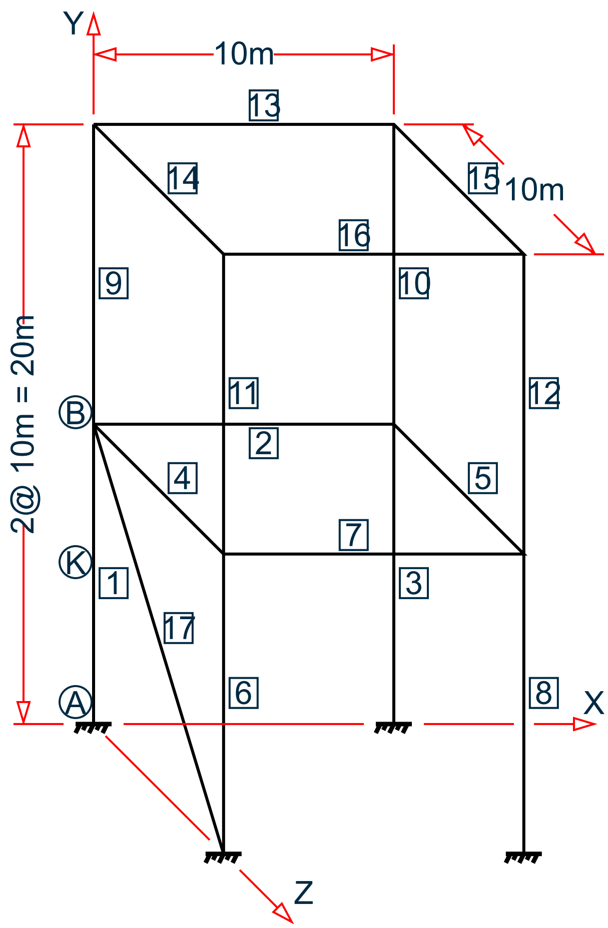


Table 563: Variation of wind intensity over structure height

Height (m)	Intensity of wind load (kN/m^2)
≤5	1.5
5 - 10	2
10 - 20	3

Validation

When open structures are subjected to wind load, wind load incident on the structure is applied as uniformly distributed loads on the members along the projected axis (i.e., in PX/PY/PZ direction). Wind intensity multiplied by the effective area of the member facing the wind load gives the uniformly distributed member load on the member.

Wind Load on Members 1,3, 6 & 8

Wind intensity varies between the length of the member, so different parts of the member will be subjected to uniformly distributed loads of different magnitudes.

Let the midpoint of member 1 be K. So, $AK = CK = 5\ m$

Up to Height 5 m (part AK)

Intensity of wind load = $1.5\ kN/m^2$
Area of the member facing the wind = $0.5\ m$
$w_{AK} = 1.5 \times 0.5 = 0.75\ kN/m$

Up to Height 10 m (part KB)

Intensity of Wind load = $2\ kN/m^2$
Area of the member facing the wind = $0.5\ m$
$w_{KB} = 2 \times 0.5 = 1\ kN/m$

Due to same orientation, member load on member 3, 6, and 8 will also be same as that of member 1.

<u>Wind Load on Member 9, 10, 11 & 12</u>

Intensity of wind load = $3\ kN/m^2$

Area of the member facing the wind = $0.5\ m$

$w = 3 \times 0.5 = 1.5\ kN/m$

Due to same orientation, member load on member 10, 11, and 12 will also be same as that of member 9.

<u>Wind Load on Member 4 & 5</u>

Intensity of wind load = $2\ kN/m^2$

Area of the member facing the wind = $0.5\ m$

$w = 2 \times 0.5 = 1.0\ kN/m$

Due to same orientation, member load on member 5 will also be same as that of member 4.

<u>Wind Load on Member 14 & 15</u>

Intensity of wind load = $3\ kN/m^2$

Area of the member facing the wind = $0.5\ m$

$w = 3 \times 0.5 = 1.5\ kN/m$

Due to same orientation, member load on member 15 will also be same as that of member 14.

<u>Wind Load on Member 2, 7, 13, & 16</u>

These members are aligned along the +X axis. Since the structure is subjected to wind load from +X direction only, the area of those members facing the wind is zero. So, there will be no load due to wind on these members.

<u>Wind Load on Member 17</u>

Since the member is rotated at a beta angle of 45°, the area of the member faceing the wind = $0.5\ m/cos45° = 0.707\ m$

Wind intensity varies between the length of the member, so different parts of the member will be subjected to uniformly distributed loads of different magnitudes.

Up to heigh 5m

Wind intensity = $1.5\ kN/m^2$
$w = 1.5 \times 0.707 = 1.061\ kN/m$

Up to height 10m

Wind intensity = $2\ kN/m^2$

$w = 2 \times 0.707 = 1.414\ kN/m$

Results

Table 564: Comparison of uniformly distributed load due to wind from +X on each member (kN/m)

Member No	Hand Calculation	STAAD.Pro	Difference	Comments
1, 3, 6 & 8	Up to 5m: 0.75 5m - 10m: 1	Up to 5m: 0.75 5m - 10m: 1	none	
4 & 5	1	1	none	
9, 10, 11, 12, 14, & 15	1.5	1.5	none	
17	Up to 5m: 1.061 5m - 10m: 1.414	Up to 5m: 1.0607 5m - 10m: 1.4142	negligible	

Note: Members with no loading under this load condition are not displayed.

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\06 Loading\Wind Load\Wind On Open Structure.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Aug-18
END JOB INFORMATION
*****************************************************************************
*
* This problem is created to determine the uniformly distributed load
* on the members when the structure is subjected to wind load
*****************************************************************************
*
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10; 9 0 20 0; 10 10 20 0; 11 0 20 10; 12 10 20 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8; 9 2 9; 10 3 10;
11 6 11; 12 7 12; 13 9 10; 14 9 11; 15 10 12; 16 11 12; 17 2 5;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
```

```
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 17 PRIS YD 0.5 ZD 0.5
CONSTANTS
BETA 45 MEMB 17
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 FIXED
DEFINE WIND LOAD
TYPE 1 WIND 1
INT 1.5 2 3 HEIG  5  10  20
EXP 1 JOINT 1 TO 12
LOAD 1 LOADTYPE Wind  TITLE LOAD CASE 1
WIND LOAD X 1 TYPE 1 OPEN
PERFORM ANALYSIS PRINT LOAD DATA
FINISH
```

STAAD.Pro Output

```
   LOADING     1  LOADTYPE WIND  TITLE LOAD CASE 1
   -----------
   MEMBER LOAD - UNIT KN   METE
   MEMBER     UDL       L1      L2       CON        L      LIN1      LIN2
      1     0.7500 PX   0.00    5.00
      1     1.0000 PX   5.00   10.00
      3     0.7500 PX   5.00   10.00
      3     1.0000 PX   0.00    5.00
      4     1.0000 PX   0.00   10.00
      5     1.0000 PX   0.00   10.00
      6     0.7500 PX   0.00    5.00
      6     1.0000 PX   5.00   10.00
      8     0.7500 PX   5.00   10.00
      8     1.0000 PX   0.00    5.00
      9     1.5000 PX   0.00   10.00
     10     1.5000 PX   0.00   10.00
     11     1.5000 PX   0.00   10.00
     12     1.5000 PX   0.00   10.00
     14     1.5000 PX   0.00   10.00
     15     1.5000 PX   0.00   10.00
     17     1.0607 PX   7.07   14.14
     17     1.4142 PX   0.00    7.07
```

Related Links

- *G.16.3 Wind Load Generator* (on page 2136)
- *TR.32.12.3 Generation of Wind Loads* (on page 2603)

V.07 Nonlinear Analysis

V. 2D Frame 2 Step P-Delta Displacement

To verify P-Delta analysis for a ten story plane frame.

Reference

Naeim, F., *The Seismic Design Handbook*, Van Nostrand Reinhold, 1989.

Related Links

- *G.17.2.1 P-Delta Analysis* (on page 2141)

Problem

Find the lateral displacement of the 10th story after two iterations of P-Delta analysis.

E = 29,000 ksi

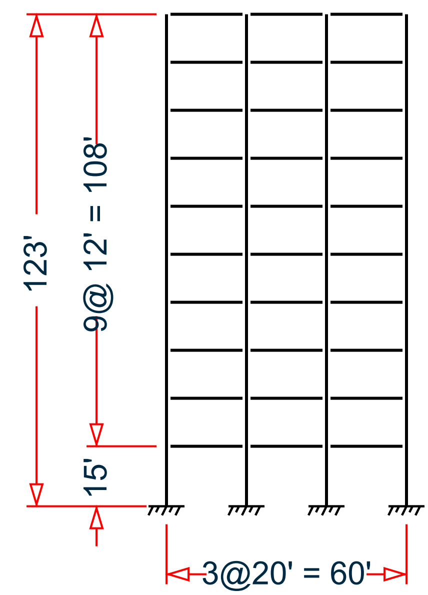


Figure 473: Ten story, plane frame

Comparison

Table 565: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Displacement in X dir. at top (Node 41), δ (in)	8.508	8.65347	1.7%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\07 Nonlinear Analysis\2D Frame 2 Step P-Delta Displacement.STD` is typically installed with the program.

```
STAAD PLANE : 2-ITERATION PDELTA ANALYSIS OF A 10 STOREY PLANE FRAME
START JOB INFORMATION
ENGINEER DATE 17-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0; 3 40 0 0; 4 60 0 0; 5 0 15 0; 6 20 15 0; 7 40 15 0;
8 60 15 0; 9 0 27 0; 10 20 27 0; 11 40 27 0; 12 60 27 0; 13 0 39 0;
14 20 39 0; 15 40 39 0; 16 60 39 0; 17 0 51 0; 18 20 51 0; 19 40 51 0;
20 60 51 0; 21 0 63 0; 22 20 63 0; 23 40 63 0; 24 60 63 0; 25 0 75 0;
26 20 75 0; 27 40 75 0; 28 60 75 0; 29 0 87 0; 30 20 87 0; 31 40 87 0;
32 60 87 0; 33 0 99 0; 34 20 99 0; 35 40 99 0; 36 60 99 0; 37 0 111 0;
38 20 111 0; 39 40 111 0; 40 60 111 0; 41 0 123 0; 42 20 123 0;
43 40 123 0; 44 60 123 0;
MEMBER INCIDENCES
1 1 5; 2 2 6; 3 3 7; 4 4 8; 5 5 6; 6 6 7; 7 7 8; 8 5 9; 9 6 10; 10 7 11;
11 8 12; 12 9 10; 13 10 11; 14 11 12; 15 9 13; 16 10 14; 17 11 15;
18 12 16; 19 13 14; 20 14 15; 21 15 16; 22 13 17; 23 14 18; 24 15 19;
25 16 20; 26 17 18; 27 18 19; 28 19 20; 29 17 21; 30 18 22; 31 19 23;
32 20 24; 33 21 22; 34 22 23; 35 23 24; 36 21 25; 37 22 26; 38 23 27;
39 24 28; 40 25 26; 41 26 27; 42 27 28; 43 25 29; 44 26 30; 45 27 31;
46 28 32; 47 29 30; 48 30 31; 49 31 32; 50 29 33; 51 30 34; 52 31 35;
53 32 36; 54 33 34; 55 34 35; 56 35 36; 57 33 37; 58 34 38; 59 35 39;
60 36 40; 61 37 38; 62 38 39; 63 39 40; 64 37 41; 65 38 42; 66 39 43;
67 40 44; 68 41 42; 69 42 43; 70 43 44;
MEMBER PROPERTY AMERICAN
61 TO 63 68 TO 70 TABLE ST W16X40
47 TO 49 54 TO 56 TABLE ST W21X44
33 TO 35 40 TO 42 TABLE ST W21X50
5 TO 7 12 TO 14 19 TO 21 26 TO 28 TABLE ST W24X55
1 4 8 11 TABLE ST W14X109
15 18 22 25 TABLE ST W14X82
29 32 36 39 TABLE ST W14X68
43 46 50 53 TABLE ST W14X53
57 60 64 67 TABLE ST W14X43
2 3 9 10 TABLE ST W14X159
16 17 23 24 TABLE ST W14X109
30 31 37 38 TABLE ST W14X90
44 45 51 52 TABLE ST W14X68
58 59 65 66 TABLE ST W14X48
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
UNIT INCHES KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 TO 4 FIXED
UNIT FEET KIP
LOAD 1 LATERAL LOADS
SELFWEIGHT Y -1
JOINT LOAD
5 FX 2.97
9 FX 5.34
13 FX 7.71
17 FX 10.08
21 FX 12.45
25 FX 14.83
29 FX 17.2
33 FX 19.57
37 FX 21.94
41 FX 30.22
MEMBER LOAD
68 TO 70 UNI GY -3
5 TO 7 12 TO 14 19 TO 21 26 TO 28 33 TO 35 40 TO 42 47 TO 49 -
54 TO 56 61 TO 63 UNI GY -3.6
PDELTA 2 ANALYSIS SMALLDELTA
PRINT SUPPORT REACTION
PRINT JOINT DISPLACEMENTS LIST 5 9 13 17 21 25 29 33 37 41
FINISH
```

STAAD Output

```
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1    -24.00    167.66      0.00      0.00      0.00    266.10
      2    1    -44.43    728.95      0.00      0.00      0.00    450.70
      3    1    -44.18    735.97      0.00      0.00      0.00    450.22
      4    1    -29.70    562.02      0.00      0.00      0.00    299.68
   ************** END OF LATEST ANALYSIS RESULT **************
    77. PRINT JOINT DISPLACEMENTS LIST 5 9 13 17 21 25 29 33 37 41
  JOINT    DISPLACE LIST     5
      : 2-ITERATION PDELTA ANALYSIS OF A 10 STOREY PLANE FRAME -- PAGE NO.
5
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      5    1    0.82891  -0.03236   0.00000   0.00000   0.00000  -0.00489
      9    1    1.67612  -0.05711   0.00000   0.00000   0.00000  -0.00505
     13    1    2.62313  -0.08857   0.00000   0.00000   0.00000  -0.00526
     17    1    3.53893  -0.11821   0.00000   0.00000   0.00000  -0.00526
     21    1    4.53306  -0.15114   0.00000   0.00000   0.00000  -0.00599
     25    1    5.50759  -0.18037   0.00000   0.00000   0.00000  -0.00583
     29    1    6.48197  -0.21244   0.00000   0.00000   0.00000  -0.00576
     33    1    7.32004  -0.23784   0.00000   0.00000   0.00000  -0.00526
```

```
    37    1    8.09379  -0.25973   0.00000   0.00000   0.00000  -0.00516
    41    1    8.65347  -0.26998   0.00000   0.00000   0.00000  -0.00483
```

V. Column Buckling Factor

Determine the buckling factor for a pinned-pinned column subject to an axial load.

Details

A 30 ft tall column with pinned ends is subject to a 160 kip axial load. The section is a W10x45 with a modulus of elasticity of 29,730 ksi.

Validation

Critical elastic buckling capacity:

$$P_{cr} = \pi^2 EI/\ell^2$$

$$P_{cr} = \pi^2(29{,}730)(248)/(30\times12)^2 = 561.5 \text{ k}$$

Buckling factor: $P_{cr}/P = 561.5\ /\ 160 = 3.509$

Results

Table 566:

Parameter	Reference	STAAD.Pro	Difference	Comments
Buckling factor	3.509	3.50976562	negligible	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples \Verification Models\07 Nonlinear Analysis\Column Buckling Factor.std` is typically installed with the program.

```
STAAD PLANE : BUCKLING FACTOR FOR PINNED-PINNED COLUMN WITH AN AXIAL LOAD
SET SHEAR
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0 5 0 30 0;
MEMBER INCIDENCES
1 1 2 4
UNIT KIP INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29730
POISSON 0.3
```

```
DENSITY 0.286E-3
ALPHA 6e-006
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 4 TABLE ST W10X45
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
5 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
5 FY -160
PERFORM BUCKLING ANALYSIS MAXSTEP 20
FINISH
```

STAAD Output

```
    BUCKLING ITERATIONS FOR CASE NO.     1
 ITERATION    BUCKLING FACTOR      UPPER BOUND        LOWER BOUND
     0         1.00000000E+00     0.00000000E+00     0.00000000E+00
     1         2.00000000E+00     1.01000000E+10     1.00000000E+00
     2         4.00000000E+00     1.01000000E+10     2.00000000E+00
     3         3.00000000E+00     4.00000000E+00     2.00000000E+00
     4         3.50000000E+00     4.00000000E+00     3.00000000E+00
     5         3.75000000E+00     4.00000000E+00     3.50000000E+00
     6         3.62500000E+00     3.75000000E+00     3.50000000E+00
     7         3.56250000E+00     3.62500000E+00     3.50000000E+00
     8         3.53125000E+00     3.56250000E+00     3.50000000E+00
     9         3.51562500E+00     3.53125000E+00     3.50000000E+00
    10         3.50781250E+00     3.51562500E+00     3.50000000E+00
    11         3.51171875E+00     3.51562500E+00     3.50781250E+00
    12         3.50976562E+00     3.51171875E+00     3.50781250E+00
    13         3.50976562E+00     3.51171875E+00     3.50976562E+00
   CONVERGED
```

Related Links

- *G.17.2.2 Buckling Analysis* (on page 2145)

V. Column Pushover Displacement

Compare theoretical solution to a pushover analysis to the STAAD.Pro solution.

Reference

Hand calculation.

Related Links

- *G.17.4 Pushover Analysis* (on page 2168)

Problem

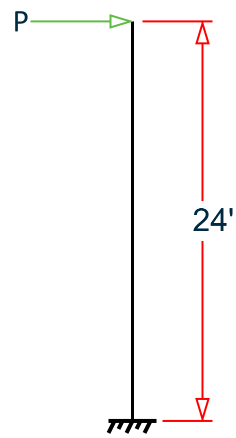


Figure 474: Cantilever member model

A transverse load P is applied to a cantilever member and increased until the member fails.

Member length = 24 inch

Section = Wide flange W16X77

Material = Steel

Expected yield strength = 36 ksi

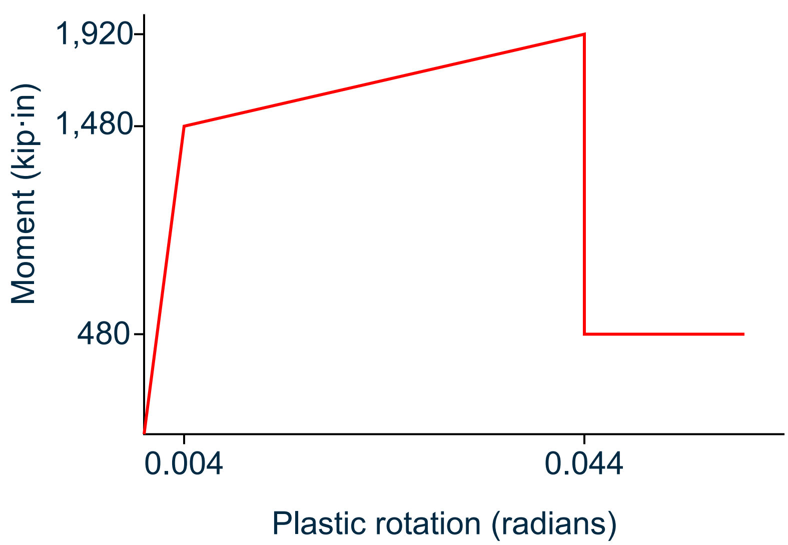


Figure 475: Plastic Rotation (in radians) vs. Moment (in in·kips) plot for moment hinge

Results from STAAD.Pro

Note: Pushover analysis requires the STAAD.Pro Advanced Analysis license.

The analysis results are saved at an interval of 0.1 inch deflection of the cantilever tip. The following results are displayed in the Postprocessing workflow by selecting the **Layouts** > **Pushover-Graphs** tool in the **Dynamics** group on the **Results** ribbon tab.

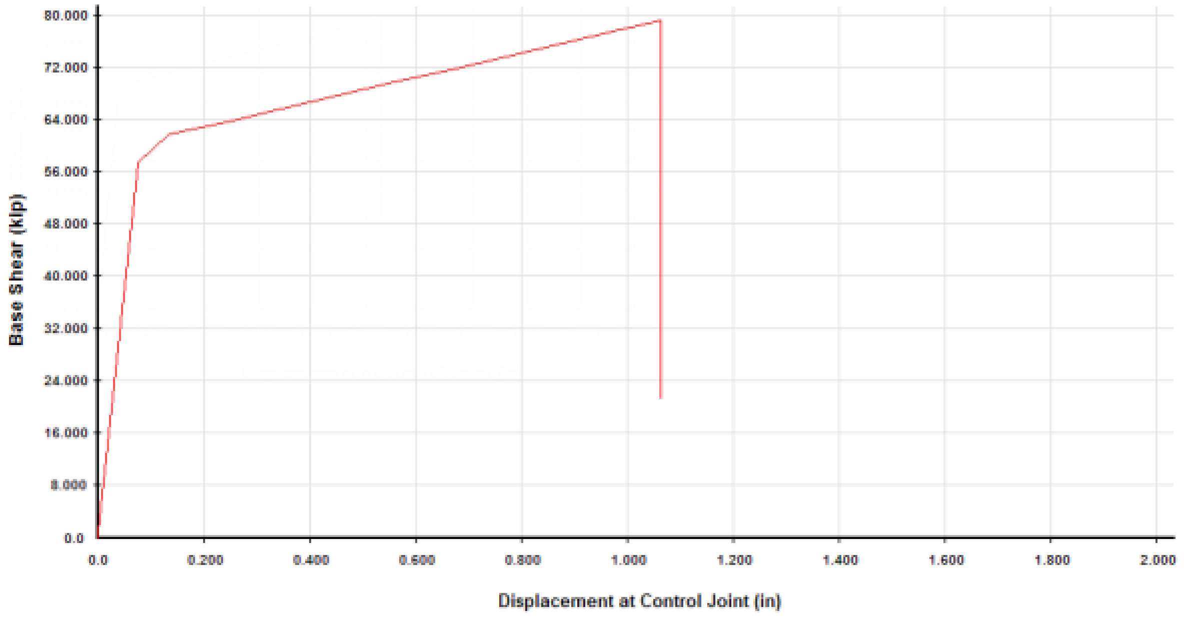


Figure 476: Capacity curve (Displacement at Control Joint, inches vs. Base Shear, kips) as calculated by STAAD.Pro

Table 567: Tabular form of the different points in Capacity curve as reported by STAAD.Pro

Load Step	Displacement in	Base Shear kip
1	0	0.153
2	0.001	1.153
3	0.078	57.837
4	0.136	61.953
5	0.243	63.979
6	0.349	65.979
7	0.455	67.979
8	0.562	69.979
9	0.668	71.979
10	0.774	73.979
11	0.880	75.979

Load Step	Displacement in	Base Shear kip
12	0.986	77.979
13	1.065	79.479
14	1.065	21.510

Results from hand calculation

<u>At Load Step 3</u>

Equation of elastic deflection at cantilever tip:

$$\delta_z = \frac{PL^3}{3EI} + \frac{PL}{GA_v}$$

where

P = 57.837 kip
L = 24 in.
E = 29,000 ksi
I = 138.0 in.^{4}
G = 11,154 ksi
A_v = 10.4323 in.^{2}

Thus elastic deformation:

$$\delta_z = \frac{57.837 \times 24^3}{3 \times 29,000 \times 138} + \frac{57.837 \times 24}{11,154 \times 10.4323} = 0.066 + 0.012 = 0.078\text{in.}$$

<u>At Load Step 13</u>

Equation of elastic deflection at cantilever tip

$$\delta_z = \frac{PL^3}{3EI} + \frac{PL}{GA_v}$$

where

P = 79.479 kip
(The other values are as listed for Load Step 3)

Thus elastic deformation at cantilever tip:

$$\delta_z = \frac{79.479 \times 24^3}{3 \times 29,000 \times 138} + \frac{79.479 \times 24}{11,154 \times 10.4323} = 0.092 + 0.016 = 0.108\text{in.}$$

Plastic rotation = 0.04 radian.

Since STAAD.Pro assumes small displacements (sinθ = θ), plastic deformation at cantilever tip

$$\delta_{z\ plastic} = L \times \theta = 24 \times 0.04 = 0.96 \text{ inch}$$

Total deflection = $\delta_{z\ elastic} + \delta_{z\ plastic} = 0.108 + 0.96 = 1.068\ inch$

Comparison

Table 568: Comparison of results

Load Step	Force P (free end) kips	Deflection (free end) in		Percent Difference
		STAAD.Pro Advanced Analysis	Hand Calculation	
3	57.837	0.078	0.078	none
13	79.479	1.065	1.068	negligible

Note: The deflection results from STAAD.Pro shown in the above table were obtained in the Postprocessing workflow.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\07 Nonlinear Analysis\Column Pushover Displacement.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Jul-07
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 24 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X77
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
DEFINE PUSHOVER DATA
FRAME 2
FYE 36.000000 ALL
HINGE PROPERTY MOMENT
TYPE 1
A 0 0 B 1 1 C 11 1.3 D 11 0.32 E 15 0.32 YM 1480 YR 0.004
IO 2 LS 7 CP 10.5
```

```
HINGE TYPE 1 ALL
GNONL 1
DISP Z 2 JOINT 2
LOADING PATTERN 1
LDSTEP 1
SPECTRUM PARAMETERS
DAMPING 2.0000
SC 4
SS 1.1
S1 1.6
SAVE LOADSTEP RESULT DISP 0.100000
END PUSHOVER DATA
LOAD 1 LOADTYPE Gravity
SELFWEIGHT Z 1
LOAD 2 LOADTYPE Push
JOINT LOAD
2 FZ 1
PERFORM PUSHOVER ANALYSIS
FINISH
```

STAAD Output

```
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS           2  NUMBER OF MEMBERS       1
     NUMBER OF PLATES           0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES         0  NUMBER OF SUPPORTS      1
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       6
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
 MORE MODES WERE REQUESTED THAN THERE ARE FREE MASSES.
 NUMBER OF MODES REQUESTED             =     6
 NUMBER OF EXISTING MASSES IN THE MODEL =     3
 NUMBER OF MODES THAT WILL BE USED      =     3
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      STAAD SPACE                                              -- PAGE NO.
3
              CALCULATED FREQUENCIES FOR LOAD CASE       3
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                     306.367                 0.00326
         2                     544.518                 0.00184
         3                    1865.405                 0.00054
            MODAL WEIGHT (MODAL MASS TIMES g) IN KIP          GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       0.000000E+00  0.000000E+00  7.674960E-02    7.674960E-02
         2       7.674960E-02  0.000000E+00  0.000000E+00    7.674960E-02
         3       0.000000E+00  7.674960E-02  0.000000E+00    7.674960E-02
 MASS PARTICIPATION FACTORS
                      MASS  PARTICIPATION FACTORS IN PERCENT
                      --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00   0.00 100.00    0.000    0.000  100.000
             2   100.00   0.00   0.00  100.000    0.000  100.000
             3     0.00 100.00   0.00  100.000  100.000  100.000
   ***WARNING: MEMBER #       1 HAS FAILED IN " DEFORMATION-CONTROLLED
```

```
ACTION.
   *** WARNING : STRUCTURE HAS REACHED AN UNSTABLE STATE WHERE
                 A VERY SMALL INCREASE IN PUSH LOAD IS CAUSING LARGE
DISPLACEMENT.
```

V. Direct Analysis of a Beam

Verify the maximum bending moment and deflection of a beam using the direct analysis method.

Reference

American Institute of Steel Construction. 2005. *Specifications for Structural Steel Buildings.* Chicago, IL:AISC. p. 16.1-435, Case 1

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

Details

A 40' long, simply-supported beam is subject to a uniformly distributed load of 100 lbs/ft and axial compression of 100 kips. The section is a W10x60 ($I_z = 341\ in^4$).

Validation

The maximum moment and deflection both occur at mid-span.

<u>Maximum moment</u>

$$M_{MAX} = M_0\left[\frac{2(\sec u - 1)}{u^2}\right]$$

where

$$M_0 = \frac{wL^2}{8} = \frac{0.1(40)^2}{8} = 20\ ft \cdot k$$

$$u = \sqrt{\frac{PL^2}{4EI}} = \sqrt{\frac{100 \times (40)^2 \times (12)^2}{4 \times 29{,}000 \times 341}} = 0.763$$

$$\frac{2(\sec u - 1)}{u^2} = \frac{2[\sec(0.763) - 1]}{0.763^2} = 1.318$$

Therefore, the maximum moment is: $20 \times 1.318 = 26.36\ ft \cdot k$

<u>Maximum deflection</u>

$$\delta_{MAX} = \delta_0\left[\frac{12(2\sec u - u^2 - 2)}{5u^4}\right]$$

where

$$\delta_0 = \frac{5wL^4}{384EI} = \frac{5(0.1)(40)^4(12)^3}{384(29{,}000)(341)} = 0.582\ in$$

$$\frac{12(2\sec u - u^2 - 2)}{5u^4} = \frac{12(2\sec(0.763) - (0.763)^2 - 2)}{5(0.763)^4} = 1.310$$

Therefore, the maximum deflection is: $0.582 \times 1.310 = 0.763\ in$

Results

Table 569: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
M_{MAX} (ft·k)	26.36	26.308	negligible	
δ_{MAX} (in)	0.763	0.757	negligible	

Note: The maximum moment and deflection results are taken from the Postprocessing workflow Beam Results and Displacements pages, respectively.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\07 Nonlinear Analysis\Direct Analysis of a Beam.std` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 17-Apr-08
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W10X60
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MX MY MZ
DEFINE DIRECT ANALYSIS
FYLD 7200 ALL
END
LOAD 1 LOADTYPE None  TITLE APPLIED LOADS
MEMBER LOAD
1 UNI GY -0.1
JOINT LOAD
1 FX 100
2 FX -100
```

```
PERFORM DIRECT ANALYSIS LRFD PDITER 15 REDUCEDEI 1
SECTION 0.5 ALL
PRINT MEMBER SECTION FORCES ALL
PRINT SECTION DISPL ALL
FINISH
```

STAAD.Pro Output

```
   MEMBER FORCES AT INTERMEDIATE SECTIONS
   ------------------------------------
   ALL UNITS ARE -- KIP  FEET
 MEMB  LOAD  SEC       AXIAL   SHEAR-Y   SHEAR-Z       MOM-X       MOM-
Y       MOM-Z
     1    1  0.50     100.00     -0.00      0.00        0.00        0.00
-20.00
   ************** END OF LATEST ANALYSIS RESULT **************
    38. PRINT SECTION DISPL ALL
  SECTION  DISPL    ALL
      STAAD PLANE                                              -- PAGE NO.
4
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - INCH
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     1       0.0000    0.0000    0.0000   -0.0078   -0.1984    0.0000
                    -0.0156   -0.3822    0.0000   -0.0234   -0.5386    0.0000
                    -0.0312   -0.6577    0.0000   -0.0390   -0.7321    0.0000
                    -0.0468   -0.7574    0.0000   -0.0545   -0.7321    0.0000
                    -0.0623   -0.6577    0.0000   -0.0701   -0.5386    0.0000
                    -0.0779   -0.3822    0.0000   -0.0857   -0.1984    0.0000
                    -0.0935    0.0000    0.0000
     MAX LOCAL  DISP =    0.75736   AT     240.00  LOAD    1   L/DISP=    633
         ************ END OF SECT DISPL RESULTS ***********
    39. FINISH
```

V. Direct Analysis of a Column

Verify the maximum bending moment and deflection of a column using the direct analysis method.

Reference

American Institute of Steel Construction. 2005. *Specifications for Structural Steel Buildings.* Chicago, IL:AISC. p. 16.1-435, Case 2

Related Links

- *G.17.2.1.4 AISC 360 Direct Analysis* (on page 2144)

Details

A 25' tall cantilevered column (i.e., a flagpole) is subject to an axial load of 30 kips. The section is a W8x18 ($I_z = 61.9\ in^4$).

Validation

A notional load factor of 0.003 is used. Therefore, the horizontal notional load at the top (free end) of the column is $H = 0.03 \times 30 = 0.09\ kips$.

Maximum moment

The maximum moment occurs at the fixed base.

$$M_{MAX} = M_0\left[\frac{\tan a}{a}\right]$$

where

$$M_0 = HL = 0.09 \times 25 = 2.25\ ft \cdot k$$

$$a = \sqrt{\frac{PL^2}{EI}} = \sqrt{\frac{30 \times (25)^2 \times (12)^2}{29{,}000 \times 61.9}} = 1.226$$

$$\frac{\tan a}{a} = \frac{\tan(1.226)}{1.226} = 2.273$$

Therefore, the maximum moment is: $2.25 \times 2.273 = 5.115\ ft \cdot k = 61.37\ in \cdot k$

Maximum deflection

The maximum deflection occurs at the free end.

$$\delta_{MAX} = \delta_0\left[\frac{3(\tan, a, -, a)}{a^3}\right]$$

where

$$\delta_0 = \frac{HL^3}{3EI} = \frac{(0.09)(25)^3(12)^3}{3(29{,}000)(61.9)} = 0.451\ in$$

$$\frac{3(\tan, a, -, a)}{a^3} = \frac{3(\tan(1.226) - 1.226)}{1.226^3} = 2.537$$

Therefore, the maximum deflection is: $0.451 \times 2.537 = 1.145\ in$

Results

Table 570: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
M_{MAX} (in·k)	61.37	61.36	negligible	
δ_{MAX} (in)	1.145	1.14538	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\07 Nonlinear Analysis\Direct Analysis of a Column.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jun-09
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 300 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X18
SUPPORTS
1 FIXED
2 FIXED BUT FX FY MX MY MZ
DEFINE DIRECT ANALYSIS
FYLD 50 LIST 1
NOTIONAL LOAD FACTOR 0.002
END
LOAD 1 LOADTYPE None  TITLE VERTICAL LOAD
JOINT LOAD
2 FY -30
perf analysi
change
LOAD 2 LOADTYPE Live REDUCIBLE TITLE VERT + NOTIONAL
REPEAT LOAD
1 1.0
NOTIONAL LOAD
1 X 0.003
PERFORM DIRECT ANALYSIS LRFD
load list 2
print joint disp list 2
print memb force list 1
FINISH
```

STAAD.Pro Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      2    2    1.14538  -0.05900   0.00000   0.00000   0.00000  -0.00586
   ************** END OF LATEST ANALYSIS RESULT **************
    46. PRINT MEMB FORCE LIST 1
  MEMB     FORCE    LIST     1
      STAAD SPACE                                              -- PAGE NO.
4
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
```

```
    1    2     1     30.00      0.09      0.00      0.00      0.00      61.36
               2    -30.00     -0.09      0.00      0.00      0.00       0.00
 ************** END OF LATEST ANALYSIS RESULT **************
  47. FINISH
```

V. Single Column P-Delta Analysis

Determine the support forces at the base of the column.

Details

A 12 ft tall column with a fixed base and free end is subject to a 3 kip axial load and a 5 kip lateral load. The section is a W8x10 using steel with a modulus of elasticity of 29,000 ksi.

Validation

Moment at the base including P-Delta

$$M = H \times L + P \times \Delta$$

where

H	=	the lateral load
L	=	the length of the column
P	=	the axial compression load
Δ	=	the displacement due to the lateral load at the free end of the column

$$\frac{H L^3}{3EI} = \frac{5 \times (144)^3}{3 \times 29{,}000 \times 30.80} = 5.57 \text{ in}$$

Therefore, the moment is:

$$M = 5\ (144) + 3\ (5.57) = 736.72 \text{ in}\cdot\text{kip}$$

Results

Table 571:

Parameter	Reference	STAAD.Pro	Difference	Comment
Base moment (ft·kip)	736.72	736.72	none	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples`

`\Verification Models\07 Nonlinear Analysis\Single Column P-Delta Analysis.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-May-08
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 12 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
SET SHEAR
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X10
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
PRINT MEMBER PROPERTIES ALL
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -3
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 3 LOADTYPE None  VERTICAL
REPEAT LOAD
1 1.0 2 1.0
PDELTA ANALYSIS LARGEDELTA PRINT STATICS CHECK
unit kips inches
PRINT JOINT DISPLACEMENTS ALL
PRINT MEMBER FORCES
FINISH
```

STAAD Output

```
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIPS INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    3     1      3.00      5.12     0.00      0.00      0.00     736.72
                 2     -3.00     -5.12     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
```

Related Links

- *G.17.2.1 P-Delta Analysis* (on page 2141)

V.08 Dynamic Analysis

V. Beam Subject to response spectrum

Find the maximum moment due to the time history loading and compare theoretical answers to the STAAD solution.

Reference

1. Biggs, John M., *Introduction to Structural Dynamics*, McGraw Hill, 1964, pp. 256-263
2. Blevins, Robert D., *Formulas for Natural Frequency and Mode Shape*, Van Nostrand-Reinhold, 1979.

Problem

The supports of a simply supported beam are subjected to an acceleration time history. The maximum bending moment in the beam is computed for the first mode of the structure. This problem demonstrates the capabilities of STAAD to calculate the correct modal response of a structure utilizing response spectrum data.

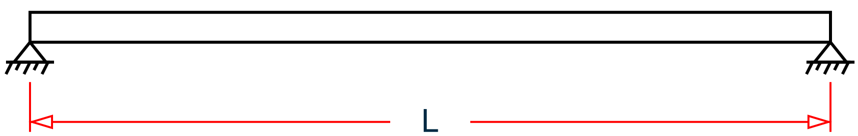


Figure 477: Simple span beam

$L = 240$ in.

The STAAD model consists of 11 nodes and 10 elastic beam elements. Node 1 is completely restrained with the exception of having rotational freedom in the Z direction, the remaining nodes are restrained except for X and Y displacements and Z rotations. Node 11 is additionally restrained against displacements in the Y direction to provide for the simple support condition . Only the contribution of the first mode of the structure is considered.

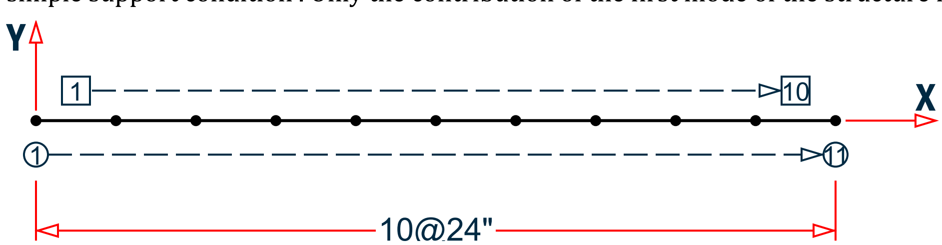


Figure 478: Finite element model

Theoretical Solution

Material Properties

$E = 30 \times 10^6$ lb/in^2

$EI = 1.0 \times 10^{10}$ lb-in^2

$m = 0.2$ lb-sec^2 / in^2

$h = 14.0$ in.

From Reference 2, Table 8-1, page 108, the fundamental frequency of the beam is:

$$f_i = \frac{\lambda_i^2}{2\pi l^2}\sqrt{\frac{EI}{m}} = \frac{9.869}{2\pi(240)^2}\sqrt{\frac{1.0(10)^{10}}{0.2}} = 6.098hz$$

The modal participation factor for the fundamental mode is:

$$\Gamma = \frac{\int_0^1 m\phi(x)\ d\ x}{\int_0^1 m\phi^2(x)\ d\ x}$$

Where the first mode shape, $\varphi(x) = sin(\pi x / 1)$

$$\Gamma = \frac{m\int_0^1 \sin\frac{\pi x}{1}\ d\ x}{m\int_0^1 \sin^2\frac{\pi x}{1}\ d\ x} = 4 \Big/ \pi$$

The maximum relative modal displacement is given by:

$A_{max} = \Gamma u^0{}_{max}$

where

$u^0{}_{max}$	=	$y''_{so} / \omega^2 (DLF)_{max}$
ω	=	$2\pi * 6.098Hz$
y''_{so}	=	$1.0g$
$(DLF)_{max}$	=	1.648 at 6.098 Hz

therefore:

$A_{max} = 4(1.648)(386.4)/[\pi(2\pi)^2(6.098)^2] = 0.5523$ in.

The bending moment

$M = -EI\delta^2 u/(\delta x^2)$

Where u for the first mode = A sin(πx/l)

$\delta^2 u/(\delta x^2) = -\pi^2/l^2 A \cdot \sin(\pi x/l)$

$M = AEI \cdot (\pi^2/l^2) \cdot \sin(\pi x/l)$

$M_{max} = A_{max} EI \cdot (\pi^2/l^2)$

at $x = l/2$

$M_{max} = 1(10)^{10} \cdot (0.5523) \cdot \pi^2/(240^2) = 946.351(10)^3$ lb·in

at x=l/2

Comparison

Table 572: Comparison of results

Solution	Theory	STAAD.Pro	Difference
Bending Moment (kip-inch)	946.351	947.088	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Beam Subject to response spectrum.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* RESPONSE OF A SIMPLY SUPPORTED BEAM TO A SHOCK SPECTRUM
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 24 0 0; 3 48 0 0; 4 72 0 0; 5 96 0 0; 6 120 0 0; 7 144 0 0;
8 168 0 0; 9 192 0 0; 10 216 0 0; 11 240 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11;
MEMBER PROPERTY AMERICAN
1 TO 10 PRIS AX 20.4082 IX 40 IY 3.6139 IZ 333.333 YD 14 ZD 1.45777
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
DENSITY 3.78672
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 1
SUPPORTS
1 FIXED BUT MZ
2 TO 10 FIXED BUT FX FY MZ
11 FIXED BUT FX MZ
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SPECTRUM SRSS Y 1 ACC SCALE 386.4 DAMP 0.001
0.15 1.648; 0.17 1.648;
PERFORM ANALYSIS
PRINT MEMBER FORCES LIST 5
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE           FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                      6.069                 0.16476
  RESPONSE SPECTRUM LOAD     1
     RESPONSE LOAD CASE      1
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN        GENERALIZED
      MODE           X             Y             Z             WEIGHT
         1       0.000000E+00  1.478714E+04  0.000000E+00    9.273617E+03
     SRSS          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   1.761987E+04  1.669251E+04  0.000000E+00 POUN
     MISSING WEIGHT X Y Z  -1.761987E+04 -1.905373E+03  0.000000E+00 POUN
       MODAL WEIGHT X Y Z   0.000000E+00  1.478714E+04  0.000000E+00 POUN
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
              1                      1.64933       0.00100
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN POUN LENGTH IN INCH
  -----------------------------------------------------------
                                                            MOMENTS ARE
ABOUT THE ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX
MY          MZ
      1      0.165        0.00    24388.86        0.00        0.00
0.00  2926662.97
      STAAD SPACE                                                -- PAGE NO.
4
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN POUN
           --------------------------------------        ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y       Z
   1    0.00 88.59  0.00    0.000   88.585    0.000      0.00 24388.86
0.00

---------------------------
                                  TOTAL SRSS  SHEAR      0.00 24388.86
0.00
                                  TOTAL 10PCT SHEAR      0.00 24388.86
0.00
                                  TOTAL ABS   SHEAR      0.00 24388.86
0.00
    34. PRINT MEMBER FORCES LIST 5
  MEMBER   FORCES   LIST     5
      STAAD SPACE                                                -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- POUN INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      5    1     5      0.00   1931.41     0.00      0.00      0.00  900734.25
                 6      0.00   1931.41     0.00      0.00      0.00  947088.00
```

V. First Modal Frequency of a Cantilever Beam

To calculate the Natural frequency of vibration for a rectangular cantilever beam with a mass at the free end.

Reference

Hand calculation using known formulas.

Problem

Find the natural frequency of vibration, ω, of the cantilever beam.

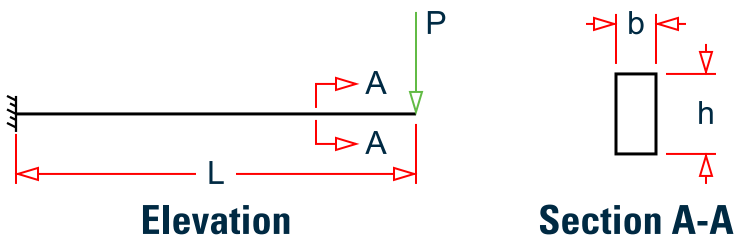


Figure 479: Cantilever beam with mass at free end

E = 30,000 ksi

h = 12.0 in

b = 6.0 in

P = 10.0 k

L = 120 in

Hand Calculations

Stiffness at free end:

$$k = 3EI/L^3 = 45 \text{ k/in}$$

Mass

$$m = w/g = 10.0 \text{ k} / (386.4 \text{ k-sec}^2/\text{in}) = 0.02588 \text{ k-sec}^2/\text{in}$$

Circular frequency:

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{45}{0.02588}} = 41.7 \text{rad/sec}$$

= 6.637 cycles/sec

Comparison

Table 573: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Frequency, f (Hz)	6.637	6.633	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\First Modal Frequency of a Cantilever Beam.STD` is typically installed with the program.

```
STAAD PLANE A RECTANGULAR CANTILEVER BEAM WITH A MASS AT THE FREE END
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
INPUT WIDTH 72
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 5 0 0; 3 10 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 2 PRIS YD 12 ZD 6
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.32e+06
POISSON 0.290909
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
CUT OFF MODE SHAPE 1
UNIT FEET KIP
LOAD 1 VERTICAL LOAD
MEMBER LOAD
2 CON Y -10 5
1 2 UNI GY -0.001
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
*PRINT MODE SHAPES
FINISH
```

STAAD Output

```
                 CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                       6.633                 0.15076
              MODAL WEIGHT (MODAL MASS TIMES g) IN KIP         GENERALIZED
      MODE            X             Y             Z              WEIGHT
         1       0.000000E+00  1.000514E+01  0.000000E+00    1.000299E+01
 MASS PARTICIPATION FACTORS
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00  99.98   0.00    0.000   99.976    0.000
```

V. Modal Frequencies of a Cantilever Beam

To find the first three natural frequencies of vibration for a cantilever beam with a uniform mass.

Reference

Thomson, W.T., *Vibration Theory and Applications*, Prentice-Hall, Inc., 1965. Also compared with ANSYS® Finite Element software.

Problem

Find the first three natural frequencies, f_1, f_2 and f_3 of the cantilever beam.

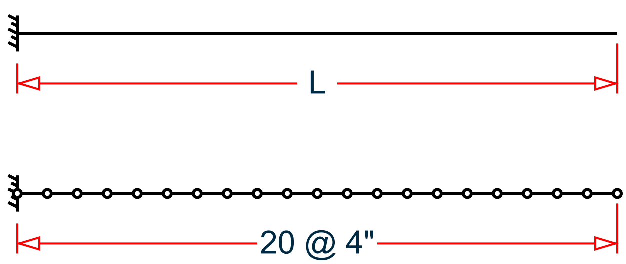


Figure 480: Model for dynamic beam no. 4

E = 30,000 ksi

I = 1.3333 in^4

A = 4 in^2

w = 1.124 lb/in

L = 80 in

Comparison

Table 574: Comparison of results

Result Type	Theory	ANSYS®	STAAD.Pro	Difference
Frequency, f_1 (Hz)	10.247	10.247	10.237	none
Frequency, f_2 (Hz)	64.221	64.197	63.974	none
Frequency, f_3 (Hz)	179.82	180.14	178.672	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Frequencies of a Cantilever Beam.STD` is typically installed with the program.

```
STAAD PLANE : FREQUENCIES OF A CANTILEVERED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE:  THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS",
*      PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 8 0 0; 4 12 0 0; 5 16 0 0; 6 20 0 0; 7 24 0 0;
8 28 0 0; 9 32 0 0; 10 36 0 0; 11 40 0 0; 12 44 0 0; 13 48 0 0;
14 52 0 0; 15 56 0 0; 16 60 0 0; 17 64 0 0; 18 68 0 0; 19 72 0 0;
20 76 0 0; 21 80 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 4 IZ 1.3333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
CUT OFF FREQUENCY 200
CUT OFF MODE SHAPE 3
LOAD 1 UNIFORM MASS FOR MODAL ANALYSIS
MEMBER LOAD
1 TO 20 UNI GY 1.124
```

```
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
      MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                      10.237                 0.09768
        2                      63.974                 0.01563
        3                     178.672                 0.00560
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN        GENERALIZED
     MODE           X             Y             Z              WEIGHT
        1       0.000000E+00  5.507472E+01  0.000000E+00    2.252359E+01
        2       0.000000E+00  1.693755E+01  0.000000E+00    2.278794E+01
        3       0.000000E+00  5.820203E+00  0.000000E+00    2.323735E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
          MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
            1    0.00  62.82   0.00    0.000   62.819    0.000
            2    0.00  19.32   0.00    0.000   82.138    0.000
            3    0.00   6.64   0.00    0.000   88.777    0.000
```

V. Modal Frequencies of a Simply Supported Beam

To find the natural frequencies of vibration for a simply supported beam.

Reference

Roark's Formulas for Stress and Strain, Warren C. Young, McGraw Hill, 6th edition.

Problem

Find the first five flexural natural frequencies of the simple beam. Neglect shear deformation and rotary inertia.

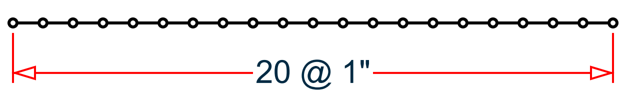


Figure 481: Model for dynamic beam no. 7

E = 10,000 ksi

density = 0.1 lb/in^3

A_x = 2.0 in^2

$I_x = 0.6667\ in^4$

$L = 20$ in

Hand Calculations

Weight

$w_{weight} = Ax * density = 2.0\ (0.1) = 0.2$ lb/in

$w_{mass} = 0.2\ /(386.4) = 0.000518$

From Table 36, Item 1b of the reference:

$$f_n = \frac{k_n}{2\pi}\sqrt{\frac{EI}{wl^4}} = \frac{k_n}{2\pi}\sqrt{\frac{10(10)^6(0.6667)}{0.000518(20)^4}} = 45.16 \cdot k_n$$

Table 575: Modal stiffness and natural frequencies

Mode	k_c	Frequency (Hz)
1	9.87	445.7
2	39.5	1,783.7
3	88.8	4,010.0
4	158	7,134.9
5	247	11,154

Comparison

Table 576: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Frequency, f_1 (Hz)	445.7	445.495	none
Frequency, f_2 (Hz)	1,783.7	1,781.968	none
Frequency, f_3 (Hz)	4,010.0	4,009.310	none
Frequency, f_4 (Hz)	7,134.9	7,127.074	none
Frequency, f_5 (Hz)	11,154	11,133.978	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Frequencies of a Simply Supported Beam.STD` is typically installed with the program.

```
STAAD PLANE : NATURAL FREQUENCIES OF A S.SUPPORTED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE: W.C.YOUNG., "ROARK'S FORMULAS FOR STRESS &amp; STRAIN", 6TH ED.
*  CASE 1B, TABLE 36, PAGE 714
*
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 12 11 0 0; 13 12 0 0; 14 13 0 0;
15 14 0 0; 16 15 0 0; 17 16 0 0; 18 17 0 0; 19 18 0 0; 20 19 0 0;
21 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.33
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 IZ 0.666667
SUPPORTS
1 21 FIXED BUT MZ
2 TO 20 FIXED BUT FY MZ
CUT OFF MODE SHAPE 0
CUT OFF FREQUENCY 12000
LOAD 1
SELFWEIGHT Y -1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                     445.495                 0.00224
         2                    1781.968                 0.00056
         3                    4009.310                 0.00025
         4                    7127.074                 0.00014
         5                   11133.978                 0.00009
```

```
          MODAL WEIGHT (MODAL MASS TIMES g) IN POUN        GENERALIZED
    MODE           X             Y             Z              WEIGHT
       1       0.000000E+00  3.228953E+00  0.000000E+00    2.000000E+00
       2       0.000000E+00  5.965927E-28  0.000000E+00    2.000000E+00
       3       0.000000E+00  3.469944E-01  0.000000E+00    2.000000E+00
       4       0.000000E+00  7.262741E-31  0.000000E+00    2.211146E+00
       5       0.000000E+00  1.165685E-01  0.000000E+00    2.000000E+00
MASS PARTICIPATION FACTORS
                   MASS  PARTICIPATION FACTORS IN PERCENT
                   --------------------------------------
         MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
           1     0.00  84.97   0.00    0.000   84.972    0.000
           2     0.00   0.00   0.00    0.000   84.972    0.000
           3     0.00   9.13   0.00    0.000   94.104    0.000
           4     0.00   0.00   0.00    0.000   94.104    0.000
           5     0.00   3.07   0.00    0.000   97.171    0.000
```

V. Modal Response of a 3D Frame

Find the natural frequencies of a space frame. Compare the results from subspace iteration method used by STAAD.Pro with the results from EASE2® and ANSYS®.

Reference

1. Problem 1, from the ASME 1972 Program Verification and Qualification Library
2. DeSalvo, G.J., and Swanson, J.A., *ANSYS Engineering Analysis System Examples Manual*, Swanson Analysis Systems, Inc., 1979, Example Problem No. 2.
3. Peterson, F.E., *EASE2 Elastic Analysis for Structural Engineering Example Problem Manual*, Engineering Analysis Corporation, 1981, Example 2.03.

Problem

A three dimensional frame is analyzed for its natural frequencies and the associated mode shapes using the subspace iteration method offered by STAAD.Pro.

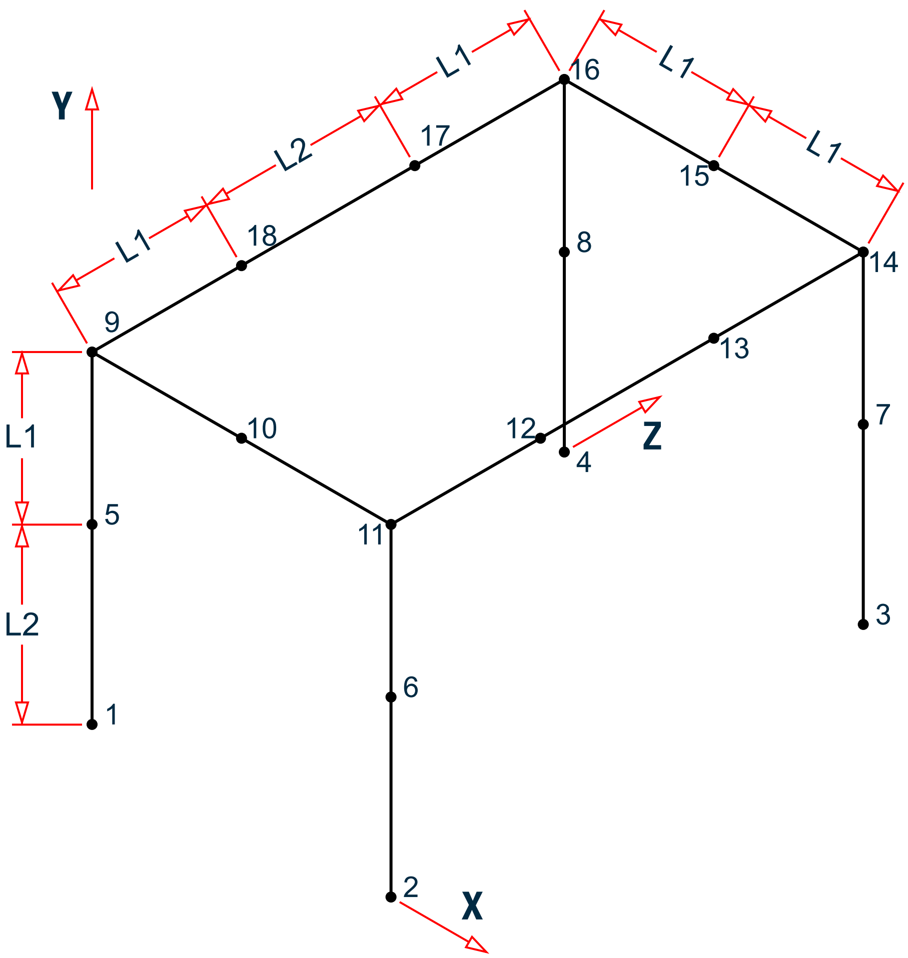


Figure 482: Frame model

L1 = 8.625 ft
L2 = 10 ft

All dimensions are in inches.

The only entity present in the model is the general purpose three dimensional beam.

Cross-section Properties

All the beam elements have the same cross section:

Outside radius, $R_o = 1.1875$ *in.*
Inside radius, $R_i = 1.0335$ *in.*
Area, $A = \pi(R_o^{\ 2} - R_i^{\ 2}) = 1.074532\ in^2$
Moment of Inertia, $I = \pi/4(R_o^{\ 4} - R_i^{\ 4})$
$I = 0.665747\ \text{in}^4$
Torsion constant = twice the inertia, for closed circular sections = $1.331494\ in^4$

The expression for the shear flexibility factor is derived from the ratio of maximum shear stress to the average shear stress:

$\alpha = AQ/Ib$

Where the four items – A, Q, I, and b are properties of the half cross-section about the centerline. For the circular section they are:

$A = \pi/2(Ro^2 - Ri^2)$

$Q = 2/3(Ro^3 - Ri^3)$

$I = \pi/8(Ro^4 - Ri^4)$

$I = 2(Ro - Ri)$

The final expression for the shear flexibility factor, for a circular tube section is:

$$\alpha = \frac{AQ}{Ib} = \frac{4}{3}\frac{R_o^3 - R_i^3}{\left(R_o^2 - R_i^2\right)\left(R_o - R_i\right)}$$

$\alpha = 1.993620$

Hence the shear area is entered as

AY = Cross Section Area / α = 1.074532 / 1.99362 = 0.538985 in^2

Comparison

The following table compares Twenty-Four Natural Frequencies

Table 577: Comparison of results

Mode Number	DeSalvo and Swanson[2]	Peterson[3]	STAAD.Pro
1	111.52	111.53	111.236
2	115.95	115.96	115.801
3	137.60	137.61	137.169
4	218.02	218.03	215.799
5	404.23	404.23	404.270
6	422.70	422.72	422.642
7	451.72	451.75	451.599
8	553.99	554.07	549.012
9	735.70	735.81	733.589
10	762.32	762.44	758.563
11	852.57	852.72	851.354

Mode Number	DeSalvo and Swanson[2]	Peterson[3]	STAAD.Pro
12	894.08	894.26	892.281
13	910.21	910.40	893.060
14	916.98	917.18	910.829
15	940.02	940.02	932.286
16	959.98	960.27	956.391
17	971.15	971.44	964.012
18	976.92	977.22	967.267
19	1012.2	1012.5	981.536
20	1028.4	1028.8	1009.256
21	1123.6	1123.9	1070.562
22	1134.5	1134.9	1123.161
23	1164.1	1164.4	1149.368
24	1216.7	1217.2	1199.882

In both references by DeSalvo and Swanson[2] and Peterson[3], the number of dynamic degrees of freedom has been reduced from 42 to 24, by means of the Guyan method. No such reduction is performed in the STAAD.Pro model.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a 3D Frame.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* Natural Modes of a space frame
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 27.25 0 0; 3 0 10 0; 4 27.25 10 0; 5 0 18.625 0;
6 8.625 18.625 0; 7 18.625 18.625 0; 8 27.25 18.625 0; 9 0 18.625 8.625;
10 27.25 18.625 8.625; 11 0 0 17.25; 12 27.25 0 17.25; 13 0 10 17.25;
14 27.25 10 17.25; 15 0 18.625 17.25; 16 8.625 18.625 17.25;
17 18.625 18.625 17.25; 18 27.25 18.625 17.25;
MEMBER INCIDENCES
1 1 3; 2 2 4; 3 3 5; 4 4 8; 5 5 6; 6 6 7; 7 7 8; 8 5 9; 9 8 10; 10 9 15;
11 10 18; 12 11 13; 13 12 14; 14 13 15; 15 14 18; 16 15 16; 17 16 17;
```

```
18 17 18;
MEMBER PROPERTY AMERICAN
1 TO 17 -
18 PRIS AX 1.07453 AY 0.538985 AZ 0.538985 IX 1.33149 IY 0.665747 -
IZ 0.665747
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.79e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 24
SUPPORTS
1 2 11 12 FIXED
LOAD 1
JOINT LOAD
3 4 13 14 FX 3.4517 FY 3.4517 FZ 3.4517
6 7 16 17 FX 3.4517 FY 3.4517 FZ 3.4517
9 10 FX 3.4517 FY 3.4517 FZ 3.4517
5 8 15 18 FX 9.7973 FY 9.7973 FZ 9.7973
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                     111.236                  0.00899
         2                     115.801                  0.00864
         3                     137.169                  0.00729
         4                     215.799                  0.00463
         5                     404.270                  0.00247
         6                     422.642                  0.00237
         7                     451.599                  0.00221
         8                     549.012                  0.00182
         9                     733.589                  0.00136
        10                     758.563                  0.00132
        11                     851.354                  0.00117
        12                     892.281                  0.00112
        13                     893.060                  0.00112
        14                     910.829                  0.00110
        15                     932.286                  0.00107
        16                     956.391                  0.00105
        17                     964.012                  0.00104
        18                     967.267                  0.00103
        19                     981.536                  0.00102
        20                    1009.256                  0.00099
        21                    1070.562                  0.00093
        22                    1123.161                  0.00089
        23                    1149.368                  0.00087
        24                    1199.882                  0.00083
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       6.984127E+01  6.404211E-24  1.284664E-24    6.127270E+01
```

```
         2       1.547059E-22  1.789605E-20  7.005318E+01    5.765801E+01
         3       1.064176E-21  1.226078E-19  9.256770E-23    7.213569E+01
         4       9.147252E-20  1.045721E-17  1.002150E-20    7.915618E+01
         5       4.553346E-19  4.244579E-03  4.985566E-20    1.532638E+01
         6       5.447627E-22  1.825470E+01  1.949591E-25    1.469096E+01
         7       1.160472E-20  1.375280E-18  5.112414E-03    1.450510E+01
         8       1.699531E-22  8.948109E-20  2.966451E-01    1.825927E+01
         9       1.055938E-23  9.521535E+00  1.626382E-22    1.177226E+01
        10       9.808838E-04  8.794438E-20  3.656936E-22    1.174657E+01
        11       1.321191E+00  9.714550E-18  1.273002E-20    1.582128E+01
        12       6.082169E-21  1.911260E+00  1.769785E-20    1.624941E+01
        13       1.722239E+00  1.063875E-16  1.119734E-19    3.802991E+01
        14       4.611355E-20  4.549234E-17  5.528088E-21    2.195002E+01
        15       4.392235E-20  6.381431E-17  2.363042E-01    1.628835E+01
        16       1.400997E-20  4.122974E+00  1.604295E-20    1.648079E+01
        17       8.893063E-19  1.239356E-15  1.393843E-18    1.622824E+01
        18       1.975827E-18  2.780487E-15  3.083966E+00    1.640512E+01
      STAAD SPACE                                              -- PAGE NO.
4
        19       3.491399E-01  1.038217E-15  5.994489E-19    2.835638E+01
        20       5.049440E-19  1.122183E+01  4.036566E-19    2.560377E+01
        21       3.350978E-18  3.211568E-15  7.101758E-19    3.464248E+01
        22       3.635973E-01  3.238417E-15  1.516123E-18    3.137071E+01
        23       8.666280E-17  9.060816E-14  3.752768E-17    1.555328E+01
        24       6.382000E-02  1.876943E-13  5.827892E-17    2.520961E+01
      STAAD SPACE                                              -- PAGE NO.
5
 MASS PARTICIPATION FACTORS
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
          MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z
            1    94.76   0.00   0.00   94.756    0.000    0.000
            2     0.00   0.00  95.04   94.756    0.000   95.044
            3     0.00   0.00   0.00   94.756    0.000   95.044
            4     0.00   0.00   0.00   94.756    0.000   95.044
            5     0.00   0.01   0.00   94.756    0.006   95.044
            6     0.00  24.77   0.00   94.756   24.773   95.044
            7     0.00   0.00   0.01   94.756   24.773   95.051
            8     0.00   0.00   0.40   94.756   24.773   95.453
            9     0.00  12.92   0.00   94.756   37.691   95.453
           10     0.00   0.00   0.00   94.758   37.691   95.453
           11     1.79   0.00   0.00   96.550   37.691   95.453
           12     0.00   2.59   0.00   96.550   40.284   95.453
           13     2.34   0.00   0.00   98.887   40.284   95.453
           14     0.00   0.00   0.00   98.887   40.284   95.453
           15     0.00   0.00   0.32   98.887   40.284   95.774
           16     0.00   5.59   0.00   98.887   45.878   95.774
           17     0.00   0.00   0.00   98.887   45.878   95.774
           18     0.00   0.00   4.18   98.887   45.878   99.958
           19     0.47   0.00   0.00   99.360   45.878   99.958
           20     0.00  15.23   0.00   99.360   61.103   99.958
           21     0.00   0.00   0.00   99.360   61.103   99.958
           22     0.49   0.00   0.00   99.854   61.103   99.958
           23     0.00   0.00   0.00   99.854   61.103   99.958
           24     0.09   0.00   0.00   99.940   61.103   99.958
```

V. Modal Response of a Beam

Find the natural frequencies for a beam and compare theoretical answers to the STAAD.Pro solution.

Reference

Timoshenko, S., *Vibration Problems in Engineering*, Third Edition, D. Van Nostrand Company, Inc., 1955, page 322

Problem

The first five natural frequencies and the associated mode shapes are computed for the flexural motion of a simply supported beam.

Figure 483: Simple beam diagram

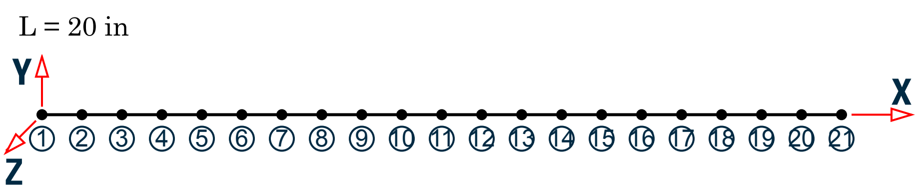


Figure 484: Finite element model

The simply supported beam is divided into twenty spanwise beam elements. At nodes 1 and 21, all degrees of freedom except the rotation about the Z axis are restrained. For the remaining nodes, only the translation along Y and the rotation about Z are permitted. Both shear deformation and rotary inertia have been excluded from the model. The mass matrix is a diagonal matrix.

Cross-section Properties

Rectangular Section: 1 inch Width x 2 inch Depth

Area = 2 in^2

$$J = \frac{b^3 a}{16}\left\{\frac{16}{3} + 3.36\left(\frac{b}{a}\right)\left[1 - \frac{1}{12}\left(\frac{b}{a}\right)^4\right]\right\}$$

where

a	=	2
b	=	1
J	=	0.4578 inch^4
I_2	=	$1 \times 2^3/12$ inch^4
I_3	=	$2 \times 1^3/12$ inch^4

Theoretical Results

The natural bending frequencies, for a uniform beam with hinged ends, are given by:

$$f_n = \frac{\pi m^2}{2l^2}\sqrt{\frac{EIg}{A\gamma}}$$

where

f_n	=	natural frequency for mode n, in cycles per second
l	=	span of the beam
E	=	elastic modulus
I	=	cross-section moment of inertia
g	=	gravitational constant
A	=	cross-section area
γ	=	weight density

The parameters used in the frequency equation are:

$l = 20\ in$
$E = 10x10^6\ psi$
$I = 0.6667\ in^4$
$g = 386.4\ in/s^2$
$A = 2.0\ in^2$
$\gamma = 0.1\ lbs/in^3$

from which:

$$f_n = n^2 x\ 445.686$$

Comparison

The table below shows the natural frequencies computed from the theoretical equation and the subspace iteration method available within STAAD.Pro. Frequencies are in cycles per second.

Table 578: Comparison of results

Mode Number	Theoretical	STAAD.Pro	Difference
1	445.686	445.506	negligible
2	1,782.74	1,782.012	negligible
3	4,011.17	4,009.410	negligible
4	7,130.97	7,127.250	negligible
5	11,142.1	11,134.253	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Beam.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
```

```
END JOB INFORMATION
* Natural modes of a simple beam
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 2 0 0; 4 3 0 0; 5 4 0 0; 6 5 0 0; 7 6 0 0; 8 7 0 0;
9 8 0 0; 10 9 0 0; 11 10 0 0; 12 11 0 0; 13 12 0 0; 14 13 0 0;
15 14 0 0; 16 15 0 0; 17 16 0 0; 18 17 0 0; 19 18 0 0; 20 19 0 0;
21 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 IZ 0.6667
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 5
SUPPORTS
1 21 FIXED BUT MZ
2 TO 20 FIXED BUT FY MZ
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                     445.506                 0.00224
         2                    1782.012                 0.00056
         3                    4009.410                 0.00025
         4                    7127.250                 0.00014
         5                   11134.253                 0.00009
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE            X             Y             Z              WEIGHT
         1       0.000000E+00  3.228953E+00  0.000000E+00    2.000000E+00
         2       0.000000E+00  6.313263E-28  0.000000E+00    2.000000E+00
         3       0.000000E+00  3.469944E-01  0.000000E+00    2.000000E+00
         4       0.000000E+00  3.242349E-30  0.000000E+00    2.211146E+00
         5       0.000000E+00  1.165685E-01  0.000000E+00    2.000000E+00
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
             MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
               1     0.00  84.97   0.00    0.000   84.972    0.000
               2     0.00   0.00   0.00    0.000   84.972    0.000
               3     0.00   9.13   0.00    0.000   94.104    0.000
```

```
              4     0.00   0.00   0.00    0.000   94.104    0.000
              5     0.00   3.07   0.00    0.000   97.171    0.000
```

V. Modal Response of a Circular Plate

Find the natural frequencies of a circular plate and compare theoretical answers to the STAAD solution.

Reference

Blevins, Robert D., *Formulas for Natural Frequency and Mode Shape*, Van Nostrand Reinhold Company, 1979, Page 240.

Problem

A flat circular plate is simply supported around the entire perimeter. The first six modes and their associated natural frequencies are to be computed using the subspace iteration method offered by STAAD. This problem demonstrates that the natural frequencies of an axi-symmetric structure can be accurately computed utilizing a 180 degree model with the appropriate boundary conditions.

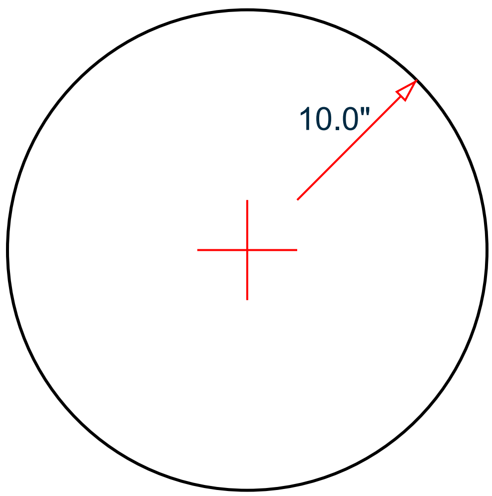


Figure 485: Diagram of circular plate

The 180 degree sector was modeled using radial lines at intervals of 15 degrees. Tangential lines were then located utilizing a relationship such that the aspect ratio of the quad-plate elements was approximately 1.0. All rotations normal to the plane of the plate were restrained. In-plane translations for all nodes were restrained because the theoretical solution does not consider in-plane effects. Rotations about the global Y axis for the nodes at X=0.0 were restrained because this is a symmetry boundary. Z translation of all nodes on the outside radius were restrained to provide for the simply-supported condition.

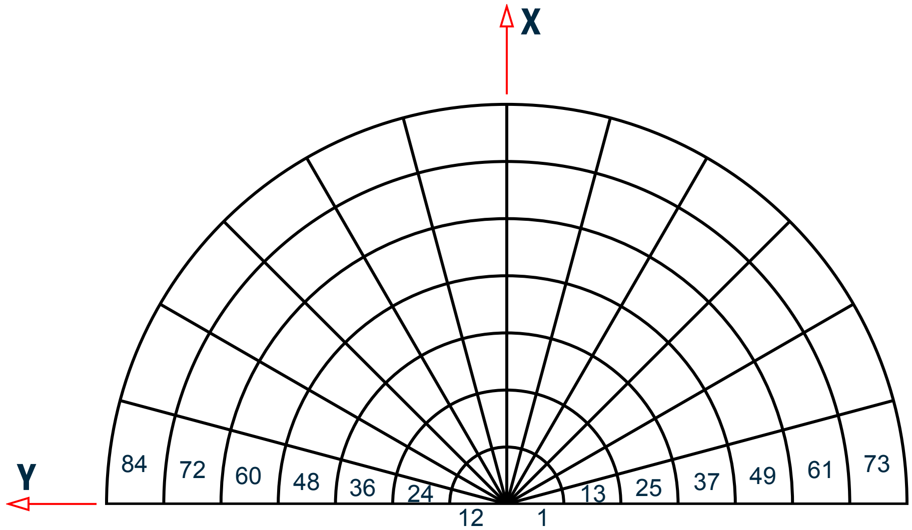


Figure 486: Finite model for half circular plate

It should be noted that the outside edge of the plate is a series of secant lines instead of a true arc. This will result in a loss of about 1% of the plate's true mass or about .5% of the mass that is effective for this problem.

Therefore, it is not unlikely that a few natural frequencies will be lower than the theoretical values instead of higher which is typical for a finite element analysis using plate elements. In addition, a true simple-support condition for this problem would require restraining the component of rotation that is radial to the outside edge.

Theoretical Calculations

From the reference case 2 in Table 11-1, the first six natural frequencies of the plate are described by the following equation:

$$f_{ij} = \frac{\lambda_{ij}^2}{2\pi a^2}\sqrt{\frac{Eh^3}{12\gamma\left(1-\nu^2\right)}}$$

= dimensionless parameter associated with the mode indices i,j

where

i	=	number of nodal diameters in this mode shape
j	=	number of nodal circles in this mode shape not counting the boundary
ν	=	Poisson's ratio
E	=	elastic modulus
h	=	plate thickness
γ	=	mass of plate per unit area
a	=	radius of plate

The numerical values used for this example are:

v = 0.30

E = 10.0×10P^6 psi

h = 0.10 inches

$$\gamma = \frac{\left(0.10 \text{ lb}_m / \text{in}^3\right)\left(0.10 \text{ in}\right)}{\left(386.4 \text{ lb}_m \cdot \text{in/lb}_f \cdot \text{s}^2\right)} = 2.588 \times 10^{-5} \text{ in/lb}_f \cdot \text{s}^2 \Big/ \text{in}^3$$

a = 10.0 inches

with the numerical values used above

$$\frac{1}{2\pi a^2}\sqrt{\frac{Eh^3}{12\gamma\left(1-\nu^2\right)}} = \frac{1}{2\pi(10.0)^2}\sqrt{\frac{\left(10.0 \cdot 10^6\right)(0.10)^3}{12\left(2.588 \cdot 10^{-5}\right)\left[1-(0.3)^2\right]}} = 9.467 \text{ cycles/sec.}$$

λ^2_{ij} is tabulated from the reference as follows:

Table 579: Values of λ^2_{ij}

Mode Number	λ^2_{ij}	Number of Nodal Diameters (i)	Number of Nodal Circles (j)
1	4.977	0	0
2	13.94	1	0
3	25.65	2	0
4	29.76	0	1
5[l]		3	0
6	48.51	1	1

[l] not tabulated in the reference

Comparison

Table 580: Comparison of results

Mode Number	Theoretical	STAAD.Pro	Difference
1	47.12	46.211	1.9%
2	132.0	130.541	1.1%
3	242.8	240.146	1.1%
4	281.7	283.028	0.5%
5 *		373.197	n/a*
6	459.3	466.579	1.6%

*The reference did not tabulate a value of for the fifth mode of the structure, hence a comparison with the theoretical value of this mode cannot be made.

All anti-symmetric mode shapes for the 360 degree circular plate were captured by the 180 degree model with a phase angle included in the calculation. Some of the difference between the theoretical and STAAD.Pro frequencies is attributed to the loss of mass due to the piecewise secant representation of the outer radius and, since this mass is about 1 percent lower than for a true circular plate, it is not surprising that the first few modes are lower than the theoretical solution.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Circular Plate.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* Natural Frequencies of a Circular Plate
UNIT INCHES POUND
JOINT COORDINATES
1 -6.39758e-07 -1 0; 2 0.258818 -0.965926 0; 3 0.5 -0.866026 0;
4 0.707107 -0.707107 0; 5 0.866025 -0.5 0; 6 0.965926 -0.258819 0;
7 1 0 0; 8 0.965926 0.258819 0; 9 0.866025 0.5 0;
10 0.707107 0.707107 0; 11 0.5 0.866026 0; 12 0.258818 0.965926 0;
13 -6.39758e-07 1 0; 14 -1.43562e-06 -2.244 0; 15 0.580789 -2.16754 0;
16 1.122 -1.94336 0; 17 1.58675 -1.58675 0; 18 1.94336 -1.122 0;
19 2.16754 -0.58079 0; 20 2.244 0 0; 21 2.16754 0.58079 0;
22 1.94336 1.122 0; 23 1.58675 1.58675 0; 24 1.122 1.94336 0;
25 0.580789 2.16754 0; 26 -1.43562e-06 2.244 0;
27 -2.23148e-06 -3.488 0; 28 0.902759 -3.36915 0; 29 1.744 -3.0207 0;
30 2.46639 -2.46639 0; 31 3.0207 -1.744 0; 32 3.36915 -0.902761 0;
33 3.488 0 0; 34 3.36915 0.902761 0; 35 3.0207 1.744 0;
36 2.46639 2.46639 0; 37 1.744 3.0207 0; 38 0.902759 3.36915 0;
39 -2.23148e-06 3.488 0; 40 -2.90386e-06 -4.539 0;
41 1.17478 -4.38434 0; 42 2.2695 -3.93089 0; 43 3.20956 -3.20956 0;
44 3.93089 -2.2695 0; 45 4.38434 -1.17478 0; 46 4.539 0 0;
47 4.38434 1.17478 0; 48 3.93089 2.2695 0; 49 3.20956 3.20956 0;
50 2.2695 3.93089 0; 51 1.17478 4.38434 0; 52 -2.90386e-06 4.539 0;
53 -3.77841e-06 -5.906 0; 54 1.52858 -5.70476 0; 55 2.953 -5.11475 0;
56 4.17617 -4.17617 0; 57 5.11475 -2.953 0; 58 5.70476 -1.52859 0;
59 5.906 0 0; 60 5.70476 1.52859 0; 61 5.11475 2.953 0;
62 4.17617 4.17617 0; 63 2.953 5.11475 0; 64 1.52858 5.70476 0;
65 -3.77841e-06 5.906 0; 66 -4.91654e-06 -7.685 0;
67 1.98902 -7.42314 0; 68 3.8425 -6.65541 0; 69 5.43411 -5.43412 0;
70 6.6554 -3.8425 0; 71 7.42314 -1.98903 0; 72 7.685 0 0;
73 7.42314 1.98903 0; 74 6.6554 3.8425 0; 75 5.43411 5.43412 0;
76 3.8425 6.65541 0; 77 1.98902 7.42314 0; 78 -4.91654e-06 7.685 0;
79 -6.39758e-06 -10 0; 80 2.58818 -9.65926 0; 81 5 -8.66026 0;
82 7.07107 -7.07107 0; 83 8.66025 -5 0; 84 9.65926 -2.58819 0;
85 10 0 0; 86 9.65926 2.58819 0; 87 8.66025 5 0; 88 7.07107 7.07107 0;
89 5 8.66026 0; 90 2.58818 9.65926 0; 91 -6.39758e-06 10 0; 1000 0 0 0;
ELEMENT INCIDENCES SHELL
1 1000 1 2; 2 1000 2 3; 3 1000 3 4; 4 1000 4 5; 5 1000 5 6; 6 1000 6 7;
7 1000 7 8; 8 1000 8 9; 9 1000 9 10; 10 1000 10 11; 11 1000 11 12;
12 1000 12 13; 13 1 14 15 2; 14 2 15 16 3; 15 3 16 17 4; 16 4 17 18 5;
17 5 18 19 6; 18 6 19 20 7; 19 7 20 21 8; 20 8 21 22 9; 21 9 22 23 10;
22 10 23 24 11; 23 11 24 25 12; 24 12 25 26 13; 25 14 27 28 15;
```

```
26 15 28 29 16; 27 16 29 30 17; 28 17 30 31 18; 29 18 31 32 19;
30 19 32 33 20; 31 20 33 34 21; 32 21 34 35 22; 33 22 35 36 23;
34 23 36 37 24; 35 24 37 38 25; 36 25 38 39 26; 37 27 40 41 28;
38 28 41 42 29; 39 29 42 43 30; 40 30 43 44 31; 41 31 44 45 32;
42 32 45 46 33; 43 33 46 47 34; 44 34 47 48 35; 45 35 48 49 36;
46 36 49 50 37; 47 37 50 51 38; 48 38 51 52 39; 49 40 53 54 41;
50 41 54 55 42; 51 42 55 56 43; 52 43 56 57 44; 53 44 57 58 45;
54 45 58 59 46; 55 46 59 60 47; 56 47 60 61 48; 57 48 61 62 49;
58 49 62 63 50; 59 50 63 64 51; 60 51 64 65 52; 61 53 66 67 54;
62 54 67 68 55; 63 55 68 69 56; 64 56 69 70 57; 65 57 70 71 58;
66 58 71 72 59; 67 59 72 73 60; 68 60 73 74 61; 69 61 74 75 62;
70 62 75 76 63; 71 63 76 77 64; 72 64 77 78 65; 73 66 79 80 67;
74 67 80 81 68; 75 68 81 82 69; 76 69 82 83 70; 77 70 83 84 71;
78 71 84 85 72; 79 72 85 86 73; 80 73 86 87 74; 81 74 87 88 75;
82 75 88 89 76; 83 76 89 90 77; 84 77 90 91 78;
ELEMENT PROPERTY
1 TO 84 THICKNESS 0.1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.1
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
* centre of circle
1000 FIXED BUT FZ MX
* Interior nodes
2 TO 12 15 TO 25 28 TO 38 41 TO 51 54 TO 64 67 TO 77 FIXED BUT FZ MX MY
* nodes along circumference
79 TO 91 FIXED BUT MX MY
* nodes along diameter except the two at the ends of the diameter.
1 13 14 26 27 39 40 52 53 65 66 78 FIXED BUT FZ MX
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       46.211                  0.02164
         2                      130.541                  0.00766
         3                      240.146                  0.00416
         4                      283.028                  0.00353
         5                      373.197                  0.00268
         6                      466.579                  0.00214
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE           X             Y             Z               WEIGHT
         1       0.000000E+00  0.000000E+00  1.053697E+00    4.537237E-01
         2       0.000000E+00  0.000000E+00  2.214294E-10    3.944628E-01
```

```
         3       0.000000E+00  0.000000E+00  1.884687E-07    3.935446E-01
         4       0.000000E+00  0.000000E+00  1.275807E-01    1.752589E-01
         5       0.000000E+00  0.000000E+00  1.060135E-08    4.222042E-01
         6       0.000000E+00  0.000000E+00  7.897355E-10    2.165119E-01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00   0.00  86.78    0.000    0.000   86.780
             2     0.00   0.00   0.00    0.000    0.000   86.780
             3     0.00   0.00   0.00    0.000    0.000   86.780
             4     0.00   0.00  10.51    0.000    0.000   97.287
             5     0.00   0.00   0.00    0.000    0.000   97.287
             6     0.00   0.00   0.00    0.000    0.000   97.287
```

V. Modal Response of a Rectangular Plate

Find the natural frequencies of a rectangular plate and compare theoretical answers to the STAAD.Pro solution.

Reference

Blevins, Robert D., *Formulas for Natural Frequency and Mode Shape*, Van Nostrand Reinhold Company, 1979, page 258.

Problem

A flat rectangular plate is simply supported on all four sides. The first six modes and their associated natural frequencies are to be computed for this structure using the subspace iteration method offered by STAAD.Pro. This problem also demonstrates that the mesh refinement can be chosen to accurately calculate modes of interest based on the expected mode shapes.

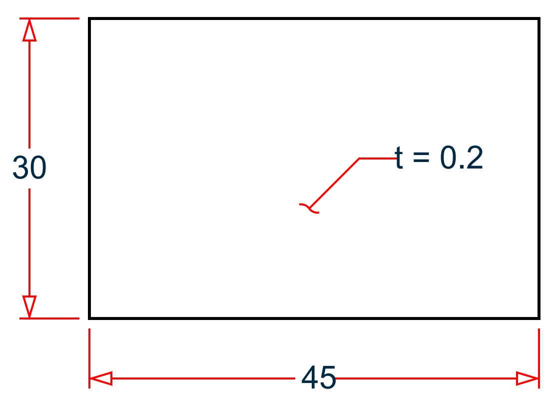


Figure 487: Simply supported, rectangular plate

L = 45 in., W = 30 in., t = 0.2 in.

A plate with an aspect ratio of 1.5 was used so that comparison could be made with theoretical results tabulated for plates in the reference. An equally spaced mesh was utilized in both the x and the y dimensions of the plate. The number of elements in each dimension was determined on the basis of the highest mode of interest. Since the number of half-waves in the sixth mode is 3 in the length dimension and 2 in the width dimension, a node spacing of 3.75 inches results in each half- wave being represented by four elements which means that no element will be expected to deform in double curvature. The simply supported edge condition requires that translation normal to the plane of the plate be restrained for these edge nodes. Rotations normal to the plate were restrained for all nodes.

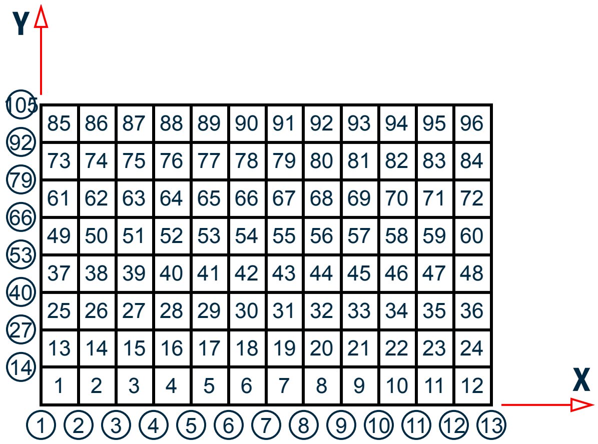


Figure 488: Model: finite element meshed

Theoretical Calculations

From the reference case 16 in Table 11-4, the first six natural frequencies of the plate are described by the following equations:

dimensionless parameter associated with the mode indices i, j

$$f_{ij} = \frac{\lambda_{ij}^2}{2\pi a^2}\sqrt{\frac{Eh^3}{12\gamma\left(1 - \nu^2\right)}}$$

where

i	=	number of half-waves in this mode shape along the horizontal axis
j	=	number of half-waves in this mode shape along the vertical axis
ν	=	Poisson’s ratio
E	=	elastic modulus
h	=	plate thickness
γ	=	mass of material per unit area
a	=	length of plate
b	=	width of plate

The numerical values used for this example are:

$\nu = 0.30$
$E = 30.0\times10^6$ psi
$h = 0.2$ inches

$$\gamma = \frac{\left(0.282\ \mathrm{lb}_m / \mathrm{in}^3\right)\left(0.20\ \mathrm{in}\right)}{\left(386.4\ \mathrm{lb}_m \cdot \mathrm{in/lb}_f \cdot \mathrm{s}^2\right)} = 0.146 \times 10^{-4}\ \mathrm{in/lb}_f \cdot \mathrm{s}^2 \Big/ \mathrm{in}^3$$

$a = 45.0\ in$
$b = 30.0\ in$

with the numerical values used above

$$\frac{1}{2\pi a^2}\sqrt{\frac{Eh^3}{12\gamma\left(1-\nu^2\right)}} = \frac{1}{2\pi(10.0)^2}\sqrt{\frac{\left(30.0\cdot 10^6\right)(0.20)^3}{12\left(1.460\cdot 10^{-4}\right)\left[1-(0.3)^2\right]}} = 0.9644 \text{ cycles/sec.}$$

$\lambda^2{}_{ij}$ is tabulated from the reference as follows:

Table 581: Values of $\lambda^2{}_{ij}$

Mode Number	$\lambda^2{}_{ij}$	Number of Half-Waves in Length (i)	Number of Half-Waves in Width (j)
1	32.08	1	1
2	61.69	2	1
3	98.70	1	2
4	111.0	3	1
5	128.3	2	2
6	177.7	3	2

Comparison

Table 582: Comparison of results

Mode Number	Theoretical	STAAD.Pro	Difference
1	30.94	30.599	1.1%
2	59.49	58.724	1.3%
3	95.18	95.063	negligible
4	107.1	106.277	0.8%
5	123.7	122.092	1.3%
6	171.3	168.009	1.9%

As noted earlier, the node spacing was based on the highest mode of interest. It follows that the difference between the theoretical and STAAD.Pro frequencies generally increases with increasing mode sequence.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Modal Response of a Rectangular Plate.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
* Natural frequencies of a rectangular plate
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 3.75 0 0; 3 7.5 0 0; 4 11.25 0 0; 5 15 0 0; 6 18.75 0 0;
7 22.5 0 0; 8 26.25 0 0; 9 30 0 0; 10 33.75 0 0; 11 37.5 0 0;
12 41.25 0 0; 13 45 0 0; 14 0 3.75 0; 15 3.75 3.75 0; 16 7.5 3.75 0;
17 11.25 3.75 0; 18 15 3.75 0; 19 18.75 3.75 0; 20 22.5 3.75 0;
21 26.25 3.75 0; 22 30 3.75 0; 23 33.75 3.75 0; 24 37.5 3.75 0;
25 41.25 3.75 0; 26 45 3.75 0; 27 0 7.5 0; 28 3.75 7.5 0; 29 7.5 7.5 0;
30 11.25 7.5 0; 31 15 7.5 0; 32 18.75 7.5 0; 33 22.5 7.5 0;
34 26.25 7.5 0; 35 30 7.5 0; 36 33.75 7.5 0; 37 37.5 7.5 0;
38 41.25 7.5 0; 39 45 7.5 0; 40 0 11.25 0; 41 3.75 11.25 0;
42 7.5 11.25 0; 43 11.25 11.25 0; 44 15 11.25 0; 45 18.75 11.25 0;
46 22.5 11.25 0; 47 26.25 11.25 0; 48 30 11.25 0; 49 33.75 11.25 0;
50 37.5 11.25 0; 51 41.25 11.25 0; 52 45 11.25 0; 53 0 15 0;
54 3.75 15 0; 55 7.5 15 0; 56 11.25 15 0; 57 15 15 0; 58 18.75 15 0;
59 22.5 15 0; 60 26.25 15 0; 61 30 15 0; 62 33.75 15 0; 63 37.5 15 0;
64 41.25 15 0; 65 45 15 0; 66 0 18.75 0; 67 3.75 18.75 0;
68 7.5 18.75 0; 69 11.25 18.75 0; 70 15 18.75 0; 71 18.75 18.75 0;
72 22.5 18.75 0; 73 26.25 18.75 0; 74 30 18.75 0; 75 33.75 18.75 0;
76 37.5 18.75 0; 77 41.25 18.75 0; 78 45 18.75 0; 79 0 22.5 0;
80 3.75 22.5 0; 81 7.5 22.5 0; 82 11.25 22.5 0; 83 15 22.5 0;
84 18.75 22.5 0; 85 22.5 22.5 0; 86 26.25 22.5 0; 87 30 22.5 0;
88 33.75 22.5 0; 89 37.5 22.5 0; 90 41.25 22.5 0; 91 45 22.5 0;
92 0 26.25 0; 93 3.75 26.25 0; 94 7.5 26.25 0; 95 11.25 26.25 0;
96 15 26.25 0; 97 18.75 26.25 0; 98 22.5 26.25 0; 99 26.25 26.25 0;
100 30 26.25 0; 101 33.75 26.25 0; 102 37.5 26.25 0; 103 41.25 26.25 0;
104 45 26.25 0; 105 0 30 0; 106 3.75 30 0; 107 7.5 30 0; 108 11.25 30 0;
109 15 30 0; 110 18.75 30 0; 111 22.5 30 0; 112 26.25 30 0; 113 30 30 0;
114 33.75 30 0; 115 37.5 30 0; 116 41.25 30 0; 117 45 30 0;
ELEMENT INCIDENCES SHELL
1 1 2 15 14; 2 2 3 16 15; 3 3 4 17 16; 4 4 5 18 17; 5 5 6 19 18;
6 6 7 20 19; 7 7 8 21 20; 8 8 9 22 21; 9 9 10 23 22; 10 10 11 24 23;
11 11 12 25 24; 12 12 13 26 25; 13 14 15 28 27; 14 15 16 29 28;
15 16 17 30 29; 16 17 18 31 30; 17 18 19 32 31; 18 19 20 33 32;
19 20 21 34 33; 20 21 22 35 34; 21 22 23 36 35; 22 23 24 37 36;
23 24 25 38 37; 24 25 26 39 38; 25 27 28 41 40; 26 28 29 42 41;
27 29 30 43 42; 28 30 31 44 43; 29 31 32 45 44; 30 32 33 46 45;
31 33 34 47 46; 32 34 35 48 47; 33 35 36 49 48; 34 36 37 50 49;
35 37 38 51 50; 36 38 39 52 51; 37 40 41 54 53; 38 41 42 55 54;
39 42 43 56 55; 40 43 44 57 56; 41 44 45 58 57; 42 45 46 59 58;
43 46 47 60 59; 44 47 48 61 60; 45 48 49 62 61; 46 49 50 63 62;
47 50 51 64 63; 48 51 52 65 64; 49 53 54 67 66; 50 54 55 68 67;
51 55 56 69 68; 52 56 57 70 69; 53 57 58 71 70; 54 58 59 72 71;
55 59 60 73 72; 56 60 61 74 73; 57 61 62 75 74; 58 62 63 76 75;
59 63 64 77 76; 60 64 65 78 77; 61 66 67 80 79; 62 67 68 81 80;
63 68 69 82 81; 64 69 70 83 82; 65 70 71 84 83; 66 71 72 85 84;
67 72 73 86 85; 68 73 74 87 86; 69 74 75 88 87; 70 75 76 89 88;
71 76 77 90 89; 72 77 78 91 90; 73 79 80 93 92; 74 80 81 94 93;
75 81 82 95 94; 76 82 83 96 95; 77 83 84 97 96; 78 84 85 98 97;
79 85 86 99 98; 80 86 87 100 99; 81 87 88 101 100; 82 88 89 102 101;
83 89 90 103 102; 84 90 91 104 103; 85 92 93 106 105; 86 93 94 107 106;
87 94 95 108 107; 88 95 96 109 108; 89 96 97 110 109; 90 97 98 111 110;
91 98 99 112 111; 92 99 100 113 112; 93 100 101 114 113;
94 101 102 115 114; 95 102 103 116 115; 96 103 104 117 116;
ELEMENT PROPERTY
```

```
1 TO 96 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
DENSITY 0.282
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 6
CUT OFF FREQUENCY 1000
SUPPORTS
* Corner nodes
1 13 105 117 FIXED BUT MX MY
* Nodes along y=0 and y=30
2 TO 12 106 TO 116 FIXED BUT MX MY
* Nodes along x=0
14 27 40 53 66 79 92 FIXED BUT MX MY
* Nodes along x=45
26 39 52 65 78 91 104 FIXED BUT MX MY
* Interior nodes
15 TO 25 28 TO 38 41 TO 51 54 TO 64 67 TO 77 80 TO 90 93 TO 102 -
103 FIXED BUT FZ MX MY
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                      30.599                 0.03268
         2                      58.724                 0.01703
         3                      95.063                 0.01052
         4                     106.277                 0.00941
         5                     122.092                 0.00819
         6                     168.009                 0.00595
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       0.000000E+00  0.000000E+00  4.833474E+01    1.915440E+01
         2       0.000000E+00  0.000000E+00  9.476868E-17    1.920213E+01
         3       0.000000E+00  0.000000E+00  4.308438E-14    1.925532E+01
         4       0.000000E+00  0.000000E+00  4.837131E+00    1.927455E+01
         5       0.000000E+00  0.000000E+00  3.971700E-13    1.927602E+01
         6       0.000000E+00  0.000000E+00  2.329330E-10    1.936312E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00   0.00  79.15    0.000    0.000   79.146
             2     0.00   0.00   0.00    0.000    0.000   79.146
             3     0.00   0.00   0.00    0.000    0.000   79.146
```

```
              4       0.00    0.00    7.92     0.000     0.000    87.066
              5       0.00    0.00    0.00     0.000     0.000    87.066
              6       0.00    0.00    0.00     0.000     0.000    87.066
```

V. Natural Frequency of a 2D Truss

To calculate the Natural frequency of vibration for a two story truss.

Reference

Hand calculation using known formula.

Problem

Find the natural frequency of vibration, *f*, of the truss.

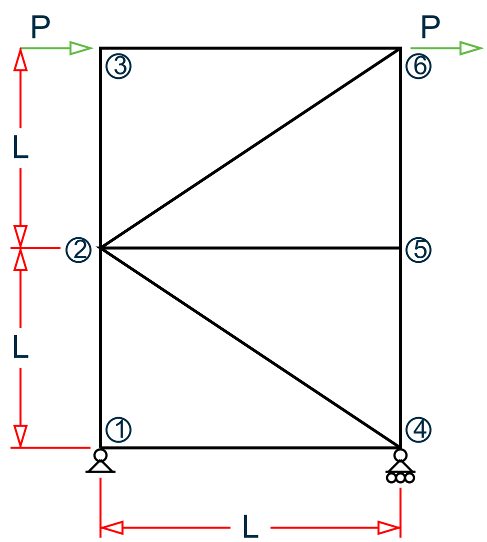


Figure 489: Space truss

E = 30,000 ksi

L = 10 ft

P = 10 kips

All members are L4x4x5/16

Hand Calculation

Calculate average deflection at top:

$$\delta_{avg} = (0.45255 \text{ in} + 0.43535 \text{ in})/2 = 0.444 \text{ in}$$

Stiffness of truss:

$$k = P/\delta_{avg} = 20.0 \text{ kip}/0.444 \text{ in} = 45.5 \text{ k/in}$$

Mass:

$$M = w/g = 20.0/386.4 = 0.05175 \text{ k-sec}^2\text{/in}$$

Frequency:

$$f = \frac{\sqrt{k / m}}{2\pi} = \frac{\sqrt{45.05 / 0.05175}}{2\pi} = 4.696\text{Hz}$$

Comparison

Table 583: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Frequency, *f* (Hz)	4.696	4.687	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of a 2D Truss.STD` is typically installed with the program.

```
STAAD TRUSS : A TWO STORY TRUSS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 0 20 0; 4 10 0 0; 5 10 10 0; 6 10 20 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 5; 4 5 6; 5 1 4; 6 2 5; 7 3 6; 8 2 4; 9 2 6;
MEMBER PROPERTY AMERICAN
1 TO 9 TABLE ST L40405
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
4 FIXED BUT FX FZ MX MY MZ
CUT OFF MODE SHAPE 0
LOAD 1
JOINT LOAD
3 6 FX 10
MODAL CALCULATION REQUESTED
```

```
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 3 6
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       4.687                 0.21337
         2                      34.019                 0.02940
            MODAL WEIGHT (MODAL MASS TIMES g) IN KIP          GENERALIZED
      MODE            X             Y             Z              WEIGHT
         1       1.999221E+01  0.000000E+00  0.000000E+00    1.924051E+01
         2       7.793822E-03  0.000000E+00  0.000000E+00    1.924051E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1    99.96   0.00   0.00   99.961    0.000    0.000
             2     0.04   0.00   0.00  100.000    0.000    0.000
    33. PRINT JOINT DISPLACEMENTS LIST 3 6
  JOINT    DISPLACE LIST     3
      : A TWO STORY TRUSS                                      -- PAGE NO.
4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = TRUSS
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      3    1    0.45368   0.06897   0.00000   0.00000   0.00000   0.00000
      6    1    0.43644  -0.06897   0.00000   0.00000   0.00000   0.00000
```

V. Natural Frequency of a Simply Supported Beam

To find the fundamental frequency of vibration for a simply supported beam with a uniform mass.

Reference

Thomson, W.T., *Vibration Theory and Applications*, Prentice-Hall, Inc., 1965. Also compared with ANSYS® Finite Element Software.

Problem

Find the fundamental frequency, f, of a simply supported beam of uniform cross-section.

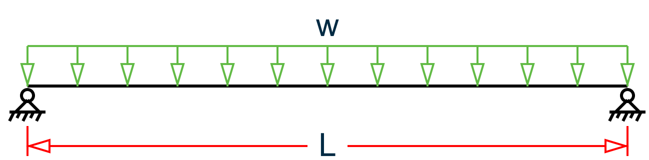


Figure 490: Model for dynamic beam no. 3

E = 30,000 ksi

$I = 1.3333\ in^4$

$A = 4\ in^2$

$w = 1.124\ lb/in$

$L = 80\ in$

Comparison

Table 584: Comparison of results

Result Type	Theory	ANSYS®	STAAD.Pro	Difference
Frequency, f (Hz)	28.766	28.767	28.7438 (Rayleigh)	none
			28.761 (Eigensolution)	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of a Simply Supported Beam.STD` is typically installed with the program.

```
STAAD PLANE :FUNDAMENTAL FREQUENCY OF A SIMPLY SUPPORTED BEAM
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE:  THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS",
*      PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 20 0 0; 3 40 0 0; 4 60 0 0; 5 80 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5;
MEMBER PROPERTY AMERICAN
1 TO 4 PRIS AX 4 IZ 1.33333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 5 PINNED
CUT OFF FREQUENCY 100
CUT OFF MODE SHAPE 3
LOAD 1 DYNAMIC ANALYSIS (NATURAL FREQUENCY)
MEMBER LOAD
1 TO 4 UNI GY 1.124
```

```
CALCULATE RAYLEIGH FREQUENCY
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                      28.761                  0.03477
         2                     114.242                  0.00875
         3                     242.560                  0.00412
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1       0.000000E+00  6.551152E+01  0.000000E+00    4.496000E+01
         2       0.000000E+00  1.870800E-31  0.000000E+00    4.496000E+01
         3       0.000000E+00  1.928479E+00  0.000000E+00    4.496000E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00  97.14   0.00    0.000   97.140    0.000
             2     0.00   0.00   0.00    0.000   97.140    0.000
             3     0.00   2.86   0.00    0.000  100.000    0.000
   **********************************************************
   *                                                        *
   * RAYLEIGH FREQUENCY FOR LOADING     1 =   28.74379 CPS  *
   * MAX DEFLECTION =  0.01499 INCH GLO Y,  AT JOINT     3  *
   *                                                        *
   **********************************************************
```

V. Natural Frequency of Beam on Springs

Find the period of free vibration for a beam supported on two springs with a point mass.

Reference

Timoshenko, S., Young, D., and Weaver, W., *Vibration Problems in Engineering*, John Wiley & Sons, 4th edition, 1974. page 11, problem 1.1-3.

Problem

A simple beam is supported by two spring as shown in the figure. Neglecting the distributed mass of the beam, calculate the period of free vibration of the beam subjected to a load of W.

EI = 30,000.0 ksi

A = 7.0 ft

B = 3.0 ft.

W = 1,000 lbfK = 300.0 lb/in.

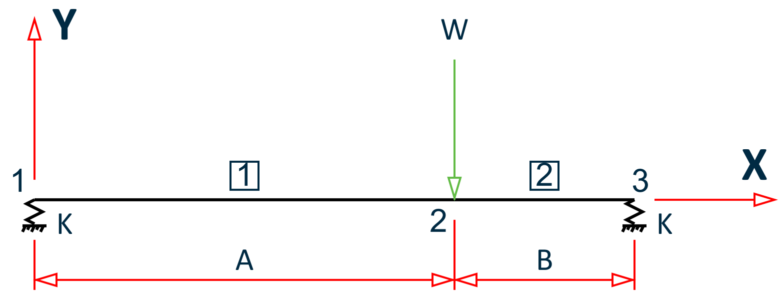


Figure 491: Beam supported on springs

Comparison

Table 585: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Period (sec)	0.533	0.53317	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Natural Frequency of Beam on Springs.STD` is typically installed with the program.

```
STAAD PLANE : NATURAL FREQUENCY OF BEAM ON SPRINGS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*   REFERENCE 'VIBRATION PROBLEMS IN ENGINEERING' BY
*   TIMOSHENKO,YOUNG,WEAVER. (4TH EDITION, PAGE 11, PROB 1.1-3)
*   THE ANSWER IN THE BOOK IS T = 0.533 sec., viz., F = 1.876 CPS
*
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 7 0 0; 3 10 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
UNIT INCHES POUND
SUPPORTS
1 3 FIXED BUT MZ KFY 300
MEMBER PROPERTY AMERICAN
1 2 PRIS AX 1 IZ 1
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 1
```

```
LOAD 1 1000 LB LOAD AT JOINT 2
JOINT LOAD
2 FY -1000
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       1.876                  0.53317
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN         GENERALIZED
      MODE           X              Y              Z              WEIGHT
         1       0.000000E+00  9.999999E+02  0.000000E+00    9.999999E+02
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1     0.00 100.00   0.00    0.000  100.000    0.000
```

V. Rayleigh Natural Frequency of a Cantilever Beam

To calculate the Natural frequency of vibration using the Rayleigh method for a light cantilever beam with a mass at the free end.

Reference

Thomson, W.T., *Vibration Theory and Applications*, Prentice-Hall, Inc., 1965.

Problem

Find the natural frequency of vibration, f, of a mass, m, attached to the end of a light cantilever beam of length, L, and flexural stiffness, EI.

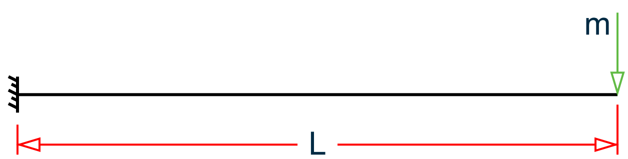


Figure 492: Model for dynamic beam no. 2

E = 30,000 ksi

I = 1.3333 in^4

m = 0.1 lb-sec^2/in

L = 30 in

Comparison

Table 586: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Frequency, f (Hz)	33.553	33.5365	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Rayleigh Natural Frequency of a Cantilever Beam.STD` is typically installed with the program.

```
STAAD PLANE NATURAL FREQUENCY OF A CANTILEVERED MASS
START JOB INFORMATION
ENGINEER DATE 14-Sep-18
END JOB INFORMATION
*
*  REFERENCE:  THOMSON, W.T., "VIBRATION THEORY AND APPLICATIONS",
*      PRENTICE HALL INC., ENGLEWOODS, NEW JERSEY, 1965
*
INPUT WIDTH 72
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 PRIS IZ 1.33333
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3e+07
POISSON 0.290909
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1 NATURAL FREQUENCY
JOINT LOAD
2 FY -38.64
CALCULATE RAYLEIGH FREQUENCY
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
FINISH
```

STAAD Output

```
   ********************************************************
   *                                                      *
```

```
 * RAYLEIGH FREQUENCY FOR LOADING     1 =   33.53649 CPS  *
 * MAX DEFLECTION =  0.00870 INCH GLO Y,  AT JOINT     2  *
 *                                                        *
 **********************************************************
```

V. Steady State Loading on a Beam

Calculate the deflections at two points along the beam at steady-state condition.

Reference

1. Blevins, R. D., *Formulas for natural Frequency and Mode Shape*, Van Nostrand Reinhold, 1979, pp. 108, 455-486.
2. Warburton, G. B., *The Dynamical Behavior of Structures*, Pergamon Press, 1964, pp. 10-15, 85, 86.

Problem

Determine the steady-state displacements of the quarter and mid-span points of a fixed-fixed beam subjected to a parabolically varying distributed load operating at a 7.5 Hz frequency.

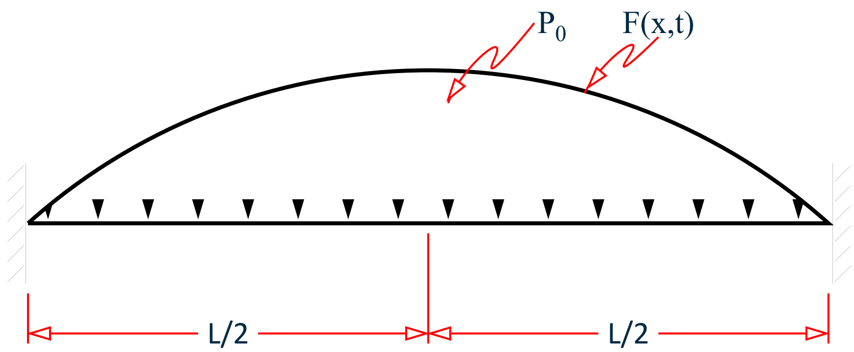


Figure 493: Beam with harmonic distributed load

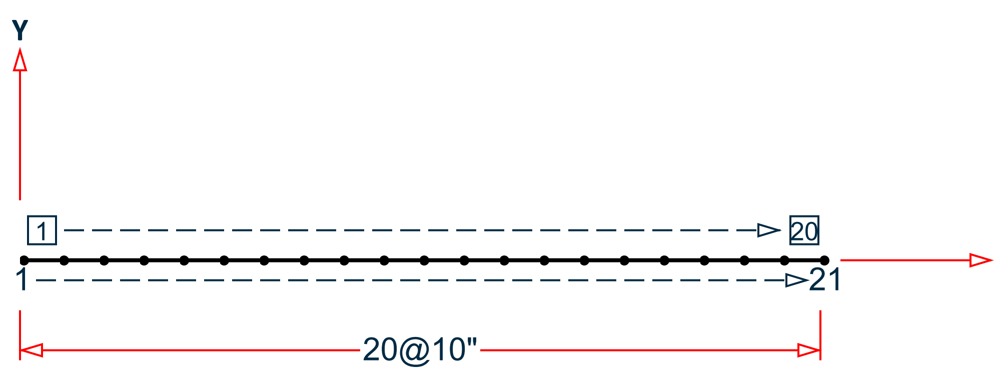


Figure 494: Model: divide span 20@10.0”

$E = 10.0\text{x}10^6$ psi

L = 200 inches

I = 2/3 in^4

A = 2 in^2

P_o = 0.1 lbf/in^3

g = 386.4 in/sec^2

The DYNRE2 run utilizes all 19 modes calculated by STAAD (Steady State analysis) for which a value of $1.0(10)^{-10}$ times critical damping is assigned. A single forcing frequency equal to 7.5 Hz is specified for the distributed load. This load is distributed to the nodes by calculating the total integrated load for each beam and lumping one-half of this force to the respective i and j nodes.

$$F_i = F_j = \int_{x_i}^{x_j} \frac{P(x)}{2}\, d\, x = \frac{1}{2}\int_{x_i}^{x_j} \frac{4}{l^2}\left(xl - x^2\right) d\, x$$

$$F_i = F_j = \left| \frac{x^2}{l} - \frac{2x^3}{3l^2} \right|_{x_i}^{x_j} = \frac{x_j^2 - x_i^2}{200} - \frac{x_j^3 - x_i^3}{60,\ 000}$$

Theoretical Solution

The theoretical solution for this example is taken from:

1. Blevins, R. D., “Formulas for natural Frequency and Mode Shape,” Van Nostrand Reinhold, 1979, pp 108, 455-486.
2. Warburton, G. B., “The Dynamical Behavior of Structures,” Pergamon Press, 1964, pp. 10-15, 85, 86.

The natural frequencies of the system are calculated using the equations from reference 1 page 108 and reference 2 page 85.

$$f_i = \frac{\lambda_1^2}{2\pi l^2}\sqrt{\frac{EI}{m}}$$

where

$$m = \rho A/g$$

$$f_i = \frac{\lambda_i^2}{2\pi(200)^2}\sqrt{\frac{1.0(10)^6\left(2/3\right)}{0.10(2.0/386.4)}}$$

$$= 4.5156271*10^{-1}\lambda_i^{\ 2}$$

$\lambda_i$satisfies the characteristic equation:

$$\cos\lambda \cosh\lambda - 1 = 0$$

Table 587: Calculated natural frequency for each mode

i	λ_i	ω_i	f_i
1	4.730041	63.47865	10.10294
2	7.853205	174.9814	27.84915
3	10.99561	343.0334	54.59546

i	λ_i	ω_i	f_i
4	14.13717	567.0517	90.24907
5	17.27876	847.0773	134.8165
6	20.42035	1183.108	188.2975
7	23.56194	1575.144	250.6919
8	26.70354	2023.185	321.9998

The mode shapes are:

$$\phi_i = \cosh\frac{\lambda_i x}{l} - \cos\frac{\lambda_i x}{l} - \sigma_i\left(\sinh\frac{\lambda_i x}{l} - \sin\frac{\lambda_i x}{l}\right)$$

Where:

$$\sigma_i = \frac{\cosh\lambda_i - \cos\lambda_i}{\sinh\lambda_i - \sin\lambda_i}$$

The response of mode i to a harmonic force:

$$\eta_i = \frac{\int_l \phi_i(x)P(x)\,d\,x}{\omega_i^2 \int\{\phi_i(x)\}^2 m\,d\,x} - \frac{\sin(\omega t - \psi_i)}{\sqrt{\left(1 - \frac{\omega^2}{\omega_i^2}\right)^2 + \left(\frac{c\omega}{k}\right)^2}}$$

Where ψ_i is the response phase lag relative to the applied force and c is the damping.

Since c = 0.0, $\psi_i = 0.0$

Upon substitution and rearranging terms:

$$\eta_i = \frac{\int_l \frac{4P_o}{l^2}\left(xl - x^2\right)\phi_i\left(x\right)\,d\,x}{\frac{\rho A}{g}\left(\omega_i^2 - \omega^2\right)\int_l\{\phi_i(x)\}^2\,d\,x}\sin\omega t$$

From reference 1, page 466, case c and page 467, case 29:

$$\frac{4P_o}{l^2}\int_l x\phi_i(x)\,d\,x - \frac{4P_o}{l^2}\int_l x^2\phi_i\left(x\right)\,d\,x = \frac{8P_0}{\beta_i^2 l}\left[1 + (-i)^i - (-i)^i\sigma_i\beta_i l\right] - \frac{8(-l)^i P_0}{\beta_i^2 l}\left[2 - \sigma_i\beta_i l\right]$$

Since the load is symmetric, this expression is zero for I = 2, 4, 6 ...;

Therefore, i = 1,3,5……, and: $(-1)^i = -1$

And from the reference, $\beta_i = \lambda_i / l$

So:

$$\int_l \phi_i(x)P(x)\,d\,x = \frac{16P_o l}{\lambda_i^2}$$

From Reference 1, page 457 case 5:

$$\int_l \{\phi_i(x)\}^2 \, d \; x = 1$$

Therefore:

$$\omega = 7.5(2\pi) = 47.1239 \text{ radians/sec}$$

$$\eta_i = \frac{16P_o}{m\lambda_i^2\left(\omega_i^2 - \omega^2\right)} \sin \omega t = \frac{16(-1.0)\sin \omega t}{\frac{0.10(2)}{386.4}\lambda_i^2\left(\omega_i^2 - 2,\ 220.661\right)} = \frac{30,\ 912 \cdot \sin \omega t}{\lambda_i^2\left(\omega_i^2 - 2,\ 220.661\right)}$$

Table 588: Mode shapes

i	λ_i	ω_i	$\eta_i(t)$	σ_i	φ(1/4)	φ(1/2)
1	4.730041	63.47865	$-7.63815(10)^{-1} \sin \omega t$	0.9825022	0.8631319	1.5881463
3	10.99561	343.0334	$-2.21457(10)^{-3} \sin \omega t$	0.9999664	1.3708047	-1.4059984
5	17.27876	847.0773	$-1.44744(10)^{-4} \sin \omega t$	0.9999999	-0.5278897	1.4145675
7	23.56194	1575.144	$-2.24623(10)^{-5} \sin \omega t$	1.0000000	-1.3037973	-1.4141982

$$y\left(x,\ t\right) = \sum_{i=1,3,5,7} \eta_i(t)\phi_i(x)$$

Table 589: Steady state displacements for 1/4 point and 1/2 point (nodes 6 and 11, respectively)

i	φ(1/4)η_i(t)	φ(1/2)η_i(t)
1	-0.6592727	-1.2130493
3	-0.0030357	0.0031137
5	0.0000764	-0.0002048
7	0.0000293	-0.0000318
Summation	-0.6622028	-1.2101086

Comparison

Table 590: Comparison of results

Location	Theory	STAAD Advanced Analysis
Node 6 (X = 50 inches)	0.66220	0.65963
Node 11 (X = 100 inches)	1.21011	1.20545

Steady-state analysis requires the STAAD.Pro Advanced Analysis Plus license.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Steady State Loading on a Beam.STD` is included in the STAAD.Pro installation folder.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Mar-06
END JOB INFORMATION
* FIXED BEAM SUBJECTED TO A HARMONIC LOAD WITH A PARABOLIC DISTRIBUTION X
*                    NUMBER OF NODES             21                     X
*                    HIGH NODE NUMBER            21                     X
*                    NODES FULLY RESTRAINED       2                     X
*                    NUMBER OF BEAM ELEMENTS     20                     X
*                    NUMBER OF EIGENVECTORS      19
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 20 0 0; 4 30 0 0; 5 40 0 0; 6 50 0 0; 7 60 0 0;
8 70 0 0; 9 80 0 0; 10 90 0 0; 11 100 0 0; 12 110 0 0; 13 120 0 0;
14 130 0 0; 15 140 0 0; 16 150 0 0; 17 160 0 0; 18 170 0 0; 19 180 0 0;
20 190 0 0; 21 200 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 9 10;
10 10 11; 11 11 12; 12 12 13; 13 13 14; 14 14 15; 15 15 16; 16 16 17;
17 17 18; 18 18 19; 19 19 20; 20 20 21;
MEMBER PROPERTY AMERICAN
1 TO 20 PRIS AX 2 AY 0 AZ 0 IX 0.001 IY 0.666667 IZ 0.166667
SUPPORTS
2 TO 20 FIXED BUT FY MZ
1 21 FIXED
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 1e+07
POISSON 0.3
DENSITY 0.0999194
END DEFINE MATERIAL
CONSTANTS
BETA 90 ALL
MATERIAL MATERIAL1 ALL
CUT OFF MODE SHAPE 7
CUT OFF FREQUENCY 500
LOAD 1
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
MODAL CALCULATION REQUESTED
PERFORM STEADY STATE ANALYSIS
BEGIN STEADY FORCE
STEADY FORCE FREQ 7.5 DAMP 1e-10
JOINT LOAD
2 FY 1.8666
3 FY 3.5666
4 FY 5.0666
5 FY 6.3666
6 FY 7.4666
```

```
7 FY 8.3666
8 FY 9.0666
9 FY 9.5666
10 FY 9.8666
11 FY 9.9666
12 FY 9.8666
13 FY 9.5666
14 FY 9.0666
15 FY 8.3666
16 FY 7.4666
17 FY 6.3666
18 FY 5.0666
19 FY 3.5666
20 FY 1.8666
END
PRINT JOINT DISPLACEMENTS LIST 6 11
FINISH
```

STAAD Output

```
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         21  NUMBER OF MEMBERS      20
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     21
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      38
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   ***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM
            THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
 EIGEN METHOD   : SUBSPACE
 -------------------------
 NUMBER OF MODES REQUESTED             =     7
 NUMBER OF EXISTING MASSES IN THE MODEL =    19
 NUMBER OF MODES THAT WILL BE USED      =     7
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      STAAD SPACE                                               -- PAGE NO.
3
               CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                      10.103                   0.09898
         2                      27.849                   0.03591
         3                      54.591                   0.01832
         4                      90.230                   0.01108
         5                     134.747                   0.00742
         6                     188.093                   0.00532
         7                     250.166                   0.00400
            MODAL WEIGHT (MODAL MASS TIMES g) IN POUN          GENERALIZED
      MODE           X             Y             Z               WEIGHT
         1       0.000000E+00  2.759074E+01  0.000000E+00    1.584634E+01
         2       0.000000E+00  1.241542E-21  0.000000E+00    1.763389E+01
         3       0.000000E+00  5.287477E+00  0.000000E+00    1.757978E+01
         4       0.000000E+00  5.783031E-23  0.000000E+00    1.788088E+01
         5       0.000000E+00  2.138447E+00  0.000000E+00    1.901368E+01
         6       0.000000E+00  1.058584E-19  0.000000E+00    1.837854E+01
```

```
         7       0.000000E+00  1.145139E+00  0.000000E+00    1.756698E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
          MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
            1     0.00  72.67   0.00    0.000   72.666    0.000
            2     0.00   0.00   0.00    0.000   72.666    0.000
            3     0.00  13.93   0.00    0.000   86.591    0.000
            4     0.00   0.00   0.00    0.000   86.591    0.000
            5     0.00   5.63   0.00    0.000   92.223    0.000
            6     0.00   0.00   0.00    0.000   92.223    0.000
            7     0.00   3.02   0.00    0.000   95.239    0.000
    44. BEGIN STEADY FORCE
    45. STEADY FORCE FREQ 7.5 DAMP 1E-10
    46. JOINT LOAD
      STAAD SPACE                                             -- PAGE NO.
4
    47. 2 FY 1.8666
    48. 3 FY 3.5666
    49. 4 FY 5.0666
    50. 5 FY 6.3666
    51. 6 FY 7.4666
    52. 7 FY 8.3666
    53. 8 FY 9.0666
    54. 9 FY 9.5666
    55. 10 FY 9.8666
    56. 11 FY 9.9666
    57. 12 FY 9.8666
    58. 13 FY 9.5666
    59. 14 FY 9.0666
    60. 15 FY 8.3666
    61. 16 FY 7.4666
    62. 17 FY 6.3666
    63. 18 FY 5.0666
    64. 19 FY 3.5666
    65. 20 FY 1.8666
    66. END
*DIRECTIONS FOR WHICH AMPLITUDE VS. FREQUENCY DATA WAS ENTERED =    0   2
0   0   0   0
*DIRECTIONS FOR WHICH AMPLITUDE VS. PHASE LAG DATA WAS ENTERED =    0   0
0   0   0   0
               FORCE  DIRECTION NUMBER    2
              FREQUENCY      AMPLITUDE      PHASE ANGLE
         1   0.749800E+01   0.100000E+01   0.000000E+00
         2   0.750200E+01   0.100000E+01   0.000000E+00
      STAAD SPACE                                             -- PAGE NO.
5
                        7 MODES (EIGENVECTORS) HAVE BEEN SELECTED.
 MODE       NATURAL FREQUENCY         GENERALIZED WEIGHT        DAMPING
DAMPED FREQUENCY
 NO.       (HZ)        (RAD/SEC)     (WEIGHT)       (MASS)
COEFFICIENT       (HZ)
    1  1.010292E+01  6.347852E+01  1.584634E+01  4.104327E-02  1.000000E-10
1.010292E+01
    2  2.784865E+01  1.749782E+02  1.763389E+01  4.567317E-02  1.000000E-10
2.784865E+01
    3  5.459144E+01  3.430081E+02  1.757978E+01  4.553302E-02  1.000000E-10
5.459144E+01
```

```
    4  9.022966E+01  5.669297E+02  1.788088E+01  4.631289E-02  1.000000E-10
9.022966E+01
    5  1.347470E+02  8.466405E+02  1.901368E+01  4.924693E-02  1.000000E-10
1.347470E+02
    6  1.880927E+02  1.181821E+03  1.837854E+01  4.760188E-02  1.000000E-10
1.880927E+02
    7  2.501660E+02  1.571839E+03  1.756698E+01  4.549988E-02  1.000000E-10
2.501660E+02
                     PARTICIPATION FACTORS FOR EACH MODE
   MODE NO.       X             Y             Z             MX
MY            MZ
         1  0.000000E+00  0.218577E+04  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         2  0.000000E+00 -0.139214E-07  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         3  0.000000E+00  0.382113E+03  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         4  0.000000E+00 -0.416926E-08  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         5  0.000000E+00 -0.148269E+03  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         6  0.000000E+00  0.128546E-06  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
         7  0.000000E+00 -0.829074E+02  0.000000E+00  0.000000E+00  0.000000E
+00  0.000000E+00
    67. PRINT JOINT DISPLACEMENTS LIST 6 11
  JOINT    DISPLACE LIST     6
      STAAD SPACE                                                -- PAGE NO.
6
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      6    1    0.00000   0.65963   0.00000   0.00000   0.00000   0.01831
     11    1    0.00000   1.20545   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
```

V. Steady State - With Damping

Determine amplitude of motion of a beam due to harmonic force.

Reference

Paz, Mario. 1985. *Structural Dynamics Theory and Computation. 2nd Edition*. New York, NY:Van Nostrand Reinhold. pp54-55, Example 3.5

Problem

A 10' long, simply-supported beam simulating a damped oscillator.

Material modulus of elasticity, $E = 30{,}000\ ksi$
Section moment of inertia, $I_z = 120\ in^4$
Weight applied at mid-span, $F_y = 3.86\ kips$

A steady state analysis is performed for the harmonic forcing function $7000\ sin\ 60t$ with the forcing frequency = $\omega = 60\ rad/sec = 9.5493\ cyc/sec$ & the magnitude = $7000\ lb = 7\ kip$ applied at the midpoint of the beam. Damping is considered for this problem and the value of 0.1 is specified.

Comparison

Result	Reference	STAAD.Pro	Difference	Comments
Steady State Amplitude (in)	0.1075	0.10749	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Steady State - With Damping.std` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Mar-17
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
SET SHEAR
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 5 0 0;
MEMBER INCIDENCES
1 1 3; 2 3 2;
START USER TABLE
TABLE 1
UNIT FEET KIP
PRISMATIC
Beam
0.001 0.005787 0.001 0.001 0.001 0.001 0 0
END
DEFINE MATERIAL START
ISOTROPIC XYZ
E 4.32e+006
END DEFINE MATERIAL
MEMBER PROPERTY
1 2 UPTABLE 1 Beam
CONSTANTS
MATERIAL XYZ ALL
SUPPORTS
1 2 PINNED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
3 FY -3.86
END DEFINE REFERENCE LOADS
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
REFERENCE LOAD
R1 1.0
MODAL CALCULATION REQUESTED
PERFORM STEADY STATE ANALYSIS
```

```
BEGIN STEADY FORCE
STEADY FORCE FREQ 9.5493 DAMP 0.1
JOINT LOAD
3 FY 7
END
PRINT JOINT DISPLACEMENTS LIST 3
FINISH
```

STAAD.Pro Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      3    1    0.00000   0.10749   0.00000   0.00000   0.00000   0.00000
```

V. Time History - Blast Loading

Determine the dynamic response of a tower subjected to blast loading.

Reference

Paz, Mario. 1985. *Structural Dynamics Theory and Computation. 2nd Edition*. New York, NY:Van Nostrand Reinhold. pp72-74

Problem

Material modulus of elasticity (assumed steel), $E = 29{,}000\ ksi$ and Poisson's ratio, $\nu = 0.3$
Length (assumed) = 10 ft
Section moment of inertia, $I_z = 1{,}986.4\ in^4$ (given $K = 100\ kip/in$, rearrange $K = 3EI/L^3$ to solve for $I = KEL^3/3$
Horizontal weight applied at top of 10' tower, $F_x = 38.6\ kips$

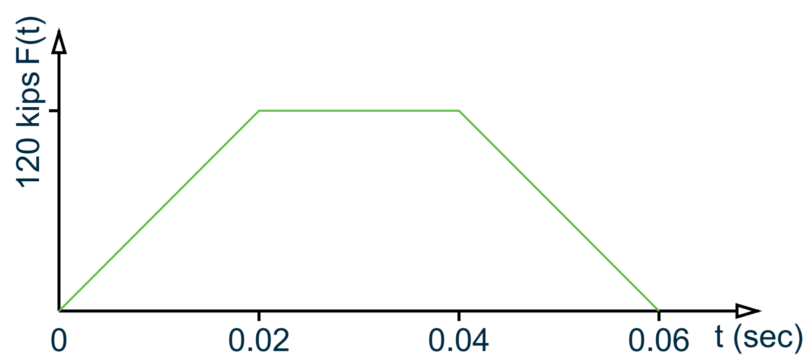


Figure 495: Segmented linear function

Comparison

Result		Reference	STAAD.Pro*	Difference	Comments
Nodal displacement at top (in.) at t =	0	0.000	0.000	none	
	0.02	0.078	0.078	none	
	0.04	0.512	0.512	none	
	0.06	1.134	1.134	none	
	0.08	1.395	1395	none	
	0.10	1.117	1.117	none	

* The STAAD.Pro results are taken from the text data output file with the extension `.tim`.

Note: Apart from `.tim` file, time-displacement graphs in postprocessing also display the results graphically.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Time History - Blast Loading.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Mar-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT FEET KIP
PRISMATIC
CANTILEVER
1e+009 0.095785 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-006
DAMP 1.1e-006
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 CANTILEVER
CONSTANTS
MATERIAL STEEL MEMB 1
SUPPORTS
1 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 FX 38.6
END DEFINE REFERENCE LOADS
CUT OFF MODE SHAPE 0
CUT OFF TIME 0.1999
CUT OFF FREQUENCY 1000
DEFINE TIME HISTORY DT 0.0001
TYPE 1 FORCE SAVE
0 0 0.02 120 0.04 120 0.06 0 0.08 0 0.1 0 0.2 0
ARRIVAL TIME
0
DAMPING 1.1e-006
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
REFERENCE LOAD
R1 1.0
TIME LOAD
2 FX 1 1 1.000000
PERFORM ANALYSIS
FINISH
```

STAAD.Pro Output

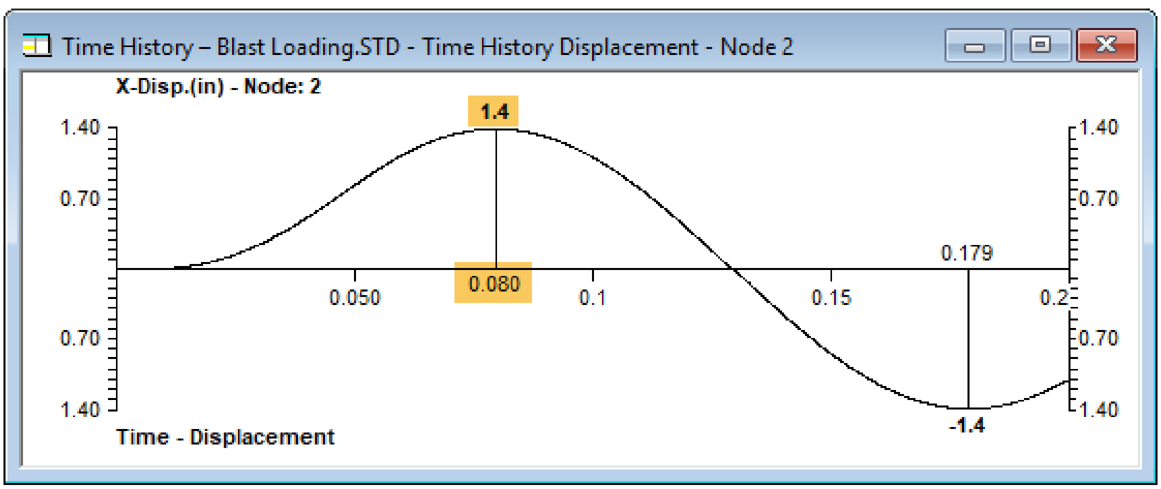


Figure 496: Horizontal displacement vs time graph in Postprocessing workflow

```
                CALCULATED FREQUENCIES FOR LOAD CASE       1
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                     5.033                   0.19867
             MODAL WEIGHT (MODAL MASS TIMES g) IN KIP          GENERALIZED
      MODE            X             Y             Z              WEIGHT
```

```
        1       3.860000E+01  0.000000E+00  0.000000E+00    3.860000E+01
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1   100.00   0.00   0.00  100.000    0.000    0.000
    A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
    MODE       DAMPING
 *WARNING* Damping Ratio reset from .0000011 to 0.0 for modes
      1      0.00000000
   TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00010 SECONDS
   NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =    1
   TIME DURATION OF TIME HISTORY ANALYSIS =      0.200 SECONDS
   NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =     1999
    55. FINISH
  BASE SHEAR UNITS ARE -- KIP  FEET
  MAXIMUM BASE SHEAR  X=   1.395310E+02  Y=   0.000000E+00  Z=   0.000000E+00
      AT TIMES              0.178900           0.000000          0.000000
      STAAD SPACE                                          -- PAGE NO.
4
```

V. Time History - Ground Acceleration

Determine the response of a two-story portal frame subjected to suddenly applied constant acceleration of 0.28 g at its base.

Reference

Paz, Mario. 1985. *Structural Dynamics Theory and Computation. 2nd Edition*. New York, NY:Van Nostrand Reinhold. pp196-201, 213, & 217-219.

Problem

Material modulus of elasticity, $E = 30,000\ ksi$ and Poisson's ratio, $\nu = 0.3$
Section moment of inertia, $I_{z1} = 248.6\ in^4$, $I_{z2} = 106.3\ in^4$
Mass at first story above ground, $F_x = 52.5\ kips$, at roof story $F_x = 25.5\ kips$
Constant ground acceleration = $0.28g = 9.03917\ ft/sec^2$

The ground acceleration is applied as a time vs. acceleration function with the data pairs supplied up to $t = 2\ sec$.

Comparison

Result	Reference	STAAD.Pro	Difference	Comments
Max. displacement at first floor (in); node 2	1.426	1.42605	negligible	
Max. displacement at roof (in); node 5	1.789	1.78064	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\08 Dynamic Analysis\Time History - Ground Acceleration.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0; 3 30 15 0; 4 30 0 0; 5 0 25 0; 6 30 25 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6;
START USER TABLE
TABLE 1
UNIT FEET KIP
PRISMATIC
COLUMN1
1e+009 0.011989 0.001 0.001 0.001 0.001 0.5 0.5
COLUMN2
1e+009 0.005126 0.001 0.001 0.001 0.001 0.5 0.5
TABLE 2
UNIT FEET KIP
PRISMATIC
BEAM
0.001 1e+009 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC XYZSTEEL
E 4.32e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-006
DAMP 0.03
G 1.60615e+006
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 UPTABLE 1 COLUMN1
4 5 UPTABLE 1 COLUMN2
2 6 UPTABLE 2 BEAM
CONSTANTS
MATERIAL XYZSTEEL MEMB 1 2 2 TO 6
SUPPORTS
1 4 FIXED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 FX 52.5
5 FX 25.5
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 3 TYPE RIG HEI 15
DIA 4 TYPE RIG HEI 25
```

```
DEFINE TIME HISTORY DT 0.0013888
TYPE 1 ACCELERATION SAVE
0 9.03917 0.1 9.03917 0.2 9.03917 0.3 9.03917 0.4 9.03917 0.5 9.03917
0.6 9.03917 0.7 9.03917 0.8 9.03917 0.9 9.03917 1 9.03917
1.1 9.03917 1.2 9.03917 1.3 9.03917 1.4 9.03917 1.5 9.03917
1.6 9.03917 1.7 9.03917 1.8 9.03917 1.9 9.03917 2 9.03917
ARRIVAL TIME
0
DAMPING 1.1e-006
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
REFERENCE LOAD
R1 1.0
GROUND MOTION X 1 1 1.000000
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 2 5
FINISH
```

STAAD.Pro Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      2    1   -1.42605   0.00000   0.00000   0.00000   0.00000   0.00000
      5    1   -1.78064   0.00000   0.00000   0.00000   0.00000   0.00000
```

V. Time History - Rectangular Pulse Force

Determine the maximum displacement at the top and maximum bending stress in the columns of a one-story portal frame subject to a rectangular pulse force.

Reference

Chopra, Anil K. 1995. *Dynamics of Structures: Theory and Applications to Earthquake Engineering*. Upper Saddle River, NJ:Prentice Hall, Example 4.1, p 136.

Problem

From the reference:

> A one-story building, idealized as a 12-ft high frame with two columns hinged at the base and a rigid beam, has a natural period of 0.5 sec. Each column is an America standard wide-flange steel section W8x18. Its properties for bending about its major axis are $I_x = 61.9\ in^4$, $S = I_x/c = 15.2\ in^3$; $E = 30{,}000\ ksi$. Neglecting damping, determine the maximum response of this frame due to a rectangular pulse force of amplitude 4 kips and duration $t_d = 0.2\ sec$. The response quantities of interest are displacement at the top of the frame and maximum bending stress in the columns.

The following time vs. force function is applied for the rectangular pulse force:

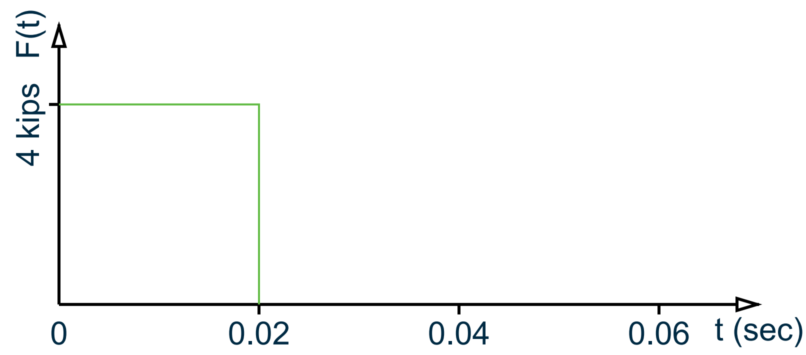


Figure 497: Rectangular pulse force

Calculations

The natural frequency of the structure is:

$$\omega_n = 2\pi/T_n = 2\pi/0.5 = 12.57 \text{ rad/sec.}$$

Also, the lateral stiffness of the frame:

$$k = 2\frac{3EI}{L^3} = 2\frac{3 \times 30{,}000 \times 61.9}{12 \times 12^3} = 3.73 \text{ kips/in}$$

Solving $\omega_n = \sqrt{\frac{k}{m}}$ for m, $m = \frac{k}{\omega_n^2} = \frac{3.73 \times 10^3}{12.57^2} = 23.62\ lbs$.

Thus, the force applied at the top of the frame as the mass load is $W = m \times g = 23.62\ lbs \times 32.2\ ft/s^2 \times 12\ in/ft = 9{,}122\ lbs\text{-}force = 9.122\ kips$.

Comparison

Result	Reference	STAAD.Pro	Difference	Comments
Max. dynamic deformation (in)	2.04	2.03965	negligible	
Max. bending moment (kip·in)	547.8	547.98	negligible	
Max. bending stress (ksi)	36.04	36.0	negligible	

Modeling notes:

- A new material with the modulus of elasticity specified is used in place of the built-in steel material.
- For the generation of the STAAD.Pro model, the cross-section of the beam is take as 6" × 6" and the cross-section properties are assigned using a user-provided table for prismatic shapes. A massive (10^6 ft^4) moment of inertia about local z and y axes is specified to approximate an infinitely rigid beam.
- Pinned supports are supplied at the base of each column. A fixed-but allowing only translation along the global X axis is assigned to the tops of the columns / beam ends.
- As STAAD.Pro defaults to a damping ratio of 0.05 if no arrival time is specified (i.e., an arrival time of $t = 0$ seconds), a negligible damping of $1.1 \times (10)^{-6}$ is assigned.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\08 Dynamic Analysis\Time History - Rectangular Pulse Force.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 06-Mar-17
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 12 0; 3 5 12 0; 4 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT FEET KIP
PRISMATIC
BEAM
1e+06 1e+06 0.001 0.001 0.001 0.001 0.5 0.5
END
DEFINE MATERIAL START
ISOTROPIC XYZ
E 4.32e+06
POISSON 0.3
DENSITY 0.00283
DAMP 0.03
ALPHA 6.5e-06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
2 UPTABLE 1 BEAM
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W8X18
CONSTANTS
MATERIAL XYZ ALL
SUPPORTS
1 4 PINNED
2 3 FIXED BUT FX
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE REF LOAD CASE 1
JOINT LOAD
2 FX 9.11751
END DEFINE REFERENCE LOADS
DEFINE TIME HISTORY DT 0.0001
TYPE 1 FORCE SAVE
0 4 0.2 4 0.2 0 0.4 0 0.6 0
ARRIVAL TIME
0
DAMPING 1.1e-06
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
REFERENCE LOAD
R1 1.0
TIME LOAD
2 FX 1 1 1.000000
```

```
PERFORM ANALYSIS
UNIT INCHES KIP
PRINT JOINT DISPLACEMENTS LIST 2 3
PRINT MEMBER FORCES LIST 1 3
PRINT MEMBER STRESSES LIST 1 3
FINISH
```

STAAD.Pro Output

```
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      2    1    2.03965   0.00000   0.00000   0.00000   0.00000   0.00000
      3    1    2.03965   0.00000   0.00000   0.00000   0.00000   0.00000
   ************** END OF LATEST ANALYSIS RESULT **************
    55. PRINT MEMBER FORCES LIST 1 3
  MEMBER   FORCES   LIST     1
      STAAD PLANE                                            -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00      3.81     0.00      0.00      0.00       0.00
                 2      0.00     -3.81     0.00      0.00      0.00     547.98
      3    1     3      0.00      3.81     0.00      0.00      0.00     547.98
                 4      0.00     -3.81     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    56. PRINT MEMBER STRESSES LIST 1 3
  MEMBER   STRESSES LIST     1
      STAAD PLANE                                            -- PAGE NO.
6
  MEMBER STRESSES
  ---------------
  ALL UNITS ARE KIP /SQ INCH
 MEMB   LD  SECT    AXIAL    BEND-Y     BEND-Z   COMBINED  SHEAR-Y  SHEAR-Z
      1    1   .0      0.0        0.0       0.0       0.0      2.2      0.0
             1.0      0.0 C      0.0      36.0      36.0      2.2      0.0
      3    1   .0      0.0        0.0      36.0      36.0      2.2      0.0
             1.0      0.0 C      0.0       0.0       0.0      2.2      0.0
```

V.09 Steel Design

V. Australia

V. AS4100 1998 - Bending Capacity

To check the bending capacity of the UB shape per AS4100-1998.

Reference

Gorenc,B., R.Tinyou and A.Syam, *Steel Designer's Handbook*, 8th edition, UNSW Press,

problem 5.3, page 112.

Related Links

- *D2.B. Australian Codes - Steel Design per AS 4100 - 1998* (on page 1356)

Problem

Check the suitability of a 360UB50.7 Grade 300 for the beam in Example 5.2 of the reference. This solution relies mainly on AS 4100 and minimally on AISC [1999a], for example, to get basic section properties. Omit deflection checks.

Comparison

Table 591: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Bending capacity, Mnz (kN·m)	138	137.5	0.3%
Bending moment (kN·m)	125	124.7	0.2%
Utilization ratio	0.906	0.907	0.1%

The reference book does not compute the ratio.

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Bending Capacity.STD` is typically installed with the program.

```
STAAD PLANE BENDING CAPACITY PER AS4100 1998
*
* REFERENCE : Steel Designer's Handbook, B.Gorenc, R.Tinyou and A.Syam,
*             8th edition, example problem 5.3, page 112
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF A UB SHAPE IN BENDING PER
*              THE AS4100-1998 CODE
*
START JOB INFORMATION
ENGINEER DATE 22-Jul-14
END JOB INFORMATION
*
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
*
```

```
MEMBER INCIDENCES
1 1 2;
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST UB360X50.7
PRINT MEMBER PROPERTIES ALL
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 2 PINNED
*******************************
LOAD 1 LOAYDTYPE DEAD TITLE PERMANENT LOADS - G
MEMBER LOAD
* UB Self weight
1 UNI GY -0.54
* Slab
1 UNI GY -3.328
*
LOAD 2 LOADTYPE Live  TITLE IMPOSED LOADS - Q
MEMBER LOAD
* Imposed Action
1 UNI GY -4.8
* Point Load
1 CON GY -10
*******************************
LOAD COMBINATION 3 ULC
1 1.2 2 1.5
*******************************
PERFORM ANALYSIS
*
UNIT MMS NEWTON
PARAMETER 1
CODE AUSTRALIAN 1998
*
* Grade 300 steel
FYLD 300 ALL
*
* Twist restraint factor
KT 1.04 ALL
*
* Effective length Le
UNT 5820 ALL
*
* Moment modification factor
ALM 1.41 ALL
*
TRACK 2 ALL
```

```
CHECK CODE ALL
*
* Published capacity:- Mb = 138 (pg 114)
* Therefore Utilisation ratio = 125/138 = 0.906
*
FINISH
```

STAAD Output

```
    *
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.
  |-----------------------------------------------------------------------|
  |                                                    Y        PROPERTIES |
  |*************                                       |        IN  CM  UNIT |
  |            *  |=============================|   ===|===     ------------ |
  |MEMBER   1  *  |  AUSTRALIAN SECTIONS        |      |        AX=0.6470E+2 |
  |            *  | ST  UB360X50.7              |      |   --Z  AY=0.2599E+2 |
  |DESIGN CODE *  |                             |      |        AZ=0.2622E+2 |
  |AS4100 1998 *  ===============================   ===|===     PY=0.1730E+3 |
  |            *                                                PZ=0.8970E+3 |
  |            *  |&lt;---LENGTH (ME=     8.00 --->|                RY=0.3852E
+1 |
  |*************                                                RZ=0.1481E+2 |
  |                                                             Iw=0.2848E+6 |
  |                  124.7( KN-METR)                                         |
  |PARAMETER        |                  �3                       FORCE/MOMENT |
  |IN  NEWTON MM    |              �3      �3                   IN  KN METRE |
  |---------------  +                                           -------------|
  | KL/R-Y=  207.7  |          �3              �3               PNC=0.2405E+3|
  | KL/R-Z=   54.0  +      �3                      �3           PNT=0.1747E+4|
  | UNL   = 5820.0  |                                           pn =0.0000E+0|
  | MAIN  =  180.0  +                                           MNZ=0.1375E+3|
  | PHI   =   0.90  |   �3                             �3       mnz=-.1247E+3|
  | FULT  =  440.0  +                                           MNY=0.4547E+2|
  | FYLD  =  300.0  |�0                                    �0   mny=0.0000E+0|
  | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   VZ =0.6371E+3|
  | SKT   =   1.00   -6.9                                       vz =0.0000E+0|
  | SKL   =   1.00              ABSOLUTE MZ ENVELOPE            VY =0.3938E+3|
  | SKR   =   1.00                 (WITH LOAD NO.)              vy =0.7500E+1|
  |                                                                          |
  | Section Type:  Compact - about Z axis;     Compact - about Y axis        |
  |                                                                          |
  |                       Parameters used to calculate RATIO                 |
  |                       ---------------------------------                  |
  | Ns=0.1723E+4  Msz=0.2691E+3  Msy=0.5053E+2  Mbz=0.1528E+3                |
  |               Miz=0.1130E-4  Miy=0.1130E-4  Moz=0.1130E-4                |
  |              Nciz=0.1433E+4 Nciy=0.2672E+3  Ncz=0.1433E+4  Ncy=0.2672E+3 |
  |              Vvmy=0.4376E+3 Vvmz=0.7079E+3                               |
  | ALPHA,M= 1.410  ALPHA,B= 0.000  ALPHA,SZ= 0.403                          |
  |                                                                          |
  |                         MAX FORCE/ MOMENT SUMMARY ( KN-METR)             |
  |                         -------------------------                        |
  |                  AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z   |
  |      VALUE        0.0        54.9        0.0         0.0       124.7     |
```

```
|   LOCATION       0.0         0.0        0.0         0.0         4.0      |
|    LOADING        0           3          0           0           3       |
|**********************************************************************|
|*                      DESIGN SUMMARY ( KN-METR)                      *|
|*                      --------------                                 *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/         *|
|*         FX              MY              MZ          LOCATION         *|
|*        ======================================================      *|
|*          PASS       AS-8.4.4.2       0.907            3             *|
|*          0.00 C          0.0       -124.7            4.00           *|
|**********************************************************************|
|----------------------------------------------------------------------|
   BENDING CAPACITY PER AS4100 1998                         -- PAGE NO.
5
    *
```

V. AS4100 1998 - Bending Capacity for Non-compact Section

Verify the bending capacity for a non-compact section per AS4100-1998.

Given

The member is 8 m long simply-supported beam. The section is a 900WB218. The member is subject to major axis bending due to uniform dead load of 4.17 kN/m and live load of 8 kN/m as well as concentrated dead load of 104 kN and live load of 140 kN at mid-span.

<u>Section Properties</u>

Overall depth of section, $D = 910\ mm$
Width of section, $B = 350\ mm$
Thickness of web, $t_w = 12\ mm$
Thickness of flange, $t_f = 25\ mm$
Area, $A_g = 27{,}800\ mm^2$
Moment of inertia about major axis, $I_{zz} = 4{,}060\ (10)^6\ mm^4$
Moment of inertia about minor axis, $I_{yy} = 179\ (10)^6\ mm^4$
Elastic section modulus about major axis, $Z_{xx} = 8.92\ (10)^6\ mm^3$
Elastic section modulus about minor axis, $Z_{yy} = 1.02\ (10)^6\ mm^3$
Plastic section modulus about major axis, $S_{xx} = 9.96\ (10)^6\ mm^3$
Plastic section modulus about minor axis, $S_{yy} = 1.56\ (10)^6\ mm^3$
Torsion constant, $J = 4.02\ (10)^6\ mm^4$
Warping moment, $I_w = 35.05\ (10)^{12}\ mm^4$
Radius of gyration, $r_x = \sqrt{I_{zz}/A} = 382.2\ mm$
Radius of gyration, $r_y = \sqrt{I_{yy}/A} = 80.2\ mm$

<u>Material Properties</u>

$E = 200{,}000\ MPa$
$G = 80{,}000\ MPa$
$f_y = 360\ MPa$ (flange); $f_y = 400\ MPa$ (web)

Validation

Section Slenderness Check

$$\frac{b}{2} = \frac{(350 - 12)}{2} = 169 \text{ mm}$$

Flange section slenderness parameter, $\lambda_{ef} = \frac{b}{t_f}\sqrt{\frac{f_y}{250}} = \frac{169}{25}\sqrt{\frac{360}{250}} = 8.112 > \lambda_{ep} = 8$, therefore the flange is non-compact.

$$d = D - 2\times t_f = 910 - 25\times 2 = 860 \text{ mm}$$

Web section slenderness parameter, $\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{860}{12}\sqrt{\frac{400}{250}} = 90.65 > \lambda_{ep} = 82$, therefore the web is non-compact.

Section Bending Capacity Major AxisThe plastic section capacity, Z_{cx}, is the minimum of S_{xx} or $1.5\times Z_{xx} = 1.5(8.92 \times 10^6) = 13.38 \times 10^6 \ mm^3$; so $Z_{cx} = 9.96 \times 10^6 \ mm^3$.

From Table 5.2 of AS 4100, $\lambda_{ey} = 115$ and $\lambda_{ep} = 82$.

For sections where $\lambda_{ep} < \lambda < \lambda_{ey}$, the effective section modulus is calculated as:

$$Z_{ez} = Z_{xx} + \frac{\lambda_{ey} - \lambda_{ew}}{\lambda_{ey} - \lambda_{ep}}\left(Z_{cx} - Z_{xx}\right) = \left[8.92 + \frac{115 - 90.65}{115 - 82}\left(9.96 - 8.92\right)\right]10^6 = 9.69 \times 10^6 \text{ mm}^3$$

$$M_{sx} = Z_{ez} \times f_y = 9.69 \times 360 = 3{,}488 \text{ kN·m}$$

$$\phi M_{sx} = 0.9 \times 3{,}488 = 3{,}139 \text{ kN·m}$$

Section Bending Capacity Minor AxisThe plastic section capacity, Z_{cy} is the minimum of S_{yy} or $1.5\times Z_{yy} = 1.5(1.02 \times 10^6) = 1.53 \times 10^6 \ mm^3$; so $Z_{cy} = 1.53 \times 10^6 \ mm^3$.

For sections where $\lambda_{ep} < \lambda < \lambda_{ey}$, the effective section modulus is calculated as:

$$Z_{ey} = Z_{yy} + \frac{\lambda_{ey} - \lambda_{ew}}{\lambda_{ey} - \lambda_{ep}}\left(Z_{cy} - Z_{yy}\right) = \left[1.02 + \frac{115 - 90.65}{115 - 82}\left(1.53 - 1.02\right)\right]10^6 = 1.53 \times 10^6 \text{ mm}^3$$

$$M_{sy} = Z_{ey} \times f_y = 1.53 \times 360 = 552.4 \text{ kN·m}$$

$$\phi M_{sy} = 0.9 \times 552.4 = 497.1 \text{ kN·m}$$

Member Bending Capacity

Twist restraint factor,

$$k_t = 1 + \left(\frac{2 \times d}{l}\right)\left(\frac{0.5 \times t_f}{t_w}\right)^3 = 1 + \left(\frac{2 \times 860}{8000}\right)\left(\frac{0.5 \times 25}{12}\right)^3 = 1.24$$

From Table 5.6.3(2) of AS 4100, Load Height Factors, $k_l = 1.4$.

From Table 5.6.3(3) of AS 4100, Lateral Rotation Restraint Factors, $k_r = 1.0$.

Effective length of the member:

$$l_e = k_t k_l k_r l = 1.24 \times 1.4 \times 1.0 \times 8{,}000 = 13{,}900 \text{ mm}$$

$$M_o = \sqrt{A(B + C)} \qquad \text{(Eqn 5.6.1.1(3))}$$

where

$$A = \frac{\pi^2 E I_y}{l_e^2} = \frac{\pi^2 \times 200 \times 10^3 179 \times 10^6}{(13{,}900)^2} = 1.829 \times 10^6 \ N$$

$$B = \frac{\pi^2 E I_w}{l_e^2} = \frac{\pi^2 \times 200 \times 10^3 35 \times 10^{12}}{(13{,}900)^2} = 357.6 \times 10^9 \ N \cdot mm^2$$

$$C = G \times J = 80 \times (10)^3 4.02 \times (10)^6 = 321.6 \times (10)^9 \ N \cdot mm^2$$

$$M_o = \sqrt{1.829 \times (10)^6 \left[357.6 \times 10^9 + 321.6 \times (10)^9\right]} = 1{,}116 \times (10)^6 \text{ N}\cdot\text{mm} = 1{,}116 \text{ kN}\cdot\text{m}$$

The slenderness reduction factor is then:

$$a_s = 0.6\left[\sqrt{\frac{M_s^2}{M_o^2} + 3} - \frac{M_s}{M_o}\right] = 0.6\left[\sqrt{\frac{(3{,}488)^2}{(1{,}116)^2} + 3} - \frac{3{,}488}{1{,}116}\right] = 0.268 \quad \text{(Eqn. 5.6.1.1(2))}$$

The moment shape factor is then:

$$M_1 = M_3 = 436.8 \text{ kN}\cdot\text{m}$$

$$M_2 = M_{max} = 805.6 \text{ kN}\cdot\text{m}$$

$$a_m = \frac{1.7 M_{max}}{\sqrt{M_2^2 + M_3^2 + M_4^2}} = \frac{1.7 \times 805.6}{\sqrt{(436.8)^2 + (805.6)^2 + (436.8)^2}} = 1.35$$

The member moment capacity:

$$\phi M_b = \phi a_m a_s M_s = 0.9 \times 1.35 \times 0.268 \times 3{,}488 = 1{,}135 \text{ kN}\cdot\text{m} \quad \text{(Eqn 5.6.1.1(1))}$$

Comparison

Table 592: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comment
Nominal section capacity major axis, M_{sx} (kN·m)	3,139	3,139	none	
Nominal section capacity major axis, M_{sy} (kN·m)	497.1	495.7831	negligible	
Member bending capacity, M_{bx} (kN·m)	1,135	1,134.4	negligible	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Bending Capacity for Non-Compact section.std` is typically installed with the program.

The following design parameters are used:

- The steel grade of Fy = 350 MPa is given by `SGR 8` (AS/NZS 3678 400).
- The load height occuring at the top flange is specified by `LHT 1`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 8.00002 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
G 7.7221e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST WB900X218
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE Dead  TITLE DEAD
MEMBER LOAD
1 UNI GY -4.17
1 CON GY -104
LOAD 2 LOADTYPE Dead  TITLE LIVE
MEMBER LOAD
1 UNI GY -8
1 CON GY -140
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.5
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
LHT 1 PMEMB 1
SGR 8 PMEMB 1
TRACK 2 PMEMB 1
PBRACE TOP 0 P 1 P PMEMB 1
PBRACE BOTTOM 0 P 1 P PMEMB 1
CHECK CODE PMEMB 1
FINISH
```

STAAD Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

==============================================================================
=======
   Designation: ST   WB900X218                (AUSTRALIAN SECTIONS)
   Governing Load Case:     3*
   Governing Criteria: AS-8.4.4.2
   Governing Ratio:   0.710  (PASS)
   SECTION PROPERTIES

==============================================================================
=======
    d:       909.9999 mm   bf:       350.0000 mm
   tf:        25.0000 mm   tw:        12.0000 mm
   Ag:     27800.0000 mm2   J:     4.0200E+06 mm4             Iw:    35.0493E
+12 mm6
   Iz:     4.0600E+09 mm4  Sz:     9.9600E+06 mm3 (plastic)   Zz:     8.9231E
+06 mm3 (elastic)
   rz:   382.1559E+00 mm
   Iy:   179.0000E+06 mm4  Sy:     1.5600E+06 mm3 (plastic)   Zy:     1.0229E
+06 mm3 (elastic)
   ry:    80.2424E+00 mm
   MATERIAL PROPERTIES

==============================================================================
=======
   Material Standard        :  AS 3678
   Nominal Grade            :  400
   Residual Stress Category :  HW (Heavily welded longitudinally)
   E (#)      :199947.000 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   360.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   400.000 MPa    [AS 4100 Table 2.1]
   fu         :   480.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

==============================================================================
=======
   Actual slenderness   :  99.698
   Allowable slenderness: 400.000
      STAAD SPACE                                              -- PAGE NO.
4
   BENDING
```

```
=================================================================================
=======
   Section Bending Capacity
   Critical Load Case:     3*          Critical Ratio:   0.257
   Critical Location:       4.000 m from Start.
   Mz* =  -805.6343E+00 KNm                      My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: Noncompact        Y-Axis Section Slenderness: 
Noncompact
   Zez =     9.6881E+06 mm3                      Zey =     1.5302E+06 mm3
   &#981;Msz =     3.1390E+03 KNm                     &#981;Msy =   495.7831E
+00 KN[AS 4100 5.2.1]
   Member Bending Capacity
   Critical Load Case:     3*          Critical Ratio:   0.710
   Critical Location:       4.000 m from Start.
   Crtiical Segment/Sub-segment:
   Location (Type):   0.00 m(P )-  8.00 m(P )
   Length:    8.00 m
   Mz* =  -805.6343E+00 KNm                      My* =     0.0000E+00 KNm
   kt   =      1.24                         [AS4100 Table 5.6.3(1)]
   kl   =      1.40                         [AS4100 Table 5.6.3(2)]
   kr   =      1.00                         [AS4100 Table 5.6.3(3)]
   le   =     13.92 m                       [AS4100 5.6.3]
   αm   =     1.349                         [AS4100 5.6.1.1(a)(iii)]
   Mo   =     1.1120E+03 KNm                [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.268                         [AS4100 5.6.1.1(a)(iv)]
   φMbz =     1.1344E+03 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

=================================================================================
=======
   Section Shear Capacity
   Critical Load Case:     3*          Critical Ratio:   0.111
   Critical Location:       0.667 m from Start.
   Vy*  =   224.0802E+00 KN
   φVvy  =     2.0266E+03 KN                [AS 4100 5.11.2]
   φVvmy =     2.0266E+03 KN                [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
   φVvz  =     3.4020E+03 KN                [AS 4100 5.11.2]
   φVvmz =     3.4020E+03 KN                [AS 4100 5.12.3]
      STAAD SPACE                                             -- PAGE NO.     
5
   AXIAL

=================================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
   Critical Location:       0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =    21.4645E+03 mm2              [AS 4100 6.2.3 / 6.2.4]
   kf    =     0.772                        [AS 4100 6.2.2]
   An    =    27.8000E+03 mm2
   φNs   =     6.9545E+03 KN                [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =      8.00 m
   Ly    =      8.00 m
   Lez   =      8.00 m
```

```
   Ley   =      8.00 m
   αb    =      0.50                       [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    22.073                       [AS 4100 6.3.3]
   αa,z  =     8.186                       [AS 4100 6.3.3]
   λ,z   =    26.166                       [AS 4100 6.3.3]
    h ,z   =     0.041                       [AS 4100 6.3.3]
    x ,z   =     6.660                       [AS 4100 6.3.3]
   αc,z  =     0.957                       [AS 4100 6.3.3]
   φNcz  = 0.6655E+4 KN                    [AS 4100 6.3.3]
   λn,y  =   105.125                       [AS 4100 6.3.3]
   αa,y  =    16.742                       [AS 4100 6.3.3]
   λ,y   =   113.496                       [AS 4100 6.3.3]
    h ,y   =     0.326                       [AS 4100 6.3.3]
    x ,y   =     0.917                       [AS 4100 6.3.3]
   αc,y  =     0.457                       [AS 4100 6.3.3]
   φNcy  = 0.3175E+4 KN                    [AS 4100 6.3.3]
   φNc   = N/A                             [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =    27.8000E+03 mm2
   φNt   =     9.0072E+03 KN               [AS 4100 7.2]
      STAAD SPACE                                            -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

================================================================================
=======
   Section Combined Capacity
   Critical Condition: Cl 8.3.2
   Critical Load Case:     3*          Critical Ratio:   0.257
   Critical Location:        4.000 m from Start.
   N* =     0.0000E+00 KN    Mz* =  -805.6343E+00 KNm      My* =     0.0000E
+00 KNm
   φNs   =     6.9545E+03 KN               [AS 4100 8.3.1]
   φMsz  =     3.1390E+03 KNm
   φMsy  =   495.7831E+00 KNm
   φMrz  =     3.1390E+03 KNm              [AS 4100 8.3.2]
   φMry  =   495.7831E+00 KNm              [AS 4100 8.3.3]
   Member Combined Capacity - In-plane
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case:     3*          Critical Ratio:   0.710
   Critical Location:        4.000 m from Start.
   N* =     0.0000E+00 KN    Mz* =  -805.6343E+00 KNm      My* =     0.0000E
+00 KNm
   φMbz  =     1.1344E+03 KNm
   φNt   =     9.0072E+03 KN               [AS 4100 8.4.4.2]
   φMozt =     1.1344E+03 KNm              [AS 4100 8.4.4.2]
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
```

```
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case:     3*          Critical Ratio:   0.619
   Critical Location:        4.000 m from Start.
   N* =     0.0000E+00 KN    Mz* =  -805.6343E+00 KNm      My* =     0.0000E
+00 KNm
   φMtz  =     1.1344E+03 KNm              [AS 4100 8.4.5.2]
   φMry  =   495.7831E+00 KNm              [AS 4100 8.4.5.2]
      STAAD SPACE                                        -- PAGE NO.
7

******************************************************************************
**
```

V. AS4100 1998 - Cantilever subject to tension and bending

Verify the tension and out-of-plane bending capacities of an I section cantilever beam per the AS 4100 1998 code.

Details

The cantilevered member is 2 m in length. The section is a 250UC89.5 of grade 300 steel. The member is subject to 434 kN in tension.

Section Properties

Cross-sectional area, $A_g = 11,400\ mm^2$
Plastic section modulus, $S_{xx} = 1.23 \times 10^6\ mm^3$

Material Properties

E = 199,947 MPa
G = 80,000 MPa
$F_{y,web}$ = 300 MPa
$F_{y,flange}$ = 280 MPa
F_u = 440 MPa

Validation

Axial Tension Capacity

Gross yielding:

$$N_{ty} = A_g f_y = 11,400 \times 280(10)^{-3} = 3,190 \text{ kN}$$

Fracture:

Assume net area, A_n, is equal to the gross area.

K_t = correction factor for distribution of forces = 1.0

$$N_{tf} = 0.85 K_t A_n f_u = 0.85 \times 1.0 \times 11,400 \times 440(10)^{-3} = 4,264 \text{ kN}$$

N_t is the lesser of N_{ty} and N_{tf}, $N_t = 3,190\ kN$

$$\phi N_t = 0.9 \times 3,190 = 2,870 \text{ kN}$$

Section Bending Capacity

$$M_{sx} = S_x f_y = 1.23(10)^6 280 = 344 \text{ kN·m}$$

$$\phi M_{sx} = 0.9 \times 344 = 310 \text{ kN·m}$$

Axial tensile load, N, mitigating flexural-torsional buckling:

$$M_{ox} = M_{bx} \times \left(1 + \frac{N}{\phi N_t}\right) = 336\left(1 + \frac{434}{2{,}870}\right) = 387 \text{ kN·m}$$

$$\phi M_{ox} = 0.9 \times 387 = 348 \text{ kN·m}$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Axial tension capacity (kN)	2,870	2,872	negligible	
Member combined out-of-plane bending and tension (kN·m)	310	309.96	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Cantilever subject to tension and bending.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jul-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
G 7.7221e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST UC250X89.5
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON X 434 2 0.28
JOINT LOAD
2 FY -88
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
TRACK 2 PMEMB 1
PBRACE TOP 0 F 1 U PMEMB 1
PBRACE BOTTOM 0 F 1 U PMEMB 1
ALM 1 PMEMB 1
CHECK CODE PMEMB ALL
FINISH
```

STAAD.Pro Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

==============================================================================
=======
   Designation: ST   UC250X89.5               (AUSTRALIAN SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: AS-5.1
   Governing Ratio:   0.581  (PASS)
   SECTION PROPERTIES

==============================================================================
=======
    d:       260.0000 mm   bf:       256.0000 mm
   tf:        17.3000 mm   tw:        10.5000 mm
   Ag:     11400.0000 mm2   J:     1.0400E+06 mm4             Iw:   712.7298E
+09 mm6
   Iz:   143.0000E+06 mm4  Sz:     1.2300E+06 mm3 (plastic)   Zz:     1.1000E
+06 mm3 (elastic)
   rz:   111.9994E+00 mm
   Iy:    48.4000E+06 mm4  Sy:   575.0000E+03 mm3 (plastic)   Zy:   378.1250E
+03 mm3 (elastic)
   ry:    65.1584E+00 mm
   MATERIAL PROPERTIES
```

```
================================================================================
=======
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)      :199947.000 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   280.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   320.000 MPa    [AS 4100 Table 2.1]
   fu         :   440.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

================================================================================
=======
   Actual slenderness   :  30.694
   Allowable slenderness: 400.000
      STAAD SPACE                                              -- PAGE NO.
4
   BENDING

================================================================================
=======
   Section Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.568
   Critical Location:        0.000 m from Start.
   Mz* =   176.0000E+00 KNm                      My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: Compact           Y-Axis Section Slenderness:
Compact
   Zez =     1.2300E+06 mm3                      Zey =   567.1875E+03 mm3
   φMsz =   309.9600E+00 KNm                     φMsy =   142.9313E+00 KN[AS
4100 5.2.1]
   Member Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.581
   Critical Location:        0.000 m from Start.
   Crtiical Segment/Sub-segment:
   Location (Type):   0.00 m(F )-  2.00 m(U )
   Length:    2.00 m
   Mz* =   176.0000E+00 KNm                      My* =     0.0000E+00 KNm
   kt   =      1.00                         [AS4100 Table 5.6.3(1)]
   kl   =      1.00                         [AS4100 Table 5.6.3(2)]
   kr   =      1.00                         [AS4100 Table 5.6.3(3)]
   le   =      2.00 m                       [AS4100 5.6.3]
   αm   =     1.000                         [AS4100 5.6.1.1(a)(iii)]
   Mo   =     3.2222E+03 KNm                [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.977                         [AS4100 5.6.1.1(a)(iv)]
   φMbz =   302.8551E+00 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

================================================================================
=======
   Section Shear Capacity
   Critical Load Case:     1*          Critical Ratio:   0.187
   Critical Location:        0.000 m from Start.
   Vy*  =    88.0000E+00 KN
   φVvy  =   471.7441E+00 KN                [AS 4100 5.11.2]
   φVvmy =   471.7441E+00 KN                [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
```

```
   φVvz  =     1.3393E+03 KN                [AS 4100 5.11.2]
   φVvmz =     1.3393E+03 KN                [AS 4100 5.12.3]
      STAAD SPACE                                              -- PAGE NO.
5
   AXIAL

==============================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =    11.4000E+03 mm2               [AS 4100 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =    11.4000E+03 mm2
   φNs   =     2.8728E+03 KN                [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =      2.00 m
   Ly    =      2.00 m
   Lez   =      2.00 m
   Ley   =      2.00 m
   αb    =      0.00                        [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    18.898                        [AS 4100 6.3.3]
   αa,z  =     5.352                        [AS 4100 6.3.3]
   λ,z   =    18.898                        [AS 4100 6.3.3]
    h ,z   =     0.018                        [AS 4100 6.3.3]
    x ,z   =    12.039                        [AS 4100 6.3.3]
   αc,z  =     0.982                        [AS 4100 6.3.3]
   φNcz  = 0.2821E+4 KN                     [AS 4100 6.3.3]
   λn,y  =    32.484                        [AS 4100 6.3.3]
   αa,y  =    15.285                        [AS 4100 6.3.3]
   λ,y   =    32.484                        [AS 4100 6.3.3]
    h ,y   =     0.062                        [AS 4100 6.3.3]
    x ,y   =     4.576                        [AS 4100 6.3.3]
   αc,y  =     0.934                        [AS 4100 6.3.3]
   φNcy  = 0.2684E+4 KN                     [AS 4100 6.3.3]
   φNc   = N/A                              [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*          Critical Ratio:   0.151
   Critical Location:        0.000 m from Start.
   N*    =  -434.0000E+00 KN
   kt    =      1.00                        [User defined]
   An    =    11.4000E+03 mm2
   φNt   =     2.8728E+03 KN                [AS 4100 7.2]
      STAAD SPACE                                              -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

==============================================================================
=======
   Section Combined Capacity
   Critical Condition: Cl 8.3.2
   Critical Load Case:     1*          Critical Ratio:   0.568
   Critical Location:        0.000 m from Start.
   N* =  -434.0000E+00 KN    Mz* =   176.0000E+00 KNm      My* =     0.0000E
+00 KNm
   φNs   =     2.8728E+03 KN                [AS 4100 8.3.1]
   φMsz  =   309.9600E+00 KNm
```

```
   φMsy  =   142.9313E+00 KNm
   φMrz  =   309.9600E+00 KNm              [AS 4100 8.3.2]
   φMry  =   142.9313E+00 KNm              [AS 4100 8.3.3]
   Member Combined Capacity - In-plane
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case:     1*         Critical Ratio:   0.568
   Critical Location:        0.000 m from Start.
   N* =  -434.0000E+00 KN    Mz* =   176.0000E+00 KNm      My* =     0.0000E
+00 KNm
   φMbz  =   302.8551E+00 KNm
   φNt   =     2.8728E+03 KN               [AS 4100 8.4.4.2]
   φMozt =   309.9600E+00 KNm              [AS 4100 8.4.4.2]
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case:     1*         Critical Ratio:   0.453
   Critical Location:        0.000 m from Start.
   N* =  -434.0000E+00 KN    Mz* =   176.0000E+00 KNm      My* =     0.0000E
+00 KNm
   φMtz  =   309.9600E+00 KNm              [AS 4100 8.4.5.2]
   φMry  =   142.9313E+00 KNm              [AS 4100 8.4.5.2]
      STAAD SPACE                                        -- PAGE NO.
7

**************************************************************************
**
```

V. AS4100 1998 - Combined Section and Member In-Plane Capacity

Verify the bending capacity in major and minor axis bending, biaxial bending, and in-plane bending for a UC member subject an axial load per AS4100-1998.

Given

The member is 7 m long and part of a steel frame structure. The section is a 200UC52.2. The member is subject to an axial load of 143.9 kN.

Section Properties

Depth of section, $D = 206\ mm$
Width of section, $B = 204\ mm$
Thickness of web, $t_w = 8\ mm$
Thickness of flange, $t_f = 12.5\ mm$
Area, $A_g = 6{,}660\ mm^2$
Moment of inertia about major axis, $I_{zz} = 52.8\ (10)^6\ mm^4$
Moment of inertia about minor axis, $I_{yy} = 17.7\ (10)^6\ mm^4$
Elastic section modulus about major axis, $Z_{xx} = 513\ (10)^3\ mm^3$
Elastic section modulus about minor axis, $Z_{yy} = 174\ (10)^3\ mm^3$

Plastic section modulus about major axis, $S_{xx} = 570\ (10)^3\ mm^3$
Plastic section modulus about minor axis, $S_{yy} = 264\ (10)^3\ mm^3$
Radius of gyration, $r_x = \sqrt{I_{zz}/A} = 89.0\ mm$
Radius of gyration, $r_y = \sqrt{I_{yy}/A} = 51.6\ mm$

Material Properties

$f_y = 300\ MPa$ (flange); $f_y = 320\ MPa$ (web)

Validation

Combined Major Axis Section Moment Capacity

Nominal axial capacity: $\phi N_s = 0.9 \times 6.66 \times 300 = 1,798\ kN$

Nominal section capacity: $\phi M_{sx} = 0.9 \times 570 \times 300 = 153.9\ kN{\cdot}m$

The section is compact, so the alternate method is applicable.

$$\phi M_{\text{rx}} = \phi 1.18 M_{\text{sx}}\left(1 - \frac{N}{\phi N_s}\right) \le \phi M_{\text{sx}}$$

$$\phi M_{\text{rx}} = 1.18 \times 153.9\left(1 - \frac{143.9}{1{,}798}\right) = 167.1\ \text{kN}{\cdot}\text{m} > \phi M_{\text{sx}}$$

$$\phi M_{\text{rx}} = \phi M_{\text{sx}} = 153.9\ \text{kN}{\cdot}\text{m}$$

Combined Minor Axis Section Moment Capacity

Nominal section capacity: $\phi M_{sy} = \phi S_{yy} F_y \le \phi 1.5 Z_{yy} F_y$

$$\phi M_{\text{sy}} = 0.9 \times 264 \times 300 = 71.3\ \text{kN}{\cdot}\text{m} > 0.9 \times 1.5 \times 174 \times 300 = 70.5\ \text{kN}{\cdot}\text{m}$$

The section is compact, so the alternate method is applicable.

$$\phi M_{\text{ry}} = \phi 1.19 M_{\text{sy}}\left(1 - \frac{N}{\phi N_s}\right)^2 \le \phi M_{\text{sy}}$$

$$\phi M_{\text{ry}} = 1.19 \times 70.5\left(1 - \frac{143.9}{1{,}798}\right)^2 = 71.0\ \text{kN}{\cdot}\text{m} > \phi M_{\text{sy}}$$

$$\phi M_{\text{ry}} = \phi M_{\text{sy}} = 70.5\ \text{kN}{\cdot}\text{m}$$

Member In-plane Capacity Check

The section is compact, so the alternate method is applicable. The larger end moment = $111.1\ kN{\cdot}m$ and the smaller end moment = $108.1\ kN{\cdot}m$.

$$\beta_{\text{mx}} = \frac{108.1}{111.1} = 0.973$$

$$\phi M_{\text{ix}} = \phi M_s\left\{\left[1 - \left(\frac{1 + \beta_m}{2}\right)^3\right]\left(1 - \frac{N}{\phi N_c}\right) + 1.18\left(1 + \frac{1 + \beta_m}{2}\right)^3\sqrt{\left(1 - \frac{N}{\phi N_c}\right)}\right\} \le \phi M_{\text{rx}}$$

$$\phi M_{\text{ix}} = 153.9\left\{\left[1 - \left(\frac{1 + 0.973}{2}\right)^3\right]\left(1 - \frac{143.9}{1{,}146}\right) + 1.18\left(1 + \frac{1 + 0.973}{2}\right)^3\sqrt{\left(1 - \frac{143.9}{1{,}146}\right)}\right\} = 168.4\ \text{kN}{\cdot}\text{m} > \phi M_{\text{rx}}$$

$$\phi M_{\text{ix}} = \phi M_{\text{rx}} = 153.9\ \text{kN}{\cdot}\text{m}$$

Comparison

Table 593: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comment
Combined major axis section moment capacity (kN·m)	153.9	153.9	none	
Combined minor axis section moment capacity (kN·m)	153.9	153.9	none	
Menber in-plane capacity check (kN·m)	153.9	153.9	none	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Combined Section and Member In-Plane Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 7 0; 3 12 7 0; 4 12 0 0; 5 0 12 0; 6 12 12 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 3 6; 6 5 6;
DEFINE PMEMBER
1 PMEMBER 4
4 PMEMBER 5
6 PMEMBER 6
2 PMEMBER 7
5 PMEMBER 8
3 PMEMBER 9
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
G 7.88462e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AUSTRALIAN
```

```
4 5 TABLE ST UC150X30.0
1 3 TABLE ST UC200X52.2
6 TABLE ST UB250X25.7
2 TABLE ST UB310X40.4
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
6 CON GY -40
2 CON GY -80
JOINT LOAD
5 6 FY -20
2 3 FY -50
5 FX 10
2 FX 30
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
TRACK 2 PMEMB 9
CHECK CODE PMEMB 9
FINISH
```

STAAD Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     9
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

==============================================================================
=======
   Designation: ST   UC200X52.2               (AUSTRALIAN SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: SLENDERNESS
   Governing Ratio:   0.754  (PASS)
   SECTION PROPERTIES

==============================================================================
=======
    d:       206.0000 mm   bf:       204.0000 mm
   tf:        12.5000 mm   tw:         8.0000 mm
   Ag:      6660.0000 mm2   J:   325.0000E+03 mm4            Iw:   165.6820E
+09 mm6
   Iz:    52.8000E+06 mm4  Sz:   570.0000E+03 mm3 (plastic)   Zz:   512.6214E
+03 mm3 (elastic)
```

```
   rz:    89.0389E+00 mm
   Iy:    17.7000E+06 mm4  Sy:   264.0000E+03 mm3 (plastic)   Zy:   173.5294E
+03 mm3 (elastic)
   ry:    51.5525E+00 mm
   MATERIAL PROPERTIES

==============================================================================
=======
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)      :204999.984 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   300.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   320.000 MPa    [AS 4100 Table 2.1]
   fu         :   440.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

==============================================================================
=======
   Actual slenderness   : 135.784
   Allowable slenderness: 180.000
      STAAD SPACE                                             -- PAGE NO.
4
   BENDING

==============================================================================
=======
   Section Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.722
   Critical Location:       0.000 m from Start.
   Mz* =   111.0612E+00 KNm                      My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: Compact           Y-Axis Section Slenderness:
Compact
   Zez =   570.0000E+03 mm3                      Zey =   260.2941E+03 mm3
   φMsz =   153.9000E+00 KNm                     φMsy =    70.2794E+00 KN[AS
4100 5.2.1]
   Member Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.722
   Critical Location:       0.000 m from Start.
   Crtiical Segment/Sub-segment:
   Location (Type):   0.00 m(F )-  7.00 m(F )
   Length:    7.00 m
   Mz* =   111.0612E+00 KNm                      My* =     0.0000E+00 KNm
   kt   =      1.00                         [AS4100 Table 5.6.3(1)]
   kl   =      1.00                         [AS4100 Table 5.6.3(2)]
   kr   =      1.00                         [AS4100 Table 5.6.3(3)]
   le   =      7.00 m                       [AS4100 5.6.3]
   αm   =     2.435                         [AS4100 5.6.1.1(a)(iii)]
   Mo   =   154.9263E+00 KNm                [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.570                         [AS4100 5.6.1.1(a)(iv)]
   φMbz =   153.9000E+00 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

==============================================================================
=======
   Section Shear Capacity
   Critical Load Case:     1*          Critical Ratio:   0.110
```

```
   Critical Location:        0.000 m from Start.
   Vy*  =    31.3075E+00 KN
   φVvy  =   284.7744E+00 KN                [AS 4100 5.11.2]
   φVvmy =   284.7744E+00 KN                [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
   φVvz  =   826.2000E+00 KN                [AS 4100 5.11.2]
   φVvmz =   826.2000E+00 KN                [AS 4100 5.12.3]
      STAAD SPACE                                                   -- PAGE NO.
5
   AXIAL

==============================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*          Critical Ratio:   0.269
   Critical Location:        0.000 m from Start.
   N*    =   143.8336E+00 KN
   Ae    =     6.6600E+03 mm2               [AS 4100 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =     6.6600E+03 mm2
   φNs   =     1.7982E+03 KN                [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =      7.00 m
   Ly    =      7.00 m
   Lez   =      7.00 m
   Ley   =      7.00 m
   αb    =      0.00                        [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    86.121                        [AS 4100 6.3.3]
   αa,z  =    18.714                        [AS 4100 6.3.3]
   λ,z   =    86.121                        [AS 4100 6.3.3]
    h ,z   =     0.237                        [AS 4100 6.3.3]
    x ,z   =     1.175                        [AS 4100 6.3.3]
   αc,z  =     0.637                        [AS 4100 6.3.3]
   φNcz  = 0.1146E+4 KN                     [AS 4100 6.3.3]
   λn,y  =   148.744                        [AS 4100 6.3.3]
   αa,y  =    12.969                        [AS 4100 6.3.3]
   λ,y   =   148.744                        [AS 4100 6.3.3]
    h ,y   =     0.441                        [AS 4100 6.3.3]
    x ,y   =     0.764                        [AS 4100 6.3.3]
   αc,y  =     0.298                        [AS 4100 6.3.3]
   φNcy  = 0.5353E+3 KN                     [AS 4100 6.3.3]
   φNc   = N/A                              [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                        [User defined]
   An    =     6.6600E+03 mm2
   φNt   =     1.7982E+03 KN                [AS 4100 7.2]
      STAAD SPACE                                                   -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

==============================================================================
=======
   Section Combined Capacity
   Critical Condition: Cl 8.3.2
   Critical Load Case:     1*          Critical Ratio:   0.722
```

```
   Critical Location:        0.000 m from Start.
   N* =   143.8336E+00 KN    Mz* =   111.0612E+00 KNm      My* =     0.0000E
+00 KNm
   φNs   =     1.7982E+03 KN               [AS 4100 8.3.1]
   φMsz  =   153.9000E+00 KNm
   φMsy  =    70.2794E+00 KNm
   φMrz  =   153.9000E+00 KNm              [AS 4100 8.3.2]
   φMry  =    70.2794E+00 KNm              [AS 4100 8.3.3]
   Member Combined Capacity - In-plane
   Critical Load Case:     1*          Critical Ratio:   0.722
   Critical Location:        0.000 m from Start.
   N* =   143.8336E+00 KN    Mz* =   111.0612E+00 KNm      My* =     0.0000E
+00 KNm
   φNcz  =     1.1463E+03 KN               [AS 4100 8.4.2.2]
   φMiz  =   153.9000E+00 KNm              [AS 4100 8.4.2.2]
   φNcy  =   535.3019E+00 KN               [AS 4100 8.4.2.2]
   φMiy  =    70.2794E+00 KNm              [AS 4100 8.4.2.2]
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case:     1*          Critical Ratio:   0.722
   Critical Location:        0.000 m from Start.
   N* =   143.8336E+00 KN    Mz* =   111.0612E+00 KNm      My* =     0.0000E
+00 KNm
   φMbz  =   153.9000E+00 KNm
   φNcy  =   535.3019E+00 KN
   φMozc =   153.9000E+00 KNm              [AS 4100 8.4.4.1]
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case:     1*          Critical Ratio:   0.633
   Critical Location:        0.000 m from Start.
   N* =   143.8336E+00 KN    Mz* =   111.0612E+00 KNm      My* =     0.0000E
+00 KNm
   φMcz  =   153.9000E+00 KNm              [AS 4100 8.4.5.1]
   φMiy  =    70.2794E+00 KNm              [AS 4100 8.4.5.1]
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
      STAAD SPACE                                            -- PAGE NO.
7

******************************************************************************
**
```

V. AS4100 1998 - Compression Capacity

Verify the capacity of a tubular compression member subjected to axial load only per AS4100-1998.

Given

The member 3.8 m long with pinned ends (no end moments). The section is a 219.1x6.4CHS. The member is subject to an axial load of 1,030 kN.

Section Properties

Depth of section, $d = 219.1\ mm$

Thickness, $t = 6.4\ mm$
Area, $A_g = 4{,}280\ mm^2$
Moment of inertia about major axis, $I_{zz} = 22.2\ (10)^6\ mm^4$
Radius of gyration, $r_x = \sqrt{I_{zz}/A} = 75.2\ mm$

Material Properties

$F_y = 350\ MPa$

Validation

Nominal Section Capacity Check

$$\lambda_e = \left(\frac{d_o}{t}\right)\left(\frac{F_y}{250}\right) = \frac{219.1}{6.5} \times \frac{350}{250} = 47.9$$

$$\lambda_{ey} = 82$$

$$d_{e1} = d_0\sqrt{\frac{\lambda_{ey}}{\lambda_e}} = 219.1\sqrt{\frac{82}{47.9}} = 287 \le d_o = 219.1 \text{ mm}$$

$$d_{e1} = d_0\left(\frac{3 \times \lambda_{ey}}{\lambda_e}\right)^2 = 219.1\left(\frac{3 \times 82}{47.9}\right)^2 = 5{,}780 \text{ mm}$$

The effective depth is the minimum of d_{e1} and d_{e2}, or $219.1\ mm$.

$$d_i = d_e - 2 \times t = 219.1 - 2(6.4) = 206.3 \text{ mm}$$

The effective area,

$$A_e = \frac{\pi {d_e}^2}{4} - \frac{\pi {d_i}^2}{4} = \frac{\pi(219.1)^2}{4} - \frac{\pi(206.3)^2}{4} = 4{,}280 \text{ mm}^2$$

$$K_f = \frac{A_e}{A_g} = \frac{4{,}280}{4{,}280} = 1$$

Net area = $4{,}280\ mm^2$

Nominal section capacity:

$$N_s = K_f \times A_n \times F_y = 1 \times 4{,}280 \times 350(10)^{-3} = 1{,}500 \text{ kN}$$

$$\phi N_s = 0.9 \times 1{,}500 = 1{,}350 \text{ kN}$$

Axial Member Capacity

The effective length for the member pinned at both ends:

$$l_e = K_e \times L = 1 \times 3{,}800 = 3{,}800 \text{ mm}$$

$$\frac{l_e}{r_x} = \frac{3{,}800}{75.2} = 50.5$$

Modified slenderness ratio:

$$\lambda_n = \frac{l_e}{r_x}\sqrt{\frac{K_f F_y}{250}} = 50.5\sqrt{\frac{1 \times 350}{250}} = 59.8$$

The member section has a constant a_b. The residual stress for cold-formed (non-stress relieved) CHS category. Kf = 1. Therefore, from AS 4100 table 6.3.3(1), $a_b = -0.5$.

From interpolating Table 6.6 or AS 4100 Table 6.3.3(3), the slenderness reduction factor, $a_c = 0.863$.

$$N_c = \alpha_c \times N_s = 0.863 \times 1{,}500 = 1{,}290 \text{ kN}$$

$$N_c < N_s, \text{ OK}$$

$$\phi N_c = 0.9 \times 1{,}290 = 1{,}160 \text{ kN}$$

Comparison

Table 594: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comment
Nominal section capacity (kN)	1,350	1,348.2	negligible	
Axial member capacity (kN)	1,160	1,163	negligible	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3.80001 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
G 7.7221e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AUSTRALIAN
1 TABLE ST 219.1x6.4CHS
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GX 1030 0
1 CON GX -1030 3.8
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
TRACK 2 PMEMB 1
SGR 5 PMEMB 1
CHECK CODE PMEMB ALL
FINISH
```

STAAD Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

==============================================================================
=======
   Designation: ST   219.1X6.4CHS             (AUSTRALIAN COLDFORMED SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: COMPRESSION
   Governing Ratio:   0.886  (PASS)
   SECTION PROPERTIES

==============================================================================
=======
    d:       219.1000 mm   bf:         0.0000 mm
   tf:         6.4000 mm   tw:         6.4000 mm
   Ag:      4280.0000 mm2   J:    48.4000E+06 mm4             Iw:     0.0000E
+00 mm6
   Iz:    24.2000E+06 mm4  Sz:   290.0000E+03 mm3 (plastic)   Zz:   220.9037E
+03 mm3 (elastic)
   rz:    75.1945E+00 mm
   Iy:    24.2000E+06 mm4  Sy:   290.0000E+03 mm3 (plastic)   Zy:   220.9037E
+03 mm3 (elastic)
   ry:    75.1945E+00 mm
   MATERIAL PROPERTIES

==============================================================================
```

```
=======
   Material Standard        :  AS 1163
   Nominal Grade            :  350
   Residual Stress Category :  CF (Cold formed)
   E (#)      :199947.000 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   350.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   350.000 MPa    [AS 4100 Table 2.1]
   fu         :   430.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

==============================================================================
=======
   Actual slenderness   :  50.536
   Allowable slenderness: 180.000
      STAAD SPACE                                                 -- PAGE NO.
4
   BENDING

==============================================================================
=======
   Section Bending Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Mz* =     0.0000E+00 KNm                      My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: N/A               Y-Axis Section Slenderness:
N/A
   Zez =     0.0000E+00 mm3                      Zey =     0.0000E+00 mm3
   φMsz =     0.0000E+00 KNm                     φMsy =     0.0000E+00 KN[AS
4100 5.2.1]
   Member Bending Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Crtiical Segment/Sub-segment:
   Location (Type):   0.00 m(U )-  0.00 m(U )
   Length:    0.00 m
   Mz* =     0.0000E+00 KNm                      My* =     0.0000E+00 KNm
   kt   =      0.00                         [AS4100 Table 5.6.3(1)]
   kl   =      0.00                         [AS4100 Table 5.6.3(2)]
   kr   =      0.00                         [AS4100 Table 5.6.3(3)]
   le   =      0.00 m                       [AS4100 5.6.3]
   αm   =     0.000                         [AS4100 5.6.1.1(a)(iii)]
   Mo   =     0.0000E+00 KNm                [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.000                         [AS4100 5.6.1.1(a)(iv)]
   φMbz =     0.0000E+00 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

==============================================================================
=======
   Section Shear Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Vy*  =     0.0000E+00 KN
   φVvy  =     0.0000E+00 KN                [AS 4100 5.11.2]
   φVvmy =     0.0000E+00 KN                [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
   φVvz  =     0.0000E+00 KN                [AS 4100 5.11.2]
   φVvmz =     0.0000E+00 KN                [AS 4100 5.12.3]
```

```
      STAAD SPACE                                              -- PAGE NO.
5
   AXIAL

==============================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*          Critical Ratio:   0.886
   Critical Location:        0.317 m from Start.
   N*    =     1.0300E+03 KN
   Ae    =     4.2766E+03 mm2              [AS 4100 6.2.3 / 6.2.4]
   kf    =     0.999                       [AS 4100 6.2.2]
   An    =     4.2800E+03 mm2
   φNs   =     1.3471E+03 KN               [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =      3.80 m
   Ly    =      3.80 m
   Lez   =      3.80 m
   Ley   =      3.80 m
   αb    =     -0.50                       [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    59.771                       [AS 4100 6.3.3]
   αa,z  =    20.639                       [AS 4100 6.3.3]
   λ,z   =    49.451                       [AS 4100 6.3.3]
    h ,z   =     0.117                       [AS 4100 6.3.3]
    x ,z   =     2.350                       [AS 4100 6.3.3]
   αc,z  =     0.863                       [AS 4100 6.3.3]
   φNcz  = 0.1163E+4 KN                    [AS 4100 6.3.3]
   λn,y  =    59.771                       [AS 4100 6.3.3]
   αa,y  =    20.639                       [AS 4100 6.3.3]
   λ,y   =    49.451                       [AS 4100 6.3.3]
    h ,y   =     0.117                       [AS 4100 6.3.3]
    x ,y   =     2.350                       [AS 4100 6.3.3]
   αc,y  =     0.863                       [AS 4100 6.3.3]
   φNcy  = 0.1163E+4 KN                    [AS 4100 6.3.3]
   φNc   = N/A                             [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =     4.2800E+03 mm2
   φNt   =     1.3482E+03 KN               [AS 4100 7.2]
      STAAD SPACE                                              -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

==============================================================================
=======
   Section Combined Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   N* =     0.0000E+00 KN    Mz* =     0.0000E+00 KNm      My* =     0.0000E
+00 KNm
   φNs   =     0.0000E+00 KN               [AS 4100 8.3.1]
   φMsz  =     0.0000E+00 KNm
   φMsy  =     0.0000E+00 KNm
   φMrz  =     0.0000E+00 KNm              [AS 4100 8.3.2]
   φMry  =     0.0000E+00 KNm              [AS 4100 8.3.3]
```

```
   Member Combined Capacity - In-plane
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
      STAAD SPACE                                             -- PAGE NO.
7

******************************************************************************
**
```

V. AS4100 1998 - SHS section subject to compression

Verify the axial compression capacities of an SHS section beam per the AS 4100 1998 code.

Details

The member is 3.8 m in length, with pinned supports at both ends. The section is a 200 x 200 x 5.0 SHS of grade 250 steel. The member is subject to 1,030 kN in compression.

Section Properties

Depth of section, $b = d = 200\ mm$
Thickness, $t = 5\ mm$
Cross-sectional area, $A_g = 3{,}810\ mm^2$
Radius of gyration, $r_x = 79.2\ mm$

Material Properties

E = 199,947 MPa
G = 80,000 MPa
F_y = 250 MPa
F_u = 320 MPa

Validation

Nominal Section Capacity

Clear width: $b_{cw} = b - 2t = 200 - 2\times5 = 190\ mm$

$$\lambda_e = \frac{b_{\mathrm{cw}}}{t}\sqrt{\frac{f_y}{250}} = \frac{190}{5}\sqrt{\frac{250}{250}} = 38$$

$$\lambda_{\mathrm{ey}} = 40$$

Effective width, b_e, for each compression element:

$$b_e = b_{cw}\sqrt{\frac{\lambda_{ey}}{\lambda_e}} \leq b_{cw} = 190\sqrt{\frac{40}{38}} = 195 > 190$$

Use $b_e = 190\ mm$

Effective area, $A_e = 4 \times b_e \times t = 4 \times 190 \times 5 = 3{,}800\ mm^2$

$$K_f = \frac{A_e}{A_g} = \frac{3{,}800}{3{,}810} = 1.0$$

Assume the net area, A_n, is equal to the gross area (i.e., no holes).

The nominal section capacity:

$$N_s = K_f A_n f_y = 1.0 \times 3{,}800 \times 250(10)^{-3} = 950 \text{ kN}$$

$$\phi N_s = 0.9 \times 950 = 855 \text{ kN}$$

<u>Axial Member Capacity</u>

The effective length, $l_e = K_e \times L = 1 \times 3{,}800\ mm = 3{,}800\ mm$

The modified slenderness ratio:

$$\lambda_n = \frac{l_e}{r}\sqrt{\frac{K_f f_y}{250}} = \frac{3{,}800}{79.2}\sqrt{\frac{1 \times 250}{250}} = 47.97$$

Member section constant, a_b, for shape of section, residual stress cold-formed (non-stress relieved) CHS category, and $K_f = 1.0$ from AS 4100 Table 6.3.3.(2): $a_b = -0.5$.

From linear interpolation of Table 6.3.3(3), the slenderness reduction factor, $a_c = 0.913$.

$$N_c = a_c N_s = 0.913 \times 950 = 867.4 \text{ kN}$$

$$\phi N_c = 0.9 \times 867.4 = 780.6 \text{ kN}$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Nominal section capacity (kN)	855	857.2499	negligible	
Axial member capacity (kN)	780.6	782.7	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - SHS section subject to compression.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Mar-21
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3.80001 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
G 7.7221e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AUSTRALIAN
1 TABLE ST 200X200X5.0SHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GX 1030 0
1 CON GX -1030 3.8
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
TRACK 2 PMEMB 1
SGR 0 PMEMB 1
CHECK CODE PMEMB ALL
FINISH
```

STAAD.Pro Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

==============================================================================
=======
```

```
   Designation: ST   200X200X5.0SHS           (AUSTRALIAN COLDFORMED SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: COMPRESSION
   Governing Ratio:   1.316 *(FAIL)
   SECTION PROPERTIES

==============================================================================
=======
    d:       200.0000 mm   bf:       200.0000 mm
   tf:         5.0000 mm   tw:         5.0000 mm
   Ag:      3809.9998 mm2   J:    37.8000E+06 mm4             Iw:     0.0000E
+00 mm6
   Iz:    23.9000E+06 mm4  Sz:   277.0000E+03 mm3 (plastic)   Zz:   239.0000E
+03 mm3 (elastic)
   rz:    79.2021E+00 mm
   Iy:    23.9000E+06 mm4  Sy:   277.0000E+03 mm3 (plastic)   Zy:   239.0000E
+03 mm3 (elastic)
   ry:    79.2021E+00 mm
   MATERIAL PROPERTIES

==============================================================================
=======
   Material Standard        :  AS 1163
   Nominal Grade            :  250
   Residual Stress Category :  CF (Cold formed)
   E (#)      :199947.000 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   250.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   250.000 MPa    [AS 4100 Table 2.1]
   fu         :   320.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

==============================================================================
=======
   Actual slenderness   :  47.979
   Allowable slenderness: 180.000
      STAAD SPACE                                                 -- PAGE NO.
4
   BENDING

==============================================================================
=======
   Section Bending Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Mz* =     0.0000E+00 KNm                       My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: N/A                Y-Axis Section Slenderness:
N/A
   Zez =     0.0000E+00 mm3                       Zey =     0.0000E+00 mm3
   φMsz =     0.0000E+00 KNm                      φMsy =     0.0000E+00 KN[AS
4100 5.2.1]
   Member Bending Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Crtiical Segment/Sub-segment:
   Location (Type):   0.00 m(U )-  0.00 m(U )
   Length:    0.00 m
   Mz* =     0.0000E+00 KNm                       My* =     0.0000E+00 KNm
```

```
   kt   =      0.00                        [AS4100 Table 5.6.3(1)]
   kl   =      0.00                        [AS4100 Table 5.6.3(2)]
   kr   =      0.00                        [AS4100 Table 5.6.3(3)]
   le   =      0.00 m                      [AS4100 5.6.3]
   αm   =     0.000                        [AS4100 5.6.1.1(a)(iii)]
   Mo   =     0.0000E+00 KNm               [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.000                        [AS4100 5.6.1.1(a)(iv)]
   φMbz =     0.0000E+00 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

==============================================================================
=======
   Section Shear Capacity
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Vy*  =     0.0000E+00 KN
   φVvy  =     0.0000E+00 KN               [AS 4100 5.11.2]
   φVvmy =     0.0000E+00 KN               [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
   φVvz  =     0.0000E+00 KN               [AS 4100 5.11.2]
   φVvmz =     0.0000E+00 KN               [AS 4100 5.12.3]
      STAAD SPACE                                                   -- PAGE NO.
5
   AXIAL

==============================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*          Critical Ratio:   1.316
   Critical Location:        0.317 m from Start.
   N*    =     1.0300E+03 KN
   Ae    =     3.8100E+03 mm2              [AS 4100 6.2.3 / 6.2.4]
   kf    =     1.000                       [AS 4100 6.2.2]
   An    =     3.8100E+03 mm2
   φNs   =   857.2499E+00 KN               [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =      3.80 m
   Ly    =      3.80 m
   Lez   =      3.80 m
   Ley   =      3.80 m
   αb    =     -0.50                       [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    47.979                       [AS 4100 6.3.3]
   αa,z  =    20.013                       [AS 4100 6.3.3]
   λ,z   =    37.972                       [AS 4100 6.3.3]
    h ,z   =     0.080                       [AS 4100 6.3.3]
    x ,z   =     3.533                       [AS 4100 6.3.3]
   αc,z  =     0.913                       [AS 4100 6.3.3]
   φNcz  = 0.7827E+3 KN                    [AS 4100 6.3.3]
   λn,y  =    47.979                       [AS 4100 6.3.3]
   αa,y  =    20.013                       [AS 4100 6.3.3]
   λ,y   =    37.972                       [AS 4100 6.3.3]
    h ,y   =     0.080                       [AS 4100 6.3.3]
    x ,y   =     3.533                       [AS 4100 6.3.3]
   αc,y  =     0.913                       [AS 4100 6.3.3]
   φNcy  = 0.7827E+3 KN                    [AS 4100 6.3.3]
   φNc   = N/A                             [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*          Critical Ratio:   0.000
```

```
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =     3.8100E+03 mm2
   φNt   =   857.2499E+00 KN               [AS 4100 7.2]
      STAAD SPACE                                            -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

==============================================================================
=======
   Section Combined Capacity
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   N* =     0.0000E+00 KN    Mz* =     0.0000E+00 KNm      My* =     0.0000E
+00 KNm
   φNs   =     0.0000E+00 KN               [AS 4100 8.3.1]
   φMsz  =     0.0000E+00 KNm
   φMsy  =     0.0000E+00 KNm
   φMrz  =     0.0000E+00 KNm              [AS 4100 8.3.2]
   φMry  =     0.0000E+00 KNm              [AS 4100 8.3.3]
   Member Combined Capacity - In-plane
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case: N/A            Critical Ratio: N/A
   Critical Location: N/A
      STAAD SPACE                                            -- PAGE NO.
7

******************************************************************************
**
   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. AS4100 1998 - Single Angle Section in Tension

To check the tensile capacity of a single angle per AS4100-1998.

Reference

Gorenc,B., R.Tinyou and A.Syam, *Steel Designer's Handbook*, 8th edition, UNSW Press, problem 7.2, page 198.

Related Links

- *D2.B. Australian Codes - Steel Design per AS 4100 - 1998* (on page 1356)

Problem

Select a section for a diagonal member of a truss. Verify its capacity. Use on equal leg angle in Grade 300 steel with one line of M20/S fasteners in one leg. Axial load of 300 kN is applied.

Comparison

Table 595: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Applied tensile load (kN)	300	300.3	0.1%
Tensile capacity, 0.9×Nt (kN)	492	492	none
Ratio	0.610	0.610	none

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Australia\AS4100 1998 - Single Angle Section in Tension.STD` is typically installed with the program.

```
STAAD SPACE AXIAL TENSILE CAPACITY PER AS4100 1998
*
* REFERENCE : Steel Designer's Handbook, B.Gorenc, R.Tinyou and A.Syam,
*             8th edition, example problem 7.2, page 198
*
*  OBJECTIVE : TO DETERMINE THE CAPACITY OF A SINGLE ANGLE IN TENSION
*              PER THE AS4100-1998 CODE
*
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+008
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-006
DAMP 0.03
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST A125X125X8
PRINT MEMBER PROPERTY ALL
*
CONSTANTS
```

```
MATERIAL STEEL MEMB 1
*
SUPPORTS
1 FIXED
*****************************************
LOAD 1 LOADTYPE DEAD TITLE PERMANENT LOAD
JOINT LOAD
2 FX 99.0
*
LOAD 2 LOADTYPE LIVE TITLE IMPOSED LOAD
JOINT LOAD
2 FX 121
*
LOAD COMBINATION 3 ULS
1 1.2 2 1.5
*
PERFORM ANALYSIS
UNIT MMS NEWTON
*
PARAMETER 1
CODE AUSTRALIAN 1998
*
* KT from AS 4100-1998 Table 7.3.2 case (i)
KT 0.85 MEMB 1
*
* NSF = An/Ag = 1720/1900
NSF 0.905 MEMB 1
***
TRACK 2.0 ALL
CHECK CODE MEMB 1
*
* UTILIZATION RATIO = 300/492 = 0.610
*
FINISH
```

STAAD Output

```
    *
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL             BE USED FOR ROLLED
SECTIONS.
  |--------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES   |
  |*************                                       |        IN  CM  UNIT |
  |            *  |=============================|   ==| |==     ------------ |
  |MEMBER   1  *  |  AUSTRALIAN SECTIONS        |     | |       AX=0.1900E+2 |
  |            *  | ST  A125X125X8              |     | |  --Z  AY=0.6500E+1 |
  |DESIGN CODE *  |                             |     | |       AZ=0.6500E+1 |
  |AS4100 1998 *  ===============================   ==| |==     PY=0.8091E+2 |
  |            *                                                PZ=0.4058E+2 |
  |            *  |<---LENGTH (ME=     4.00 --->|                  RY=0.4931E
+1 |
  |*************                                                RZ=0.2480E+1 |
  |                                                             Iw=0.0000E+0 |
  |                  0.0( KN-METR)                                           |
```

```
  |PARAMETER        |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  FORCE/MOMENT
|
  |IN  NEWTON MM    |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  IN  KN METRE
|
  |---------------  +�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0
-------------|
  | KL/R-Y=   81.1  |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  PNC=0.9995E
+2|
  | KL/R-Z=  161.3  +�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  PNT=0.4920E
+3|
  | UNL   = 4000.0  |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  pn =-.3003E
+3|
  | MAIN  =  180.0  +�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  MNZ=0.6846E
+1|
  | PHI   =   0.90  |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  mnz=0.0000E
+0|
  | FULT  =  440.0  +�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  MNY=0.1828E
+2|
  | FYLD  =  320.0  |�0 �0  �0  �0  �0  �0  �0  �0  �0  �0  �0  mny=0.0000E
+0|
  | NSF   =   0.90  +---+---+---+---+---+---+---+---+---+---|   VZ =0.1123E+3|
  | SKT   =   1.00    0.0                                      vz =0.0000E+0|
  | SKL   =   1.00               ABSOLUTE MZ ENVELOPE          VY =0.1123E+3|
  | SKR   =   1.00                  (WITH LOAD NO.)            vy =0.0000E+0|
  |                                                                         |
  | Section Type:  Noncompact - about Z axis;  Noncompact - about Y axis    |
  |                                                                         |
  |                     Parameters used to calculate RATIO                  |
  |                     ----------------------------------                  |
  | Ns=0.5349E+3  Msz=0.9458E+1  Msy=0.2031E+2  Mbz=0.7607E+1               |
  |               Miz=0.0000E+0  Miy=0.0000E+0  Moz=0.0000E+0               |
  |              Nciz=0.1111E+3 Nciy=0.2725E+3  Ncz=0.1111E+3  Ncy=0.2725E+3 |
  |              Vvmy=0.4942E-2 Vvmz=0.4942E-2                              |
  | ALPHA,M= 1.000  ALPHA,B= 1.000  ALPHA,SZ= 0.804                         |
  |                                                                         |
  |                       MAX FORCE/ MOMENT SUMMARY ( KN-METR)              |
  |                       -------------------------                         |
  |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
  |      VALUE    -300.3        0.0        0.0         0.0         0.0      |
  |   LOCATION       4.0        0.0        0.0         0.0         0.0      |
  |    LOADING        3          0          0           0           0       |
  |*************************************************************************|
  |*                           DESIGN SUMMARY ( KN-METR)                   *|
  |*                           --------------                              *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/           *|
  |*         FX              MY             MZ          LOCATION           *|
  |*        =====================================================          *|
  |*          PASS        TENSION          0.610            3              *|
  |*        300.30 T          0.0          0.0            0.00             *|
  |*************************************************************************|
  |-------------------------------------------------------------------------|
     AXIAL TENSILE CAPACITY PER AS4100 1998                  -- PAGE NO.
5
    *
```

V. AS4100 1998 - Welded section subject to bending

Verify the bending capacity of a welded girder per the AS 4100 1998 code.

Details

The member is 21 m in length, with pinned supports at both ends. The section has top & bottom flanges of 450 mm × 60 mm and a web of 1,390 mm × 32 mm. The plates are all grade 280 steel. The member is subject to major axis bending. The section is partially restrained at the supports and at the third points along the span. The point loads are applied at the top flange.

Material Properties

E = 199,947 MPa
G = 80,000 MPa
F_y = 280 MPa
F_u = 440 MPa

Validation

Section Properties

Depth of section, $D = 1{,}390 + 2 \times 60 = 1{,}510\ mm$
Cross-sectional area, $A_g = 1{,}390 \times 32 + 2 \times 450 \times 60 = 98{,}480\ mm^2$
Moment of inertia about major axis:

$$I_{zz} = 2\left[\frac{b_f {t_f}^3}{12} + b_f t_f \left(\frac{d_w + t_f}{2}\right)^2\right] + \frac{t_w {d_w}^3}{12} = 2\left[\frac{450(60)^3}{12} + 450 \times 60\left(\frac{1{,}390 + 60}{2}\right)^2\right] + \frac{32(1{,}390)^3}{12} = 35.6(10)^9\ \mathrm{mm}$$

Moment of inertia about minor axis:

$$I_{yy} = \frac{d_w {t_w}^3}{12} + 2\frac{t_f {b_f}^3}{12} = \frac{1{,}390 \times (32)^3}{12} + 2\frac{60(450)^3}{12} = 915(10)^6\ \mathrm{mm}^4$$

Elastic section modulus, $Z_{xx} = \frac{I_{xx}}{D/2} = \frac{35.6(10)^9}{1{,}510 / 2} = 47.1(10)^6\ mm^3$

Plastic section modulus,

$$S_{xx} = 2\left[\frac{{d_w}^2 t_w}{8} + b_f t_f \left(\frac{d_w - t_f}{2}\right)\right] = 2\left[\frac{(1{,}390)^2 32}{8} + 450 \times 60\left(\frac{1{,}390 - 60}{2}\right)\right] = 54.6(10)^6\ \mathrm{mm}^3$$

Warping constant:

$$I_w = \frac{I_y (d_w + t_f)^2}{4} = \frac{915(10)^6 (1{,}390 + 60)^2}{4} = 481(10)^{12}\ \mathrm{mm}^6$$

Torsional constant:

$$J = \frac{2 b_f {t_f}^3 + d_w {t_w}^3}{3} = \frac{2(450)(60)^3 + 1{,}390(32)^3}{3} = 80.0(10)^6\ \mathrm{mm}^4$$

Section Slenderness Ratio

Flange section slenderness parameter:

$$\frac{b}{2} = \frac{b_f - t_w}{2} = \frac{450 - 32}{2} = 209 \text{ mm}$$

$$\lambda_{ef} = \frac{b}{t_f}\sqrt{\frac{f_y}{250}} = \frac{209}{60}\sqrt{\frac{280}{250}} = 3.69 < \lambda_{ep} = 8$$

Hence, the flange is compact.

Web section slenderness parameter:

$$\lambda_{ew} = \frac{d_w}{t_w}\sqrt{\frac{f_y}{250}} = \frac{1{,}390}{32}\sqrt{\frac{280}{250}} = 46.0 < \lambda_{ep} = 82$$

Hence, the web is compact.

<u>Section Bending Capacity</u>

Z_{cx} is the lesser of S_{xx} and $1.5 \times Z_{xx} = 1.5(47.1 \times 10^6) = 70.7(10^6)$.

$$M_{sx} = Z_{cx} f_y = 54.6(10)^6 280(10)^{-6} = 15{,}290 \text{ kN·m}$$

$$\phi M_{sx} = 0.9 \times 15{,}290 = 13{,}760 \text{ kN·m}$$

<u>Member Bending Capacity</u>

The length between braced points, $l = 7{,}000$ *mm*

$$k_t = 1 + \left(\frac{2d}{l}\right)\left(\frac{0.5t_f}{t_w}\right)^3 = 1 + \left(\frac{2 \times 1{,}390}{7{,}000}\right)\left(\frac{0.5 \times 60}{32}\right)^3 = 1.33$$

Top flange load height position: $k_l = 1.4$. No lateral rotational restraint: $k_r = 1.0$.

The effective length, $l_e = k_t k_l k_r l = 1.33 \times 1.4 \times 1.0 \times 7{,}000 = 13{,}000$ *mm*.

The reference buckling moment:

$$M_o = \sqrt{\frac{\pi^2 EI_y}{l_e^2}\left[GJ + \frac{\pi^2 EI_w}{l_e^2}\right]} = \frac{3{,}800}{79.2}\sqrt{\frac{1 \times 250}{250}}$$

$$= \sqrt{\frac{\pi^2 \times 199{,}947 \times 915(10)^6}{(13{,}000)^2}\left[80{,}000 \times 80(10)^6 + \frac{\pi^2 \times 199{,}947 \times 481(10)^{12}}{(13{,}000)^2}\right]}(10)^{-6} = 11{,}330 \text{ kN·m}$$

The moment modification factor:

$$a_m = \frac{1.7M_m}{\sqrt{(M_2)^2 + (M_3)^2 + (M_4)^2}}$$

where

M_M	=	the maximum design moment in the segment, = $M_3 = 6{,}142.5$ *kN·m* (from STAAD.Pro analysis)
$M_2, = M_4$	=	the design moment at the quarter points in the segment, = $5{,}862.5$ *kN·m* (from STAAD.Pro analysis)

$$a_m = \frac{1.7 \times 6{,}142.5}{\sqrt{2 \times (5{,}862.5)^2 + (6{,}142.5)^2}} = 1.012$$

The slenderness reduction factor per AS 4100 Eq. 5.6.1.1(2):

$$a_s = 0.6\left[\sqrt{\frac{{M_s}^2}{{M_o}^2} + 3} - \frac{M_s}{M_o}\right] = 0.6\left[\sqrt{\frac{(15{,}290)^2}{(11{,}330)^2} + 3} - \frac{15{,}290}{11{,}330}\right] = 0.508$$

$$\phi M_b = \phi a_m a_s M_s = 0.9 \times 1.012 \times 0.508 \times 15{,}290 = 7{,}074 \text{ kN·m}$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Nominal section capacity major axis, M_{sx}(kN·m)	13,760	13,760.9	negligible	
Member bending capacity, M_{bx} (kN·m)	7,074	7,072	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Australia\AS4100 1998 - Welded section subject to bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jul-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.00001 0 0; 3 14 0 0; 4 21 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE PMEMBER
1 TO 3 PMEMBER 1
START USER TABLE
TABLE 2
UNIT METER KN
WIDE FLANGE
1510X450X60X32
0.09848 1.51 0.032 0.45 0.06 0.0355616 0.000915046 7.99825e-05 0.04832 -
0.036 0.45 0.06
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+08
POISSON 0.3
DENSITY 76.8191
ALPHA 6.5e-06
DAMP 0.03
G 7.7221e+07
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY
```

```
1 TO 3 UPTABLE 2 1510X450X60X32
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
4 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 TO 3 UNI GY -40
JOINT LOAD
2 3 FY -250
MEMBER LOAD
1 TO 3 CON GY -250
PERFORM ANALYSIS
PARAMETER 1
CODE AUSTRALIAN
TRACK 2 PMEMB 1
LHT 1 PMEMB 1
IST 4 PMEMB 1
PBRACE TOP 0 P 0.333 P 0.666 P 1 P PMEMB 1
PBRACE BOTTOM 0 P 0.333 P 0.666 P 1 P PMEMB 1
CHECK CODE PMEMB ALL
FINISH
```

STAAD.Pro Output

```
                    STAAD.Pro CODE CHECKING - (  AS4100-1998   )
V2.3
                    ****************************************************
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [AS4100 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per AS4100 1.4.
   DESIGN SUMMARY

===============================================================================
=======
   Designation: ST   1510X450X60X32           (UPT)
   Governing Load Case:     1*
   Governing Criteria: AS-8.4.4.2
   Governing Ratio:   0.869  (PASS)
   SECTION PROPERTIES

===============================================================================
=======
    d:      1510.0000 mm   bf:       450.0000 mm
   tf:        60.0000 mm   tw:        32.0000 mm
   Ag:     98480.0000 mm2   J:    79.9825E+06 mm4            Iw:   480.9710E
+12 mm6
   Iz:    35.5616E+09 mm4  Sz:    54.6068E+06 mm3 (plastic)   Zz:    47.1015E
+06 mm3 (elastic)
```

```
   rz:   600.9199E+00 mm
   Iy:   915.0460E+06 mm4  Sy:     6.4308E+06 mm3 (plastic)   Zy:     4.0669E
+06 mm3 (elastic)
   ry:    96.3934E+00 mm
   MATERIAL PROPERTIES

==============================================================================
=======
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  LW (Lightly welded longitudinally)
   E (#)      :199947.000 MPa    [AS 4100 1.4]
   G          : 80000.000 MPa    [AS 4100 1.4]
   fy, flange :   280.000 MPa    [AS 4100 Table 2.1]
   fy, web    :   280.000 MPa    [AS 4100 Table 2.1]
   fu         :   440.000 MPa    [AS 4100 Table 2.1]
   SLENDERNESS

==============================================================================
=======
   Actual slenderness   : 217.857
   Allowable slenderness: 400.000
      STAAD SPACE                                              -- PAGE NO.
4
   BENDING

==============================================================================
=======
   Section Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.446
   Critical Location:       10.500 m from Start.
   Mz* =    -6.1425E+03 KNm                      My* =     0.0000E+00 KNm
   Z-Axis Section Slenderness: Compact           Y-Axis Section Slenderness:
Compact
   Zez =    54.6068E+06 mm3                      Zey =     6.1003E+06 mm3
   φMsz =    13.7609E+03 KNm                     φMsy =     1.5373E+03 KN[AS
4100 5.2.1]
   Member Bending Capacity
   Critical Load Case:     1*          Critical Ratio:   0.869
   Critical Location:       10.500 m from Start.
   Crtiical Segment/Sub-segment:
   Location (Type):   6.99 m(P )- 13.99 m(P )
   Length:    6.99 m
   Mz* =    -6.1425E+03 KNm                      My* =     0.0000E+00 KNm
   kt   =      1.33                        [AS4100 Table 5.6.3(1)]
   kl   =      1.40                        [AS4100 Table 5.6.3(2)]
   kr   =      1.00                        [AS4100 Table 5.6.3(3)]
   le   =     13.00 m                      [AS4100 5.6.3]
   αm   =     1.012                        [AS4100 5.6.1.1(a)(iii)]
   Mo   =    11.3341E+03 KNm               [AS4100 5.6.1.1(a)(iv)]
   αsz  =     0.508                        [AS4100 5.6.1.1(a)(iv)]
   φMbz =     7.0722E+03 KNm (&lt;= φMsz)      [AS4100 5.6.1.1(a)]
   SHEAR

==============================================================================
=======
   Section Shear Capacity
   Critical Load Case:     1*          Critical Ratio:   0.152
```

```
   Critical Location:        0.583 m from Start.
   Vy*  =     1.0217E+03 KN
   φVvy  =     6.7254E+03 KN               [AS 4100 5.11.2]
   φVvmy =     6.7254E+03 KN               [AS 4100 5.12.3]
   Vz*  =     0.0000E+00 KN
   φVvz  =     8.1648E+03 KN               [AS 4100 5.11.2]
   φVvmz =     8.1648E+03 KN               [AS 4100 5.12.3]
      STAAD SPACE                                          -- PAGE NO.
5
   AXIAL

==============================================================================
=======
   Section Compression Capacity
   Critical Load Case:     1*         Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =    92.7036E+03 mm2              [AS 4100 6.2.3 / 6.2.4]
   kf    =     0.941                       [AS 4100 6.2.2]
   An    =    98.4800E+03 mm2
   φNs   =    23.3613E+03 KN               [AS 4100 6.2.1]
   Member Compression Capacity
   Lz    =     21.00 m
   Ly    =     21.00 m
   Lez   =     21.00 m
   Ley   =     21.00 m
   αb    =      1.00                       [AS 4100 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    35.883                       [AS 4100 6.3.3]
   αa,z  =    16.856                       [AS 4100 6.3.3]
   λ,z   =    52.739                       [AS 4100 6.3.3]
    h ,z   =     0.128                       [AS 4100 6.3.3]
    x ,z   =     2.142                       [AS 4100 6.3.3]
   αc,z  =     0.847                       [AS 4100 6.3.3]
   φNcz  = 0.1979E+5 KN                    [AS 4100 6.3.3]
   λn,y  =   223.694                       [AS 4100 6.3.3]
   αa,y  =     9.070                       [AS 4100 6.3.3]
   λ,y   =   232.764                       [AS 4100 6.3.3]
    h ,y   =     0.715                       [AS 4100 6.3.3]
    x ,y   =     0.628                       [AS 4100 6.3.3]
   αc,y  =     0.133                       [AS 4100 6.3.3]
   φNcy  = 0.3109E+4 KN                    [AS 4100 6.3.3]
   φNc   = N/A                             [AS 4100 6.3.3 / AS 4600 3.4.1(b)]
   Section Tension Capacity
   Critical Load Case:     1*         Critical Ratio:   0.000
   Critical Location:        0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =    98.4800E+03 mm2
   φNt   =    24.8170E+03 KN               [AS 4100 7.2]
      STAAD SPACE                                          -- PAGE NO.
6
   COMBINED BENDING AND AXIAL

==============================================================================
=======
   Section Combined Capacity
   Critical Condition: Cl 8.3.2
   Critical Load Case:     1*         Critical Ratio:   0.446
```

```
   Critical Location:       10.500 m from Start.
   N* =     0.0000E+00 KN    Mz* =    -6.1425E+03 KNm      My* =     0.0000E
+00 KNm
   φNs   =    23.3613E+03 KN               [AS 4100 8.3.1]
   φMsz  =    13.7609E+03 KNm
   φMsy  =     1.5373E+03 KNm
   φMrz  =    13.7609E+03 KNm              [AS 4100 8.3.2]
   φMry  =     1.5373E+03 KNm              [AS 4100 8.3.3]
   Member Combined Capacity - In-plane
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Out-of-plane(tension)
   Critical Load Case:     1*          Critical Ratio:   0.869
   Critical Location:       10.500 m from Start.
   N* =     0.0000E+00 KN    Mz* =    -6.1425E+03 KNm      My* =     0.0000E
+00 KNm
   φMbz  =     7.0722E+03 KNm
   φNt   =    24.8170E+03 KN               [AS 4100 8.4.4.2]
   φMozt =     7.0722E+03 KNm              [AS 4100 8.4.4.2]
   Member Combined Capacity - Biaxial(compression)
   Critical Load Case: N/A             Critical Ratio: N/A
   Critical Location: N/A
   Member Combined Capacity - Biaxial(tension)
   Critical Load Case:     1*          Critical Ratio:   0.821
   Critical Location:       10.500 m from Start.
   N* =     0.0000E+00 KN    Mz* =    -6.1425E+03 KNm      My* =     0.0000E
+00 KNm
   φMtz  =     7.0722E+03 KNm              [AS 4100 8.4.5.2]
   φMry  =     1.5373E+03 KNm              [AS 4100 8.4.5.2]
      STAAD SPACE                                            -- PAGE NO.
7

******************************************************************************
**
   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. Canadian

V. CSA S16-01 - Axial Tension

Design of a double-angle member subject to tension load.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 3.1

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

Design a tension diagonal in an all-welded truss. Attached to WT 265x61.5 chords (t_w = 13.1 mm).

$L = 4$ m

$T_f = 630$ kN

The material used is G40.21 300W steel (F_y = 300 MPa, F_u = 450 MPa) (FYLD and FU)

Calculations

Gross area is used for welded connections.

$$T_r = \max \begin{cases} \phi A F_y = 0.9 \times 300 \times A = 270A \\ \phi_u A F_u = 0.75 \times 450 \times A = 338A \end{cases}$$

$T_r \geq T_f$, so solve for A:

$$A = \frac{630 \times (10)^3}{270} = 2,\ 333\text{mm}^2$$

Select a pair of angles with at least this gross area: try a 2- 76x64x9.5, with long legs back-to-back:

$A = 2{,}480\ \text{mm}^2$

$r_{min} = 13.3$ mm

Calculate the effective net area due to shear lag. The average weld length is taken as *(120 mm + 250 mm)/2 = 185 mm*.

The long leg does not have a reduction in area shear lag, thus:

$A_{n2} = 1.00\times(76 - 9.5) \times 9.5 = 632\ \text{mm}^2$

The outstanding (short) leg has an eccentricity of half the leg length, or *64/2 = 32 mm*.

$$A_{n3} = \left(1 - \frac{32}{250}\right) \times 64 \times 9.5 = 530\text{mm}^2$$

The total net area for the pair of angles for the rupture limit state is:

$A_{ne} = 632 + 530 = 1{,}162\ \text{mm}^2$

For the pair of angles, $A_{ne} = 2\times1{,}1162 = 2{,}324\ mm^2$; the ratio of net section area to gross area = 2,324 / 2,480 = 0.937 (NSF).

Checking the yielding limit state:

$T_r = 0.90\times2{,}480\times300 = 670$ kN

Checking the rupture limit state:

$T_r = 0.75\times2{,}324\times450 = 784$ kN

Yielding governs, and the critical ratio is: *630/670 = 0.94*.

Comparison

Table 596: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
T_f (kN)	670	669.6	negligible
Critical Ratio	0.94	0.941	negligible

Note: If the member SELECT facility is used in place of a code CHECK, STAAD.Pro will actually select a 2-L 102x89x6.4 member, which has a critical ratio of 0.997 (per Cl. 13.2) and an area of 2,340 mm^2, thus more economical (the NSF used for this size is 0.923).

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Axial Tension.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE LD L76X64X9.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2001
```

```
FYLD 300000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L76X64X9.5               (CANADIAN SECTIONS)
                           PASS     CSA-13.9.A         0.941         1
                      630.00 T          0.00           0.00        4.00
      STAAD SPACE                                              -- PAGE NO.
6
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  2.48E+01   MEMBER LENGTH =  4.00E+02
   IZ =  1.38E+02   SZ =  2.66E+01   PZ =  4.78E+01
   IY =  1.67E+02   SY =  2.62E+01   PY =  4.72E+01
   IX =  7.51E+00   CW =  0.00E+00
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 450.0   E =  2.05E+05   G =  7.88E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  6.696E+02   CR2 =  1.421E+02   SECTION CLASS 3
   CRZ =  1.421E+02   CTORFLX =  1.421E+02
   TENSILE CAPACITY     =  6.696E+02   COMPRESSIVE CAPACITY =  1.421E+02
   FACTORED MOMENT RESISTANCE : MRY =  7.082E+00   MRZ =  7.176E+00
                                MU =  3.531E+01
   FACTORED SHEAR RESISTANCE  : VRY =  2.588E+02   VRZ =  2.038E+02
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =  154.350   KL/RZ =  169.588   ALLOWABLE KL/R =  300.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  4.000
   OMEGA-1 (Y-AXIS) = 0.00   OMEGA-1 (Z-AXIS) = 0.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  7.00E+00
    41. FINISH
      STAAD SPACE                                              -- PAGE NO.
7
             *********** END OF THE STAAD.Pro RUN ***********
```

V. CSA S16-01 - Beam Shear Capacity

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.3

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

The W530x82 beam has a maximum shear force, $v = 540\ kN$ due to factored loads.

$$L = 11.0 \text{ m}$$

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`)

Calculations

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{528 - 2(13.3)}{9.5} = 52.8 < 439\sqrt{\frac{k_v}{F_y}} = 439\sqrt{\frac{5.34}{350}} = 54.2$$

Therefore the shear capacity is calculated as:

$$V_r = \phi \times A_w \times F_y = 0.9 \times 528 \times 9.5 \times 350 = 1{,}043 \text{ kN}$$

Calculate Equivalent Concentrated Load

The given end moments and assumed distributed loads (which was determined to given an equivalent mid-span moment of the given example) do not result in a shear force of 540 kN. Therefore, an additional concentrated load is added near the first support.

$$R_1 = \frac{w_f L}{2} + \frac{M_1 - M_2}{L} = \frac{40.17 \times 11.0}{2} + \frac{540 - 185}{11.0} = 253.2 \text{ kN}$$

$$P = 540 \text{ kN} - 253.2 \text{ kN} = 287 \text{ kN}$$

The STAAD.Pro will have this load applied at 1 mm away from the left support.

Comparison

Table 597: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
V_r (kN)	1,043	1,042.826	negligible

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Beam Shear Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2001
FYLD 350000 ALL
FU 450000 ALL
SHE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
```

```
                      ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W530X82                  (CANADIAN SECTIONS)
                           FAIL     CSA-13.8.2+       4.092         1
                       0.00 C          0.00        540.00        0.00
      STAAD SPACE                                               -- PAGE NO.
6
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  1.05E+02   MEMBER LENGTH =  1.10E+03
   IZ =  4.77E+04   SZ =  1.81E+03   PZ =  2.06E+03
   IY =  2.03E+03   SY =  1.94E+02   PY =  3.03E+02
   IX =  5.18E+01   CW =  1.34E+06
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 350.0   FU = 450.0   E =  2.05E+05   G =  7.88E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  3.308E+03   CR2 =  2.965E+02   SECTION CLASS 2
   CRZ =  2.638E+03   CTORFLX =  2.965E+02
   TENSILE CAPACITY     =  3.308E+03   COMPRESSIVE CAPACITY =  2.965E+02
   FACTORED MOMENT RESISTANCE : MRY =  9.545E+01   MRZ =  1.319E+02
                               MU =  1.466E+02
   FACTORED SHEAR RESISTANCE  : VRY =  1.043E+03   VRZ =  7.705E+02
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =  250.172   KL/RZ =   51.609   ALLOWABLE KL/R =  300.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) = 11.000
   OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  2.542E+02   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  5.28E+01
```

V. CSA S16-01 - Cantilever with Biaxial Loading

A slender, cantilever beam subjected to a uniform load. Static analysis, 3D beam element.

Reference

CISC Example 1, page 5-91, Limit State Design, CSA-S16.1-94

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

A cantilever beam of length 4 meter is subjected to uniformly distributed load of 3 kN/Meter in both major and minor axis. Axial compression of 8 kN is also applied to the member. A user-defined steel section "Sect_Class-4" is assigned to the member.

Given

Design forces

8.0 kN (Compression)
6.0 kN·m (Bending-Y)
6.0 kN·m (Bending-Z)
6.0 kN (Shear-Y)
6.0 kN (Shear-Z)

Section Properties(Sect_Class-4):

Area = 2,766 mm^2
Depth of section, D = 150 mm
Thickness of web T_w = 7 mm
Width of flange B_f = 150 mm
Thickness of flange T_f = 6 mm
Moment of inertia about Z axis, $I_z = 10.87\times10^6$ mm^4
Moment of inertia about Y axis, $I_y = 3.38\times10^6$ mm^4
Moment of inertia about X axis, $I_x = 37.38\times10^3$ mm^4
Warping constant, $C_w = 17.52\times10^9$ mm^6
Torsion constant, $J = \frac{2B_f T_f^{\,3} + \left(D - T_f\right)t_w^{\,3}}{3} = \frac{2 \times 150 \times 6^3 + \left(150 - 6\right)7^3}{3} = 38.06 \times 10^3\ mm^4$

Member Length L = 2 m, Unbraced length = 100mm.

Material

Fy (FYLD) = 300 MPa
E = 205,000 MPa
G = 78,850 MPa

Calculations

Slenderness Ratio

Effective Length factor along Local Y-Axis = KY = 1

Effective Length factor along Local Z-Axis = KZ = 1

Slenderness ratio about Z axis, L/Rz = 31.9

Slenderness ratio about Y axis, L/Ry = 57.22

Maximum Slenderness Ratio, L/Rmax = 57.22

Section Classification

$$B_f / T_f = 150 \times 0.5 / 6 = 12.5 > \frac{200}{\sqrt{F_y}} = 11.54$$

Flange is Class 4.

$$d/T_w = (150 - 2.0\times6)/7 = 19.7$$

$$\frac{1{,}100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{\phi C_y}\right) = \frac{1{,}100}{\sqrt{300}}\left(1 - 0.39\frac{8{,}000}{0.9 \times 2{,}766 \times 300}\right) = 63.5$$

Web is Class 1.

Overall section is Class 4 section.

Check against axial compression (Clause 13.3.3)

Effective width, $B_{eff} = 200 \times \frac{T_f}{\sqrt{F_y}} = 69.24$

Effective area, $A_{eff} = 69.24 \times 6 \times 4 + (150 - 2 \times 6) \times 7 = 2{,}628\ mm^4$.

Effective yield stress, $F_{y,eff} = \frac{40{,}000}{(0.5 \times B_f\ /\ T_f)^4} = 256\ MPa$.

As per Clause 13.3.3(a),

Elastic critical buckling, Fe = $\pi^4\times$E/ L_Rmax^4 = 617.956 MPa.

Non-dimensional slenderness ratio, $\lambda = \sqrt{F_y / F_e} = 0.697$

Axial compressive resistance, $C_r = \phi A_{eff} \times F_y \times \left[1 + 0.697^{(2\times1.34)}\right]^{(-1/1.34)} = 557.9\ kN$.

As per Clause 13.3.3(b),

Elastic critical buckling, Fe = $\pi^4\times$E/ L_Rmax^4 = 617.956 MPa.

Effective non-dimensional slenderness ratio, $\lambda = \sqrt{F_y / F_e} = 0.697$

Axial compressive resistance, $C_r = \phi \times A \times F_{y,eff} \times \left[1 + 0.644^{(2\times1.34)}\right]^{(-1/1.34)} = 521.8\ kN$.

Axial compressive resistance = $min(557.9,\ 521.8) = 521.8\ kN$.

Check against bending (Clause 13.5(c))

As the web of the section meets the requirement of Class 3 and flange exceeds Class 3 limit, flexural resistance should be calculated as per clause 13.5(c).iii.

Effective moment of inertia about Z axis,

$$I_{z,eff} = 2 \times \left[\frac{2 \times 69.24 \times 6^3}{12} + 2 \times \frac{2 \times 69.24 \times 6 \times (150 - 6)^2}{4}\right] + \frac{7 \times (150 - 2 \times 6)^3}{12} = 10.15 \times (10)^6\ \mathrm{mm}^4$$

Effective section modulus about Z axis,

$$S_{z,eff} = 10.15 \times (10)^6 \times 2 / 150 = 135.4 \times (10)^3\ \mathrm{mm}^3$$

Effective moment of inertia about Y axis,

$$I_{y,eff} = \frac{2 \times 6 \times (2 \times 69.24)^3}{12} + \frac{0.5\left(150 - 6\right)7^3}{12} = 2.658 \times (10)^6 \text{ mm}^4$$

Effective section modulus about Y axis,

$$S_{y,eff} = 2.658 \times (10)^6 / 69.24 = 38.38 \times (10)^3 \text{ mm}^3$$

Major axis bending resistance if member is laterally supported,

$$M_{rz1} = \phi S_{z,eff} F_y = 0.9 \times 135.4 \times (10)^3 \times 300 = 36.55 \text{ kN} \cdot \text{m}$$

Minor axis bending resistance,

$$M_{ry} = \phi S_{y,eff} F_y = 0.9 \times 38.38 \times (10)^3 \times 300 = 10.36 \text{ kN} \cdot \text{m}$$

If the member is laterally unsupported major axis bending resistance is determined by clause 13.6(b).

As the value of one of the end moments is 0.0, $\omega_2 = 1.75$.

Where, as per clause 13.6(a),

$$EI_y GJ = 205{,}000 \times 38.8(10)^6 \times 78{,}850 \times 38.06(10)^3 = 23.87 \times 10^{21} \text{ MPa}^2 \cdot \text{mm}^8$$

$$\left(\frac{\pi E}{L}\right)^2 I_y C_w = \left(\frac{\pi \times 205{,}000}{2{,}000}\right)^2 3.38 \times 10^6 \times 17.52 \times 10^9 = 6.14 \times 10^{21} \text{ MPa}^2 \cdot \text{mm}^8$$

$$M_u = \frac{\omega_2 \pi}{L} \sqrt{E I_y G J + \left(\frac{\pi E}{L}\right)^2 I_y C_w}$$

$$= \frac{1.75 \times \pi}{2{,}000} \sqrt{23.87 \times 10^{21} + 6.14 \times 10^{21}} = 476.2 \times 10^6 \text{ N} \cdot \text{mm} = 476.2 \text{ kN} \cdot \text{m}$$

$$M_y = S_z F_y = \frac{10.87(10)^6 \times 2}{150} 300 = 43.48 \text{ kN} \cdot \text{m}$$

Since $M_u > 0.65 \times M_y = 28.26$, Moment of resistance

$$M_{rz2} = 1.15 \times 0.9 \times 43.48 \times \left(1 - 0.28 \frac{43.48}{248}\right) = 42.79 \text{kN} \cdot \text{m}$$

Mrz2 should not be more than Mrz1. Since, Mrz2 > Mrz1 in this example, Mrz2 = Mrz1.

$$M_{rz2} = 36.55 \text{ kN-m}$$

Comparison

Table 598: Comparison of results

Criteria	Hand Calculation	STAAD.Pro Result	Comments
Axial compressive resistance (kN)	521.7	521.9	negligible
Major axis bending resistance (kN-m)	36.55	36.57	negligible
Minor axis bending resistance (kN-m)	10.36	10.38	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Cantilever with Biaxial Loading.STD` is typically installed with the program.

```
STAAD SPACE VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91
START JOB INFORMATION
ENGINEER DATE 16-FEB-10
END JOB INFORMATION
* CISC EXAMPLE 1 PAGE 5-91, LIMIT STATES DESIGN, CSA-S16.1-94
* SIMPLY SUPPORTED BEAM WITH UNIFORM LOAD
* LIVE LOAD DEFLECTION OF L/300
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 2000 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER NEWTON
WIDE FLANGE
SECT_CLASS-4
0.002766 0.15 0.007 0.15 0.006 1.08696e-05 3.37894e-06 3.7378e-08 -
0.00105 0.0018
END
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.05e+08
POISSON 0.3
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 SECT_CLASS-4
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
UNIT METER KN
LOAD 1 LC1
MEMBER LOAD
1 UNI GY -3
1 UNI GZ -3
JOINT LOAD
2 FX -8
PERFORM ANALYSIS
LOAD LIST 1
PRINT MEMBER FORCES ALL
PARAMETER 1
CODE CANADIAN 2001
CB 0 ALL
```

```
TRACK 2 ALL
FYLD 300000 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

The design output from STAAD.Pro:

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   SECT_CLASS-4             (UPT)
                           PASS     CSA-13.8.3C        0.760         1
                        8.00 C         -6.00          6.00        0.00
     VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91        -- PAGE NO.
5
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  2.77E+01   MEMBER LENGTH =  2.00E+02
   IZ =  1.09E+03   SZ =  1.45E+02   PZ =  1.63E+02
   IY =  3.38E+02   SY =  4.51E+01   PY =  6.92E+01
   IX =  3.74E+00   CW =  1.75E+04
   EFFECTIVE MEMBER PROPERTIES FOR CLASS-4 SECTION(UNIT = CM)
   ----------------------------------------------------------
   EFFECTIVE CROSS SECTION AREA =  2.63E+01
   EFFECTIVE IZ =  1.02E+03   EFFECTIVE SZ =  1.35E+02
   EFFECTIVE IY =  2.66E+02   EFFECTIVE SY =  3.85E+01
   EFFECTIVE YIELD STRESS = 256.0  MPA
   COMPRESSIVE CAPACITIES FOR CLASS 4 SECTION(UNIT = MPA)
   ------------------------------------------------------
   BASED ON EFFECTIVE AREA
   CR1 =  7.098E+02   CR2 =  5.582E+02   CRZ =  6.705E+02
   CTORFLX =  5.582E+02
   BASED ON EFFECTIVE YIELD STRENGTH
   CR1 =  6.373E+02   CR2 =  5.219E+02   CRZ =  6.084E+02
   CTORFLX =  5.219E+02
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 345.0   E =  2.05E+05   G =  7.88E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  6.373E+02   CR2 =  5.219E+02   SECTION CLASS 4
   CRZ =  6.084E+02   CTORFLX =  5.219E+02
   TENSILE CAPACITY     =  7.300E+02   COMPRESSIVE CAPACITY =  5.219E+02
   FACTORED MOMENT RESISTANCE : MRY =  1.038E+01   MRZ =  3.657E+01
                              MU =  2.486E+02
   FACTORED SHEAR RESISTANCE  : VRY =  1.871E+02   VRZ =  3.208E+02
   MISCELLANEOUS INFORMATION
   --------------------------
```

```
NET SECTION FACTOR FOR TENSION =  1.000
KL/RY =   57.222   KL/RZ =   31.904   ALLOWABLE KL/R =  200.000
UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  2.000
OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.75
SHEAR FORCE (KNS) : Y AXIS =  6.000E+00   Z AXIS =  6.000E+00
SLENDERNESS RATIO OF WEB (H/W) =  1.97E+01
```

V. CSA S16-01 - Shear Capacity Combined Stresses

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 8.6

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

The W250x73 column has the following concentrated loads

$C_f = 900$ kN

And a uniform moment of:

$M_{fx} = 180$ kN·m

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`). The beam is braced against lateral-torsional buckling only at the ends (`LAT 0`, default)

Calculations

Axial Capacity

Determine the buckling load of the column:

$$\left(\frac{KL}{r}\right)_x = \frac{1.0(3,\ 600)}{110} = 32.7$$

$$\left(\frac{KL}{r}\right)_y = \frac{1.0(3,\ 600)}{64.6} = 55.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)_x^2} = \frac{\pi^2(200,\ 000)}{(55.7)^2} = 635.6 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{635.6}} = 0.742$$

$$C_r = \phi A F_y\left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.742^{(2\times1.34)}\right]^{-1/1.34} = 2,\ 216 \times (10)^6 \text{ N} = 2,\ 216 \text{ kN}$$

Moment Modifier

Uniform moment:

$$\omega_1 = 1.0$$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{\pi^2 \times 200,\ 000 \times 113 \times 10^6}{(1.0 \times 3,\ 600)^2} = 17,\ 211\text{kN}$$

$$U_{1x} = \frac{1.0}{1 - \frac{900}{17,\ 211}} = 1.06$$

Bending Capacity

$$\omega_2 = 1.0$$

$$EI_yGJ = 200{,}000 \times 38.8 \times 76.92 \times 575 = 343.2 \times 10^{21}\ \text{MPa}^2 \cdot \text{mm}^8$$

$$\left(\frac{\pi E}{L}\right)^2 I_y C_w = \left(\frac{\pi \times 200,\ 000}{3,\ 600}\right)^2 38.8 \times 553 = 653.6 \times 10^{21}\ \text{MPa}^2 \cdot \text{mm}^8$$

$$M_u = \frac{\omega_2 \pi}{L}\sqrt{EI_yGJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w}$$

$$= \frac{1.0 \times \pi}{3{,}600}\sqrt{343.2 \times 10^{21} + 653.6 \times 10^{21}} = 871 \times 10^6\ \text{N} \cdot \text{mm} = 871\ \text{kN} \cdot \text{m}$$

Plastic moment capacity:

$$M_p = Z_xF_y = 985 \times 10^3 \times 350 = 345 \times 10^6\ \text{N} \cdot \text{mm} = 345\ \text{kN} \cdot \text{m}$$

$$M_u = 871\ \text{kN} \cdot \text{m} > 0.67 \times M_p = 0.67 \times 345 = 231\ \text{kN} \cdot \text{m}$$

Therefore, the moment capacity is calculated as:

$$M_r = 1.15\phi M_p\left(1 - \frac{0.28M_p}{M_u}\right) = 1.15 \times 0.9 \times 345\left(1 - \frac{0.28 \times 345}{871}\right) = 317\ \text{kN} \cdot \text{m}$$

Check the capacity based on the strength cross-section:

$$M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310\ \text{kN} \cdot \text{m}$$

This governs capacity.

Combined Stress Ratio

$$\frac{900}{2,\ 216} + \frac{0.85 \times 1.06 \times 180}{310} = 0.406 + 0.523 = 0.929 < 1.0$$

Comparison

Table 599: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_x$	32.7	32.624	negligible

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_y$	55.7	55.675	negligible
C_r (kN)	2,216	2,217	negligible
M_u (kN·m)	871	871.4	negligible
M_{rx} (kN·m)	310	310.3	negligble
Stress Ratio	0.929	0.926	negligble

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Shear Capacity Combined Stresses.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.00e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY 900 MZ 180
2 FY -900 MZ -180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2001
SSY 1 ALL
```

```
SSZ 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
      STAAD SPACE                                               -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W250X73                 (CANADIAN SECTIONS)
                           PASS     CSA-13.8.2C        0.926         1
                      900.00 C          0.00        180.00        3.60
      STAAD SPACE                                               -- PAGE NO.
6
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  9.28E+01   MEMBER LENGTH =  3.60E+02
   IZ =  1.13E+04   SZ =  8.93E+02   PZ =  9.85E+02
   IY =  3.88E+03   SY =  3.06E+02   PY =  4.63E+02
   IX =  5.75E+01   CW =  5.53E+05
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 350.0   FU = 450.0   E =  2.00E+05   G =  7.69E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  2.923E+03   CR2 =  2.217E+03   SECTION CLASS 2
   CRZ =  2.710E+03   CTORFLX =  2.217E+03
   TENSILE CAPACITY     =  2.923E+03   COMPRESSIVE CAPACITY =  2.217E+03
   FACTORED MOMENT RESISTANCE : MRY =  1.458E+02   MRZ =  3.103E+02
                              MU =  8.714E+02
   FACTORED SHEAR RESISTANCE  : VRY =  4.523E+02   VRZ =  1.001E+03
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =   55.675   KL/RZ =   32.624   ALLOWABLE KL/R =  200.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  3.600
   OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  2.61E+01
```

V. CSA S16-01 - Short Column Compression

Determine the factored compressive resistance of a short column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.1

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

A W250x73 section is used for a pedestal with a height of 1.1 m.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{1,100}{110} = 10$$

$$\frac{L}{r_y} = \frac{1,100}{64.6} = 17.03$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$F_e = \frac{\pi^2 E}{kL\ / r_y} = \frac{\pi^2 205 \times 10^3}{1 \times 17.03} = 6,978 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{6,978}} = 0.224$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.224^{(2 \times 1.34)}\right]^{-1/1.34} = 2.884 \times (10)^6 \text{ N} = 2,\ 884 \text{ kN}$$

Comparison

Table 600: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	10	9.968	negligible
KL/r, minor	17.03	17.012	negligible
C_r (kN)	2,884	2,884	none

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Short Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
```

```
PARAMETER 1
CODE CANADIAN 2001
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       *******************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W250X73                (CANADIAN SECTIONS)
                           PASS     CSA-13.8.3C        0.000         1
                        1.00 C          0.00           0.00        1.10
      STAAD SPACE                                                -- PAGE NO.
6
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  9.28E+01   MEMBER LENGTH =  1.10E+02
   IZ =  1.13E+04   SZ =  8.93E+02   PZ =  9.85E+02
   IY =  3.88E+03   SY =  3.06E+02   PY =  4.63E+02
   IX =  5.75E+01   CW =  5.53E+05
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 350.0   FU = 450.0   E =  2.05E+05   G =  7.88E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  2.884E+03   CR2 =  2.914E+03   SECTION CLASS 1
   CRZ =  2.914E+03   CTORFLX =  2.884E+03
   TENSILE CAPACITY     =  2.923E+03   COMPRESSIVE CAPACITY =  2.884E+03
   FACTORED MOMENT RESISTANCE : MRY =  1.458E+02   MRZ =  3.103E+02
                                 MU =  7.934E+03
   FACTORED SHEAR RESISTANCE  : VRY =  4.523E+02   VRZ =  1.001E+03
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =   17.012   KL/RZ =    9.968   ALLOWABLE KL/R =  200.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  1.100
   OMEGA-1 (Y-AXIS) = 0.00   OMEGA-1 (Z-AXIS) = 0.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  2.61E+01
```

V. CSA S16-01 - Slender Column Compression

Determine the factored compressive resistance of a slender column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.2

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

A W250x73 section is used for a pedestal with a height of 11.0 m.

The material used is G40.21 350W steel (F_y = *350 MPa*) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{11,\ 000}{110} = 100$$

$$\frac{L}{r_y} = \frac{11,\ 000}{64.6} = 170.3$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$F_e = \frac{\pi^2 \times 205,\ 000}{(1.0 \times 170.3)^2} = 69.78 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{69.78}} = 2.24$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 2.24^{(2\times1.34)}\right]^{-1/1.34} = 0.529 \times (10)^6 \text{ N} = 537.3 \text{ kN}$$

Comparison

Table 601: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	100	99.684	negligible
KL/r, minor	170.3	170.118	negligible
C_r (kN)	537.3	538.2	negligible

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Slender Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2001
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W250X73                    (CANADIAN SECTIONS)
                           PASS     CSA-13.8.3C        0.002         1
                        1.00 C          0.00           0.00       11.00
      STAAD SPACE                                              -- PAGE NO.
6
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  9.28E+01   MEMBER LENGTH =  1.10E+03
   IZ =  1.13E+04   SZ =  8.93E+02   PZ =  9.85E+02
   IY =  3.88E+03   SY =  3.06E+02   PY =  4.63E+02
   IX =  5.75E+01   CW =  5.53E+05
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 350.0   FU = 450.0   E =  2.05E+05   G =  7.88E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  5.382E+02   CR2 =  1.267E+03   SECTION CLASS 1
   CRZ =  1.267E+03   CTORFLX =  5.382E+02
   TENSILE CAPACITY     =  2.923E+03   COMPRESSIVE CAPACITY =  1.267E+03
   FACTORED MOMENT RESISTANCE : MRY =  1.458E+02   MRZ =  1.694E+02
                                MU =  1.882E+02
   FACTORED SHEAR RESISTANCE  : VRY =  4.523E+02   VRZ =  1.001E+03
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =  170.118   KL/RZ =   99.684   ALLOWABLE KL/R =  200.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) = 11.000
   OMEGA-1 (Y-AXIS) = 0.00   OMEGA-1 (Z-AXIS) = 0.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  2.61E+01
```

V. CSA S16-01 - Wide Flange Beam Interaction Ratio 1

Reference

CAN/CSA-S16.1-94, National Standard of Canada, Limit States Design of Steel Structures. The Canadian Standards Association, 1994 with CISC (Canadian Institute of Steel Construction) handbook. CISC Example 1 page 5-91.

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

Find the interaction ratio, beam resistance and beam deflection.

Given

E = 200000 MPa (STEEL)
F_y = 300 Mpa CSA G40.21-M

Simply supported beam has a 8.0 m span; Ky is 1.0, Kz 1.0, unsupported length 1.0 m

Allowable Live Load deflection, L/300 = 8000/300 = 27 mm

Factored Uniform Load IS 7 kN/m DEAD, 15 kN/m LIVE.

Steel section is W410X54

Comparison

Table 602: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Interaction Ratio	0.88	0.882	negligible
Beam Resistance (kN·m)	284	283.5	negligible
Beam Deflection (mm)	21	21.39	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Wide Flange Beam Interaction Ratio 1.STD` is typically installed with the program.

```
STAAD SPACE VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91
START JOB INFORMATION
ENGINEER DATE 21-Sep-18
END JOB INFORMATION
* CISC EXAMPLE 1 PAGE 5-91, LIMIT STATES DESIGN, CSA-S16.1-94
* SIMPLY SUPPORTED BEAM WITH UNIFORM LOAD
* LIVE LOAD DEFLECTION OF L/300
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 8000 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY CANADIAN
1 TABLE ST W410X54
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
```

```
E 1.99947e+08
POISSON 0.3
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MY MZ
UNIT METER KN
LOAD 1 DEAD
MEMBER LOAD
1 UNI GY -7
LOAD 2 LIVE
MEMBER LOAD
1 UNI GY -15
LOAD COMB 3 1.25DL + 1.5 LL
1 1.25 2 1.5
PERFORM ANALYSIS
LOAD LIST 2
PRINT SECTION DISPL
LOAD LIST 3
PARAMETER 1
CODE CANADIAN 2001
TRACK 2 ALL
UNL 1 ALL
FYLD 300000 ALL
BEAM 1 ALL
CHECK CODE ALL
STEEL TAKE OFF ALL
FINISH
```

STAAD Output

```
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - CM
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     2       0.0000    0.0000    0.0000    0.0000   -0.5623    0.0000
                     0.0000   -1.0822    0.0000    0.0000   -1.5237    0.0000
                     0.0000   -1.8590    0.0000    0.0000   -2.0681    0.0000
                     0.0000   -2.1392    0.0000    0.0000   -2.0681    0.0000
                     0.0000   -1.8590    0.0000    0.0000   -1.5237    0.0000
                     0.0000   -1.0822    0.0000    0.0000   -0.5623    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    2.13916   AT     400.00  LOAD    2   L/DISP=    373
         ************ END OF SECT DISPL RESULTS ***********
    39. LOAD LIST 3
    40. PARAMETER 1
    41. CODE CANADIAN 2001
    42. TRACK 2 ALL
    43. UNL 1 ALL
    44. FYLD 300000 ALL
    45. BEAM 1 ALL
    46. CHECK CODE ALL
 STEEL DESIGN
      VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91        -- PAGE NO.
```

```
4
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W410X54                  (CANADIAN SECTIONS)
                           PASS     CSA-13.8.2+        0.882         3
                        0.00 C          0.00        -250.00        4.00
      VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91        -- PAGE NO.
5
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  6.81E+01   MEMBER LENGTH =  8.00E+02
   IZ =  1.86E+04   SZ =  9.23E+02   PZ =  1.05E+03
   IY =  1.01E+03   SY =  1.14E+02   PY =  1.77E+02
   IX =  2.26E+01   CW =  3.88E+05
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 345.0   E =  2.00E+05   G =  7.69E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  1.839E+03   CR2 =  2.646E+02   SECTION CLASS 1
   CRZ =  1.556E+03   CTORFLX =  2.646E+02
   TENSILE CAPACITY     =  1.797E+03   COMPRESSIVE CAPACITY =  2.646E+02
   FACTORED MOMENT RESISTANCE : MRY =  4.779E+01   MRZ =  2.835E+02
                              MU =  3.952E+03
   FACTORED SHEAR RESISTANCE  : VRY =  5.386E+02   VRZ =  4.604E+02
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =  207.732   KL/RZ =   48.407   ALLOWABLE KL/R =  300.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  1.000
   OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  5.08E+01
    47. STEEL TAKE OFF ALL
      VERIFICATION CISC 1994 HANDBOOK EXAMPLE PAGE 5-91        -- PAGE NO.
6
 STEEL TAKE OFF
   STEEL TAKE-OFF
   --------------
      PROFILE                 LENGTH(METE)      WEIGHT(KN  )
   In Steel Takeoff the density of steel is assumed for members with no
density.
   ST  W410X54                     8.00            4.185
                                                ----------------
                                   TOTAL =         4.185
```

V. CSA S16-01 - Wide Flange Beam Interaction Ratio 2

Reference

Handbook of Steel Construction, 9th Edition, H Markham, Ont.: Canadian Institute of Steel Construction, 2007.. p. 4-114

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

Find the interaction ratio, beam and column resistance.

Given

E = 200,000 MPa (STEEL).
F_y = 300 MPa CSA G40.21-M

Simply supported beam/column has a 3.7 m span, Ky is 1.0, Kz 1.0. Factored axial load is 2000 kN and end moments of 200 kN·m and 300 kN·m.

Steel section is W310X118

Comparison

Table 603: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Interaction Ratio	0.940	0.941	negligible
Beam Resistance (kN·m)	606	605.5	negligible
Column Resistance (kN)	3,850	3,849	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Wide Flange Beam Interaction Ratio 2.STD` is typically installed with the program.

```
STAAD SPACE VERIFICATION CISC HANDBOOK 9th Edition PAGE 4-114
START JOB INFORMATION
ENGINEER DATE 14-Nov-18
JOB NAME Verification Example S16-01 : Compression + Major Bending
JOB CLIENT Bentley Systems Inc
JOB REF CISC Hand Book [9th Edition] 4th Printing (Page 4-114)
JOB NO CISC
JOB REV 1.0
JOB PART 1
END JOB INFORMATION
```

```
*
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.7 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X118
*
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
*
*
UNIT MMS NEWTON
DEFINE MATERIAL START
ISOTROPIC STEEL_ASTM_A992
E 200000
POISSON 0.3
DENSITY 7.68191e-05
ALPHA 6.5e-06
DAMP 0.03
G 76902.7
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
*
CONSTANTS
MATERIAL STEEL_ASTM_A992 ALL
*
*
UNIT METER KN
LOAD 1 AXIAL
JOINT LOAD
2 FY -2000
LOAD 2 BENDING MAJOR
JOINT LOAD
2 MZ 200
1 MZ 300
LOAD 3 LOADTYPE None  TITLE AXIAL+BENDINGMAJOR
REPEAT LOAD
1 1.0 2 1.0
*
PERFORM ANALYSIS
*
LOAD LIST 3
*
*
UNIT MMS NEWTON
PARAMETER 1
CODE CANADIAN 2001
TRACK 2 ALL
LY 3700 ALL
LZ 3700 ALL
CMZ 0.4 ALL
```

```
FYLD 345 ALL
CHECK CODE ALL
*
STEEL MEMBER TAKE OFF ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W310X118                 (CANADIAN SECTIONS)
                           PASS     CSA-13.8.2C        0.941         3
                     2000.00 C          0.00         300.00        0.00
      VERIFICATION CISC HANDBOOK 9TH EDITION PAGE 4-114        -- PAGE NO.
5
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  1.50E+02   MEMBER LENGTH =  3.70E+02
   IZ =  2.75E+04   SZ =  1.75E+03   PZ =  1.95E+03
   IY =  9.02E+03   SY =  5.88E+02   PY =  8.93E+02
   IX =  1.60E+02   CW =  1.97E+06
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 345.0   FU = 396.7   E =  2.00E+05   G =  7.69E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  4.658E+03   CR2 =  3.849E+03   SECTION CLASS 2
   CRZ =  4.443E+03   CTORFLX =  3.849E+03
   TENSILE CAPACITY     =  4.553E+03   COMPRESSIVE CAPACITY =  3.849E+03
   FACTORED MOMENT RESISTANCE : MRY =  2.773E+02   MRZ =  6.055E+02
                                MU =  2.300E+03
   FACTORED SHEAR RESISTANCE  : VRY =  7.657E+02   VRZ =  1.571E+03
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =   47.714   KL/RZ =   27.326   ALLOWABLE KL/R =  200.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  3.700
   OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 0.40   OMEGA-2 = 1.00
   SHEAR FORCE (KNS) : Y AXIS =  1.351E+02   Z AXIS =  0.000E+00
   SLENDERNESS RATIO OF WEB (H/W) =  2.32E+01
```

V. CSA S16-01 - Wide Flange Beam Interaction Ratio 3

Reference

Handbook of Steel Construction, 9th Edition, H Markham, Ont.: Canadian Institute of Steel Construction, 2007. p4-115.

Related Links

- *D4.B. Canadian Codes - Steel Design per CSA Standard CAN/CSA-S16-01* (on page 1419)

Problem

Find the interaction ratio, beam, and column resistance.

Given

E = 200,000 MPa (STEEL)
F_y = 345 MPa (ASTM A992)

Simply supported beam/column has a 3.7 m span, Ky is 1.0, Kz 1.0, Lu = 3.7 m

The factored axial load is 2,000 kN and end moments of 200 kN·m and 300 kN·m in the strong axis and 50 kN·m at each end in the weak axis.

Steel section is W310X129.

Comparison

Table 604: CAN/CSA-S16 comparison

Criteria	Reference	STAAD.Pro	Difference
Interaction Ratio	0.990	0.990	none
Beam Resistance, Weak axis (kN·m)	308	307.7	negligible
Beam Resistance, Strong axis (kN·m)	671	670.7	negligible
Column Resistance (kN)	4,240	4,242	negligible

STAAD Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-01 - Wide Flange Beam Interaction Ratio 3.STD` is typically installed with the program.

```
STAAD SPACE VERIFICATION CISC HANDBOOK 9th Edition PAGE 4-115
START JOB INFORMATION
ENGINEER DATE 14-Nov-18
JOB NAME Verification Example S16-01 : Compression + Bending(s)
```

```
JOB CLIENT Bentley Systems Inc
JOB REF CISC Hand Book [9th Edition] 4th Printing (Page 4-115)
JOB NO CISC
JOB REV 1.0
JOB PART 1
END JOB INFORMATION
*
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.7 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X129
*
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
*
*
UNIT MMS NEWTON
DEFINE MATERIAL START
ISOTROPIC STEEL_ASTM_A992
E 200000
POISSON 0.3
DENSITY 7.68191e-05
ALPHA 6.5e-06
DAMP 0.03
G 76902.7
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
*
CONSTANTS
MATERIAL STEEL_ASTM_A992 ALL
*
*
UNIT METER KN
LOAD 1 AXIAL
JOINT LOAD
2 FY -2000
LOAD 2 BENDING MAJOR
JOINT LOAD
2 MZ 200
1 MZ 300
LOAD 3 BENDING MINOR
JOINT LOAD
2 MX 50
1 MX 50
LOAD 4 LOADTYPE None  TITLE AXIAL+BENDING(S)
REPEAT LOAD
1 1.0 2 1.0 3 1.0
*
PERFORM ANALYSIS
*
```

```
LOAD LIST 4
*
*
UNIT MMS NEWTON
PARAMETER 1
CODE CANADIAN 2001
TRACK 2 ALL
LY 3700 ALL
LZ 3700 ALL
SSY 0 ALL
SSZ 0 ALL
FYLD 345 ALL
CHECK CODE ALL
*
STEEL MEMBER TAKE OFF ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (CAN/CSA-S16-01  )
V2.1
                       ********************************************
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W310X129                 (CANADIAN SECTIONS)
                           PASS     CSA-13.8.2C        0.990         4
                     2000.00 C        -50.00        300.00        0.00
      VERIFICATION CISC HANDBOOK 9TH EDITION PAGE 4-115        -- PAGE NO.
5
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   CROSS SECTION AREA =  1.65E+02   MEMBER LENGTH =  3.70E+02
   IZ =  3.08E+04   SZ =  1.94E+03   PZ =  2.16E+03
   IY =  1.00E+04   SY =  6.49E+02   PY =  9.91E+02
   IX =  2.13E+02   CW =  2.21E+06
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 345.0   FU = 396.7   E =  2.00E+05   G =  7.69E+04
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CR1 =  5.123E+03   CR2 =  4.242E+03   SECTION CLASS 1
   CRZ =  4.892E+03   CTORFLX =  4.242E+03
   TENSILE CAPACITY     =  5.008E+03   COMPRESSIVE CAPACITY =  4.242E+03
   FACTORED MOMENT RESISTANCE : MRY =  3.077E+02   MRZ =  6.707E+02
                              MU =  2.638E+03
   FACTORED SHEAR RESISTANCE  : VRY =  8.537E+02   VRZ =  1.731E+03
   MISCELLANEOUS INFORMATION
   --------------------------
   NET SECTION FACTOR FOR TENSION =  1.000
   KL/RY =   47.527   KL/RZ =   27.081   ALLOWABLE KL/R =  200.000
   UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  3.700
```

```
OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
SHEAR FORCE (KNS) : Y AXIS =  1.351E+02   Z AXIS =  2.703E+01
SLENDERNESS RATIO OF WEB (H/W) =  2.11E+01
```

V.CSA S16-09 - Axial Tension

Design of a double-angle member subject to tension load.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 3.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

Design a tension diagonal in an all-welded truss (`SNU 0` for welded). Attached to WT 265x61.5 chords (t_w = 13.1 mm).

$$L = 4 \text{ m}$$

$$T_f = 630 \text{ kN}$$

The material used is G40.21 300W steel (F_y = *300 MPa*, F_u = *450 MPa*) (`FYLD` and `FU`)

Calculations

Gross area is used for welded connections.

$$T_r = \max \begin{cases} \phi A F_y = 0.9 \times 300 \times A = 270A \\ \phi_u A F_u = 0.75 \times 450 \times A = 338A \end{cases}$$

$T_r \geq T_f$, so solve for A:

$$A = \frac{630 \times (10)^3}{270} = 2,\ 333\text{mm}^2$$

Select a pair of angles with at least this gross area: try a 2- 76x64x9.5, with long legs back-to-back:

$$A = 2{,}480 \text{ mm}^2$$

$$r_{min} = 13.3 \text{ mm}$$

Calculate the effective net area due to shear lag. The average weld length is taken as $(120\ mm + 250\ mm)/2 = 185\ mm$.

The long leg does not have a reduction in area shear lag, thus:

$$A_{n2} = 1.00 \times (76 - 9.5) \times 9.5 = 632 \text{ mm}^2$$

The outstanding (short) leg has an eccentricity of half the leg length, or $64/2 = 32\ mm$.

$$A_{n3} = \left(1 - \frac{32}{250}\right) \times 64 \times 9.5 = 530\text{mm}^2$$

The total net area for the pair of angles for the rupture limit state is:

$$A_{ne} = 632 + 530 = 1,162 \text{ mm}^2$$

For the pair of angles, $A_{ne} = 2×1,1162 = 2,324\ mm^2$; the ratio of net section area to gross area = 2,324 / 2,480 = 0.937 (NSF).

Checking the yielding limit state:

$$T_r = 0.90×2,480×300 = 670 \text{ kN}$$

Checking the rupture limit state:

$$T_r = 0.75×2,324×450 = 784 \text{ kN}$$

Yielding governs, and the critical ratio is: $630/670 = 0.94$.

Comparison

Table 605: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
T_f (kN)	670	669.6	negligible
Critical Ratio	0.94	0.941	negligible

Note: If the member SELECT facility is used in place of a code CHECK, STAAD.Pro will actually select a 2-L 102x89x6.4 member, which has a critical ratio of 0.997 (per Cl. 13.2) and an area of 2,340 mm^2, thus more economical (the NSF used for this size is 0.923).

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Axial Tension.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
```

```
1 TABLE LD L76X64X9.5
*1 TABLE T W530x123
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 300000 ALL
FU 450000 ALL
SNUG 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.941(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.9.1
  SECTION: LD   L76X64X9.5              (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.941(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.9.1
  DESIGN FORCES:   Fx:    630.00(T)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      11.195 AYY:      13.616 CW:
0.000
                            SZZ:      26.578 SYY:      26.229
                            IZZ:     137.969 IYY:     166.556
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      300.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):     4.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     169.588  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 3
  FLANGE:          Class 3
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
```

```
LOCATION(MET):
  YIELDING:      630.000     669.600      0.941    Cl. 13.2
1        0.000
  RUPTURE:       630.000     837.000      0.753    Cl. 13.2
1        0.000
      STAAD SPACE                                          -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:            0.000     141.791      0.000    Cl. 13.3
1        0.000
  MINOR:            0.000     166.729      0.000    Cl. 13.3
1        0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          24.800    169.796      70.178      2.068        1.340
  MINOR:          24.800    154.539      84.719      1.882        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
                  0.000     164.708      0.000    Cl. 13.3          1
0.000
  INTERMEDIATE:   Fe:(N MM) λ           n:
                  83.506    1.895       1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:            0.000     226.444      0.000    Cl. 13.4.3
1        0.000
  MINOR:            0.000     196.070      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          13.616    1.200       0.000        0.000
0.000       0.000
  MINOR:          11.195
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    7.18E+00      0.000    Cl. 13.5(b)
1        0.000
  MINOR:        0.00E+00    7.08E+00      0.000    Cl. 13.5(b)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        1.44E+01    0.00E+00    0.80E+01
  MINOR:        0.00E+00    0.00E+00    0.79E+01
  UNIT: KN MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    1.17E+01      0.000    Cl. 13.6(e)(i)
1        0.000
  INTERMEDIATE:  Iyc(CM):   w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        4.07E+01  2.50   1.00     0.59   3.65E+01  0.02   5.58E+00
0.52  25.18
  UNIT: KN MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
```

```
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.941    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.000    Cl. 13.8.3         1       0.000
  MEMBER STRENGTH:             0.000    Cl. 13.8.3         1       0.000
  LTB STRENGTH:                0.000    Cl. 13.8.3         1       0.000
  FLEX AND SHEAR:              0.000    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.000    Cl. 13.8           1       0.000
      STAAD SPACE                                              -- PAGE NO.
7
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   6.30E+02  Tr:   6.70E+02  Mfz:   0.00E+00  Mrz:
7.18E+00
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00  0.00E+00  0.00E+00 6.70E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  MEM STR: 0.00E+00  0.00E+00  0.00E+00 1.42E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  LTB STR: 0.00E+00  0.00E+00  0.00E+00 1.65E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   7.18E+00  Vfy:   0.00E+00  Vry:   2.26E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   7.18E+00  Mfy:   0.00E+00  Mry:   7.08E
+00
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V.CSA S16-09 - Beam Bending

Determine the uniformly factored load that the member can resist.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W310x52 beam spans 7.3 m. Both ends of the beam are supported by columns connected standard web angle connections.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Section Classification

Evaluate the slenderness effects of the beam flanges:

$$\frac{b}{2t} = \frac{167}{2 \times 13.2} = 6.3 < \frac{170}{\sqrt{350}} = 9.1$$

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{318 - 2(13.2)}{7.6} = 38.4 < \frac{1,700}{\sqrt{350}} = 90.9$$

Both the beam flanges and web are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 5.7.

Bending Capacity

Factored moment resistance is:

$$M_r = \phi Z_x F_y = 0.90 \times 841 \times (10)^3 \times 350 = 265 \times (10)^3 \text{ N·mm} = 265 \text{ kN·m}$$

Equate this to the bending moment due to a uniformly distributed load and solve for the load:

$$M_r \geq \frac{w_f L^2}{8}$$

$$w_f = \frac{8M_r}{L^2} = \frac{8 \times 265}{(7.3)^2} = 39.8 \text{ kN/m}$$

Comparison

Table 606: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
M_r (kN·m)	265	265	none
w_f (kN/m)	39.8	39.76*	negligible

Note: (*) STAAD.Pro does calculate the allowable uniform load on the beam. Instead, this is done by applying a uniform load incrementally until the critical ratio is ≈ 1.0. The, this uniform load is divided by the resulting critical ratio to normalize the distributed load capacity of the beam. Thus, *39.8/1.001 = 39.67 kN/m*

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Beam Bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
```

```
1 0 0 0; 2 7.3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -39.8
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
 *MEMBER NO:     1    CRITICAL RATIO:  1.001(FAIL)     LOAD:     1
  LOCATION (MET): 3.65  CONDITION: Cl. 13.8
  SECTION: ST   W310X52                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  1.001(FAIL)     LOAD CASE:     1  LOCATION (MET): 3.65
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:  -2.65E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      44.088 AYY:      23.165 CW:
237980.875
                            SZZ:     748.428 SYY:     123.353
                            IZZ:   11900.002 IYY:    1030.000
```

```
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):     7.300
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     185.766  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE:         Class 1
  FLANGE:          Class 1
  WEB:             Class 1
  UNIT: KN MET
  TENSION:         FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:         0.000    2101.050      0.000    Cl. 13.2
1        0.000
  RUPTURE:          0.000    2251.125      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:     FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:            0.000    1587.513      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:            0.000     322.454      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:    Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:           65.224     54.653      677.375      0.719        1.340
  MINOR:           65.224    185.766       58.630      2.443        1.340
  FLEX TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
                   0.000    1023.539      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:    Fe:(N MM) λ           n:
                 253.108    1.176        1.340
  SHEAR:           FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         -145.270     502.453      0.289    Cl. 13.4.1.1
1        7.300
  MINOR:            0.000     833.263      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:    Aw:(CM)    Kv:           Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:           23.165    5.348       0.044       326.992
653.857      231.000
  MINOR:           44.088
  UNIT: KN MET
  YIELDING:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         2.65E+02    2.65E+02      1.001    Cl. 13.5(a)
1        3.650
  MINOR:         0.00E+00    5.95E+01      0.000    Cl. 13.5(a)
1        0.000
```

```
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        2.94E+02    0.00E+00    0.00E+00
  MINOR:        6.62E+01    0.00E+00    0.00E+00
  UNIT: KN MET
  INTERACTION:                 RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                 1.001    Cl. 13.9.1         1       3.650
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.851    Cl. 13.8.2         1       3.650
  MEMBER STRENGTH:             0.851    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.729    Cl. 14.6(a)        1       3.042
  BIAXIAL FLEX:                 1.001    Cl. 13.8           1       3.650
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.10E+03  Mfz:   2.65E+02  Mrz:
2.65E+02
      STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00  2.65E+02  0.00E+00 2.05E+03 2.65E+02 5.95E+01 0.85 4.52E
+03 3.91E+02 0.60 0.60 1.00 1.00
  MEM STR: 0.00E+00  2.65E+02  0.00E+00 3.22E+02 2.65E+02 5.95E+01 0.85 4.52E
+03 3.91E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   2.58E+02  Mrz:   2.65E+02  Vfy:   2.42E+01  Vry:   5.02E
+02
  BIAX FLEX:   Mfz:   2.65E+02  Mrz:   2.65E+02  Mfy:   0.00E+00  Mry:   5.95E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V.CSA S16-09 - Beam Shear Capacity

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.3

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W530x82 beam has a maximum shear force, $v = 540\ kN$ due to factored loads.

L = 11.0 m

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{528 - 2(13.3)}{9.5} = 52.8 < 439\sqrt{\frac{k_v}{F_y}} = 439\sqrt{\frac{5.34}{350}} = 54.2$$

Therefore the shear capacity is calculated as:

$$V_r = \phi \times A_w \times F_y = 0.9 \times 528 \times 9.5 \times 350 = 1{,}043 \text{ kN}$$

Calculate Equivalent Concentrated Load

The given end moments and assumed distributed loads (which was determined to given an equivalent mid-span moment of the given example) do not result in a shear force of 540 kN. Therefore, an additional concentrated load is added near the first support.

$$R_1 = \frac{w_f L}{2} + \frac{M_1 - M_2}{L} = \frac{40.17 \times 11.0}{2} + \frac{540 - 185}{11.0} = 253.2 \text{ kN}$$

$$P = 540 \text{ kN} - 253.2 \text{ kN} = 287 \text{ kN}$$

The STAAD.Pro will have this load applied at 1 mm away from the left support.

Comparison

Table 607: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
V_r (kN)	1,043	1,042.826	negligible

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Beam Shear Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
```

```
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                     STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                   *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
 *MEMBER NO:     1    CRITICAL RATIO:  1.803(FAIL)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W530X82                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  1.803(FAIL)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   254.22      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      55.594 AYY:      48.896 CW:
1340255.500
                            SZZ:    1806.818 SYY:     194.258
                            IZZ:   47700.008 IYY:    2030.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     250.172  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
```

```
  COMPRESSION:     Class 4
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    3307.500      0.000    Cl. 13.2
1        0.000
  RUPTURE:         0.000    3543.750      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000    2253.274      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:           0.000     253.222      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          89.688     51.609     759.622      0.679        1.340
  MINOR:          89.688    250.172      32.328      3.290        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
                  0.000     854.388      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe:(N MM) λ            n:
                 125.187    1.672        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        -538.417    1042.826      0.516    Cl. 13.4.1.1       1
11.000
  MINOR:           0.000    1050.727      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          48.896    5.349       0.048       237.748
345.654      231.000
  MINOR:          55.594
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -5.40E+02    6.49E+02      0.832    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        7.21E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -5.40E+02    2.99E+02      1.803    Cl. 13.6(a)(ii
1        0.000
  INTERMEDIATE:  Iyc(CM):   w 2:     w 3:      b x:     Mu:     rt:
```

```
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.29   0.00     0.00   3.33E+02  0.00   0.00E+00
0.00   0.00
  UNIT: KN MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.832    Cl. 13.9.1         1       0.000
  MEMBER STRENGTH:             1.803    Cl. 13.9.2         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.707    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.707    Cl. 13.8.2         1       0.000
  LTB STRENGTH:                1.533    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              1.422    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                1.803    Cl. 13.8           1       0.000
      STAAD SPACE                                              -- PAGE NO.
7
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.31E+03  Mfz:  -5.40E+02  Mrz:
6.49E+02
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 2.83E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.53E+02 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.60 0.74 1.00 1.00
  LTB STR: 0.00E+00 -5.40E+02  0.00E+00 2.53E+02 2.99E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   2.99E+02  Vfy:   2.54E+02  Vry:   1.04E
+03
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   2.99E+02  Mfy:   0.00E+00  Mry:   9.54E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V.CSA S16-09 - Select a Beam

Select a wide-flange shape for the given beam span.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.2

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A beam must be selected to span 11.0 m. The end moment of the span are:

$$M_1 = -540 \text{ kN·m}$$

$$M_2 = -185 \text{ kN·m}$$

And the maximum moment in the span is:

$$M_3 = 256 \text{ kN}\cdot\text{m}$$

The beam is braced at 2.5 m from the center of each supporting column (thus $L_b = 6.0\ m$) (`LT 6.0`).

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`)

Calculations

Assume a Class 2 section:

$$M_r = \phi M_p = \phi Z_x F_y$$

Thus,

$$Z_x \geq \frac{540(10)^6}{0.9 \times 350} = 1,714 \times (10)^3 \text{ mm}^3$$

Try a W460x82:

$$Z_x = 1{,}830 \times (10)^3 \text{ mm}^3, \text{ tf} = 16.0 \text{ mm}, \text{ bf} = 191.0 \text{ mm}, \text{ tw} = 9.9 \text{ mm}, \text{ D} = 460.0 \text{ mm}$$

Evaluate the slenderness effects of the beam flanges:

$$\frac{b}{2t} = \frac{191}{2 \times 16.0} = 5.7 < \frac{170}{\sqrt{350}} = 9.1$$

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{460 - 2(16.0)}{9.9} = 43.2 < \frac{1,700}{\sqrt{350}} = 90.9$$

The assumptions are valid and this beam is sufficient to support the given moments.

$$M_r = 0.9 \times 1{,}830 \times (10)^3 \times 350\ [(10)^{-6}] = 576.5 \text{ kN}\cdot\text{m}$$

Ratio = $540\ /\ 576.5 = 0.937$

Comparison

Table 608: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Section	W460x82 *	W460x82	none
M_r (kN·m)	576.5 **	576	negligible
Critical Ratio	0.937 **	0.937	none

Notes:

*The reference also tries a W530x82, which is stiffer for the same weight. STAAD.Pro selects the shallower beam.
** The reference does not calculate the resisting moment capacity nor the critical ratio for either section size, but they are evaluated here for completeness.

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Select a Beam.STD` is typically installed with the program.

In order to model this beam in STAAD.Pro, we need to determine the distributed load which would result in the same mid-span moment on a beam with the given end moments. From the AISC Streel Construction Manual, p 3-222:

$$M_3 = \frac{w_f L^2}{8} - \frac{(M_1 + M_2)}{2} + \frac{(M_1 - M_2)^2}{2w_f L^2}$$

$$256 = \frac{w_f (11.0)^2}{8} - \frac{(540 + 185)}{2} + \frac{(540 - 185)^2}{2w_f (11.0)^2}$$

Solving this yields $w_f = 40.033\ kN/m$

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -40.033
JOINT LOAD
1 MZ 540
2 MZ -185
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
```

```
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
SELECT ALL
PERFORM ANALYSIS
FINISH
```

STAAD.Pro Output

```
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.832(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W530X82                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.832(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   252.45      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      55.594 AYY:      48.896 CW:
1340255.500
                            SZZ:    1806.818 SYY:     194.258
                            IZZ:   47700.008 IYY:    2030.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     250.172  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    3307.500      0.000    Cl. 13.2
1        0.000
  RUPTURE:         0.000    3543.750      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
```

```
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000    2253.274      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:           0.000     253.222      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)        λ          n:
  MAJOR:          89.688     51.609      759.622      0.679        1.340
  MINOR:          89.688    250.172       32.328      3.290        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
                  0.000     854.388      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe:(N MM) λ            n:
                 125.187    1.672        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         252.454    1042.826      0.242    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1050.727      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          48.896    5.349       0.048       237.748
345.654      231.000
  MINOR:          55.594
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -5.40E+02    6.49E+02      0.832    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        7.21E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  INTERACTION:               RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:               0.832    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:               0.707    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:            0.707    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:             0.715    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:               0.832    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.31E+03  Mfz:  -5.40E+02  Mrz:
6.49E+02
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
```

```
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 2.83E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.53E+02 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.60 0.74 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   6.49E+02  Vfy:   2.52E+02  Vry:   1.04E
+03
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   6.49E+02  Mfy:   0.00E+00  Mry:   9.54E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
    45. SELECT ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO MEMBER SELECTION - S16-09 (v1.1)
                     **********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.937(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W460x82                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.937(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   252.45      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      61.120 AYY:      43.956 CW:
1340255.500
                            SZZ:    1608.696 SYY:     194.764
                            IZZ:   37000.004 IYY:    1860.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     260.108  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE:         Class 1
  FLANGE:          Class 1
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    3276.000      0.000    Cl. 13.2
1        0.000
  RUPTURE:         0.000    3510.000      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                              -- PAGE NO.
9
                     STAAD.PRO MEMBER SELECTION - (          S16-09)   v1.1
```

```
                     ***********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000    2261.248      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:           0.000     253.374      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          96.728     58.319      594.890      0.767        1.340
  MINOR:          96.728    260.107       29.905      3.421        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
                  0.000    1279.603      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe:(N MM) λ           n:
                 194.448    1.342        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         252.454     946.776      0.267    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1155.168      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          43.956    5.347       0.042       290.187
514.950      231.000
  MINOR:          61.120
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -5.40E+02    5.76E+02      0.937    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        6.41E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  INTERACTION:               RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:               0.937    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:               0.796    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:            0.796    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:             0.802    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:               0.937    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.28E+03  Mfz:  -5.40E+02  Mrz:
5.76E+02
      STAAD SPACE                                         -- PAGE NO.
10
                     STAAD.PRO MEMBER SELECTION - (          S16-09)   v1.1
                     ***********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
```

```
              Cf:        Mfz:        Mfy:      Cr:      Mrz:      Mry:      b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 3.05E+03 5.76E+02 9.54E+01 0.85 6.19E
+03 3.11E+02 0.74 0.60 1.00 1.00
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.53E+02 5.76E+02 9.54E+01 0.85 6.19E
+03 3.11E+02 0.60 0.74 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   5.76E+02  Vfy:   2.52E+02  Vry:   9.47E
+02
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   5.76E+02  Mfy:   0.00E+00  Mry:   9.54E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V.CSA S16-09 - Shear Capacity Combined Stresses

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 8.6

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W250x73 column has the following concentrated loads

$$C_f = 900 \text{ kN}$$

And a uniform moment of:

$$M_{fx} = 180 \text{ kN·m}$$

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`). The beam is braced against lateral-torsional buckling only at the ends (`LAT 0`, default)

Calculations

Axial Capacity

Determine the buckling load of the column:

$$\left(\frac{KL}{r}\right)_x = \frac{1.0(3,\ 600)}{110} = 32.7$$

$$\left(\frac{KL}{r}\right)_y = \frac{1.0(3,\ 600)}{64.6} = 55.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)_x^2} = \frac{\pi^2(200,\ 000)}{(55.7)^2} = 635.6 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{635.6}} = 0.742$$

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.742^{(2\times1.34)}\right]^{-1/1.34} = 2,\ 216 \times (10)^6 \text{ N} = 2,\ 216 \text{ kN}$$

Moment Modifier

Uniform moment:

$$\omega_1 = 1.0$$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{\pi^2 \times 200,\ 000 \times 113 \times 10^6}{(1.0 \times 3,\ 600)^2} = 17,\ 211\text{kN}$$

$$U_{1x} = \frac{1.0}{1 - \frac{900}{17,\ 211}} = 1.06$$

Bending Capacity

$$\omega_2 = 1.0$$

$$EI_yGJ = 200{,}000\times38.8\times76.92\times575= 343.2\times10^{21} \text{ MPa}^2 \cdot\text{mm}^8$$

$$\left(\frac{\pi E}{L}\right)^2 I_y C_w = \left(\frac{\pi \times 200,\ 000}{3,\ 600}\right)^2 38.8 \times 553 = 653.6 \times 10^{21} \text{ MPa}^2 \cdot \text{mm}^8$$

$$M_u = \frac{\omega_2 \pi}{L}\sqrt{EI_yGJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w}$$

$$= \frac{1.0 \times \pi}{3{,}600}\sqrt{343.2 \times 10^{21} + 653.6 \times 10^{21}} = 871 \times 10^6 \text{ N}\cdot\text{mm} = 871 \text{ kN}\cdot\text{m}$$

Plastic moment capacity:

$$M_p = Z_x F_y = 985\times10^3 \times 350 = 345\times10^6 \text{ N}\cdot\text{mm} = 345 \text{ kN}\cdot\text{m}$$

$$M_u = 871 \text{ kN}\cdot\text{m} > 0.67\times M_p = 0.67\times345 = 231 \text{ kN}\cdot\text{m}$$

Therefore, the moment capacity is calculated as:

$$M_r = 1.15\phi M_p\left(1 - \frac{0.28M_p}{M_u}\right) = 1.15 \times 0.9 \times 345\left(1 - \frac{0.28 \times 345}{871}\right) = 317 \text{ kN}\cdot\text{m}$$

Check the capacity based on the strength cross-section:

$$M_{rx} = 0.9\times985\times10^3 \times350 = 310 \text{ kN}\cdot\text{m}$$

This governs capacity.

Combined Stress Ratio

$$\frac{900}{2,\ 216} + \frac{0.85 \times 1.06 \times 180}{310} = 0.406 + 0.523 = 0.929 < 1.0$$

Comparison

Table 609: Verification Problem comparison

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_x$	33	32.624	negligible
$(KL/r)_y$	56	55.675	negligible
C_r (kN)	2,216	2,220	negligible
C_e (kN)	17,211	17,200	negligible
Stress Ratio	0.929	0.926	negligible

Note: Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Shear Capacity Combined Stresses.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.00e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
*MEMBER PROPERTY CANADIAN
*1 TABLE ST W250X73
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
```

```
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY 900 MZ 180
2 FY -900 MZ -180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
SSY 1 ALL
SSZ 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.926(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.926(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:    900.00(C)    Fy:     0.00      Fz:      0.00
                  Mx:   0.00E+00      My:  0.00E+00     Mz:   1.80E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):     3.600
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      55.675  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
```

```
1        0.000
  RUPTURE:          0.000    3132.000      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         900.000    2709.553      0.332    Cl. 13.3
1        0.000
  MINOR:         900.000    2217.120      0.406    Cl. 13.3
1        0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          92.800     32.624    1854.621      0.434        1.340
  MINOR:          92.800     55.675     636.808      0.741        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000     452.349      0.000    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          20.537    5.360       0.070       480.943
1414.474      231.000
  MINOR:          72.136
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -1.80E+02    3.10E+02      0.580    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:        My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:       -1.80E+02    3.17E+02      0.567    Cl. 13.6(a)(i)
1        0.000
  INTERMEDIATE:  Iyc(CM):   w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  1.00   0.00     0.00   8.71E+02  0.00   0.00E+00
0.00   0.00
  UNIT: KN MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.580    Cl. 13.9.1         1       0.000
  MEMBER STRENGTH:             0.580    Cl. 13.9.2         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.828    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.926    Cl. 13.8.2         1       0.000
  LTB STRENGTH:                0.926    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.422    Cl. 14.6(a)        1       0.000
```

```
  BIAXIAL FLEX:                0.580    Cl. 13.8          1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:  -1.80E+02  Mrz:
3.10E+02
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 9.00E+02 -1.80E+02  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.60 1.72E
+04 5.91E+03 1.00 0.60 1.06 1.00
  MEM STR: 9.00E+02 -1.80E+02  0.00E+00 2.22E+03 3.10E+02 1.46E+02 0.85 1.72E
+04 5.91E+03 0.60 1.00 1.06 0.71
  LTB STR: 9.00E+02 -1.80E+02  0.00E+00 2.22E+03 3.10E+02 1.46E+02 0.85 1.72E
+04 5.91E+03 1.00 0.60 1.06 0.71
  FLEX SHEAR:  Mfz:  -1.80E+02  Mrz:   3.10E+02  Vfy:   0.00E+00  Vry:   4.52E
+02
  BIAX FLEX:   Mfz:  -1.80E+02  Mrz:   3.10E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V.CSA S16-09 - Short Column Compression

Determine the factored compressive resistance of a short column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W250x73 section is used for a pedestal with a height of 1.1 m.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{1,\ 100}{110} = 10$$

$$\frac{L}{r_y} = \frac{1,\ 100}{64.6} = 17$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$\lambda = \left(\frac{KL}{r}\right)_{\max}\sqrt{\frac{F_y}{\pi^2 E}} = 17\sqrt{\frac{350}{\pi^2 200,\ 000}} = 0.227$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y\left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.227^{(2\times1.34)}\right]^{-1/1.34} = 2.883 \times (10)^6 \text{ N} = 2,\ 883 \text{ kN}$$

Comparison

Table 610: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	10	9.968	negligible
KL/r, minor	17	17.012	negligible
λ, critical	0.227	0.224	1.3%
C_r (kN)	2,883	2,884.358	negligible

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Short Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
*1 TABLE ST L203x203x25
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
*2 FY -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.000(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.000(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:      0.00(C)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
```

```
  ACTUAL MEMBER LENGTH(MET):     1.100
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      17.012  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.001    2913.816      0.000    Cl. 13.3
1       0.000
  MINOR:           0.001    2884.358      0.000    Cl. 13.3
1       0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          92.800      9.968    20360.980     0.131        1.340
  MINOR:          92.800     17.012     6991.204     0.224        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000     452.349      0.000    Cl. 13.4.1.1
1       0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1       0.000
  INTERMEDIATE:   Aw:(CM)    Kv:           Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          20.537    5.552       0.224       489.474
1465.103      231.000
  MINOR:          72.136
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    3.10E+02      0.000    Cl. 13.5(a)
1       0.000
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1       0.000
  INTERMEDIATE:   Mp:         Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    3.55E+02      0.000    Cl. 13.6(a)(i)
1       0.000
```

```
  INTERMEDIATE:  Iyc(CM):   w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.50   0.00     0.00   1.98E+04  0.00   0.00E+00
0.00   0.00
  UNIT: KN MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.000    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.000    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.000    Cl. 13.8.2         1       0.000
  LTB STRENGTH:                0.000    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.000    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.000    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   0.00E+00  Mrz:
3.10E+02
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 1.00E-03  0.00E+00  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.69 1.89E
+05 6.49E+04 0.60 0.60 1.00 1.00
  MEM STR: 1.00E-03  0.00E+00  0.00E+00 2.88E+03 3.10E+02 1.46E+02 0.69 1.89E
+05 6.49E+04 0.60 0.60 1.00 1.00
  LTB STR: 1.00E-03  0.00E+00  0.00E+00 2.88E+03 3.10E+02 1.46E+02 0.69 1.89E
+05 6.49E+04 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   3.10E+02  Vfy:   0.00E+00  Vry:   4.52E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   3.10E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V.CSA S16-09 - Slender Column Compression

Determine the factored compressive resistance of a slender column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.2

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W250x73 section is used for a pedestal with a height of 11.0 m.

The material used is G40.21 350W steel (F_y = *350 MPa*) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{11,\ 000}{110} = 100$$

$$\frac{L}{r_y} = \frac{11,\ 000}{64.6} = 170$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$F_e = \frac{\pi^2 \times 200,\ 000}{(1.0 \times 170)^2} = 68.3 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{68.3}} = 2.26$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 2.26^{(2\times 1.34)}\right]^{-1/1.34} = 0.529 \times (10)^6 \text{ N} = 529 \text{ kN}$$

Comparison

Table 611: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	100	99.684	negligible
KL/r, minor	170	170.118	negligible
λ, critical	2.26	2.237	1.0%

Criteria	Reference	STAAD.Pro	Difference
C_r (kN)	529	538.161	1.7%

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Slender Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
```

```
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.000(PASS)     LOAD:     1
  LOCATION (MET): 0.00  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.000(PASS)     LOAD CASE:     1  LOCATION (MET): 0.00
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:      0.00(C)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     170.118  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.001    1266.716      0.000    Cl. 13.3
1       0.000
  MINOR:           0.001     538.161      0.000    Cl. 13.3
1       0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          92.800     99.684     203.610      1.311        1.340
  MINOR:          92.800    170.118      69.912      2.237        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:           0.000     452.349      0.000    Cl. 13.4.1.1
1       0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1       0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
```

```
  MAJOR:          20.537    5.342       0.023       480.151
1409.819      231.000
  MINOR:          72.136
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    3.10E+02      0.000    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        0.00E+00    2.83E+02      0.000    Cl. 13.6(a)(i)
1        0.000
  INTERMEDIATE:  Iyc(CM):   w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.50   0.00     0.00   4.66E+02  0.00   0.00E+00
0.00   0.00
  UNIT: KN MET
  INTERACTION:               RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:               0.000    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:               0.000    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:            0.000    Cl. 13.8.2         1       0.000
  LTB STRENGTH:               0.000    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:             0.000    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:               0.000    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   0.00E+00  Mrz:
3.10E+02
      STAAD SPACE                                           -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 1.00E-03  0.00E+00  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  MEM STR: 1.00E-03  0.00E+00  0.00E+00 5.38E+02 3.10E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  LTB STR: 1.00E-03  0.00E+00  0.00E+00 5.38E+02 2.83E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   2.83E+02  Vfy:   0.00E+00  Vry:   4.52E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   2.83E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V.CSA S16-09 - Wide Flange Capacity Combined Stresses

Check the capacity of a wide-flange column under combined stresses.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 8.5

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W250x73 column has the following concentrated loads applied at the top of the column:

$$C_f = 900 \text{ kN}$$

$$M_{fx} = 180 \text{ kN·m}$$

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`). The column is assumed to be braced laterally against lateral-torsional buckling (`LAT 1`).

$$L = 3.6 \text{ m}$$

Calculations

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Axial Capacity

Determine the buckling load of the column, assuming that weak axis bucking will not control due to bracing:

$$\left(\frac{KL}{r}\right)_x = \frac{1.0(3,\ 600)}{110} = 32.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)_x^2} = \frac{\pi^2 (200,\ 000)}{(32.7)^2} = 1,\ 847 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{1,\ 847}} = 0.435$$

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.435^{(2\times 1.34)}\right]^{-1/1.34} = 2,\ 716 \times (10)^6 \text{ N} = 2,\ 716 \text{ kN}$$

Moment Modifier

$\kappa = -0/180 = 0$

$\omega_1 = 0.6 - 0.4\kappa = 0.6 \leq 1.0$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{\pi^2 \times 200,\ 000 \times 113 \times 10^6}{(1.0 \times 3,\ 600)^2} = 17,\ 211\text{kN}$$

$$U_{1x} = \frac{0.6}{1 - \left(\frac{900}{17,\ 210}\right)} = 0.633$$

Bending Capacity

From a previous example, the W250x73 is a Class 2 section. It is laterally supported along its length, thus:

$M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN}\cdot\text{m}$

Combined Stress Ratio

$$\frac{900}{2,\ 716} + \frac{0.85 \times 0.633 \times 180}{310} = 0.331 + 0.312 = 0.644 < 1.0$$

This is the check for overall member strength. However, there is another case that actually controls in this instance: checking for cross sectional strength.

$C_r = 0.9 \times 9{,}280 \times 350 = 2{,}923 \text{ kN}$

$U_{1x} = 1.0$

$$\frac{900}{2,\ 923} + \frac{0.85 \times 1.0 \times 180}{310} = 0.308 + 0.494 = 0.801 < 1.0$$

Comparison

Table 612: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_x$	33	32.624	negligible
$(KL/r)_y$	56	55.675	negligible
C_r (kN)	2,716	2,715.935	negligible
C_e (kN)	17,211	17,600	2.3%
Stress Ratio	0.644	0.643	none
Stress Ratio (Critical)	0.801	0.801	none

Note: Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\S16 2009\CSA S16-09 - Wide Flange Capacity Combined Stresses.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY 900
*1 MZ 180
2 FY -900 MZ 180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2009
FYLD 350000 ALL
LAT 1 ALL
SSZ 1 ALL
SSY 1 ALL
FU 450000 ALL
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - S16-09 (v1.1)
                     ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.801(PASS)     LOAD:     1
  LOCATION (MET): 3.60  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                  (CANADIAN SECTIONS)
  UNIT: KN MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.801(PASS)     LOAD CASE:     1  LOCATION (MET): 3.60
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:    900.00(C)    Fy:    50.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:  -1.80E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD:      350.000   FU:      450.000
  ACTUAL MEMBER LENGTH(MET):     3.600
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      55.675  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 2
  FLANGE:          Class 2
  WEB:             Class 1
  UNIT: KN MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-09)   v1.1
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:         900.000    2715.935      0.331    Cl. 13.3
1       0.000
  MINOR:         900.000    2233.940      0.403    Cl. 13.3
1       0.000
  INTERMEDIATE:   Ag:(CM)     KL/r:     Fe:(N MM)       λ          n:
  MAJOR:          92.800     32.624     1900.987      0.429       1.340
```

```
  MINOR:          92.800     55.675      652.728      0.732        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:          50.000     452.349      0.111    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:   Aw:(CM)    Kv:          Ka:       Fcri:(N MM)  Fcre:(N MM)
Fs:(N MM)
  MAJOR:          20.537    5.360       0.070       480.943
1414.474      231.000
  MINOR:          72.136
  UNIT: KN MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  MAJOR:        1.80E+02    3.10E+02      0.580    Cl. 13.5(a)
1        3.600
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:         Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET):
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.580    Cl. 13.9.1         1       3.600
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.801    Cl. 13.8.2         1       3.600
  MEMBER STRENGTH:             0.715    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.472    Cl. 14.6(a)        1       3.600
  BIAXIAL FLEX:                0.580    Cl. 13.8           1       3.600
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   1.80E+02  Mrz:
3.10E+02
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 9.00E+02  1.80E+02  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.60 1.76E
+04 6.06E+03 0.60 0.60 1.00 1.00
  MEM STR: 9.00E+02  1.80E+02  0.00E+00 2.23E+03 3.10E+02 1.46E+02 0.85 1.76E
+04 6.06E+03 0.60 0.60 0.63 0.70
  FLEX SHEAR:  Mfz:   1.80E+02  Mrz:   3.10E+02  Vfy:   5.00E+01  Vry:   4.52E
+02
      STAAD SPACE                                              -- PAGE NO.
7
  BIAX FLEX:   Mfz:   1.80E+02  Mrz:   3.10E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Axial Tension

Design of a double-angle member subject to tension load.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 3.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

Design a tension diagonal in an all-welded truss (`SNU 0` for welded). Attached to WT 265x61.5 chords (t_w = 13.1 mm).

$$L = 4 \text{ m}$$

$$T_f = 630 \text{ kN}$$

The material used is G40.21 300W steel (F_y = *300 MPa*, F_u = *450 MPa*) (`FYLD` and `FU`)

Calculations

Gross area is used for welded connections.

$$T_r = \max\begin{cases} \phi AF_y = 0.9 \times 300 \times A = 270A \\ \phi_u AF_u = 0.75 \times 450 \times A = 338A \end{cases}$$

$T_r \geq T_f$, so solve for A:

$$A = \frac{630 \times (10)^3}{270} = 2,\ 333\text{mm}^2$$

Select a pair of angles with at least this gross area: try a 2- 76x64x9.5, with long legs back-to-back:

$$A = 2{,}480 \text{ mm}^2$$

$$r_{min} = 13.3 \text{ mm}$$

Calculate the effective net area due to shear lag. The average weld length is taken as *(120 mm + 250 mm)/2 = 185 mm*.

The long leg does not have a reduction in area shear lag, thus:

$$A_{n2} = 1.00 \times (76 - 9.5) \times 9.5 = 632 \text{ mm}^2$$

The outstanding (short) leg has an eccentricity of half the leg length, or *64/2 = 32 mm*.

$$A_{n3} = \left(1 - \frac{32}{250}\right) \times 64 \times 9.5 = 530\text{mm}^2$$

The total net area for the pair of angles for the rupture limit state is:

$$A_{ne} = 632 + 530 = 1{,}162 \text{ mm}^2$$

For the pair of angles, A_{ne} = *2×1,1162 = 2,324 mm²* ; the ratio of net section area to gross area = 2,324 / 2,480 = 0.937 (`NSF`).

Checking the yielding limit state:

$$T_r = 0.90 \times 2{,}480 \times 300 = 670 \text{ kN}$$

Checking the rupture limit state:

$$T_r = 0.75 \times 2{,}324 \times 450 = 784 \text{ kN}$$

Yielding governs, and the critical ratio is: $630/670 = 0.94$.

Comparison

Table 613: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
T_f (kN)	670	669.6	negligible
Critical Ratio	0.94	0.941	negligible

Note: If the member SELECT facility is used in place of a code CHECK, STAAD.Pro will actually select a 2-L 102x89x6.4 member, which has a critical ratio of 0.997 (per Cl. 13.2) and an area of 2,340 mm^2, thus more economical (the NSF used for this size is 0.923).

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Axial Tension.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Apr-14
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE LD L76X64X9.5
*1 TABLE T W530x123
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
```

```
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 300000 ALL
FU 450000 ALL
SNUG 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.941(PASS)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.9.1
  SECTION: LD   L76X64X9.5              (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.941(PASS)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.9.1
  DESIGN FORCES:   Fx:    630.00(T)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      11.195 AYY:      13.616 CW:
0.000
                            SZZ:      26.578 SYY:      26.229
                            IZZ:     137.969 IYY:     166.556
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      300.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):     4.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     169.588  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE:         Class 3
  FLANGE:          Class 3
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:      630.000     669.600      0.941    Cl. 13.2
1       0.000
  RUPTURE:       630.000     837.000      0.753    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
```

```
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.000     141.791      0.000    Cl. 13.3
1        0.000
  MINOR:           0.000     166.729      0.000    Cl. 13.3
1        0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):          λ            n:
  MAJOR:          24.800    169.796       70.178      2.068        1.340
  MINOR:          24.800    154.539       84.719      1.882        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
                  0.000     164.708      0.000    Cl. 13.3           1
0.000
  INTERMEDIATE:   Fe(N MM):       λ            n:
                  83.506    1.895       1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:           0.000     226.444      0.000    Cl. 13.4.3
1        0.000
  MINOR:           0.000     196.070      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          13.616    1.200       0.000         0.000
0.000       0.000
  MINOR:          11.195
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    7.18E+00      0.000    Cl. 13.5(b)
1        0.000
  MINOR:        0.00E+00    7.08E+00      0.000    Cl. 13.5(b)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        1.44E+01    0.00E+00    0.80E+01
  MINOR:        0.00E+00    0.00E+00    0.79E+01
  UNIT: KN   MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    1.17E+01      0.000    Cl. 13.6(e)(i)
1        0.000
  INTERMEDIATE:  Iyc(CM):      w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        4.07E+01  2.50   1.00     0.59   3.65E+01  0.02   5.58E+00
0.52  25.18
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.941    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.000    Cl. 13.8.3         1       0.000
  MEMBER STRENGTH:             0.000    Cl. 13.8.3         1       0.000
  LTB STRENGTH:                0.000    Cl. 13.8.3         1       0.000
  FLEX AND SHEAR:              0.000    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.000    Cl. 13.8           1       0.000
      STAAD SPACE                                             -- PAGE NO.
```

```
7
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   6.30E+02  Tr:   6.70E+02  Mfz:   0.00E+00  Mrz:
7.18E+00
  FLEXURE AND AXIAL COMPRESSION:
              Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00  0.00E+00  0.00E+00 6.70E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  MEM STR: 0.00E+00  0.00E+00  0.00E+00 1.42E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  LTB STR: 0.00E+00  0.00E+00  0.00E+00 1.65E+02 7.18E+00 7.08E+00  N/A  1.74E
+02 2.11E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   7.18E+00  Vfy:   0.00E+00  Vry:   2.26E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   7.18E+00  Mfy:   0.00E+00  Mry:   7.08E
+00
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Beam Bending

Determine the uniformly factored load that the member can resist.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W310x52 beam spans 7.3 m. Both ends of the beam are supported by columns connected standard web angle connections.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Section Classification

Evaluate the slenderness effects of the beam flanges:

$$\frac{b}{2t} = \frac{167}{2 \times 13.2} = 6.3 < \frac{170}{\sqrt{350}} = 9.1$$

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{318 - 2(13.2)}{7.6} = 38.4 < \frac{1,\ 700}{\sqrt{350}} = 90.9$$

Both the beam flanges and web are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 5.7.

Bending Capacity

Factored moment resistance is:

$$M_r = \phi Z_x F_y = 0.90 \times 841 \times (10)^3 \times 350 = 265 \times (10)^3 \text{ N}\cdot\text{mm} = 265 \text{ kN}\cdot\text{m}$$

Equate this to the bending moment due to a uniformly distributed load and solve for the load:

$$M_r \geq \frac{w_f L^2}{8}$$

$$w_f = \frac{8M_r}{L^2} = \frac{8 \times 265}{(7.3)^2} = 39.8 \text{ kN/m}$$

Comparison

Table 614: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
M_r (kN·m)	265	265	none
w_f (kN/m)	39.8	39.76*	negligible

Note: (*) STAAD.Pro does calculate the allowable uniform load on the beam. Instead, this is done by applying a uniform load incrementally until the critical ratio is ≈ 1.0. The, this uniform load is divided by the resulting critical ratio to normalize the distributed load capacity of the beam. Thus, $39.8/1.001 = 39.67\ kN/m$

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Beam Bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -39.8
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
   WARNING: The current units are in KN   METE. All the design results are in
KN   MET  (unless otherwise noted).
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
 *MEMBER NO:     1    CRITICAL RATIO:  1.001(FAIL)     LOAD:     1
  LOCATION : 3.65  CONDITION: Cl. 13.8
  SECTION: ST   W310X52                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  1.001(FAIL)     LOAD CASE:     1  LOCATION : 3.65
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:  -2.65E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      44.088 AYY:      23.165 CW:
237980.875
                            SZZ:     748.428 SYY:     123.353
                            IZZ:   11900.002 IYY:    1030.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):     7.300
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
```

```
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     185.766  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE      MAJOR         MINOR
  SECTION:    Class 1        Class 1
  FLANGE:     Class 1        Class 1
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    2101.050      0.000    Cl. 13.2
1        0.000
  RUPTURE:         0.000    2251.125      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                                -- PAGE NO.
6
                        STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                        ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.000    1587.513      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:           0.000     322.454      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):         λ            n:
  MAJOR:          65.224     54.653     677.375      0.719        1.340
  MINOR:          65.224    185.766      58.630      2.443        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
                  0.000    1023.539      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe(N MM):       λ            n:
                 253.108    1.176        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        -145.270     502.453      0.289    Cl. 13.4.1.1
1        7.300
  MINOR:           0.000     833.263      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          23.165    5.348       0.044       326.992
653.857      231.000
  MINOR:          44.088
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        2.65E+02    2.65E+02      1.001    Cl. 13.5(a)
1        3.650
  MINOR:        0.00E+00    5.95E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:         Se:         My:
  MAJOR:        2.94E+02    0.00E+00    0.00E+00
  MINOR:        6.62E+01    0.00E+00    0.00E+00
  UNIT: KN   MET
```

```
  INTERACTION:                  RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                  1.001    Cl. 13.9.1         1       3.650
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                  0.851    Cl. 13.8.2         1       3.650
  MEMBER STRENGTH:               0.851    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:                0.729    Cl. 14.6(a)        1       3.042
  BIAXIAL FLEX:                  1.001    Cl. 13.8           1       3.650
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.10E+03  Mfz:   2.65E+02  Mrz:
2.65E+02
      STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00  2.65E+02  0.00E+00 2.05E+03 2.65E+02 5.95E+01 0.85 4.52E
+03 3.91E+02 0.60 0.60 1.00 1.00
  MEM STR: 0.00E+00  2.65E+02  0.00E+00 3.22E+02 2.65E+02 5.95E+01 0.85 4.52E
+03 3.91E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   2.58E+02  Mrz:   2.65E+02  Vfy:   2.42E+01  Vry:   5.02E
+02
  BIAX FLEX:   Mfz:   2.65E+02  Mrz:   2.65E+02  Mfy:   0.00E+00  Mry:   5.95E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Beam Shear Capacity

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.3

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W530x82 beam has a maximum shear force, $v = 540\ kN$ due to factored loads.

$$L = 11.0 \text{ m}$$

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{528 - 2(13.3)}{9.5} = 52.8 < 439\sqrt{\frac{k_v}{F_y}} = 439\sqrt{\frac{5.34}{350}} = 54.2$$

Therefore the shear capacity is calculated as:

$$V_r = \phi \times A_w \times F_y = 0.9 \times 528 \times 9.5 \times 350 = 1{,}043 \text{ kN}$$

Calculate Equivalent Concentrated Load

The given end moments and assumed distributed loads (which was determined to given an equivalent mid-span moment of the given example) do not result in a shear force of 540 kN. Therefore, an additional concentrated load is added near the first support.

$$R_1 = \frac{w_f L}{2} + \frac{M_1 - M_2}{L} = \frac{40.17 \times 11.0}{2} + \frac{540 - 185}{11.0} = 253.2 \text{ kN}$$

$$P = 540 \text{ kN} - 253.2 \text{ kN} = 287 \text{ kN}$$

The STAAD.Pro will have this load applied at 1 mm away from the left support.

Comparison

Table 615: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
V_r (kN)	1,043	1,042.826	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Beam Shear Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
 *MEMBER NO:     1    CRITICAL RATIO:  1.803(FAIL)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W530X82                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  1.803(FAIL)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   254.22      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      55.594 AYY:      48.896 CW:
1340255.500
                            SZZ:    1806.818 SYY:     194.258
                            IZZ:   47700.008 IYY:    2030.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     250.172  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 2        Class 2
  FLANGE:     Class 2        Class 2
  WEB:               Class 1
```

```
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    3307.500      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3543.750      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.000    2253.274      0.000    Cl. 13.3.5(a)
1       0.000
  MINOR:           0.000     253.222      0.000    Cl. 13.3.5(a)
1       0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):         λ            n:
  MAJOR:          89.688     51.609      759.622      0.679        1.340
  MINOR:          89.688    250.172       32.328      3.290        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
                  0.000     854.388      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe(N MM):       λ            n:
                 125.187    1.672         1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        -538.417    1042.826      0.516    Cl. 13.4.1.1       1
11.000
  MINOR:           0.000    1050.727      0.000    AISC G2-1
1       0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          48.896    5.349       0.048       237.748
345.654      231.000
  MINOR:          55.594
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:       -5.40E+02    6.49E+02      0.832    Cl. 13.5(a)
1       0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1       0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        7.21E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:       -5.40E+02    2.99E+02      1.803    Cl. 13.6(a)(ii
1       0.000
  INTERMEDIATE:  Iyc(CM):      w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.29   0.00     0.00   3.33E+02  0.00   0.00E+00
0.00   0.00
  UNIT: KN   MET
```

```
  INTERACTION:                    RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                    0.832    Cl. 13.9.1        1        0.000
  MEMBER STRENGTH:                 1.803    Cl. 13.9.2        1        0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                    0.707    Cl. 13.8.2        1        0.000
  MEMBER STRENGTH:                 0.707    Cl. 13.8.2        1        0.000
  LTB STRENGTH:                    1.533    Cl. 13.8.2        1        0.000
  FLEX AND SHEAR:                  1.422    Cl. 14.6(a)       1        0.000
  BIAXIAL FLEX:                    1.803    Cl. 13.8          1        0.000
      STAAD SPACE                                                  -- PAGE NO.
7
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.31E+03  Mfz:  -5.40E+02  Mrz:
6.49E+02
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 2.83E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.25E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.60 0.74 1.00 1.00
  LTB STR: 0.00E+00 -5.40E+02  0.00E+00 2.53E+02 2.99E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   2.99E+02  Vfy:   2.54E+02  Vry:   1.04E
+03
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   2.99E+02  Mfy:   0.00E+00  Mry:   9.54E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Select a Beam

Select a wide-flange shape for the given beam span.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 5.2

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A beam must be selected to span 11.0 m. The end moment of the span are:

$$M_1 = -540 \text{ kN}\cdot\text{m}$$

$$M_2 = -185 \text{ kN}\cdot\text{m}$$

And the maximum moment in the span is:

$$M_3 = 256 \text{ kN}\cdot\text{m}$$

The beam is braced at 2.5 m from the center of each supporting column (thus $L_b = 6.0\ m$) (`LT 6.0`).

The material used is G40.21 350W steel (F_y = *350 MPa*) (`FYLD`)

Calculations

Assume a Class 2 section:

$$M_r = \phi M_p = \phi Z_x F_y$$

Thus,

$$Z_x \geq \frac{540(10)^6}{0.9 \times 350} = 1,\ 714 \times (10)^3\ \mathrm{mm}^3$$

Try a W460x82:

Z_x = 1,830×$(10)^3$ mm^3, tf = 16.0 mm, bf = 191.0 mm, tw = 9.9 mm, D = 460.0 mm

Evaluate the slenderness effects of the beam flanges:

$$\frac{b}{2t} = \frac{191}{2 \times 16.0} = 5.7 < \frac{170}{\sqrt{350}} = 9.1$$

Evaluate the slenderness effects of the beam web:

$$\frac{h}{w} = \frac{460 - 2(16.0)}{9.9} = 43.2 < \frac{1,\ 700}{\sqrt{350}} = 90.9$$

The assumptions are valid and this beam is sufficient to support the given moments.

$$M_r = 0.9 \times 1{,}830 \times (10)^3 \times 350\ [(10)^{-6}] = 576.5\ \mathrm{kN \cdot m}$$

Ratio = *540 / 576.5 = 0.937*

Comparison

Table 616: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Section	W460x82 *	W460x82	none
M_r (kN·m)	576.5 **	576	negligible
Critical Ratio	0.937 **	0.937	none

Notes:

*The reference also tries a W530x82, which is stiffer for the same weight. STAAD.Pro selects the shallower beam.
** The reference does not calculate the resisting moment capacity nor the critical ratio for either section size, but they are evaluated here for completeness.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Select a Beam.STD` is typically installed with the program.

In order to model this beam in STAAD.Pro, we need to determine the distributed load which would result in the same mid-span moment on a beam with the given end moments. From the AISC Steel Construction Manual, p 3-222:

$$M_3 = \frac{w_f L^2}{8} - \frac{(M_1 + M_2)}{2} + \frac{(M_1 - M_2)^2}{2 w_f L^2}$$

$$256 = \frac{w_f (11.0)^2}{8} - \frac{(540 + 185)}{2} + \frac{(540 - 185)^2}{2 w_f (11.0)^2}$$

Solving this yields $w_f = 40.033\ kN/m$

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -40.033
JOINT LOAD
1 MZ 540
2 MZ -185
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
SELECT ALL
```

```
PERFORM ANALYSIS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.832(PASS)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W530X82                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.832(PASS)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   252.45      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      55.594 AYY:      48.896 CW:
1340255.500
                            SZZ:    1806.818 SYY:     194.258
                            IZZ:   47700.008 IYY:    2030.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     250.172  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 2        Class 2
  FLANGE:     Class 2        Class 2
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:         FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    3307.500      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3543.750      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:     FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.000    2253.274      0.000    Cl. 13.3.5(a)
1       0.000
  MINOR:           0.000     253.222      0.000    Cl. 13.3.5(a)
1       0.000
```

```
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):         λ           n:
  MAJOR:          89.688     51.609     759.622      0.679        1.340
  MINOR:          89.688    250.172      32.328      3.290        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
                  0.000     854.388      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe(N MM):       λ           n:
                 125.187    1.672        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:         252.454    1042.826      0.242    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1050.727      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          48.896    5.349       0.048       237.748
345.654      231.000
  MINOR:          55.594
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:       -5.40E+02    6.49E+02      0.832    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        7.21E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.832    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.707    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.707    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.715    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.832    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.31E+03  Mfz:  -5.40E+02  Mrz:
6.49E+02
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:      Cr:      Mrz:      Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 2.83E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.74 0.60 1.00 1.00
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.25E+03 6.49E+02 9.54E+01 0.85 7.98E
+03 3.39E+02 0.60 0.74 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   6.49E+02  Vfy:   2.52E+02  Vry:   1.04E
+03
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   6.49E+02  Mfy:   0.00E+00  Mry:   9.54E
```

```
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
    45. SELECT ALL
 STEEL DESIGN
   WARNING: The current units are in KN   METE. All the design results are in
KN   MET  (unless otherwise noted).
      STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO MEMBER SELECTION - S16-14 (v1.0)
                     *********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.937(PASS)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.8
  SECTION: ST   W460x82                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.937(PASS)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.8
  DESIGN FORCES:   Fx:      0.00(T)    Fy:   252.45      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   5.40E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      61.120 AYY:      43.956 CW:
915745.312
                            SZZ:    1608.696 SYY:     194.764
                            IZZ:   37000.004 IYY:    1860.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     260.108  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  300.000
  SECTION CLASS:
  COMPRESSION:     Class 4
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 1        Class 1
  FLANGE:     Class 1        Class 1
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:         FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    3276.000      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3510.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                              -- PAGE NO.
9
                     STAAD.PRO MEMBER SELECTION - (          S16-14)   v1.0
                     **********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:     FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
```

```
LOCATION:
  MAJOR:           0.000    2261.248      0.000    Cl. 13.3.5(a)
1        0.000
  MINOR:           0.000     253.374      0.000    Cl. 13.3.5(a)
1        0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):          λ            n:
  MAJOR:          96.728     58.319      594.890      0.767        1.340
  MINOR:          96.728    260.107       29.905      3.421        1.340
  FLEX TOR BUCK:  FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
                  0.000    1194.071      0.000    Cl. 13.3.5(a)      1
0.000
  INTERMEDIATE:   Fe(N MM):       λ            n:
                 176.181    1.409        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:         252.454     946.776      0.267    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1155.168      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          43.956    5.347       0.042       290.187
514.950      231.000
  MINOR:          61.120
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:       -5.40E+02    5.76E+02      0.937    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    9.54E+01      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        6.41E+02    0.00E+00    0.00E+00
  MINOR:        1.06E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.937    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.796    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.796    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.802    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.937    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   3.28E+03  Mfz:  -5.40E+02  Mrz:
5.76E+02
      STAAD SPACE                                              -- PAGE NO.
10
                     STAAD.PRO MEMBER SELECTION - (          S16-14)   v1.0
                     ***********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 0.00E+00 -5.40E+02  0.00E+00 3.05E+03 5.76E+02 9.54E+01 0.85 6.19E
+03 3.11E+02 0.74 0.60 1.00 1.00
```

```
  MEM STR: 0.00E+00 -5.40E+02  0.00E+00 2.26E+03 5.76E+02 9.54E+01 0.85 6.19E
+03 3.11E+02 0.60 0.74 1.00 1.00
  FLEX SHEAR:  Mfz:  -5.40E+02  Mrz:   5.76E+02  Vfy:   2.52E+02  Vry:   9.47E
+02
  BIAX FLEX:   Mfz:  -5.40E+02  Mrz:   5.76E+02  Mfy:   0.00E+00  Mry:   9.54E
+01
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Shear Capacity Combined Stresses

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 8.6

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W250x73 column has the following concentrated loads

$C_f = 900$ kN

And a uniform moment of:

$M_{fx} = 180$ kN·m

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`). The beam is braced against lateral-torsional buckling only at the ends (`LAT 0`, default)

Calculations

Axial Capacity

Determine the buckling load of the column:

$$\left(\frac{KL}{r}\right)_x = \frac{1.0(3,\ 600)}{110} = 32.7$$

$$\left(\frac{KL}{r}\right)_y = \frac{1.0(3,\ 600)}{64.6} = 55.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)_x^2} = \frac{\pi^2(200,\ 000)}{(55.7)^2} = 635.6\text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{635.6}} = 0.742$$

$$C_r = \phi A F_y\left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.742^{(2\times1.34)}\right]^{-1/1.34} = 2,\ 216 \times (10)^6\text{ N} = 2,\ 216\text{ kN}$$

Moment Modifier

Uniform moment:

$$\omega_1 = 1.0$$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{\pi^2 \times 200,\ 000 \times 113 \times 10^6}{(1.0 \times 3,\ 600)^2} = 17,\ 211\text{kN}$$

$$U_{1x} = \frac{1.0}{1 - \frac{900}{17,\ 211}} = 1.06$$

Bending Capacity

$$\omega_2 = 1.0$$

$$EI_yGJ = 200{,}000 \times 38.8 \times 76.92 \times 575 = 343.2 \times 10^{21}\ \text{MPa}^2 \cdot \text{mm}^8$$

$$\left(\frac{\pi E}{L}\right)^2 I_y C_w = \left(\frac{\pi \times 200,\ 000}{3,\ 600}\right)^2 38.8 \times 553 = 653.6 \times 10^{21}\ \text{MPa}^2 \cdot \text{mm}^8$$

$$M_u = \frac{\omega_2 \pi}{L}\sqrt{EI_yGJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w}$$

$$= \frac{1.0 \times \pi}{3{,}600}\sqrt{343.2 \times 10^{21} + 653.6 \times 10^{21}} = 871 \times 10^6\ \text{N}\cdot\text{mm} = 871\ \text{kN}\cdot\text{m}$$

Plastic moment capacity:

$$M_p = Z_x F_y = 985 \times 10^3 \times 350 = 345 \times 10^6\ \text{N}\cdot\text{mm} = 345\ \text{kN}\cdot\text{m}$$

$$M_u = 871\ \text{kN}\cdot\text{m} > 0.67 \times M_p = 0.67 \times 345 = 231\ \text{kN}\cdot\text{m}$$

Therefore, the moment capacity is calculated as:

$$M_r = 1.15\phi M_p\left(1 - \frac{0.28M_p}{M_u}\right) = 1.15 \times 0.9 \times 345\left(1 - \frac{0.28 \times 345}{871}\right) = 317\ \text{kN}\cdot\text{m}$$

Check the capacity based on the strength cross-section:

$$M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310\ \text{kN}\cdot\text{m}$$

This governs capacity.

Combined Stress Ratio

$$\frac{900}{2,\ 216} + \frac{0.85 \times 1.06 \times 180}{310} = 0.406 + 0.523 = 0.929 < 1.0$$

Comparison

Table 617: Verification Problem comparison

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_x$	33	32.624	negligible

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_y$	56	55.675	negligible
C_r (kN)	2,216	2,220	negligible
C_e (kN)	17,211	17,200	negligible
Stress Ratio	0.929	0.926	negligible

Note: Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Shear Capacity Combined Stresses.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.00e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
*MEMBER PROPERTY CANADIAN
*1 TABLE ST W250X73
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY 900 MZ 180
2 FY -900 MZ -180
```

```
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
SSY 1 ALL
SSZ 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|-------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W250X73          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.926         LOADCASE    1      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8.2         |
|-------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                 |
|   CRITICAL RATIO:  0.926(PASS)            LOAD CASE:     1              |
|   LOCATION      :   0.00                  CONDITION: Cl. 13.8.2          |
|-------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      0.00             |
|         FX:    900.00(C)    FY:     0.00      FZ:      0.00             |
|         MX:  0.000E+00      MY: 0.000E+00     MZ:  1.800E+02           |
|-------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   0.00)            UNIT:      CM              |
|   AXX:      92.800     AZZ:      72.136      AYY:        21.758         |
|   IXX:      57.500     IZZ:   11300.001      IYY:       3880.000        |
|   PZZ:     985.000     PYY:     463.000                                 |
|   SZZ:     893.281     SYY:     305.512      CW :     552900.312        |
|-------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                         UNIT:      N MM            |
|   FYLD:     350.000     FU:   450.000      E: 199999.984  G:  76920.000 |
|-------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                      |
|   ACTUAL MEMBER LENGTH:     3.600        LZ:    3.600    LY:    3.600   |
|   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000        |
|-------------------------------------------------------------------------|
|  SECTION CLASS                                                          |
|   COMPRESSION    :            Class 1                                   |
|   FLEXURE            MAJOR              MINOR                           |
|   SECTION        : Class 2            Class 2                           |
|   FLANGE         : Class 2            Class 2                           |
|   WEB (CRITICAL) :            Class 1                                   |
|=========================================================================|
|  SLENDERNESS                                                            |
|   ACTUAL SLENDERNESS RATIO   :      55.675          LC   :        1     |
|   ALLOWABLE SLENDERNESS RATIO:     200.000          LOC  :      0.00    |
|-------------------------------------------------------------------------|
      STAAD SPACE                                                -- PAGE NO.
6
```

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                         |
|--------------------------------------------------------------------------|
|  YIELDING                                                                |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   2.92E+03       0.000      Cl. 13.2          1     0.000    |
|  RUPTURE                                                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   3.13E+03       0.000      Cl. 13.2          1     0.000    |
|--------------------------------------------------------------------------|
|                       CHECKS FOR AXIAL COMPRESSION                       |
|--------------------------------------------------------------------------|
|  FLEX BUCKLING                                                           |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:    |
|   MAJOR:   9.00E+02   2.71E+03     0.332  Cl. 13.3            1    0.000 |
|   MINOR:   9.00E+02   2.22E+03     0.406  Cl. 13.3            1    0.000 |
|  INTERMEDIATE RESULTS:                                                   |
|               Ag:(CM)     Fe:(N MM)       λ:        n:      KL/r:       |
|   MAJOR:      92.800        1854.621       0.434    1.34      32.624     |
|   MINOR:      92.800         636.808       0.741    1.34      55.675     |
|--------------------------------------------------------------------------|
|  FLEXURAL TORSIONAL BUCKLING                                             |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    9.00E+02   2.22E+03       0.406      Cl. 13.3          1     0.000    |
|  INTERMEDIATE RESULTS:                                                   |
|   Ag:(CM)    92.800   Fe:(N MM)      636.808  λ:     0.741  n: 1.34      |
|--------------------------------------------------------------------------|
|                             CHECKS FOR SHEAR                             |
|--------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                      |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   4.52E+02       0.000    Cl. 13.4.1.1        1     0.000    |
|  INTERMEDIATE RESULTS:                                                   |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)     |
|     21.758    5.360     0.070       480.943      1414.474    231.000     |
|--------------------------------------------------------------------------|
|  SHEAR ALONG Z AXIS                                                      |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   1.36E+03       0.000    AISC G2-1           1     0.000    |
|  INTERMEDIATE RESULTS:                                                   |
|   Aw:(CM)                                                                |
|     72.136                                                               |
|--------------------------------------------------------------------------|
     STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                            CHECKS FOR BENDING                            |
|--------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
```

```
|    -1.80E+02  3.103E+02      0.580    Cl. 13.5(a)         1     0.000    |
|  INTERMEDIATE RESULTS: Mp:  3.45E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00  1.458E+02      0.000    Cl. 13.5(a)         1     0.000     |
|  INTERMEDIATE RESULTS: Mp:  1.62E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|   -1.80E+02  3.103E+02      0.580   Cl. 13.6(a)(i)      1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:     
Lyr:  |
|  0.00E+00   1.00  0.00     0.00  8.712E+02  0.00  0.00E+00  0.00   0.00  |
|--------------------------------------------------------------------------|
|                         CHECKS FOR INTERACTION                           |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                               |
|                    RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:    |
|   C/S STRENGTH   : 0.493  Cl. 13.9.2      1   0.00   0.00E+00  2.92E+03  |
|   MEMBER STRENGTH: 0.580  Cl. 13.9.3      1   0.00                       |
|                        Mfz:         Mfy:         Mrz:         Mry:       |
|   C/S STRENGTH   :   -1.80E+02      0.00E+00     3.10E+02      1.46E+02  |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                           |
|                    RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:    |
|   C/S STRENGTH   : 0.828  Cl. 13.8.2      1   0.000 9.00E+02  2.92E+03   |
|   MEMBER STRENGTH: 0.852  Cl. 13.8.2      1   0.000 9.00E+02  2.71E+03   |
|   LTB STRENGTH   : 0.926  Cl. 13.8.2      1   0.000 9.00E+02  2.22E+03   |
|                        Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w 
1y:    |
|   C/S STRENGTH   : -1.80E+02  0.00E+00  3.10E+02  1.46E+02  1.00   0.60  |
|   MEMBER STRENGTH: -1.80E+02  0.00E+00  3.10E+02  1.46E+02  0.60   1.00  |
|   LTB STRENGTH   : -1.80E+02  0.00E+00  3.10E+02  1.46E+02  1.00   0.60  |
|                        b :         Cez:     Cey:         U1z:     U1y:      |
|   C/S STRENGTH   :    0.60     1.72E+04  5.91E+03     1.06     1.00       |
|   MEMBER STRENGTH:    0.85     1.72E+04  5.91E+03     1.06     0.71       |
|   LTB STRENGTH   :    0.85     1.72E+04  5.91E+03     1.06     0.71       |
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     0.00E+00  4.523E+02      0.000    Cl. 14.6            1     0.000    |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:  |
|   0.580  Cl. 13.8     1    0.00  -1.80E+02  3.10E+02  0.00E+00  1.46E+02 |
|==========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL 
MEMBERS
```

V. CSA S16-14 - Short Column Compression

Determine the factored compressive resistance of a short column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.1

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W250x73 section is used for a pedestal with a height of 1.1 m.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{1,\ 100}{110} = 10$$

$$\frac{L}{r_y} = \frac{1,\ 100}{64.6} = 17$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$\lambda = \left(\frac{KL}{r}\right)_{max} \sqrt{\frac{F_y}{\pi^2 E}} = 17\sqrt{\frac{350}{\pi^2 200,\ 000}} = 0.227$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.227^{(2\times1.34)}\right]^{-1/1.34} = 2.883 \times (10)^6 \text{ N} = 2,\ 883 \text{ kN}$$

Comparison

Table 618: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	10	9.968	negligible
KL/r, minor	17	17.012	negligible
λ, critical	0.227	0.224	1.3%
C_r (kN)	2,883	2,884.358	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Short Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
*1 TABLE ST L203x203x25
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
*2 FY -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
```

```
CODE CANADIAN 2014
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.000(PASS)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.000(PASS)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:      0.00(C)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):     1.100
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      17.012  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 2        Class 2
  FLANGE:     Class 2        Class 2
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.001    2913.816      0.000    Cl. 13.3
1       0.000
```

```
  MINOR:           0.001    2884.358      0.000    Cl. 13.3
1        0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):         λ            n:
  MAJOR:          92.800      9.968    20360.980      0.131        1.340
  MINOR:          92.800     17.012    6991.204      0.224        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:           0.000     452.349      0.000    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          20.537    5.552       0.224       489.474
1465.103      231.000
  MINOR:          72.136
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    3.10E+02      0.000    Cl. 13.5(a)
1        0.000
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    3.55E+02      0.000    Cl. 13.6(a)(i)
1        0.000
  INTERMEDIATE:  Iyc(CM):      w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.50   0.00     0.00   1.98E+04  0.00   0.00E+00
0.00   0.00
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.000    Cl. 13.9.1         1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.000    Cl. 13.8.2         1       0.000
  MEMBER STRENGTH:             0.000    Cl. 13.8.2         1       0.000
  LTB STRENGTH:                0.000    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.000    Cl. 14.6(a)        1       0.000
  BIAXIAL FLEX:                0.000    Cl. 13.8           1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   0.00E+00  Mrz:
3.10E+02
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 1.00E-03  0.00E+00  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.69 1.89E
```

```
+05 6.49E+04 0.60 0.60 1.00 1.00
  MEM STR: 1.00E-03  0.00E+00  0.00E+00 2.88E+03 3.10E+02 1.46E+02 0.69 1.89E
+05 6.49E+04 0.60 0.60 1.00 1.00
  LTB STR: 1.00E-03  0.00E+00  0.00E+00 2.88E+03 3.10E+02 1.46E+02 0.69 1.89E
+05 6.49E+04 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   3.10E+02  Vfy:   0.00E+00  Vry:   4.52E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   3.10E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Slender Column Compression

Determine the factored compressive resistance of a slender column.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 4.2

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

A W250x73 section is used for a pedestal with a height of 11.0 m.

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (FYLD)

Calculations

The nominal yield strength is used for the actual yield strength for a Group 2 section, per Table 3 and Table 4 of Appendix A of the reference.

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Column Capacity

Assume K = 1.0,

$$\frac{L}{r_x} = \frac{11,000}{110} = 100$$

$$\frac{L}{r_y} = \frac{11,\ 000}{64.6} = 170$$

The largest slenderness ration controls, thus the slenderness factor for buckling about the minor axis:

$$F_e = \frac{\pi^2 \times 200,\ 000}{(1.0 \times 170)^2} = 68.3 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_e}} = \sqrt{\frac{350}{68.3}} = 2.26$$

When n = 1.34 (Group 2 W shape),

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 2.26^{(2 \times 1.34)}\right]^{-1/1.34} = 0.529 \times (10)^6 \text{ N} = 529 \text{ kN}$$

Comparison

Table 619: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
KL/r, major	100	99.684	negligible
KL/r, minor	170	170.118	negligible
λ, critical	2.26	2.237	1.0%
C_r (kN)	529	538.161	1.7%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Slender Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Apr-14
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
```

```
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -0.001
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
   WARNING: The current units are in KN   METE. All the design results are in
KN   MET  (unless otherwise noted).
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.000(PASS)     LOAD:     1
  LOCATION : 0.00  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.000(PASS)     LOAD CASE:     1  LOCATION : 0.00
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:      0.00(C)    Fy:     0.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:   0.00E+00
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
  ACTUAL MEMBER LENGTH (MET ):    11.000
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     170.118  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
```

```
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 2        Class 2
  FLANGE:     Class 2        Class 2
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1       0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1       0.000
      STAAD SPACE                                          -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:           0.001    1266.716      0.000    Cl. 13.3
1       0.000
  MINOR:           0.001     538.161      0.000    Cl. 13.3
1       0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):          λ            n:
  MAJOR:          92.800     99.684     203.610      1.311        1.340
  MINOR:          92.800    170.118      69.912      2.237        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:           0.000     452.349      0.000    Cl. 13.4.1.1
1       0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1       0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          20.537    5.342       0.023       480.151
1409.819      231.000
  MINOR:          72.136
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    3.10E+02      0.000    Cl. 13.5(a)
1       0.000
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1       0.000
  INTERMEDIATE:   Mp:         Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  LAT TOR BUCK:   FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        0.00E+00    2.83E+02      0.000    Cl. 13.6(a)(i)
1       0.000
  INTERMEDIATE:  Iyc(CM):      w 2:     w 3:      b x:     Mu:     rt:
Myr:     Lu:     Lyr:
  MAJOR:        0.00E+00  2.50   0.00      0.00   4.66E+02  0.00   0.00E+00
0.00   0.00
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
```

```
  C/S STRENGTH:                0.000    Cl. 13.9.1        1       0.000
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.000    Cl. 13.8.2        1       0.000
  MEMBER STRENGTH:             0.000    Cl. 13.8.2        1       0.000
  LTB STRENGTH:                0.000    Cl. 13.8.2        1       0.000
  FLEX AND SHEAR:              0.000    Cl. 14.6(a)       1       0.000
  BIAXIAL FLEX:                0.000    Cl. 13.8          1       0.000
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   0.00E+00  Mrz:
3.10E+02
      STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 1.00E-03  0.00E+00  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  MEM STR: 1.00E-03  0.00E+00  0.00E+00 5.38E+02 3.10E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  LTB STR: 1.00E-03  0.00E+00  0.00E+00 5.38E+02 2.83E+02 1.46E+02 0.85 1.89E
+03 6.49E+02 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   0.00E+00  Mrz:   2.83E+02  Vfy:   0.00E+00  Vry:   4.52E
+02
  BIAX FLEX:   Mfz:   0.00E+00  Mrz:   2.83E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
```

V. CSA S16-14 - Wide Flange Capacity Combined Stresses

Check the capacity of a wide-flange column under combined stresses.

Reference

Kulak, G.L. and G.Y. Grondin. 2010. *Limit States Design in Structural Steel*. 9th Edition. Markham, ON:CISC. Example 8.5

Related Links

- *D4.E. Canadian Codes - Steel Design per CAN/CSA-S16-09/14/19* (on page 1447)

Problem

The W250x73 column has the following concentrated loads applied at the top of the column:

$C_f = 900$ kN

$M_{fx} = 180$ kN·m

The material used is G40.21 350W steel ($F_y = 350\ MPa$) (`FYLD`). The column is assumed to be braced laterally against lateral-torsional buckling (`LAT 1`).

L = 3.6 m

Calculations

Local Element Buckling

Evaluate the slenderness effects of the column web:

$$\frac{h}{w} = \frac{225}{8.6} = 26.2 < \frac{670}{\sqrt{350}} = 35.8$$

Evaluate the slenderness effects of the column flanges:

$$\frac{b}{2t} = \frac{254}{2 \times 14.2} = 8.9 < \frac{200}{\sqrt{350}} = 10.7$$

Both the column web and flanges are less than the local buckling limit, so the capacity of the column is evaluated using Eq. 4.21.

Axial Capacity

Determine the buckling load of the column, assuming that weak axis bucking will not control due to bracing:

$$\left(\frac{KL}{r}\right)_x = \frac{1.0(3,\ 600)}{110} = 32.7$$

The largest slenderness ratio controls:

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)_x^2} = \frac{\pi^2(200,\ 000)}{(32.7)^2} = 1,\ 847 \text{ kN}$$

$$\lambda = \sqrt{\frac{F_y}{F_{ey}}} = \sqrt{\frac{350}{1,\ 847}} = 0.435$$

$$C_r = \phi A F_y \left(1 + \lambda^{2n}\right)^{-1/n} = 0.90 \times 9,\ 280 \times 350\left[1 + 0.435^{(2 \times 1.34)}\right]^{-1/1.34} = 2,\ 716 \times (10)^6 \text{ N} = 2,\ 716 \text{ kN}$$

Moment Modifier

$$\kappa = \text{-}0/180 = 0$$

$$\omega_1 = 0.6 - 0.4\kappa = 0.6 \leq 1.0$$

Calculate the elastic buckling load for the bending axis:

$$C_e = \frac{\pi^2 \times 200,\ 000 \times 113 \times 10^6}{(1.0 \times 3,\ 600)^2} = 17,\ 211\text{kN}$$

$$U_{1x} = \frac{0.6}{1 - \left(\frac{900}{17,\ 210}\right)} = 0.633$$

Bending Capacity

From a previous example, the W250x73 is a Class 2 section. It is laterally supported along its length, thus:

$$M_{rx} = 0.9 \times 985 \times 10^3 \times 350 = 310 \text{ kN} \cdot \text{m}$$

Combined Stress Ratio

$$\frac{900}{2,\ 716} + \frac{0.85 \times 0.633 \times 180}{310} = 0.331 + 0.312 = 0.644 < 1.0$$

This is the check for overall member strength. However, there is another case that actually controls in this instance: checking for cross sectional strength.

$$C_r = 0.9 \times 9{,}280 \times 350 = 2{,}923 \text{ kN}$$

$$U_{1x} = 1.0$$

$$\frac{900}{2,\ 923} + \frac{0.85 \times 1.0 \times 180}{310} = 0.308 + 0.494 = 0.801 < 1.0$$

Comparison

Table 620: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
$(KL/r)_x$	33	32.624	negligible
$(KL/r)_y$	56	55.675	negligible
C_r (kN)	2,716	2,715.935	negligible
C_e (kN)	17,211	17,600	2.3%
Stress Ratio	0.644	0.643	none
Stress Ratio (Critical)	0.801	0.801	none

Note: Both the reference and hand calculations agree with the STAAD.Pro results that the section is adequate. The reference however neglects to account for section capacity checks, which control in this case.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-14 - Wide Flange Capacity Combined Stresses.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Sep-13
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
```

```
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
*1 ENFORCED BUT MX MY MZ
1 PINNED
2 ENFORCED BUT FY MX MZ
*2 ENFORCED BUT FX FZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY 900
*1 MZ 180
2 FY -900 MZ 180
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2014
FYLD 350000 ALL
LAT 1 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - S16-14 (v1.0)
                     *******************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  -------------------- START OF DESIGN OUTPUT OF MEMBER      1
--------------------
  MEMBER NO:     1    CRITICAL RATIO:  0.896(PASS)     LOAD:     1
  LOCATION : 3.60  CONDITION: Cl. 13.8.2
  SECTION: ST   W250X73                 (CANADIAN SECTIONS)
  UNIT: KN   MET
  STRENGTH CHECKS:
  CRITICAL RATIO:  0.896(PASS)     LOAD CASE:     1  LOCATION : 3.60
CONDITION: Cl. 13.8.2
  DESIGN FORCES:   Fx:    900.00(C)    Fy:    50.00      Fz:      0.00
                   Mx:   0.00E+00      My:  0.00E+00     Mz:  -1.80E+02
  UNIT: CM
  SECTION PROPERTIES:       AZZ:      72.136 AYY:      20.537 CW:
552900.312
                            SZZ:     893.281 SYY:     305.512
                            IZZ:   11300.001 IYY:    3880.000
  UNIT: NEW MM
  MATERIAL PROPERTIES:      FYLD (N MM):      350.000   FU (N MM):
450.000
```

```
  ACTUAL MEMBER LENGTH (MET ):     3.600
  PARAMETERS:                  KZ:  1.000  KY:  1.000  NSF:  1.000  SLF:
1.000
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      55.675  LOAD:     1   LOC.:
0.000
                 ALLOWABLE SLENDERNESS RATIO:  200.000
  SECTION CLASS:
  COMPRESSION:     Class 1
  FLEXURE      MAJOR          MINOR
  SECTION:    Class 2        Class 2
  FLANGE:     Class 2        Class 2
  WEB:               Class 1
  UNIT: KN   MET
  TENSION:        FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  YIELDING:        0.000    2923.200      0.000    Cl. 13.2
1        0.000
  RUPTURE:         0.000    3132.000      0.000    Cl. 13.2
1        0.000
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (          S16-14)   v1.0
                       ********************************************
 ALL UNITS ARE - KN   MET  (UNLESS OTHERWISE Noted)
  COMPRESSION:    FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION:
  MAJOR:         900.000    2715.935      0.331    Cl. 13.3
1        0.000
  MINOR:         900.000    2233.940      0.403    Cl. 13.3
1        0.000
  INTERMEDIATE:   Ag(CM):     KL/r:    Fe(N MM):         λ           n:
  MAJOR:          92.800     32.624     1900.987      0.429        1.340
  MINOR:          92.800     55.675      652.728      0.732        1.340
  SHEAR:          FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:          50.000     452.349      0.111    Cl. 13.4.1.1
1        0.000
  MINOR:           0.000    1363.370      0.000    AISC G2-1
1        0.000
  INTERMEDIATE:  Aw(CM):    Kv:          Ka:         Fcri(N MM):    Fcre(N
MM):  Fs(N MM):
  MAJOR:          20.537    5.360       0.070       480.943
1414.474      231.000
  MINOR:          72.136
  UNIT: KN   MET
  YIELDING:       FORCE:    CAPACITY:    RATIO:    CRITERIA:    LOAD CASE:
LOCATION(MET ):
  MAJOR:        1.80E+02    3.10E+02      0.580    Cl. 13.5(a)
1        3.600
  MINOR:        0.00E+00    1.46E+02      0.000    Cl. 13.5(a)
1        0.000
  INTERMEDIATE:   Mp:        Se:         My:
  MAJOR:        3.45E+02    0.00E+00    0.00E+00
  MINOR:        1.62E+02    0.00E+00    0.00E+00
  UNIT: KN   MET
  INTERACTION:                RATIO:    CRITERIA:    LOAD CASE:   LOCATION:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:                0.580    Cl. 13.9.1        1       3.600
```

```
  FLEXURE AND AXIAL COMPRESSION:
  C/S STRENGTH:                0.801    Cl. 13.8.2         1       3.600
  MEMBER STRENGTH:             0.896    Cl. 13.8.2         1       0.000
  FLEX AND SHEAR:              0.472    Cl. 14.6(a)        1       3.600
  BIAXIAL FLEX:                0.580    Cl. 13.8           1       3.600
  INTERMEDIATE:
  FLEXURE AND AXIAL TENSION:
  C/S STRENGTH:   Tf:   0.00E+00  Tr:   2.92E+03  Mfz:   1.80E+02  Mrz:
3.10E+02
  FLEXURE AND AXIAL COMPRESSION:
             Cf:       Mfz:       Mfy:     Cr:     Mrz:     Mry:     b :
Cez:     Cey:    w 1z:  w 1y: U1z: U1y:
  C/S STR: 9.00E+02  1.80E+02  0.00E+00 2.92E+03 3.10E+02 1.46E+02 0.85 1.76E
+04 6.06E+03 0.60 0.60 1.00 1.00
  MEM STR: 9.00E+02  1.80E+02  0.00E+00 2.23E+03 3.10E+02 1.46E+02 0.85 1.76E
+04 6.06E+03 0.60 0.60 1.00 1.00
  FLEX SHEAR:  Mfz:   1.80E+02  Mrz:   3.10E+02  Vfy:   5.00E+01  Vry:   4.52E
+02
      STAAD SPACE                                             -- PAGE NO.
7
  BIAX FLEX:   Mfz:   1.80E+02  Mrz:   3.10E+02  Mfy:   0.00E+00  Mry:   1.46E
+02
  -------------------- END OF DESIGN OUTPUT OF MEMBER      1
--------------------
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V. CSA S16-19 - Axial Comp and Bending Interaction

Check a wide flange column member subject to both axial compression and bending per the CSA S16-19 code.

References

Canadian Institute of Steel Construction. *Handbook of Steel Construction.* 9th Edition. 2008. Willowdale, Ont., pp. 6-52 - 6-53

Details

The column is simply supported and has a span of 3.6 m. The member is subjected to an axial load of 900 kN along with moments in both strong and weak directions of 50 kN·m at the roller end. The section used is a Canadian W250x73. The steel has a yield strength of 350 MPa (50 ksi) and ultimate strength of 450 MPa (65 ksi).

Section Properties

From the *Handbook of Steel Construction*:

$D = 253\ mm$
$B = 254\ mm$
$t_w = 8.6\ mm$
$t_f = 14.2\ mm$
$A = 9{,}280\ mm^2$
$I_x = 113\times(10)^6\ mm^4$
$I_y = 38.8\times(10)^6\ mm^4$
$S_x = 891\times(10)^3\ mm^3$

$S_y = 306\times(10)^3\ mm^3$
$Z_x = 985\times(10)^3\ mm^3$
$Z_y = 463\times(10)^3\ mm^3$
$r_x = 110\ mm$
$r_y = 64.6\ mm$
$C_w = 553\times(10)^9\ mm^6$
$J = 575\times(10)^3\ mm^4$

Material Properties

$E = 205{,}000\ MPa$
$G = 76{,}920\ MPa$

Calculations

Section Classification

$$b_{el} = B/2 = 127 \text{ mm}$$

$$h = D - 2\times t_f = 224.6 \text{ mm}$$

Compression section classification as per Table 1:

Flange classification:

$$b_{el}/t = {}^{127}\!/_{14.2} = 8.94 < \frac{200}{\sqrt{F_y}} = 10.69\text{, i.e. Class 1}$$

Web classification:

$$h\ /w = {}^{224.6}\!/_{8.6} = 26.11 < \frac{670}{\sqrt{F_y}} = 35.81\text{, i.e. Class 1}$$

Section is Class 1 for compression.

Flexural section classification as per Table 2

Flange classification (major and minor axes):

$$\frac{145}{\sqrt{F_y}} = 7.75 < b_{el}/t = {}^{127}\!/_{14.2} = 8.94 < \frac{170}{\sqrt{F_y}} = 9.09\text{, i.e. Class 2}$$

Web classification (major and minor axes):

$$h\ /w = {}^{224.6}\!/_{8.6} = 26.11 < \frac{525}{\sqrt{F_y}} = 28.06\text{, i.e. Class 1 } \textit{if} \text{ the condition } \frac{M_{fy}}{S_y} > \frac{0.9M_{fx}}{S_x}$$

$$\frac{M_{fy}}{S_y} = \frac{50}{306(10)^3} = 0.163(10)^{-3} > \frac{M_{fx}}{S_x} = \frac{50}{891(10)^3} = 0.051(10)^{-3}\text{, thus OK.}$$

Section is Class 2 for flexure.

Axial Compression Capacity

Check for axial compression per Cl. 13.3.1.1:

$$C_r = \frac{\phi A F_y}{\left(1 + \lambda^{2n}\right)^{1/n}} \qquad (13.3.1.1)$$

where

n = *1.34* for I sections

λ = $\sqrt{\frac{F_y}{F_e}}$

$$F_{ex} = \frac{\pi^2 E}{(KL_x / r_x)^2} = \frac{\pi^2 \times 205{,}000}{(1.0 \times 3{,}600 / 110)^2} = 1{,}890 \text{ MPa}$$

$$\lambda_x = \sqrt{\frac{350}{1{,}890}} = 0.430$$

$$C_{rx} = \frac{0.9 \times 9{,}280 \times 350}{\left[1 + 0.430^{(2\times1.34)}\right]^{1/1.34}} 10^{-3} = 2{,}715 \text{ kN}$$

$$F_{ey} = \frac{\pi^2 E}{(KL_y / r_y)^2} = \frac{\pi^2 \times 205{,}000}{(1.0 \times 3{,}600 / 64.6)^2} = 651 \text{ MPa}$$

$$\lambda_y = \sqrt{\frac{350}{651}} = 0.733$$

$$C_{ry} = \frac{0.9 \times 9{,}280 \times 350}{\left[1 + 0.733^{(2\times1.34)}\right]^{1/1.34}} 10^{-3} = 2{,}233 \text{ kN}$$

Check for flexural-torsional buckling per Cl. 13.3.1.2:

$$F_{ez} = \left[\frac{\pi^2 E C_w}{(KL_z)^2} + GJ\right]\frac{1}{A\bar{r}_0^2} \quad (13.3.1.2)$$

where

$\bar{r}_0^2$ = $x_0^2 + y_0^2 + r_x^2 + r_y^2$; since the section is doubly-symmetric, $x_0 = y_0 = 0$

$$\bar{r}_0^2 = (110)^2 + (64.6)^2 = 16{,}273 \text{ mm}^2$$

$$F_{ez} = \left[\frac{\pi^2 \times 205{,}000 \times 553 \times (10)^9}{(1.0 \times 3{,}600)^2} + 76{,}920 \times 575 \times (10)^3\right]\frac{1}{9{,}280 \times 16{,}273} = 865 \text{ MPa}$$

For flexural torsional buckling, F_e is the least of F_{ex}, F_{ey}, and F_{ez}. Therefore, $F_e = 651\ MPa$.

$$\lambda = 0.733$$

$$C_r = C_{ry} = 2{,}233 \text{ kN}$$

The critical ratio = C_f / C_r = 900 / 2,233 = 0.403

<u>Bending Capacity</u>

Check the bending capacity for a Class 2 section as per Cl. 13.5.a:

About major axis:

$$M_{px} = Z_x \times F_y = 985 \times (10)^3 \times 350 \times 10^{-6} = 344.8 \text{ kN}\cdot\text{m}$$

$$M_{rx} = \phi M_{px} = 0.9\ (344.8) = 310.3 \text{ kN}\cdot\text{m}$$

Ratio = M_{fx} / M_{rx} = 50 / 310.3 = 0.161

About minor axis:

$M_{py} = Z_y \times F_y = 463 \times (10)^3 \times 350 \times 10^{-6} = 162.1$ kN·m

$M_{ry} = \phi M_{py} = 0.9\ (162.1) = 145.8$ kN·m

Ratio = M_{fy} / M_{ry} = 50 / 145.8 = 0.343

<u>Combined Axial Compression & Bending</u>

Check cross-section strength for a Class 2 section per Cl. 13.8.2-a:

$$\frac{C_f}{C_r} + \frac{0.85 U_{1x} M_{fx}}{M_{rx}} + \frac{\beta U_{1y} M_{fy}}{M_{ry}} \le 1.0 \quad (13.8.2)$$

where

C_r = C_r as per Cl. 13.3 with the value of $\lambda = 0$ (Cl. 13.8.2-a-i), which reduces to:

$= \phi A F_y = 0.9 \times 9{,}280 \times 350 \times (10)^{-3} = 2{,}923$ kN

β = $0.6 + 0.4 \times \lambda_y \le 0.85$

$0.6 + 0.4\ (0.733) = 0.893$; thus limit $\beta = 0.85$

About both major and minor axes, the moment at the “start” joint, $M_A = 0$ and the moment at the “end” joint, $M_B = 50\ kN{\cdot}m$. Thus, $K = M_A\ /\ M_B = 0$.

$$\omega_1 = 0.6 - 0.4 \times K = 0.6 \quad (13.8.6)$$

$$U_1 = \frac{\omega_1}{1 - {C_f}/{C_e}} \ge 1.0 \quad (13.8.5\ \&\ 13.8.2\text{-a-iii})$$

where

C_e = $\dfrac{\pi^2 EI}{L^2}$

$$C_{ez} = \frac{\pi^2 E I_z}{L^2} = \frac{\pi^2 205{,}000 \times 113 \times (10)^3}{(3{,}600)^2} = 17{,}640 \text{ kN}$$

$$C_{ey} = \frac{\pi^2 E I_y}{L^2} = \frac{\pi^2 205{,}000 \times 38.8 \times (10)^3}{(3{,}600)^2} = 6{,}057 \text{ kN}$$

$$U_{1z} = \frac{0.6}{1 - {900}/{17{,}640}} 0.632 < 1.0$$

$$U_{1y} = \frac{0.6}{1 - {900}/{6{,}057}} 0.705 < 1.0$$

Thus, $U_1 = 1.0$ about both major and minor axes.

$$\frac{900}{2{,}923} + \frac{0.85 \times 1.0 \times (50)}{310.3} + \frac{0.85 \times 1.0 \times (50)}{145.8} = 0.736 \le 1.0$$

Check overall member strength for a Class 2 section per Cl. 13.8.2-b:

C_r used is taken from the axial compression calculations ($K = 1$).
Use U_{1z} and $U_{1y} = 1.0$ for an unbraced frame.

$$\frac{900}{2{,}233} + \frac{0.85 \times 1.0 \times (50)}{310.3} + \frac{0.85 \times 1.0 \times (50)}{145.8} = 0.831 \le 1.0$$

Check moment-only capacity for biaxial bending per Cl. 13.8.2-d:

$$\frac{M_{\mathrm{fz}}}{M_{\mathrm{rz}}} + \frac{M_{\mathrm{fy}}}{M_{\mathrm{ry}}} = \frac{50}{310.3} + \frac{50}{145.8} = 0.504 \leq 1.0$$

Comparison

Table 621: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
Elastic buckling stress about the major axis, F_{ex} (MPa)	1,890	1,900.987	negligible	
Elastic buckling stress about the minor axis, F_{ey} (MPa)	651	652.728	negligible	
Non-dimensional slenderness parameter about major axis, λ_x	0.430	0.429	negligible	
Non-dimensional slenderness parameter about minor axis, λ_y	0.733	0.732	negligible	
Factored axial compressive resistance about major axis, C_{rx} (kN)	2,715	2,720	negligible	
Factored axial compressive resistance about major axis, C_{ry} (kN)	2,233	2,230	negligible	
Flexural torsional buckling stress, F_{ez} (MPa)	651	652.728	negligible	
Factored axial compressive resistance in flexural torsional buckling, C_r (kN)	2,233	2,230	negligible	

Criteria	Reference	STAAD.Pro	Difference	Comments
Factored moment resistance about the major axis, M_{rx} (kN·m)	310.3	310.3	none	
Factored moment resistance about the minor axis, M_{ry} (kN·m)	145.8	145.8	none	
Factored axial compressive stress used for cross-section strength, C_r (kN)	2,923	2,920	negligible	
Elastic buckling strength about major axis, C_{ez} (kN)	17,640	17,600	negligible	
Elastic buckling strength about minor axis, C_{ey} (kN)	6,057	6,060	negligible	
β	0.85	0.85	none	
Cross-sectional strength ratio	0.736	0.736	none	
Factored axial compressive stress used for overall member strength, C_r (kN)	2,233	2,230	negligible	
Overall member strength ratio	0.831	0.831	none	
Biaxial flexure ratio	0.504	0.504	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Axial Comp and Bending Interaction.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- Full lateral support is specified using `LAT 1`.

- Steel strength values are specified using `FYLD 350000` and `FU 450000` (the units are kN/m^2 or kPa).

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Jan-22
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -900 MX 50 MZ 50
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|-----------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W250X73          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.831         LOADCASE    1       |
|  LOCATION:      3.60    CRITICAL CONDITION:            Cl. 13.8.2          |
|-----------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                     |
```

```
||   CRITICAL RATIO:  0.831(PASS)            LOAD CASE:     1                 |
||   LOCATION      :   3.60                  CONDITION: Cl. 13.8.2            |
||------------------------------------------------------------------------- |
||  DESIGN FORCES              LC:       1      LOC:      3.60               |
||         FX:    900.00(C)    FY:    13.89      FZ:     13.89                |
||         MX:  0.000E+00      MY: 5.000E+01     MZ: -5.000E+01               |
||------------------------------------------------------------------------- |
||  SECTION PROPERTIES (LOC:   3.60)           UNIT:      CM                 |
||   AXX:      92.800     AZZ:      72.136      AYY:        21.758           |
||   IXX:      57.500     IZZ:   11300.001      IYY:       3880.000          |
||   PZZ:     985.000     PYY:     463.000                                   |
||   SZZ:     893.281     SYY:     305.512      CW :     552900.312          |
||------------------------------------------------------------------------- |
||  MATERIAL PROPERTIES                         UNIT:      N MM              |
||   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000  |
||------------------------------------------------------------------------- |
||  DESIGN PARAMETERS                                                        |
||   ACTUAL MEMBER LENGTH:     3.600        LZ:    3.600    LY:    3.600    |
||   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000         |
||------------------------------------------------------------------------- |
||  SECTION CLASS                                                            |
||   COMPRESSION    :            Class 1                                     |
||   FLEXURE           MAJOR              MINOR                              |
||   SECTION        : Class 2            Class 2                             |
||   FLANGE         : Class 2            Class 2                             |
||   WEB (CRITICAL) :            Class 2                                     |
||========================================================================= |
||  SLENDERNESS                                                              |
||   ACTUAL SLENDERNESS RATIO   :      55.675          LC   :        1      |
||   ALLOWABLE SLENDERNESS RATIO:     200.000          LOC  :      0.00     |
||------------------------------------------------------------------------- |
      STAAD SPACE                                               -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
  Member :     1       Contd.
|--------------------------------------------------------------------------|
|                           CHECKS FOR AXIAL TENSION                       |
|--------------------------------------------------------------------------|
|  YIELDING                                                                |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   2.92E+03       0.000      Cl. 13.2          1     0.000    |
|  RUPTURE                                                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   3.13E+03       0.000      Cl. 13.2          1     0.000    |
|--------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                     |
|--------------------------------------------------------------------------|
|  FLEX BUCKLING                                                           |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:    |
|   MAJOR:   9.00E+02   2.72E+03     0.331  Cl. 13.3           1    0.000  |
|   MINOR:   9.00E+02   2.23E+03     0.403  Cl. 13.3           1    0.000  |
|  INTERMEDIATE RESULTS:                                                   |
|               Ag:(CM)     Fe:(N MM)      λ:         n:      KL/r:       |
|   MAJOR:      92.800       1900.987       0.429    1.34      32.624      |
|   MINOR:      92.800        652.728       0.732    1.34      55.675      |
||--------------------------------------------------------------------------|
```

```
||  FLEXURAL TORSIONAL BUCKLING                                                |
|    FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|     9.00E+02   2.23E+03       0.403      Cl. 13.3           1     0.000     |
|   INTERMEDIATE RESULTS:                                                     |
|    Ag:(CM)    92.800   Fe:(N MM)      652.728  λ:     0.732  n: 1.34        |
|-----------------------------------------------------------------------------|
|                            CHECKS FOR SHEAR                                 |
|-----------------------------------------------------------------------------|
|   SHEAR ALONG Y AXIS                                                        |
|    FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|     1.39E+01   4.52E+02       0.031    Cl. 13.4.1.1        1     0.000      |
|   INTERMEDIATE RESULTS:                                                     |
|    Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)       |
|      21.758    5.360     0.070       480.943      1414.474    231.000       |
|-----------------------------------------------------------------------------|
|   SHEAR ALONG Z AXIS                                                        |
|    FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|    -1.39E+01   1.36E+03       0.010    AISC G2-1           1     0.000      |
|   INTERMEDIATE RESULTS:                                                     |
|    Aw:(CM)                                                                  |
|      72.136                                                                 |
|-----------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                      ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-----------------------------------------------------------------------------|
|                           CHECKS FOR BENDING                                |
|-----------------------------------------------------------------------------|
|   YIELDING ALONG MAJOR AXIS                                                 |
|    FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|      5.00E+01  3.103E+02      0.161    Cl. 13.5(a)         1     3.600      |
|   INTERMEDIATE RESULTS: Mp:  3.45E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|-----------------------------------------------------------------------------|
|   YIELDING ALONG MINOR AXIS                                                 |
|    FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|     5.00E+01  1.458E+02      0.343    Cl. 13.5(a)         1     3.600       |
|   INTERMEDIATE RESULTS: Mp:  1.62E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|-----------------------------------------------------------------------------|
|                         CHECKS FOR INTERACTION                              |
|-----------------------------------------------------------------------------|
|   FLEXURE AND AXIAL TENSION                                                 |
|                    RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:       |
|    C/S STRENGTH   : 0.343  Cl. 13.9.2      1   3.60   0.00E+00  2.92E+03    |
|                        Mfz:          Mfy:          Mrz:          Mry:       |
|    C/S STRENGTH   :    5.00E+01      5.00E+01     3.10E+02      1.46E+02    |
|-----------------------------------------------------------------------------|
|   FLEXURE AND AXIAL COMPRESSION                                             |
|                    RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:       |
|    C/S STRENGTH   : 0.736  Cl. 13.8.2      1   3.600 9.00E+02  2.92E+03     |
|    MEMBER STRENGTH: 0.831  Cl. 13.8.2      1   0.000 9.00E+02  2.23E+03     |
|                        Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|    C/S STRENGTH   :  5.00E+01  5.00E+01  3.10E+02  1.46E+02  0.60   0.60    |
|    MEMBER STRENGTH:  5.00E+01  5.00E+01  3.10E+02  1.46E+02  0.60   0.60    |
|                        b :         Cez:     Cey:         U1z:     U1y:        |
```

```
|   C/S STRENGTH   :    0.85     1.76E+04  6.06E+03     1.00     1.00       |
|   MEMBER STRENGTH:    0.85     1.76E+04  6.06E+03     1.00     1.00       |
|---------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                          |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
|     1.39E+01  4.523E+02      0.031    Cl. 14.6           1     0.000       |
|---------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                          |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:   |
|   0.504  Cl. 13.8     1    3.60   5.00E+01  3.10E+02  5.00E+01  1.46E+02 |
|===========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V. CSA S16-19 - Axial Tension

Determine the capacity of a member subject to axial tension load per the CSA S16-19 code.

Problem

A 4 m long member is subjected to a 630 kN tensile load. The section is a Canadian W100x19. The steel is grade 300W.

Section Properties

A_g = 2,480 mm^2

Material Properties

F_y = 300 MPa
F_u = 450 MPa

Calculations

Assume the gross area for both yielding and rupture limit states. Therefore, $A_{ne} = A_g = 2{,}480\ mm^2$.

$$T_r = \min\begin{cases}\phi A_g F_y = 0.9 \times 300 \times 2{,}480(10)^{-3} = 669.6\text{ kN}\\ \phi_u A_{ne} F_u = 0.75 \times 450 \times 2{,}480(10)^{-3} = 837\text{ kN}\end{cases}$$

So, the tensile capacity = $669.6\ kN$.

The critical stress ratio $630 / 669.6 = 0.941$.

Comparison

Table 622: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
Yielding tension capacity, T_r (kN)	669.6	669.6	none	

Criteria	Reference	STAAD.Pro	Difference	Comments
Rupture tension capacity, T_r (kN)	837	837	none	
Critical Ratio	0.941	0.941	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Axial Tension.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- The parameter `SNUG 0` is used for welded or pre-tenstioned bolts (i.e., no reduction for bolt holes)
- The default value of `NSF 1` is used for net section equal to the gross area.
- The default value of `SLF 1` is used for a shear lag factor of 1 (i.e., no shear lag in connection)

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Sep-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W100x19
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 630
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
```

```
FYLD 300000 ALL
FU 450000 ALL
SNUG 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:       1      SECTION: ST  W100X19          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.941          LOADCASE    1      |
|  LOCATION:      0.00    CRITICAL CONDITION:             Cl. 13.9.2         |
|---------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                    |
|   CRITICAL RATIO:  0.941(PASS)             LOAD CASE:     1               |
|   LOCATION      :   0.00                   CONDITION: Cl. 13.9.2          |
|---------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      0.00               |
|         FX:    630.00(T)    FY:     0.00      FZ:      0.00               |
|         MX:  0.000E+00      MY: 0.000E+00     MZ:  0.000E+00              |
|---------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   0.00)              UNIT:      CM              |
|   AXX:      24.800     AZZ:      18.128      AYY:          7.526          |
|   IXX:       6.360     IZZ:     477.000      IYY:        161.000          |
|   PZZ:     103.000     PYY:      48.000                                   |
|   SZZ:      90.000     SYY:      31.262      CW :       3785.434          |
|---------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                           UNIT:      N MM            |
|   FYLD:     300.000     FU:   450.000      E: 204999.984  G:  76920.000   |
|---------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                         |
|   ACTUAL MEMBER LENGTH:     4.000        LZ:    4.000    LY:    4.000     |
|   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000          |
|---------------------------------------------------------------------------|
|  SECTION CLASS                                                             |
|   COMPRESSION    :            Class 1                                     |
|   FLEXURE           MAJOR              MINOR                              |
|   SECTION        : Class 1            Class 1                             |
|   FLANGE         : Class 1            Class 1                             |
|   WEB (CRITICAL) :            Class 1                                     |
|===========================================================================|
|  SLENDERNESS                                                               |
|   ACTUAL SLENDERNESS RATIO   :     156.990          LC   :        1       |
|   ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00      |
|---------------------------------------------------------------------------|
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|---------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL TENSION                          |
```

```
||--------------------------------------------------------------------------|
| YIELDING                                                                  |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   6.30E+02   6.70E+02       0.941      Cl. 13.2          1     0.000      |
| RUPTURE                                                                   |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   6.30E+02   8.37E+02       0.753      Cl. 13.2          1     0.000      |
|--------------------------------------------------------------------------|
|                      CHECKS FOR AXIAL COMPRESSION                         |
|--------------------------------------------------------------------------|
| FLEX BUCKLING                                                             |
|           FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:      |
|  MAJOR:   0.00E+00   3.57E+02     0.000  Cl. 13.3           1    0.000    |
|  MINOR:   0.00E+00   1.62E+02     0.000  Cl. 13.3           1    0.000    |
| INTERMEDIATE RESULTS:                                                     |
|              Ag:(CM)     Fe:(N MM)       λ:         n:      KL/r:         |
|  MAJOR:      24.800         243.221       1.111    1.34      91.207       |
|  MINOR:      24.800          82.093       1.912    1.34     156.990       |
|--------------------------------------------------------------------------|
| FLEXURAL TORSIONAL BUCKLING                                               |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   0.00E+00   1.62E+02       0.000      Cl. 13.3          1     0.000      |
| INTERMEDIATE RESULTS:                                                     |
|  Ag:(CM)    24.800   Fe:(N MM)       82.093  λ:     1.912  n: 1.34        |
|--------------------------------------------------------------------------|
|                            CHECKS FOR SHEAR                               |
|--------------------------------------------------------------------------|
| SHEAR ALONG Y AXIS                                                        |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   0.00E+00   1.34E+02       0.000    Cl. 13.4.1.1        1     0.000      |
| INTERMEDIATE RESULTS:                                                     |
|  Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)       |
|     7.526    5.343     0.026       932.501      6203.750    198.000       |
|--------------------------------------------------------------------------|
| SHEAR ALONG Z AXIS                                                        |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   0.00E+00   2.94E+02       0.000    AISC G2-1           1     0.000      |
| INTERMEDIATE RESULTS:                                                     |
|  Aw:(CM)                                                                  |
|    18.128                                                                 |
|--------------------------------------------------------------------------|
     STAAD SPACE                                                -- PAGE NO.
7
                    STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                  ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
  Member :     1       Contd.
|--------------------------------------------------------------------------|
|                           CHECKS FOR BENDING                              |
|--------------------------------------------------------------------------|
| YIELDING ALONG MAJOR AXIS                                                 |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|    0.00E+00  2.781E+01      0.000    Cl. 13.5(a)         1     0.000      |
| INTERMEDIATE RESULTS: Mp:  3.09E+01   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
| YIELDING ALONG MINOR AXIS                                                 |
|  FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|   0.00E+00  1.296E+01      0.000    Cl. 13.5(a)         1     0.000       |
```

```
|  INTERMEDIATE RESULTS: Mp:  1.44E+01   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00  2.781E+01      0.000   Cl. 13.6(a)(i)       1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:    
Lyr:  |
|  0.00E+00   2.50  0.00     0.00  8.267E+01  0.00  0.00E+00  0.00   0.00 |
|--------------------------------------------------------------------------|
|                         CHECKS FOR INTERACTION                           |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                               |
|                    RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:    |
|   C/S STRENGTH   : 0.941  Cl. 13.9.2      1   0.00   6.30E+02  6.70E+02 |
|                        Mfz:         Mfy:          Mrz:         Mry:      |
|   C/S STRENGTH   :    0.00E+00      0.00E+00     2.78E+01      1.30E+01 |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                           |
|                    RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:    |
|   C/S STRENGTH   : 0.000  Cl. 13.8.2      1   0.000 0.00E+00  6.70E+02   |
|   MEMBER STRENGTH: 0.000  Cl. 13.8.2      1   0.000 0.00E+00  1.62E+02   |
|   LTB STRENGTH   : 0.000  Cl. 13.8.2      1   0.000 0.00E+00  1.62E+02   |
|                        Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w 
1y:    |
|   C/S STRENGTH   :  0.00E+00  0.00E+00  2.78E+01  1.30E+01  0.60   0.60 |
|   MEMBER STRENGTH:  0.00E+00  0.00E+00  2.78E+01  1.30E+01  0.60   0.60 |
|   LTB STRENGTH   :  0.00E+00  0.00E+00  2.78E+01  1.30E+01  0.60   0.60 |
|                       b :         Cez:     Cey:         U1z:     U1y:      |
|   C/S STRENGTH   :    0.85     6.03E+02  2.04E+02     1.00     1.00      |
|   MEMBER STRENGTH:    0.85     6.03E+02  2.04E+02     1.00     1.00      |
|   LTB STRENGTH   :    0.85     6.03E+02  2.04E+02     1.00     1.00      |
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     0.00E+00  1.341E+02      0.000    Cl. 14.6           1     0.000     |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:  |
|   0.000  Cl. 13.8     1    0.00   0.00E+00  2.78E+01  0.00E+00  1.30E+01 |
```

V. CSA S16-19 - Axial Tension and Bending Interaction

Determine the capacity of a member subject to axial tension and biaxial moment per the CSA S16-19 code.

Problem

A 3.6 m long column is subjected to a 900 kN tensile load along with concentrated moments of 50 kN·m about both major and minor axes at one end. The section is a Canadian W250x73. The steel is grade 350W.
Section Properties

D = 253 mm
B = 254 mm
t_w = 8.6 mm
t_f = 14.2 mm

A_g = 9,280 mm^2
Z_x = 985,000 mm^3
Z_y = 463,000 mm^3

Material Properties

F_y = 350 MPa
F_u = 450 MPa

Calculations

Section Classification

Compression section classification per Table 1:

$$b_{el} = B / 2 = 127 \text{ mm}$$

$$h = D - 2 \times t_f = 235.8 \text{ mm}$$

- Flange classification:

$$\frac{b_{el}}{t} = \frac{127}{14.2} = 8.94 < \frac{200}{\sqrt{F_y}} = \frac{200}{\sqrt{350}} = 10.69$$

- Web classification:

$$\frac{h}{w} = \frac{235.8}{8.6} = 27.42 < \frac{670}{\sqrt{F_y}} = \frac{670}{\sqrt{350}} = 35.81$$

Therefore, the section is Class 1 for compression.

Flexure section classification per Table 2:

- Flange classification about major axis:

$$\frac{145}{\sqrt{F_y}} = 7.75 < \frac{b_{el}}{t} = \frac{127}{14.2} = 8.94 < \frac{170}{\sqrt{350}} = 9.09$$

  Class 2
- Flange classification about minor axis:

$$\frac{145}{\sqrt{F_y}} = 7.75 < \frac{b_{el}}{t} = \frac{127}{14.2} = 8.94 < \frac{170}{\sqrt{350}} = 9.09$$

  Class 2
- Web classification about major axis:

$$\frac{h}{w} = \frac{235.8}{8.6} = 27.42 < \frac{1{,}100}{\sqrt{F_y}} = 58.8$$

  Class 1
- Web classification about minor axis:

$$\frac{h}{w} = \frac{235.8}{8.6} = 27.42 < \frac{1{,}100}{\sqrt{F_y}} = 58.8$$

  Class 1

Therefore, the section is Class 2 for flexure.

Axial Tension

Assume the gross area for both yielding and rupture limit states. Therefore, $A_{ne} = A_g = 9{,}280\ mm^2$.

$$T_r = \min \begin{cases} \phi A_g F_y = 0.9 \times 350 \times 9{,}280(10)^{-3} = 2{,}923 \text{ kN} \\ \phi_u A_{ne} F_u = 0.75 \times 450 \times 9{,}280(10)^{-3} = 3{,}132 \text{ kN} \end{cases}$$

So, the tensile capacity = $2{,}923\ kN$.

The critical stress ratio $900\ /\ 2{,}923 = 0.308$.

Bending Capacity

For a Class 2, doubly symmetric section, the capacity is determined by Cl. 13.5a:

Major axis:

$$M_{px} = Z_x \times F_y = 985{,}000 \times 350\ (10)^{-6} = 344.8 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 344.8 = 310.3 \text{ kN·m}$$

The ratio = $50\ /\ 310.3 = 0.161$

Minor axis:

$$M_{py} = Z_y \times F_y = 463{,}000 \times 350\ (10)^{-6} = 162.1 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 162.1 = 145.8 \text{ kN·m}$$

The ratio = $50\ /\ 145.8 = 0.343$

Combined Axial Tension and Bending

Check for axial tension and bending per Cl. 13.9.2:

$$\frac{T_f}{T_r} + 0.85\frac{M_{fx}}{M_{rx}} + 0.6\frac{M_{fy}}{M_{ry}} = 0.380 + 0.85\left(0.161\right) + 0.6\left(0.343\right) = 0.651 < 1.0$$

Also, the following must be satisfied:

$$\frac{M_{fx}}{M_{rx}} + \frac{M_{fy}}{M_{ry}} = 0.161 + 0.343 = 0.504 < 1.0$$

Comparison

Table 623: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
Nominal flexure capacity about major axis, M_{px} (kN·m)	344.8	345	negligible	
Ultimate flexure capacity about major axis, M_{rx} (kN·m)	310.3	310.3	none	

Criteria	Reference	STAAD.Pro	Difference	Comments
Nominal flexure capacity about minor axis, M_{py} (kN·m)	162.1	162	negligible	
Ultimate flexure capacity about minor axis, M_{rx} (kN·m)	145.8	145.8	none	
Tension capacity, T_r (kN)	2,932	2,930	negligible	
Critical Ratio	0.651	0.651	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Axial Tension and Bending Interaction.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- The default value of `NSF 1` is used for net section equal to the gross area.
- The default value of `SLF 1` is used for a shear lag factor of 1 (i.e., no shear lag in connection)

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Oct-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3.6 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
```

```
2 ENFORCED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY 900 MX 50 MZ 50
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
LAT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W250X73          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.651         LOADCASE    1     |
|  LOCATION:      3.60    CRITICAL CONDITION:            Cl. 13.9.2        |
|--------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                  |
|   CRITICAL RATIO:  0.651(PASS)            LOAD CASE:     1               |
|   LOCATION      :   3.60                  CONDITION: Cl. 13.9.2          |
|--------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      3.60              |
|         FX:    900.00(T)    FY:    13.89      FZ:     13.89              |
|         MX:  0.000E+00      MY: 5.000E+01     MZ: -5.000E+01             |
|--------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   3.60)              UNIT:      CM             |
|   AXX:      92.800     AZZ:      72.136      AYY:         21.758         |
|   IXX:      57.500     IZZ:   11300.001      IYY:       3880.000         |
|   PZZ:     985.000     PYY:     463.000                                  |
|   SZZ:     893.281     SYY:     305.512      CW :     552900.312         |
|--------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                           UNIT:      N MM           |
|   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000  |
|--------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                       |
|   ACTUAL MEMBER LENGTH:     3.600        LZ:    3.600    LY:    3.600    |
|   KZ:  1.000         KY:  1.000      NSF:  1.000      SLF:  1.000        |
|--------------------------------------------------------------------------|
|  SECTION CLASS                                                           |
|   COMPRESSION    :            Class 1                                    |
|   FLEXURE           MAJOR               MINOR                            |
|   SECTION        : Class 2            Class 2                            |
|   FLANGE         : Class 2            Class 2                            |
|   WEB (CRITICAL) :            Class 1                                    |
|==========================================================================|
|  SLENDERNESS                                                             |
|   ACTUAL SLENDERNESS RATIO   :      55.675          LC   :       1       |
|   ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00     |
```

```
|-----------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-----------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL TENSION                           |
|-----------------------------------------------------------------------------|
|  YIELDING                                                                   |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    9.00E+02   2.92E+03       0.308      Cl. 13.2          1     0.000       |
|  RUPTURE                                                                    |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    9.00E+02   3.13E+03       0.287      Cl. 13.2          1     0.000       |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|  FLEX BUCKLING                                                              |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:       |
|   MAJOR:   0.00E+00   2.72E+03     0.000  Cl. 13.3            1    0.000    |
|   MINOR:   0.00E+00   2.23E+03     0.000  Cl. 13.3            1    0.000    |
|  INTERMEDIATE RESULTS:                                                      |
|               Ag:(CM)     Fe:(N MM)       λ:         n:      KL/r:         |
|   MAJOR:      92.800       1900.987       0.429    1.34      32.624        |
|   MINOR:      92.800        652.728       0.732    1.34      55.675        |
|-----------------------------------------------------------------------------|
|  FLEXURAL TORSIONAL BUCKLING                                                |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    0.00E+00   2.23E+03       0.000      Cl. 13.3          1     0.000       |
|  INTERMEDIATE RESULTS:                                                      |
|   Ag:(CM)    92.800   Fe:(N MM)      652.728  λ:     0.732  n: 1.34        |
|-----------------------------------------------------------------------------|
|                             CHECKS FOR SHEAR                                |
|-----------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    1.39E+01   4.52E+02       0.031    Cl. 13.4.1.1        1     0.000       |
|  INTERMEDIATE RESULTS:                                                      |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)       |
|     21.758    5.360     0.070       480.943      1414.474    231.000        |
|-----------------------------------------------------------------------------|
|  SHEAR ALONG Z AXIS                                                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|   -1.39E+01   1.36E+03       0.010    AISC G2-1           1     0.000       |
|  INTERMEDIATE RESULTS:                                                      |
|   Aw:(CM)                                                                   |
|     72.136                                                                  |
|-----------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-----------------------------------------------------------------------------|
|                             CHECKS FOR BENDING                              |
```

```
|--------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     5.00E+01  3.103E+02      0.161    Cl. 13.5(a)         1     3.600     |
|  INTERMEDIATE RESULTS: Mp:  3.45E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    5.00E+01  1.458E+02      0.343    Cl. 13.5(a)         1     3.600      |
|  INTERMEDIATE RESULTS: Mp:  1.62E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|                         CHECKS FOR INTERACTION                           |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                               |
|                  RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:      |
|   C/S STRENGTH   : 0.651  Cl. 13.9.2      1   3.60   9.00E+02  2.92E+03  |
|                      Mfz:         Mfy:          Mrz:          Mry:       |
|   C/S STRENGTH   :    5.00E+01      5.00E+01     3.10E+02      1.46E+02  |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                           |
|                  RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:      |
|   C/S STRENGTH   : 0.428  Cl. 13.8.2      1   3.600 0.00E+00  2.92E+03   |
|   MEMBER STRENGTH: 0.428  Cl. 13.8.2      1   0.000 0.00E+00  2.23E+03   |
|                      Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|   C/S STRENGTH   :  5.00E+01  5.00E+01  3.10E+02  1.46E+02  0.60   0.60  |
|   MEMBER STRENGTH:  5.00E+01  5.00E+01  3.10E+02  1.46E+02  0.60   0.60  |
|                      b :         Cez:     Cey:         U1z:     U1y:        |
|   C/S STRENGTH   :    0.85     1.76E+04  6.06E+03     1.00     1.00       |
|   MEMBER STRENGTH:    0.85     1.76E+04  6.06E+03     1.00     1.00       |
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     1.39E+01  4.523E+02      0.031    Cl. 14.6            1     0.000     |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:  |
|   0.504  Cl. 13.8     1    3.60   5.00E+01  3.10E+02  5.00E+01  1.46E+02 |
```

V. CSA S16-19 - Beam Shear Capacity

Check a the shear capacity of a wide flange beam per the CSA S16-19 code

References

1. Canadian Institute of Steel Construction. *Handbook of Steel Construction.* 9th Edition. 2008. Willowdale, Ont., pp.6-46 - 6-47
2. American Institute of Steel Construction, *Steel Construction Manual*. 15th Edition, American Institute of Steel Construction, 2017 p.3-213 & p.3-222
3. American Institute of Steel Construction, *Specification for Structural Steel Buildings.* AISC 360-16, American Institute of Steel Construction, 2016

Details

The beam is an 11 m long, simply-supported span. The member is subject to a uniform load of 40.033 kN/m along the length and a concentrated load of 352.27 kN at 55 mm before the right support. The member is also subjected to concentrated moments of 540 kN·m at the left support and -185 kN·m at the right support.

Assume that the web is stiffened at supports and that the member is laterally unsupported. Assume loads are factored.

The section used is a Canadian W530x82. The steel has a yield strength of 350 MPa (50 ksi) and ultimate strength of 450 MPa (65 ksi).

Section Properties

$d = 528\ mm$
$b = 209\ mm$
$t_f = 13.3\ mm$
$t_w = 9.5\ mm$
$I_y = 20.3\times(10)^6\ mm^4$
$Z_x = 2{,}060\ mm^3$
$J = 518\times(10)^3\ mm^4$
$C_w = 1{,}340\times(10)^9\ mm^6$

Material Properties

$E = 205{,}000\ MPa$
$G = 76{,}920\ MPa$

Calculations

Design Forces

Shear at the right end:

$$V_f = 352.27\left(\frac{10.945}{11}\right) + \frac{1}{2}(40.033 \times 11) + \frac{185 - 540}{11} = 350.5 + 220.2 - 49.1 = 538.4\text{ kN}$$

Moment at point of maximum shear (right end):

$$M_f = 185\text{ kN}\cdot\text{m}$$

Maximum moment (at left end):

$$M_{max} = 540\text{ kN}\cdot\text{m}$$

Moment at quarter points of span (for calculating moment gradient coefficient), calculated per Table 3-23 in the AISC *Steel Construction Manual*:

$$M_a = \frac{40.033(2.75)}{2}(11 - 2.75) + \left(\frac{540 - 185}{11}\right)2.75 - 540 + \frac{352.27(2.75)(0.055)}{11} = 7.7\text{ kN}\cdot\text{m}$$

$$M_b = \frac{40.033(5.5)}{2}(11 - 5.5) + \left(\frac{540 - 185}{11}\right)5.5 - 540 + \frac{352.27(5.5)(0.055)}{11} = 252.7\text{ kN}\cdot\text{m}$$

$$M_c = \frac{40.033(8.25)}{2}(11 - 8.25) + \left(\frac{540 - 185}{11}\right)8.25 - 540 + \frac{352.27(8.25)(0.055)}{11} = 194.9\text{ kN}\cdot\text{m}$$

Shear Capacity

Along major axis:

Web depth, $h = d - 2\times t_f = 528 - 2\times 13.3 = 501\ mm$

$$V_r = \phi A_w F_s \qquad (13.4.1.1)$$

where

A_w = area of the web, = $d \times t_w = 528 \times 9.5 = 5{,}016\ mm^2$
F_s = the shear stress capacity.

a = 11,000 mm (distance between stiffeners)

a/h = 11,000 / 501 = 21.9 > 1.0

The shear buckling coefficient, $k_v = 5.34 + 4\ /\ (a/h)^2 = 5.348$

$$\frac{h}{t_w} = \frac{501}{9.5} = 52.8 < 439\sqrt{\frac{k_v}{F_y}} = 439\sqrt{\frac{5.348}{350}} = 54.3$$

Therefore, $F_s = 0.66F_y = 0.66\ (350) = 231\ MPa$

$$V_r = 0.9 \times 5{,}016 \times 231\ (10)^{-3} = 1{,}043\ \text{kN}$$

The ratio along the major axis = $V_f\ /\ V_r = 538.4\ /\ 1{,}043 = 0.516$

Along minor axis: CSA S16-19 does not directly address the case of shear along the minor axis. Therefore, the method used in AISC 360-16 section G7. is used.

$$A_w = 2 \times b \times t_f = 2\ (209)\ (13.3) = 5{,}559\ \text{mm}^2$$

$$k_v = 1.2$$

$$\frac{h}{t_w} = \frac{b}{t_f} = \frac{209}{13.3} = 15.7 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{1.2 \times 205{,}000}{350}} = 29.2$$

Therefore, $C_v = 1.0$.

$$V_r = \phi\ 0.6A_w F_y C_v = 0.9 \times 5{,}559 \times 350 \times 1.0\ (10)^{-3} = 1{,}051\ \text{kN}$$

<u>Bending Capacity</u>

Calculate the bending capacity per Cl. 13.6.1:

$$M_p = Z_x F_y = 2{,}060 \times 350 \times 10^{-3} = 721\ \text{kN·m}$$

$$M_u = \frac{\omega_2 \pi}{L}\sqrt{EI_y GJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w} \qquad (13.6.1)$$

where

$$\omega_2 = \frac{4M_{max}}{\sqrt{M_{max}{}^2 + 4M_a{}^2 + 7M_b{}^2 + 4M_c{}^2}} = \frac{4(540)}{\sqrt{(540)^2 + 4(7.7)^2 + 7(252.7)^2 + 4(194.9)^2}} = 2.29 < 2.5$$

$$M_u = \frac{2.29 \times \pi}{11{,}000}\sqrt{205{,}000 \times \left(20.3 \times 10^6\right) \times (76{,}920) \times \left(518 \times 10^3\right) + \left(\frac{\pi \times 205{,}000}{11{,}000}\right)^2 \left(20.3 \times 10^6\right) \times \left(1{,}340 \times 1\right.}$$

$$= \frac{1}{1{,}530}\sqrt{165.8(10)^{21} + 93.24(10)^{21}} = 332.7\ \text{kN·m}$$

$$M_u < 0.67M_p = 0.67\ (721) = 483.1\ \text{kN·m}$$

Therefore, $M_r = \phi M_u = 0.9\ (\ 332.7) = 299.4\ kN{\cdot}m$

<u>Shear and Bending Interaction</u>

$$F_s = 231 > 0.60 \times F_y = 210\ \text{MPa}$$

Calculate the reduction factor for shear capacity due to combined shear and moment per Cl. 14.6:

$$\left[2.20 - 1.60\frac{M_f}{M_r}\right] = \left[2.20 - 1.60\frac{185}{299.4}\right] = 1.21 \tag{14.6}$$

Therefore, no reduction in the shear capacity to combined effects of bending and shear:

$$V_r = 1{,}043 \text{ kN}$$

Ratio = V_f / V_r = 538.4 / 1,043 = 0.516

Comparison

Table 624: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
Shear buckling coefficient, k_v	5.348	5.349	negligible	
Elastic shear stress capacity, F_s (MPa)	231	231	none	
Factored shear capacity along major axis, V_{ry} (kN)	1,043	1,040	negligible	
Shear ratio along major axis	0.516	0.516	none	
Factored shear capacity along minor axis, V_{rz} (kN)	1,051	1,050	negligible	
Factored shear capacity modified for shear and moment interaction, V_r (kN)	1,043	1,043	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Beam Shear Capacity.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- Steel strength values are specified using `FYLD 350000` and `FU 450000` (the units are kN/m^2 or kPa).

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jan-22
END JOB INFORMATION
INPUT WIDTH 79
```

```
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 11 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W530X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 ENFORCED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 MZ 540
2 MZ -185
MEMBER LOAD
1 UNI GY -40.033
1 CON GY -352.27 10.945
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2019
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|-----------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W530X82          (CANADIAN SECTIONS)|
|  STATUS:        FAIL    CRITICAL RATIO:  1.803         LOADCASE    1        |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8             |
|-----------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                     |
|   CRITICAL RATIO:  1.803(FAIL)            LOAD CASE:     1                  |
|   LOCATION      :   0.00                  CONDITION: Cl. 13.8               |
|-----------------------------------------------------------------------------|
|  DESIGN FORCES                LC:       1      LOC:      0.00               |
|         FX:      0.00(T)    FY:   254.22      FZ:      0.00                |
```

```
||        MX:  0.000E+00      MY: 0.000E+00     MZ:  5.400E+02               |
||-------------------------------------------------------------------------|
||  SECTION PROPERTIES (LOC:   0.00)            UNIT:      CM                |
||   AXX:     105.000     AZZ:      55.594      AYY:        50.160           |
||   IXX:      51.800     IZZ:   47700.008      IYY:      2030.000           |
||   PZZ:    2060.000     PYY:     303.000                                   |
||   SZZ:    1806.818     SYY:     194.258      CW :    1340255.500          |
||-------------------------------------------------------------------------|
||  MATERIAL PROPERTIES                         UNIT:      N MM              |
||   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000   |
||-------------------------------------------------------------------------|
||  DESIGN PARAMETERS                                                        |
||   ACTUAL MEMBER LENGTH:    11.000        LZ:   11.000    LY:   11.000     |
||   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000          |
||-------------------------------------------------------------------------|
||  SECTION CLASS                                                            |
||   COMPRESSION    :            Class 4                                     |
||   FLEXURE           MAJOR               MINOR                             |
||   SECTION        : Class 2            Class 2                             |
||   FLANGE         : Class 2            Class 2                             |
||   WEB (CRITICAL) :            Class 1                                     |
||=========================================================================|
||  SLENDERNESS                                                              |
||   ACTUAL SLENDERNESS RATIO   :     250.172          LC   :        1       |
||   ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00      |
||-------------------------------------------------------------------------|
      STAAD SPACE                                                -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
 |-------------------------------------------------------------------------|
 |                          CHECKS FOR AXIAL TENSION                        |
 |-------------------------------------------------------------------------|
 |  YIELDING                                                                |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
 |    0.00E+00   3.31E+03       0.000      Cl. 13.2          1     0.000    |
 |  RUPTURE                                                                 |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
 |    0.00E+00   3.54E+03       0.000      Cl. 13.2          1     0.000    |
 |-------------------------------------------------------------------------|
 |                        CHECKS FOR AXIAL COMPRESSION                      |
 |-------------------------------------------------------------------------|
 |  FLEX BUCKLING                                                           |
 |            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:    |
 |   MAJOR:   0.00E+00   2.25E+03     0.000  Cl. 13.3.4(a)       1    0.000 |
 |   MINOR:   0.00E+00   2.53E+02     0.000  Cl. 13.3.4(a)       1    0.000 |
 |  INTERMEDIATE RESULTS:                                                   |
 |               Ag:(CM)     Fe:(N MM)       λ:         n:      KL/r:       |
 |   MAJOR:      89.688         759.622       0.679    1.34      51.609     |
 |   MINOR:      89.688          32.328       3.290    1.34     250.172     |
 |-------------------------------------------------------------------------|
 |  FLEXURAL TORSIONAL BUCKLING                                             |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
 |    0.00E+00   2.53E+02       0.000      Cl. 13.3.4        1     0.000    |
 |  INTERMEDIATE RESULTS:                                                   |
 |   Ag:(CM)    89.688   Fe:(N MM)       32.328  λ:     3.290  n: 1.34      |
```

```
|-----------------------------------------------------------------------------|
|                                CHECKS FOR SHEAR                             |
|-----------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|   -5.38E+02   1.04E+03       0.516    Cl. 13.4.1.1        1    11.000       |
|  INTERMEDIATE RESULTS:                                                      |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)        |
|     50.160    5.349     0.048       237.748       345.654    231.000        |
|-----------------------------------------------------------------------------|
|  SHEAR ALONG Z AXIS                                                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    0.00E+00   1.05E+03       0.000    AISC G2-1           1     0.000       |
|  INTERMEDIATE RESULTS:                                                      |
|   Aw:(CM)                                                                   |
|     55.594                                                                  |
|-----------------------------------------------------------------------------|
      STAAD SPACE                                             -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   **********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-----------------------------------------------------------------------------|
|                               CHECKS FOR BENDING                            |
|-----------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                                  |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    -5.40E+02  6.489E+02      0.832    Cl. 13.5(a)         1     0.000       |
|  INTERMEDIATE RESULTS: Mp:  7.21E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|-----------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                                  |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    0.00E+00  9.545E+01      0.000    Cl. 13.5(a)         1     0.000        |
|  INTERMEDIATE RESULTS: Mp:  1.06E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|-----------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                    |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|   -5.40E+02  2.994E+02      1.803   Cl. 13.6(a)(ii)      1     0.000        |
|  INTERMEDIATE RESULTS:                                                      |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:
Lyr:  |
|  0.00E+00   2.29  0.00     0.00  3.327E+02  0.00  0.00E+00  0.00   0.00   |
|-----------------------------------------------------------------------------|
|                             CHECKS FOR INTERACTION                          |
|-----------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                                  |
|                   RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:       |
|   C/S STRENGTH   : 0.707  Cl. 13.9.2      1   0.00   0.00E+00  3.31E+03    |
|   MEMBER STRENGTH: 1.803  Cl. 13.9.3      1   0.00                         |
|                       Mfz:         Mfy:          Mrz:         Mry:         |
|   C/S STRENGTH   :   -5.40E+02      0.00E+00     6.49E+02      9.54E+01    |
|-----------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                              |
|                   RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:       |
|   C/S STRENGTH   : 0.707  Cl. 13.8.2      1   0.000 0.00E+00  2.83E+03     |
|   MEMBER STRENGTH: 0.707  Cl. 13.8.2      1   0.000 0.00E+00  2.25E+03     |
|   LTB STRENGTH   : 1.533  Cl. 13.8.2      1   0.000 0.00E+00  2.53E+02     |
```

```
|                        Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w 
1y:    |
|   C/S STRENGTH   : -5.40E+02  0.00E+00  6.49E+02  9.54E+01  0.74   0.60  |
|   MEMBER STRENGTH: -5.40E+02  0.00E+00  6.49E+02  9.54E+01  0.60   0.74  |
|   LTB STRENGTH   : -5.40E+02  0.00E+00  2.99E+02  9.54E+01  0.74   0.60  |
|                       b :         Cez:     Cey:        U1z:     U1y:      |
|   C/S STRENGTH   :    0.85     7.98E+03  3.39E+02     1.00     1.00      |
|   MEMBER STRENGTH:    0.85     7.98E+03  3.39E+02     1.00     1.00      |
|   LTB STRENGTH   :    0.85     7.98E+03  3.39E+02     1.00     1.00      |
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    -5.38E+02  1.043E+03      0.516    Cl. 14.6           1    11.000     |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:  |
|   1.803  Cl. 13.8     1    0.00  -5.40E+02  2.99E+02  0.00E+00  9.54E+01 |
|==========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL 
MEMBERS
```

V. CSA S16-19 - Cantilever Beam with Point Load

Determine the capacity of a cantilever beam subject to a point load per the CSA S16-19 code.

Details

A 7.5 m long cantilever member is subjected to load at the free end of 5 kN. The section is a Canadian W310x52. The steel is grade 350W. The section is unbraced along the length.

Section Properties

D = 318 mm
B = 167 mm
t_w = 7.6 mm
t_f = 13.2 mm
A_g = 9,280 mm^2
Z_x = 841,000 mm^3
Z_y = 189,000 mm^3
I_y = 1,030,00 mm^4
J = 308,000 mm^4
C_w = 238×10^9 mm^6

Material Properties

F_y = 350 MPa
F_u = 450 MPa

Validation

Section Classification

$$b_{el} = B / 2 = 83.5 \text{ mm}$$

$h = D - 2 \times t_f = 291.6$ mm

$C_f = 0$ (no compression force)

Flexure section classification per Table 2 of CSA S16-19:

- Flange classification about major axis:

$$\frac{b_{el}}{t_f} = \frac{83.5}{13.2} = 6.33 < \frac{145}{\sqrt{F_y}} = \frac{145}{\sqrt{350}} = 7.75$$

  Class 1

- Flange classification about minor axis:

$$\frac{b_{el}}{t_f} = \frac{83.5}{13.2} = 6.33 < \frac{145}{\sqrt{F_y}} = \frac{145}{\sqrt{350}} = 7.75$$

  Class 1

- Web classification about major axis:

$$\frac{h}{t_w} = \frac{291.6}{7.6} = 38.67 < \frac{1{,}100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{\phi C_y}\right) = 58.8$$

  Class 1

- Web classification about minor axis:

$$\frac{h}{t_w} = \frac{291.6}{7.6} = 38.67 < \frac{1{,}100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{\phi C_y}\right) = 58.8$$

  Class 1

Therefore, the section is Class 1 for flexure.

<u>Bending Capacity</u>

$$M_{fx} = 5 \times 7.5 = 37.5 \text{ kN·m}$$

For a Class 1, doubly symmetric section, the capacity is determined by Cl. 13.5:

Major axis:

$$M_{px} = Z_x \times F_y = 841{,}000 \times 350\ (10)^{-6} = 294.4 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 294.4 = 264.9 \text{ kN·m}$$

The ratio = *37.5 / 264.9 = 0.142*

Minor axis:

$$M_{py} = Z_y \times F_y = 189{,}000 \times 350\ (10)^{-6} = 66.2 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 66.2 = 59.5 \text{ kN·m}$$

Check for lateral-torsional buckling per Cl. 13.6.h i).

$$M_u = \omega_2 C_H C_B \frac{\sqrt{E I_y G J}}{L_c}$$

where

$$X = \frac{\pi}{L_c}\sqrt{\frac{E C_w}{GJ}} = \frac{\pi}{7{,}500}\sqrt{\frac{205{,}000 \times 238(10)^9}{76{,}920 \times 308{,}000}} = 0.601$$

$0.5 \leq X \leq 2.3$ for a point load

ω_2 = $3.95 + 3.52\ X = 3.95 + 3.52\ (0.601) = 6.07$

C_H = $0.76 - 0.51\ X + 0.13\ X^2 = 0.76 - 0.51\ (0.601) + 0.13\ (0.601)^2 = 0.50$

C_B = 1.0

$$M_u = 6.07(0.50)(1.0)\frac{\sqrt{205{,}000 \times 1{,}030{,}000 \times 76{,}920 \times 308{,}000}}{7{,}500 \times (10)^6} = 90.5\ \text{kN·m}$$

$$0.67 \times M_p = 0.67\ (294.4) = 197.2\ \text{kN·m} > M_u$$

$$M_r = \phi M_u = 0.9 \times 90.52 = 81.5\ \text{kN·m}$$

The ratio = $37.5\ /\ 81.5 = 0.46$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
M_{px} (kN·m)	294.4	294	negligible	
M_{rx} (kN·m)	264.9	264.9	none	
Ratio of M_{fx}/M_{rx}	0.142	0.142	none	
M_{py} (kN·m)	66.2	66.2	none	
M_{ry} (kN·m)	59.5	59.54	negligible	
M_u (kN·m)	90.5	90.52	negligible	
M_r (kN·m)	81.5	81.47	negligible	
ω_2	6.07	6.07	none	
Critical ratio	0.46	0.46	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Cantilever Beam with Point Load.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- The unbraced cantilever is specified using `CAN 1`.
- The load acts on the tension side of the bending member, thus `LHT 1`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Dec-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -5
PERFORM ANALYSIS
PARAMETER 1
CODE CANADIAN
CAN 1 ALL
LHT 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W310X52          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.460         LOADCASE    1          |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8               |
|--------------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                        |
|   CRITICAL RATIO:  0.460(PASS)            LOAD CASE:     1                     |
|   LOCATION      :   0.00                  CONDITION: Cl. 13.8                  |
|--------------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      0.00                    |
|         FX:      0.00(T)    FY:     5.00      FZ:      0.00                    |
|         MX:  0.000E+00      MY: 0.000E+00     MZ:  3.750E+01                   |
|--------------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   0.00)             UNIT:      CM                    |
|   AXX:      66.700     AZZ:      44.088      AYY:         24.168               |
|   IXX:      30.800     IZZ:   11900.002      IYY:       1030.000               |
|   PZZ:     841.000     PYY:     189.000                                        |
|   SZZ:     748.428     SYY:     123.353      CW :     237980.875               |
|--------------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                          UNIT:      N MM                  |
```

```
|   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000  |
|--------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                        |
|   ACTUAL MEMBER LENGTH:     7.500        LZ:    7.500    LY:    7.500    |
|   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000         |
|--------------------------------------------------------------------------|
|  SECTION CLASS                                                            |
|   COMPRESSION    :            Class 4                                     |
|   FLEXURE           MAJOR               MINOR                             |
|   SECTION        : Class 1            Class 1                             |
|   FLANGE         : Class 1            Class 1                             |
|   WEB (CRITICAL) :            Class 1                                     |
|==========================================================================|
|  SLENDERNESS                                                              |
|   ACTUAL SLENDERNESS RATIO   :     190.856          LC   :        1       |
|   ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00      |
|--------------------------------------------------------------------------|
      STAAD SPACE                                               -- PAGE NO.
4
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   **********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                           CHECKS FOR AXIAL TENSION                        |
|--------------------------------------------------------------------------|
|  YIELDING                                                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
|    0.00E+00   2.10E+03       0.000      Cl. 13.2          1     0.000     |
|  RUPTURE                                                                  |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
|    0.00E+00   2.25E+03       0.000      Cl. 13.2          1     0.000     |
|--------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                      |
|--------------------------------------------------------------------------|
|  FLEX BUCKLING                                                            |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:       LC:    LOC:    |
|   MAJOR:   0.00E+00   1.56E+03     0.000  Cl. 13.3.4(a)       1    0.000  |
|   MINOR:   0.00E+00   3.07E+02     0.000  Cl. 13.3.4(a)       1    0.000  |
|  INTERMEDIATE RESULTS:                                                    |
|               Ag:(CM)     Fe:(N MM)       λ:         n:      KL/r:       |
|   MAJOR:      65.224         641.730       0.739    1.34      56.150      |
|   MINOR:      65.224          55.545       2.510    1.34     190.856      |
|--------------------------------------------------------------------------|
|  FLEXURAL TORSIONAL BUCKLING                                              |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
|    0.00E+00   3.07E+02       0.000      Cl. 13.3.4        1     0.000     |
|  INTERMEDIATE RESULTS:                                                    |
|   Ag:(CM)    65.224   Fe:(N MM)       55.545  λ:     2.510  n: 1.34      |
|--------------------------------------------------------------------------|
|                               CHECKS FOR SHEAR                            |
|--------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                       |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
|    5.00E+00   5.02E+02       0.010    Cl. 13.4.1.1       1     0.000     |
|  INTERMEDIATE RESULTS:                                                    |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)      |
|     24.168    5.347     0.042       326.980       653.808    231.000     |
```

```
||--------------------------------------------------------------------------||
||  SHEAR ALONG Z AXIS                                                       |
||   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:        |
||    0.00E+00   8.33E+02       0.000    AISC G2-1           1     0.000     |
||  INTERMEDIATE RESULTS:                                                    |
||   Aw:(CM)                                                                 |
||     44.088                                                                |
||--------------------------------------------------------------------------||
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                              CHECKS FOR BENDING                          |
|--------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    -3.75E+01  2.649E+02      0.142    Cl. 13.5(a)         1     0.000    |
|  INTERMEDIATE RESULTS: Mp:  2.94E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00  5.954E+01      0.000    Cl. 13.5(a)         1     0.000     |
|  INTERMEDIATE RESULTS: Mp:  6.62E+01   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|   -3.75E+01  8.147E+01      0.460   Cl. 13.6(a)(ii)      1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:
Lyr:  |
|  0.00E+00   6.07  0.00     0.00  9.052E+01  0.00  0.00E+00  0.00   0.00  |
|--------------------------------------------------------------------------|
|                            CHECKS FOR INTERACTION                        |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                               |
|                      RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:   |
|   C/S STRENGTH   : 0.120  Cl. 13.9.2      1   0.00   0.00E+00  2.10E+03  |
|   MEMBER STRENGTH: 0.460  Cl. 13.9.3      1   0.00                        |
|                          Mfz:         Mfy:          Mrz:         Mry:     |
|   C/S STRENGTH   :   -3.75E+01      0.00E+00     2.65E+02      5.95E+01   |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                           |
|                      RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:   |
|   C/S STRENGTH   : 0.120  Cl. 13.8.2      1   0.000 0.00E+00  2.05E+03   |
|   MEMBER STRENGTH: 0.120  Cl. 13.8.2      1   0.000 0.00E+00  3.07E+02   |
|   LTB STRENGTH   : 0.391  Cl. 13.8.2      1   0.000 0.00E+00  3.07E+02   |
|                          Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|   C/S STRENGTH   : -3.75E+01  0.00E+00  2.65E+02  5.95E+01  0.60   0.60  |
|   MEMBER STRENGTH: -3.75E+01  0.00E+00  2.65E+02  5.95E+01  0.60   0.60  |
|   LTB STRENGTH   : -3.75E+01  0.00E+00  8.15E+01  5.95E+01  0.60   0.60  |
|                          b :         Cez:      Cey:        U1z:     U1y:     |
|   C/S STRENGTH   :    0.85     4.28E+03  3.70E+02     1.00     1.00      |
|   MEMBER STRENGTH:    0.85     4.28E+03  3.70E+02     1.00     1.00      |
|   LTB STRENGTH   :    0.85     4.28E+03  3.70E+02     1.00     1.00      |
```

```
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     5.00E+00  5.025E+02      0.010    Cl. 14.6            1     0.000    |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:  |
|   0.460  Cl. 13.8     1    0.00  -3.75E+01  8.15E+01  0.00E+00  5.95E+01 |
|==========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V. CSA S16-19 - Check for Bending and LTB

Determine the capacity of a laterally unbraced beam subject to bending moment per the CSA S16-19 code.

Problem

A 7.5 m long, simply supported beam is subjected to a distributed load of 12 kN/m. The section is a Canadian W310x52. The steel is grade 350W.

Section Properties

$D = 318$ mm
$B = 167$ mm
$t_w = 7.6$ mm
$t_f = 13.2$ mm
$Z_x = 841{,}000\ mm^3$
$Z_y = 189{,}000\ mm^3$
$I_y = 10.3\times10^6\ mm^4$

$J = 308{,}000\ mm^4$
$C_w = 238\times10^9\ mm^6$

Material Properties

$F_y = 350$ MPa
$F_u = 450$ MPa
$E = 205{,}000$ MPa
$G = 76{,}920$ MPa

Calculations

Section Classification

$$b_{el} = B / 2 = 83.5 \text{ mm}$$

$$h = D - 2\times t_f = 291.6 \text{ mm}$$

Flexure section classification per Table 2:

- Flange classification about major axis:

$$\frac{b_{\text{el}}}{t} = \frac{83.5}{13.2} = 6.33 < \frac{145}{\sqrt{F_y}} = 7.75$$

Class 1

- Flange classification about minor axis:

$$\frac{b_{\text{el}}}{t} = \frac{83.5}{13.2} = 6.33 < \frac{145}{\sqrt{F_y}} = 7.75$$

Class 1

- Web classification about major axis:

$$\frac{h}{w} = \frac{291.6}{7.6} = 38.37 < \frac{1,100}{\sqrt{F_y}} = 58.8$$

Class 1

- Web classification about minor axis:

$$\frac{h}{w} = \frac{291.6}{7.6} = 38.37 < \frac{1,100}{\sqrt{F_y}} = 58.8$$

Class 1

Therefore, the section is Class 1 for flexure.

Bending Capacity

$$M_{\text{fx}} = \frac{12(7.5)^2}{8} = 84.4 \text{ kN·m}$$

For a Class 2, doubly symmetric section, the capacity is determined by Cl. 13.5a:

Major axis:

$$M_{px} = Z_x \times F_y = 841{,}000 \times 350\ (10)^{-6} = 294.4 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 294.4 = 264.9 \text{ kN·m}$$

The ratio = *84.4 / 264.9 = 0.318*

Minor axis:

$$M_{py} = Z_y \times F_y = 189{,}000 \times 350\ (10)^{-6} = 66.2 \text{ kN·m}$$

$$M_{rx} = \phi M_{px} = 0.9 \times 66.2 = 59.5 \text{ kN·m}$$

Lateral Torsional Buckling

Check for lateral torsional buckling per Cl. 13.6:

$$M_u = \frac{\omega_2 \pi}{L}\sqrt{EI_yGJ + \left(\frac{\pi E}{L}\right)^2 I_y C_w} < 0.67 \times M_p$$

where

ω_2	=	1.0
EI_yGJ	=	$205{,}000 \times 10.3(10)^6 \times 76{,}920 \times 308{,}000 = 50.02\ (10)^{21}$
$\left(\frac{\pi E}{L}\right)^2 I_y C_w$	=	$\left(\frac{\pi \times 205{,}000}{7{,}500}\right)^2 10.3(10)^6 238(10)^9 = 18.08(10)^{21}$

$$\frac{(1.0)\pi}{7{,}500}\sqrt{50.02(10)^{21} + 18.08(10)^{21}} = 109.3(10)^6 \text{ N·mm} = 109.3 \text{ kN·m}$$

$$109.3 < 0.67\ (294.4) = 197.2$$

$$M_r = \phi M_u = 0.9 \times 109.3 = 98.38 \text{ kN·m}$$

Ratio = *84.4 / 98.38 = 0.858*.

Comparison

Table 625: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
Nominal flexure capacity about major axis, M_{px} (kN·m)	294.4	294	negligible	
Ultimate flexure capacity about major axis, M_{rx} (kN·m)	264.9	264.9	none	
Nominal flexure capacity about minor axis, M_{py} (kN·m)	66.2	66.2	none	
Ultimate flexure capacity about minor axis, M_{rx} (kN·m)	59.5	59.54	negligible	
Ratio	0.318	0.318	none	
Nominal lateral torsional bending capacity, M_u (kN)	109.3	109.3	none	
Ultimate lateral torsional bending capacity, M_r (kN)	98.38	98.38	none	
Critical Ratio	0.858	0.858	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Check for Bending and LTB.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- The laterally unsupported is designated by the parameter `LAT 0`.

- The value of ω_2 is specified as 1 using the parameter `CB 1`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Oct-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W310X52
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -12
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN
LAT 0 ALL
CB 1 ALL
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|-----------------------------------------------------------------------------|
|    MEMBER NO.:      1      SECTION: ST  W310X52          (CANADIAN SECTIONS)|
|   STATUS:        PASS    CRITICAL RATIO:  0.858         LOADCASE    1       |
|   LOCATION:      3.75    CRITICAL CONDITION:            Cl. 13.8            |
|-----------------------------------------------------------------------------|
|   STRENGTH CHECK SUMMARY                                                    |
```

```
|   CRITICAL RATIO:  0.858(PASS)            LOAD CASE:     1               |
|   LOCATION      :   3.75                  CONDITION: Cl. 13.8            |
|--------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      3.75              |
|         FX:      0.00(T)    FY:     0.00      FZ:      0.00               |
|         MX:  0.000E+00      MY: 0.000E+00     MZ: -8.438E+01              |
|--------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   3.75)             UNIT:      CM              |
|   AXX:      66.700     AZZ:      44.088      AYY:         24.168         |
|   IXX:      30.800     IZZ:   11900.002      IYY:       1030.000         |
|   PZZ:     841.000     PYY:     189.000                                  |
|   SZZ:     748.428     SYY:     123.353      CW :     237980.875         |
|--------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                          UNIT:      N MM            |
|   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000  |
|--------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                       |
|   ACTUAL MEMBER LENGTH:     7.500        LZ:    7.500    LY:    7.500    |
|   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000         |
|--------------------------------------------------------------------------|
|  SECTION CLASS                                                           |
|   COMPRESSION    :            Class 4                                    |
|   FLEXURE           MAJOR              MINOR                             |
|   SECTION        : Class 1            Class 1                            |
|   FLANGE         : Class 1            Class 1                            |
|   WEB (CRITICAL) :            Class 1                                    |
|==========================================================================|
|  SLENDERNESS                                                             |
|   ACTUAL SLENDERNESS RATIO   :     190.856          LC   :        1      |
|   ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00     |
|--------------------------------------------------------------------------|
     STAAD SPACE                                                -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                    ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL TENSION                        |
|--------------------------------------------------------------------------|
|  YIELDING                                                                |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   2.10E+03       0.000      Cl. 13.2          1     0.000    |
|  RUPTURE                                                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   2.25E+03       0.000      Cl. 13.2          1     0.000    |
|--------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                      |
|--------------------------------------------------------------------------|
|  FLEX BUCKLING                                                           |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:    |
|   MAJOR:   0.00E+00   1.56E+03     0.000  Cl. 13.3.4(a)       1    0.000 |
|   MINOR:   0.00E+00   3.07E+02     0.000  Cl. 13.3.4(a)       1    0.000 |
|  INTERMEDIATE RESULTS:                                                   |
|               Ag:(CM)     Fe:(N MM)       λ:        n:      KL/r:       |
|   MAJOR:      65.224        641.730       0.739    1.34      56.150      |
|   MINOR:      65.224         55.545       2.510    1.34     190.856      |
|--------------------------------------------------------------------------|
```

```
 |  FLEXURAL TORSIONAL BUCKLING                                          |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |    0.00E+00   3.07E+02       0.000      Cl. 13.3.4        1     0.000  |
 |  INTERMEDIATE RESULTS:                                                |
 |   Ag:(CM)    65.224   Fe:(N MM)       55.545  λ:     2.510  n: 1.34   |
 |-----------------------------------------------------------------------|
 |                           CHECKS FOR SHEAR                            |
 |-----------------------------------------------------------------------|
 |  SHEAR ALONG Y AXIS                                                   |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |    4.50E+01   5.02E+02       0.090    Cl. 13.4.1.1       1     0.000  |
 |  INTERMEDIATE RESULTS:                                                |
 |   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)  |
 |     24.168    5.347     0.042       326.980       653.808    231.000  |
 |-----------------------------------------------------------------------|
 |  SHEAR ALONG Z AXIS                                                   |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |    0.00E+00   8.33E+02       0.000    AISC G2-1           1     0.000  |
 |  INTERMEDIATE RESULTS:                                                |
 |   Aw:(CM)                                                             |
 |     44.088                                                            |
 |-----------------------------------------------------------------------|
      STAAD SPACE                                               -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
 |-----------------------------------------------------------------------|
 |                          CHECKS FOR BENDING                           |
 |-----------------------------------------------------------------------|
 |  YIELDING ALONG MAJOR AXIS                                            |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |     8.44E+01  2.649E+02      0.318    Cl. 13.5(a)        1     3.750  |
 |  INTERMEDIATE RESULTS: Mp:  2.94E+02   Se(CM):  0.00E+00    My:  0.00E+00|
 |-----------------------------------------------------------------------|
 |  YIELDING ALONG MINOR AXIS                                            |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |    0.00E+00  5.954E+01      0.000    Cl. 13.5(a)        1     0.000   |
 |  INTERMEDIATE RESULTS: Mp:  6.62E+01   Se(CM):  0.00E+00    My:  0.00E+00|
 |-----------------------------------------------------------------------|
 |  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                              |
 |   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:    |
 |    8.44E+01  9.838E+01      0.858   Cl. 13.6(a)(ii)      1     3.750   |
 |  INTERMEDIATE RESULTS:                                                |
 |   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:
Lyr:  |
 |  0.00E+00   1.00  0.00     0.00  1.093E+02  0.00  0.00E+00  0.00   0.00 |
 |-----------------------------------------------------------------------|
 |                        CHECKS FOR INTERACTION                         |
 |-----------------------------------------------------------------------|
 |  FLEXURE AND AXIAL TENSION                                            |
 |                    RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:  |
 |   C/S STRENGTH   : 0.271  Cl. 13.9.2      1   3.75   0.00E+00  2.10E+03 |
 |   MEMBER STRENGTH: 0.858  Cl. 13.9.3      1   0.00                     |
 |                        Mfz:         Mfy:          Mrz:         Mry:    |
 |   C/S STRENGTH   :    8.44E+01      0.00E+00     2.65E+02      5.95E+01 |
 |-----------------------------------------------------------------------|
```

```
|  FLEXURE AND AXIAL COMPRESSION                                               |
|                    RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:       |
|   C/S STRENGTH   : 0.271  Cl. 13.8.2      1   3.750 0.00E+00  2.05E+03     |
|   MEMBER STRENGTH: 0.271  Cl. 13.8.2      1   3.750 0.00E+00  3.07E+02     |
|   LTB STRENGTH   : 0.729  Cl. 13.8.2      1   3.750 0.00E+00  3.07E+02     |
|                        Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|   C/S STRENGTH   :  8.44E+01  0.00E+00  2.65E+02  5.95E+01  0.60   0.60   |
|   MEMBER STRENGTH:  8.44E+01  0.00E+00  2.65E+02  5.95E+01  0.60   0.60   |
|   LTB STRENGTH   :  8.44E+01  0.00E+00  9.84E+01  5.95E+01  0.60   0.60   |
|                        b :         Cez:     Cey:         U1z:     U1y:       |
|   C/S STRENGTH   :    0.85     4.28E+03  3.70E+02     1.00     1.00        |
|   MEMBER STRENGTH:    0.85     4.28E+03  3.70E+02     1.00     1.00        |
|   LTB STRENGTH   :    0.85     4.28E+03  3.70E+02     1.00     1.00        |
|----------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                           |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:         |
|     4.50E+01  5.025E+02      0.090    Cl. 14.6           1     0.000       |
|----------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                           |
|   RATIO:  CRITERIA:  LC:   LOC:     Mfz:      Mrz:       Mfy:      Mry:    |
|   0.858  Cl. 13.8     1    3.75   8.44E+01  9.84E+01  0.00E+00  5.95E+01  |
```

V. CSA S16-19 - CLASS4 UPT Cantilever Beam with UDL

Determine the capacity of a cantilever beam using a user-provided section subject to a distributed load per the CSA S16-19 code.

Details

A 7.5 m long cantilever member is subjected to uniformly distributed load of 1 kN. The section is an I shape with 300 mm × 15 mm flanges and a 420 mm × 15 mm web. The compression flange is laterally unbraced along the length.

Material Properties

$F_y = 650$ MPa

Validation

Section Properties

$D = 420 + 15 + 15 = 450$ mm

$A_g = 2×(300 × 15) + 420 × 15 = 15{,}300 \text{ mm}^2$

$$I_x = 2\left[\frac{300(15)^3}{12} + 300 \times 15\left(\frac{420 + 15}{2}\right)^2\right] + \frac{15(420)^3}{12} = 518.5(10)^6\ mm^4$$

$$I_y = 2\left[\frac{15(300)^3}{12}\right] + \frac{420(15)^3}{12} = 67{,}620{,}000\ mm^4$$

$$S_x = \frac{518.5(10)^6 2}{450} = 2{,}305{,}000\ mm^3$$

$$S_y = \frac{67{,}620{,}000(2)}{300} = 450{,}800\ mm^3$$

$$J = \frac{1}{3}\left[2 \times 300(15)^3 + 420(15)^3\right] = 1{,}148{,}000\ mm^4$$

$$C_w = \frac{(420 + 15)^2 (300)^3 15}{24} = 3{,}193(10)^9\ mm^6$$

<u>Section Classification</u>

$b_{el} = B / 2 = 150$ mm

$C_f = 0$ (no compression force)

Flexure section classification per Table 2 of CSA S16-19:

- Flange classification about major axis:

  $$\frac{b_{el}}{t_f} = \frac{150}{15} = 10 > \frac{200}{\sqrt{F_y}} = \frac{200}{\sqrt{650}} = 7.84$$

  Class 4
- Flange classification about minor axis:

  $$\frac{b_{el}}{t_f} = \frac{150}{15} = 10 < \frac{340}{\sqrt{F_y}} = \frac{340}{\sqrt{650}} = 13.36$$

  Class 3
- Web classification about major axis:

  $$\frac{h}{t_w} = \frac{420}{15} = 28 < \frac{1{,}100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{\phi C_y}\right) = 58.8$$

  Class 1
- Web classification about minor axis:

  $$\frac{h}{t_w} = \frac{420}{15} = 28 < \frac{1{,}100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{\phi C_y}\right) = 58.8$$

  Class 1

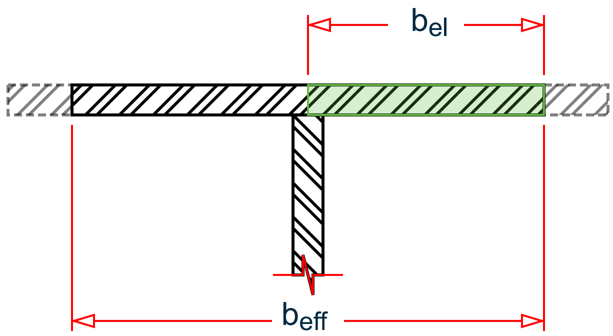
Therefore, the section is Class 4 for flexure.

<u>Bending Capacity</u>

$$M_{fx} = \frac{1.0(7.5)^2}{2} = 28.1\ \text{kN}\cdot\text{m}$$

For a Class 4, doubly symmetric section, the capacity is determined by Cl. 13.5.c. In this case, the compression flange is Class 4 but the web is Class 3, so Cl. 13.5.c.iii applies. Therefore, the effective section modulus, S_e, is used. The outstanding element length is limited to:

$$b_{el} = 200t / \sqrt{F_y} = 200 \times 15 / \sqrt{650} = 118\ \text{mm}$$



Therefore, $b_{eff} = 2\times b_{el} = 236\ mm$

$$I_{\mathrm{eff,x}} = 2\left[\frac{268(15)^3}{12} + 268\times 15\left(\frac{420+15}{2}\right)^2\right] + \frac{15(420)^3}{12} = 427.7(10)^6\ \mathrm{mm}^4$$

$$S_{\mathrm{eff,x}} = \frac{427.7(10)^6 2}{450} = 1{,}901{,}000\ \mathrm{mm}^3$$

Major axis:

$$M_{yx} = S_{eff,x}\times F_y = 1{,}901{,}000 \times 650\ (10)^{-6} = 1{,}236\ kN\cdot m$$

$$M_{rx} = \phi M_{yx} = 0.9 \times 1{,}236 = 1{,}112\ kN\cdot m$$

The ratio = *28.1 / 1,112 = 0.025*

Minor axis:

$$M_{yy} = S_y\times F_y = 450{,}800 \times 650\ (10)^{-6} = 293.0\ kN\cdot m$$

$$M_{ry} = \phi M_{yy} = 0.9 \times 293.0 = 263.7\ kN\cdot m$$

Check for lateral-torsional buckling per Cl. 13.6.h i).

$$M_u = \omega_2 C_H C_B \frac{\sqrt{EI_y GJ}}{L_c}$$

where

$$X = \frac{\pi}{L_c}\sqrt{\frac{EC_w}{GJ}} = \frac{\pi}{7{,}500}\sqrt{\frac{205{,}000\times 3{,}193(10)^9}{76{,}920\times 1{,}148{,}000}} = 1.142$$

$0.5 \le X \le 2.5$ for a distributed load

ω_2 = 5.38 + 8.71 X = 5.38 + 8.71 (1.142) = 15.77

C_H = $0.49 - 0.27\,X + 0.06\,X^2 = 0.49 - 0.27\,(1.142) + 0.06\,(1.142)^2 = 0.26$

C_B = $1.48 + 0.16\,X = .48 + 0.16\,(1.142) = 1.66$

$$M_u = 15.77(0.26)(1.66)\frac{\sqrt{205{,}000\times 67{,}620{,}000\times 76{,}920\times 1{,}148{,}000}}{7{,}500\times(10)^6} = 1{,}004\ \ kN\cdot m$$

$$0.67\times M_y = 0.67\ (1{,}236) = 827.8\ kN\cdot m < M_u$$

$M_r = 1.15\phi M_y[1 - 0.28\ M_y / M_u] = 1.15 \times 0.9 \times 1{,}236\ [\ 1 - 0.28\times(1{,}236 / 1{,}004)\ ] = 838.3$ kN·m

The ratio = *28.1 / 838.3 = 0.034*

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
M_{yx} (kN·m)	1,236	1,230	negligible	
M_{rx} (kN·m)	1,112	1,110	negligible	
Ratio of M_{fx}/M_{yx}	0.025	0.025	none	
M_{yy} (kN·m)	293.0	290.0	negligible	
M_{ry} (kN·m)	263.7	261.1	negligible	
M_u (kN·m)	1,004	1,006	negligible	
M_r (kN·m)	838.3	837.9	negligible	
ω_2	15.77	15.77	none	
Critical ratio	0.034	0.034	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - CLASS4 UPT Cantilever Beam with UDL.STD` is typically installed with the program.

The following D4.E.7 Design Parameters (on page 1458) are used:

- The cantilever is braced along the tension flange only specified using `CAN 2`.
- The load acts on the tension side of the bending member, thus `LHT 1`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 3-Dec-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER KN
WIDE FLANGE
UPTI
0.0153 0.45 0.015 0.3 0.015 0.000518535 6.76181e-05 1.1475e-06 0.00675 -
0.006 0.3 0.015
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
```

```
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 UPTABLE 1 UPTI
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -1
PERFORM ANALYSIS
PARAMETER 1
CODE CANADIAN
CAN 2 ALL
LHT 1 ALL
FYLD 650000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  UPTI             (UPT)               |
|  STATUS:        PASS    CRITICAL RATIO:  0.034         LOADCASE    1       |
|  LOCATION:      0.00    CRITICAL CONDITION:           Cl. 13.8.4          |
|---------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                   |
|   CRITICAL RATIO:  0.034(PASS)             LOAD CASE:     1               |
|   LOCATION      :   0.00                   CONDITION: Cl. 13.8.4          |
|---------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      0.00               |
|         FX:      0.00(T)    FY:     7.50      FZ:      0.00               |
|         MX:  0.000E+00      MY: 0.000E+00     MZ:  2.813E+01              |
|---------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   0.00)              UNIT:      CM               |
|   AXX:     153.000     AZZ:      90.000      AYY:         63.000          |
|   IXX:     114.750     IZZ:   51853.500      IYY:       6761.811          |
|   PZZ:    2619.000     PYY:     698.625                                   |
|   SZZ:    2304.600     SYY:     450.787      CW :    3198759.000          |
|---------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                           UNIT:      N MM             |
|   FYLD:     650.000     FU:   450.000      E: 204999.984  G:  76920.000   |
|---------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                        |
|   ACTUAL MEMBER LENGTH:     7.500        LZ:    7.500    LY:    7.500     |
|   KZ:  1.000         KY:  1.000      NSF:  1.000      SLF:  1.000         |
|---------------------------------------------------------------------------|
```

```
||  SECTION CLASS                                                          |
|    COMPRESSION    :            Class 4                                   |
|    FLEXURE          MAJOR               MINOR                            |
|    SECTION        : Class 4           Class 3                            |
|    FLANGE         : Class 4           Class 3                            |
|    WEB (CRITICAL) :            Class 1                                   |
|=========================================================================|
|   SLENDERNESS                                                            |
|    ACTUAL SLENDERNESS RATIO   :     112.817          LC   :        1      |
|    ALLOWABLE SLENDERNESS RATIO:     300.000          LOC  :      0.00     |
|-------------------------------------------------------------------------|
      STAAD SPACE                                            -- PAGE NO.
4
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                         |
|-------------------------------------------------------------------------|
|  YIELDING                                                                |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   8.95E+03       0.000      Cl. 13.2          1     0.000    |
|  RUPTURE                                                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   5.16E+03       0.000      Cl. 13.2          1     0.000    |
|-------------------------------------------------------------------------|
|                      CHECKS FOR AXIAL COMPRESSION                       |
|-------------------------------------------------------------------------|
|  FLEX BUCKLING                                                           |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:    |
|   MAJOR:   0.00E+00   5.81E+03     0.000  Cl. 13.3.4(a)       1    0.000 |
|   MINOR:   0.00E+00   1.67E+03     0.000  Cl. 13.3.4(a)       1    0.000 |
|  INTERMEDIATE RESULTS:                                                   |
|               Ag:(CM)     Fe:(N MM)       λ:        n:      KL/r:        |
|   MAJOR:     129.731       1219.039       0.730    1.34      40.740     |
|   MINOR:     129.731        158.965       2.022    1.34     112.817     |
|-------------------------------------------------------------------------|
|  FLEXURAL TORSIONAL BUCKLING                                             |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   1.67E+03       0.000      Cl. 13.3.4        1     0.000    |
|  INTERMEDIATE RESULTS:                                                   |
|   Ag:(CM)   129.731   Fe:(N MM)      158.965  λ:     2.022  n: 1.34     |
|-------------------------------------------------------------------------|
|                            CHECKS FOR SHEAR                             |
|-------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                      |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    7.50E+00   2.43E+03       0.003    Cl. 13.4.1.1       1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)     |
|     63.000    5.353     0.056       610.909      1228.900    429.000     |
|-------------------------------------------------------------------------|
|  SHEAR ALONG Z AXIS                                                      |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00   3.16E+03       0.000    AISC G2-1          1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
||   Aw:(CM)                                                               |
```

```
|      90.000                                                                 |
|-----------------------------------------------------------------------------|
      STAAD SPACE                                                 -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|-----------------------------------------------------------------------------|
|                             CHECKS FOR BENDING                              |
|-----------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                                  |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    -2.81E+01  1.110E+03      0.025    Cl. 13.5(c)         1     0.000       |
|  INTERMEDIATE RESULTS: Mp:  0.00E+00   Se(CM):  1.90E+03    My:  1.23E+03|
|-----------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                                  |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|    0.00E+00  2.611E+02      0.000    Cl. 13.5(c)         1     0.000        |
|  INTERMEDIATE RESULTS: Mp:  0.00E+00   Se(CM):  4.46E+02    My:  2.90E+02|
|-----------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                    |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|   -2.81E+01  8.379E+02      0.034   Cl. 13.6(b)(ii)      1     0.000        |
|  INTERMEDIATE RESULTS:                                                      |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:
Lyr:  |
|  0.00E+00  15.77  0.00     0.00  1.006E+03  0.00  0.00E+00  0.00   0.00  |
|-----------------------------------------------------------------------------|
|                           CHECKS FOR INTERACTION                            |
|-----------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                                  |
|                     RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:     |
|   C/S STRENGTH   : 0.025  Cl. 13.9.1      1   0.00   0.00E+00  5.16E+03   |
|   MEMBER STRENGTH: 0.034  Cl. 13.9.3      1   0.00                         |
|                         Mfz:         Mfy:          Mrz:         Mry:       |
|   C/S STRENGTH   :   -2.81E+01      0.00E+00     1.11E+03      2.61E+02   |
|-----------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                              |
|                     RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:     |
|   C/S STRENGTH   : 0.025  Cl. 13.8.4      1   0.000 0.00E+00  7.59E+03    |
|   MEMBER STRENGTH: 0.025  Cl. 13.8.4      1   0.000 0.00E+00  1.67E+03    |
|   LTB STRENGTH   : 0.034  Cl. 13.8.4      1   0.000 0.00E+00  1.67E+03    |
|                         Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|   C/S STRENGTH   : -2.81E+01  0.00E+00  1.11E+03  2.61E+02  0.60   0.60  |
|   MEMBER STRENGTH: -2.81E+01  0.00E+00  1.11E+03  2.61E+02  0.60   0.60  |
|   LTB STRENGTH   : -2.81E+01  0.00E+00  8.38E+02  2.61E+02  0.60   0.60  |
|                         b :          Cez:     Cey:          U1z:     U1y:      |
|   C/S STRENGTH   :   N/A     1.87E+04  2.43E+03     1.00     1.00          |
|   MEMBER STRENGTH:   N/A     1.87E+04  2.43E+03     1.00     1.00          |
|   LTB STRENGTH   :   N/A     1.87E+04  2.43E+03     1.00     1.00          |
|-----------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                            |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:          |
|     7.50E+00  2.432E+03      0.003    Cl. 14.6           1     0.000       |
|-----------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                            |
```

```
|   RATIO:  CRITERIA:  LC:    LOC:      Mfz:       Mrz:        Mfy:       Mry:  |
|   0.034  Cl. 13.8     1    0.00  -2.81E+01  8.38E+02  0.00E+00  2.61E+02 |
|===========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V. CSA S16-19 - Short Column Compression

Determine the capacity of a short column per the CSA S16-19 code.

Problem

A Canadian W250x73 section is used for a pedestal with a height of 1.1 m. The axial load is 1,000 kN in compression. The steel is grade 350W.

<u>Section Properties</u>

$D = 253$ mm
$B = 254$ mm
$t_w = 8.6$ mm
$t_f = 14.2$ mm
$A_g = 9{,}280$ mm^2

$I_x = 113\times10^6$ mm^4

$I_y = 38.8\times10^6$ mm^4

$J = 575{,}000$ mm^4
$C_w = 553\times10^9$ mm^6

<u>Material Properties</u>

$F_y = 350$ MPa
$F_u = 450$ MPa
$E = 205{,}000$ MPa
$G = 76{,}920$ MPa

Calculations

<u>Section Classification</u>

$$b_{el} = B / 2 = 83.5 \text{ mm}$$

$$h = D - 2\times t_f = 291.6 \text{ mm}$$

Compression section classification per Table 1

- Flange classification:

$$\frac{b_{el}}{t} = \frac{127}{14.2} = 8.94 < \frac{200}{\sqrt{F_y}} = 10.69$$

- Web classification about major axis:

$$\frac{h}{w} = \frac{235.8}{8.6} = 27.4 < \frac{670}{\sqrt{F_y}} = 35.8$$

Therefore, the section is Class 1 for compression.

Column Capacity

$$r_x = \sqrt{I_x / A} = \sqrt{\frac{113 \times (10)^6}{9{,}280}} = 110.3 \text{ mm}$$

$$r_y = \sqrt{I_y / A} = \sqrt{\frac{38.8 \times (10)^6}{9{,}280}} = 64.7 \text{ mm}$$

Assume K = 1.0,

$$\frac{KL}{r_x} = \frac{1.0 \times 1{,}100}{110.3} = 9.97$$

$$\frac{KL}{r_y} = \frac{1.0 \times 1{,}100}{64.7} = 17.0$$

Elastic buckling stress:

$$F_{\text{ex}} = \frac{\pi^2 E}{\left(KL_x / r_x\right)^2} = \frac{\pi^2 \times 205{,}000}{(9.97)^2} = 20{,}360 \text{ MPa}$$

$$F_{\text{ey}} = \frac{\pi^2 E}{\left(KL_y / r_y\right)^2} = \frac{\pi^2 \times 205{,}000}{(17.0)^2} = 6{,}991 \text{ MPa}$$

$$\lambda_x = \sqrt{F_y / F_{\text{ex}}} = \sqrt{\frac{350}{20{,}360}} = 0.131$$

$$\lambda_y = \sqrt{F_y / F_{\text{ey}}} = \sqrt{\frac{350}{6{,}991}} = 0.224$$

Per Cl. 13.3.1, $n = 1.34$ (Group 2 W shape):

$$C_{\text{rx}} = \phi A F_y \left(1 + \lambda_x^{2n}\right)^{-1/n} = 0.90 \times 9{,}280 \times 350\left[1 + 0.131^{(2\times1.34)}\right]^{-1/1.34}(10)^{-3} = 2{,}914 \text{ kN}$$

$$C_{\text{ry}} = \phi A F_y \left(1 + \lambda_y^{2n}\right)^{-1/n} = 0.90 \times 9{,}280 \times 350\left[1 + 0.224^{(2\times1.34)}\right]^{-1/1.34}(10)^{-3} = 2{,}884 \text{ kN}$$

Flexural Torsional Buckling

$$r_o^2 = r_x^2 + r_y^2 + X^2 + Y^2 = (110.3)^2 + (64.7)^2 + 0 + 0 = 16{,}360 \text{ mm}^2$$

$$F_{\text{ez}} = \left[\frac{\pi^2 E C_w}{(K_z L_z)^2} + GJ\right]\frac{1}{A \times r_o^2}$$

$$= \left[\frac{\pi^2 \times 205{,}000 \times 553(10)^9}{(1{,}100)^2} + 76{,}920 \times 575{,}000\right]\frac{1}{9{,}280 \times 16{,}360} = 6{,}382 \text{ MPa}$$

Take the minimum of F_{ex}, F_{ey}, and F_{ez} as F_{FTB}:

$$F_{\text{FTB}} = F_{\text{ez}} = 6{,}382 \text{ MPa}$$

$$\lambda_{\mathrm{FTB}} = \sqrt{F_y / F_e} = \sqrt{\frac{350}{6{,}382}} = 0.234$$

$$C_{r,\mathrm{FTB}} = \phi A F_y \left(1 + \lambda_z{}^{2n}\right)^{-1/n} = 0.90 \times 9{,}280 \times 350\left[1 + 0.234^{(2\times1.34)}\right]^{-1/1.34}(10)^{-3} = 2{,}879\ \mathrm{kN}$$

Ratio = *1,000 / 2,879 = 0.347*

Comparison

Table 626: Comparison of results

Criteria	Reference	STAAD.Pro	Difference	Comments
λ_x	0.131	0.131	none	
F_{ex} (MPa)	20,360	20,360.98	negligible	
C_{rx} (kN)	2,914	2,910	negligible	
λ_y	0.224	0.224	none	
F_{ey} (MPa)	6,991	6,991,204	negligible	
C_{ry} (kN)	2,884	2,880	negligible	
λ_{FTB}	0.234	0.234	none	
F_{FTB} (MPa)	6,382	6,381.727	negligible	
C_{FTB} (kN)	2,879	2,880	negligible	
Critical ratio	0.347	0.347	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Canada\CSA S16-19 - Short Column Compression.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Sep-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1.1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.92
```

```
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
*STRENGTH FY 350000 FU 450000 RY 1.5 RT 1.2
G 7.692e+07
END DEFINE MATERIAL
MEMBER PROPERTY CANADIAN
1 TABLE ST W250X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT SUPPORT REACTION ALL
PARAMETER 1
CODE CANADIAN 2019
FYLD 350000 ALL
FU 450000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W250X73          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.347         LOADCASE    1      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8.2         |
|---------------------------------------------------------------------------|
|  STRENGTH CHECK SUMMARY                                                   |
|   CRITICAL RATIO:  0.347(PASS)             LOAD CASE:     1               |
|   LOCATION      :   0.00                  CONDITION: Cl. 13.8.2          |
|---------------------------------------------------------------------------|
|  DESIGN FORCES              LC:       1      LOC:      0.00               |
|         FX:   1000.00(C)    FY:     0.00      FZ:      0.00               |
|         MX:  0.000E+00      MY: 0.000E+00     MZ:  0.000E+00              |
|---------------------------------------------------------------------------|
|  SECTION PROPERTIES (LOC:   0.00)              UNIT:      CM              |
|   AXX:      92.800     AZZ:      72.136      AYY:         21.758          |
|   IXX:      57.500     IZZ:   11300.001      IYY:       3880.000          |
|   PZZ:     985.000     PYY:     463.000                                   |
|   SZZ:     893.281     SYY:     305.512      CW :     552900.312          |
|---------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                           UNIT:      N MM            |
|   FYLD:     350.000     FU:   450.000      E: 204999.984  G:  76920.000   |
|---------------------------------------------------------------------------|
|  DESIGN PARAMETERS                                                        |
|   ACTUAL MEMBER LENGTH:     1.100        LZ:    1.100    LY:    1.100     |
|   KZ:  1.000        KY:  1.000      NSF:  1.000      SLF:  1.000          |
|---------------------------------------------------------------------------|
```

```
|  SECTION CLASS                                                         |
|   COMPRESSION    :            Class 1                                  |
|   FLEXURE           MAJOR               MINOR                          |
|   SECTION        : Class 2            Class 2                          |
|   FLANGE         : Class 2            Class 2                          |
|   WEB (CRITICAL) :            Class 1                                  |
|========================================================================|
|  SLENDERNESS                                                           |
|   ACTUAL SLENDERNESS RATIO   :      17.012          LC   :        1    |
|   ALLOWABLE SLENDERNESS RATIO:     200.000          LOC  :      0.00   |
|------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                   ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
  Member :     1       Contd.
|------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                        |
|------------------------------------------------------------------------|
|  YIELDING                                                              |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:     |
|    0.00E+00   2.92E+03       0.000      Cl. 13.2          1     0.000  |
|  RUPTURE                                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:     |
|    0.00E+00   3.13E+03       0.000      Cl. 13.2          1     0.000  |
|------------------------------------------------------------------------|
|                      CHECKS FOR AXIAL COMPRESSION                      |
|------------------------------------------------------------------------|
|  FLEX BUCKLING                                                         |
|            FORCE:    CAPACITY:   RATIO:    CRITERIA:      LC:    LOC:  |
|   MAJOR:   1.00E+03   2.91E+03     0.343  Cl. 13.3            1    0.000 |
|   MINOR:   1.00E+03   2.88E+03     0.347  Cl. 13.3            1    0.000 |
|  INTERMEDIATE RESULTS:                                                 |
|              Ag:(CM)     Fe:(N MM)       λ:         n:      KL/r:     |
|   MAJOR:      92.800      20360.980       0.131    1.34       9.968    |
|   MINOR:      92.800       6991.204       0.224    1.34      17.012    |
|------------------------------------------------------------------------|
|  FLEXURAL TORSIONAL BUCKLING                                           |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:     |
|    1.00E+03   2.88E+03       0.347      Cl. 13.3          1     0.000  |
|  INTERMEDIATE RESULTS:                                                 |
|   Ag:(CM)    92.800   Fe:(N MM)     6381.727  &#955;:     0.234  n:
1.34       |
|------------------------------------------------------------------------|
|                           CHECKS FOR SHEAR                             |
|------------------------------------------------------------------------|
|  SHEAR ALONG Y AXIS                                                    |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:     |
|    0.00E+00   4.52E+02       0.000    Cl. 13.4.1.1        1     0.000  |
|  INTERMEDIATE RESULTS:                                                 |
|   Aw:(CM)      Kv:       Ka:     Fcri:(N MM)  Fcre:(N MM)  Fs:(N MM)   |
|     21.758    5.552     0.224       489.474      1465.103    231.000   |
|------------------------------------------------------------------------|
|  SHEAR ALONG Z AXIS                                                    |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:     |
|    0.00E+00   1.36E+03       0.000    AISC G2-1           1     0.000  |
|  INTERMEDIATE RESULTS:                                                 |
```

```
|   Aw:(CM)                                                                |
|     72.136                                                               |
|--------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                    ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
   Member :     1       Contd.
|--------------------------------------------------------------------------|
|                           CHECKS FOR BENDING                             |
|--------------------------------------------------------------------------|
|  YIELDING ALONG MAJOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     0.00E+00  3.103E+02      0.000    Cl. 13.5(a)         1     0.000    |
|  INTERMEDIATE RESULTS: Mp:  3.45E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  YIELDING ALONG MINOR AXIS                                               |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00  1.458E+02      0.000    Cl. 13.5(a)         1     0.000     |
|  INTERMEDIATE RESULTS: Mp:  1.62E+02   Se(CM):  0.00E+00    My:  0.00E+00|
|--------------------------------------------------------------------------|
|  LATERAL TORSIONAL BUCKLING (MAJOR AXIS)                                 |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|    0.00E+00  3.103E+02      0.000   Cl. 13.6(a)(i)       1     0.000     |
|  INTERMEDIATE RESULTS:                                                   |
|   Iyc(CM):    w 2:    w 3:    b x(CM):  Mu:       rt:   Myr:     Lu:
Lyr:  |
|  0.00E+00   2.50  0.00     0.00  1.982E+04  0.00  0.00E+00  0.00   0.00  |
|--------------------------------------------------------------------------|
|                         CHECKS FOR INTERACTION                           |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL TENSION                                               |
|                   RATIO:   CRITERIA:   LC:   LOC:     Tf:        Tr:      |
|   C/S STRENGTH   : 0.000  Cl. 13.9.2      1   0.00   0.00E+00  2.92E+03  |
|                       Mfz:         Mfy:          Mrz:          Mry:       |
|   C/S STRENGTH   :    0.00E+00     0.00E+00     3.10E+02      1.46E+02   |
|--------------------------------------------------------------------------|
|  FLEXURE AND AXIAL COMPRESSION                                           |
|                   RATIO:   CRITERIA:   LC:   LOC:     Cf:        Cr:      |
|   C/S STRENGTH   : 0.342  Cl. 13.8.2      1   0.000 1.00E+03  2.92E+03   |
|   MEMBER STRENGTH: 0.347  Cl. 13.8.2      1   0.000 1.00E+03  2.88E+03   |
|   LTB STRENGTH   : 0.347  Cl. 13.8.2      1   0.000 1.00E+03  2.88E+03   |
|                       Mfz:      Mfy:      Mrz:      Mry:    w 1z:    w
1y:    |
|   C/S STRENGTH   :  0.00E+00  0.00E+00  3.10E+02  1.46E+02  0.60   0.60  |
|   MEMBER STRENGTH:  0.00E+00  0.00E+00  3.10E+02  1.46E+02  0.60   0.60  |
|   LTB STRENGTH   :  0.00E+00  0.00E+00  3.10E+02  1.46E+02  0.60   0.60  |
|                       b :          Cez:     Cey:         U1z:     U1y:      |
|   C/S STRENGTH   :    0.69     1.89E+05  6.49E+04     1.00     1.00       |
|   MEMBER STRENGTH:    0.69     1.89E+05  6.49E+04     1.00     1.00       |
|   LTB STRENGTH   :    0.69     1.89E+05  6.49E+04     1.00     1.00       |
|--------------------------------------------------------------------------|
|  SHEAR CAPACITY MODIFICATION  ( SHEAR AND FLEX )                         |
|   FORCE:      CAPACITY:     RATIO:     CRITERIA:      LC:     LOC:       |
|     0.00E+00  4.523E+02      0.000    Cl. 14.6            1     0.000    |
|--------------------------------------------------------------------------|
|  BIAXIAL FLEXURE                                                         |
```

```
|   RATIO:  CRITERIA:  LC:    LOC:      Mfz:       Mrz:        Mfy:       Mry:  |
|   0.000  Cl. 13.8     1    0.00   0.00E+00  3.10E+02  0.00E+00  1.46E+02 |
|==========================================================================|
   ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL
MEMBERS
```

V. China

V. GB500017-2017 Double Angle section with Axial Force

Verify the slenderness, strength, and stability of a double-angle section subject to axial force per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is a 2 x L100x100x7 SP 0 with a length of 3.005 m. The structure is a truss model. Member #32 assigned with double angle section (2 x L100x100x7 SP 0) is designed per GB 50017-2017. The governing load combination (#4) is 1.2 DL + 1.4 LL and the ultimate force on the member, N, is 416.2 kN of axial compression.

Material Properties

The material is Q235 type steel.

Design strength in tension, compression, and flexure: $f_y = 215\ MPa$
Design strength in shear: $f_v = 125\ MPa$

Section Properties

Side length, $b = h = 100\ mm$
Thickness, $t = 7\ mm$
Cross-sectional area, $A = 2{,}760\ mm^2$
Moment of inertia about x, $I_x = 2{,}631{,}000\ mm^4$
Moment of inertia about y, $I_y = 4{,}732{,}000\ mm^4$
Radius of gyration about x, $i_x = 30.87\ mm$
Radius of gyration about y, $i_y = 41.41\ mm$
Area moment about x, $S_x = 36{,}700\ mm^3$
Area moment about y, $S_x = 37{,}280\ mm^3$

Calculations

Slenderness Ratio

According to Table 7.4.1-1 of GB50017-2017, effective length of double angle is:

$$l_{ox} = l_x = 3{,}005 \text{ mm}$$

$$l_{oy} = l_y = 3{,}005 \text{ mm}$$

According to formula 7.2.1-1 and 2 of steel structures code, slenderness ratio of double angle with X axis and Y axis is:

$$\lambda_x = \frac{l_{0x}}{i_x} = \frac{3,005}{30.87} = 97.33$$

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{3,005}{41.41} = 72.57$$

According to formula 7.2.2-7 of steel structures code, torsional buckling equivalent slenderness ratio of double angle is:

$$\lambda_z = 3.9 \times \frac{b}{t} = 3.9 \times \frac{100}{7} = 55.71$$

Calculate the equivalent slenderness ratio of symmetry axis. Since $\lambda_y > \lambda_z$, the formula 7.2.2-5 should be used to get the λ_{yz}:

$$\lambda_{yz} = \lambda_y\left[1 + 0.16\left(\frac{\lambda_z}{\lambda_y}\right)^2\right] = 72.57 \times \left[1 + 0.16 \times \left(\frac{55.71}{72.57}\right)^2\right] = 79.42$$

The maximum slenderness ratio of the double-angle section, $\lambda_{max} = 97.33$

The limit of slenderness ratio of compression member is 150 according to table 7.4.6 of steel structures code.

$$\lambda_{\mathrm{max}} / \lambda_{\mathrm{limit}} = \frac{97.33}{150} = 0.65 < 1.0, \text{ Passed}$$

The limit of slenderness ratio of tension member is 300 according to table 7.4.7 of steel structures code.

$$\lambda_{\mathrm{max}} / \lambda_{\mathrm{limit}} = \frac{97.33}{300} = 0.32 < 1.0, \text{ Passed}$$

Tension or Compression Strength

According to formula 7.1.1-1 and 7.1.1-2 of steel structures code, the stress of double angle with gross cross-section area is:

$$\sigma = \frac{N}{A} = \frac{416.2\mathrm{kN}}{2,760\mathrm{mm}^2} = 150.8 \ \mathrm{kN} / \mathrm{mm}^2 < f = 215$$

And the ratio is:

$$\frac{\sigma}{f} = \frac{150.8}{215} = 0.70$$

The stress of double angle with net cross section area is:

$$\sigma = \frac{N}{A} = \frac{416.2\mathrm{kN}}{2,760\mathrm{mm}^2} = 150.8 \ \mathrm{kN} / \mathrm{mm}^2 < 0.7\mathrm{f}_u = 0.7 \times 370 = 259$$

And the ratio is:

$$\frac{\sigma}{0.7\mathrm{f}_u} = \frac{150.8}{259} = 0.58$$

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Compressive Strength

According to table 7.2.1-1 of steel structures code, the double angle section class is b.

The slenderness ratio of double angle on X axis, $\lambda_x = 97.33$. So,

$$\frac{\lambda}{\varepsilon_k} = \lambda_x = 97.33$$

$$\lambda_n = \frac{\lambda_x}{\pi}\sqrt{\frac{f_y}{E}} = \frac{97.33}{\pi}\sqrt{\frac{235}{206{,}000}} = 1.046$$

Class b, so, the coefficients are $a1 = 0.650$, $a2 = 0.965$, and $a3 = 0.300$.

Since $\lambda_n = 1.046 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(a_2 + a_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(a_2 + a_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 1.046^2}\left[\left(0.965 + 0.300 \times 1.046 + 1.046^2\right) - \sqrt{\left(0.965 + 0.300 \times 1.046 + 1.046^2\right)^2 - 4 \times 1.046^2}\right] = 0.572$$

The slenderness ratio of double angle on symmetry axis is $\lambda_{yz} = 79.42$. So,

$$\frac{\lambda}{\varepsilon_k} = \lambda_x = 79.42$$

$$\lambda_n = \frac{\lambda_{xy}}{\pi}\sqrt{\frac{f_y}{E}} = \frac{79.42}{\pi}\sqrt{\frac{235}{206{,}000}} = 0.854$$

Class b, so, the coefficients are $a1 = 0.650$, $a2 = 0.965$, and $a3 = 0.300$.

Since $\lambda_n = 1.046 > 0.215$, then the stability factor in xy:

$$\phi_{yz} = \frac{1}{2\lambda_n^2}\left[\left(a_2 + a_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(a_2 + a_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 0.854^2}\left[\left(0.965 + 0.300 \times 0.854 + 0.854^2\right) - \sqrt{\left(0.965 + 0.300 \times 0.854 + 0.854^2\right)^2 - 4 \times 0.854^2}\right] = 0.692$$

The minimum stability factor is 0.572.

From GB50017-2017 Clause 7.3.1-5, when $\lambda > 80\varepsilon_k$, the formula 7.3.1-7 should be used to get the value of width thickness ratio of compression member. Here, $97.33 > 80 \times 1$.

Limit value of width to thickness ratio = $5\varepsilon_k + 0.125\lambda = 5 + 0.125 \times 97.33 = 17.17$.

According to Clause 7.3.2 of GB50017-2017, $\phi Af = 0.572 \times 2{,}760 \times 215 = 339.4\ kN < N = 416.2\ kN$

Therefore, the limit value of width to thickness ratio without magnification.

According to formula 7.3.1-7 of GB50017-2017, $w = b - 2t = 100 - 2 \times 7 = 86\ mm$

$$\frac{w}{t} = \frac{86}{7} = 12.29 < 17.17$$

$$\frac{\text{Actual value}}{\text{Limit value}} = \frac{12.29}{17.17} = 0.72 < 1.0$$

The double angle members are composed of equilateral single angle. So, the check process of the height thickness ratio of compression web is consistent with that of width thickness ratio of compression flange. Thus, OK.

Stability of the compression member: according to formula (7.2.1) of GB50017-2017:

$$\frac{N}{\phi Af} = \frac{416{,}200}{0.572 \times 2{,}760 \times 215} = 1.23 > 1.0, \text{ Failed}$$

Shear Strength

According to formula (7.2.7) of GB50017-2017,

$$V = \frac{A \times f}{85\varepsilon_k} = \frac{2{,}760 \times 215}{85 \times 1} = 6{,}981 \text{ N}$$

Then, according to formula (6.1.3) of GB50017-2017,

$$\tau = \frac{VS}{It_w} = \frac{6{,}981 \times 36{,}700}{2{,}630{,}000 \times 14} = 6.957 \text{ N/mm}^2$$

Therefore, the ratio is:

$$\frac{\tau}{f_v} = \frac{6.957}{125} = 0.06$$

Comparison

Table 627: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Compression Slenderness	0.64	0.65	none	
Tension Slenderness	0.32	0.32	none	
Truss Strength	0.70	0.70	none	
Compression Flange Slenderness	0.72	0.72	none	
Compression Web Slenderness	0.72	0.72	none	
Compression Stability	1.23	1.23	none	
Shear strength	0.06	0.06	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\China\GB500017-2017 Double Angle section with Axial Force.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
```

```
1 0 0 0; 2 1.66667 0 0; 3 3.33333 0 0; 4 5 0 0; 5 6.66667 0 0; 6 8.33333 0 0;
7 10 0 0; 8 1.66667 2.5 0; 9 3.33333 2.5 0; 10 5 2.5 0; 11 6.66667 2.5 0;
12 8.33333 2.5 0; 13 0 0 2; 14 1.66667 0 2; 15 3.33333 0 2; 16 5 0 2;
17 6.66667 0 2; 18 8.33333 0 2; 19 10 0 2; 20 1.66667 2.5 2; 21 3.33333 2.5 2;
22 5 2.5 2; 23 6.66667 2.5 2; 24 8.33333 2.5 2;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 8 9; 8 9 10; 9 10 11; 10 11 12;
11 1 8; 12 8 3; 13 3 10; 14 10 5; 15 5 12; 16 12 7; 17 2 8; 18 4 10; 19 6 12;
20 3 9; 21 5 11; 22 13 14; 23 14 15; 24 15 16; 25 16 17; 26 17 18; 27 18 19;
28 20 21; 29 21 22; 30 22 23; 31 23 24; 32 13 20; 33 20 15; 34 15 22; 35 22
17;
36 17 24; 37 24 19; 38 14 20; 39 16 22; 40 18 24; 41 15 21; 42 17 23; 43 1 13;
44 2 14; 45 3 15; 46 4 16; 47 5 17; 48 6 18; 49 7 19; 50 8 20; 51 9 21;
52 10 22; 53 11 23; 54 12 24;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 10 22 TO 31 43 TO 54 TABLE ST PIP152X8.0
12 15 17 TO 21 33 36 38 TO 42 TABLE ST L80X80X6
11 16 32 37 TABLE SD L100X100X7
13 14 34 35 TABLE ST L100X100X6
CONSTANTS
MATERIAL STEEL ALL
MEMBER TRUSS
11 TO 21 32 TO 42
SUPPORTS
1 7 13 19 PINNED
LOAD 1 LOADTYPE None  TITLE DL
JOINT LOAD
2 TO 6 8 TO 12 14 TO 18 20 TO 24 FY -20
8 TO 12 20 TO 24 FY -40
LOAD 2 LOADTYPE None  TITLE LL
JOINT LOAD
8 TO 12 20 TO 24 FY -30
LOAD COMB 3 F : 1.20DL
1 1.2
LOAD COMB 4 F : 1.20DL+1.40LL
1 1.2 2 1.4
LOAD COMB 5 F : 1.00DL
1 1.0
LOAD COMB 6 F : 1.00DL+1.40LL
1 1.0 2 1.4
LOAD COMB 7 F : 1.20DL+0.98LL
1 1.2 2 0.98
LOAD COMB 8 F : 1.00DL+0.98LL
1 1.0 2 0.98
LOAD COMB 9 F : 1.35DL
1 1.35
LOAD COMB 10 F : 1.35DL+0.98LL
1 1.35 2 0.98
```

```
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.

[GROUP=1]
Name(Parameter Name)=DOUBLEANGLE
Type(Member Type)=2
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=ALL
CheckDispLoadCase(Displacement Loads Case No.)=ALL
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
```

```
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=0
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=400
DJ1(Start Node Number in Major Axis)=0
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=1
miuy(Effective Length Factor for Column in Minor Axis)=1
Lateral(Member in Frame Without Sidesway or not)=1
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=1
rFlange(Limit Ratio of Width to Thickness for Flange)=0
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=0
```

```
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=1
Members(Member Number)=32
```

Chinese Steel Design Workflow Report

Member Check

Member No.	32

Member Type: Truss Member
Project Name: GB500017-2017 Double Angle section with Axial Force
Code for design of steel structure: GB50017-2017

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc	
1	Compression Stability	215	263.4	1.23	1.23		4	
2	Compression Flange Slenderness		17.2	12.3	0.72	0.72		4
3	Compression Web Slenderness	17.2	12.3	0.72	0.72		4	
4	Truss Strength	215	150.8	0.7	0.7		4	
5	Compression Slenderness	150	97.3	0.65	0.65		1	
6	Tension Slenderness	300	97.3	0.32	0.32		1	
7	Compression Shear	125	7	0.06	0.06		1	
Max(Ratio/Failure ratio)		**1.225**						

V. GB500017-2017 H Section Subject to Bending

Verify the strength, stability, and deflection of an H section subject to bending per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is an HN500x200 with a length of 2.5 m. The structure is a two-story frame. Member #40 assigned with an H-section (HN500x200) is designed per GB 50017-2017. The section has an unbraced length of 1.0 m about the x direction and 2.5 m about the y direction.

Material Properties

The material is Q235 type steel.

Design strength in tension, compression, and flexure: $f_y = 215\ MPa$
Design strength in shear: $f_v = 125\ MPa$

Section Properties

Section depth, h = *500 mm*
Section width, b = *200 mm*
Flange thickness, t_f = *16 mm*
Web thickness, t_w = *10 mm*
Cross-sectional area, A = *11,225 mm²*
Moment of inertia about x, I_x = *468,110,000 mm⁴*
Moment of inertia about y, I_y = *21,380,000 mm⁴*
Radius of gyration about x, i_x = *204.2 mm*
Radius of gyration about y, i_y = *43.6 mm*
Area moment about x, S_x = *1,872,400 mm³*
Area moment about y, S_x = *213,800 mm³*

Calculations

Slenderness Ratio

The effective length is:

$$l_{oy} = l_y = 2{,}500 \text{ mm}$$

According to the formula (c.0.1-2) in Appendix C, the slenderness ratio is:

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{2{,}500}{43.6} = 57.3$$

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Overall Stability Coefficient

According to the formula (c.0.5-1) in Appendix C of *Standard for design of steel structures*, the overall stability coefficient of the beam determined by the bending around the strong axis is calculated as:

$$\phi_{bx} = \phi_b = 1.07 - \frac{\lambda_y^2}{44{,}000\varepsilon_k^2} = 1.07 - \frac{(57.3)^2}{44{,}000} = 0.995$$

Plastic Development Coefficient

According to clause 6.1.2 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3,

$$y_x = 1.05$$

$$y_y = 1.20$$

Check Web Thickness to Height Ratio

The calculated height of the web does not include the chamfering arc segment,

$$h_0 = 500 \text{ mm} - 2\times(16 \text{ mm}) = 468 \text{ mm}$$

$$\frac{h_0}{t_f} = \frac{468}{10} = 46.8$$

According to table 3.5.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of height thickness ratio is:

$$93\varepsilon_k = 93$$

Ratio: $\frac{46.8}{93} = 0.50$

Check Flange Thickness to Width Ratio

The overhanging width of flange does not include chamfering arc segment,

$$b_0 = \frac{b - t_w}{2} = \frac{200 - 10}{2} = 95 \text{ mm}$$

$$\frac{b_0}{t_f} = \frac{95}{16} = 5.94$$

According to table 3.5.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of width thickness ratio is:

$$13\varepsilon_k = 13$$

Ratio: $\frac{5.94}{13} = 0.46$

Check Overall Stability

The controlling load condition for the overall stability is combination 59. F : 1.35DL+0.84WF:

$$M_x = -111.2 \text{ kN·m}$$

$$M_y = 2.68 \text{ kN·m}$$

$$\frac{M_x}{\phi_{bx} W_x f} + \frac{M_y}{\gamma_y W_y f} \quad \text{(Cl. 6.2.3)}$$

$$\frac{-111.2(10)^6}{0.995 \times 1,872,400 \times 215} + \frac{2.67(10)^6}{1.2 \times 213,800 \times 215} = 0.33$$

Deflection

From the STAAD analysis, the maximum displacement of the beam is 5.8 mm. The total length of the physical beam member is 10 m.

$$\text{Deflection ratio } = \frac{5.8}{10,000} = 1 / 1,730$$

According to table B.1.1 of Appendix B of Standard for design of steel structures, the allowable deflection ratio is *1 / 400*.

$$\frac{1 / 1,730}{1 / 400} = 0.23$$

Equivalent Stress in Beam

The controlling load condition for equivalent stress is combination 56. F : 1.35DL+0.98LL+0.84WL:

$$M = -111.2 \text{ kN·m}$$

$$V = 43.3 \text{ kN}$$

According to clause 6.1.5 of *Standard for design of steel structures* and taking the top of web as the calculation point, the distance from calculated point to neutral axis of beam:

$$y_1 = \frac{h_w}{2} = \frac{h - 2t_f}{2} = \frac{500 - 2 \times 16}{2} = 234 \text{ mm}$$

The normal stress:

$$\sigma = \frac{M}{I_n} y_1 = \frac{-111{,}200{,}000 \text{ N·mm}}{468{,}110{,}000 \text{ mm}^4} \times 234 \text{ mm} = -55.6 \text{ N/mm}^2$$

According to clause 6.1.3 of *Standard for design of steel structures* and taking the top of web as the calculation point, calculate the area moment of the above rough section to the neutral axis at the shear stress:

$$S = 774{,}400 \text{ mm}^3$$

Shear stress:

$$\tau = \frac{VS}{It_w} = \frac{43{,}300 \text{ N} \times 774{,}400 \text{ mm}^3}{468{,}110{,}000 \text{ mm}^4 \times 10\text{mm}} = 7.163 \text{ N/mm}^2$$

According to clause 6.1.5 of *Standard for design of steel structures*, the local compressive stress $\sigma_c = 0$, so, $\beta_1 = 1.1$. The resultant stress is:

$$\sqrt{\sigma^2 + \sigma_c^2 - \sigma\sigma_c + 3\tau^2}$$

$$= \sqrt{(-55.6)^2 + 0^2 - 0 \times (-55.6) + 3 \times (7.16)^2} = 57.0 \text{ N/mm}^2$$

The allowable stress:

$$\beta_1 f = 1.1 \times 215 \text{ N/mm}^2 = 237 \text{ N/mm}^2$$

$$\sqrt{\sigma^2 + \sigma_c^2 - \sigma\sigma_c + 3\tau^2} < \beta_1 f$$

Ratio: $\frac{\sqrt{\sigma^2 + \sigma_c^2 - \sigma\sigma_c + 3\tau^2}}{\beta_1 f} = \frac{57.0}{237} = 0.24$

Shear Strength

The controlling load condition for shear is combination 10. F : 1.20DL+1.40LL+0.84WL:

$$V_y = 49.9 \text{ kN}$$

$$V_z = 540.3 \text{ kN}$$

Take the neutral axis as the calculation point of shear stress, calculate the area moment:

$$S_y = 1{,}048{,}000 \text{ mm}^3$$

According to clause 6.1.3 of *Standard for design of steel structures*, shear stress

$$\tau = \frac{VS}{It_w} = \frac{49{,}900 \text{ N} \times 1{,}048{,}000 \text{ mm}^3}{468{,}110{,}000 \text{ mm}^4 \times 10\text{mm}} = 11.18 \text{ N/mm}^2$$

$$\tau > f_v = 125 \text{ N/mm}^2$$

Therefore, the ratio is:

$$\frac{\tau}{f_v} = \frac{11.18}{125} = 0.09$$

Bending Strength

The controlling load condition for bending is combination 59. F : 1.35DL+0.84WF:

$$M_x = -111.2 \text{ kN·m}$$

$$M_y = 2.68 \text{ kN·m}$$

According to Clause 6.1.1 of *Standard for design of steel structures*,

$$\frac{M_x}{\gamma_x W_{nx}} + \frac{M_y}{\gamma_y W_{ny}} = \frac{111{,}200{,}000}{1.05 \times 1{,}872{,}400} + \frac{2{,}680{,}000}{1.20 \times 213{,}800} = 67.0 \text{ N/mm}^2$$

Ratio: $\dfrac{\dfrac{M_x}{\gamma_x W_{nx}} + \dfrac{M_y}{\gamma_y W_{ny}}}{f} = \dfrac{67.0}{215} = 0.31$

Comparison

Table 628: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Web Slenderness	0.50	0.50	none	
Flange Slenderness	0.46	0.46	none	
Overall Stability	0.33	0.33	none	
Beam Deflection	0.23	0.23	none	
Equivalent Stress	0.24	0.24	none	
Shear Strength	0.09	0.09	none	
Bending Strength	0.31	0.31	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ \Verification Models\09 Steel Design\China\GB500017-2017 H-section with Bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4.5 0; 3 0 9 0; 5 10 0 0; 6 10 4.5 0; 7 10 9 0; 17 0 0 6;
18 0 4.5 6; 19 0 9 6; 20 10 0 6; 21 10 4.5 6; 22 10 9 6; 23 2.5 4.5 6;
24 5 4.5 6; 25 7.5 4.5 6; 26 2.5 4.5 0; 27 5 4.5 0; 28 7.5 4.5 0; 29 2.5 9 0;
30 5 9 0; 31 7.5 9 0; 32 2.5 9 6; 33 5 9 6; 34 7.5 9 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 4 2 26; 5 3 29; 7 5 6; 8 6 7; 23 3 19; 24 6 21; 25 7 22;
30 17 18; 31 18 19; 32 18 23; 33 19 32; 34 20 21; 35 21 22; 36 23 24; 37 24
25;
38 25 21; 39 26 27; 40 27 28; 41 28 6; 42 29 30; 43 30 31; 44 31 7; 45 32 33;
46 33 34; 47 34 22; 48 23 26; 49 24 27; 50 25 28; 51 32 29; 52 33 30; 53 34
31;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
```

```
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 2 7 8 30 31 34 35 TABLE ST HW400X408
4 5 32 33 36 TO 47 TABLE ST HN500X200
23 TO 25 48 TO 53 TABLE ST HN300X150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 17 20 FIXED
DEFINE WIND LOAD
*{ TYPE AUTO, DON'T MODIFY FOLLOWING DATA
TYPE 2
INT 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 HEIG 0.9 1.8 2.7 3.6 -
4.5 5.4 6.3 7.2 8.1 9
*{ END MAIN TYPE
*{ END INCLINE TYPE
LOAD 1 LOADTYPE None  TITLE DL
SELFWEIGHT Y -1
MEMBER LOAD
23 TO 25 48 TO 53 UNI GY -12.5
LOAD 2 LOADTYPE None  TITLE LL
MEMBER LOAD
23 TO 25 48 TO 53 UMOM GY -10
LOAD 3 WL AUTO WIND LOAD
*{ THE FIRST AUTO BUILD WIND LOAD
*{ 自□定□的□荷□的□型号
*{ TYPE NO : 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17
*{ 建筑□构的□高度
*{ STRUCTURE HIGH : 9
*{ 是否考□□振系数的影响
*{ IS BETAZ : 0
*{ 建筑□构基本自振周期
*{ BASE PERIOD : 0.5
*{ 基本□□□
*{ BASE PRESS : 0.35
*{ □地土粗造度□□
*{ SOIL TYPE : B
*{ 四个□向的迎□面□度
*{ AWEATHER WIDTH : 6 6 10 10
*{ 四个□向的迎□面□度
*{ AWEATHER WIDTH : 6 6 10 10
*{ □构□型（空□或平面）
*{ STRUCTURE TYPE 2
*{ □荷□体型系数，四个□向，四个面
*{ LEFTWIND MIUS : 0.8 -0.5 -0.6 -0.6
*{ RIGHTWIND MIUS : -0.5 0.8 -0.6 -0.6
*{ FRONTWIND MIUS : -0.6 -0.6 0.8 -0.5
*{ BACKWIND MIUS : -0.6 -0.6 -0.5 0.8
*{ □算□□□度/高度曲□的方式
*{ INTENSITY FLAG : 1
*{ 用□指定需要□算□□□度的点数
*{ INTENSITY NUMBER : 10
```

```
*{ 等距离□算□□□度□的距离□
*{ INTENSITY ISOMETRY : 5
*{ 按照□高□算各点□□□度□的□高度
*{ INTENSITY FLOOR : 4.5
WIND LOAD X 0.8 TYPE 2
WIND LOAD -X 0.5 TYPE 2
WIND LOAD -Z 0.6 TYPE 2
WIND LOAD -Z -0.6 TYPE 2
LOAD 4 WR AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD X -0.5 TYPE 2
WIND LOAD -X -0.8 TYPE 2
WIND LOAD -Z 0.6 TYPE 2
WIND LOAD -Z -0.6 TYPE 2
LOAD 5 WF AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD -X -0.6 TYPE 2
WIND LOAD -X 0.6 TYPE 2
WIND LOAD -Z -0.8 TYPE 2
WIND LOAD Z -0.5 TYPE 2
LOAD 6 WB AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD -X -0.6 TYPE 2
WIND LOAD -X 0.6 TYPE 2
WIND LOAD -Z 0.5 TYPE 2
WIND LOAD Z 0.8 TYPE 2
LOAD COMB 7 F : 1.20DL
1 1.2
LOAD COMB 8 F : 1.20DL+1.40LL
1 1.2 2 1.4
LOAD COMB 9 F : 1.20DL+0.84WL
1 1.2 3 0.84
LOAD COMB 10 F : 1.20DL+1.40LL+0.84WL
1 1.2 2 1.4 3 0.84
LOAD COMB 11 F : 1.20DL+0.84WR
1 1.2 4 0.84
LOAD COMB 12 F : 1.20DL+1.40LL+0.84WR
1 1.2 2 1.4 4 0.84
LOAD COMB 13 F : 1.20DL+0.84WF
1 1.2 5 0.84
LOAD COMB 14 F : 1.20DL+1.40LL+0.84WF
1 1.2 2 1.4 5 0.84
LOAD COMB 15 F : 1.20DL+0.84WB
1 1.2 6 0.84
LOAD COMB 16 F : 1.20DL+1.40LL+0.84WB
1 1.2 2 1.4 6 0.84
LOAD COMB 17 F : 1.00DL
1 1.0
LOAD COMB 18 F : 1.00DL+1.40LL
1 1.0 2 1.4
LOAD COMB 19 F : 1.00DL+0.84WL
1 1.0 3 0.84
LOAD COMB 20 F : 1.00DL+1.40LL+0.84WL
1 1.0 2 1.4 3 0.84
LOAD COMB 21 F : 1.00DL+0.84WR
1 1.0 4 0.84
LOAD COMB 22 F : 1.00DL+1.40LL+0.84WR
1 1.0 2 1.4 4 0.84
```

```
LOAD COMB 23 F : 1.00DL+0.84WF
1 1.0 5 0.84
LOAD COMB 24 F : 1.00DL+1.40LL+0.84WF
1 1.0 2 1.4 5 0.84
LOAD COMB 25 F : 1.00DL+0.84WB
1 1.0 6 0.84
LOAD COMB 26 F : 1.00DL+1.40LL+0.84WB
1 1.0 2 1.4 6 0.84
LOAD COMB 27 F : 1.20DL+0.98LL
1 1.2 2 0.98
LOAD COMB 28 F : 1.20DL+0.98LL+0.84WL
1 1.2 2 0.98 3 0.84
LOAD COMB 29 F : 1.20DL+0.98LL+0.84WR
1 1.2 2 0.98 4 0.84
LOAD COMB 30 F : 1.20DL+0.98LL+0.84WF
1 1.2 2 0.98 5 0.84
LOAD COMB 31 F : 1.20DL+0.98LL+0.84WB
1 1.2 2 0.98 6 0.84
LOAD COMB 32 F : 1.00DL+0.98LL
1 1.0 2 0.98
LOAD COMB 33 F : 1.00DL+0.98LL+0.84WL
1 1.0 2 0.98 3 0.84
LOAD COMB 34 F : 1.00DL+0.98LL+0.84WR
1 1.0 2 0.98 4 0.84
LOAD COMB 35 F : 1.00DL+0.98LL+0.84WF
1 1.0 2 0.98 5 0.84
LOAD COMB 36 F : 1.00DL+0.98LL+0.84WB
1 1.0 2 0.98 6 0.84
LOAD COMB 37 F : 1.20DL+1.40WL
1 1.2 3 1.4
LOAD COMB 38 F : 1.20DL+0.98LL+1.40WL
1 1.2 2 0.98 3 1.4
LOAD COMB 39 F : 1.20DL+1.40WR
1 1.2 4 1.4
LOAD COMB 40 F : 1.20DL+0.98LL+1.40WR
1 1.2 2 0.98 4 1.4
LOAD COMB 41 F : 1.20DL+1.40WF
1 1.2 5 1.4
LOAD COMB 42 F : 1.20DL+0.98LL+1.40WF
1 1.2 2 0.98 5 1.4
LOAD COMB 43 F : 1.20DL+1.40WB
1 1.2 6 1.4
LOAD COMB 44 F : 1.20DL+0.98LL+1.40WB
1 1.2 2 0.98 6 1.4
LOAD COMB 45 F : 1.00DL+1.40WL
1 1.0 3 1.4
LOAD COMB 46 F : 1.00DL+0.98LL+1.40WL
1 1.0 2 0.98 3 1.4
LOAD COMB 47 F : 1.00DL+1.40WR
1 1.0 4 1.4
LOAD COMB 48 F : 1.00DL+0.98LL+1.40WR
1 1.0 2 0.98 4 1.4
LOAD COMB 49 F : 1.00DL+1.40WF
1 1.0 5 1.4
LOAD COMB 50 F : 1.00DL+0.98LL+1.40WF
1 1.0 2 0.98 5 1.4
LOAD COMB 51 F : 1.00DL+1.40WB
1 1.0 6 1.4
```

```
LOAD COMB 52 F : 1.00DL+0.98LL+1.40WB
1 1.0 2 0.98 6 1.4
LOAD COMB 53 F : 1.35DL
1 1.35
LOAD COMB 54 F : 1.35DL+0.98LL
1 1.35 2 0.98
LOAD COMB 55 F : 1.35DL+0.84WL
1 1.35 3 0.84
LOAD COMB 56 F : 1.35DL+0.98LL+0.84WL
1 1.35 2 0.98 3 0.84
LOAD COMB 57 F : 1.35DL+0.84WR
1 1.35 4 0.84
LOAD COMB 58 F : 1.35DL+0.98LL+0.84WR
1 1.35 2 0.98 4 0.84
LOAD COMB 59 F : 1.35DL+0.84WF
1 1.35 5 0.84
LOAD COMB 60 F : 1.35DL+0.98LL+0.84WF
1 1.35 2 0.98 5 0.84
LOAD COMB 61 F : 1.35DL+0.84WB
1 1.35 6 0.84
LOAD COMB 62 F : 1.35DL+0.98LL+0.84WB
1 1.35 2 0.98 6 0.84
LOAD COMB 63 D : 1.00DL
1 1.0
LOAD COMB 64 D : 1.00DL+1.00LL
1 1.0 2 1.0
LOAD COMB 65 D : 1.00DL+0.60WL
1 1.0 3 0.6
LOAD COMB 66 D : 1.00DL+1.00LL+0.60WL
1 1.0 2 1.0 3 0.6
LOAD COMB 67 D : 1.00DL+0.60WR
1 1.0 4 0.6
LOAD COMB 68 D : 1.00DL+1.00LL+0.60WR
1 1.0 2 1.0 4 0.6
LOAD COMB 69 D : 1.00DL+0.60WF
1 1.0 5 0.6
LOAD COMB 70 D : 1.00DL+1.00LL+0.60WF
1 1.0 2 1.0 5 0.6
LOAD COMB 71 D : 1.00DL+0.60WB
1 1.0 6 0.6
LOAD COMB 72 D : 1.00DL+1.00LL+0.60WB
1 1.0 2 1.0 6 0.6
LOAD COMB 73 D : 1.00DL+0.70LL
1 1.0 2 0.7
LOAD COMB 74 D : 1.00DL+0.70LL+0.60WL
1 1.0 2 0.7 3 0.6
LOAD COMB 75 D : 1.00DL+0.70LL+0.60WR
1 1.0 2 0.7 4 0.6
LOAD COMB 76 D : 1.00DL+0.70LL+0.60WF
1 1.0 2 0.7 5 0.6
LOAD COMB 77 D : 1.00DL+0.70LL+0.60WB
1 1.0 2 0.7 6 0.6
LOAD COMB 78 D : 1.00DL+1.00WL
1 1.0 3 1.0
LOAD COMB 79 D : 1.00DL+0.70LL+1.00WL
1 1.0 2 0.7 3 1.0
LOAD COMB 80 D : 1.00DL+1.00WR
1 1.0 4 1.0
```

```
LOAD COMB 81 D : 1.00DL+0.70LL+1.00WR
1 1.0 2 0.7 4 1.0
LOAD COMB 82 D : 1.00DL+1.00WF
1 1.0 5 1.0
LOAD COMB 83 D : 1.00DL+0.70LL+1.00WF
1 1.0 2 0.7 5 1.0
LOAD COMB 84 D : 1.00DL+1.00WB
1 1.0 6 1.0
LOAD COMB 85 D : 1.00DL+0.70LL+1.00WB
1 1.0 2 0.7 6 1.0
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.

[GROUP=1]
Name(Parameter Name)=MAINBEAM
Type(Member Type)=1
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
```

```
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=ALL
CheckDispLoadCase(Displacement Loads Case No.)=63 To 85
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=0
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=400
DJ1(Start Node Number in Major Axis)=0
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=0
miuy(Effective Length Factor for Column in Minor Axis)=0
Lateral(Member in Frame Without Sidesway or not)=0
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
rFlange(Limit Ratio of Width to Thickness for Flange)=0
```

```
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=0
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=1
Members(Member Number)=40
```

Chinese Steel Design Workflow Report

Member Check

Member No.	40

Member Type: Bending Member
Project Name: GB500017-2017 H-section with Bending
Code for design of steel structure: GB50017-2017

Section

Section Name HN500X200

Steel No.

Material Name Q235

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc
1	Web Slenderness	93	46.8	0.5	0.5		1
2	Flange Slenderness	13	5.9	0.46	0.46		1
3	Overall Stability	215	70.1	0.33	0.33		59
4	Beam Bending Strength	215	67	0.31	0.31		59
5	Equivalent Stress	215	51.8	0.24	0.24		56
6	Beam Deflection	1/400	1/1706.2	0.23	0.23		80
7	Shear Strength	125	11.2	0.09	0.09		10
Max(Ratio/Failure ratio)		**0.503**					

V. GB500017-2017 H-section with Combined Axial and Bending

Verify the strength, stability, and slenderness of an H section subject to combined axial and bending per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is an HN400x408 with a length of 9 m. The structure is a two-story frame. Member #1 assigned with an H-section (HN400x408) is designed per GB 50017-2017. The section has an unbraced length of 4.5 m about the x direction and 9 m about the y direction.

Material Properties

The material is Q235 type steel.

Design strength in tension, compression, and flexure: $f_y = 205\ MPa$
Design strength in shear: $f_v = 120\ MPa$

Section Properties

Section depth, $h = 400\ mm$
Section width, $b = 408\ mm$
Flange thickness, $t_f = 21\ mm$
Web thickness, $t_w = 21\ mm$
Cross-sectional area, $A = 25{,}069\ mm^2$
Moment of inertia about x, $I_x = 708{,}880{,}000\ mm^4$
Moment of inertia about y, $I_y = 238{,}090{,}000\ mm^4$
Radius of gyration about x, $i_x = 168.2\ mm$
Radius of gyration about y, $i_y = 97.5\ mm$
Area moment about x, $S_x = 3{,}544{,}400\ mm^3$
Area moment about y, $S_x = 1{,}167{,}100\ mm^3$

Calculations

Slenderness Ratio

The effective length is:

$$l_{ox} = \mu_x \times l_x = 1.581 \times 4{,}500 \text{ mm} = 7{,}115 \text{ mm}$$

$$l_{oy} = \mu_y \times l_y = 1.319 \times 9{,}000 \text{ mm} = 11{,}870 \text{ mm}$$

The slenderness ratio is:

$$\lambda_x = \frac{l_{0x}}{i_x} = \frac{7{,}115}{168.2} = 42.3 < 150$$

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{11{,}870}{97.5} = 121.8 < 150$$

Ratio for tension slenderness: 42.3 / 150 = 0.28

Ratio for compression slenderness: 121.8 / 150 = 0.81

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Overall Stability Coefficient

According to the formula (C.0.1) in Appendix C of *Standard for design of steel structures*, the overall stability coefficient of the beam determined by the bending around the strong axis is calculated as:

$$\phi_{bx} = \beta_b \frac{4320}{\lambda_y^2} \times \frac{Ah}{W_y}\left[\sqrt{1 + \left(\frac{\lambda_y t_1}{4.4h}\right)^2} + \eta_b\right]\varepsilon_k$$

$$= 0.765 \times \frac{4{,}320}{(121.8)^2} \times \frac{250{,}691 \times 400}{354{,}4400}\left[\sqrt{1 + \left(\frac{121.8 \times 21}{4.4 \times 400}\right)^2} + 0\right] \times 1 = 1.113 > 0.6$$

According to the formula (c.0.1-7) in Appendix C of *Standard for design of steel structures*,

$$\phi_{bx}' = 1.07 - \frac{0.282}{1.113} = 0.817 < 1.0$$

So, for the section in x-x direction, $\phi_{bx} = 1.0$.

Stability Factor for Axial Compression

According to the formula (D.0.5) in Appendix D of *Standard for design of steel structures*,

For section y-y direction:

$$\lambda_n = \frac{\lambda_y}{\pi}\sqrt{\frac{f_y}{E}} = \frac{121.8}{\pi}\sqrt{\frac{225}{206{,}000}} = 1.281$$

Class c, so, the coefficients are $\alpha 1 = 0.730$, $\alpha 2 = 1.126$, and $\alpha 3 = 0.302$.

Since $\lambda_n = 1.281 > 0.215$, then the stability factor in y:

$$\phi_y = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 1.281^2}\left[\left(1.216 + 0.302 \times 1.281 + 1.281^2\right) - \sqrt{\left(1.216 + 0.302 \times 1.281 + 1.281^2\right)^2 - 4 \times 1.281^2}\right] = 0.382$$

For section x-x direction:

$$\lambda_n = \frac{\lambda_y}{\pi}\sqrt{\frac{f_y}{E}} = \frac{42.3}{\pi}\sqrt{\frac{225}{206{,}000}} = 0.445$$

Class b, so, the coefficients are $\alpha 1 = 0.650$, $\alpha 2 = 0.965$, and $\alpha 3 = 0.300$.

Since $\lambda_n = 0.445 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 0.445^2}\left[\left(0.965 + 0.300 \times 0.445 + 0.445^2\right) - \sqrt{\left(0.965 + 0.300 \times 0.445 + 0.445^2\right)^2 - 4 \times 0.445^2}\right] = 0.893$$

Equivalent Moment Factor

The critical buckling load:

$$N_{cry} = \frac{\pi^2 E I_y}{\left(\mu_y l_y\right)^2} = \frac{\pi^2 \times 206{,}000 \times 238{,}090{,}000}{(1.319 \times 9{,}000)^2} = 3{,}435{,}000 \text{ N}$$

$$N_{crx} = \frac{\pi^2 EI_x}{(\mu_x l_x)^2} = \frac{\pi^2 \times 206000 \times 708{,}880{,}000}{(1.581 \times 4{,}500)^2} = 28{,}474{,}000 \text{ N}$$

For an unbraced frame:

$$\beta_{my} = 1 - \frac{0.36N}{N_{cry}} = 1 - 0.36 \times \frac{181.1}{3{,}435} = 0.981$$

$$\beta_{mx} = 1 - \frac{0.36N}{N_{crx}} = 1 - 0.36 \times \frac{181.1}{28{,}474} = 0.998$$

Because there is a reverse bending point, $M1$ and $M2$ have different signs:

$$\beta_{tx} = 0.65 + 0.35 M_2 / M_1 = 0.65 - 0.35 \times 12.66 / 174.5 = 0.625$$

$$\beta_{ty} = 0.65 + 0.35 M_{2y} / M_{1y} = 0.65 - 0.35 \times 7.39 / 57.06 = 0.605$$

Section influence coefficient, $\eta = 1.0$

Plastic Development Coefficient

According to clause 6.1.2 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3,

$$\gamma_x = 1.05$$

$$\gamma_y = 1.20$$

Check Web Thickness to Height Ratio

Calculated the web height:

$$h_0 = 400 - 21 - 21 = 358 \text{ mm}$$

$$\frac{h_0}{t_w} = \frac{458}{21} = 17.05$$

According to table 3.5.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of height thickness ratio is:

$$40 + 18 \times \alpha_0^{1.5}$$

where

$$\sigma_{max} = \frac{N_x}{A} + \frac{M_x}{W_{z\text{-}web}} = \frac{176{,}800}{25{,}069} + \frac{16{,}000 \times (400 - 21 - 21)}{708{,}800{,}000 \times 2} = 7.056$$

where

$$a_{min} = \frac{N_x}{A} - \frac{M_x}{W_{z\text{-}web}} = \frac{176{,}800}{25{,}069} - \frac{16{,}000 \times (400 - 21 - 21)}{708{,}800{,}000 \times 2} = 7.048$$

where

$$a_0 = \frac{a_{max} - a_{min}}{a_{max}} = \frac{7.056 - 7.048}{7.056} = 0.001$$

Therefore, the limit value is: $40 + 18 \times 0.001^{1.5} = 40.00$

Ratio: $\frac{17.05}{40.00} = 0.43$

Check Flange Thickness to Width Ratio

$$b_0 = \frac{b - t_w}{2} = \frac{408 - 21}{2} = 194 \text{ mm}$$

$$\frac{b_0}{t_f} = \frac{193}{21} = 9.21$$

According to table 3.5.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of width thickness ratio is:

$$13\varepsilon_k = 13$$

Ratio: $\frac{9.21}{13} = 0.71$

<u>Shear Strength</u>

The controlling load condition for shear is combination 10. F : 1.20DL+1.40LL+0.84WL:

$V_x = 46.5$ kN

$V_y = 9.52$ kN

Take the neutral axis as the calculation point of shear stress, calculate the area moment:

$S_x = 1{,}960{,}000 \text{ mm}^3$

$S_y = 893{,}700 \text{ mm}^3$

According to clause 6.1.3 of *Standard for design of steel structures*, shear stress

$$\tau_x = \frac{VS}{It_w} = \frac{46{,}500 \text{ N} \times 1{,}960{,}000 \text{ mm}^3}{708{,}880{,}000 \text{ mm}^4 \times 21\text{mm}} = 6.12 \text{ N/mm}^2$$

$$\tau_y = \frac{VS}{It_w} = \frac{9{,}526 \text{ N} \times 893{,}700 \text{ mm}^3}{238{,}090{,}000 \text{ mm}^4 \times 21\text{mm}} = 0.85 \text{ N/mm}^2$$

$\tau_{max} = 6.12 < f_v = 120 \text{ N/mm}^2$

Therefore, the ratio is:

$$\frac{\tau}{f_v} = \frac{6.12}{120} = 0.05$$

<u>Check In-plain Stability</u>

The controlling load condition for in-plane stability is combination 14. F : 1.2DL+1.4LL+0.84WF:

$M_x = 174.8$ kN·m

$M_y = 58.75$ kN·m

N = 181.2 kN

According to clause 8.2.5 of *Standard for design of steel structures*,

$$N_{Ex} = \frac{\pi^2 EA}{1.1\lambda_x^2} = \frac{\pi^2 206{,}000 \times 25{,}069}{1.1 \times 42.3^2} = 25{,}900 \text{ kN}$$

$$\frac{N}{\phi_x Af} + \frac{\beta_{mx} M_x}{\gamma_x W_x \left(1 - 0.8\frac{N}{N_{Ex}}\right) f} + \eta \frac{\beta_{ty} M_y}{\varphi_{by} W_y f} \qquad \text{(Cl. 8.2.5)}$$

$$= \frac{181.2}{0.893 \times 25{,}069 \times 205} + \frac{0.998 \times 174{,}8 \times 10^9}{1.05 \times 3{,}544{,}400 \times \left(1 - 0.8\frac{181.2}{25{,}900}\right) \times 205} + 1.0 \times \frac{0.605 \times 58.75 \times 10^6}{1.0 \times 1{,}167{,}110 \times 205} = 0.42 < 1.0$$

<u>Check Out-of-plain Stability</u>

The controlling load condition for in-plane stability is combination 14. F : 1.2DL+1.4LL+0.84WF:

$M_x = 174.8$ kN·m

$M_y = 58.75$ kN·m

N = 181.2 kN

According to clause 8.2.5 of *Standard for design of steel structures*,

$$N_{Ex} = \frac{\pi^2 EA}{1.1\lambda_x^2} = \frac{\pi^2 206{,}000 \times 25{,}069}{1.1 \times 121.8^2} = 3{,}123 \text{ kN}$$

$$\frac{N}{\phi_x A f} + \frac{\beta_{mx} M_x}{\gamma_x W_x \left(1 - 0.8\frac{N}{N_{Ex}}\right) f} + \eta \frac{\beta_{ty} M_y}{\varphi_{by} W_y f} \quad \text{(Cl. 8.2.5)}$$

$$= \frac{181.2 \times 1000}{0.382 \times 25{,}069 \times 205} + \frac{0.981 \times 58.75 \times 10^9}{1.2 \times 1{,}167{,}100 \times \left(1 - 0.8\frac{181.1}{3{,}123}\right) \times 205} + 1.0 \times \frac{0.625 \times 174.8 \times 10^9}{0.817 \times 3{,}544{,}400 \times 205} = 0.49 < 1.0$$

<u>Strength of the Member</u>

The controlling load condition for strength is combination 14. F : 1.2DL+1.4LL+0.84WF:

According to Clause 8.1.1 of *Standard for design of steel structures*,

$$\frac{N}{A_f f} + \frac{M_x}{r_x W_{nx} f} + \frac{M_y}{r_y W_{ny} f} = \frac{181.2}{25{,}069 \times 205} \frac{174.8 \times 10^6}{1.05 \times 3{,}544{,}400 \times 205} + \frac{58.75 \times 10^6}{1.20 \times 1{,}167{,}100 \times 205} = 0.47$$

Comparison

Table 629: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Column Strength	0.47	0.47	none	
In-plane Stability	0.42	0.42	none	
Out-plane Stability	0.49	0.49	none	
Compression Slenderness	0.81	0.81	none	
Tension Slenderness	0.41	0.41	none	
Shear Strength	0.05	0.05	none	
Flange Slenderness	0.71	0.71	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ \Verification Models\09 Steel Design\China\GB500017-2017 H-section with Combined Axial and Bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4.5 0; 3 0 9 0; 5 10 0 0; 6 10 4.5 0; 7 10 9 0; 17 0 0 6;
18 0 4.5 6; 19 0 9 6; 20 10 0 6; 21 10 4.5 6; 22 10 9 6; 23 2.5 4.5 6;
24 5 4.5 6; 25 7.5 4.5 6; 26 2.5 4.5 0; 27 5 4.5 0; 28 7.5 4.5 0; 29 2.5 9 0;
30 5 9 0; 31 7.5 9 0; 32 2.5 9 6; 33 5 9 6; 34 7.5 9 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 4 2 26; 5 3 29; 7 5 6; 8 6 7; 23 3 19; 24 6 21; 25 7 22;
30 17 18; 31 18 19; 32 18 23; 33 19 32; 34 20 21; 35 21 22; 36 23 24; 37 24
25;
38 25 21; 39 26 27; 40 27 28; 41 28 6; 42 29 30; 43 30 31; 44 31 7; 45 32 33;
46 33 34; 47 34 22; 48 23 26; 49 24 27; 50 25 28; 51 32 29; 52 33 30; 53 34
31;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 2 7 8 30 31 34 35 TABLE ST HW400X408
4 5 32 33 36 TO 47 TABLE ST HN500X200
23 TO 25 48 TO 53 TABLE ST HN300X150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 17 20 FIXED
DEFINE WIND LOAD
*{ TYPE AUTO, DON'T MODIFY FOLLOWING DATA
TYPE 2
INT 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 HEIG 0.9 1.8 2.7 3.6 -
4.5 5.4 6.3 7.2 8.1 9
*{ END MAIN TYPE
*{ END INCLINE TYPE
LOAD 1 LOADTYPE None  TITLE DL
SELFWEIGHT Y -1
MEMBER LOAD
23 TO 25 48 TO 53 UNI GY -12.5
LOAD 2 LOADTYPE None  TITLE LL
MEMBER LOAD
23 TO 25 48 TO 53 UMOM GY -10
LOAD 3 WL AUTO WIND LOAD
*{ THE FIRST AUTO BUILD WIND LOAD
```

```
*{ 自□定□的□荷□的□型号
*{ TYPE NO : 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17
*{ 建筑□构的□高度
*{ STRUCTURE HIGH : 9
*{ 是否考□□振系数的影响
*{ IS BETAZ : 0
*{ 建筑□构基本自振周期
*{ BASE PERIOD : 0.5
*{ 基本□□□
*{ BASE PRESS : 0.35
*{ □地土粗造度□□
*{ SOIL TYPE : B
*{ 四个□向的迎□面□度
*{ AWEATHER WIDTH : 6 6 10 10
*{ 四个□向的迎□面□度
*{ AWEATHER WIDTH : 6 6 10 10
*{ □构□型（空□或平面）
*{ STRUCTURE TYPE 2
*{ □荷□体型系数，四个□向，四个面
*{ LEFTWIND MIUS : 0.8 -0.5 -0.6 -0.6
*{ RIGHTWIND MIUS : -0.5 0.8 -0.6 -0.6
*{ FRONTWIND MIUS : -0.6 -0.6 0.8 -0.5
*{ BACKWIND MIUS : -0.6 -0.6 -0.5 0.8
*{ □算□□□度/高度曲□的方式
*{ INTENSITY FLAG : 1
*{ 用□指定需要□算□□□度的点数
*{ INTENSITY NUMBER : 10
*{ 等距离□算□□□度□的距离□
*{ INTENSITY ISOMETRY : 5
*{ 按照□高□算各点□□□度□的□高度
*{ INTENSITY FLOOR : 4.5
WIND LOAD X 0.8 TYPE 2
WIND LOAD -X 0.5 TYPE 2
WIND LOAD -Z 0.6 TYPE 2
WIND LOAD -Z -0.6 TYPE 2
LOAD 4 WR AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD X -0.5 TYPE 2
WIND LOAD -X -0.8 TYPE 2
WIND LOAD -Z 0.6 TYPE 2
WIND LOAD -Z -0.6 TYPE 2
LOAD 5 WF AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD -X -0.6 TYPE 2
WIND LOAD -X 0.6 TYPE 2
WIND LOAD -Z -0.8 TYPE 2
WIND LOAD Z -0.5 TYPE 2
LOAD 6 WB AUTO WIND LOAD
*{ AUTO BUILD WIND LOAD
WIND LOAD -X -0.6 TYPE 2
WIND LOAD -X 0.6 TYPE 2
WIND LOAD -Z 0.5 TYPE 2
WIND LOAD Z 0.8 TYPE 2
LOAD COMB 7 F : 1.20DL
1 1.2
LOAD COMB 8 F : 1.20DL+1.40LL
1 1.2 2 1.4
LOAD COMB 9 F : 1.20DL+0.84WL
```

```
1 1.2 3 0.84
LOAD COMB 10 F : 1.20DL+1.40LL+0.84WL
1 1.2 2 1.4 3 0.84
LOAD COMB 11 F : 1.20DL+0.84WR
1 1.2 4 0.84
LOAD COMB 12 F : 1.20DL+1.40LL+0.84WR
1 1.2 2 1.4 4 0.84
LOAD COMB 13 F : 1.20DL+0.84WF
1 1.2 5 0.84
LOAD COMB 14 F : 1.20DL+1.40LL+0.84WF
1 1.2 2 1.4 5 0.84
LOAD COMB 15 F : 1.20DL+0.84WB
1 1.2 6 0.84
LOAD COMB 16 F : 1.20DL+1.40LL+0.84WB
1 1.2 2 1.4 6 0.84
LOAD COMB 17 F : 1.00DL
1 1.0
LOAD COMB 18 F : 1.00DL+1.40LL
1 1.0 2 1.4
LOAD COMB 19 F : 1.00DL+0.84WL
1 1.0 3 0.84
LOAD COMB 20 F : 1.00DL+1.40LL+0.84WL
1 1.0 2 1.4 3 0.84
LOAD COMB 21 F : 1.00DL+0.84WR
1 1.0 4 0.84
LOAD COMB 22 F : 1.00DL+1.40LL+0.84WR
1 1.0 2 1.4 4 0.84
LOAD COMB 23 F : 1.00DL+0.84WF
1 1.0 5 0.84
LOAD COMB 24 F : 1.00DL+1.40LL+0.84WF
1 1.0 2 1.4 5 0.84
LOAD COMB 25 F : 1.00DL+0.84WB
1 1.0 6 0.84
LOAD COMB 26 F : 1.00DL+1.40LL+0.84WB
1 1.0 2 1.4 6 0.84
LOAD COMB 27 F : 1.20DL+0.98LL
1 1.2 2 0.98
LOAD COMB 28 F : 1.20DL+0.98LL+0.84WL
1 1.2 2 0.98 3 0.84
LOAD COMB 29 F : 1.20DL+0.98LL+0.84WR
1 1.2 2 0.98 4 0.84
LOAD COMB 30 F : 1.20DL+0.98LL+0.84WF
1 1.2 2 0.98 5 0.84
LOAD COMB 31 F : 1.20DL+0.98LL+0.84WB
1 1.2 2 0.98 6 0.84
LOAD COMB 32 F : 1.00DL+0.98LL
1 1.0 2 0.98
LOAD COMB 33 F : 1.00DL+0.98LL+0.84WL
1 1.0 2 0.98 3 0.84
LOAD COMB 34 F : 1.00DL+0.98LL+0.84WR
1 1.0 2 0.98 4 0.84
LOAD COMB 35 F : 1.00DL+0.98LL+0.84WF
1 1.0 2 0.98 5 0.84
LOAD COMB 36 F : 1.00DL+0.98LL+0.84WB
1 1.0 2 0.98 6 0.84
LOAD COMB 37 F : 1.20DL+1.40WL
1 1.2 3 1.4
LOAD COMB 38 F : 1.20DL+0.98LL+1.40WL
```

```
1 1.2 2 0.98 3 1.4
LOAD COMB 39 F : 1.20DL+1.40WR
1 1.2 4 1.4
LOAD COMB 40 F : 1.20DL+0.98LL+1.40WR
1 1.2 2 0.98 4 1.4
LOAD COMB 41 F : 1.20DL+1.40WF
1 1.2 5 1.4
LOAD COMB 42 F : 1.20DL+0.98LL+1.40WF
1 1.2 2 0.98 5 1.4
LOAD COMB 43 F : 1.20DL+1.40WB
1 1.2 6 1.4
LOAD COMB 44 F : 1.20DL+0.98LL+1.40WB
1 1.2 2 0.98 6 1.4
LOAD COMB 45 F : 1.00DL+1.40WL
1 1.0 3 1.4
LOAD COMB 46 F : 1.00DL+0.98LL+1.40WL
1 1.0 2 0.98 3 1.4
LOAD COMB 47 F : 1.00DL+1.40WR
1 1.0 4 1.4
LOAD COMB 48 F : 1.00DL+0.98LL+1.40WR
1 1.0 2 0.98 4 1.4
LOAD COMB 49 F : 1.00DL+1.40WF
1 1.0 5 1.4
LOAD COMB 50 F : 1.00DL+0.98LL+1.40WF
1 1.0 2 0.98 5 1.4
LOAD COMB 51 F : 1.00DL+1.40WB
1 1.0 6 1.4
LOAD COMB 52 F : 1.00DL+0.98LL+1.40WB
1 1.0 2 0.98 6 1.4
LOAD COMB 53 F : 1.35DL
1 1.35
LOAD COMB 54 F : 1.35DL+0.98LL
1 1.35 2 0.98
LOAD COMB 55 F : 1.35DL+0.84WL
1 1.35 3 0.84
LOAD COMB 56 F : 1.35DL+0.98LL+0.84WL
1 1.35 2 0.98 3 0.84
LOAD COMB 57 F : 1.35DL+0.84WR
1 1.35 4 0.84
LOAD COMB 58 F : 1.35DL+0.98LL+0.84WR
1 1.35 2 0.98 4 0.84
LOAD COMB 59 F : 1.35DL+0.84WF
1 1.35 5 0.84
LOAD COMB 60 F : 1.35DL+0.98LL+0.84WF
1 1.35 2 0.98 5 0.84
LOAD COMB 61 F : 1.35DL+0.84WB
1 1.35 6 0.84
LOAD COMB 62 F : 1.35DL+0.98LL+0.84WB
1 1.35 2 0.98 6 0.84
LOAD COMB 63 D : 1.00DL
1 1.0
LOAD COMB 64 D : 1.00DL+1.00LL
1 1.0 2 1.0
LOAD COMB 65 D : 1.00DL+0.60WL
1 1.0 3 0.6
LOAD COMB 66 D : 1.00DL+1.00LL+0.60WL
1 1.0 2 1.0 3 0.6
LOAD COMB 67 D : 1.00DL+0.60WR
```

```
1 1.0 4 0.6
LOAD COMB 68 D : 1.00DL+1.00LL+0.60WR
1 1.0 2 1.0 4 0.6
LOAD COMB 69 D : 1.00DL+0.60WF
1 1.0 5 0.6
LOAD COMB 70 D : 1.00DL+1.00LL+0.60WF
1 1.0 2 1.0 5 0.6
LOAD COMB 71 D : 1.00DL+0.60WB
1 1.0 6 0.6
LOAD COMB 72 D : 1.00DL+1.00LL+0.60WB
1 1.0 2 1.0 6 0.6
LOAD COMB 73 D : 1.00DL+0.70LL
1 1.0 2 0.7
LOAD COMB 74 D : 1.00DL+0.70LL+0.60WL
1 1.0 2 0.7 3 0.6
LOAD COMB 75 D : 1.00DL+0.70LL+0.60WR
1 1.0 2 0.7 4 0.6
LOAD COMB 76 D : 1.00DL+0.70LL+0.60WF
1 1.0 2 0.7 5 0.6
LOAD COMB 77 D : 1.00DL+0.70LL+0.60WB
1 1.0 2 0.7 6 0.6
LOAD COMB 78 D : 1.00DL+1.00WL
1 1.0 3 1.0
LOAD COMB 79 D : 1.00DL+0.70LL+1.00WL
1 1.0 2 0.7 3 1.0
LOAD COMB 80 D : 1.00DL+1.00WR
1 1.0 4 1.0
LOAD COMB 81 D : 1.00DL+0.70LL+1.00WR
1 1.0 2 0.7 4 1.0
LOAD COMB 82 D : 1.00DL+1.00WF
1 1.0 5 1.0
LOAD COMB 83 D : 1.00DL+0.70LL+1.00WF
1 1.0 2 0.7 5 1.0
LOAD COMB 84 D : 1.00DL+1.00WB
1 1.0 6 1.0
LOAD COMB 85 D : 1.00DL+0.70LL+1.00WB
1 1.0 2 0.7 6 1.0
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
```

```
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.

[GROUP=1]
Name(Parameter Name)=MAINBEAM
Type(Member Type)=1
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=ALL
CheckDispLoadCase(Displacement Loads Case No.)=63 To 85
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
```

```
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=0
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=400
DJ1(Start Node Number in Major Axis)=0
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=0
miuy(Effective Length Factor for Column in Minor Axis)=0
Lateral(Member in Frame Without Sidesway or not)=0
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
rFlange(Limit Ratio of Width to Thickness for Flange)=0
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=0
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=1
Members(Member Number)=40
```

Chinese Steel Design Workflow Report

Member Check

Member No.	1

Member Type: Column Member
Project Name: GB500017-2017 H-section with Combined Axial and Bending
Code for design of steel structure: GB50017-2017

Section

Section Name HW400X408

Steel No.

Material Name Q235

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc
1	Compression Slenderness	150	121.8	0.81	0.81		1
2	Compression Flange Slenderness	13	9.2	0.71	0.71		1
3	Stability Out-plane	205	99.8	0.49	0.49		14
4	Column Strength	205	96.1	0.47	0.47		14
5	Compression Web Slenderness	40	17	0.43	0.43		65
6	Stability In-plane	205	85.6	0.42	0.42		14
7	Tension Slenderness	300	121.8	0.41	0.41		1
8	Shear Strength	120	6.1	0.05	0.05		20
Max(Ratio/Failure ratio)		**0.812**					

V. GB500017-2017 Pipe section with Combined Axial and Bending

Verify the strength, stability, and slenderness of a pipe section subject to combined axial and bending per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is a PIP299x10.0 with a length of 4 m. The structure is a portal frame. Member #3 assigned with a pipe section (PIP299x10.0) is designed per GB 50017-2017. The section has an unbraced length of 4 m in either direction. The governing load case #1 has the following ultimate loads on the member:

$F_x = 24.45\ kN$
$F_y = 34.94\ kN$
$N = 93.30\ kN$
$M_x = -76.7\ kN \cdot m$ (at member end)
$M_y = 117.8\ kN \cdot m$ (at member end)

<u>Material Properties</u>

The material is Q235 type steel.

Design strength in tension, compression, and flexure: f_y = *215 MPa*
Design strength in shear: f_v = *125 MPa*

Section Properties

Outside diameter, D = *299 mm*
Wall thickness, t = *10 mm*
Cross-sectional area, A = *9,079* mm^2
Moment of inertia about x, I = *94,902,00* mm^4
Radius of gyration about x, = *102.2 mm*
Area moment about x, S = *417,600* mm^3

Calculations

Slenderness Ratio

The effective length is:

$$l_{ox} = \mu_x \times l_x = 1.297 \times 4{,}000 \text{ mm} = 5{,}188 \text{ mm}$$

$$l_{oy} = \mu_y \times l_y = 2.0383 \times 4{,}000 \text{ mm} = 8{,}150 \text{ mm}$$

The slenderness ratio is:

$$\lambda_x = \frac{l_{0x}}{i_x} = \frac{5{,}188}{102.2} = 50.73 < 150$$

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{8{,}150}{102.2} = 79.72 < 150$$

According to table 7.4.6 of *Standard for design of steel structures*, the allowable slenderness ratio of compression members is $\lambda_{clim} = 150$.
Ratio: $\lambda_{max} / \lambda_{clim} = 79.92 / 150 = 0.53$

According to table 7.4.7 of *Standard for design of steel structures*, the allowable slenderness ratio of tension members is $\lambda_{tlim} = 300$.
Ratio: $\lambda_{max} / \lambda_{tlim} = 79.92 / 300 = 0.27$

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Check Diameter to Thickness Ratio

$$\frac{D}{t} = \frac{299}{10} = 29.9$$

According to table 3.5.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of height thickness ratio is:

$$93\varepsilon_k = 93$$

Ratio: $\frac{29.9}{90} = 0.33$

Plastic Development Coefficient

According to clause 8.1.1-2 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3,

$$\gamma_m = 1.15$$

Check Member Strength

According to clause 8.1.1-2 of *Standard for design of steel structures*,

$$\sigma = \frac{N}{A_n} + \frac{\sqrt{M_x^2 + M_y^2}}{\gamma_m W_n} = \frac{N}{\text{A}} + \frac{\sqrt{M_x^2 + M_y^2}}{\gamma_m W}$$

$$= \frac{93.3\times10^3}{9{,}079} + \frac{\sqrt{(76.7, \times, 10^6)^2 + (117.8, \times, 10^6)^2}}{1.15\times634{,}800} = 202.8 \text{ N/mm}^2$$

Ratio: $\frac{\sigma}{f} = \frac{202.8}{215} = 0.94$

Check Member Stability According to table 7.2.1-1 of *Standard for design of steel structures*, the section is “a” for this section.

According to the formula (D.0.5) in Appendix D of *Standard for design of steel structures*,

For section x-x direction:

$$\lambda_n = \frac{\lambda_y}{\pi}\sqrt{\frac{f_y}{E}} = \frac{50.73}{\pi}\sqrt{\frac{225}{206{,}000}} = 0.545$$

Class a, so, the coefficients are $\alpha 1 = 0.41$, $\alpha 2 = 0.986$, and $\alpha 3 = 0.152$.

Since $\lambda_n = 0.545 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2\times0.545^2}\left[\left(0.986 + 0.152\times0.545 + 0.545^2\right) - \sqrt{\left(0.986 + 0.152\times0.445 + 0.545^2\right)^2 - 4\times0.545^2}\right] = 0.914$$

For section y-y direction:

$$\lambda_n = \frac{\lambda_y}{\pi}\sqrt{\frac{f_y}{E}} = \frac{79.72}{\pi}\sqrt{\frac{225}{206{,}000}} = 0.857$$

Class a, so, the coefficients are $\alpha 1 = 0.41$, $\alpha 2 = 0.986$, and $\alpha 3 = 0.152$.

Since $\lambda_n = 0.875 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2\times0.857^2}\left[\left(0.986 + 0.152\times0.857 + 0.857^2\right) - \sqrt{\left(0.986 + 0.152\times0.857 + 0.857^2\right)^2 - 4\times0.857^2}\right] = 0.785$$

So, $\phi_{min} = 0.785$

Equivalent Moment Factor

According to formula 8.2.4-6 of *Standard for design of steel structures*:

$$N_E = \frac{\pi^2 EA}{\lambda^2} = \frac{\pi^2\times206{,}000\times9{,}079}{79.72^2} = 2{,}904{,}600 \text{ N}$$

According to the note of formula 8.2.4 of *Standard for design of steel structures*:

M_{1x} = -76.7 kN·m

M_{2x} = 63.06 kN·m

M_{1y} = 117.8 kN·m

M_{2y} = -0.006 kN·m

According to formula 8.2.4-4 of *Standards for design of steel structures*:

$$\beta_x = 1 - 0.35\sqrt{\frac{N}{N_E}} + \sqrt{\frac{N}{N_E}}\frac{W_{2x}}{W_{1x}} = 1 - 0.35 \times \sqrt{\frac{93.3}{2{,}905}} + 0.35 \times \sqrt{\frac{93.3}{2{,}905}}\frac{63.06}{-76.7} = 0.886$$

According to formula 8.2.4-5 of *Standards for design of steel structures*:

$$\beta_y = 1 - 0.35\sqrt{\frac{N}{N_E}} + \sqrt{\frac{N}{N_E}}\frac{W_{2y}}{W_{1y}} = 1 - 0.35 \times \sqrt{\frac{93.3}{2{,}905}} + 0.35 \times \sqrt{\frac{-0.006}{117.8}}\frac{63.06}{-76.7} = 0.937$$

According to formula 8.2.4-5 of *Standards for design of steel structures*:

$\beta = \beta_x\beta_y = 0.886 \times 0.937 = 0.830$

According to formula 8.2.1-2 of *Standard for design of steel structures*:

$$N'_{Ex} = \frac{\pi^2 EA}{1.1\lambda_{max}^2} = \frac{\pi^2 \times 206{,}000 \times 9{,}079}{1.1 \times 79.72^2} = 2{,}640{,}500 \text{ N}$$

According to the note of formula 8.2.4-2 of *Standard for design of steel structures*:

M_{xA} = 63.06 kN·m

M_{yA} = -0.006 kN·m

M_{xB} = -76.7 kN·m

M_{yB} = 117.8 kN·m

$$\sqrt{M_{xA}^2 + M_{yA}^2} = \sqrt{(63.06)^2 + (-0.006)^2} = 63.06 \text{ kN·m}$$

$$\sqrt{M_{xB}^2 + M_{yB}^2} = \sqrt{(-76.7)^2 + (117.8)^2} = 140.6 \text{ kN·m}$$

M_{max} = 140.6 kN·m

According to the note of formula 8.2.4-2 of *Standard for design of steel structures*:

$$\frac{N}{\phi A f} + \frac{\beta M}{\gamma_m W\left(1 - 0.8\frac{N}{N'_{Ex}}\right)f}$$

$$= \frac{93.3 \times 10^3}{0.875 \times 9{,}079 \times 215} + \frac{0.830 \times 140.6 \times 10^6}{1.15 \times 634{,}792\left(1, \ - , \ \frac{93.3}{2{,}641}\right)215} = 0.826$$

<u>Shear Strength</u>

Take the neutral axis as the calculation point of shear stress, calculate the area moment:

$S = 417{,}605 \text{ mm}^3$

According to clause 6.1.3 of *Standard for design of steel structures*, shear stress

$$\tau_x = \frac{VS}{It_w} = \frac{29{,}449 \times 417{,}605}{94{,}902{,}000 \times 20} = 6.48 \text{ N/mm}^2$$

$$\tau_x = \frac{VS}{It_w} = \frac{34{,}940 \times 417{,}605}{94{,}902{,}000 \times 20} = 7.69 \text{ N/mm}^2$$

$$\tau_{max} = 7.69 < f_v = 125 \text{ N/mm}^2$$

Therefore, the ratio is:

$$\frac{\tau}{f_v} = \frac{7.89}{125} = 0.06$$

Comparison

Table 630: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Compression Slenderness	0.53	0.53	none	
Tension Slenderness	0.27	0.27	none	
Diameter thickness ratio	0.33	0.33	none	
Column Strength	0.94	0.94	none	
In-plane stability	0.83	0.83	none	
Out-plane stability	0.83	0.83	none	
Shear Strength	0.06	0.06	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\China\GB500017-2017 Pipe section with Combined Axial and Bending.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 6 4 0; 4 6 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
```

```
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TABLE ST TUB30030010.0
3 TABLE ST PIP299X10.0
2 TABLE ST HN300X150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -10
JOINT LOAD
2 3 FX 30 FY -50 FZ 30
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.
```

```
[GROUP=1]
Name(Parameter Name)=PIPE
Type(Member Type)=3
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=1
CheckDispLoadCase(Displacement Loads Case No.)=1
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=1
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=400
DJ1(Start Node Number in Major Axis)=0
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
```

```
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=0
miuy(Effective Length Factor for Column in Minor Axis)=0
Lateral(Member in Frame Without Sidesway or not)=0
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
rFlange(Limit Ratio of Width to Thickness for Flange)=0
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=0
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=1
Members(Member Number)=3
```

Chinese Steel Design Workflow Report

Member Check

Member No.	3

Member Type: Column Member
Project Name: GB500017-2017 Pipe section with Combined Axial and Bending
Code for design of steel structure: GB50017-2017

Section

Section Name PIP299X10.0

Steel No.

Material Name Q235

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc
1	Column Strength	215	202.8	0.94	0.94		1
2	Stability In-plane	215	177.6	0.83	0.83		1
3	Stability Out-plane	215	177.6	0.83	0.83		1
4	Compression Slenderness	150	79.7	0.53	0.53		1
5	Compression Flange Slenderness	90	29.9	0.33	0.33		1
6	Compression Web Slenderness	90	29.9	0.33	0.33		1
7	Tension Slenderness	300	79.7	0.27	0.27		1
8	Shear Strength	125	7.7	0.06	0.06		1
Max(Ratio/Failure ratio)		**0.943**					

V. GB500017-2017 Single Angle section with Axial Force

Verify the slenderness, strength, and stability of a single-angle section subject to axial force per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is an L100x100x6 with a length of 3.005 m. The structure is a truss model. Member #34 assigned with single angle section (L100x100x6) is designed per GB 50017-2017. The governing load combination (#4) is 1.2 DL + 1.4 LL and the ultimate force on the member, N, is 92.06 kN of axial compression.

The material is Q235 type steel.

Design strength in tension, compression, and flexure: $f_y = 215\ MPa$
Design strength in shear: $f_v = 125\ MPa$

Calculations

<u>Section Properties</u>

Side length, $b = h = 100\ mm$
Thickness, $t = 6\ mm$
Cross-sectional area, $A = 1{,}193\ mm^2$
Moment of inertia about x, $I_x = 479{,}000\ mm^4$
Moment of inertia about y, $I_y = 1{,}820{,}000\ mm^4$
Radius of gyration about x, $i_x = 20.04\ mm$
Radius of gyration about y, $i_y = 39.06\ mm$
Area moment about x, $S_x = 15{,}890\ mm^3$
Area moment about y, $S_x = 31{,}690\ mm^3$

<u>Slenderness Ratio</u>

According to Table 7.4.1-1 of GB50017-2017, effective length of double angle is:

$$l_{ox} = \mu_x \times l_x = 0.8 \times 3{,}005 \text{ mm} = 2{,}404 \text{ mm}$$

$$l_{oy} = \mu_y \times l_y = 1.0 \times 3{,}005 \text{ mm} = 3{,}005 \text{ mm}$$

According to formula 7.2.1-1 and 2 of steel structures code, slenderness ratio of double angle with X axis and Y axis is:

$$\lambda_x = \frac{l_{0x}}{i_x} = \frac{2{,}404}{20.04} = 119.8$$

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{3{,}005}{39.06} = 76.94$$

According to formula 7.2.2-3 and 7.2.2.-4 of steel structures code, torsional buckling equivalent slenderness ratio of single angle is calculated as follows.

The relative coordinates from the centroid to the heel (corner) of the angle are:

$$x_c = 27.23 \text{ mm}$$

$$y_c = 27.23 \text{ mm}$$

The distance from the centroid to the shear center, y_s:

$$y_z^2 = 2 \times \left(x_c - \frac{t}{2}\right)^2 = 2 \times \left(27.23 - \frac{6}{2}\right)^2 = 1{,}174 \text{ mm}^2$$

The polar radius of gyration of the shear center, i_0:

$$i_0^2 = i_0^2 + i_x^2 + i_y^2 = 1{,}174 + 20.04^2 + 39.06^2 = 3{,}101 \text{ mm}^2$$

Polar moment of inertia of gross angle to the shear center, I_0:

$$I_0 = i_0^2 A = 3{,}101 \times 1{,}193 = 3{,}700{,}000 \text{ mm}^4$$

The free torsion constant of shear center on gross angle section. The correction factor for the free torsion factor, $k = 1.2$.

$$I_t = (I_{t1} + I_{t2})k = \left(\frac{bt^3}{3} + \frac{(b - t)t^3}{3}\right)k = \frac{k(2b - t)t^3}{3} = \frac{1.2 \times (2 \times 100 - 6) \times (6)^3}{3} = 16{,}760 \text{ mm}^4$$

Fan moment of inertia of the gross angle section to shear center, I_ω:

$$I_t = \frac{t^3 b^3}{18} = \frac{(6)^3(100)^3}{18} = 12{,}000{,}000 \text{ mm}^6$$

Length for torsional buckling, $l_\omega = 2{,}404\ mm$. The torsional buckling equivalent slenderness ratio, λ_z^2:

$$\lambda_z^2 = \frac{I_0}{I_t / 25.7 + I_\omega / l_\omega^2} = \frac{3{,}700{,}000}{16{,}762 / 25.7 + 12{,}000{,}000 / 2{,}404^2} = 5{,}655$$

Calculate the slenderness ratio for flexural torsional buckling, λ_{yz}:

$$\lambda_{yz} = \sqrt{\frac{\left(\lambda_y^2 + \lambda_z^2\right) + \sqrt{\left(\lambda_y^2 + \lambda_z^2\right)^2 - 4\left(1 - \frac{y_s^2}{i_0^2}\right)\lambda_y^2 \lambda_z^2}}{2}}$$

$$= \sqrt{\frac{\left(120.0^2 + 5{,}655\right) + \sqrt{\left(120.0^2 + 5{,}655\right)^2 - 4 \times \left(1 - \frac{1{,}174}{3{,}101}\right) \times 120.0^2 \times 5{,}655}}{2}} = 130.7$$

The maximum slenderness ratio of the single angle section, $\lambda_{max} = 130.7$

The limit of slenderness ratio of compression member is 150 according to table 7.4.6 of steel structures code.

$$\lambda_{\text{max}} / \lambda_{\text{limit}} = \frac{130.7}{150} = 0.87 < 1.0, \text{ Passed}$$

The limit of slenderness ratio of tension member is 300 according to table 7.4.7 of steel structures code.

$$\lambda_{\text{max}} / \lambda_{\text{limit}} = \frac{130.7}{300} = 0.44 < 1.0, \text{ Passed}$$

<u>Tension or Compression Strength</u>

According to Table 7.1.3 of GB50017-2017, the $\eta = 0.85$.

According to formula 7.1.1-1 of steel structures code, the stress of single angle with gross cross-section area is:

$$\sigma = \frac{N}{\eta A} = \frac{92{,}060}{0.85 \times 2{,}760} = 90.79$$

According to clause 7.6.1-1 of steel structures code, reduction factor of strength design value of angle is 0.85.

$$0.85 f = 0.85 \text{ x } 215 = 182.8 \text{ N/mm}^2$$

And the ratio is:

$$\frac{\sigma}{0.85 f} = \frac{90.79}{187.8} = 0.50$$

According to formula 7.1.1-2 of the steel structures code, the strength ratio is:

$$0.85 f_u = 0.85 \times 370 = 314.5 \text{ N/mm}^2$$

And the ratio is:

$$\frac{\sigma}{0.7\text{f}_u} = \frac{90.79}{0.7 \times 314.5} = 0.41$$

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Compressive Strength

According to table 7.2.1-1 of steel structures code, the double angle section class is b.

The slenderness ratio of double angle on X axis, $\lambda_x = 76.93$. So,

$$\frac{\lambda}{\varepsilon_k} = \lambda_y = 76.93$$

$$\lambda_n = \frac{\lambda_x}{\pi}\sqrt{\frac{f_y}{E}} = \frac{76.93}{\pi}\sqrt{\frac{235}{206{,}000}} = 0.827$$

Class b, so, the coefficients are $a1 = 0.650$, $a2 = 0.965$, and $a3 = 0.300$.

Since $\lambda_n = 0.827 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(a_2 + a_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(a_2 + a_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 0.827^2}\left[\left(0.965 + 0.300 \times 0.827 + 0.827^2\right) - \sqrt{\left(0.965 + 0.300 \times 0.827 + 0.827^2\right)^2 - 4 \times 0.827^2}\right] = 0.708$$

The slenderness ratio of double angle on symmetry axis is $\lambda_{yz} = 130.7$. So,

$$\frac{\lambda}{\varepsilon_k} = \lambda_x = 130.7$$

$$\lambda_n = \frac{\lambda_{\text{xy}}}{\pi}\sqrt{\frac{f_y}{E}} = \frac{130.7}{\pi}\sqrt{\frac{235}{206{,}000}} = 1.405$$

Class b, so, the coefficients are $a1 = 0.650$, $a2 = 0.965$, and $a3 = 0.300$.

Since $\lambda_n = 1.405 > 0.215$, then the stability factor in xy:

$$\phi_{yz} = \frac{1}{2\lambda_n^2}\left[\left(a_2 + a_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(a_2 + a_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 1.405^2}\left[\left(0.965 + 0.300 \times 1.405 + 1.405^2\right) - \sqrt{\left(0.965 + 0.300 \times 1.405 + 1.405^2\right)^2 - 4 \times 1.405^2}\right] = 0.384$$

The minimum stability factor is 0.384.

Compression Flange Width to Thickness Ratio

From GB50017-2017 Clause 7.3.1-5, when $\lambda = 80\varepsilon_k$, the formula 7.3.1-7 should be used to obtain the value of the width to thickness of a compression member. Here, $\lambda_{yz} = 130.7$ and $\varepsilon_k = 1$.

So, the limiting value of the width to thickness ratio = $5\varepsilon_k + 0.125\lambda = 5 + 0.125 \times 130.7 = 21.34$.

According to Clause 7.3.2 of GB50017-2017,

$$\phi A \times f = 0.384 \times 1{,}193 \times 215 = 98{,}494$$

$N = 92{,}061 < \phi A \times f$, so the magnification factor $a = \sqrt{\frac{98{,}494}{92{,}061}} = 1.034$

Therefore, Limit value of width to thickness ratio should be $1.034 \times 21.34 = 22.07$.

The width per formula 7.3.1-7 of GB50017-2017, $w = b - 2t = 100 - 2 \times 6 = 88\ mm$.

The actual width to thickness ratio is $\frac{88}{6} = 14.67 < 22.07$

So the ratio is $\frac{14.67}{22.07} = 0.66$.

Since the section is a equal leg angle, the check for the height to thickness ratio is the same; therefore ok.

Stability of Compression Member

According to formula 7.6.1-2 of GB50017-2017,

$$\eta = 0.6 + 0.0015\lambda = 0.6 + 0.0015 \times 130.7 = 0.796$$

The reduction factor of stability bearing capacity, ρ_e must be calculated as $\frac{w}{t} = 14.67 > 14\varepsilon_k = 14$.

$$\rho_e = 1.3 - \frac{0.3w}{14t\varepsilon_k} = 1.3 - \frac{0.3 \times 88}{14 \times 6 \times 1} = 0.986$$

The member capacity according to formula 7.6.1-1 of GB50017-2017:

$$\rho_e \eta \phi A f = 0.986 \times 0.796 \times 0.384 \times 1{,}193 \times 215 \times 10^{-3} = 77.30\ kN$$

So the ratio is $\frac{92.06}{77.30} = 1.19 > 1.0$; Failed.

Comparison

Table 631: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Compression Slenderness	0.87	0.87	none	
Tension Slenderness	0.44	0.44	none	
Truss Strength	0.50	0.50	none	

Result Type	Reference	STAAD.Pro	Difference	Comment
Compression Flange Slenderness	0.66	0.66	none	
Compression Web Slenderness	0.66	0.66	none	
Compression Stability	1.19	1.19	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\China\GB500017-2017 Single Angle section with Axial Force.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1.66667 0 0; 3 3.33333 0 0; 4 5 0 0; 5 6.66667 0 0; 6 8.33333 0 0;
7 10 0 0; 8 1.66667 2.5 0; 9 3.33333 2.5 0; 10 5 2.5 0; 11 6.66667 2.5 0;
12 8.33333 2.5 0; 13 0 0 2; 14 1.66667 0 2; 15 3.33333 0 2; 16 5 0 2;
17 6.66667 0 2; 18 8.33333 0 2; 19 10 0 2; 20 1.66667 2.5 2; 21 3.33333 2.5 2;
22 5 2.5 2; 23 6.66667 2.5 2; 24 8.33333 2.5 2;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 8 9; 8 9 10; 9 10 11; 10 11 12;
11 1 8; 12 8 3; 13 3 10; 14 10 5; 15 5 12; 16 12 7; 17 2 8; 18 4 10; 19 6 12;
20 3 9; 21 5 11; 22 13 14; 23 14 15; 24 15 16; 25 16 17; 26 17 18; 27 18 19;
28 20 21; 29 21 22; 30 22 23; 31 23 24; 32 13 20; 33 20 15; 34 15 22; 35 22
17;
36 17 24; 37 24 19; 38 14 20; 39 16 22; 40 18 24; 41 15 21; 42 17 23; 43 1 13;
44 2 14; 45 3 15; 46 4 16; 47 5 17; 48 6 18; 49 7 19; 50 8 20; 51 9 21;
52 10 22; 53 11 23; 54 12 24;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TO 10 22 TO 31 43 TO 54 TABLE ST PIP152X8.0
12 15 17 TO 21 33 36 38 TO 42 TABLE ST L80X80X6
11 16 32 37 TABLE SD L100X100X7
13 14 34 35 TABLE ST L100X100X6
CONSTANTS
MATERIAL STEEL ALL
MEMBER TRUSS
```

```
11 TO 21 32 TO 42
SUPPORTS
1 7 13 19 PINNED
LOAD 1 LOADTYPE None  TITLE DL
JOINT LOAD
2 TO 6 8 TO 12 14 TO 18 20 TO 24 FY -20
8 TO 12 20 TO 24 FY -40
LOAD 2 LOADTYPE None  TITLE LL
JOINT LOAD
8 TO 12 20 TO 24 FY -30
LOAD COMB 3 F : 1.20DL
1 1.2
LOAD COMB 4 F : 1.20DL+1.40LL
1 1.2 2 1.4
LOAD COMB 5 F : 1.00DL
1 1.0
LOAD COMB 6 F : 1.00DL+1.40LL
1 1.0 2 1.4
LOAD COMB 7 F : 1.20DL+0.98LL
1 1.2 2 0.98
LOAD COMB 8 F : 1.00DL+0.98LL
1 1.0 2 0.98
LOAD COMB 9 F : 1.35DL
1 1.35
LOAD COMB 10 F : 1.35DL+0.98LL
1 1.35 2 0.98
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
```

```
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.

[GROUP=1]
Name(Parameter Name)=TRUSS WEB MEMBER
Type(Member Type)=2
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=ALL
CheckDispLoadCase(Displacement Loads Case No.)=ALL
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=0
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=400
DJ1(Start Node Number in Major Axis)=0
```

```
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=0.8
miuy(Effective Length Factor for Column in Minor Axis)=1
Lateral(Member in Frame Without Sidesway or not)=1
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
rFlange(Limit Ratio of Width to Thickness for Flange)=0
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=1
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=0.85
Members(Member Number)=34
```

Chinese Steel Design Workflow Report

Member Check

Member No.	34

Member Type: Truss Member
Project Name: GB500017-2017 Single Angle section with Axial Force
Code for design of steel structure: GB50017-2017

Section

Section Name L100X100X6

Steel No.

Material Name Q235

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc
1	Compression Stability	168.7	200.8	1.19	1.19		4
2	Compression Slenderness	150	130.7	0.87	0.87		1
3	Compression Web Slenderness	22.1	14.7	0.66	0.66		4
4	Compression Flange Slenderness	22.2	14.7	0.66	0.66		4
5	Truss Strength	182.8	90.8	0.5	0.5		4
6	Tension Slenderness	300	130.7	0.44	0.44		1
Max(Ratio/Failure ratio)		**1.19**					

V. GB500017-2017 Tube section with Combined Axial and Bending

Verify the strength, stability, and slenderness of a tube section subject to combined axial and bending per GB50017-2017.

Reference

MOHURD. 2017. *GB 50017-2017 Standard for design of steel structures* . Beijing, China: Ministry of Housing and Urban-Rural Development

Problem

The section is a TUB 300x10.0 with a length of 4 m. The structure is a portal frame. Member #1 assigned with a tube section (TUB300x10.0) is designed per GB 50017-2017. The section has an unbraced length of 4 m in either direction. The governing load case #1 has the following ultimate loads on the member:

$F_x = 30.55\ kN$
$F_y = 25.06\ kN$
$N = 66.69\ kN$
$M_x = 83.44\ kN{\cdot}m$ (at member end)
$M_y = 122.2\ kN{\cdot}m$ (at member end)

The material is Q235 type steel.

Design strength in tension, compression, and flexure: $f_y = 215\ MPa$

Design strength in shear: $f_v = 125\ MPa$

Calculations

Section Properties

Section depth, $H = 300\ mm$
Section width, $D = 300\ mm$
Wall thickness, $t = 10\ mm$
Cross-sectional area, $A = 11{,}260\ mm^2$
Moment of inertia about x, $I_x = I_y = 155{,}200{,}00\ mm^4$
Radius of gyration about x, $r_x = r_y = 118.2\ mm$
Section modulus about x, $W_x = W_y = 1{,}035{,}000\ mm^3$

Slenderness Ratio

The effective length is:

$$l_{ox} = \mu_x \times l_x = 1.402 \times 4{,}000 \text{ mm} = 5{,}188 \text{ mm}$$

$$l_{oy} = \mu_y \times l_y = 2.0383 \times 4{,}000 \text{ mm} = 8{,}150 \text{ mm}$$

The slenderness ratio is:

$$\lambda_x = \frac{l_{0x}}{i_x} = \frac{\mu_x l}{i_x} = \frac{1.402 \times 4{,}000}{117.4} = 47.77 < 150$$

$$\lambda_y = \frac{l_{0y}}{i_y} = \frac{\mu_y l}{i_y} = \frac{2.038 \times 4{,}000}{117.4} = 69.44 < 150$$

According to table 7.4.6 of *Standard for design of steel structures*, the allowable slenderness ratio of compression members is $\lambda_{clim} = 150$.

Ratio: $\lambda_{max} / \lambda_{clim} = 69.44 / 150 = 0.46$

According to table 7.4.7 of *Standard for design of steel structures*, the allowable slenderness ratio of tension members is $\lambda_{tlim} = 300$.

Ratio: $\lambda_{max} / \lambda_{tlim} = 69.44 / 300 = 0.23$

Steel Grade Correction Factor

The steel type is Q235. According to note 1 in table 3.5.1 of *Standard for design of steel structures* and table 1 of *Standard for design of steel structures. Commentary 2.2*, the steel grade correction coefficient is $\varepsilon_k = 1$.

Check Flange Width to Thickness Ratio of Compression Member

$$\frac{B - 2t}{t} = \frac{300 - 2 \times 10}{10} = 28$$

According to table 3.5.1, note 2 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3, so the limit value of height thickness ratio is:

$$40\varepsilon_k = 40$$

Ratio: $\frac{28}{40} = 0.70$

Note: Since the section is a square tube, the web heigh to flange thickness check is the same.

Plastic Development Coefficient

According to clause 8.1.1 of *Standard for design of steel structures*, and the width thickness ratio of cross-section plate is grade S3,

$$\gamma_x = 1.05$$

$$\gamma_y = 1.05$$

Check Member Strength

According to clause 8.1.1 of *Standard for design of steel structures*,

$$\sigma = \frac{N}{A_n} + \frac{M_x}{\gamma_x W_{nx}} + \frac{M_y}{\gamma_y W_{ny}} = \frac{N}{A} + \frac{M_x}{\gamma_x W_x} + \frac{M_y}{\gamma_y W_y}$$

$$= \frac{66.69\times10^3}{11{,}260} + \frac{83.44\times10^6}{1.05\times1{,}034{,}600} + \frac{122.2\times10^6}{1.05\times1{,}034{,}600} = 195.2 \text{ N/mm}^2$$

Ratio: $\frac{\sigma}{f} = \frac{195.2}{215} = 0.91$

Check In-Plane Stability of Member According to table 7.2.1-1 of *Standard for design of steel structures*, the section is “b” for this section.

According to the formula (D.0.5) in Appendix D of *Standard for design of steel structures*, for the x-x direction:

$$\lambda_n = \frac{\lambda_x}{\pi}\sqrt{\frac{f_y}{E}} = \frac{47.77}{\pi}\sqrt{\frac{235}{206{,}000}} = 0.514$$

Class b, so, the coefficients are $\alpha 1 = 0.650$, $\alpha 2 = 0.965$, and $\alpha 3 = 0.300$.

Since $\lambda_n = 0.514 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 0.514^2}\left[\left(0.965 + 0.300 \times 0.514 + 0.514^2\right) - \sqrt{\left(0.965 + 0.300 \times 0.514 + 0.514^2\right)^2 - 4 \times 0.514^2}\right] = 0.866$$

According to the formula 8.2.1-8 of *Standard for design of steel structures*,

$$N_{cr} = \frac{\pi^2 EI}{(\mu l)^2} = \frac{\pi^2 EI_x}{(\mu_x l)^2}$$

$$= \frac{\pi^2 \times 206{,}000 \times 155{,}190{,}000}{(1.402 \times 4{,}000)} = 10{,}033 \text{ kN}$$

According to the formula 8.2.1-10 of *Standard for design of steel structures*,

$$\beta_{mx} = 1 - 0.36N / N_{cr} = 1 - 0.36 \times 66.69 / (10{,}033) = 0.998$$

According to the formula 8.2.1-12 and 8.2.1-5 note of *Standard for design of steel structures*,

$$\beta_{ty} = 0.65 + 0.35 \times \frac{M_2}{M_1} = 0.65 + 0.35 \times \frac{6{,}000}{122{,}200{,}000} = 0.65$$

According to the formula 8.2.1-12 of *Standard for design of steel structures*,

$$N'_{Ex} = \frac{\pi^2 EA}{1.1\lambda_x^2} = \frac{\pi^2 \times 206{,}000 \times 11{,}260}{1.1 \times (47.77)^2} = 9{,}121 \text{ kN}$$

According to the formula 8.2.5, $\phi_{by} = 1.0$.

According to the formula 8.2.1, $\eta = 0.7$.

According to the formula 8.2.5-1,

$$\frac{N}{\phi_x Af} + \frac{\beta_{mx} M_x}{\gamma_x W_x \left(1 - 0.8\frac{N}{N'_{Ex}}\right) f} + \eta \frac{\beta_{ty} M_y}{\phi_{by} W_y f}$$

$$\frac{66.69 \times 10^3}{0.866 \times 11{,}260215} + \frac{0.998 \times 83.44 \times 10^6}{1.05 \times 1{,}034{,}600\left(1 - 0.8 \times \frac{66.69}{9{,}121}\right)215} + 0.7\frac{0.65 \times 122.2 \times 10^6}{1.0 \times 1{,}034{,}600 \times 215} = 0.64$$

Check Out-of-Plane Stability of Member According to table 7.2.1-1 of *Standard for design of steel structures*, the section is "b" for this section.

According to the formula (D.0.5) in Appendix D of *Standard for design of steel structures*, for the y-y direction:

$$\lambda_n = \frac{\lambda_y}{\pi}\sqrt{\frac{f_y}{E}} = \frac{69.44}{\pi}\sqrt{\frac{235}{206{,}000}} = 0.747$$

Class b, so, the coefficients are $\alpha 1 = 0.650$, $\alpha 2 = 0.965$, and $\alpha 3 = 0.300$.

Since $\lambda_n = 0.747 > 0.215$, then the stability factor in x:

$$\phi_x = \frac{1}{2\lambda_n^2}\left[\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right) - \sqrt{\left(\alpha_2 + \alpha_3\lambda_n + \lambda_n^2\right)^2 - 4\lambda_n^2}\right]$$

$$= \frac{1}{2 \times 0.747^2}\left[\left(0.965 + 0.300 \times 0.747 + 0.747^2\right) - \sqrt{\left(0.965 + 0.300 \times 0.747 + 0.747^2\right)^2 - 4 \times 0.747^2}\right] = 0.754$$

According to the formula 8.2.1-8 of *Standard for design of steel structures*,

$$N_{cr} = \frac{\pi^2 EI}{(\mu l)^2} = \frac{\pi^2 EI_y}{(\mu_y l)^2}$$

$$= \frac{\pi^2 \times 206{,}000 \times 155{,}190{,}000}{(2.038 \times 4{,}000)} = 4{,}748 \text{ kN}$$

According to the formula 8.2.1-10 of *Standard for design of steel structures*,

$$\beta_{my} = 1 - 0.36N / N_{cr} = 1 - 0.36 \times 66.69 / (4{,}748) = 0.995$$

According to the formula 8.2.1-12 and 8.2.1-5 note of *Standard for design of steel structures*,

$$\beta_{tx} = 1.0$$

According to the formula 8.2.1-12 of *Standard for design of steel structures*,

$$N'_{Ey} = \frac{\pi^2 EA}{1.1\lambda_y^2} = \frac{\pi^2 \times 206{,}000 \times 11{,}260}{1.1 \times (69.44)^2} = 4{,}316 \text{ kN}$$

According to the formula 8.2.5, $\phi_{bx} = 1.0$.

According to the formula 8.2.1, $\eta = 0.7$.

According to the formula 8.2.5-2,

$$\frac{N}{\phi_x A f} + \eta \frac{\beta_{tx} M_x}{\phi_{\mathrm{bx}} W_x f} + \frac{\beta_{my} M_y}{\gamma_y W_y \left(1 - 0.8 \frac{N}{N'_{Ey}}\right) f}$$

$$\frac{66.69 \times 10^3}{0.754 \times 11{,}260215} + 0.7 \frac{1.0 \times 83.44 \times 10^6}{1.0 \times 1{,}034{,}600 \times 215} + \frac{0.995 \times 122.2 \times 10^6}{1.05 \times 1{,}034{,}600 \left(1 - 0.8 \times \frac{66.69}{4{,}316}\right) 215} = 0.83$$

<u>Shear Strength</u>

Take the neutral axis as the calculation point of shear stress, calculate the area moment:

$$\mathrm{S} = 417{,}605 \text{ mm}^3$$

According to clause 6.1.3 of *Standard for design of steel structures*, shear stress

$$\tau_x = \frac{VS}{I t_w} = \frac{V S_x}{I_y (2t)} = \frac{30.55 \times 10^3 \times 631{,}000}{155{,}190{,}000 \times 2 \times 10} = 6.21 \text{ N/mm}^2$$

$$\tau_y = \frac{VS}{I t_w} = \frac{V S_x}{I_x (2t)} = \frac{25.06 \times 10^3 \times 631{,}000}{155{,}190{,}000 \times 2 \times 10} = 5.09 \text{ N/mm}^2$$

$$\tau_{max} = 7.69 < f_v = 125 \text{ N/mm}^2$$

Therefore, the ratio is:

$$\frac{\tau_{max}}{f_v} = \frac{6.21}{125} = 0.05$$

Comparison

Table 632: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comment
Compression Slenderness	0.46	0.46	none	
Tension Slenderness	0.23	0.23	none	
Flange width to thickness ratio	0.70	0.70	none	
Web height to thickness ratio	0.70	0.70	none	
Column Strength	0.91	0.91	none	
In-plane stability	0.64	0.64	none	
Out-plane stability	0.83	0.83	none	
Shear Strength	0.05	0.05	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\China\GB500017-2017 Tube section with Combined Axial and Bending.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 6 4 0; 4 6 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY CHINESE
1 TABLE ST TUB30030010.0
3 TABLE ST PIP299X10.0
2 TABLE ST HN300X150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -10
JOINT LOAD
2 3 FX 30 FY -50 FZ 30
PERFORM ANALYSIS
FINISH
```

Chinese steel design parameters (`.gsp` file):

```
[[version=2207]
*{ The below data is for code check general information, please do not modify
it.
[CodeCheck]
SeismicGrade=None
BeamBendingStrength=1
BeamShearStrength=1
BeamEquivalentStress=1
BeamOverallStability=1
BeamSlendernessWeb=1
BeamSlendernessFlange=1
TrussStrength=1
TrussStability=1
TrussShearStrength=1
ColumnStrength=1
```

```
ColumnStabilityMzMy=1
ColumnStabilityMyMz=1
PressedTrussSlenderness=1
TensionTrussSlenderness=1
ColumnSlendernessFlange=1
ColumnSlendernessWeb=1
BeamDeflection=1
SelectAll=0
GroupOptimize=0
FastOptimize=0
Iteration=0
SecondaryMembers=
SectCollectionOrder=0
[CheckOptionAngle]
PrimaryAxis=60.000000
SecondaryAxis=60.000000
ExtendLine=10.000000
*{ The above data is for code check general information, please do not modify
it.

[GROUP=1]
Name(Parameter Name)=TUBE
Type(Member Type)=3
Principle(Principle Rules)=0
SteelNo()=Q235
SectionSlendernessRatioGrade(Section Slenderness Ratio Grade)=3
Fatigue(Fatigue Calculation)=0
Optimization(Perform optimized design)=0
MaxFailure(Failure Ratio)=1
MinTooSafe(Safety Ratio)=0.3
CheckLoadCase(Force Loads Case No.)=1
CheckDispLoadCase(Displacement Loads Case No.)=1
BeamBendingStrength()=1
BeamShearStrength()=1
BeamEquivalentStress()=1
BeamOverallStability()=1
BeamSlendernessFlange(b/t on beam)=1
BeamSlendernessWeb(h0/tw on beam)=1
TrussStrength(Axial Force Strength)=1
SecondaryMoment(Secondary Moment of Truss)=0
TrussStability(Solid-web Axial Compression Stability)=1
TrussShearStrength(Axial Shear Strength)=1
PressedTrussSlenderness(Pressed Member Slenderness)=1
TensionTrussSlenderness(Tension Member Slenderness)=1
ColumnStrength(Column Member Strength)=1
ColumnStabilityMzMy(Column Stability In-plane)=1
ColumnStabilityMyMz(Column Stability Out-plane)=1
ColumnSlendernessFlange(b/t on column)=1
ColumnSlendernessWeb(h0/tw on column)=1
CheckItemAPPENDIX_B11(Beam Deflection)=1
UseAntiSeismic(Use Seismic Adjusting Factor)=0
GamaReStr(Seismic Adjusting Factor of Load-bearing Capacity for Strength)=0
GamaReSta(Seismic Adjusting Factor of Load-bearing Capacity for Stability)=0
SLevel(Grade of Seismic Resistance)=0
lmdc(Slenderness Limit of Compression Member)=0
lmdt(Slenderness Limit of Tension Member)=0
Lmd831(Slenderness of Seismic Column)=0
Lmd841(Slenderness of Seismic Brace)=0
```

```
Lmd9213(Slenderness of Seismic Single-story Plant)=0
LmdH28(Slenderness of Seismic Multi-story Plant)=0
rz(Plastic Development Factor in Major Axis)=0
ry(Plastic Development Factor in Minor Axis)=0
gamaSharp(Plastic Development Factor of sharp side)=0
betamz(the equivalent moment factor in Major Axis plane)=0
betamy(the equivalent moment factor in Minor Axis plane)=0
betatz(the equivalent moment factor out Major Axis plane)=1
betaty(the equivalent moment factor out Minor Axis plane)=0
HasHorLoadZ(Has Horizontal Load in Z-Axis)=0
HasHorLoadY(Has Horizontal Load in Y-Axis)=0
DFF(Deflection Limit of Beam)=150
DJ1(Start Node Number in Major Axis)=0
DJ2(End Node Number in Major Axis)=0
Horizontal(Check for Deflection in Minor Axis)=0
Cantilever(Cantilever Member)=0
fabz(Overall Stability Factor in Major Axis of Bending Member)=0
faby(Overall Stability Factor in Minor Axis of Bending Member)=0
StressFeature(Select the Stress Feature to calulate stability factor of
beam)=1
faz(Overall Stability Factor in Major Axis of Axial Compression Member=0
fay(Overall Stability Factor in Minor Axis of Axial Compression Member)=0
lz(Unbraced Length in Major Axis)=0
ly(Unbraced Length in Minor Axis)=0
miuz(Effective Length Factor for Column in Major Axis)=0
miuy(Effective Length Factor for Column in Minor Axis)=0
Lateral(Member in Frame Without Sidesway or not)=0
APZ(Gyration Radius Calculation as Z-Axis Parallel Leg)=0
rFlange(Limit Ratio of Width to Thickness for Flange)=0
rWeb(Limit Ratio of High to Thickness for Web)=0
BucklingStrength(Axis forced member bulking strength)=0
ZSectType(Section Type in Z-Axis)=0
YSectType(Section Type in Y-Axis)=0
HSectWebInTrussPlane(Web of H in Truss Plane)=0
rAn(Net Factor of Section Area)=1
rWnz(Net Factor of Resistance Moment in Z-Axis)=1
rWny(Net Factor of Resistance Moment in Y-Axis)=1
CapReduce(Seismic Reduction Factor of Load-bearing Capacity for Brace)=1
AngleReduce(Angle Strength Reduce)=0
LAglConSta(Connect Type of unequal single angle)=0
LAngleStrength(Reduction Factor of Angle Strength)=0
LAngleStability(Reduction Factor of Angle Stability)=0
rTrussSectReduce(Effective Factor of Axial Force Section)=1
Members(Member Number)=1
```

Chinese Steel Design Workflow Report

Member Check

Member No.	1

Member Type: Column Member
Project Name: GB500017-2017 Tube section with Combined Axial and Bending
Code for design of steel structure: GB50017-2017

Section

Section Name	TUB30030010.0

Steel No.

Material Name	Q235

Results

No.	Item	Limit v.	Actual v.	Ratio	R./Failure R.	Results	Lc
1	Column Strength	215	195.3	0.91	0.91		1
2	Stability Out-plane	215	177.7	0.83	0.83		1
3	Compression Flange Slenderness	40	28	0.7	0.7		1
4	Compression Web Slenderness	40	28	0.7	0.7		1
5	Stability In-plane	215	137.7	0.64	0.64		1
6	Compression Slenderness	150	69.4	0.46	0.46		1
7	Tension Slenderness	300	69.4	0.23	0.23		1
8	Shear Strength	125	6.2	0.05	0.05		1
Max(Ratio/Failure ratio)		**0.908**					

V. Europe

V. EC3 - Pinned column using non-slender UKC section

Verify the design of a pinned column per the EC3 2005 design code.

Details

The column is 8 m with pinned ends. The member is a 305 x 305 x 97 UKC shape with S275 grade steel. The axial load is 2,000 kN and $L_{cry}/L = 0.7$.

Partial safety factors:

$$\Gamma_{M0} = 1.0$$

$$\Gamma_{M1} = 1.0$$

Validation

<u>Effective Length Calculations</u>

$$L_{cry} = 0.7 \times 8{,}000 = 5{,}600 \text{ mm}$$

$L_{crz} = 1.0 \times 8{,}000 = 8{,}000$ mm

Yield Strength

Steel grade S275

The maximum thickness = *15.4 mm < 40 mm*; therefore *fy = 275 N/mm² [EN 1993-1-1:2005 Table 3.1]*

Section Classification

$\varepsilon = \sqrt{(235/fy)} = 0.92$ [EN 1993-1-1:2005 Table 5.2]

Outstanding flange (flange under uniform compression):

$c = (b-t_w-2r)/2 = (305.3 - 9.9 - 2 \times 15.2)/2 = 132.3$ mm

'

$c/t_f = 132.3/15.4 = 8.6 < 10\ \varepsilon$

Class 2 element *[EN 1993-1-1:2005 Table 5.2 Sheet 2]*

Internal compression part (web in bending):

$c = h-2t_f-2r = 307.9-2 \times 15.4 - 2 \times 15.2 = 246.7$ mm

$c/t_w = 246.7/9.9 = 24.92 < 33\ \varepsilon$

Class 1 element *[EN 1993-1-1:2005 Table 5.2 Sheet 1]*

Therefore, section is a Class 2 section.

Axial Resistance

Design axial resistance:

$N_{c,Rd} = A \times fy/\Gamma_{m0} = 123.4\ cm^2(10^2) \times 275\ kN/mm^2 / 1.0 \times 10^{-3} = 2{,}283\ kN > 2{,}000\ kN$ [EN 1993-1-1:2005 Cl. 6.2.4]

Lateral Buckling

Determine the elastic critical moment, M_{cr}:

$$N_{bRd} = \frac{\chi A fy}{\Gamma_{M1}}$$ [EN 1993-1-1:2005 Cl. 6.3.1.1]

$N_{cr} = (\pi^2 \times E \times Iy)/Ly^2 = [\pi^2 \times 205{,}000 \times 7{,}308(10^4)] / (56{,}002 \times 1{,}000) = 4715$ kN

$\lambda = (A \times fy/Ncr)^{0.5} = (123 \times 10^2 \times 275 / 4{,}715 \times 10^3)^{0.5} = 0.847$

$h/b = 307.9/305.3 = 1 < 1.2\ \&\ t_f < 100$ mm

$\alpha = 0.49$ [EN 1993-1-1:2005 Table 6.3]

$\phi = 0.5\,[1 + \alpha(\lambda - 0.2) + \lambda^2]$ [EN 1993-1-1:2005 Cl. 6.3.2.3]

$\phi = 0.5\,[1 + 0.49(0.847 - 0.2) + 0.847^2] = 1.017$

$$\chi = \frac{1}{\varnothing + \sqrt{(\varnothing^2 - \lambda^2)}} = \frac{1}{1.017 + \sqrt{(1.017^2 - 0.847^2)}} = 0.639 < 1.0$$

$$N_{bRd} = \frac{0.639 \times 123 \times 10^2 275 \times 10^{-3}}{1.0} = 2{,}161\ kN$$

Critical Ratio

$P / N_{bRd} = 2{,}000 / 2{,}161 = 0.925$

Results

Table 633: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Critical Ratio	0.925	0.925	none	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 - Pinned column using non-slender UKC section.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 04-Jan-08
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 8 0;
MEMBER INCIDENCES
1 2 1;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST UC305X305X97
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -2000
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
SGR 1 ALL
LY 5.6 ALL
GM1 1 ALL
GM0 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                  -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST  UC305X305X97 (BRITISH SECTIONS)
                            PASS     EC-6.3.1.1         0.925         1
                      2000.00 C         0.00           0.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    800.00
      Gross Area =  123.00          Net Area =  123.00
                                      z-axis          y-axis
      Moment of inertia        :    22200.002       7310.001
      Plastic modulus          :     1590.000        726.000
      Elastic modulus          :     1442.027        478.873
      Shear Area               :       84.627         35.174
      Radius of gyration       :       13.435          7.709
      Effective Length         :      800.000        560.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 2
      Squash Load              :   3382.50
      Axial force/Squash load  :     0.591
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         59.5            72.6
      Compression Capacity     :       2677.4          2161.6
      Tension Capacity         :       3382.5          3382.5
      Moment Capacity          :        437.3           199.7
      Reduced Moment Capacity  :        202.6           156.4
      Shear Capacity           :       1343.6           558.5
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  307.5
      co-efficients C1 & K : C1 =1.132 K =1.0, Effective Length= 8.000
      Lateral Torsional Buckling Curve : CURVE b
      Elastic Critical Moment for LTB,              Mcr   =   442.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  5058.6
      Critical Load For Torsional-Flexural Buckling, NcrTF =  5058.6
      STAAD PLANE                                                  -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.3.1.1     0.925     1  2000.0      0.0     0.0     0.0     0.0
    Torsion has not been considered in the design.
```

```
************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.C.5 Member Design* (on page 1480)

V. EC3 - Pipe Section with Conc Load

Verify the adequacy of a circular hollow section used as a beam subject to concentrated loads per EN 1993-1-1:2005 (no national annex used).

Details

The member is a 6 m long beam with pinned ends. The beam supports 10kN concentrated loads at the third points. The steel is grade S275. The section is a European 114.3X8CHS.

Section Properties

A = 26.7 cm^2
Outer diameter = 114.3 mm
t = 8 mm
$I_z = I_y = 379\ cm^4$
$Z_z = Z_y = 90.6\ cm^3$
$C_w = 0\ cm^6$

$$r_z = r_y = \sqrt{I/A} = \sqrt{\frac{379}{26.7}} = 3.768\ \ cm$$

Partial safety factors:

$$\Gamma_{M0} = 1.0$$

$$\Gamma_{M1} = 1.0$$

$$\Gamma_{M2} = 1.25$$

Validation

Section Classification

$$\varepsilon = \sqrt{\frac{235}{f_y}} = \sqrt{\frac{235}{275}} = 0.924$$

As per Table 5.2:

$$d / t = 114.3 / 8 = 14.3 < 50\times\varepsilon^2 = 50\ (0.924)^2 = 42.7$$

Hence, this is a Class 1 section.

Slenderness Ratio

The slenderness ratio = $kL\ /\ r = 1 \times 6{,}000\ /\ 37.68 = 159.3$

Axial Tension

Determine axial tension capacity per Cl. 6.2.3.

$$N_{\mathrm{pl,Rd}} = A_g \times f_y / \gamma_{\mathrm{M0}} = 2{,}670 \times 275 / 1.0 \times (10)^{-3} = 734.3\ \mathrm{kN}$$

$$N_{u,Rd} = 0.9A_{net} \times f_u / \gamma_{M2} = 0.9 \times 2{,}670 \times 295 / 1.25 \times (10)^{-3} = 567.1 \text{ kN}$$

The tensile capacity is the minimum of $N_{pl,Rd}$ and $N_{u,Rd}$, thus: $N_{t,Rd} = 567.1\ kN$.

No axial tension in section, so by observation no need to check ratio.

<u>Axial Compression</u>

Determine axial compression capacity per Cl. 6.2.4 for a Class 1 section.

$$N_{c,Rd} = A_g \times f_y / \gamma_{M0} = 2{,}670 \times 275 / 1.0 \times (10)^{-3} = 734.3 \text{ kN}$$

Next, check the flexural buckling resistance per Cl. 6.3.1.3:

$$N_{b,Rd} = \chi \times A \times f_y / \gamma_{M1}$$

From Table 6.1: the imperfection factor, $a = 0.21$.

$$\bar{\lambda} = \sqrt{A \times f_y / N_{cr}} = L_{cr} / i \times \lambda_1$$

where

L_{cr}	=	6,000 mm
i	=	37.68 mm
λ_1	=	$93.9 \times \varepsilon = 93.9 \times 0.924 = 86.8$

$$\bar{\lambda} = 1.834$$

$$\Phi = 0.5\left[1 + a\left(\bar{\lambda} - 0.2\right) + \bar{\lambda}^2\right] = 0.5\left[1 + 0.21\left(1.834 - 0.2\right) + (1.834)^2\right] = 2.354$$

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}} = 0.2611$$

$$N_{b,Rd} = 0.2611 \times 2{,}670 \times 275 / 1.0 = 191.7 \text{ kN}$$

The compression capacity is the minimum of $N_{c,Rd}$ and $N_{b,Rd}$, thus: $N_{c,Rd} = 191.7\ kN$.

No axial compression in section, so by observation no need to check ratio.

<u>Bending Capacity</u>

Maximum bending moment in the section (center third of span): $M_{Ed} = 20.0\ kN{\cdot}m$.

Check for bending capacity per Cl. 6.2.5:

For a Class 1 section:

$$M_{c,Rd} = W_{ply} \times f_y / \gamma_{M0} = 90.6 \times 275 / 1.0 \times (10)^{-3} = 24.92 \text{ kN}{\cdot}\text{m}$$

Ratio per Eq. 6.12: $M_{Ed}\ /\ M_{c,Rd} = 20.0\ /\ 24.92 = 0.803$

<u>Shear Capacity</u>

Maximum shear in the section: $V_{Ed} = 10\ kN$

$$A_v = 2A / \pi = 1{,}700 \text{ mm}^2$$

Check for shear capacity per Cl. 6.2.6 for plastic design (Class 1):

$$V_{c,Rd} = V_{pl,Rd} = A_v \times \left(f_y / \sqrt{3} \right) / \gamma_{M0} = 1{,}700 \times \left(275 / \sqrt{3} \right) / 1.0 = 269.9 \text{ kN}$$

Ratio per Eq. 6.17: $V_{Ed}\ /\ V_{c,Rd} = 10.0\ /\ 26.9 = 0.037$

<u>Lateral Torsional Buckling</u>

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 GI_t}{\pi^2 EI_z} + (C_2 Z_g)^2} - C_2 Z_g\right]$$

where

C_1	=	1.0
C_2	=	1.0
$\frac{\pi^2 EI_y}{(kL)^2}$	=	$213,000$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	0
$\frac{(kL)^2 GI_T}{\pi^2 EI_y}$	=	$2,806,000$
$C_2 Z_g$	=	$1.0 \times 114.3 / 2 = 57.15$

Therefore, $M_{cr} = 1.0 \times 213,000\left[\sqrt{0 + 2,806,000 + 57.15^2} - 57.15\right] = 344.8\ kN \cdot m$

For CHS sections, $\chi_{LT} = 1.0$.

$$M_{b,Rd} = \chi_{LT} W_y \times f_y / \gamma_{M1} = 1.0 \times 90.6 \times 275 / 1.0 \times (10)^{-3} = 24.92\ \mathrm{kN \cdot m}$$

Ratio per Eq. 6.12: $M_{Ed} / M_{b,Rd} = 20.0 / 24.92 = 0.803$

Results

Table 634: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Section Class	Class 1	Class 1	none	
Slenderness Ratio	159.3	159.3	none	
Tension Capacity (kN)	567.1	567.1	none	
Compression Capacity (kN)	191.7	191.7	none	
Moment Capacity (kN·m)	24.92	24.9	negligible	
Shear Area (cm^2)	17.0	16.99	negligible	
Shear Capacity (kN)	269.9	269.9	none	
M_B (kN·m)	24.92	24.9	negligible	
Ratio per Cl. 6.2.5	0.803	0.803	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio per Cl. 6.2.6 (Y)	0.037	0.037	none	
Ratio per Cl. 6.3.2 LTB	0.803	0.803	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 - Pipe Section with Conc Load.std` is typically installed with the program.

The following design parameters are used:

- Symmetric point loads about mid-span: `CMM 5`
- Use Cl. 6.3.2.2 to determine χ_{LT}: `MTH 1`
- The values of `C1 1.0` and `C2 1.0` are specified.
- The values of F_y and F_u are specified directly using `PY 275000` and `FU 295000`.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 07-May-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST 114.3X8CHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10 2
1 CON GY -10 4
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
C1 1 ALL
C2 1 ALL
CMM 5 ALL
MTH 1 ALL
```

```
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   114.3X8CHS  (EUROPEAN SECTIONS)
                           PASS     EC-6.2.5           0.803         1
                        0.00           0.00          20.00        2.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =   26.70          Net Area =   26.70
                                      z-axis          y-axis
      Moment of inertia        :      379.000         379.000
      Plastic modulus          :       90.600          90.600
      Elastic modulus          :       66.317          66.317
      Shear Area               :       16.998          16.998
      Radius of gyration       :        3.768           3.768
      Effective Length         :      600.000         600.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    734.25
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        159.3           159.3
      Compression Capacity     :        191.7           191.7
      Tension Capacity         :        567.1           567.1
      Moment Capacity          :         24.9            24.9
      Reduced Moment Capacity  :         24.9            24.9
      Shear Capacity           :        269.9           269.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =   24.9
      co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 6.000
      Lateral Torsional Buckling Curve :
      Compression buckling curves:     z-z:  Curve a   y-y:  Curve a
      STAAD PLANE                                              -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
```

```
 CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
EC-6.2.5       0.803     1     0.0     10.0     0.0    20.0     0.0
EC-6.2.6-(Y)   0.037     1     0.0     10.0     0.0     0.0     0.0
EC-6.3.2 LTB   0.803     1     0.0     10.0     0.0    20.0     0.0
 Torsion has not been considered in the design.

************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 - Simply supported laterally unrestrained beam

Verify the design of a pinned column per the EC3 2005 design code.

Details

The beam is a 5.7 m simply supported span. The member is a UKB 356 x171 x51shape with S275 grade steel. The member is uniformly loaded with 9.58 kN/m dead load and 6.25 kN/m live load.

Partial safety factors:

$\Gamma_{M0} = 1.0$

$\Gamma_{M1} = 1.0$

Validation

Factored Loads

$w = 1.35 (9.58) + 1.5 (6.25) = 22.31$ kN/m

$M_{z,Ed} = 22.31 (5.7)^2 / 8 = 90.6$ kN·m

$V_{z,Ed} = 22.31 (5.7)/2 = 63.58$ kN

Yield Strength

Steel grade S275

The maximum thickness = *15.4 mm < 40 mm*; therefore *fy = 275 N/mm²* *[EN 1993-1-1:2005 Table 3.1]*

Section Classification

$\varepsilon = \sqrt{(235/fy)} = 0.92$ [EN 1993-1-1:2005 Table 5.2]

Outstanding flange (flange under uniform compression):

$c = (b-t_w-2r)/2 = (171.5 – 7.4 – 2 \times 10.2)/2 = 71.85$ mm

$c /t_f = 71.85 / 11.5 = 6.25 < 9 \varepsilon$

Class 1 element *[EN 1993-1-1:2005 Table 5.2 Sheet 2]*

Internal compression part (web in bending):

$c = h-2t_f-2r = 355-2 \times 11.5 – 2 \times 10.2 = 311.6$ mm

$c / t_w = 311.6 / 7.4 = 42.1 < 72 \varepsilon$

Class 1 element *[EN 1993-1-1:2005 Table 5.2 Sheet 1]*

Therefore, section is a Class 2 section.

Moment Resistance

Design bending resistance:

$M_{c,\,Rd} = W_{pl} \times fy/\Gamma_{m0} = 896\ cm^3(10^3)\times 275\ kN/mm^2 / 1.0 \times 10^{-6} = 246.4\ kN\cdot m > 90.6\ kN\cdot m$ [EN 1993-1-1:2005 Cl. 6.2.4]

Ratio *90.6 / 246.4 = 0.368*

Lateral Torsional Buckling

Determine the elastic critical moment, M_{cr}:

$$M_{cr} = C1\frac{\pi^2 E\, I_y}{(kl)^2}\left\{\sqrt{\left[\left(\frac{k}{Kw}\right)^2\left(\frac{Iw}{Iy}\right) + \frac{(kl)^2 GIt}{\pi^2 E\, I_y} + (C2z_g)^2\right]} - C2z_g\right\}$$ [EN 1993-1-1:2005 Annex F]

where

k	=	$kw = 1$
z_g	=	h/2 = 177.5 mm
$C1$	=	1.132 [EN 1993-1-1:2005 Table F.1.2]
$C2$	=	0.459 [EN 1993-1-1:2005 Table F.1.2]

$M_{cr} = 121.74\ kN\cdot m$

$$\lambda_{LT} = \sqrt{W_{pLz} \times fy / M_{cr}} = \sqrt{\frac{896,\ 000 \times 275}{121.74(10)^6}} = 1.423$$

$h/b = 355 / 171.5 = 2.07 > 2$

Use buckling curve c. *[EN 1993-1-1:2005 Cl. 6.3.2.3 & Table 6.5]*

$\alpha_{LT} = 0.49$ [EN 1993-1-1:2005 Table 6.3]

$\phi = 0.5\ [1 + \alpha(\lambda_{LT} - \lambda_{LT0}) + \beta\lambda^2]$ [EN 1993-1-1:2005 Cl. 6.3.2.3]

$\phi = 0.5\ [1 + 0.49(1.423 - 0.4) + 0.75\times 1.423^2] = 1.510$

$$\chi_{LT} = \frac{1}{\varnothing_{LT} + \sqrt{(\varnothing_{LT}{}^2 - \lambda_{LT}{}^2)}} = \frac{1}{1.510 + \sqrt{(1.510^2 - 1.423^2)}} = 0.420 < 1.0$$

$\chi_{LT} = 0.420 < 0.494$OK

$f = 1 - 0.5\ (1 - kc)[1 - 2(\lambda_{LT} - 0.8)^2] = 0.99 < 1$

where

kc	=	0.94

$\chi_{LT} = \chi_{LT} / f = 0.42$

$M_{b,\,Rd} = \chi_{LT} W_{pl,z} \times fy/\Gamma_{m1} = 0.42\times 896\ cm^3(10^3)\times 275\ kN/mm^2 / 1.0 \times 10^{-6} = 103.5\ kN\cdot m > 90.6\ kN\cdot m$ [EN 1993-1-1:2005 Cl. 6.2.4]

Ratio = *90.6 / 103.5 = 0.875*

Check Shear Resistance

In the absence of torsion shear area

$A_v = A - 2bt_f + (t_w + 2r)\ t_f = 6490 - 2 \times 171.5\times 11.5 + (7.4+2 \times 10.2) \times 11. = 2,865\ mm^2$

$\eta\ h_w\ t_w = 1 \times 332 \times 7.4 = 2,457\ mm^2 > A_v$

Shear plastic resistance

$V_{pl,Rd} = A_v\ (fy/\sqrt{3})/\ \Gamma_{M0} = 2,865 \times (275/\sqrt{3}) / 1.0 = 454.9\ kN > 63.58\ kN$

Ratio = $63.58 \;/\; 454.9 = 0.140$

$hw / tw = 332/7.4 = 44.86 < 66.24$ (i.e. 72 ε) [EN 1993-1-1:2005 Cl. 6.2.6(6)]

Hence shear buckling check not required.

$V_{Ed} = 63.58 < 0.5 \times 454.9$ [EN 1993-1-1:2005 Cl. 6.2.8(2)]

Hence combined shear and bending checks not required.

<u>Check for Deflection Under Service Load</u>

$G + Q = 9.58 + 6.25 = 15.83$ kN/m

$$\delta = \frac{5 \times 15.83 \times 5,\ 700^4}{384 \times 205,\ 000 \times 14,\ 135(10)^4} = 7.3 \text{ mm} < L \;/300 = \frac{5,\ 700}{300} = 19 \text{ mm}$$

Results

Table 635: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio for Bending	0.368	0.368	none	
Ratio for LTB	0.875	0.877	negligible	
Ratio for Shear	0.140	0.140	none	
Deflection Limit (mm)	19	19	none	

STAAD Input

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 - Simply supported laterally unrestrained beam.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 03-Jan-08
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5.7 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
```

```
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST UB356X171X51
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
1 UNI GY -9.58
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
1 UNI GY -6.25
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.35 2 1.5
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
GM0 1 ALL
GM1 1 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE EN 1993-1-1:2005
SGR 1 ALL
DFF 300 ALL
TRACK 4 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST  UB356X171X51 (BRITISH SECTIONS)
                           PASS     EC-6.3.2 LTB       0.877         3
                        0.00           0.00         -90.60        2.85
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    570.00
      Gross Area =   64.90          Net Area =   64.90
```

```
                                     z-axis          y-axis
      Moment of inertia        :    14100.002        968.000
      Plastic modulus          :      896.000        174.000
      Elastic modulus          :      794.366        112.886
      Shear Area               :       35.500         28.652
      Radius of gyration       :       14.740          3.862
      Effective Length         :      570.000        570.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   1784.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                     z-axis          y-axis
      Slenderness ratio (KL/r) :         38.7          147.6
      Compression Capacity     :       1678.4          496.1
      Tension Capacity         :       1784.8         1784.8
      Moment Capacity          :        246.4           47.9
      Reduced Moment Capacity  :        246.4           47.9
      Shear Capacity           :        563.6          454.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  103.3
      co-efficients C1 & K : C1 =1.132 K =1.0, Effective Length= 5.700
      Lateral Torsional Buckling Curve : CURVE c
      Elastic Critical Moment for LTB,              Mcr   =   121.4
      Compression buckling curves:     z-z:  Curve a   y-y:  Curve b
      Critical Load For Torsional Buckling,         NcrT  =  1574.5
      Critical Load For Torsional-Flexural Buckling, NcrTF =  1574.5
      STAAD PLANE                                                -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.368     3     0.0      0.0     0.0   -90.6     0.0
   EC-6.2.6-(Y)   0.140     3     0.0     63.6     0.0     0.0     0.0
   EC-6.3.2 LTB   0.877     3     0.0      0.0     0.0   -90.6     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
    43. LOAD LIST 4
    44. PARAMETER 2
    45. CODE EN 1993-1-1:2005
    46. SGR 1 ALL
    47. DFF 300 ALL
    48. TRACK 4 ALL
    49. CHECK CODE ALL
 STEEL DESIGN
                        STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
5
 DEFLECTION CHECKS        (LENGTH UNITS - METE)
 MEMBER     TABLE         RESULT   ACTUAL    DEFL.LIMIT DEFL.LEN/  LOAD/
                                   DEFL.     RATIO       DFF    LOCATION
 =======================================================================
      1 ST  UB356X171X51  (BRITISH SECTIONS)
                           PASS      0.007     0.019     5.700       4
```

```
                                                     0.394   300.000     2.85
   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.C.5 Member Design* (on page 1480)

V. EC3 - Tube Section with Axial Load

Verify the adequacy of a cantilever tube section subject to axial force and bending per EN 1993-1-1:2005 (no national annex used).

Details

The member is a 5 m long cantilever. The member is subject to a 25 kN axial compressive load along with moments of 10 kN·m about the major axis and 5 kN·m about the minor axis at the free end. The steel is grade S275. The section is a European TUB120806.

Section Properties

$A = 23.4\ cm^2$
Depth, D = 120 mm
Width, B = 80 mm
t = 6.3 mm
$I_z = 447\ cm^4$
$I_y = 234\ cm^4$
$Z_z = 91\ cm^3$
$Z_y = 68.2\ cm^3$
$J = 486\ cm^4$
$C_w = 0\ cm^6$

$$r_z = \sqrt{I_z / A} = \sqrt{\frac{447}{23.4}} = 4.371\ cm$$

$$r_y = \sqrt{I_y / A} = \sqrt{\frac{234}{23.4}} = 3.162\ cm$$

Partial safety factors:

$$\Gamma_{M0} = 1.0$$

$$\Gamma_{M1} = 1.0$$

$$\Gamma_{M2} = 1.25$$

Validation

Section Classification

$$\alpha = 1$$

$$C = D - 2t = 120 - 2 \times 6.3 = 107.4\ mm$$

$$\varepsilon = \sqrt{\frac{235}{f_y}} = \sqrt{\frac{235}{275}} = 0.924$$

As per Table 5.2:

$$C / t = 107.4 / 6.3 = 17.05 < 50\times\varepsilon^2 = 50\ (0.924)^2 = 42.7$$

$$C/t = \frac{107.4}{6.3} = 17.05 < \frac{396\varepsilon}{13a - 1} = 30.36$$

Hence, this is a Class 1 section.

<u>Slenderness Ratio</u>

The slenderness ratio = $kL / r = 1 \times 5{,}000 / 31.62 = 158.1$

<u>Axial Tension</u>

Determine axial tension capacity per Cl. 6.2.3.

$$N_{pl,Rd} = A_g \times f_y / \gamma_{M0} = 2{,}340 \times 275 / 1.0 \times (10)^{-3} = 643.5 \text{ kN}$$

$$N_{u,Rd} = 0.9 A_{net} \times f_u / \gamma_{M2} = 0.9 \times 2{,}340 \times 295 / 1.25 \times (10)^{-3} = 497.0 \text{ kN}$$

The tensile capacity is the minimum of $N_{pl,Rd}$ and $N_{u,Rd}$, thus: $N_{t,Rd} = 497.0\ kN$.

No axial tension in section, so by observation no need to check ratio.

<u>Axial Compression</u>

Determine axial compression capacity per Cl. 6.2.4 for a Class 1 section.

$$N_{c,Rd} = A_g \times f_y / \gamma_{M0} = 2{,}340 \times 275 / 1.0 \times (10)^{-3} = 643.5 \text{ kN}$$

Next, check the flexural buckling resistance per Cl. 6.3.1.3:

$$N_{b,Rd} = \chi \times A \times f_y / \gamma_{M1}$$

From Table 6.1: the imperfection factor, $a = 0.21$.

Along the Z axis:

$$\bar{\lambda}_z = \sqrt{A \times f_y / N_{cr}} = L_{cr} / i_z \times \lambda_1$$

where

L_{cr}	=	5,000 mm
i_z	=	43.71 mm
λ_1	=	$93.9\times\varepsilon = 93.9 \times 0.924 = 86.8$

$$\bar{\lambda}_z = 1.318$$

$$\Phi = 0.5\left[1 + a\left(\bar{\lambda}_z - 0.2\right) + \bar{\lambda}_z^2\right] = 0.5\left[1 + 0.21\left(1.318 - 0.2\right) + (1.318)^2\right] = 1.486$$

$$\chi_z = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}_z^2}} = 0.4605$$

$$N_{b,Rd} = 0.4605 \times 2{,}340 \times 275 / 1.0 = 296.3 \text{ kN}$$

Along the Y axis:

$$\bar{\lambda}_y = \sqrt{A \times f_y / N_{cr}} = L_{cr} / i_y \times \lambda_1$$

where

i_y	=	31.62 mm

$$\bar{\lambda}_z = 1.822$$

$$\Phi = 0.5\left[1 + a\left(\bar{\lambda}_y - 0.2\right) + \bar{\lambda}_y^2\right] = 0.5\left[1 + 0.21\left(1.822 - 0.2\right) + (1.822)^2\right] = 2.330$$

$$\chi_y = \frac{1}{\Phi + \sqrt{\Phi^2 - \lambda_y^2}} = 0.2644$$

$$N_{\mathrm{b,Rd}} = 0.2644 \times 2{,}340 \times 275 / 1.0 = 170.2 \text{ kN}$$

The compression capacity is the minimum of $N_{c,Rd}$ and $N_{b,Rd}$, thus: $N_{c,Rd} = 170.2\ kN$.

Ratio per Eq. 6.9: $M_{Ed} / M_{c,Rd} = 25.0 / 170.2 = 0.147$

<u>Bending Capacity</u>

Along Z axis:

Maximum bending moment in the section: $M_{Ed,z} = 10.0\ kN{\cdot}m$.

Check for bending capacity per Cl. 6.2.5:

For a Class 1 section:

$$M_{\mathrm{c,Rd}} = W_{\mathrm{ply}} \times f_y / \gamma_{\mathrm{M0}} = 91 \times 275 / 1.0 \times (10)^{-3} = 25.03 \text{ kN}{\cdot}\text{m}$$

Ratio per Eq. 6.12: $M_{Ed} / M_{c,Rd} = 10.0 / 25.03 = 0.400$

Along Y axis:

Maximum bending moment in the section: $M_{Ed,y} = 5.0\ kN{\cdot}m$.

Check for bending capacity per Cl. 6.2.5:

For a Class 1 section:

$$M_{\mathrm{c,Rd}} = W_{\mathrm{ply}} \times f_y / \gamma_{\mathrm{M0}} = 68.2 \times 275 / 1.0 \times (10)^{-3} = 18.76 \text{ kN}{\cdot}\text{m}$$

Ratio per Eq. 6.12: $M_{Ed} / M_{c,Rd} = 5.0 / 18.76 = 0.267$

<u>Shear Capacity</u>

Along Z direction:

$$A_v = \frac{A \times B}{D + B} = \frac{2{,}340 \times 80}{120 + 80} = 936 \text{ mm}^2$$

Check for shear capacity per Cl. 6.2.6 for plastic design (Class 1):

$$V_{\mathrm{c,Rd}} = V_{\mathrm{pl,Rd}} = A_v \times \left(f_y / \sqrt{3} \right) / \gamma_{\mathrm{M0}} = 936 \times \left(275 / \sqrt{3} \right) / 1.0 = 148.6 \text{ kN}$$

Along Y direction:

$$A_v = \frac{A \times D}{D + B} = \frac{2{,}340 \times 120}{120 + 80} = 1{,}404 \text{ mm}^2$$

Check for shear capacity per Cl. 6.2.6 for plastic design (Class 1):

$$V_{\mathrm{c,Rd}} = V_{\mathrm{pl,Rd}} = A_v \times \left(f_y / \sqrt{3} \right) / \gamma_{\mathrm{M0}} = 1{,}404 \times \left(275 / \sqrt{3} \right) / 1.0 = 222.9 \text{ kN}$$

No shear in section, so by observation no need to check ratio.

<u>Lateral Torsional Buckling</u>

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 G I_t}{\pi^2 E I_z} + (C_2 Z_g)^2} - C_2 Z_g \right]$$

where

C_1	=	*1.0*
C_2	=	*1.0*
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*189,400*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*0*
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	*2,023,000*
$C_2 Z_g$	=	*1.0×40 = 40*

Therefore, $M_{cr} = 1.0 \times 189,400 \left[\sqrt{0 + 2,023,000 + 40^2} - 40\right] = 261.8\ kN \cdot m$

As per Cl. 6.3.2.1, tube sections are not susceptible to lateral-torsional buckling, so $\chi_{LT} = 1.0$.

$$M_{\mathrm{b,Rd}} = \chi_{\mathrm{LT}} W_y \times f_y / \gamma_{\mathrm{M1}} = 1.0 \times 91 \times 275 / 1.0 \times (10)^{-3} = 25.03\ \mathrm{kN \cdot m}$$

Ratio per Eq. 6.12: $M_{Ed}\ /\ M_{b,Rd} = 10.0\ /\ 24.92 = 0.400$

<u>Check for Interaction</u>

From Table B.3 in Annex B of EC3:

$$\Psi = 1.0$$

$$\mathrm{C_{my}} = \mathrm{C_{mz}} = \mathrm{C_{mLT}} = 0.6 + 0.4 \times \Psi = 1.0$$

$$\mathrm{N_{Rk}} = \mathrm{A} \times \mathrm{f_y} = 643.5\ \mathrm{kN}$$

From Table B.1 of Annex B, the interaction factors:

$$K_{\mathrm{zz}} = C_{\mathrm{mz}} \frac{1 + (\lambda_z - 0.2) N_{\mathrm{Ed}}}{\chi_z N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 1.0 \frac{1 + (1.318 - 0.2)25}{0.4605 \times 643.51} = 1.094 > 1 + \frac{0.8 N_{\mathrm{Ed}}}{\chi_z N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 1.067$$

$$\mathrm{K_{zz}} = 1.067$$

$$K_{\mathrm{yy}} = C_{\mathrm{my}} \frac{1 + (\lambda_y - 0.2) N_{\mathrm{Ed}}}{\chi_y N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 1.0 \frac{1 + (1.822 - 0.2)25}{0.2644 \times 643.51} = 1.238 > 1 + \frac{0.8 N_{\mathrm{Ed}}}{\chi_y N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 1.118$$

$$\mathrm{K_{yy}} = 1.118$$

$$\mathrm{K_{yz}} = 0.6 \times \mathrm{K_{zz}} = 0.641$$

$$\mathrm{K_{zy}} = 0.6 \times \mathrm{K_{yy}} = 0.671$$

For K_{yz}, consider Table B.2 as well:

$$K_{\mathrm{yz}} = 1 - \frac{0.1 \lambda_y}{C_{\mathrm{mLT}} - 0.25} \frac{N_{\mathrm{Ed}}}{\chi_y N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 0.964 < 1 - \frac{0.1}{C_{\mathrm{mLT}} - 0.25} \frac{N_{\mathrm{Ed}}}{\chi_y N_{\mathrm{Rk}} \gamma_{\mathrm{M1}}} = 0.980$$

So, $K_{yz} = 0.980$

Check for Clause 6.3.3-661:

$$\frac{N_{\mathrm{Ed}}}{\chi_z \frac{N_{\mathrm{Rk}}}{\gamma_{\mathrm{M1}}}} + K_{\mathrm{zz}} \frac{M_{\mathrm{z,Ed}}}{\chi_{\mathrm{LT}} \frac{M_{\mathrm{z,Rk}}}{\gamma_{\mathrm{M1}}}} + K_{\mathrm{zy}} \frac{M_{\mathrm{y,Ed}}}{\frac{M_{\mathrm{y,Rk}}}{\gamma_{\mathrm{M1}}}} = \frac{25}{0.4605 \times 643.5} + 1.067 \frac{10}{1.0 \times 25.03} + 0.671 \frac{5}{18.76} = 0.690$$

Check for Clause 6.3.3-662:

$$\frac{N_{\mathrm{Ed}}}{x_y \frac{N_{\mathrm{Rk}}}{\gamma_{\mathrm{M1}}}} + K_{\mathrm{yz}} \frac{M_{\mathrm{z,Ed}}}{x_{\mathrm{LT}} \frac{M_{\mathrm{z,Rk}}}{\gamma_{\mathrm{M1}}}} + K_{\mathrm{yy}} \frac{M_{\mathrm{y,Ed}}}{\frac{M_{\mathrm{y,Rk}}}{\gamma_{\mathrm{M1}}}} = \frac{25}{0.2644 \times 643.5} + 0.980 \frac{10}{1.0 \times 25.03} + 1.118 \frac{5}{18.76} = 0.837$$

Results

Table 636: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Section Class	Class 1	Class 1	none	
Slenderness Ratio	158.1	158.1	none	
Tension Capacity (kN)	497.0	497	none	
Compression Capacity (Major) (kN)	296.3	296.3	none	
Compression Capacity (Minor) (kN)	170.2	170.2	none	
Moment Capacity (Major) (kN·m)	25.03	25	negligible	
Moment Capacity (Minor) (kN·m)	18.76	18.8	negligible	
Shear Area (Major) ($cm^{2)}$	9.36	9.36	none	
Shear Area (Minor) ($cm^{2)}$	14.04	14.04	none	
Shear Capacity (Major) (kN)	148.6	148.6	none	
Shear Capacity (Minor) (kN)	222.9	222.9	none	
M_{cr} (kN·m)	261.8	258.5	negligible	
M_B (kN·m)	25.03	25	negligible	
Ratio per Cl. 6.3.1.1	0.147	0.147	none	
Ratio per Cl. 6.2.9.1	0.400	0.4	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio per Cl. 6.3.3-661	0.690	0.69	none	
Ratio per Cl. 6.3.3-662	0.837	0.837	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 - Tube Section with Axial Load.std` is typically installed with the program.

The following design parameters are used:

- Fixed end supports: `CMM 2`
- Cantilever member: `CMN 0.7`
- Use Cl. 6.3.2.2 to determine χ_{LT}: `MTH 1`
- The values of `C1 1.0` and `C2 1.0` are specified.
- The values of F_y and F_u are specified directly using `PY 275000` and `FU 295000`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-May-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST TUB120806
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
CMM 2 ALL
C1 1 ALL
C2 1 ALL
```

```
FU 295000 ALL
PY 275000 ALL
MTH 1 ALL
CMN 0.7 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   TUB120806   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.3-662       0.837         1
                       25.00 C          5.00         -10.00       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   23.40          Net Area =   23.40
                                      z-axis          y-axis
      Moment of inertia        :      447.000         234.000
      Plastic modulus          :       91.000          68.200
      Elastic modulus          :       74.500          58.500
      Shear Area               :        9.360          14.040
      Radius of gyration       :        4.371           3.162
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    643.50
      Axial force/Squash load  :     0.039
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        114.4           158.1
      Compression Capacity     :        296.3           170.2
      Tension Capacity         :        497.0           497.0
      Moment Capacity          :         25.0            18.8
      Reduced Moment Capacity  :         25.0            18.8
      Shear Capacity           :        148.6           222.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =   25.0
      co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : CURVE d
      Elastic Critical Moment for LTB,               Mcr   =   258.5
      Compression buckling curves:     z-z:  Curve a   y-y:  Curve a
```

```
      STAAD SPACE                                                -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX        VY       VZ      MZ       MY
   EC-6.3.1.1     0.147     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.2.9.1     0.400     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-661   0.690     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-662   0.837     1    25.0      0.0     0.0   -10.0     5.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 - Tube Section with UDL

Verify the adequacy of a tube section used as a beam subject to a uniform load per EN 1993-1-1:2005 (no national annex used).

Details

The member is a 5 m long beam with fixed ends. The member is subject to a 10 kN/m uniform load. The steel is grade S275. The section is a European TUB1201205.

Section Properties

$A = 22.9\ cm^2$
Depth, D = 120 mm
Width, B = 120 mm
t = 5 mm
$I_z = I_y = 503\ cm^4$
$Z_z = Z_y = 97.6\ cm^3$
$C_w = 0\ cm^6$

$$r_z = r_y = \sqrt{I/A} = \sqrt{\frac{503}{22.9}} = 4.687\ \ cm$$

Partial safety factors:

$$\Gamma_{M0} = 1.0$$

$$\Gamma_{M1} = 1.0$$

$$\Gamma_{M2} = 1.25$$

Validation

Section Classification

$$\varepsilon = \sqrt{\frac{235}{f_y}} = \sqrt{\frac{235}{275}} = 0.924$$

As per Table 5.2:

$$\alpha = 1$$

$$C = D - 2 \times t = 110\ mm$$

$$C/t = \frac{110}{5} = 22 < \frac{396 \times \varepsilon}{13a - 1} = \frac{396 \times 0.924}{12} = 30.36$$

Hence, this is a Class 1 section.

Slenderness Ratio

The slenderness ratio = $kL / r = 1 \times 5{,}000 / 46.87 = 106.7$

Axial Tension

Determine axial tension capacity per Cl. 6.2.3.

$$N_{\mathrm{pl,Rd}} = A_g \times f_y / \gamma_{\mathrm{M0}} = 2{,}290 \times 275 / 1.0 \times (10)^{-3} = 630.0 \text{ kN}$$

$$N_{\mathrm{u,Rd}} = 0.9A_{\mathrm{net}} \times f_u / \gamma_{\mathrm{M2}} = 0.9 \times 2{,}290 \times 450 / 1.25 \times (10)^{-3} = 742.0 \text{ kN}$$

The tensile capacity is the minimum of $N_{pl,Rd}$ and $N_{u,Rd}$, thus: $N_{t,Rd} = 630.0\ kN$.

No axial tension in section, so by observation no need to check ratio.

Axial Compression

Determine axial compression capacity per Cl. 6.2.4 for a Class 1 section.

$$N_{\mathrm{c,Rd}} = A_g \times f_y / \gamma_{\mathrm{M0}} = 2{,}290 \times 275 / 1.0 \times (10)^{-3} = 630.0 \text{ kN}$$

Next, check the flexural buckling resistance per Cl. 6.3.1.3:

$$N_{\mathrm{b,Rd}} = \chi \times A \times f_y / \gamma_{\mathrm{M1}}$$

From Table 6.1: the imperfection factor, $a = 0.21$.

$$\bar{\lambda} = \sqrt{A \times f_y / N_{\mathrm{cr}}} = L_{\mathrm{cr}} / i \times \lambda_1$$

where

L_{cr}	=	5,000 mm
i	=	46.87 mm
λ_1	=	$93.9 \times \varepsilon = 93.9 \times 0.924 = 86.8$

$$\bar{\lambda} = 1.229$$

$$\Phi = 0.5\left[1 + a\left(\bar{\lambda} - 0.2\right) + \bar{\lambda}^2\right] = 0.5\left[1 + 0.21\left(1.229 - 0.2\right) + (1.229)^2\right] = 1.363$$

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}} = 0.512$$

$$N_{\mathrm{b,Rd}} = 0.512 \times 2{,}290 \times 275 / 1.0 = 322.4 \text{ kN}$$

The compression capacity is the minimum of $N_{c,Rd}$ and $N_{b,Rd}$, thus: $N_{c,Rd} = 322.4\ kN$.

No axial compression in section, so by observation no need to check ratio.

Bending Capacity

Maximum bending moment in the section (at fixed supports): $M_{Ed} = 20.83\ kN{\cdot}m$.

Check for bending capacity per Cl. 6.2.5:

For a Class 1 section:

$$M_{\mathrm{c,Rd}} = W_{\mathrm{ply}} \times f_y / \gamma_{\mathrm{M0}} = 97.6 \times 275 / 1.0 \times (10)^{-3} = 26.84 \text{ kN}{\cdot}\text{m}$$

Ratio per Eq. 6.12: $M_{Ed} / M_{c,Rd} = 20.83 / 26.84 = 0.776$

Shear Capacity

Maximum shear in the section (at fixed supports): $V_{Ed} = 25\ kN$

$$A_v = \frac{A \times D}{D + B} = \frac{2{,}290 \times 120}{120 + 120} = 1{,}145\ \text{mm}^2$$

Check for shear capacity per Cl. 6.2.6 for plastic design (Class 1):

$$V_{c,Rd} = V_{pl,Rd} = A_v \times \left({}^{f_y}\!/\!_{\sqrt{3}}\right) / \gamma_{M0} = 1{,}145 \times \left({}^{275}\!/\!_{\sqrt{3}}\right) / 1.0 = 181.8\ \text{kN}$$

Ratio per Eq. 6.17: $V_{Ed}\ /\ V_{c,Rd} = 25.0\ /\ 181.8 = 0.138$

Lateral Torsional Buckling

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL\ )^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(kL\ )^2 GI_t}{\pi^2 EI_z} + (C_2 Z_g)^2} - C_2 Z_g\right]$$

where

C_1	=	1.0
C_2	=	1.0
$\frac{\pi^2 EI_y}{(kL\ )^2}$	=	162,800
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	0
$\frac{(kL\ )^2 GI_T}{\pi^2 EI_y}$	=	375,300
$C_2 Z_g$	=	$1.0 \times 60 = 60$

Therefore, $M_{cr} = 1.0 \times 162{,}800\left[\sqrt{0 + 375{,}300 + 60^2} - 60\right] = 904.6\ kN \cdot m$

From Table 6.4, buckling curve "d" is used. From Table 6.3, $a_{LT} = 0.76$ for buckling curve d.

As per Cl. 6.3.2.1, tube sections are not susceptible to lateral-torsional buckling, so $\chi_{LT} = 1.0$.

$$M_{b,Rd} = \chi_{LT} W_y \times f_y / \gamma_{M1} = 1.0 \times 97.6 \times 275 / 1.0 \times (10)^{-3} = 26.84\ \text{kN} \cdot \text{m}$$

Ratio per Eq. 6.12: $M_{Ed}\ /\ M_{b,Rd} = 20.83\ /\ 26.84 = 0.776$

Results

Table 637: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Shear Area (cm^2)	11.45	11.45	none	
Section Class	Class 1	Class 1	none	
Slenderness Ratio	106.7	106.7	none	
Tension Capacity (kN)	630.0	629.8	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Compression Capacity (kN)	322.4	322.4	none	
Moment Capacity (kN·m)	26.84	26.8	negligible	
Shear Capacity (kN)	181.8	181.8	none	
M_{cr} (kN·m)	904.6	905.7	negligible	
M_B (kN·m)	26.84	26.8	negligible	
Ratio per Cl. 6.2.5	0.776	0.776	none	
Ratio per Cl. 6.2.6 (Y)	0.138	0.138	none	
Ratio per Cl. 6.3.2 LTB	0.776	0.776	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 - Tube Section with UDL.std` is typically installed with the program.

The following design parameters are used:

- Fixed end supports with a uniform load: `CMM 2`
- Fixed end supports: `CMN 0.5`
- Use Cl. 6.3.2.2 to determine χ_{LT}: `MTH 1`
- The values of `C1 1.0` and `C2 1.0` are specified.
- The values of F_y and F_u are specified directly using `PY 275000` and `FU 295000`.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 04-May-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
```

```
MEMBER PROPERTY EUROPEAN
1 TABLE ST TUB1201205
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
C1 1 ALL
C2 1 ALL
CMM 2 ALL
CMN 0.5 ALL
MTH 1 ALL
PY 275000 ALL
FU 450000 ALL
KC 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NOT USED
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   TUB1201205  (EUROPEAN SECTIONS)
                           PASS     EC-6.2.5           0.776         1
                        0.00          0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   22.90          Net Area =   22.90
                                      z-axis          y-axis
      Moment of inertia        :      503.000         503.000
      Plastic modulus          :       97.600          97.600
      Elastic modulus          :       83.833          83.833
      Shear Area               :       11.450          11.450
      Radius of gyration       :        4.687           4.687
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    629.75
```

```
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00         GM2 :  1.25
                                     z-axis         y-axis
      Slenderness ratio (KL/r) :       106.7          106.7
      Compression Capacity     :       322.4          322.4
      Tension Capacity         :       629.8          629.8
      Moment Capacity          :        26.8           26.8
      Reduced Moment Capacity  :        26.8           26.8
      Shear Capacity           :       181.8          181.8
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =   26.8
      co-efficients C1 & K : C1 =1.000 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : CURVE d
      Elastic Critical Moment for LTB,              Mcr   =   905.7
      Compression buckling curves:     z-z:  Curve a   y-y:  Curve a
      STAAD PLANE                                              -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.2.5       0.776     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.138     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.776     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Belgian NA - Channel Section with Conc Load

Calculate the bending capacity of a channel section beam under subject to a pair of concentrated load per the Belgian NA to EC3

Details

The section is a C15X50, grade S275 steel. The member is a 6 m span, simply supported beam is subject to two concentrated loads of 10 kN each at the 1/3 points (2 m from either support).

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{1{,}123(10)^3 275}{1.0(10)^6} = 308.7 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2}\left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j)\right]$$

where

C_1	=	1.046
C_2	=	0.42
C_3	=	1.0

$\frac{\pi^2 EI_y}{(kL)^2}$ = $257,320$

$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$ = $28,852$

$\frac{(kL)^2 GI_T}{\pi^2 EI_y}$ = $330,055$

C_2Z_g = $0.42 \times 0.5 \times 381 = 80.01$

β_f = $\frac{b_c^3 + t_c}{b_c^3 + t_c + b_t^3 + t_t} = 0.5$ (From SN030a-EN-EU)

Z_j = $0.4 \times h_s (2 \times \beta_f - 1) = 0$, since $\beta_f \geq 0.5$

C_3Z_j = $1.0\ (0) = 0$

Therefore, $M_{cr} = 1.046 \times 257,320 \left[\sqrt{28,852 + 330,055 + (80.01)^2} - (80.01)\right] = 141.2\ kN \cdot m$

Results

Table 638:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	308.7	308.7	none	
Critical moment, M_{cr} (kN·m)	141.2	142.2	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Belgian NA - Channel Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Simply-supported span: `CMN 1` and `C3 1`
- Concentrated loads at 1/3 points supports: `CMM 5`
- The value of `C2 0.42` is specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 18-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
```

```
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10 2
1 CON GY -10 4
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 8 ALL
C2 0.42 ALL
C3 1 ALL
CMM 5 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
*KC 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   C15X50      (AISC SECTIONS)
                           PASS     EC-6.3.2 LTB       0.228         1
                        0.00           0.00        -20.00        2.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =   94.84          Net Area =   94.84
                                      z-axis          y-axis
```

```
      Moment of inertia        :    16815.750        457.855
      Plastic modulus          :     1122.514        133.391
      Elastic modulus          :      882.716         61.711
      Shear Area               :       20.800         69.290
      Radius of gyration       :       13.316          2.197
      Effective Length         :      600.000        600.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   2608.06
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00         GM2 :  1.25
                                    z-axis          y-axis
      Slenderness ratio (KL/r) :        45.1          273.1
      Compression Capacity     :      2170.6          227.2
      Tension Capacity         :      2014.4         2014.4
      Moment Capacity          :       308.7           36.7
      Reduced Moment Capacity  :       308.7           36.7
      Shear Capacity           :       330.2         1097.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =   87.6
      co-efficients C1 & K : C1 =1.046 K =1.0, Effective Length= 6.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,              Mcr   =   142.2
      Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  4811.4
      Critical Load For Torsional-Flexural Buckling, NcrTF =  4544.2
      STAAD PLANE                                                -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.2.5       0.065     1     0.0     10.0     0.0   -20.0     0.0
   EC-6.2.6-(Y)   0.009     1     0.0     10.0     0.0     0.0     0.0
   EC-6.3.2 LTB   0.228     1     0.0     10.0     0.0   -20.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 Belgian NA - Column with Axial Load

Calculate the axial and bending capacities and interaction ratios of a column using the Belgian NA to EC3.

Details

The 5 m tall column as a fixed base and is free to translate and rotate at the top. The column is subject to a 25 kN compressive load and moments of 5 kN·m about the X axis and 10 kN·m about the Z axis. The section is an HD320X127, grade S275 steel.

Validation

<u>Bending Capacity</u>

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149(10)^3 275}{1.0(10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	*2.578*
C_2	=	*1.554*
$\frac{\pi^2 EI_y}{(kL)^2}$	=	*7,477,200*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*22,394*
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	*23,740*
$C_2 Z_g$	=	*1.554×160 = 248.6*

Therefore, $M_{cr} = 1{,}540.6\ kN{\cdot}m$

From the Belgian NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times 10^3 \times 275}{1{,}540.6 \times 10^6}} = 0.619$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34\times(0.619 - 0.4) + 0.75 \times 0.619^2] = 0.681$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.908$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.908 \times 2{,}149(10)^3 \times 275}{1.0} = 536.8 \text{ kN·m}$$

Compression Capacity

Critical compressive values:

$$N_{cry} = \sqrt{\frac{\pi^2 EI_y}{(kl)^2}} = 7{,}477 \text{ kN}$$

$$N_{crz} = \sqrt{\frac{\pi^2 EI_z}{(kl)^2}} = 24{,}943 \text{ kN}$$

$$\lambda_{cy} = \sqrt{\frac{A \times f_y}{N_{cry}}} = 0.770$$

$$\lambda_{cz} = \sqrt{\frac{A \times f_y}{N_{crz}}} = 0.422$$

Determine χ_y

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for y-y axis, use curve "c". Hence, $\alpha = 0.49$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha\ (\lambda - 0.2) +\lambda^2)] = 0.936$

$$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.681 \quad \text{(governs)}$$

Determine χ_z

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for z-z axis, use curve "b". Hence, $\alpha = 0.34$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha\ (\lambda - 0.2) +\lambda^2)] = 0.627$

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.917 \quad \text{(does not govern)}$$

Compression capacity:

$$\chi_y \times N_{Rk} = 0.681 \times 7{,}477 = 3{,}020.2\ \text{kN}$$

Compression ratio: $25\ kN\ /\ 3{,}020.2\ kN = 0.008$

<u>Critical Axial Loads for Flexure and Flexural Torsional Buckling</u>

From NCCI document SN001a-EN-FU:

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 EI_w}{l_T^2}\right)$$

where

$$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2;\ \text{here } y_0 = z_0 = 0$$

$i_y = 75.68\ mm$ and $i_z = 138.23\ mm$

$$= 24{,}835\ \text{mm}^2$$

$$N_{cr,T} = \frac{1}{24{,}835}\left[\frac{20{,}500 \times 225.1(10)^4}{2.6} + \frac{\pi^2(205{,}000)2.069(10)^{12}}{5{,}000^2}\right] = 13{,}889\ \text{kN}$$

$$N_{cr,TF} = \frac{i_0^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{\left(N_{cr,y} + N_{cr,T}\right)^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_0^2}}\right]$$

$$= \frac{1}{2}\left[24{,}943 + 13{,}889 - \sqrt{(24{,}943 + 13{,}889)^2 - 4\left(24{,}943\right)\left(13{,}889\right)^1/_1}\right] = 13{,}889\ \text{kN}$$

<u>Interaction Check</u>

From Annex A, Table A.1 of EC3, the auxiliary terms:

$$\mu_z = \frac{1 - \frac{N_{Ed}}{N_{crz}}}{1 - \chi_z * \frac{N_{Ed}}{N_{crz}}} = 0.9999$$

$$\mu_y = \frac{1 - \frac{N_{Ed}}{N_{cry}}}{1 - x_y * \frac{N_{Ed}}{N_{cry}}} = 0.9989$$

$\chi_{LT} = 0.908294$

$W_z = w_{plz}/w_{elz} = 2149/1926 = 1.1158$

$W_y = w_{ply}/w_{ely} = 939/615.933 = 1.525 > 1.5$, so, $w_y = 1.5$

$\psi=1.0$

$$c_{mz,0} = 0.79 + 0.21 \times 1 + 0.36(1 - 0.33)\frac{25}{24{,}942} = 1.0002$$

Also, $c_{mz,0} > 1 - \frac{N_{Ed}}{N_{cr,z}}$, OK

$$c_{my,0} = 0.79 + 0.21 \times 1 + 0.36(1 - 0.33)\frac{25}{7{,}477} = 1.0008$$

Also, $c_{my,0} > 1 - \frac{N_{Ed}}{N_{cr,y}}$, OK

Determination of λ_0

$C_1 = 1$, $C_2 = 0$

$M_{cr} = 1{,}606$ kN

$$\lambda_0 = \sqrt{\frac{2{,}149 \times 10^3 \times 275}{1{,}606 \times 10^6}} = 0.6066$$

$$\lambda_{0,limit} = 0.2\sqrt{C1}\sqrt[4]{\left(1 - \frac{N_{Ed}}{N_{cry}}\right)*\left(1 - \frac{N_{Ed}}{N_{crTF}}\right)} = 0.321$$

$\lambda_0 > \lambda_{0,limit}$

$a_{LT} = 1 - I_t/I_z = 0.993$

$$\varepsilon_z = \frac{M_{Z,Ed}}{N_{Ed}} \times \frac{A}{W_{elz}} = 3.350$$

$$C_{mz} = C_{mz,0} + \left(1 - C_{mz,0}\right)\frac{\sqrt{\varepsilon_Z * a_{LT}}}{1 + \varepsilon_Z * a_{LT}} = 1.000$$

$$C_{my} = C_{my,0} = 1.001$$

$$C_{mLT} = C_{mz}^2 \times \frac{a_{LT}}{\sqrt{\left(1 - \frac{N_{Ed}}{N_{cry}}\right) \times \left(1 - \frac{N_{Ed}}{N_{crT}}\right)}} = 0.995 < 1$$

So, $C_{mLT} = 1.0$

λ_{max} = the maximum of λ_y and λ_z, therefore $\lambda_{max} = \lambda_y = 0.7702$

$$b_{LT} = 0.5a_{LT}\lambda_0^2\frac{M_{yEd}}{x_{LT} \times M_{ply.Rd}} \times \frac{M_{zEd}}{M_{plz.Rd}} = 0.5 \times 0.993 \times 0.6066^2\frac{5 \times 10}{0.908 \times 258.3 \times 591} \cong 0$$

$$\eta_{pl} = \frac{N_{Ed}}{\frac{N_{Rk}}{GM1}} = \frac{25}{4{,}436 / 1.0} = 0.0056$$

$$C_{zz} = 1 = \left(w_z + 1\right)\left[\left(2 - \frac{1.6}{w_Z} C_{mz}{}^2 \lambda_{max} - \frac{1.6}{w_Z} C_{mz}{}^2 \lambda_{max}{}^2\right)\eta_{pl} - b_{LT}\right]$$

$$= 1 + (1.1158 - 1)\left[\left(2 - \frac{1.6}{1.1158} 1^2 \times 0.7702 - \frac{1.6}{1.1158} 1^2 \times 0.7702^2\right)0.00564 - 0\right] \cong 1.0$$

$W_{elz}/W_{plz} = 0.8962 < C_{ZZ}$. So, $C_{ZZ} = 1.0$

$$C_{LT} = 10a_{LT} \frac{\lambda_0{}^2}{5 + \lambda_y{}^4} \times \frac{M_{z,Ed}}{C_{mz} \times x_{LT} \times M_{plz,Rd}} = 0.0127$$

$$C_{zy} = 1 + (w_y - 1)\left[\left(2 - 14 \frac{C_{my}{}^2 \times \lambda_{max}{}^2}{w_y{}^5}\right)\eta_{pl} - C_{LT}\right] = 0.9962$$

Check $C_{zy} > 0.6\sqrt{\frac{W_y}{W_z}} \times \frac{W_{ely}}{W_{ply}} = 0.456$, OK

$$d_{LT} = 2a_{LT} \frac{\lambda_0}{0.1 + \lambda_y{}^4} \times \frac{\mathrm{M_{z,Ed}}}{\mathrm{C_{mz}} \times \chi_{\mathrm{LT}} \times \mathrm{M_{plz,Rd}}} \times \frac{\mathrm{M_{y,Ed}}}{\mathrm{C_{my}} {*} \mathrm{M_{ply,Rd}}}$$

$$= 2 \times 0.9927 \frac{0.6066}{0.1 + 0.7702^2} \times \frac{10}{1 \times 0.9083 \times 591} \times \frac{5}{1.0008 \times 258.3} = 0.00096$$

$$C_{yz} = 1 + (w_z - 1)\left[\left(2 - 14 \frac{C_{mz}{}^2 \times \lambda_{max}{}^2}{w_z{}^5}\right)\eta_{pl} - d_{LT}\right] = 0.9981$$

$$e_{LT} = 1.7a_{LT} \frac{\lambda_0}{0.1 + \lambda_y{}^4} \times \frac{M_{z,Ed}}{\mathrm{C_{mz}} \times \chi_{\mathrm{LT}} \times \mathrm{M_{plz,.Rd}}} = 0.0422$$

$$C_{yy} = 1 + \left(w_y - 1\right)\left[\left(2 - \frac{1.6}{w_y} C_{my}{}^2 \times \lambda_{max} - \frac{1.6}{w_y} \times C_{my}{}^2 \times \lambda_{max}{}^2\right)\eta_{pl} - e_{LT}\right]$$

$$= 1 + (1.5 - 1)\left[\left(2 - \frac{1.6}{1.5} 1.001^2 \times 0.7702 - \frac{1.6}{1.5} 1.001^2 \times 0.7702^2\right)0.00564 - 0.0422\right] = 0.9804$$

$W_{ely}/W_{ply} = 0.656 < C_{yy}$. So, $C_{yy} = 0.9804$

Interaction factors:

$$K_{zz} = C_{mz} C_{mLT} \frac{\mu_z}{1 - \frac{N_{Ed}}{N_{crz}}} \times \frac{1}{C_{zz}} = 1.001$$

$$K_{zy} = C_{my} \frac{\mu_z}{1 - \frac{N_{Ed}}{N_{cry}}} \times \frac{1}{C_{zy}} 0.6\sqrt{\frac{W_y}{W_z}} = 0.7012$$

$$K_{yz} = C_{mz} C_{mLT} \frac{\mu_y}{1 - \frac{N_{Ed}}{N_{crz}}} \times \frac{1}{C_{yz}} 0.6\sqrt{\frac{W_z}{W_y}} = 0.5185$$

$$K_{yy} = C_{my} \frac{\mu_y}{1 - \frac{N_{Ed}}{N_{cry}}} \times \frac{1}{C_{yy}} = 1.023$$

Check for Clause 6.3.3-661:

$$\frac{N_{Ed}}{x_Z \frac{N_{Rk}}{\gamma M_1}} + K_{ZZ} \frac{M_{z,Ed}}{x_{LT} \frac{M_{z,Rk}}{\gamma M_1}} + K_{zy} \frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.9174 \times 4{,}436} + 1.001 \frac{10}{0.9083 \times 591} + 0.7012 \frac{5}{258.3} = 0.038$$

$$\frac{N_{Ed}}{x_y \frac{N_{Rk}}{\gamma M_1}} + K_{yz} \frac{M_{z,Ed}}{x_{LT} \frac{M_{z,Rk}}{\gamma M_1}} + K_{yy} \frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.6809 \times 4{,}436} + 0.519 \frac{10}{0.9083 \times 591} + 1.0231 \frac{5}{258.3} = 0.038$$

Results

Table 639:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,540.5	1,541.5	negligible	
Bending capacity, M_B (kN·m)	536.8	536.8	none	
Critical load for torsional buckling, $N_{cr,T}$ (kN)	13,889	13,898.0	negligible	
Critical load for torsional-flexural buckling, $N_{cr,TF}$	13,889	13,898.0	negligible	
Compression interaction, Cl. 6.3.1.1	0.008	0.008	none	
Bending and compression interaction, Cl 6.3.3-661	0.038	0.038	none	
Bending and compression interaction, Cl. 6.3.3-662	0.038	0.037	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Europe\EC3 Belgian NA - Column with Axial Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Fixed support: `CMM 2.0`
- Fixed base and free at other end: `CMN 0.7`
- The value of `C2 1.554` is specified

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 8
CMM 2 ALL
C2 1.554 ALL
FU 295000 ALL
PY 275000 ALL
CMN 0.7 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                               -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.3-661       0.038         1
                       25.00 C          5.00        -10.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                       z-axis           y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                       z-axis           y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       4078.2          3045.5
      Tension Capacity         :       3426.0          3426.0
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  536.8
      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1541.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                               -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.3.1.1     0.008     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-661   0.038     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-662   0.037     1    25.0      0.0      0.0   -10.0     5.0
```

```
EC-6.3.2 LTB    0.019      1     25.0       0.0      0.0    -10.0      5.0
 Torsion has not been considered in the design.

************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 Belgian NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the Belgian NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	*1.35*
C_2	=	*0.59*
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*7,477,200*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*22,394*
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	*23,737*
$C_2 Z_g$	=	*0.59×160 = 94.4*

Therefore, $M_{cr} = 1{,}416 \ kN \cdot m$

From the Belgian NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}416 \times (10)^6}} = 0.646$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34\times(0.646 - 0.4) + 1 \times 0.646^2] = 0.698$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}{}^{2} - \beta\lambda_{LT}{}^{2}}\right)^{-1} = 0.896$

From Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1.0$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.896 < \frac{1}{\lambda_{LT}{}^{2}} = 2.396$, OK

$$M_B = \frac{x_{\mathrm{LT}} w_y f_y}{\gamma_{\mathrm{M1}}} = \frac{0.896 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 529.4 \text{ kN·m}$$

Results

Table 640:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,416	1,416	none	
Bending capacity, M_B (kN·m)	529.4	529.4	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Belgian NA - I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Simply-supported span: `CMN 1`
- Simply-supported span with concentrated load at midspan: `CMM 3`

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 17-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
```

```
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -30
PERFORM ANALYSIS
PARAMETER 2
CODE EN 1993-1-1:2005
NA 8 ALL
CMM 3 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.071         1
                        0.00           0.00        -37.50        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
```

```
     Axial force/Squash load   :     0.000
     GM0 :  1.00          GM1 :  1.00         GM2 :  1.25
                                       z-axis         y-axis
     Slenderness ratio (KL/r) :         36.2           66.1
     Compression Capacity     :       4078.2         3045.5
     Tension Capacity         :       3426.0         3426.0
     Moment Capacity          :        591.0          258.3
     Reduced Moment Capacity  :        591.0          258.3
     Shear Capacity           :       1301.9          821.3
  BUCKLING CALCULATIONS (units - kN,m)
     Lateral Torsional Buckling Moment       MB =  529.4
     co-efficients C1 & K : C1 =1.350 K =1.0, Effective Length= 5.000
     Lateral Torsional Buckling Curve : Curve b
     Elastic Critical Moment for LTB,              Mcr   =  1416.0
     Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
     Critical Load For Torsional Buckling,         NcrT  = 13898.0
     Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
     STAAD PLANE                                                -- PAGE NO.
4
  CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
   CLAUSE         RATIO  LOAD     FX       VY       VZ      MZ      MY
  EC-6.2.5        0.063     1     0.0     15.0     0.0   -37.5     0.0
  EC-6.2.6-(Y)    0.018     1     0.0     15.0     0.0     0.0     0.0
  EC-6.3.2 LTB    0.071     1     0.0     15.0     0.0   -37.5     0.0
   Torsion has not been considered in the design.

  ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 Belgian NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the Belgian NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1 = 2.578

C_2	=	1.554
k	=	0.5
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$29,909,000$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$5,599$
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	$5,934$
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1,712\ kN \cdot m$

From the Belgian NA, $\lambda_{LT,\ 0} = 0.2, \beta = 1.0$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2,149 \times (10)^3 \times 275}{1,712 \times (10)^6}} = 0.588$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,\ 0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34\times(0.589 - 0.2) + 1 \times 0.588^2] = 0.739$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.843$

From Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.588 - 0.8)^2\right] = 0.955$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.883 < \frac{1}{\lambda_{LT}^2} = 2.897$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.883 \times 2,149(10)^3 \times 275}{1.0} = 521.8\ \mathrm{kN \cdot m}$$

If Cl. 6.3.2.2 is used to determine χ_{LT}:

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.3 of Eurocode 3, $\alpha_{LT} = 0.21$.

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - 0.2) + \lambda_{LT}^2] = 0.5[1+0.21\times(0.588 - 0.2) +1\times0.588^2] = 0.713$$

$$\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \lambda_{LT}^2}\right)^{-1} = 0.895$$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.895 \times 2,149(10)^3 \times 275}{1.0 \times (10)^6} = 528.9\ \mathrm{kN \cdot m}$$

Results

Table 641: Comparison of results for EC3 Belgian NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,712	1,712.9	negligible	
Bending capacity, M_B (kN·m)	521.8	522.1	negligible	
Bending capacity, M_B (kN·m)	528.9	528.8	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Belgian NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Fully fixed span: `CMN 0.5`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The value of `C2 1.554` is specified
- The parameter KC instructs the program to calculate the value of k_c
- A second design parameter set is used with `MTH 1` to use Cl 6.3.2.2 to determine Φ_{LT}

> **Note:** The previously set parameters are still used as they are not specified again with a different value, which is the default behavior in STAAD.Pro batch design.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
```

```
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 8
C2 1.554 ALL
CMM 2 ALL
CMN 0.5 ALL
PY 275000 ALL
KC 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE EN 1993-1-1:2005
NA 8
MTH 1 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                                  -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.040         1
                        0.00           0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
```

```
  DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
     Section Class            :   CLASS 1
     Squash Load              :   4435.75
     Axial force/Squash load  :     0.000
     GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                   z-axis          y-axis
     Slenderness ratio (KL/r) :       36.2            66.1
     Compression Capacity     :     4078.2          3045.5
     Tension Capacity         :     4180.9          4180.9
     Moment Capacity          :      591.0           258.3
     Reduced Moment Capacity  :      591.0           258.3
     Shear Capacity           :     1301.9           821.3
  BUCKLING CALCULATIONS (units - kN,m)
     Lateral Torsional Buckling Moment       MB =  522.1
     co-efficients C1 & K : C1 =2.578 K =0.5, Effective Length= 5.000
     Lateral Torsional Buckling Curve : Curve b
     Elastic Critical Moment for LTB,              Mcr   =  1712.9
     Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
     Critical Load For Torsional Buckling,         NcrT  = 13898.0
     Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
     STAAD SPACE                                             -- PAGE NO.
4
  CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
   CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
  EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
  EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
  EC-6.3.2 LTB   0.040     1     0.0     25.0     0.0    20.8     0.0
   Torsion has not been considered in the design.

  ************** END OF TABULATED RESULT OF DESIGN **************
   39. PARAMETER 2
   40. CODE EN 1993-1-1:2005
   41. NA 8
   42. MTH 1 ALL
   43. CHECK CODE ALL
 STEEL DESIGN
                        STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     STAAD SPACE                                             -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.039        1
                        0.00           0.00          20.83        0.00
 =======================================================================
  MATERIAL DATA
     Grade of steel           =  USER
     Modulus of elasticity    =  205 kN/mm2
     Design Strength  (py)    =  275  N/mm2
  SECTION PROPERTIES (units - cm)
     Member Length =    500.00
     Gross Area =  161.30          Net Area =  161.30
                                   z-axis          y-axis
```

```
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4180.9          4180.9
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  528.8
      co-efficients C1 & K : C1 =2.578 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve a
      Elastic Critical Moment for LTB,              Mcr   =  1712.9
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                                -- PAGE NO.
6
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.039     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 Belgian NA - Tee Section

Calculate the bending capacity of a beam using an Tee section subject to a uniformly distributed load and varying end moments per the Belgian NA to EC3.

Details

The section is an Indian INST100 with $f_y = 300\ MPa$. The member is a 5 m, simply-supported span under a 2 kN/m uniform load. The left support has a -5.0 kN·m moment applied and the right support has a 2.5 kN·m moment applied.

Validation

Moment capacity:

$$M_{\text{ckd}} = \frac{W_{\text{ply}} f_y}{\gamma_{M0}} = \frac{45.5 \times (10)^3 \times 300}{1.0 \times (10)^6} = 13.7 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2} \left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j) \right]$$

where

C_1	=	1.0
C_2	=	1.0
C_3	=	1.0
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$64{,}421$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$\left(\frac{1}{1}\right)^2 \frac{30.76}{79.6} = 0.386$
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	$\frac{(1 \times 5{,}000)^2 \times 205{,}000 \times 7.33 \times (10)^4}{2.6 \times \pi^2 \times 205{,}000 \times 79.6 \times (10)^4} = 89{,}714$
$C_2 Z_g$	=	$1.0 \times 10 \;/\; 2 = 5$
Ψ_f	=	1 ($I_{ft} = 0$, as there is no tension flange present in the Tee section)
h_s	=	D - 0.5×t_f = 100 - 0.5 × 10 = 95 mm
z_j	=	$0.8 \times \Psi_f \times h_s / 2 = 0.8 \times 1 \times 95 \;/\; 2 = 38 \; mm$
$C_3 Z_j$	=	$1.0 \times 38 = 38$

Therefore, $M_{cr} = 1.0 \times 64{,}421 \left[\sqrt{0.386 + 89{,}714 + (5 - 38)^2} - (5 - 38) \right] = 21.5 \; kN{\cdot}m$

Results

Table 642:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	13.7	13.7	none	
Critical moment, M_{cr} (kN·m)	21.5	21.5	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Belgian NA - Tee Section.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Simply-supported span: `CMN 1.0`
- Varying end moments and a uniformly distributed load: `CMM 7`

- The values of `C1 1.0`, `C2 1.0`, and `C3 1.0` are specified

> **Note:** Since these parameters are specified, `MU` is *not* specified as it would be ignored in favor of the direct coefficient specifications. Refer to D5.D.10.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1564) for additional details.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 20-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISNT100
*1 TABLE ST ISNT150
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 2.5
1 MZ -5
MEMBER LOAD
1 UNI GY -2
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 8
CMM 7 ALL
C1 1 ALL
C2 1 ALL
C3 1 ALL
CMN 1 ALL
PY 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         *******************************************
```

```
                        NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   ISNT100     (INDIAN SECTIONS)
                           FAIL     EC-6.3.2 LTB       1.263         1
                       0.00           0.00        -10.03        2.08
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  300  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   19.20         Net Area =   19.20
                                     z-axis          y-axis
      Moment of inertia        :      180.000         79.600
      Plastic modulus          :       45.500         26.600
      Elastic modulus          :       25.210         15.920
      Shear Area               :       13.333         10.900
      Radius of gyration       :        3.062          2.036
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    576.00
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                     z-axis          y-axis
      Slenderness ratio (KL/r) :       163.3          245.6
      Compression Capacity     :       116.5           55.1
      Tension Capacity         :       497.7          497.7
      Moment Capacity          :        13.7            8.0
      Reduced Moment Capacity  :        13.7            8.0
      Shear Capacity           :       230.9          188.8
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =    7.9
      co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,              Mcr   =    21.5
      Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  4275.7
      Critical Load For Torsional-Flexural Buckling, NcrTF =    64.4
      STAAD PLANE                                                -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.735     1     0.0      0.3     0.0   -10.0     0.0
   EC-6.2.6-(Y)   0.029     1     0.0      5.5     0.0     2.5     0.0
   EC-6.3.2 LTB   1.263     1     0.0      0.3     0.0   -10.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 Belgian NA - Varying End Moments

Calculate the bending capacity of a beam using an I section subject to a concentrated load and varying end moments per the Belgian NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 10 kN/m uniform load. The left support has a -10.0 kN·m moment applied and the right support has a 8.0 kN·m moment applied.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	*2.578*
C_2	=	*1.554*
k_w	=	$k = 1$
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*7,477,200*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*22,394*
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	*23,737*
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}541\ kN \cdot m$

$$\mu = \frac{FL}{4M} = \frac{4 \times 5}{4 \times 10} = 0.5 > 0$$

From the graph, $C_1 = 1.184$, $C_2 = 0.17$

$$C_2 z_g = 0.17 \times 160 = 27.2$$

Therefore, $M_{cr} = 1{,}676\ kN \cdot m$

Results

Table 643:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,676	1,676.1	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Belgian NA - Varying End Moments-CMM8.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Belgian NA is specified using `NA 8`
- Simply-supported span: `CMN 1.0`
- Concentrated load with varying end moments: `CMM 8`
- The parameter `MU 0.5` is used for the case with varying end moments
- The value of `C2 0.17` is specified; based on calculation of μ

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 20-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -4
JOINT LOAD
2 MZ 8
```

```
1 MZ -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 8 ALL
CMM 8 ALL
MU 0.5 ALL
C2 0.17 ALL
CMN 1 ALL
PY 275000 ALL
*KC 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NBN EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NBN EN 1993-1-1 ANB : 2018
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.026        1
                        0.00           0.00        -14.00        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       4078.2          3045.5
      Tension Capacity         :       4180.9          4180.9
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
```

```
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  543.6
      co-efficients C1 & K : C1 =1.184 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1676.1
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                             -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.024     1     0.0      1.6     0.0   -14.0     0.0
   EC-6.2.6-(Y)   0.003     1     0.0     -2.4     0.0   -13.0     0.0
   EC-6.3.2 LTB   0.026     1     0.0      1.6     0.0   -14.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.10 Belgian National Annex to EC3* (on page 1563)

V. EC3 French NA - Channel Section with Conc Load

Calculate the bending capacity of a channel section beam under subject to a pair of concentrated load per the French NA to EC3

Details

The section is a C15X50, grade S275 steel. The member is a 6 m span, simply supported beam is subject to two concentrated loads of 10 kN each at the 1/3 points (2 m from either support).

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{1{,}123(10)^3 275}{1.0(10)^6} = 308.7 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2} \left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j) \right]$$

where

C_1	=	*1.046*
C_2	=	*0.42*
C_3	=	*1.0*
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*257,320*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*28,852*

$\frac{(kL)^2 GI_T}{\pi^2 EI_y}$ = $330{,}055$

C_2Z_g = $0.42 \times 0.5 \times 381 = 80.01$

β_f = $\frac{b_c^3 + t_c}{b_c^3 + t_c + b_t^3 + t_t} = 0.5$ (From SN030a-EN-EU)

Z_j = $0.4 \times h_s\,(2 \times \beta_f - 1) = 0$, since $\beta_f \geq 0.5$

C_3Z_j = $1.0\,(0) = 0$

Therefore, $M_{cr} = 1.046 \times 257{,}320 \left[\sqrt{28{,}852 + 330{,}055 + (80.01)^2} - (80.01)\right] = 141.2\ kN \cdot m$

Results

Table 644:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	308.7	308.7	none	
Critical moment, M_{cr} (kN·m)	141.2	142.2	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 French NA - Channel Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Simply-supported span: `CMN 1` and `C3 1`
- Concentrated loads at 1/3 points supports: `CMM 5`
- The value of `C2 0.42` is specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 15-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10 2
1 CON GY -10 4
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4
C2 0.42 ALL
C3 1 ALL
CMM 5 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                  -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   C15X50      (AISC SECTIONS)
                           PASS     EC-6.3.2 LTB       0.228         1
                        0.00            0.00        -20.00        2.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =   94.84          Net Area =   94.84
                                      z-axis           y-axis
      Moment of inertia        :    16815.750        457.855
      Plastic modulus          :     1122.514        133.391
      Elastic modulus          :      882.716         61.711
      Shear Area               :       20.800         69.290
      Radius of gyration       :       13.316          2.197
      Effective Length         :      600.000        600.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
```

```
      Section Class            :   CLASS 1
      Squash Load              :   2608.06
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        45.1           273.1
      Compression Capacity     :      2170.6           227.2
      Tension Capacity         :      2014.4          2014.4
      Moment Capacity          :       308.7            36.7
      Reduced Moment Capacity  :       308.7            36.7
      Shear Capacity           :       330.2          1097.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =   87.6
      co-efficients C1 & K : C1 =1.046 K =1.0, Effective Length= 6.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,              Mcr   =   142.2
      Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  4811.4
      Critical Load For Torsional-Flexural Buckling, NcrTF =  4544.2
      STAAD PLANE                                                 -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.2.5       0.065     1     0.0     10.0     0.0   -20.0     0.0
   EC-6.2.6-(Y)   0.009     1     0.0     10.0     0.0     0.0     0.0
   EC-6.3.2 LTB   0.228     1     0.0     10.0     0.0   -20.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 French NA - Column with Axial Load

Calculate the axial and bending capacities and interaction ratios of a column using the French NA to EC3.

Details

The 5 m tall column as a fixed base and is free to translate and rotate at the top. The column is subject to a 25 kN compressive load and moments of 5 kN·m about the X axis and 10 kN·m about the Z axis. The section is an HD320X127, grade S275 steel.

Validation

Bending Capacity

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149(10)^3 275}{1.0(10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554
$\frac{\pi^2 EI_y}{(kL)^2}$	=	$7,477,200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22,394$
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	$23,740$
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1,540.6\ kN \cdot m$

From the French NA, $\lambda_{LT,\ 0} = 0.2 + 0.1 \times b\ /\ h = 0.2 + 0.1 \times 300\ /\ 327 = 0.292, \beta = 1.0$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2,149 \times 10^3 \times 275}{1,540.6 \times 10^6}} = 0.619$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3,
$\alpha_{LT} = 0.4 - 0.2 \frac{b}{h} \lambda_{LT}{}^2 = 0.4 - 0.2 \frac{300}{327} (0.619)^2 = 0.323$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,\ 0}) + \beta \times \lambda_{LT}{}^2] = 0.5[1 + 0.323 \times (0.619 - 0.292) + 0.75 \times 0.619^2] = 0.746$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}{}^2 - \beta \lambda_{LT}{}^2}\right)^{-1} = 0.861$

So, $kc = 0.9$ and thus modification factor, $f = 1 - 0.5(1 - 0.9) \times [1 - 2(0.619 - 0.8)^2] = 0.953$

$$\chi_{LT,\text{mod}} = \chi_{LT}\ /\ f = 0.903 < 1/\lambda_{LT}{}^2 = 2.603$$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.903 \times 2,149(10)^3 \times 275}{1.0} = 533.8.\ \text{kN} \cdot \text{m}$$

Compression Capacity

Critical compressive values:

$$N_{\text{cry}} = \sqrt{\frac{\pi^2 EI_y}{(kl)^2}} = 7,477\ \text{kN}$$

$$N_{\text{crz}} = \sqrt{\frac{\pi^2 EI_z}{(kl)^2}} = 24,943\ \text{kN}$$

$$\lambda_{\text{cy}} = \sqrt{\frac{A \times f_y}{\text{N}_{\text{cry}}}} = 0.770$$

$$\lambda_{\text{cz}} = \sqrt{\frac{A \times f_y}{\text{N}_{\text{crz}}}} = 0.422$$

Determine χ_y

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for y-y axis, use curve "c". Hence, $\alpha = 0.49$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha (\lambda - 0.2) +\lambda^2)] = 0.936$

$$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.681 \quad \text{(governs)}$$

Determine χ_z

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for z-z axis, use curve "b". Hence, $\alpha = 0.34$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha (\lambda - 0.2) +\lambda^2)] = 0.627$

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.917 \quad \text{(does not govern)}$$

Compression capacity:

$$\chi_y \times N_{Rk} = 0.681 \times 7{,}477 = 3{,}020.2\ \text{kN}$$

Compression ratio: $25\ kN\ /\ 3{,}020.2\ kN = 0.008$

<u>Critical Axial Loads for Flexure and Flexural Torsional Buckling</u>

From NCCI document SN001a-EN-FU:

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 EI_w}{l_T^2}\right)$$

where

$$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2; \text{ here } y_0 = z_0 = 0$$

$i_y = 75.68\ mm$ and $i_z = 138.23\ mm$

$$= 24{,}835\ \text{mm}^2$$

$$N_{cr,T} = \frac{1}{24{,}835}\left[\frac{20{,}500 \times 225.1(10)^4}{2.6} + \frac{\pi^2(205{,}000)2.069(10)^{12}}{5{,}000^2}\right] = 13{,}889\ \text{kN}$$

$$N_{cr,TF} = \frac{i_0^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{(N_{cr,y} + N_{cr,T})^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_0^2}}\right]$$

$$= \frac{1}{2}\left[24{,}943 + 13{,}889 - \sqrt{(24{,}943 + 13{,}889)^2 - 4(24{,}943)(13{,}889)^1/_1}\right] = 13{,}889\ \text{kN}$$

<u>Interaction Check</u>

From Annex A, Table A.1 of EC3, we get the auxiliary terms:

$$\mu_z = \frac{1 - \frac{N_{Ed}}{N_{crz}}}{1 - \chi_z \frac{N_{Ed}}{N_{crz}}} = 1.000$$

$$\mu_y = \frac{1 - \frac{N_{Ed}}{N_{cry}}}{1 - x_y \frac{N_{Ed}}{N_{cry}}} = 0.999$$

$$\mathrm{w_z = w_{plz} / w_{elz}} = 1.116$$

$$\mathrm{w_y = w_{ply} / w_{ely}} = 1.525 > 1.5$$

The beam has uniform loading and fixed end supports and $\psi=1.0$.

$$C_{mz,0} = 0.79 + 0.21 \times 1 + 0.36\left(1 - 0.33\right)\frac{25}{24,\ 943} = 1.000 > 1 - \frac{N_{Ed}}{N_{cr,z}}$$

$$C_{my,0} = 0.79 + 0.21 \times 1 + 0.36\left(1 - 0.33\right)\frac{25}{4,\ 477} = 1.001 > 1 - \frac{N_{Ed}}{N_{cr,y}}$$

Determine λ_0: $C_1 = 1,\ C_2 = 0,\ M_{cr} = 1{,}606\ kN \cdot m$

$$\lambda_0 = \sqrt{\frac{2,\ 149 \times 10^3 \times 275}{1,\ 606 \times 10^6}} = 0.607$$

$$\lambda_{0,\min} = 0.2\sqrt{C_1}\sqrt[4]{\left(1 - \frac{N_{Ed}}{N_{cr,y}}\right) \times \left(1 - \frac{N_{Ed}}{N_{cr\mathrm{TF}}}\right)} = 0.321$$

$$a_{LT} = 1 - I_t / I_z = 0.993$$

$$\varepsilon_z = \frac{M_{z,Ed}}{N_{Ed}} \times \frac{A}{w_{elz}} = 3.350$$

$$C_{mz} = C_{mz,0} + (1 - C_{mz,0})\frac{\sqrt{\varepsilon_z a_{LT}}}{1 + \varepsilon_z a_{LT}} = 1.000$$

$$\mathrm{C_{my} = C_{my,0}} = 1.001$$

$$C_{mLT} = C_{mz}^2 \frac{a_{LT}}{\sqrt{\left(1 - \frac{N_{Ed}}{N_{cr,y}}\right) \times \left(1 - \frac{N_{Ed}}{N_{crT}}\right)}} = 0.995 < 1$$

So, $C_{mLT} = 1.0, \lambda_{max} = max.(\lambda_z,\lambda_y) = \lambda_y = 0.770$

$$b_{LT} = 0.5 a_{LT} \lambda_0^2 \frac{M_{yEd}}{x_{LT} \times M_{ply.Rd}} \times \frac{M_{zEd}}{M_{plz.Rd}} = 0.5 \times 0.993 \times 0.607^2 \frac{5}{0.909 \times 258.3} \times \frac{10}{591} \approx 0$$

$$\eta_{pl} = \frac{N_{Ed}}{N_{Rk} / \Gamma_{M1}} = \frac{25}{4,\ 436} = 0.006$$

$$C_{zz} = 1 + (w_z - 1)\left[\left(2 - \frac{1.6}{w_z} C_{mz}^2 \lambda_{\max} - \frac{1.6}{w_z} C_{mz}^2 \lambda_{\max}^2\right)\eta_{pl} - b_{LT}\right]$$

$$= 1 + (1.116 - 1)\left[\left(2 - \frac{1.6}{1.116} \times 1^2 \times 0.770 - \frac{1.6}{1.116} \times 1^2 \times 0.770^2\right)0.006 - 0\right] = 1.000$$

$$c_{LT} = 10 a_{LT} \frac{\lambda_0^2}{5 + \lambda_y^4} \times \frac{M_{z,Ed}}{C_{mz} x_{LT} M_{plz,Rd}} = 0.013$$

$$C_{zy} = 1 + (w_y - 1)\left[\left(2 - 14\frac{C_{my}{}^2 \lambda_{max}{}^2}{w_y{}^5}\right)\eta_{pl} - c_{LT}\right] = 0.996$$

$$C_{zy} > 0.6\sqrt{\frac{w_y}{w_z}} \times \frac{w_{ely}}{w_{ply}} = 0.456$$

$$d_{LT} = 2a_{LT}\frac{\lambda_0}{0.1 + \lambda_y{}^4} \times \frac{M_{z,Ed}}{C_{mz} \times \chi_{LT} \times M_{plz,Rd}} \times \frac{M_{y,Ed}}{C_{my} \times M_{ply,Rd}}$$

$$= 2 \times 0.993\frac{0.607}{0.1 + 0.770^4}\frac{10}{1 \times 0.903 \times 591}\frac{5}{1.000 \times 258.3} = 0.001$$

$$C_{yz} = 1 + (w_z - 1)\left[\left(2 - 14\frac{C_{mz}{}^2 \lambda_{max}{}^2}{w_z{}^5}\right)\eta_{pl} - d_{LT}\right] = 0.998$$

$$e_{LT} = 1.7a_{LT}\frac{\lambda_0}{0.1 + \lambda_y{}^4} \times \frac{M_{z,Ed}}{C_{mz} \times \chi_{LT} \times M_{plz,Rd}} = 1.7 \times 0.993\frac{0.607}{0.1 + 0.770^4}\frac{10}{1 \times 0.903 \times 591} = 0.043$$

$$C_{yy} = 1 + (w_y - 1)\left[\left(2 - \frac{1.6}{w_y}C_{my}{}^2\lambda_{max} - \frac{1.6}{w_y}C_{my}{}^2\lambda_{max}{}^2\right)\eta_{pl} - e_{LT}\right]$$

$$= 1 + (1.5 - 1)\left[\left(2 - \frac{1.6}{1.5} \times 1.001^2 \times 0.770 - \frac{1.6}{1.5} \times 1.001^2 \times 0.770^2\right)0.006 - 0.043\right] = 0.980$$

$W_{ely}/W_{ply} = 0.656 < C_{yy}$. So, $C_{yy} = 0.980$

Interaction factors:

$$K_{zz} = C_{mz}C_{mLT}\frac{\mu_z}{1 - \frac{N_{Ed}}{N_{crz}}}\frac{1}{C_{zz}} = 1.001$$

$$K_{zy} = C_{my}\frac{\mu_z}{1 - \frac{N_{Ed}}{N_{cry}}}\frac{1}{C_{zy}}0.6\sqrt{\frac{w_y}{w_z}} = 0.701$$

$$K_{yz} = C_{mz}C_{mLT}\frac{\mu_y}{1 - \frac{N_{Ed}}{N_{crz}}}\frac{1}{C_{yz}}0.6\sqrt{\frac{w_z}{w_y}} = 0.518$$

$$K_{yy} = C_{my}\frac{\mu_y}{1 - \frac{N_{Ed}}{N_{cry}}}\frac{1}{C_{yy}} = 1.023$$

Check for Clause 6.3.3-661:

$$\frac{N_{Ed}}{x_Z\frac{N_{Rk}}{\gamma M_1}} + K_{ZZ}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{zy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.9174 \times 4{,}436} + 1.001\frac{10}{0.903 \times 591} + 0.701\frac{5}{258.3} = 0.038$$

Check for Clause 6.3.3-662:

$$\frac{N_{Ed}}{x_y \frac{N_{Rk}}{\gamma M_1}} + K_{yz}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{yy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.6809 \times 4{,}436} + 0.518\frac{10}{0.903 \times 591} + 1.023\frac{5}{258.3} = 0.038$$

Results

Table 645:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,540.6	1,536.7	negligible	
Bending capacity, M_B (kN·m)	533.7	534.4	none	
Critical load for torsional buckling, $N_{cr,T}$ (kN)	13,889	13,898.0	negligible	
Critical load for torsional-flexural buckling, $N_{cr,TF}$	13,889	13,898.0	negligible	
Compression interaction, Cl. 6.3.1.1	0.008	0.008	none	
Bending and compression interaction, Cl 6.3.3-661	0.038	0.038	none	
Bending and compression interaction, Cl. 6.3.3-662	0.038	0.037	3%	Rounding difference in small numbers.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 French NA - Column with Axial Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Fixed support: `CMM 2.0`

- Fixed base and free at other end: `CMN 0.7`
- The value of `C2 1.554` is specified
- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4
CMM 2 ALL
C2 1.554 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
CMN 0.7 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   HD320X127   (EUROPEAN SECTIONS)
                          PASS     EC-6.3.3-661      0.038        1
                      25.00 C         5.00        -10.00       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2           66.1
      Compression Capacity     :      4078.2         3045.5
      Tension Capacity         :      3426.0         3426.0
      Moment Capacity          :       591.0          258.3
      Reduced Moment Capacity  :       591.0          258.3
      Shear Capacity           :      1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  534.4
      co-efficients C1 & K : C1 =2.570 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve :
      Elastic Critical Moment for LTB,              Mcr   =  1536.7
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                              -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE         RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.3.1.1     0.008     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-661   0.038     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-662   0.037     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.2 LTB   0.019     1    25.0      0.0      0.0   -10.0     5.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 French NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the French NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	1.35
C_2	=	0.59
$\frac{\pi^2 E I_y}{(kL)^2}$	=	7,477,200
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	22,394
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	23,737
$C_2 Z_g$	=	0.59×160 = 94.4

Therefore, $M_{cr} = 1{,}416\ kN·m$

From the French NA, $\lambda_{LT,0} = 0.2 + 0.1 \times b / h = 0.2 + 0.1 \times 300 / 327 = 0.292, \beta = 1.0$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}416 \times (10)^6}} = 0.646$

H/b = 320/300 = 1.067 < 2

$\alpha_{LT} = 0.4 - 0.2 \times b / h \times \lambda_{LT}^2 = 0.4 - 0.2 \times 300 / 327 \times 0.646^2 = 0.324$

From Cl. 6.3.2.3 of Eurocode3:

$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.324 \times (0.646 - 0.292) + 1 \times 0.646^2] = 0.766$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.849$

$$M_B = \frac{\chi_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.849 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 501.9 \text{ kN·m}$$

Results

Table 646:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,416	1,416	none	
Bending capacity, M_B (kN·m)	501.9	502.6	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 National Annex\French NA\EC3 French NA-I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Simply-supported span: `CMN 1`
- Simply-supported span with concentrated load at midspan: `CMM 3`

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 21-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -30
PERFORM ANALYSIS
```

```
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4
CMM 3 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.05.00.**                   *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    AUG 14, 2020                  *
             *           Time=     9: 1:39                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE
INPUT FILE: EC3 French NA-I Section with Conc Load.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 21-JUNE-20
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT METER KN
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 5 0 0
     9. MEMBER INCIDENCES
    10. 1 1 2
    11. DEFINE MATERIAL START
    12. ISOTROPIC STEEL
    13. E 2.05E+008
    14. POISSON 0.3
    15. DENSITY 76.8195
    16. ALPHA 1.2E-005
    17. DAMP 0.03
    18. END DEFINE MATERIAL
    19. MEMBER PROPERTY EUROPEAN
    20. 1 TABLE ST HD320X127
    21. CONSTANTS
    22. MATERIAL STEEL ALL
    23. SUPPORTS
    24. 1 2 PINNED
    25. LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
    26. MEMBER LOAD
    27. 1 CON GY -30
    28. PERFORM ANALYSIS
      STAAD PLANE                                              -- PAGE NO.
2
```

```
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          2  NUMBER OF MEMBERS       1
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       2
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   29. PARAMETER 1
   30. CODE EN 1993-1-1:2005
   31. NA 4
   32. CMM 3 ALL
   33. CMN 1 ALL
   34. FU 295000 ALL
   35. PY 275000 ALL
   36. TRACK 2 ALL
   37. CHECK CODE ALL
 STEEL DESIGN
                        STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.13 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.075
1
                        0.00           0.00         -37.50        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00         GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2           66.1
      Compression Capacity     :       4078.2         3045.5
      Tension Capacity         :       3426.0         3426.0
      Moment Capacity          :        591.0          258.3
```

```
      Reduced Moment Capacity  :        591.0          258.3
      Shear Capacity           :       1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  502.6
      co-efficients C1 & K : C1 =1.350 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve :
      Elastic Critical Moment for LTB,               Mcr   =  1416.0
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,          NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                                -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.2.5       0.063     1     0.0     15.0     0.0   -37.5     0.0
   EC-6.2.6-(Y)   0.018     1     0.0     15.0     0.0     0.0     0.0
   EC-6.3.2 LTB   0.075     1     0.0     15.0     0.0   -37.5     0.0
    Torsion and deflections have not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
    38. FINISH
      STAAD PLANE                                                -- PAGE NO.
5
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= AUG 14,2020   TIME=   9: 1:40 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *      Copyright (c) 1997-2017 Bentley Systems, Inc.       *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 French NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the French NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554
$\frac{\pi^2 EI_y}{(kL)^2}$	=	7,477,200
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	22,394
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	23,737
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}539\ kN·m$

From the French NA, $\lambda_{LT,0} = 0.2 + 0.1 \times b / h = 0.2 + 0.1 \times 300 / 327 = 0.292, \beta = 1.0$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}539 \times (10)^6}} = 0.620$

$H/b = 320/300 = 1.067 < 2$

$\alpha_{LT} = 0.4 - 0.2 \times b / h \times \lambda_{LT}^2 = 0.4 - 0.2 \times 300 / 327 \times 0.620^2 = 0.330$

From Cl. 6.3.2.3 of Eurocode3:

$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.330 \times (0.620 - 0.292) + 1 \times 0.620^2] = 0.746$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.861$

$k_c = 0.9$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.620 - 0.8)^2\right] = 0.953$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.903 < \frac{1}{\lambda_{LT}^2} = 2.603$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.903 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 533.8 \text{ kN·m}$$

Results

Table 647: Comparison of results for EC3 French NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,539	1,539.7	negligible	
Bending capacity, M_B (kN·m)	533.8	534.6	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 French NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The value of `C2 1.554` is specified
- The parameter KC specifies a value of 0.9 for k_c
- The yield strength is directly specified by `PY 275000`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-May-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
1 UNI GY -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4
CMM 2 ALL
PY 275000 ALL
KC 0.9 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                                  -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.039         1
                        0.00           0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4180.9          4180.9
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  534.6
      co-efficients C1 & K : C1 =2.570 K =1.0, Effective Length= 5.000
```

```
      Lateral Torsional Buckling Curve :
      Elastic Critical Moment for LTB,              Mcr   =  1539.7
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                             -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE         RATIO  LOAD      FX       VY       VZ      MZ      MY
   EC-6.2.5        0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)    0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB    0.039     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 French NA - Tee Section

Calculate the bending capacity of a beam using an Tee section subject to a uniformly distributed load and varying end moments per the French NA to EC3.

Details

The section is an Indian INST100 with $f_y = 300\ MPa$. The member is a 5 m, simply-supported span under a 2 kN/m uniform load. The left support has a -5.0 kN·m moment applied and the right support has a 2.5 kN·m moment applied.

Validation

Moment capacity:

$$M_{\mathrm{ckd}} = \frac{W_{\mathrm{ply}} f_y}{\gamma_{M0}} = \frac{45.5 \times (10)^3 \times 300}{1.0 \times (10)^6} = 13.7\ \mathrm{kN \cdot m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2} \left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j) \right]$$

where

C_1	=	1.0
C_2	=	1.0
C_3	=	1.0
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$64{,}421$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$\left(\frac{1}{1}\right)^2 \frac{30.76}{79.6} = 0.386$
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	$\frac{(1 \times 5{,}000)^2 \times 205{,}000 \times 7.33 \times (10)^4}{2.6 \times \pi^2 \times 205{,}000 \times 79.6 \times (10)^4} = 89{,}714$

C_2Z_g	=	$1.0 \times 10\ /\ 2 = 5$
Ψ_f	=	1 ($I_{ft} = 0$, as there is no tension flange present in the Tee section)
h_s	=	D - 0.5×t_f = 100 - 0.5 × 10 = 95 mm
z_j	=	$0.8 \times \Psi_f \times h_s / 2 = 0.8 \times 1 \times 95\ /\ 2 = 38\ mm$
C_3Z_j	=	$1.0 \times 38 = 38$

Therefore, $M_{cr} = 1.0 \times 64{,}421 \left[\sqrt{0.386 + 89{,}714 + (5 - 38)^2} - (5 - 38) \right] = 21.5\ kN \cdot m$

Results

Table 648:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	13.7	13.7	none	
Critical moment, M_{cr} (kN·m)	21.5	21.5	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 French NA - Tee Section.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Simply-supported span: `CMN 1.0`
- Varying end moments and a uniformly distributed load: `CMM 7`
- The values of `C1 1.0`, `C2 1.0`, and `C3 1.0` are specified

> **Note:** Since these parameters are specified, `MU` is *not* specified as it would be ignored in favor of the direct coefficient specifications. Refer to D5.D.10.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1564) for additional details.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 24-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
```

```
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISNT100
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 2.5
1 MZ -5
MEMBER LOAD
1 UNI GY -2
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4 ALL
CMM 7 ALL
C1 1 ALL
C2 1 ALL
C3 1 ALL
CMN 1 ALL
PY 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   ISNT100     (INDIAN SECTIONS)
                           FAIL     EC-6.3.2 LTB       1.263         1
                        0.00           0.00        -10.03        2.08
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  300  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   19.20          Net Area =   19.20
                                      z-axis          y-axis
      Moment of inertia       :      180.000          79.600
      Plastic modulus         :       45.500          26.600
      Elastic modulus         :       25.210          15.920
      Shear Area              :       13.333          10.900
```

```
    Radius of gyration       :       3.062          2.036
    Effective Length         :     500.000        500.000
 DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
    Section Class            :   CLASS 1
    Squash Load              :    576.00
    Axial force/Squash load  :     0.000
    GM0 :  1.00         GM1 :  1.00          GM2 :  1.25
                                   z-axis          y-axis
    Slenderness ratio (KL/r) :       163.3          245.6
    Compression Capacity     :       116.5           55.1
    Tension Capacity         :       497.7          497.7
    Moment Capacity          :        13.7            8.0
    Reduced Moment Capacity  :        13.7            8.0
    Shear Capacity           :       230.9          188.8
 BUCKLING CALCULATIONS (units - kN,m)
    Lateral Torsional Buckling Moment       MB =    7.9
    co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 5.000
    Lateral Torsional Buckling Curve : Curve d
    Elastic Critical Moment for LTB,              Mcr   =    21.5
    Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
    Critical Load For Torsional Buckling,         NcrT  =  4275.7
    Critical Load For Torsional-Flexural Buckling, NcrTF =    64.4
    STAAD PLANE                                              -- PAGE NO.
5
 CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
  CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
 EC-6.2.5       0.735     1     0.0      0.3     0.0   -10.0     0.0
 EC-6.2.6-(Y)   0.029     1     0.0      5.5     0.0     2.5     0.0
 EC-6.3.2 LTB   1.263     1     0.0      0.3     0.0   -10.0     0.0
  Torsion has not been considered in the design.

 ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 French NA - Varying End Mom CMM8

Calculate the bending capacity of a beam using an I section subject to a concentrated load and varying end moments per the French NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 10 kN/m uniform load. The left support has a -10.0 kN·m moment applied and the right support has a 8.0 kN·m moment applied.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

k_w	=	$k = 1$
$\frac{\pi^2 EI_y}{(kL)^2}$	=	7,477,200
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	22,394
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	23,737
$C_2 Z_g$	=	$0.17 \times 160 = 27.2$

$$\mu = \frac{FL}{4M} = \frac{4 \times 5}{4 \times 10} = 0.5 > 0$$

From the graph, $C_1 = 1.184$, $C_2 = 0.17$

$$C_2 z_g = 0.17 \times 160 = 27.2$$

Therefore, $M_{cr} = 1{,}676\ kN \cdot m$

Results

Table 649:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,676	1,676.5	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 French NA - Varying End Mom CMM8.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The French NA is specified using `NA 4`
- Simply-supported span: `CMN 1.0`
- Concentrated load with varying end moments: `CMM 8`
- The parameter `MU 0.5` is used for the case with varying end moments
- The value of `C2 0.17` is specified; based on calculation of μ

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 22-June-20
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -4
JOINT LOAD
2 MZ 8
1 MZ -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 4
CMM 8 ALL
MU 0.5 ALL
C2 0.17 ALL
CMN 1 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - NF EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NF EN 1993-1-1/NA
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                               -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.027        1
                       0.00            0.00        -14.00        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
```

```
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30         Net Area =  161.30
                                    z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.25
                                    z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4180.9          4180.9
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  516.2
      co-efficients C1 & K : C1 =1.184 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve :
      Elastic Critical Moment for LTB,              Mcr   =  1676.1
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                              -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE         RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5        0.024     1     0.0      1.6     0.0   -14.0     0.0
   EC-6.2.6-(Y)    0.003     1     0.0     -2.4     0.0   -13.0     0.0
   EC-6.3.2 LTB    0.027     1     0.0      1.6     0.0   -14.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.6 French National Annex to EC3* (on page 1537)

V. EC3 German NA - Built up Section with UDL

Calculate the bending capacity of a beam using a built-up I section subject to a uniformly distributed load per the German NA to EC3.

Details

The section built up from 300 mm × 20.5 mm flange plates and a 279 mm × 11.5 mm web. Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Section Properties

Moment of inertia about major axis,

$$I_z = 2\left[\frac{30 \times (2.05)^3}{12} + 30(2.05)\left(\frac{27.9 + 2.05}{2}\right)^2\right] + \frac{1.15(27.9)^3}{12} = 29{,}707 \text{ cm}^4$$

Moment of inertia about minor axis,

$$I_z = 2\left[\frac{2.05 \times (30)^3}{12}\right] + \frac{27.9(1.15)^3}{12} = 9{,}229 \text{ cm}^4$$

Elastic section modulus about major axis, $Z_z = \frac{29{,}707}{32/2} = 1{,}857\ cm^3$

Elastic section modulus about minor axis, $Z_y = \frac{9{,}229}{30/2} = 615.2\ cm^3$

Plastic section modulus about major axis,

$$W_{ply} = \frac{30(32)^2}{4} - \frac{(30 - 1.15)(27.9)^2}{4} = 2{,}066 \text{ cm}^3$$

Torsional constant,

$$J = 2\left[\frac{30(2.05)^3}{3}\right] + \frac{27.9(1.15)^3}{3} = 186.4 \text{ cm}^4$$

Warping constant,

$$I_w = \frac{(32 - 2.05)^2(30)^3 2.05}{24} = 2{,}068{,}700 \text{ cm}^6$$

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}066 \times (10)^3 \times 275}{1.0 \times (10)^6} = 568.1 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2}\left\{\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g\right\}$$

where

C_1	=	0.712
C_2	=	0.652
k	=	0.5
$\frac{\pi^2 EI_y}{(kL)^2}$	=	29,909,000
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	5,599
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	5,934
$C_2 Z_g$	=	0.652×160 = 104.3

Therefore, $M_{cr} = 966.7\ kN·m$

From the German NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}066 \times (10)^3 \times 275}{966.7 \times (10)^6}} = 0.767$

$H/b = 320/300 = 1.067 < 2$. So, use buckling curve "c". From Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.49$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - \lambda_{LT,0}) + \beta\times \lambda_{LT}^2] = 0.5[1 + 0.49\times(0.767 - 0.2) + 1 \times 0.767^2] = 0.810$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.785$

Calculate k_c: $k_c = \sqrt{\frac{1}{C_1}} = \sqrt{1/0.712} = 1.185 > 1.0$; use the value from Table 6.6 of EC3: $k_c = 0.9$.

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.767 - 0.8)^2\right] = 0.950$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.883 < \frac{1}{\lambda_{LT}^2} = 1.702$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.826 \times 2{,}066(10)^3 \times 275}{1.1} = 426.5 \text{ kN·m}$$

Results

Table 650: Comparison of results for EC3 German NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	568.1	568.1	none	
Critical moment, M_{cr} (kN·m)	966.7	967.3	negligible	
Bending capacity, M_B (kN·m)	426.5	426.5	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 German NA - Built up Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The German NA is specified using `NA 10`
- Fully fixed span: `CMN 0.5`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The value of `C2 0.652` is specified
- The parameter `SBLT 1` indicates a built-up section
- The parameter `KC 0` instructs the program to calculate the value of k_c

- The parameter `PY 275000` indicates an f_y of 275 MPa

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER KN
WIDE FLANGE
BUILTUP1
0.01613 0.32 0.0115 0.3 0.0205 0.0003082 9.239e-005 2.251e-006 0.00368 0.0082
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 BUILTUP1
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 10
C2 0.652 ALL
CMM 2 ALL
CMN 0.5 ALL
SBLT 1 ALL
KC 0 ALL
*MTH 1 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - DIN EN 1993-1-1:2010-12
                         ********************************************
                         NATIONAL ANNEX - DIN EN 1993-1-1/NA:2010-12
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
4
```

```
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
*** WARNING:CMN PARAM INVALID FOR NATIONAL ANNEX
NATIONAL ANNEX ONLY DEALS WITH END RESTRAINT FACTORS OF K = KW = 1.
HENCE WILL USE ANNEX F FROM DD ENV 1993-1-1:1992
      1 ST   BUILTUP1    (UPT)
                          PASS     EC-6.2.6-(Y)       0.049         1
                       0.00           0.00          20.83        0.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  USER
     Modulus of elasticity    =  205 kN/mm2
     Design Strength  (py)    =  275  N/mm2
  SECTION PROPERTIES (units - cm)
     Member Length =    500.00
     Gross Area =  161.30          Net Area =  161.30
                                     z-axis          y-axis
     Moment of inertia        :    30820.002        9239.001
     Plastic modulus          :     2065.718         931.725
     Elastic modulus          :     1926.250         615.933
     Shear Area               :       82.000          32.085
     Radius of gyration       :       13.823           7.568
     Effective Length         :      500.000         500.000
  DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
     Section Class            :   CLASS 1
     Squash Load              :   4435.75
     Axial force/Squash load  :     0.000
     GM0 :  1.00          GM1 :  1.10          GM2 :  1.25
                                     z-axis          y-axis
     Slenderness ratio (KL/r) :         36.2            66.1
     Compression Capacity     :       3707.4          2768.6
     Tension Capacity         :       4180.9          4180.9
     Moment Capacity          :        568.1           256.2
     Reduced Moment Capacity  :        568.1           256.2
     Shear Capacity           :       1301.9           509.4
  BUCKLING CALCULATIONS (units - kN,m)
     Lateral Torsional Buckling Moment       MB =  426.5
     co-efficients C1 & K : C1 =0.712 K =0.5, Effective Length= 5.000
     Lateral Torsional Buckling Curve : Curve c
     Elastic Critical Moment for LTB,              Mcr   =   967.3
     Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
     Critical Load For Torsional Buckling,          NcrT  = 13898.0
     Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
     STAAD SPACE                                              -- PAGE NO.
5
  CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
   CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
  EC-6.2.5       0.037     1     0.0     25.0     0.0    20.8     0.0
  EC-6.2.6-(Y)   0.049     1     0.0     25.0     0.0    20.8     0.0
  EC-6.3.2 LTB   0.049     1     0.0     25.0     0.0    20.8     0.0
   Torsion has not been considered in the design.

  ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 German NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the German NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	1.348
C_2	=	0.63
$\frac{\pi^2 EI_y}{(kL)^2}$	=	$7{,}477{,}200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22{,}394$
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	$23{,}737$
$C_2 Z_g$	=	$0.63 \times 160 = 100.8$

Therefore, $M_{cr} = 1{,}375\ kN\cdot m$

From the German NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}375 \times (10)^6}} = 0.656$

$H/b = 320/300 = 1.067 < 2$

So from Table 6.5 of Eurocode 3: $\alpha_{LT} = 0.34$

From Cl. 6.3.2.3 of Eurocode3:

$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34\times(0.656 - 0.4) + 0.75 \times 0.656^2] = 0.705$

$kc = 1$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.891$

Modification factor: $f = 1 - 0.5(1 - 1) \times [1 - 2(0.656 - 0.8)^2] = 1.0$

$$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.891 < \frac{1}{\lambda_{LT}{}^2} = 2.897, \text{OK}$$

$$M_B = \frac{x_{\mathrm{LT}} w_y f_y}{\gamma_{\mathrm{M1}}} = \frac{0.891 \times 2{,}149 \times (10)^3 \times 275}{1.1 \times (10)^6} = 478.9 \text{ kN·m}$$

Results

Table 651:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,375	1,376	negligible	
Bending capacity, M_B (kN·m)	478.9	478.9	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 German NA - I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The German NA is specified using `NA 10`
- Simply-supported span: `CMN 1`
- Simply-supported span with concentrated load at midspan: `CMM 3`
- User-supplied value of kc: `KC 0`

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 02-Aug-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -30
PERFORM ANALYSIS
PARAMETER 2
CODE EN 1993-1-1:2005
NA 10
C1 1.348 ALL
C2 0.63 ALL
CMM 3 ALL
CMN 1 ALL
KC 1 ALL
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - DIN EN 1993-1-1:2010-12
                         ********************************************
                         NATIONAL ANNEX - DIN EN 1993-1-1/NA:2010-12
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.078        1
                        0.00           0.00        -37.50        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.10          GM2 :  1.25
```

```
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      3707.4          2768.6
      Tension Capacity         :      3426.0          3426.0
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  478.9
      co-efficients C1 & K : C1 =1.348 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1376.0
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                               -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.063     1     0.0     15.0     0.0   -37.5     0.0
   EC-6.2.6-(Y)   0.018     1     0.0     15.0     0.0     0.0     0.0
   EC-6.3.2 LTB   0.078     1     0.0     15.0     0.0   -37.5     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 German NA - Column with Axial Load

Calculate the axial and bending capacities and interaction ratios of a column using the German NA to EC3.

Details

The 5 m tall column as a fixed base and is free to translate and rotate at the top. The column is subject to a 25 kN compressive load and moments of 5 kN·m about the X axis and 10 kN·m about the Z axis. The section is an HD320X127, grade S275 steel.

Validation

Bending Capacity

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149(10)^3 275}{1.0(10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554

$\dfrac{\pi^2 E I_y}{(kL)^2}$	=	7,477,200
$\left(\dfrac{k}{k_w}\right)^2 \dfrac{I_w}{I_y}$	=	22,394
$\dfrac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	23,740
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}540.6\ kN \cdot m$

From the German NA, $\lambda_{LT,\ 0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\dfrac{w_y f_y}{M_{cr}}} = \sqrt{\dfrac{2{,}149 \times 10^3 \times 275}{1{,}540.6 \times 10^6}} = 0.619$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - \lambda_{LT,\ 0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.619 - 0.4) + 0.75 \times 0.619^2] = 0.681$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.908$

$kc = {}^{1}\big/_{\sqrt{C_1}} = {}^{1}\big/_{\sqrt{2.578}} = 0.623$ and thus modification factor, $f = 1 - 0.5(1 - 0.623) \times [1 - 2(0.619 - 0.8)^2] = 0.824$

$$\chi_{LT,\text{mod}} = \chi_{LT} / f = 0.824 = 1.102 > 1$$

So, $\chi_{LT,mod} = 1.0$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{1.0 \times 2{,}149(10)^3 \times 275}{1.1} = 537.3 \text{ kN} \cdot \text{m}$$

<u>Compression Capacity</u>

Critical compressive values:

$$N_{\text{cry}} = \sqrt{\frac{\pi^2 E I_y}{(kl)^2}} = 7{,}477 \text{ kN}$$

$$N_{\text{crz}} = \sqrt{\frac{\pi^2 E I_z}{(kl)^2}} = 24{,}943 \text{ kN}$$

$N_{c,Rd} = A_g f_y\ /\ \gamma_{M0} = 16{,}130 \times 275\ /\ 1.0\ (10)^{-3} = 4{,}436\ kN$ for a Class 1 section.

Check the flexural buckling resistance per Cl. 6.3.1.1.

$N_{b,Rd} = \chi \times A \times f_y\ /\ \gamma_{M1}$ for a Class 1 section.

Buckling curves for I section are found in Table 6.2. Imperfection factor, α, values are found in Table 6.1.

Along Z

$$h/b = 1.067 < 2$$

$$t_f = 20.5 < 100$$

So, for the z-z axis, use curve “b”. Hence, *a = 0.34* (*EC3 Table 6.2*).

$$\lambda_{cz} = \sqrt{\frac{A \times f_y}{N_{crz}}} = \frac{L_{cr}}{i_z \times \lambda_1}$$

where

L_{cr}	=	5,000 mm
i_z	=	138.2 mm
λ_1	=	$93.9\varepsilon = 93.9\sqrt{\frac{235}{f_y}} = 93.9\sqrt{\frac{235}{275}} = 86.80$

Thus, $\lambda_{cz} = \frac{5,000}{138.2 \times 86.80} = 0.417$

$$\phi = 0.5[1+\alpha(\lambda_z - 0.2) + \lambda^2)] = 0.624$$

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}} = 0.919$$

Compression capacity about Z:

$$N_{b,Rd} = \chi_z \times A \times f_y / \gamma_{M1} = 0.919 \times 16,130 \times 275 / 1.1 = 3,707.5 \text{ kN}$$

Along Y

For the y-y axis, use curve “c”. Hence, *a = 0.49* (*EC3 Table 6.2*).

$$\lambda_{cz} = \sqrt{\frac{A \times f_y}{N_{crz}}} = \frac{L_{cr}}{i_z \times \lambda_1}$$

where

L_{cr}	=	5,000 mm
i_y	=	75.68 mm
λ_1	=	$93.9\varepsilon = 93.9\sqrt{\frac{235}{f_y}} = 93.9\sqrt{\frac{235}{275}} = 86.80$

Thus, $\lambda_{cy} = \frac{5,000}{75.68 \times 86.80} = 0.761$

$$\phi = 0.5[1+\alpha(\lambda_z - 0.2) + \lambda^2)] = 0.827$$

$$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}} = 0.687$$

Compression capacity about Z:

$$N_{b,Rd} = \chi_z \times A \times f_y / \gamma_{M1} = 0.687 \times 16,130 \times 275 / 1.1 = 2,768.6 \text{ kN}$$

Thus, the compression capacity, $N_{b,Rd} = 2,768.6\ kN$

Compression ratio: $25\ kN\ /\ 2,768.6\ kN = 0.009$

Critical Axial Loads for Flexure and Flexural Torsional Buckling

From NCCI document SN001a-EN-FU:

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 EI_w}{l_T^2}\right)$$

where

i_o^2	=	$i_y^2 + i_z^2 + y_o^2 + z_o^2$; here $y_0 = z_0 = 0$

$i_y = 75.68\ mm$ and $i_z = 138.23\ mm$

$$= 24{,}835\ \text{mm}^2$$

$$N_{cr,T} = \frac{1}{24{,}835}\left[\frac{20{,}500 \times 225.1(10)^4}{2.6} + \frac{\pi^2(205{,}000)2.069(10)^{12}}{5{,}000^2}\right] = 13{,}889\ \text{kN}$$

$$N_{cr,TF} = \frac{i_0^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{\left(N_{cr,y} + N_{cr,T}\right)^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_0^2}}\right]$$

$$= \frac{1}{2}\left[24{,}943 + 13{,}889 - \sqrt{(24{,}943 + 13{,}889)^2 - 4(24{,}943)(13{,}889)^1/_1}\right] = 13{,}889\ \text{kN}$$

<u>Interaction Check</u>

From Annex B, Table B.3 of EN 1993:1-1:2005,

$W_z = W_{plz} / W_{elz} = 2{,}149 / 1{,}926 = 1.116$

$W_y = W_{ply} / W_{ely} = 939 / 615.9 = 1.525 > 1.5;\ W_y = 1.5$

$\psi = 1.0$

So, $C_{mz} = 0.6 + 0.4\psi = 1.0$

$C_{mLT} = C_{mz} = 1.0$

$C_{my} = 0.6 + 0.4\psi = 1.0$

Interaction factors:

$$K_{zz} = C_{mz}\left[1 + (\lambda_z - 0.2)\frac{N_{Ed}}{x_z \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (0.422 - 0.2)\frac{25}{0.917 \times 4,\ 436}\right] = 1.001$$

$$K_{yy} = C_{my}\left[1 + (2\lambda_y - 0.6)\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (2 \times 0.770 - 0.6)\frac{25}{0.681 \times 4,\ 436}\right] = 1.008$$

$$K_{yz1} = 0.6K_{zz} = 0.601$$

$$K_{yz2} = 1 + \frac{0.1 \times \lambda_y}{C_{mLT} - 0.25}\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}} = 1 - \frac{0.1 \times 0.770}{(1 - 0.25)}\frac{25}{0.681 \times 4,\ 436} = 0.999$$

So, $K_{yz} = maximum(K_{yz1}, K_{yz2}) = 0.999$

$K_{zy} = 0.6 \times K_{yy} = 0.6(1.008) = 0.605$

Check for Clause 6.3.3-661:

$$\frac{N_{Ed}}{x_Z\frac{N_{Rk}}{\gamma M_1}} + K_{ZZ}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{zy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25 \times 1.1}{0.9174 \times 4{,}436} + 1.001\frac{10 \times 1.1}{1.0 \times 591} + 0.605\frac{5 \times 1.1}{258.3} = 0.038$$

Check for Clause 6.3.3-662:

$$\frac{N_{Ed}}{x_y\frac{N_{Rk}}{\gamma M_1}} + K_{yz}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{yy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25 \times 1.1}{0.681 \times 4{,}436} + 0.999\frac{10 \times 1.1}{1.0 \times 591} + 1.008\frac{5 \times 1.1}{258.3} = 0.049$$

Results

Table 652:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,540.6	1,541.5	negligible	
Bending capacity, M_B (kN·m)	537.3	537.2	negligible	
Critical load for torsional buckling, $N_{cr,T}$ (kN)	13,889	13,898.0	negligible	
Critical load for torsional-flexural buckling, $N_{cr,TF}$	13,889	13,898.0	negligible	
Compression interaction, Cl. 6.3.1.1	0.009	0.009	none	
Bending and compression interaction, Cl 6.3.3-661	0.038	0.038	none	
Bending and compression interaction, Cl. 6.3.3-662	0.049	0.049	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 German NA - Column with Axial Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The German NA is specified using `NA 10`
- Fixed support: `CMM 2.0`
- Fixed base and free at other end: `CMN 0.7`
- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Aug-2021
```

```
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 10
CMM 2 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
CMN 0.7 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - DIN EN 1993-1-1:2010-12
                         ********************************************
                         NATIONAL ANNEX - DIN EN 1993-1-1/NA:2010-12
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.3-662       0.049        1
                      25.00 C          5.00        -10.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
```

```
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30         Net Area =  161.30
                                     z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006
      GM0 :  1.00          GM1 :  1.10          GM2 :  1.25
                                     z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2           66.1
      Compression Capacity     :       3707.4         2768.6
      Tension Capacity         :       3426.0         3426.0
      Moment Capacity          :        591.0          258.3
      Reduced Moment Capacity  :        591.0          258.3
      Shear Capacity           :       1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  537.2
      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1541.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                             -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.3.1.1     0.009     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-661   0.038     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.3-662   0.049     1    25.0      0.0     0.0   -10.0     5.0
   EC-6.3.2 LTB   0.019     1    25.0      0.0     0.0   -10.0     5.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 German NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the German NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	0.712
C_2	=	0.652
k	=	0.5
$\frac{\pi^2 EI_y}{(kL)^2}$	=	$29{,}909{,}000$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$5{,}599$
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	$5{,}934$
$C_2 Z_g$	=	$0.652 \times 160 = 104.3$

Therefore, $M_{cr} = 966.7\ kN·m$

From the German NA, $\lambda_{LT,\ 0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{966.7 \times (10)^6}} = 0.782$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $a_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,\ 0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.782 - 0.2) + 1 \times 0.782^2] = 0.794$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta \lambda_{LT}^2}\right)^{-1} = 0.827$

So, $kc = 1 / \sqrt{C_1} = 1 / \sqrt{0.712} = 1.182 > 1$ but this value cannot be used, so from Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.782 - 0.8)^2\right] = 0.950$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.871 < \frac{1}{\lambda_{LT}^2} = 1.635$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.871 \times 2{,}149(10)^3 \times 275}{1.1} = 467.7 \text{ kN·m}$$

Results

Table 653: Comparison of results for EC3 German NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	966.7	967.3	negligible	
Bending capacity, M_B (kN·m)	467.7	467.8	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 German NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The German NA is specified using `NA 10`
- Fully fixed span: `CMN 0.5`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The value of `C2 0.652` is specified
- The parameter `KC 0` instructs the program to calculate the value of k_c
- A second design parameter set is used with `MTH 1` to use Cl 6.3.2.2 to determine Φ_{LT}

> **Note:** The previously set parameters are still used as they are not specified again with a different value, which is the default behavior in STAAD.Pro batch design.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 1-Aug-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
```

```
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 10
C2 0.652 ALL
CMM 2 ALL
CMN 0.5 ALL
FU 300000 ALL
KC 0 ALL
*MTH 1 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - DIN EN 1993-1-1:2010-12
                         ********************************************
                         NATIONAL ANNEX - DIN EN 1993-1-1/NA:2010-12
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *** WARNING:CMN PARAM INVALID FOR NATIONAL ANNEX
 NATIONAL ANNEX ONLY DEALS WITH END RESTRAINT FACTORS OF K = KW = 1.
 HENCE WILL USE ANNEX F FROM DD ENV 1993-1-1:1992
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.045         1
                        0.00           0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
```

```
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.10          GM2 :  1.25
                                       z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       3707.4          2768.6
      Tension Capacity         :       3484.1          3484.1
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  467.8
      co-efficients C1 & K : C1 =0.712 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =   967.3
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                               -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.045     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Malaysian NA - Channel Section with Conc Load

Calculate the bending capacity of a channel section beam under subject to a pair of concentrated load per the Malaysian NA to EC3

Details

The section is a C15X50, grade S275 steel. The member is a 6 m span, simply supported beam is subject to two concentrated loads of 10 kN each at the 1/3 points (2 m from either support).

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{1{,}123(10)^3 275}{1.0(10)^6} = 308.7 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2} \left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j) \right]$$

where

C_1	=	*1.046*
C_2	=	*0.42*
C_3	=	*1.0*

$$\frac{\pi^2 EI_y}{(kL)^2} = 257{,}320$$

$$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y} = 28{,}852$$

$$\frac{(kL)^2 GI_T}{\pi^2 EI_y} = 330{,}055$$

$$C_2 Z_g = 0.42 \times 0.5 \times 381 = 80.01$$

$$\beta_f = \frac{b_c{}^3 + t_c}{b_c{}^3 + t_c +- b_t{}^3 + t_t} = 0.5 \text{ (From SN030a-EN-EU)}$$

$$Z_j = 0.4 \times h_s\,(2 \times \beta_f - 1) = 0\text{, since } \beta_f \geq 0.5$$

$$C_3 Z_j = 1.0\,(0) = 0$$

Therefore, $M_{cr} = 1.046 \times 257{,}320 \left[\sqrt{28{,}852 + 330{,}055 + (80.01)^2} - (80.01)\right] = 141.2\ kN{\cdot}m$

Results

Table 654:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	308.7	308.7	none	
Critical moment, M_{cr} (kN·m)	141.2	142.2	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Malaysian NA - Channel Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Malaysian NA is specified using `NA 9`
- Simply-supported span: `CMN 1` and `C3 1`
- Concentrated loads at 1/3 points supports: `CMM 5`
- The value of `C2 0.42` is specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 25-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
```

```
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10 2
1 CON GY -10 4
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 9
C2 0.42 ALL
C3 1 ALL
CMM 5 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
*KC 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - MS EN 1993-1-1:2010
                        ********************************************
                        NATIONAL ANNEX - NA to MS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   C15X50      (AISC SECTIONS)
                           PASS     EC-6.3.2 LTB       0.186         1
                        0.00           0.00         -20.00        2.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =   94.84          Net Area =   94.84
                                      z-axis          y-axis
```

```
      Moment of inertia        :    16815.750        457.855
      Plastic modulus          :     1122.514        133.391
      Elastic modulus          :      882.716         61.711
      Shear Area               :       20.800         69.290
      Radius of gyration       :       13.316          2.197
      Effective Length         :      600.000        600.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   2608.06
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.20
                                       z-axis          y-axis
      Slenderness ratio (KL/r) :          45.1          273.1
      Compression Capacity     :        2170.6          227.2
      Tension Capacity         :        2098.3         2098.3
      Moment Capacity          :         308.7           36.7
      Reduced Moment Capacity  :         308.7           36.7
      Shear Capacity           :         330.2         1097.9
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  107.3
      co-efficients C1 & K : C1 =1.046 K =1.0, Effective Length= 6.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,              Mcr   =   142.2
      Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  4811.4
      Critical Load For Torsional-Flexural Buckling, NcrTF =  4544.2
      STAAD PLANE                                             -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.065     1     0.0     10.0     0.0   -20.0     0.0
   EC-6.2.6-(Y)   0.009     1     0.0     10.0     0.0     0.0     0.0
   EC-6.3.2 LTB   0.186     1     0.0     10.0     0.0   -20.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Malaysian NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the Malaysian NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	1.348
C_2	=	0.63
$\frac{\pi^2 EI_y}{(kL)^2}$	=	$7,477,200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22,394$
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	$23,737$
$C_2 Z_g$	=	$0.63 \times 160 = 100.8$

Therefore, $M_{cr} = 1,375\ kN \cdot m$

From the Malaysian NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2,149 \times (10)^3 \times 275}{1,375 \times (10)^6}} = 0.656$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.656 - 0.4) + 1 \times 0.656^2] = 0.705$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.891$

So, $kc = 1 / \sqrt{C_1} = 1 / \sqrt{1.348} = 0.861$

Modification factor: $f = 1 - 0.5(1 - 0.861) \times [1 - 2(0.656 - 0.8)^2] = 0.934$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.955 < \frac{1}{\lambda_{LT}^2} = 2.897$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.955 \times 2,149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 564.3\ \mathrm{kN \cdot m}$$

Results

Table 655:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,375	1,376	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Bending capacity, M_B (kN·m)	564.3	564.3	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Malaysian NA - I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Malaysian NA is specified using `NA 9`
- Simply-supported span: `CMN 1`
- Simply-supported span with concentrated load at midspan: `CMM 3`
- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 23-MAR-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -30
PERFORM ANALYSIS
PARAMETER 2
CODE EN 1993-1-1:2005
NA 9
CMM 3 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - MS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to MS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.066         1
                        0.00          0.00         -37.50        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.20
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2           66.1
      Compression Capacity     :       4078.2         3045.5
      Tension Capacity         :       3568.8         3568.8
      Moment Capacity          :        591.0          258.3
      Reduced Moment Capacity  :        591.0          258.3
      Shear Capacity           :       1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  564.3
      co-efficients C1 & K : C1 =1.348 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1376.0
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,          NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                                -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
```

```
   EC-6.2.5        0.063     1     0.0     15.0     0.0   -37.5     0.0
   EC-6.2.6-(Y)    0.018     1     0.0     15.0     0.0     0.0     0.0
   EC-6.3.2 LTB    0.066     1     0.0     15.0     0.0   -37.5     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Malaysian NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the Malaysian NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	*0.712*
C_2	=	*0.652*
k	=	*0.5*
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*29,909,000*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*5,599*
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	*5,934*
$C_2 Z_g$	=	$0.652 \times 160 = 104.3$

Therefore, $M_{cr} = 966.7\ kN\cdot m$

From the Malaysian NA, $\lambda_{LT,0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{966.7 \times (10)^6}} = 0.782$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $a_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.782 - 0.2) + 1 \times 0.782^2] = 0.794$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}{}^{2} - \beta\lambda_{LT}{}^{2}}\right)^{-1} = 0.827$

So, $kc = 1 / \sqrt{C_1} = 1 / \sqrt{0.712} = 1.182 > 1$ but this value cannot be used, so from Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.782 - 0.8)^2\right] = 0.950$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.871 < \frac{1}{\lambda_{LT}{}^{2}} = 1.635$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.871 \times 2{,}149(10)^3 \times 275}{1.0} = 514.4 \text{ kN}\cdot\text{m}$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	966.7	967.3	negligible	
Bending capacity, M_B (kN·m)	514.4	514.5	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Europe\EC3 Malaysian NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Malaysian NA is specified using `NA 9`
- Fully fixed span: `CMN 0.5`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The parameter `KC 0` instructs the program to calculate the value of k_c

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
```

```
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 9
CMM 2 ALL
CMN 0.5 ALL
KC 0 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - MS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to MS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *** WARNING:CMN PARAM INVALID FOR NATIONAL ANNEX
 NATIONAL ANNEX ONLY DEALS WITH END RESTRAINT FACTORS OF K = KW = 1.
 HENCE WILL USE ANNEX F FROM DD ENV 1993-1-1:1992
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.040         1
                        0.00            0.00          20.83       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
```

```
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.20
                                   z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2           66.1
      Compression Capacity     :      4078.2         3045.5
      Tension Capacity         :      4355.1         4355.1
      Moment Capacity          :       591.0          258.3
      Reduced Moment Capacity  :       591.0          258.3
      Shear Capacity           :      1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  514.5
      co-efficients C1 & K : C1 =0.712 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =   967.3
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                              -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.040     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Malaysian NA - Tee Section

Calculate the bending capacity of a beam using an Tee section subject to a uniformly distributed load and varying end moments per the Malaysian NA to EC3.

Details

The section is an Indian INST100 with $f_y = 300\ MPa$. The member is a 5 m, simply-supported span under a 2 kN/m uniform load. The left support has a -5.0 kN·m moment applied and the right support has a 2.5 kN·m moment applied.

Validation

Moment capacity:

$$M_{\text{ckd}} = \frac{W_{\text{ply}} f_y}{\gamma_{M0}} = \frac{45.5 \times (10)^3 \times 300}{1.0 \times (10)^6} = 13.7\ \text{kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2} \left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j) \right]$$

where

C_1	=	1.0
C_2	=	1.0
C_3	=	1.0
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$64{,}421$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$\left(\frac{1}{1}\right)^2 \frac{30.76}{79.6} = 0.386$
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	$\frac{(1 \times 5{,}000)^2 \times 205{,}000 \times 7.33 \times (10)^4}{2.6 \times \pi^2 \times 205{,}000 \times 79.6 \times (10)^4} = 89{,}714$
$C_2 Z_g$	=	$1.0 \times 10 \;/\; 2 = 5$
Ψ_f	=	1 ($I_{ft} = 0$, as there is no tension flange present in the Tee section)
h_s	=	D - 0.5×t_f = 100 - 0.5 × 10 = 95 mm
z_j	=	$0.8 \times \Psi_f \times h_s / 2 = 0.8 \times 1 \times 95 \;/\; 2 = 38\ mm$
$C_3 Z_j$	=	$1.0 \times 38 = 38$

Therefore, $M_{cr} = 1.0 \times 64{,}421 \left[\sqrt{0.386 + 89{,}714 + (5 - 38)^2} - (5 - 38) \right] = 21.5\ kN \cdot m$

Results

Table 656:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	13.7	13.7	none	
Critical moment, M_{cr} (kN·m)	21.5	21.5	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Malaysian NA - Tee Section.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Malaysian NA is specified using `NA 9`
- Simply-supported span: `CMN 1.0`
- Varying end moments and a uniformly distributed load: `CMM 7`
- The values of `C1 1.0`, `C2 1.0`, and `C3 1.0` are specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 24-Mar-21
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISNT100
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 2.5
1 MZ -5
MEMBER LOAD
1 UNI GY -2
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 9
CMM 7 ALL
C1 1 ALL
C2 1 ALL
C3 1 ALL
CMN 1 ALL
PY 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - MS EN 1993-1-1:2010
                        ********************************************
                        NATIONAL ANNEX - NA to MS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   ISNT100     (INDIAN SECTIONS)
                           FAIL     EC-6.3.2 LTB       1.064        1
                       0.00           0.00         -10.03        2.08
 =======================================================================
```

```
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  300  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   19.20          Net Area =   19.20
                                    z-axis          y-axis
      Moment of inertia        :     180.000          79.600
      Plastic modulus          :      45.500          26.600
      Elastic modulus          :      25.210          15.920
      Shear Area               :      13.333          10.900
      Radius of gyration       :       3.062           2.036
      Effective Length         :     500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    576.00
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00          GM2 :  1.20
                                    z-axis          y-axis
      Slenderness ratio (KL/r) :       163.3           245.6
      Compression Capacity     :       116.5            55.1
      Tension Capacity         :       518.4           518.4
      Moment Capacity          :        13.7             8.0
      Reduced Moment Capacity  :        13.7             8.0
      Shear Capacity           :       230.9           188.8
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =    9.4
      co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,               Mcr   =    21.5
      Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
      Critical Load For Torsional Buckling,          NcrT  =  4275.7
      Critical Load For Torsional-Flexural Buckling, NcrTF =    64.4
      STAAD PLANE                                             -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.735     1     0.0      0.3     0.0   -10.0     0.0
   EC-6.2.6-(Y)   0.029     1     0.0      5.5     0.0     2.5     0.0
   EC-6.3.2 LTB   1.064     1     0.0      0.3     0.0   -10.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. EC3 Polish NA - Column with Axial Load

Calculate the axial and bending capacities and interaction ratios of a column using the Polish NA to EC3.

Details

The 5 m tall column as a fixed base and is free to translate and rotate at the top. The column is subject to a 25 kN compressive load and moments of 5 kN·m about the X axis and 10 kN·m about the Z axis. The section is an HD320X127, grade S275 steel.

Validation

Bending Capacity

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149(10)^3 275}{1.0(10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$7{,}477{,}200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22{,}394$
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	$23{,}740$
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}540.6 \text{ kN·m}$

From the Polish NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times 10^3 \times 275}{1{,}540.6 \times 10^6}} = 0.619$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1 + \alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.619 - 0.4) + 0.75 \times 0.619^2] = 0.681$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.908$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.908 \times 2{,}149(10)^3 \times 275}{1.0} = 536.8 \text{ kN·m}$$

Compression Capacity

Critical compressive values:

$$N_{cry} = \sqrt{\frac{\pi^2 E I_y}{(kl)^2}} = 7{,}477 \text{ kN}$$

$$N_{crz} = \sqrt{\frac{\pi^2 E I_z}{(kl)^2}} = 24{,}943 \text{ kN}$$

$$\lambda_{cy} = \sqrt{\frac{A \times f_y}{N_{cry}}} = 0.770$$

$$\lambda_{cz} = \sqrt{\frac{A \times f_y}{N_{crz}}} = 0.422$$

Determine χ_y

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for y-y axis, use curve "c". Hence, $\alpha = 0.49$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha\ (\lambda - 0.2) + \lambda^2)] = 0.936$

$$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.681 \text{ (governs)}$$

Determine χ_z

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for z-z axis, use curve "b". Hence, $\alpha = 0.34$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha\ (\lambda - 0.2) + \lambda^2)] = 0.627$

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.917 \text{ (does not govern)}$$

Compression capacity:

$$\chi_y \times N_{Rk} = 0.681 \times 7{,}477 = 3{,}020.2 \text{ kN}$$

Compression ratio: $25\ kN\ /\ 3{,}020.2\ kN = 0.008$

Critical Axial Loads for Flexure and Flexural Torsional Buckling

From NCCI document SN001a-EN-FU:

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 EI_w}{l_T^2}\right)$$

where

$$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2; \text{ here } y_0 = z_0 = 0$$

$i_y = 75.68\ mm$ and $i_z = 138.23\ mm$

$$= 24{,}835 \text{ mm}^2$$

$$N_{cr,T} = \frac{1}{24{,}835}\left[\frac{20{,}500 \times 225.1(10)^4}{2.6} + \frac{\pi^2(205{,}000)2.069(10)^{12}}{5{,}000^2}\right] = 13{,}889 \text{ kN}$$

$$N_{cr,TF} = \frac{i_0^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{\left(N_{cr,y} + N_{cr,T}\right)^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_0^2}}\right]$$

$$= \frac{1}{2}\left[24{,}943 + 13{,}889 - \sqrt{(24{,}943 + 13{,}889)^2 - 4(24{,}943)(13{,}889)^1/_1}\right] = 13{,}889 \text{ kN}$$

Interaction Check

From Annex B, Table B.3 of EN 1993:1-1:2005,

$W_z = W_{plz} / W_{elz} = 2{,}149 / 1{,}926 = 1.116$

$W_y = W_{ply} / W_{ely} = 939 / 615.9 = 1.525 > 1.5; W_y = 1.5$

$\psi = 1.0$

So, $C_{mz} = 0.6 + 0.4\psi = 1.0$

$C_{mLT} = C_{mz} = 1.0$

$C_{my} = 0.6 + 0.4\psi = 1.0$

Interaction factors:

$$K_{zz} = C_{mz}\left[1 + \left(\lambda_z - 0.2\right)\frac{N_{Ed}}{x_z \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (0.422 - 0.2)\frac{25}{0.681 \times 4,\ 436}\right] = 1.001$$

$$K_{yy} = C_{my}\left[1 + \left(2\lambda_y - 0.6\right)\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (2 \times 0.770 - 0.6)\frac{25}{0.681 \times 4,\ 436}\right] = 1.008$$

$$K_{yz1} = 0.6K_{zz} = 0.601$$

$$K_{yz2} = 1 + \frac{0.1 \times \lambda_y}{C_{mLT} - 0.25}\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}} = 1 - \frac{0.1 \times 0.770}{(1 - 0.25)}\frac{25}{0.681 \times 4,\ 436} = 0.999$$

So, $K_{yz} = maximum(K_{yz1}, K_{yz2}) = 0.999$

$K_{zy} = 0.6 \times K_{yy} = 0.6(1.008) = 0.605$

Check for Clause 6.3.3-661:

$$\frac{N_{Ed}}{x_Z\frac{N_{Rk}}{\gamma M_1}} + K_{ZZ}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{zy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.9174 \times 4{,}436} + 1.001\frac{10}{1.0 \times 591} + 0.605\frac{5}{258.3} = 0.035$$

Check for Clause 6.3.3-662:

$$\frac{N_{Ed}}{x_y\frac{N_{Rk}}{\gamma M_1}} + K_{yz}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{yy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.6809 \times 4{,}436} + 0.999\frac{10}{1.0 \times 591} + 1.008\frac{5}{258.3} = 0.045$$

<u>Check for PN-NA 20.2</u>

About the Z-Z axis:

$$\eta = {}^{N_{\mathrm{Ed}}}\!/\!_{N_{\mathrm{Rd}}} = \frac{25\ \mathrm{kN}}{4{,}436\ \mathrm{kN}} = 0.0056$$

$$m_z = {}^{M_{\mathrm{zEd}}}\!/\!_{M_{\mathrm{zRd}}} = \frac{10\ \mathrm{kN \cdot m}}{591\ \mathrm{kN \cdot m}} = 0.0169$$

$$m_y = {}^{M_{\mathrm{yEd}}}\!/\!_{M_{\mathrm{yRd}}} = \frac{5\ \mathrm{kN \cdot m}}{258.3\ \mathrm{kN \cdot m}} = 0.0194$$

$$\Delta_0 = 0.1 + 0.2\left({}^{W_{\mathrm{plz}}}\!/\!_{W_{\mathrm{elz}}} - 1\right) = 0.1 + 0.2(1.116 - 1) = 0.123$$

Per the Polish NA, the following must be satisfied:

$$\frac{\eta}{x_z} + \frac{C_{\mathrm{mz}} \times m_z}{x_{\mathrm{LT}}} + C_{\mathrm{my}} \times m_y \le 1 - \Delta_0$$

$$\frac{0.0056}{0.917} + \frac{1 \times 0.0169}{0.908} + 1 \times 0.0194 = 0.044 \leq 1 - 0.123 = 0.877$$

The ratio of the inequality sides is then: $0.044 \ / \ 0.877 = 0.050$

About the Y-Y axis:

$$\Delta_0 = 0.1 + 0.2\left({}^{W_{\text{ply}}}\!\big/_{W_{\text{ely}}} - 1\right) = 0.1 + 0.2(1.525 - 1) = 0.205$$

Per the Polish NA, the following must be satisfied:

$$\frac{\eta}{x_y} + \frac{C_{\text{my}} \times m_y}{x_{\text{LT}}} + C_{\text{mz}} \times m_y \leq 1 - \Delta_0$$

$$\frac{0.0056}{0.6809} + \frac{1 \times 0.0194}{0.908} + 1 \times 0.0169 = 0.046 \leq 1 - 0.20 = 0.80$$

The ratio of the inequality sides is then: $0.046 \ / \ 0.80 = 0.058$

Results

Table 657:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,540.5	1,541.5	negligible	
Bending capacity, M_B (kN·m)	591.0	591.0	none	
Critical load for torsional buckling, $N_{cr,T}$ (kN)	13,889	13,898.0	negligible	
Critical load for torsional-flexural buckling, $N_{cr,TF}$	13,889	13,898.0	negligible	
Compression interaction, Cl. 6.3.1.1	0.008	0.008	none	
Bending and compression interaction, Cl 6.3.3-661	0.035	0.036	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Bending and compression interaction, Cl. 6.3.3-662	0.045	0.046	none	
PN-NA 20.2	0.058	0.058	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Polish NA - Column with Axial Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Polish NA is specified using `NA 6`
- Fixed support: `CMM 2.0`
- Fixed base and free at other end: `CMN 0.7`
- D5.D.8.7 Clause 6.3.3(5) – Interaction factors kyy, kyz, kzy, and kzz (on page 1555) performed by the program per the Polish NA: `PLG 1`
- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Feb-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320x127
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 6 ALL
```

```
CMM 2 ALL
FU 295000 ALL
PY 275000 ALL
CMN 0.7 ALL
PLG 1 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - PN-EN 1993-1-1:2006
                         ********************************************
                         NATIONAL ANNEX - NA FOR PN-EN 1993-1-1:2006
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.3          0.058         1
                                    PN-NA 20.2
                      25.00 C          5.00        -10.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006
      GM0 :  1.00         GM1 :  1.00          GM2 :  0.97
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       4078.2          3045.5
      Tension Capacity         :       4435.8          4435.8
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  536.8
      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
```

```
      Elastic Critical Moment for LTB,               Mcr   =  1541.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,          NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                              -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY       VZ      MZ       MY
   EC-6.3.1.1     0.008     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-661   0.036     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-662   0.046     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.2 LTB   0.019     1    25.0      0.0      0.0   -10.0     5.0
   ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NA FOR PN-EN 1993-1-1:2006 ]
(units- kN,m):
   EC CLAUSE           NA-CLAUSE        RATIO   LOAD      FX       VY
VZ     MZ       MY
   EC-6.3.3      PN-NA 20.2            0.058     1      25.0      0.0
0.0   -10.0     5.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.8 Polish National Annex to EC3* (on page 1550)

V. EC3 Polish NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the Polish NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	1.683
C_2	=	1.645
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$7{,}477{,}200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22{,}394$

$$\frac{(kL)^2 GI_t}{\pi^2 EI_y} = 23{,}737$$

$$C_2 Z_g = 1.645 \times 160 = 263.2$$

Therefore, $M_{cr} = 962.9\ kN \cdot m$

From the Polish NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{962.9 \times (10)^6}} = 0.783$

$$H/b = 320/300 = 1.067 < 2$$

So from Table 6.3 of Eurocode 3, $a_{LT} = 0.21$.

From Cl. 6.3.2.2 of Eurocode3:

$$\Phi_{LT} = 0.5\left[1 + a_{LT}\left(\lambda_{LT} - 0.2\right) + \lambda_{LT}^2\right] = 0.5\left[1 + 0.21\left(0.783 - 0.2\right) + (0.783)^2\right] = 0.868$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \lambda_{LT}^2}\right)^{-1} = \frac{1}{0.868 + \sqrt{(0.868)^2 - (0.783)^2}} = 0.805$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.805 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 475.7\ \mathrm{kN \cdot m}$$

Results

Table 658:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	962.9	963.4	negligible	
Bending capacity, M_B (kN·m)	475.7	475.8	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Polish NA - I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Polish NA is specified using `NA 6`
- The C_1 and C_2 values are directly specified: `C1 1.683` and `C2 1.645`
- Fixed-fixed span with concentrated load at midspan: `CMM 4`

- Use Cl. 6.3.2.2 to evaluate χ_{LT}: `MTH 1`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Feb-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -30
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 6 ALL
C1 1.683 ALL
C2 1.645 ALL
CMM 4 ALL
MTH 1 ALL
FU 295000 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - PN-EN 1993-1-1:2006
                        ********************************************
                        NATIONAL ANNEX - NA FOR PN-EN 1993-1-1:2006
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                            -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.039        1
```

```
                        0.00            0.00          18.75         0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                     z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00         GM1 :  1.00          GM2 :  0.97
                                     z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2           66.1
      Compression Capacity     :      4078.2         3045.5
      Tension Capacity         :      4435.8         4435.8
      Moment Capacity          :       591.0          258.3
      Reduced Moment Capacity  :       591.0          258.3
      Shear Capacity           :      1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  475.8
      co-efficients C1 & K : C1 =1.683 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve a
      Elastic Critical Moment for LTB,              Mcr   =   963.4
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                               -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.032     1     0.0     15.0     0.0    18.8     0.0
   EC-6.2.6-(Y)   0.018     1     0.0     15.0     0.0    18.8     0.0
   EC-6.3.2 LTB   0.039     1     0.0     15.0     0.0    18.8     0.0
   ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NA FOR PN-EN 1993-1-1:2006 ]
(units- kN,m):
   EC CLAUSE           NA-CLAUSE        RATIO   LOAD      FX       VY
VZ     MZ       MY
    Torsion has not been considered in the design.
                        _________________________
   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.8 Polish National Annex to EC3* (on page 1550)

V. EC3 Polish NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the Polish NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 GI_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554
$\frac{\pi^2 EI_y}{(kL)^2}$	=	7,477,200
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	22,394
$\frac{(kL)^2 GI_t}{\pi^2 EI_y}$	=	23,737
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}539 \ kN·m$

From the Polish NA, $\lambda_{LT,0} = 0.4, \beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}539 \times (10)^6}} = 0.620$

$H/b = 320/300 = 1.067 < 2$

So, from Table 6.5 of Eurocode 3, $a_{LT} = 0.34$ (buckling curve “b”)

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5\left[1 + a_{LT}\left(\lambda_{LT} - \lambda_{LT,0}\right) + \beta \times \lambda_{LT}^2\right] = 0.5\left[1 + 0.34\left(0.620 - 0.4\right) + 0.75 \times (0.620)^2\right] = 0.681$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta \times \lambda_{LT}^2}\right)^{-1} = \frac{1}{0.681 + \sqrt{(0.681)^2 - 0.75(0.620)^2}} = 0.908$

For the Polish NA, the value of k_c is evaluated as:

$$k_c = \sqrt{C_{\mathrm{mLT}}}$$

where

a_s	=	M_s / M_h = -10.42 / 20.83 = -0.5 for $-1 \le a_s \le 1$; per Table B.3 of Annex B of EC3.
ψ	=	1 for $0 \le \psi \le 1$
C_{mLT}	=	$0.1 - 0.8 \times a_s = 0.5 > 0.4$, thus use $C_{mLT} = 0.5$

So, $k_c = \sqrt{0.5} = 0.707$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.707)\left[1 - 2(0.620 - 0.8)^2\right] = 0.863$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = \frac{0.908}{0.863} = 1.05 > 1$, thus use $\chi_{LT,mod} = 1$

$$M_B = \frac{x_{\mathrm{LT}} w_y f_y}{\gamma_{\mathrm{M1}}} = \frac{1.0 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591\ \mathrm{kN \cdot m}$$

Results

Table 659: Comparison of results for EC3 French NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,539	1,541.5	negligible	
Bending capacity, M_B (kN·m)	591.0	591.0	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Polish NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Polish NA is specified using `NA 6`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The value of `C2 1.554` is specified
- The yield strength is directly specified by `PY 275000`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 1-Feb-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 6
C2 1.554 ALL
CMM 2 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.2.5           0.035         1
                        0.00            0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
```

```
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                   z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4435.8          4435.8
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  591.0
      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1541.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                             -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.035     1     0.0     25.0     0.0    20.8     0.0
   ADDITIONAL CHECKS AS PER NATIONAL ANNEX [NA FOR PN-EN 1993-1-1:2006 ]
(units- kN,m):
   EC CLAUSE           NA-CLAUSE        RATIO   LOAD      FX       VY
VZ     MZ       MY
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.8 Polish National Annex to EC3* (on page 1550)

V. EC3 Singapore NA - Channel Section with Conc Load

Calculate the bending capacity of a channel section beam under subject to a pair of concentrated load per the Singaporean NA to EC3

Details

The section is a C15X50, grade S275 steel. The member is a 6 m span, simply supported beam is subject to two concentrated loads of 10 kN each at the 1/3 points (2 m from either support).

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{1{,}123(10)^3 275}{1.0(10)^6} = 308.7 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2}\left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j)\right]$$

where

C_1	=	1.046
C_2	=	0.42
C_3	=	1.0
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$257,320$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$28,852$
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	$330,055$
$C_2 Z_g$	=	$0.42 \times 0.5 \times 381 = 80.01$
β_f	=	$\frac{b_c^3 + t_c}{b_c^3 + t_c +\text{-}\ b_t^3 + t_t} = 0.5$ (From SN030a-EN-EU)
Z_j	=	$0.4 \times h_s (2 \times \beta_f - 1) = 0$, since $\beta_f \geq 0.5$
$C_3 Z_j$	=	$1.0\ (0) = 0$

Therefore, $M_{cr} = 1.046 \times 257,320\left[\sqrt{28,852 + 330,055 + (80.01)^2} - (80.01)\right] = 141.2\ kN \cdot m$

Results

Table 660:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	308.7	308.7	none	
Critical moment, M_{cr} (kN·m)	141.2	142.2	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Singapore NA - Channel Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`
- Simply-supported span: `CMN 1` and `C3 1`
- Concentrated loads at 1/3 points supports: `CMM 5`
- The value of `C2 0.42` is specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 03-Jul-20
```

```
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
*2 FIXED BUT FX FZ MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10 2
1 CON GY -10 4
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 7
C2 0.42 ALL
C3 1 ALL
CMM 5 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
 STEEL DESIGN
                        STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                        ********************************************
                        NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   C15X50      (AISC SECTIONS)
                           PASS     EC-6.3.2 LTB       0.186         1
```

```
                        0.00            0.00         -20.00        2.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  USER
     Modulus of elasticity    =  205 kN/mm2
     Design Strength  (py)    =  275  N/mm2
  SECTION PROPERTIES (units - cm)
     Member Length =    600.00
     Gross Area =   94.84          Net Area =   94.84
                                    z-axis          y-axis
     Moment of inertia        :    16815.750        457.855
     Plastic modulus          :     1122.514        133.391
     Elastic modulus          :      882.716         61.711
     Shear Area               :       20.800         69.290
     Radius of gyration       :       13.316          2.197
     Effective Length         :      600.000        600.000
  DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
     Section Class            :   CLASS 1
     Squash Load              :   2608.06
     Axial force/Squash load  :     0.000
     GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                    z-axis          y-axis
     Slenderness ratio (KL/r) :        45.1          273.1
     Compression Capacity     :      2170.6          227.2
     Tension Capacity         :      2289.1         2289.1
     Moment Capacity          :       308.7           36.7
     Reduced Moment Capacity  :       308.7           36.7
     Shear Capacity           :       330.2         1097.9
  BUCKLING CALCULATIONS (units - kN,m)
     Lateral Torsional Buckling Moment       MB =  107.3
     co-efficients C1 & K : C1 =1.046 K =1.0, Effective Length= 6.000
     Lateral Torsional Buckling Curve : Curve d
     Elastic Critical Moment for LTB,              Mcr   =   142.2
     Compression buckling curves:     z-z:  Curve c   y-y:  Curve c
     Critical Load For Torsional Buckling,         NcrT  =  4811.4
     Critical Load For Torsional-Flexural Buckling, NcrTF =  4544.2
     STAAD PLANE                                                -- PAGE NO.
5
  CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
   CLAUSE         RATIO  LOAD     FX       VY       VZ      MZ      MY
  EC-6.2.5        0.065     1     0.0     10.0     0.0   -20.0     0.0
  EC-6.2.6-(Y)    0.009     1     0.0     10.0     0.0     0.0     0.0
  EC-6.3.2 LTB    0.186     1     0.0     10.0     0.0   -20.0     0.0
   Torsion has not been considered in the design.

  ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. EC3 Singapore NA - Column with Axial Load

Calculate the axial and bending capacities and interaction ratios of a column using the Singaporean NA to EC3.

Details

The 5 m tall column as a fixed base and is free to translate and rotate at the top. The column is subject to a 25 kN compressive load and moments of 5 kN·m about the X axis and 10 kN·m about the Z axis. The section is an HD320X127, grade S275 steel.

Validation

Bending Capacity

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149(10)^3 275}{1.0(10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	2.578
C_2	=	1.554
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$7{,}477{,}200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22{,}394$
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	$23{,}740$
$C_2 Z_g$	=	$1.554 \times 160 = 248.6$

Therefore, $M_{cr} = 1{,}540.6$ $kN·m$

From the Singaporean NA, $\lambda_{LT,0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times 10^3 \times 275}{1{,}540.6 \times 10^6}} = 0.619$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.619 - 0.4) + 0.75 \times 0.619^2] = 0.681$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.908$

So, $kc = 1 / \sqrt{C_1} = 1 / \sqrt{2.578} = 0.623$ and thus modification factor, $f = 0.824$

$\chi_{LT,mod} = \chi_{LT} / f = 1.10 > 1$, thus use $\chi_{LT} = 1$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{1.0 \times 2{,}149(10)^3 \times 275}{1.0} = 591.0 \text{ kN·m}$$

Compression Capacity

Critical compressive values:

$$N_{cry} = \sqrt{\frac{\pi^2 E I_y}{(kl)^2}} = 7{,}477 \text{ kN}$$

$$N_{crz} = \sqrt{\frac{\pi^2 E I_z}{(kl)^2}} = 24{,}943 \text{ kN}$$

$$\lambda_{cy} = \sqrt{\frac{A \times f_y}{N_{cry}}} = 0.770$$

$$\lambda_{cz} = \sqrt{\frac{A \times f_y}{N_{crz}}} = 0.422$$

Determine χ_y

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for y-y axis, use curve "c". Hence, $\alpha = 0.49$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha (\lambda - 0.2) +\lambda^2)] = 0.936$

$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.681$ (governs)

Determine χ_z

$h/b = 1.067 < 1.2$ and $t_f = 20.5\ mm < 100\ mm$

So, for z-z axis, use curve "b". Hence, $\alpha = 0.34$ [Table 6.2, EC3]

So, $\phi = 0.5[1+\alpha (\lambda - 0.2) +\lambda^2)] = 0.627$

$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}}\ 0.917$ (does not govern)

Compression capacity:

$$\chi_y \times N_{Rk} = 0.681 \times 7{,}477 = 3{,}020.2 \text{ kN}$$

Compression ratio: $25\ kN\ /\ 3{,}020.2\ kN = 0.008$

<u>Critical Axial Loads for Flexure and Flexural Torsional Buckling</u>

From NCCI document SN001a-EN-FU:

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 E I_w}{l_T^2}\right)$$

where

$i_o^2 = i_y^2 + i_z^2 + y_o^2 + z_o^2$; here $y_0 = z_0 = 0$

$i_y = 75.68\ mm$ and $i_z = 138.23\ mm$

$= 24{,}835 \text{ mm}^2$

$$N_{cr,T} = \frac{1}{24{,}835}\left[\frac{20{,}500 \times 225.1(10)^4}{2.6} + \frac{\pi^2(205{,}000)2.069(10)^{12}}{5{,}000^2}\right] = 13{,}889 \text{ kN}$$

$$N_{cr,TF} = \frac{i_0^2}{2\left(i_y^2 + i_z^2\right)}\left[N_{cr,y} + N_{cr,T} - \sqrt{\left(N_{cr,y} + N_{cr,T}\right)^2 - 4N_{cr,y}N_{cr,T}\frac{i_y^2 + i_z^2}{i_0^2}}\right]$$

$$= \tfrac{1}{2}\left[24{,}943 + 13{,}889 - \sqrt{(24{,}943 + 13{,}889)^2 - 4(24{,}943)(13{,}889)^1/_1}\right] = 13{,}889 \text{ kN}$$

<u>Interaction Check</u>

From Annex B, Table B.3 of EN 1993:1-1:2005,

$W_z = W_{plz} / W_{elz} = 2{,}149 / 1{,}926 = 1.116$

$W_y = W_{ply} / W_{ely} = 939 / 615.9 = 1.525 > 1.5$; $W_y = 1.5$

$\psi = 1.0$

So, $C_{mz} = 0.6 + 0.4\psi = 1.0$

$C_{mLT} = C_{mz} = 1.0$

$C_{my} = 0.6 + 0.4\psi = 1.0$

Interaction factors:

$$K_{zz} = C_{mz}\left[1 + (\lambda_z - 0.2)\frac{N_{Ed}}{x_z \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (0.422 - 0.2)\frac{25}{0.681 \times 4,\ 436}\right] = 1.001$$

$$K_{yy} = C_{my}\left[1 + (2\lambda_y - 0.6)\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}}\right] = 1\left[1 + (2 \times 0.770 - 0.6)\frac{25}{0.681 \times 4,\ 436}\right] = 1.008$$

$$K_{yz1} = 0.6K_{zz} = 0.601$$

$$K_{yz2} = 1 + \frac{0.1 \times \lambda_y}{C_{mLT} - 0.25}\frac{N_{Ed}}{x_y \times N_{Rk} \times \gamma_{M1}} = 1 - \frac{0.1 \times 0.770}{(1 - 0.25)}\frac{25}{0.681 \times 4,\ 436} = 0.999$$

So, $K_{yz} = maximum(K_{yz1}, K_{yz2}) = 0.999$

$K_{zy} = 0.6 \times K_{yy} = 0.6(1.008) = 0.605$

Check for Clause 6.3.3-661:

$$\frac{N_{Ed}}{x_Z\frac{N_{Rk}}{\gamma M_1}} + K_{ZZ}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{zy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.9174 \times 4{,}436} + 1.001\frac{10}{1.0 \times 591} + 0.605\frac{5}{258.3} = 0.035$$

Check for Clause 6.3.3-662:

$$\frac{N_{Ed}}{x_y\frac{N_{Rk}}{\gamma M_1}} + K_{yz}\frac{M_{z,Ed}}{x_{LT}\frac{M_{z,Rk}}{\gamma M_1}} + K_{yy}\frac{M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma M_1}} = \frac{25}{0.6809 \times 4{,}436} + 0.999\frac{10}{1.0 \times 591} + 1.008\frac{5}{258.3} = 0.045$$

Results

Table 661:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,540.5	1,541.5	negligible	
Bending capacity, M_B (kN·m)	591.0	591.0	none	
Critical load for torsional buckling, $N_{cr,T}$ (kN)	13,889	13,898.0	negligible	
Critical load for torsional-flexural buckling, $N_{cr,TF}$	13,889	13,898.0	negligible	
Compression interaction, Cl. 6.3.1.1	0.008	0.008	none	
Bending and compression interaction, Cl 6.3.3-661	0.035	0.035	none	
Bending and compression interaction, Cl. 6.3.3-662	0.045	0.045	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Europe\EC3 Singapore NA - Column with Axial Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`
- Fixed support: `CMM 2.0`
- Fixed base and free at other end: `CMN 0.7`
- The value of `C2 1.554` is specified

- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Jul-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -25 MX 5 MZ 10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 7
CMM 2 ALL
GM0 1 ALL
C2 1.554 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
CMN 0.7 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   HD320X127   (EUROPEAN SECTIONS)
                          PASS     EC-6.3.3-662       0.045        1
                      25.00 C         5.00        -10.00       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                        z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.006
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                        z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2           66.1
      Compression Capacity     :       4078.2         3045.5
      Tension Capacity         :       3893.2         3893.2
      Moment Capacity          :        591.0          258.3
      Reduced Moment Capacity  :        591.0          258.3
      Shear Capacity           :       1301.9          821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  591.0
      co-efficients C1 & K : C1 =2.578 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1541.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                                -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE         RATIO  LOAD     FX       VY       VZ      MZ      MY
   EC-6.3.1.1     0.008     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.2.9.1     0.020     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-661   0.035     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.3-662   0.045     1    25.0      0.0      0.0   -10.0     5.0
   EC-6.3.2 LTB   0.017     1    25.0      0.0      0.0   -10.0     5.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. EC3 Singapore NA - I Section with Conc Load

Calculate the bending capacity of a beam subject to a concentrated load per the Singaporean NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 30 kN concentrated load at midpsan.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	1.348
C_2	=	0.63
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$7{,}477{,}200$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$22{,}394$
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	$23{,}737$
$C_2 Z_g$	=	$0.63 \times 160 = 100.8$

Therefore, $M_{cr} = 1{,}375\ kN \cdot m$

From the Singaporean NA, $\lambda_{LT,0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{1{,}375 \times (10)^6}} = 0.656$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $\alpha_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT} (\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34 \times (0.656 - 0.4) + 1 \times 0.656^2] = 0.705$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.891$

So, $kc = 1 / \sqrt{C_1} = 1 / \sqrt{1.348} = 0.861$ but this value cannot be used, so from Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1 - 0.5\left(1 - 0.861\right) \times \left[1 - 2(0.656 - 0.8)^2\right] = 0.934$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.955 < \frac{1}{\lambda_{LT}^2} = 2.897$, OK

$$M_B = \frac{\chi_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.955 \times 2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 564.3 \text{ kN}\cdot\text{m}$$

Results

Table 662:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,375	1,376	negligible	
Bending capacity, M_B (kN·m)	564.3	564.3	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Europe\EC3 Singapore NA - I Section with Conc Load.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`
- Simply-supported span: `CMN 1`
- Simply-supported span with concentrated load at midspan: `CMM 3`
- Program calculated kc per Table 6.6 of EN 1993-1-1:2005: `KC 0`

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 12-Jul-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
*2 FIXED BUT FX FZ MZ
LOAD 1 LOADTYPE None  TITLE  DL
MEMBER LOAD
```

```
1 CON GY -30
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 7
CMM 3 ALL
CMN 1 ALL
FU 295000 ALL
PY 275000 ALL
KC 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                                -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.066         1
                        0.00           0.00         -37.50        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       4078.2          3045.5
      Tension Capacity         :       3893.2          3893.2
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
```

```
      Lateral Torsional Buckling Moment       MB =  564.3
      co-efficients C1 & K : C1 =1.348 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1376.0
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                             -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE         RATIO  LOAD      FX       VY      VZ      MZ      MY
   EC-6.2.5        0.063     1     0.0     15.0     0.0   -37.5     0.0
   EC-6.2.6-(Y)    0.018     1     0.0     15.0     0.0     0.0     0.0
   EC-6.3.2 LTB    0.066     1     0.0     15.0     0.0   -37.5     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. EC3 Singapore NA - I Section with UDL

Calculate the bending capacity of a beam using an I section subject to a uniformly distributed load per the Singaporean NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, fixed-fixed span subject to a 10 kN/m uniform load.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

C_1	=	*0.712*
C_2	=	*0.652*
k	=	*0.5*
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*29,909,000*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*5,599*

$$\frac{(kL)^2 GI_t}{\pi^2 EI_y} = 5{,}934$$

$$C_2 Z_g = 0.652 \times 160 = 104.3$$

Therefore, $M_{cr} = 966.7\ kN \cdot m$

From the Singaporean NA, $\lambda_{LT,0} = 0.4$, $\beta = 0.75$

So, $\lambda_{LT} = \sqrt{\frac{w_y f_y}{M_{cr}}} = \sqrt{\frac{2{,}149 \times (10)^3 \times 275}{966.7 \times (10)^6}} = 0.782$

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.5 of Eurocode 3, $a_{LT} = 0.34$.

From Cl. 6.3.2.3 of Eurocode3:

$$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - \lambda_{LT,0}) + \beta \times \lambda_{LT}^2] = 0.5[1 + 0.34\times(0.782 - 0.2) + 1 \times 0.782^2] = 0.794$$

So, $\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\lambda_{LT}^2}\right)^{-1} = 0.827$

So, $kc = 1/\sqrt{C_1} = 1/\sqrt{0.712} = 1.182 > 1$ but this value cannot be used, so from Table 6.6 of the Eurocode 3, $k_c = 0.9$

Modification factor: $f = 1 - 0.5(1 - k_c)\left[1 - 2(\lambda_{LT} - 0.8)^2\right] = 1 - 0.5(1 - 0.9)\left[1 - 2(0.782 - 0.8)^2\right] = 0.950$

$\chi_{LT,mod} = \frac{\chi_{LT}}{f} = 0.871 < \frac{1}{\lambda_{LT}^2} = 1.635$, OK

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.871 \times 2{,}149(10)^3 \times 275}{1.0} = 514.4\ \mathrm{kN \cdot m}$$

If Cl. 6.3.2.2 is used to determine χ_{LT}:

$H/b = 320/300 = 1.067 < 2$. So, from Table 6.3 of Eurocode 3, $a_{LT} = 0.21$.

$$\Phi_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT} - 0.2) + \lambda_{LT}^2] = 0.5[1+0.21\times(0.782 - 0.2) + 1\times0.782^2] = 0.867$$

(Cl. 6.3.2.2)

$$\chi_{LT} = \left(\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \lambda_{LT}^2}\right)^{-1} = 0.806$$

$$M_B = \frac{x_{LT} w_y f_y}{\gamma_{M1}} = \frac{0.806 \times 2{,}149(10)^3 \times 275}{1.0 \times (10)^6} = 476.3\ \mathrm{kN \cdot m}$$

Results

Table 663: Comparison of results for EC3 Singaporean NA - I Section with UDL

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	966.7	967.3	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Bending capacity, M_B (kN·m)	514.4	514.5	negligible	
Bending capacity, M_B (kN·m)	476.3	476.3	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Singapore NA - I Section with UDL.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`
- Fully fixed span: `CMN 0.5`
- Uniformly distributed load w/ fixed-fixed supports: `CMM 2`
- The parameter KC instructs the program to calculate the value of k_c
- A second design parameter set is used with `MTH 1` to use Cl 6.3.2.2 to determine Φ_{LT}

> **Note:** The previously set parameters are still used as they are not specified again with a different value, which is the default behavior in STAAD.Pro batch design.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-June-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -10
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
```

```
CODE EN 1993-1-1:2005
NA 7
CMM 2 ALL
CMN 0.5 ALL
KC 0 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE EN 1993-1-1:2005
NA 7
CMM 2 ALL
CMN 0.5 ALL
KC 0 ALL
MTH 1 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *** WARNING:CMN PARAM INVALID FOR NATIONAL ANNEX
 NATIONAL ANNEX ONLY DEALS WITH END RESTRAINT FACTORS OF K = KW = 1.
 HENCE WILL USE ANNEX F FROM DD ENV 1993-1-1:1992
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.040         1
                        0.00            0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004       9239.001
      Plastic modulus          :     2149.000        939.100
      Elastic modulus          :     1926.250        615.933
      Shear Area               :       81.998         51.728
      Radius of gyration       :       13.823          7.568
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
```

```
      GM0 :  1.00         GM1 :  1.00         GM2 :  1.10
                                    z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4435.8          4435.8
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  514.5
      co-efficients C1 & K : C1 =0.712 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =   967.3
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                              -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.040     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.
                        _________________________
   ************** END OF TABULATED RESULT OF DESIGN **************
    39. PARAMETER 2
    40. CODE EN 1993-1-1:2005
    41. NA 7
    42. CMM 2 ALL
    43. CMN 0.5 ALL
    44. KC 0 ALL
    45. MTH 1 ALL
    46. PY 275000 ALL
    47. TRACK 2 ALL
    48. CHECK CODE ALL
 STEEL DESIGN
                        STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                        ********************************************
                        NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD SPACE                                              -- PAGE NO.
6
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *** WARNING:CMN PARAM INVALID FOR NATIONAL ANNEX
 NATIONAL ANNEX ONLY DEALS WITH END RESTRAINT FACTORS OF K = KW = 1.
 HENCE WILL USE ANNEX F FROM DD ENV 1993-1-1:1992
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.044         1
                        0.00            0.00          20.83        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
```

```
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                     z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                     z-axis          y-axis
      Slenderness ratio (KL/r) :        36.2            66.1
      Compression Capacity     :      4078.2          3045.5
      Tension Capacity         :      4435.8          4435.8
      Moment Capacity          :       591.0           258.3
      Reduced Moment Capacity  :       591.0           258.3
      Shear Capacity           :      1301.9           821.3
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  476.3
      co-efficients C1 & K : C1 =0.712 K =0.5, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve a
      Elastic Critical Moment for LTB,              Mcr   =   967.3
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD SPACE                                             -- PAGE NO.
7
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.035     1     0.0     25.0     0.0    20.8     0.0
   EC-6.2.6-(Y)   0.030     1     0.0     25.0     0.0    20.8     0.0
   EC-6.3.2 LTB   0.044     1     0.0     25.0     0.0    20.8     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. EC3 Singapore NA - Tee Section

Calculate the bending capacity of a beam using an Tee section subject to a uniformly distributed load and varying end moments per the Singaporean NA to EC3.

Details

The section is an Indian INST100 with $f_y = 300\ MPa$. The member is a 5 m, simply-supported span under a 2 kN/m uniform load. The left support has a -5.0 kN·m moment applied and the right support has a 2.5 kN·m moment applied.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{45.5 \times (10)^3 \times 300}{1.0 \times (10)^6} = 13.7 \text{ kN}\cdot\text{m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(k_z L)^2}\left[\sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_T}{\pi^2 E I_z} + (C_2 Z_g - C_3 Z_j)^2} - (C_2 Z_g - C_3 Z_j)\right]$$

where

C_1	=	1.0
C_2	=	1.0
C_3	=	1.0
$\frac{\pi^2 E I_y}{(kL)^2}$	=	$64{,}421$
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	$\left(\frac{1}{1}\right)^2 \frac{30.76}{79.6} = 0.386$
$\frac{(kL)^2 G I_T}{\pi^2 E I_y}$	=	$\frac{(1\times 5{,}000)^2 \times 205{,}000 \times 7.33 \times (10)^4}{2.6 \times \pi^2 \times 205{,}000 \times 79.6 \times (10)^4} = 89{,}714$
$C_2 Z_g$	=	$1.0\times 10\ /\ 2 = 5$
Ψ_f	=	1 ($I_{ft} = 0$, as there is no tension flange present in the Tee section)
h_s	=	D - 0.5×t_f = 100 - 0.5 × 10 = 95 mm
z_j	=	$0.8\times\Psi_f\times h_s/2 = 0.8\times 1\times 95\ /\ 2 = 38\ mm$
$C_3 Z_j$	=	$1.0\times 38 = 38$

Therefore, $M_{cr} = 1.0\times 64{,}421\left[\sqrt{0.386 + 89{,}714 + (5 - 38)^2} - (5 - 38)\right] = 21.5\ kN\cdot m$

Results

Table 664:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	13.7	13.7	none	
Critical moment, M_{cr} (kN·m)	21.5	21.5	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Singapore NA - Tee Section.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`

- Simply-supported span: `CMN 1.0`
- Varying end moments and a uniformly distributed load: `CMM 7`
- The values of `C1 1.0`, `C2 1.0`, and `C3 1.0` are specified

Note: Since these parameters are specified, `MU` is *not* specified as it would be ignored in favor of the direct coefficient specifications. Refer to D5.D.10.2 Clause 6.3.2.2 –Elastic critical moment and imperfection factors for LTB checks (on page 1564) for additional details.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-Jul-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISNT100
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 2.5
1 MZ -5
MEMBER LOAD
1 UNI GY -2
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE EN 1993-1-1:2005
NA 7
CMM 7 ALL
C1 1 ALL
C2 1 ALL
C3 1 ALL
CMN 1 ALL
PY 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
4
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
 *     1 ST   ISNT100     (INDIAN SECTIONS)
                           FAIL     EC-6.3.2 LTB       1.064         1
                        0.00           0.00        -10.03        2.08
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  300  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =   19.20          Net Area =   19.20
                                    z-axis          y-axis
      Moment of inertia        :      180.000         79.600
      Plastic modulus          :       45.500         26.600
      Elastic modulus          :       25.210         15.920
      Shear Area               :       13.333         10.900
      Radius of gyration       :        3.062          2.036
      Effective Length         :      500.000        500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :    576.00
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                    z-axis          y-axis
      Slenderness ratio (KL/r) :        163.3          245.6
      Compression Capacity     :        116.5           55.1
      Tension Capacity         :        565.5          565.5
      Moment Capacity          :         13.7            8.0
      Reduced Moment Capacity  :         13.7            8.0
      Shear Capacity           :        230.9          188.8
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =    9.4
      co-efficients C1 & K : C1 =1.000 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve d
      Elastic Critical Moment for LTB,              Mcr   =    21.5
      Compression buckling curves:     z-z:  Curve c    y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  =  4275.7
      Critical Load For Torsional-Flexural Buckling, NcrTF =    64.4
      STAAD PLANE                                              -- PAGE NO.
5
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.735     1     0.0      0.3     0.0   -10.0     0.0
   EC-6.2.6-(Y)   0.029     1     0.0      5.5     0.0     2.5     0.0
   EC-6.3.2 LTB   1.064     1     0.0      0.3     0.0   -10.0     0.0
    Torsion has not been considered in the design.
```

```
  ************** END OF TABULATED RESULT OF DESIGN **************
   44. FINISH
     STAAD PLANE                                                 -- PAGE NO.
6
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:55:56 ****
        ************************************************************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. EC3 Singapore NA - Varying End Mom CMM8

Calculate the bending capacity of a beam using an I section subject to a concentrated load and varying end moments per the Singaporean NA to EC3.

Details

The section is a HD320X127, Grade S275 steel. The member is a 5 m, simply-supported span subject to a 10 kN/m uniform load. The left support has a -10.0 kN·m moment applied and the right support has a 8.0 kN·m moment applied.

Validation

Moment capacity:

$$M_{ckd} = \frac{W_{ply} f_y}{\gamma_{M0}} = \frac{2{,}149 \times (10)^3 \times 275}{1.0 \times (10)^6} = 591.0 \text{ kN·m}$$

The critical moment is given by:

$$M_{cr} = C_1 \frac{\pi^2 EI}{(kL)^2} \left\{ \sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I} + \frac{(kL)^2 G I_t}{\pi^2 EI} + (C_2 Z_g)^2} - C_2 Z_g \right\}$$

where

k_w	=	$k = 1$
$\frac{\pi^2 E I_y}{(kL)^2}$	=	*7,477,200*
$\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_y}$	=	*22,394*
$\frac{(kL)^2 G I_t}{\pi^2 E I_y}$	=	*23,737*
$C_2 Z_g$	=	$0.17 \times 160 = 27.2$

$$\mu = \frac{FL}{4M} = \frac{4 \times 5}{4 \times 10} = 0.5 > 0$$

From the graph, $C_1 = 1.184$, $C_2 = 0.17$

$$C_2 z_g = 0.17 \times 160 = 27.2$$

Therefore, $M_{cr} = 1{,}676\ kN \cdot m$

Results

Table 665:

Result Type	Reference	STAAD.Pro	Difference	Comments
Moment capacity, M_{ckd} (kN·m)	591.0	591.0	none	
Critical moment, M_{cr} (kN·m)	1,676	1,676.5	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Europe\EC3 Singapore NA - Varying End Mom CMM8.std` is typically installed with the program.

The following D5.C.6 Design Parameters (on page 1502) are used:

- The Singaporean NA is specified using `NA 7`
- Simply-supported span: `CMN 1.0`
- Concentrated load with varying end moments: `CMM 8`
- The value of `C2 0.17` is specified

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 03-Jul-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 PINNED
*2 FIXED BUT FX FZ MZ
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
1 CON GY -4
JOINT LOAD
2 MZ 8
```

```
1 MZ -10
PERFORM ANALYSIS
PARAMETER 1
CODE EN 1993-1-1:2005
NA 7
CMM 8 ALL
C1 1.184 ALL
C2 0.17 ALL
C3 1 ALL
CMN 1 ALL
PY 275000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.PRO CODE CHECKING - SS EN 1993-1-1:2010
                         ********************************************
                         NATIONAL ANNEX - NA to SS EN 1993-1-1:2010
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      STAAD PLANE                                              -- PAGE NO.
3
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HD320X127   (EUROPEAN SECTIONS)
                           PASS     EC-6.3.2 LTB       0.025         1
                        0.00           0.00        -14.00        2.50
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  USER
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    500.00
      Gross Area =  161.30          Net Area =  161.30
                                      z-axis          y-axis
      Moment of inertia        :    30820.004        9239.001
      Plastic modulus          :     2149.000         939.100
      Elastic modulus          :     1926.250         615.933
      Shear Area               :       81.998          51.728
      Radius of gyration       :       13.823           7.568
      Effective Length         :      500.000         500.000
   DESIGN DATA (units - kN,m)   EUROCODE NO.3 /2005
      Section Class            :   CLASS 1
      Squash Load              :   4435.75
      Axial force/Squash load  :     0.000
      GM0 :  1.00          GM1 :  1.00          GM2 :  1.10
                                      z-axis          y-axis
      Slenderness ratio (KL/r) :         36.2            66.1
      Compression Capacity     :       4078.2          3045.5
      Tension Capacity         :       4435.8          4435.8
      Moment Capacity          :        591.0           258.3
      Reduced Moment Capacity  :        591.0           258.3
      Shear Capacity           :       1301.9           821.3
```

```
   BUCKLING CALCULATIONS (units - kN,m)
      Lateral Torsional Buckling Moment       MB =  564.6
      co-efficients C1 & K : C1 =1.184 K =1.0, Effective Length= 5.000
      Lateral Torsional Buckling Curve : Curve b
      Elastic Critical Moment for LTB,              Mcr   =  1676.5
      Compression buckling curves:     z-z:  Curve b   y-y:  Curve c
      Critical Load For Torsional Buckling,         NcrT  = 13898.0
      Critical Load For Torsional-Flexural Buckling, NcrTF = 13898.0
      STAAD PLANE                                             -- PAGE NO.
4
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   EC-6.2.5       0.024     1     0.0      1.6     0.0   -14.0     0.0
   EC-6.2.6-(Y)   0.003     1     0.0     -2.4     0.0   -13.0     0.0
   EC-6.3.2 LTB   0.025     1     0.0      1.6     0.0   -14.0     0.0
    Torsion has not been considered in the design.

   ************** END OF TABULATED RESULT OF DESIGN **************
```

Related Links

- *D5.D.9 Singaporean National Annex to EC3* (on page 1556)

V. India

V. IS 800 2007 LSD - Angle - Flexural Torsional Buckling

Verify the design of a single angle subject to axial compress per the IS 800:2007 limit state method.

References

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

BIS. *IS 808: Dimensions for Hot Rolled Steel Beam, Column, Channel, and Angle Sections.* 1898. New Delhi. p.12

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The member is a 2.5 m long cantilever subject to a 100 kN compressive force. The section used is a ISA 150x115x16. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, a modulus of elasticity of 205×(10)^3 MPa, and a Poisson's ratio of 0.3.

Section PropertiesFrom IS: 808-1989, Table 6.1:

$A_x = 40$ cm^2

$c_z = 3.07$ cm

$c_y = 4.81$ cm

$I_x = 878$ cm^4

$I_y = 447$ cm^4

$r_u = 5.21$ cm

$r_y = 2.44$ cm

Validation

Check for axial compression about the major principal axis

The non-dimensional slenderness ratio,

$$\lambda_z = \sqrt{\frac{f_y\left(K_zL/r_z\right)^2}{\pi^2 E}} = \sqrt{\frac{250\left(1.0 \times 250/5.21\right)^2}{\pi^2 205{,}000}} 0.533$$

Per Table 10, buckling class is “c” for an angle section. Per Table 7, for buckling class “c”, $a = 0.49$

$$\phi_z = 0.5\left[1 + a\left(\lambda_z - 0.2\right) + \lambda_z^2\right] = 0.5\left[1 + 0.49\left(0.533 - 0.2\right) + (0.533)^2\right] = 0.724$$

$$\mathrm{X}_z = \left(\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}\right)^{-1} = \frac{1}{0.724 + \sqrt{(0.724)^2 - (0.533)^2}} = 0.825$$

$$f_{cdz} = \frac{x_z \times f_y}{\gamma_{m0}} = \frac{0.825 \times 250(10)^6}{1.1} = 187.4 \text{ MPa} \qquad \text{(Cl. 7.1.2.1)}$$

The axial compression capacity with respect to the major principal axis is:

$$P_{dz} = A_x \times f_{cdz} = 40(10)^2 \text{ cm}^2 \times 187.4 \text{ MPa}(10)^{-3} = 749.7 \text{ MPa}$$

Check for axial compression about the minor principal axis

The non-dimensional slenderness ratio,

$$\lambda_y = \sqrt{\frac{f_y\left(K_zL/r_z\right)^2}{\pi^2 E}} = \sqrt{\frac{250\left(1.0 \times 250/2.44\right)^2}{\pi^2 205{,}000}} 1.139$$

$$\phi_y = 0.5\left[1 + a\left(\lambda_y - 0.2\right) + \lambda_y^2\right] = 0.5\left[1 + 0.49\left(1.139 - 0.2\right) + (1.139)^2\right] = 1.379$$

$$\mathrm{X}_y = \left(\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}\right)^{-1} = \frac{1}{1.379 + \sqrt{(1.379)^2 - (1.139)^2}} = 0.464$$

$$f_{cdy} = \frac{x_y \times f_y}{\gamma_{m0}} = \frac{0.464 \times 250(10)^6}{1.1} = 105.4 \text{ MPa} \qquad \text{(Cl. 7.1.2.1)}$$

The axial compression capacity with respect to the minor principal axis is:

$$P_{dy} = A_x \times f_{cdy} = 40(10)^2 \text{ cm}^2 \times 105.4 \text{ MPa}(10)^{-3} = 421.8 \text{ MPa}$$

Check for axial compression against flexural torsional buckling

Since the member being design is an angle loaded through one leg, the axial compression capacity with respect to flexural torsional buckling must be checked. The end connection has two or more bolts (or is welded) and is assumed to be unrestrained (“hinged”). Therefore, from Table 12: $k_1 = 0.7$, $k_2 = 0.6$, and $k_3 = 5$.

$$\lambda_e = \sqrt{k_1 + k_2 \times \lambda_{vv}^2 + k_3 \times \lambda_\phi^2} \qquad \text{(Cl. 7.5.1.2)}$$

$$\varepsilon = \sqrt{250 / f_y} = 1$$

$$\lambda_{\phi} = \frac{(b+d)/2t}{\varepsilon\sqrt{\frac{\pi^2 \times E}{250}}} = \frac{(150+115)/(2 \times 16)}{1.0\sqrt{\frac{\pi^2 \times 205{,}000}{250}}} = 0.092$$

$$\lambda_{vv} = \frac{L \;/r_y}{\varepsilon\sqrt{\frac{\pi^2 \times E}{250}}} = \frac{250/2.44}{1.0\sqrt{\frac{\pi^2 \times 205{,}000}{250}}} = 1.139$$

So, $\lambda_e = \sqrt{0.7 + 0.6 \times 1.139^2 + 5 \times 0.092^2} = 1.233$

$$\phi = 0.5\left[1 + a\left(\lambda_e - 0.2\right) + \lambda_e{}^2\right] = 0.5\left[1 + 0.49\left(1.233 - 0.2\right) + (1.233)^2\right] = 1.513 \qquad \text{(Cl. 7.1.2.1)}$$

$$\chi = \left(\phi + \sqrt{\phi^2 - \lambda_e{}^2}\right)^{-1} = \frac{1}{1.513 + \sqrt{(1.513)^2 - (1.233)^2}} = 0.418$$

$$f_{cd} = \frac{x \times f_y}{\gamma_{m0}} = \frac{0.418 \times 250(10)^6}{1.1} = 95.06 \text{ MPa}$$

The axial compression capacity under flexural torsional buckling is:

$$P_{dft} = A_x \times f_{cd} = 40(10)^2 \text{ cm}^2 \times 95.06 \text{ MPa}(10)^{-3} = 380.2 \text{ MPa}$$

The minimum capacity governs, thus flexural torsional buckling controls. The utilization ratio is

$P / P_d = \frac{100}{380.2} = 0.263$

Results

Table 666:

Result Type	Reference	STAAD.Pro	Difference	Comments
Axial compression capacity,major axis (kN)	749.7	750.202	negligible	
Axial compression , minor axis (kN)	421.8	421.773	negligible	
Axial compression capacity, flexural torsional buckling (kN)	380.2	380.234	negligible	
Ratio for compression	0.263	0.263	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - Angle - Flexural Torsional Buckling.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The beam type is a cantilever, specified by `CAN 1`
- The member is connected by a single angle leg, so `ANG 1`
- The connection is considered "hinged" using `FXTY 0`
- Two or more bolts are assumed at the connection, thus `NBL 1`
- The beam is laterally unsupported, specified by `LAT 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-May-20
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
*****************************************************************************
*This problem has been created to validate the program calculated compression
*capacity of a single angle section loaded eccentrically through one leg and
*designed using IS 800 2007 (LSD)
*****************************************************************************
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 2.5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISA150X115X16
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PRINT ANALYSIS RESULTS
SECTION 0.25 0.5 0.75
PRINT MEMBER SECTION FORCES ALL
PARAMETER 1
CODE IS800 LSD
ANG 1 ALL
CAN 1 ALL
FXTY 0 ALL
NBL 1 ALL
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|--------------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                          |
 |  Member Section:  ST   ISA150X115X16         (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.569  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Slenderness
(Compression)                                |
 |  Critical Design Forces:  (Unit: KN
METE)                                  |
 |      FX:     100.000E+00 C     FY:       0.000E+00      FZ:       0.000E
+00      |
 |      MX:       0.000E+00       MY:       0.000E+00      MZ:       0.000E
+00      |

|--------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                            |
 |  AXX:      40.000E+00          IZZ:     238.144E+00          RZZ:
2.440E+00|
 |  AYY:      24.000E+00          IYY:       1.094E+03          RYY:
5.229E+00|
 |  AZZ:      18.400E+00          IXX:      34.133E+00           CW:
465.161E+00|
 |  ZEZ:      47.777E+00          ZPZ:      82.555E
+00                              |
 |  ZEY:     105.142E+00          ZPY:     181.532E
+00                              |

|--------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                      |
 |  Actual Length:        2.500E
+00                                      |
 |  Parameters:     LZ:        2.500E+00  LY:        2.500E
+00                     |
 |                 KZ:           1.000  KY:
1.000                   |
 |  Actual Ratio: 102.46 Allowable Ratio: 180.00  LOAD:
1                       |

|--------------------------------------------------------------------------------
-----|
 |  Section Class: Semi-Compact; Flange Class: Semi-Compact; Web Class: Semi-
Compact|
```

```
|--------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                  -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|--------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                             |
 |  Member Section:  ST   ISA150X115X16         (INDIAN
SECTIONS)                    |

|--------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                   |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03               |
 |                  NSF:            1.000    ALPHA:
0.800               |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                      |
 |              Capacity:          909.091E+00     As per:  Sec.
6.2              |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                      |
 |              Capacity:            1.075E+03     As per:  Sec.
6.3              |

|--------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                               |
 |  Buckling Class:   Major: c    Minor: c    As per:Cl.
7.1.2.2                      |
 |  Parameters:     ANG:  1       NBL:  1       FXTY:
0.000                   |
 |  Major Axis:   Design Force:      100.000E+00    LC:     1  Loc:
0.000        |
 |              Capacity:          750.202E+00    As per:  Sec.
7.1.2              |
 |  Minor Axis:   Design Force:      100.000E+00    LC:     1  Loc:
0.000        |
 |              Capacity:          421.773E+00    As per:  Sec.
7.1.2              |
 |  FlxTorBuck:   Design Force:      100.000E+00    LC:     1  Loc:
0.000        |
 |              Capacity:          380.234E+00    As per:  Sec.
7.1.2              |

|--------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                          |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
```

```
0.000       |
 |              Capacity:          241.437E+00    As per:  Sec.
8.4               |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |              Capacity:          314.918E+00    As per:  Sec.
8.4               |

|-----------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                 |
 |  Parameters:   Laterally Unsupported    KX:   1.00  LX:   2.500E+00
Cantilever  |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |              Capacity:           22.597E+00    As per:  Sec.
8.2.2             |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |              Capacity:           10.858E+00    As per:  Sec.
8.2.1.2           |

|-----------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                          |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.110    As per:  Sec.
9.3.1.3                  |
 |                      LC:           1    Loc:
0.000                      |

|-----------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension               0.000       0                      0.000E
+00          |
 |  Compression           0.263       1                      0.000E
+00          |
 |  Shear Major           0.000       0                      0.000E
+00          |
 |  Shear Minor           0.000       0                      0.000E
+00          |
 |  Bend Major            0.000       0                      0.000E
+00          |
 |  Bend Minor            0.000       0                      0.000E
+00          |
 |  Sec. 9.3.1.3          0.110       1                      0.000E
+00          |
 |  Sec. 9.3.2.2 (i)      0.237       1                      0.000E
+00          |
 |  Sec. 9.3.2.2(ii)      0.133       1                      0.000E
```

```
+00             |

|--------------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - Angle - Tension with Block Shear

Verify the design of a single angle subject to tension per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The member is a 3 m long cantilever subject to a 100 kN tensile force. The section used is a ISA 150x90x15. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, and a modulus of elasticity of $205\times(10)^3$ MPa.

Validation

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Check for yielding of gross section per Cl. 6.2:

$$T_{dg} = A_g \times f_y / \gamma_{m0} = 770.5 \text{ kN} \quad \text{(Cl 6.2)}$$

Check for rupture of critical section per Cl 6.3:

$$\alpha = 0.8$$

The net section factor is given as 0.79.

$$A_{net} = 0.79 \times 33.9 \text{ cm}^2 = 26.8 \text{ cm}^2$$

$$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 719.9 \text{ kN} \quad \text{(Cl 6.3)}$$

Check for block shear per Cl 6.4.1:

The minimum net area in shear along a bolt line parallel to the external force, $A_{vn} = 2{,}040\ mm^2$.
The minimum gross area in shear along a bolt line parallel to the external force, $A_{vg} = 3{,}525\ mm^2$.
The minimum net area in tension of the angel between the bolt hole and toe or end bolt line, normal to the line of force $A_{tn} = 570\ mm^2$.
The minimum gross area in tension of the angel between the bolt hole and toe or end bolt line, normal to the line of force $A_{tg} = 675\ mm^2$.

$$T_{db\_1} = \frac{A_{vg} f_y}{\sqrt{3}\gamma_{m0}} + \frac{0.9 A_{tn} f_u}{\gamma_{m1}} = 616.8 \text{ kN}$$

$$T_{db\_2} = \frac{0.9A_{vn} f_u}{\sqrt{3}\gamma_{m1}} + \frac{A_{tg} f_y}{\gamma_{m0}} = 509.6 \text{ kN}$$

Then tension capacity in block shear, $T_{db} = min(T_{db\_1}, T_{db\_2}) = 509.6\ kN$

Tension capacity, $T_d = min(T_{dg}, T_{dn}, T_{db}) = 509.6\ kN$

The critical ratio for tension is $100\ kN\ /\ 509.6\ kN = 0.196$

Results

Table 667:

Result Type	Reference	STAAD.Pro	Difference	Comments
Tension capacity, gross yielding (kN)	770.5	770.455	negligible	
Tension capacity, rupture of critical section (kN)	719.9	719.873	negligible	
Tension capacity, block shear (kN)	509.6	509.574	negligible	
Ratio for tension	0.196	0.196	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - Angle - Tension with Block Shear.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The net section factor is specified by `NSF 0.79`
- The areas considered in the block rupture calculation are specified by:

```
AVN 2040
AVG 3525
ATN 570
ATG 675
```

- The beam type is a cantilever, specified by `CAN 1`
- The beam is laterally unsupported, specified by `LAT 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Jun-2020
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of a single angle section subjected to Axial Tension,
```

```
*designed using IS 800 2007 (LSD)
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISA150X90X15
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 100
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE IS800 LSD
AVN 2040 ALL
AVG 3525 ALL
ATN 570 ALL
ATG 675 ALL
CAN 1 ALL
LAT 0 ALL
NSF 0.79 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|------------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                           |
 |  Member Section:  ST   ISA150X90X15          (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.389  Critical Load Case:     1  Location:
0.00    |
 |  Critical Condition: Slenderness
```

```
(Tension)                                                   |
 |  Critical Design Forces:  (Unit: KN
METE)                                                 |
 |      FX:    -100.000E+00 T     FY:       0.000E+00      FZ:       0.000E
+00      |
 |      MX:       0.000E+00       MY:       0.000E+00      MZ:       0.000E
+00      |

|-------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                                         |
 |  AXX:      33.900E+00          IZZ:     126.274E+00          RZZ:
1.930E+00|
 |  AYY:      22.500E+00          IYY:     848.279E+00          RYY:
5.002E+00|
 |  AZZ:      13.500E+00          IXX:      25.425E+00           CW:
323.921E+00|
 |  ZEZ:      33.333E+00          ZPZ:      53.497E
+00                              |
 |  ZEY:      84.935E+00          ZPY:     148.166E
+00                              |

|-------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                                   |
 |  Actual Length:        3.000E
+00                                                  |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                      |
 |                  KZ:           1.000  KY:
1.000                        |
 |  Actual Ratio: 155.44 Allowable Ratio: 400.00  LOAD:
1                       |

|-------------------------------------------------------------------------------
-----|
 |  Section Class: Semi-Compact; Flange Class: Semi-Compact; Web Class: Semi-
Compact|

|-------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                               |
 |  Member Section:  ST   ISA150X90X15          (INDIAN
SECTIONS)                       |

|-------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
```

```
METE)                                                 |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                 |
 |                  NSF:            0.790    ALPHA:
0.800               |
 |        AVG:     3525.00  AVN:     2040.00  ATN:      675.00  ATG:
570.00    |
 |  Yielding   :  Design Force:      100.000E+00     LC:
1                      |
 |                Capacity:          770.455E+00     As per:  Sec.
6.2              |
 |  Rupture    :  Design Force:      100.000E+00     LC:
1                      |
 |                Capacity:          719.873E+00     As per:  Sec.
6.3              |
 |  Block Shear:  Design Force:      100.000E+00     LC:
1                      |
 |                Capacity:          509.574E+06     As per:  Sec.
6.4              |

|------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                             |
 |  Buckling Class:   Major: c    Minor: c    As per:Cl.
7.1.2.2                   |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          574.159E+00    As per:  Sec.
7.1.2              |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          193.342E+00    As per:  Sec.
7.1.2              |

|------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                         |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          177.142E+00    As per:  Sec.
8.4                |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          295.236E+00    As per:  Sec.
8.4                |

|------------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                 |
 |  Parameters:   Laterally Unsupported    KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:           17.656E+00    As per:  Sec.
8.2.2              |
```

```
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |                Capacity:            6.323E+00    As per:  Sec.
8.2.1.2           |

|---------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                           |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900         |
 |  Section Strength:    Ratio:     0.139    As per:  Sec.
9.3.1.3                 |
 |                       LC:           1    Loc:
0.000                      |

|---------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                0.139      1                        0.000E
+00           |
 |  Compression            0.000      0                        0.000E
+00           |
 |  Shear Major            0.000      0                        0.000E
+00           |
 |  Shear Minor            0.000      0                        0.000E
+00           |
 |  Bend Major             0.000      0                        0.000E
+00           |
 |  Bend Minor             0.000      0                        0.000E
+00           |
 |  Sec. 9.3.1.3           0.139      1                        0.000E
+00           |

|---------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - Channel - with LTB

Verify the design of a channel section with lateral-torsional buckling per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

BIS. *IS 808: Dimensions for Hot Rolled Steel Beam, Column, Channel, and Angle Sections.* 1898. New Delhi. p.6

CISC. “Torsional Section Properties of steel Shapes”. Canadian Institute of Steel Construction. 2002. Willowdale, Ont. p.7. http://dir.cisc-icca.ca/files/technical/techdocs/updates/torsionprop.pdf

Details

The member is a 3 m long cantilever subject to a 10 kN compressive force and uniform bending moments of 1 kN/m along both major and minor axes. The section used is a ISMC 200 and is unbraced against lateral-torsional buckling. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, a modulus of elasticity of $205 \times (10)^3$ MPa, and a Poisson's ratio, $\nu = 0.3$.

Section PropertiesFrom IS:808

Cross-sectional area, $A_x = 28.5\ cm^2$
Overall depth of the section, $h = 200\ mm$
Width of flange, $b_f = 75\ mm$
Thickness of flange, $t_f = 11.4\ mm$
Thickness of web, $t_w = 6.2\ mm$
Distance from heel to centroid, $c_y = 2.20\ cm$
Moment of inertia about major axis, $I_z = 1,830\ cm^4$
Moment of inertia about minor axis, $I_y = 139\ cm^4$*
Radius of gyration about major axis, $r_z = 8.02\ cm$
Elastic section modulus about minor axis, $Z_{ey} = 26.4\ cm^3$

* This value have been taken from the built-in section library STAAD.Pro of Indian sections.

Additional section properties:

Plastic section modulus about major axis, $Z_{pz} = 213\ cm^3$
Plastic section modulus about minor axis, $Z_{py} = 51.1\ cm^3$
Torsional moment of inertia, $I_x = 9.83\ cm^4$

Validation

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Section Classification

$$\varepsilon = \sqrt{250 / f_y} = 1$$

$$b = b_f/2 = 37.5 \text{ mm}$$

$$b / t_f = 6.58 < 9.4\varepsilon = 9.4$$

$$d_w/t_w = 25.03 < 42\varepsilon = 42$$

Hence, from Table 2 of IS 800:2007, the flange is plastic but the web is non-slender (i.e., semi-compact).

Therefore, the overall classification of the section is “semi-compact”.

Hence, from Table 10 of IS 800:2007, the buckling class about both axes is “c”.

Check Slenderness Ratio

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about the major axis, $k_z L / r_z = 37.41 < 180$

Radius of gyration about minor axis, $r_y = \sqrt{I_y / A_x} = \sqrt{139 / 28.5} = 2.21\ cm$

Slenderness ratio about the minor axis, $k_y L / r_y = 135.8 < 180$

So the slenderness is within the limit.

<u>Axial Tension Capacity</u>

Check for yielding of gross section:

$$T_{dg} = A_g \times f_y / \gamma_{m0} = 647.7\ \text{kN}$$

Check for rupture of critical section:

$$\alpha = 0.8$$

$$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 766.1\ \text{kN}$$

Block shear areas are not specified and therefore not checked.

No tension in the member, so critical ratio is 0.

<u>Check Axial Compression</u>

Axial compression capacity of major axis:

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\dfrac{f_y (k_z L / r_z)^2}{\pi^2 E}} = \sqrt{\dfrac{250(1.0 \times 3{,}000 / 80.2)^2}{\pi^2 \times 205{,}000}} = 0.416$

For buckling class "c", the imperfection factor, $\alpha = 0.49$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 0.639$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = \frac{1}{0.639 + \sqrt{0.639^2 - 0.416^2}} = 0.889$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\dfrac{\chi_z f_y}{\gamma_{m0}}, \dfrac{f_y}{\gamma_{m0}}\right) = 202.0\ MPa$

Therefore, axial compression capacity in the major axis: $P_{dz} = A_x f_{cdz} = 575.6\ kN$

The critical ratio for axial compression in major axis = $10 / 575.6 = 0.017$

Axial compression capacity of minor axis:

The non-dimensional slenderness ratio, $\lambda_y = \sqrt{\dfrac{f_y (k_y L / r_y)^2}{\pi^2 E}} = \sqrt{\dfrac{205(1.0 \times 3{,}000 / 22.2)^2}{\pi^2 \times 205{,}000}} = 1.502$

For buckling class "c", the imperfection factor, $\alpha = 0.49$

$$\phi_y = 0.5\left[1 + \alpha(\lambda_y - 0.2) + \lambda_y^2\right] = 1.961$$

$$\chi_y = \frac{1}{\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}} = 0.311$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdy} = min\left(\dfrac{\chi_y f_y}{\gamma_{m0}}, \dfrac{f_y}{\gamma_{m0}}\right) = 70.75\ MPa$

Therefore, axial compression capacity in the minor axis: $P_{dy} = A_x f_{cdy} = 201.6\ kN$, which governs.

The critical ratio for axial compression in minor axis = $10 / 201.6 = 0.050$

Check Shear Capacity

Major Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times b_f t_f = 1{,}710\ mm^2$

So, as per Cl. 8.4.1, $V_{pz} = \frac{A_{vz} f_{yw}}{\sqrt{3}} = 246.8\ kN$

Hence, shear capacity along major axis: $V_{dz} = \frac{V_{pz}}{\gamma_{m0}} = 224.4\ kN$

Critical shear ratio along major axis: $3 / 224.5 = 0.013$

Minor Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vy} = h \times t_w = 1{,}240\ mm^2$

So, as per Cl. 8.4.1, $V_{py} = \frac{A_{vy} f_{yw}}{\sqrt{3}} = 179.0\ kN$

Hence, shear capacity along major axis: $V_{dy} = \frac{V_{py}}{\gamma_{m0}} = 162.7\ kN$

As per Cl. 8.4.2, $d_w / t_w = 25.03 < 67\varepsilon$, so resistance to shear buckling need not be checked.

Shear force along both axes = $3\ kN$.

Critical shear ratio along minor axis: $3 / 162.7 = 0.018$

Check Bending Capacity

Since the section has been classified as "semi-compact", $\beta_b = Z_{ez} / Z_{pz} = 0.859$.

Major Axis Bending

The elastic lateral-torsional buckling moment:

$$M_{cr} = \sqrt{\frac{\pi^2 E I_y}{L_{LT}^2}\left(G I_t + \frac{\pi^2 E I_w}{L_{LT}^2}\right)}$$

where

G = shear modulus, $= \frac{E}{2(1 + \nu)} = 78{,}850\ MPa$

d' = $d - t_f = 188.6\ mm$

b' = $b - t_w/2 = 71.9\ mm$

a = $\frac{1}{2 + \frac{d' t_w}{3b' t_f}} = \frac{1}{2 + \frac{18.86(0.62)}{3(7.19)(1.14)}} = 0.404$

I_w = warping constant, as taken from the CISC document "Torsional Section Properties of Stee
reference), which is taken from SSRC 1998 and Galambos 1968.

$$I_w = d'^2 b'^3 t_f\left[\frac{1 - 3(0.404)}{6} + \frac{a^2}{2}\left(1 + \frac{d' t_w}{6b' t_f}\right)\right] = (18.86)^2(7.19)^3 1.14\left[\frac{1 - 3(0.404)}{6} + \frac{(0.404)^2}{2}\right.$$

$$M_{cr} = \sqrt{\frac{\pi^2(205{,}000)(141)10^4}{(3{,}000)^2}\left[(78{,}850)(9.83)10^4 + \frac{\pi^2(205{,}000)(9{,}899)10^6}{(3{,}000)^2}\right]}(10)^{-6} = 55.83\ \mathrm{kN \cdot m}$$

$$\lambda_{LT} = \min \left| \begin{array}{l} \sqrt{\frac{\beta_b Z_{pz} f_y}{M_{cr}}} = \sqrt{\frac{0.859(213)(250)}{55.83(10)^3}} = 0.905 \\ \sqrt{\frac{1.2 Z_{cz} f_y}{M_{cr}}} = \sqrt{\frac{1.2(183)(250)}{55.83(10)^3}} = 0.992 \end{array} \right.$$

Since $\lambda_{LT} > 0.4$, per 8.2.2(c), Lateral Torsional Buckling must be considered.

$$\phi_{\mathrm{LT}} = 0.5\left[1 + 0.21(\lambda_{\mathrm{LT}} - 0.2) + \lambda_{\mathrm{LT}}^2\right] = 0.984$$

$$\chi_{\mathrm{LT}} = \frac{1}{\phi_{\mathrm{LT}} + \sqrt{\phi_{\mathrm{LT}}^2 - \lambda_{\mathrm{LT}}^2}} = 0.730$$

$$f_{bd} = \frac{\chi_{LT} f_y}{\gamma_{m0}} = 165.9 \text{ MPa}$$

So, the bending strength with respect to the major axis is $M_{dz} = \beta_b Z_{pz} f_{bd} = 30.35\ kN \cdot m$

Actual bending moment about the major axis is $M_z = 4.5\ kN \cdot m$.

The critical bending ratio in the major axis is $4.5\ /\ 30.35 = 0.148$

Minor Axis Bending

Since the section has been classified as “semi-compact”, $\beta_b = Z_{ey}\ /\ Z_{py} = 0.517$.

Actual shear force in both axes, $V = 3\ kN < 0.6V_d = 89.75\ kN$. Hence, the section is subjected to low shear.

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 5.938 \text{ kN} \cdot \text{m}$$

$$\frac{1.2 Z_{ey} f_y}{\gamma_{m0}} = 8.907 \text{ kN} \cdot \text{m}$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 5.938\ kN \cdot m$

Actual bending moment about the minor axis is $M_z = 4.5\ kN \cdot m$.

The critical bending ratio in the minor axis is $4.5\ /\ 5.938 = 0.758$

Check Combined Interaction of Axial Compression and Bending

Section strength calculation as per Cl. 9.3.1.3:

Compression strength in yielding: $N_d = \frac{A_g f_y}{\gamma_{m0}} = \frac{28.5(10)^{-1}(250)}{1.1} = 647.7\ kN$

$$\frac{N}{N_d} + \frac{M_y}{M_{\mathrm{dy}}} + \frac{M_z}{M_{\mathrm{dz}}} = \frac{10}{647.7} + 0.758 + 0.148 = 0.921$$

Overall member strength in bending and axial compression

$$\mathrm{n_y} = \mathrm{F_x} / \mathrm{P_{dy}} = 0.050$$

$$\mathrm{n_z} = \mathrm{F_x} / \mathrm{P_{dz}} = 0.017$$

$$K_y = \min \left| \begin{array}{l} 1 + (\lambda_y - 0.2) n_y = 1 + (1.51 - 0.2) \cdot 0.050 = 1.066 \\ 1 + 0.8 n_y = 1 + 0.8 \cdot 0.050 = 1.040 \end{array} \right.$$

$$K_z = \min \left| \begin{array}{l} 1 + (\lambda_z - 0.2)n_z = 1 + (0.416 - 0.2) \cdot 0.017 = 1.004 \\ 1 + 0.8n_z = 1 + 0.8 \cdot 0.017 = 1.014 \end{array} \right.$$

$$K_{LT} = \max \left| \begin{array}{l} 1 - \frac{0.1\lambda_{LT}n_y}{C_{mLT} - 0.25} = 1 - \frac{0.1 \times 0.905 \times 0.050}{0.9 - 0.25} = 0.993 \\ 1 - \frac{0.1n_y}{C_{mLT} - 0.25} = 1 - \frac{0.1 \times 0.050}{0.9 - 0.25} = 0.992 \end{array} \right.$$

$$\frac{P}{P_{dy}} + K_y \frac{C_{my}M_y}{M_{dy}} + K_{LT}\frac{M_z}{M_{dz}} = 0.906$$

$$\frac{P}{P_{dz}} + 0.6K_y \frac{C_{my}M_y}{M_{dy}} + K_z \frac{C_{mz}M_z}{M_{dz}} = 0.577$$

Results

Table 668:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	135.8	135.84	negligible	
Tension capacity, gross yielding (kN)	647.7	647.727	none	
Tension capacity, rupture of critical section (kN)	766.1	766.08	negligible	
Axial compression capacity about major axis (kN)	575.6	575.661	negligible	
Axial compression capacity about minor axis (kN)	201.6	201.646	negligible	
Critical ratio for axial compression	0.050	0.050	none	
Shear capacity in major axis (kN)	224.4	224.379	negligible	
Shear capacity in minor axis (kN)	162.7	162.708	negligible	
Ratio for shear in major axis	0.013	0.013	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio for shear in minor axis	0.018	0.018	none	
Bending capacity about major axis (kN·m)	30.35	30.382	negligible	
Bending capacity about minor axis (kN·m)	5.938	5.938	none	
Ratio for bending about major axis	0.148	0.148	none	
Ratio for bending about minor axis	0.758	0.758	none	
Ratio per Cl. 9.3.1.3	0.921	0.921	none	
Ratio per Eq. 1 of Cl. 9.3.2.2	0.906	0.906	none	
Ratio per Eq.2 of Cl. 9.3.2.2	0.577	0.577	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - Channel - With LTB.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The value of `KX 1.0` is directly specified.
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The effective length for lateral-torsional buckling is specified as `LX 3` (m).
- The beam type is a cantilever, specified by `CAN 1`
- The beam is laterally unsupported, specified by `LAT 0`
- The default values of `CMX`, `CMY`, and `CMZ` of 0.9 are used (`CMZ 0.9` is directly specified).

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Jun-19
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of laterally unsupported Channel section subjected to axial
*compression and biaxial bending, designed using IS 800 2007 (LSD)
****************************************************************************
INPUT WIDTH 79
```

```
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISMC200
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES
PARAMETER 1
CODE IS800 LSD
LX 3 ALL
KX 1 ALL
CAN 1 ALL
CMZ 0.9 ALL
LAT 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                           |
 |  Member Section:  ST   ISMC200               (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.921  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Sec.
9.3.1.3                                                 |
 |  Critical Design Forces:  (Unit: KN
```

```
METE)                                           |
 |      FX:      10.000E+00 C     FY:        3.000E+00      FZ:        3.000E
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:        4.500E
+00      |

|-----------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                                  |
 |  AXX:      28.500E+00          IZZ:       1.830E+03          RZZ:
8.013E+00|
 |  AYY:      12.400E+00          IYY:     139.000E+00          RYY:
2.208E+00|
 |  AZZ:      17.100E+00          IXX:       9.830E+00           CW:
9.899E+03|
 |  ZEZ:     183.000E+00          ZPZ:     213.000E
+00                           |
 |  ZEY:      63.761E+00          ZPY:      51.100E
+00                           |

|-----------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                     |
 |  Actual Length:        3.000E
+00                                                |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                     |
 |                  KZ:           1.000  KY:
1.000                     |
 |  Actual Ratio: 135.84 Allowable Ratio: 180.00  LOAD:
1                       |

|-----------------------------------------------------------------------------
-----|
 |  Section Class: Semi-Compact; Flange Class:      Plastic; Web Class: Semi-
Compact|

|-----------------------------------------------------------------------------
-----|
      STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                         |
 |  Member Section:  ST   ISMC200               (INDIAN
SECTIONS)                     |

|-----------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                 |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
```

```
+03                 |
 |                   NSF:            1.000    ALPHA:
0.800               |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                     |
 |                Capacity:          647.727E+00     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                     |
 |                Capacity:          766.080E+00     As per:  Sec.
6.3               |

|-------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                         |
 |  Buckling Class:   Major: c    Minor: c    As per:Cl.
7.1.2.2                      |
 |  Major Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          575.661E+00    As per:  Sec.
7.1.2             |
 |  Minor Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          201.646E+00    As per:  Sec.
7.1.2             |

|-------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                           |
 |  Major Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          224.379E+00    As per:  Sec.
8.4               |
 |  Minor Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          162.708E+00    As per:  Sec.
8.4               |

|-------------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                             |
 |  Parameters:   Laterally Unsupported    KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major Axis:   Design Force:        4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:           30.382E+00    As per:  Sec.
8.2.2             |
 |  Minor Axis:   Design Force:       -4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:            5.938E+00    As per:  Sec.
8.2.1.2           |

|-------------------------------------------------------------------------------
-----|
 |  Combined
```

```
Interaction:                                                                  |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.921    As per:  Sec.
9.3.1.3                |
 |                       LC:            1    Loc:
0.000                     |
 |  Overall Member Strength (Bending
+Compression):                                   |
 |  Equation 1:          Ratio:     0.906    As per:  Sec. 9.3.2.2   Ky :
1.0397  |
 |                       LC:            1    Loc:       0.000        KLT:
0.9931  |
 |  Equation 2:          Ratio:     0.577    As per:  Sec. 9.3.2.2   Ky :
1.0397  |
 |                       LC:            1    Loc:       0.000        Kz :
1.0038  |

|------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                0.000      0                       0.000E
+00            |
 |  Compression            0.050      1                       0.000E
+00            |
 |  Shear Major            0.013      1                       0.000E
+00            |
 |  Shear Minor            0.018      1                       0.000E
+00            |
 |  Bend Major             0.148      1                       0.000E
+00            |
 |  Bend Minor             0.758      1                       0.000E
+00            |
 |  Sec. 9.3.1.3           0.921      1                       0.000E
+00            |
 |  Sec. 9.3.2.2 (i)       0.906      1                       0.000E
+00            |
 |  Sec. 9.3.2.2(ii)       0.577      1                       0.000E
+00            |

|------------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - I Section - High Shear

Verify the design of an I section beam subject to high shear per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The member is a 3 m long, simply-supported beam subject to the following forces:

compression force of 100 kN at the “roller” end, and
concentrated transverse loads of 350 kN at both of the one-third points acting along the minor axis of the section

The section used is a built-up I shape with 400 mm × 20 mm flanges, a web thickness of 8 mm, and an overall depth of 550 mm. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, and a modulus of elasticity of $205\times(10)^3$ MPa. The member is laterally supported along its length.

Validation

<u>Section Properties</u>

Depth of web, $d = 550 - 2\times20 = 510\ mm$

Cross-sectional area, $A_x = 2(400 \times 20) + 510 \times 8 = 20{,}080\ mm^2 = 200.8\ cm^2$

Moment of inertia about major axis, $I_z = \frac{2b_f t_f^3}{12} + 2b_f t_f\left(\frac{h}{2} - \frac{t_f}{2}\right)^2 + \frac{t_2 d^3}{12} = 21{,}336\ cm^4$

Moment of inertia about minor axis, $I_y = \frac{2t_f b_f^3}{12} + \frac{d t_w^3}{12} = 137\ cm^4$

Radius of gyration about major axis, $r_z = \sqrt{I_z / A_x} = 247.7\ \ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y / A_x} = 103.1\ \ mm$

Elastic section modulus about major axis, $Z_{ez} = \frac{I_z}{h / 2} = 4{,}409\ cm^3$

Elastic section modulus about minor axis, $Z_{ey} = \frac{I_z}{b_f / 2} = 1{,}067\ cm^3$

Plastic section modulus about major axis, $Z_{pz} = b_f t_f (h - t_f) + 0.25t_w(h - 2t_f)^2 = 4{,}760\ cm^3$

Plastic section modulus about minor axis, $Z_{py} = \frac{b_f^2 t_f}{2} + 0.25t_w^2(h - 2t_f) = 1{,}608\ cm^3$

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

<u>Section Classification</u>

$\varepsilon = \sqrt{250 / f_y} = 1$

$b = b_f/2 = 200$ mm

$b / t_f = 10 < 9.4\varepsilon = 9.4$ and $b / t_f = 10 > 10.5\varepsilon = 10.5$

Since the section is symmetric about both axes, the neutral axis is at mid-depth.

$d_w / t_w = 63.75 < 84\varepsilon = 84$

Hence, from Table 2 of IS 800:2007, the flange is compact and the web is plastic.

Therefore, the overall classification of the section is "compact".

$h / b_f = 1.375 > 1.2$ and $t_f = 20\ mm < 40\ mm$

Hence, from Table 10 of IS 800:2007, the buckling class about the major axis is "a" and the same about the minor axis is "b".

Check Slenderness Ratio

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about the major axis, $k_z L / r_z = 12.21 < 180$

Slenderness ratio about the minor axis, $k_y L / r_y = 29.10 < 180$

So the slenderness is within the limit.

Axial Tension Capacity

Check for yielding of gross section:

$T_{dg} = A_g \times f_y / \gamma_{m0} = 4{,}564$ kN

Check for rupture of critical section:

$\alpha = 0.8$

$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 5{,}398$ kN

Block shear areas are not specified and therefore not checked.

No tension in the member, so critical ratio is 0.

Check Axial Compression

Axial compression capacity of major axis:

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\frac{f_y (k_z L / r_z)^2}{\pi^2 E}} = 0.136$

For buckling class "a", the imperfection factor, $\alpha = 0.21$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 0.502$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = 1.014$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\frac{\chi_z f_y}{\gamma_{m0}}, \frac{f_y}{\gamma_{m0}}\right) = 227.3\ MPa$

Therefore, axial compression capacity in the major axis: $P_{dz} = A_x f_{cdz} = 4{,}564\ kN$

Axial compression capacity of minor axis:

The non-dimensional slenderness ratio, $\lambda_y = \sqrt{\frac{f_y (k_y L / r_y)^2}{\pi^2 E}} = 0.324$

For buckling class "b", the imperfection factor, $\alpha = 0.34$

$$\phi_y = 0.5\left[1 + \alpha\left(\lambda_y - 0.2\right) + \lambda_y^2\right] = 0.573$$

$$\chi_y = \frac{1}{\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}} = 0.955$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdy} = min\left(\frac{\chi_y f_y}{\gamma_{m0}}, \frac{f_y}{\gamma_{m0}}\right) = 217.1\ MPa$

Therefore, axial compression capacity in the minor axis: $P_{dy} = A_x f_{cdy} = 4{,}360\ kN$, which governs.

The critical ratio for axial compression = $100\ /\ 4{,}360 = 0.023$

Check Shear Capacity

Major Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times b_f t_f = 16{,}000\ mm^2$

So, as per Cl. 8.4.1, $V_{pz} = \frac{A_{vz} f_{yw}}{\sqrt{3}} = 2{,}309\ kN$

Hence, shear capacity along major axis: $V_{dz} = \frac{V_{pz}}{\gamma_{m0}} = 2{,}099\ kN$

Minor Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vy} = h \times t_w = 4{,}400\ mm^2$

So, as per Cl. 8.4.1, $V_{py} = \frac{A_{vy} f_{yw}}{\sqrt{3}} = 635.1\ kN$

Hence, shear capacity along major axis: $V_{dy} = \frac{V_{py}}{\gamma_{m0}} = 577.4\ kN$

As per Cl. 8.4.2, $d\ /\ t_w = 63.75 < 67\varepsilon$, so resistance to shear buckling need not be checked.

No shear force along major axis (i.e., ratio = 0).

Critical shear ratio along minor axis: $350\ /\ 577.4 = 0.606$

Check Bending Capacity

Since the section has been classified as “compact”, $\beta_b = 1$.

Major Axis Bending

The actual shear force along the major axis, $V = 350\ kN > 0.6V_d = 346.4\ kN$. Thus, the section is subjected to high shear.

$$M_{dz} = \min\left(\frac{\beta_b Z_{pz} f_y}{\gamma_{m0}}, \frac{1.2 Z_{ez} f_y}{\gamma_{m0}}\right) = 1{,}082\ \mathrm{kN \cdot m} \qquad \text{(Cl 8.2.1.1)}$$

Per Cl 9.2.2(a) for “compact” sections:

$$\beta = \left(2\frac{V_z}{V_d} - 1\right)^2 = 0.045$$

The plastic section modulus excluding shear area, $Z_{fdz} = Z_{pz} - \frac{t_w h^2}{4} = 4{,}155\ cm^3$

Plastic design strength of cross-section area excluding shear area, $M_{fdz} = \frac{Z_{fdz} f_y}{\gamma_{m0}} = 944.4\ kN{\cdot}m$

$$M_{dvz} = M_{dz} - \beta(M_{dz} - M_{fdz}) = 1{,}082 - 0.045\times(1{,}082 - 944.4) = 1{,}076\ kN{\cdot}m \qquad \text{(Cl. 8.2.1.3)}$$

So, the governing bending strength with respect to the major axis is $M_{dz} = M_{dvz} = 1{,}076\ kN{\cdot}m$

Actual bending moment about the major axis is $M_z = 350\ kN{\cdot}m$.

The critical bending ratio in the major axis is $350\ /\ 1{,}076 = 0.325$

Minor Axis Bending

There is no shear along the minor axis.

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 365.5\ kN{\cdot}m$$

$$\frac{1.2 Z_{ey} f_y}{\gamma_{m0}} = 290.9\ kN{\cdot}m$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 290.9\ kN{\cdot}m$

No bending moment about the minor axis in beam, so critical ratio is 0.

Check Combined Interaction of Axial Compression and Bending

Section strength calculation as per Cl. 9.3.1:

$$n = N / N_d = 0.022 < 0.2$$

Hence, from Table 17:

$$\alpha_1 = \max(5n\ ,\ 1) = 1$$

$$\alpha_2 = 2$$

$$M_{ndy} = M_{dy} = 290.9\ kN{\cdot}m \qquad \text{(Cl. 9.3.1.2(c))}$$

$$1.11{\cdot}M_{dz}{\cdot}(1 - n) = 1{,}168\ kN{\cdot}m > M_{dz}$$

$$M_{ndz} = M_{dz} = 1{,}076\ kN{\cdot}m$$

$$\left(\frac{M_y}{M_{ndy}}\right)^{\alpha_1} + \left(\frac{M_z}{M_{ndz}}\right)^{\alpha_2} = \left(\frac{0}{290.9}\right)^1 + \left(\frac{350}{1{,}076}\right)^2 = 0.106$$

Overall member strength in bending and axial compression

$$n_y = F_x / P_{dy} = 0.023$$

$$n_z = F_x / P_{dz} = 0.022$$

$$K_y = \min \left| \begin{array}{l} 1 + (\lambda_y - 0.2)n_y = 1 + (0.324 - 0.2)0.023 = 1.003 \\ 1 + 0.8n_y = 1 + 0.8\ {\cdot}0.023 = 1.018 \end{array} \right.$$

$$K_z = \min \left| \begin{array}{l} 1 + (\lambda_z - 0.2)n_z = 1 + (0.136 - 0.2)0.022 = 0.999 \\ 1 + 0.8n_z = 1 + 0.8\ {\cdot}0.022 = 1.018 \end{array} \right.$$

$$\frac{P}{P_{dz}} + 0.6K_y \frac{C_{my} M_y}{M_{dy}} + K_z \frac{C_{mz} M_z}{M_{dz}} = 0.314$$

Results

Table 669:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	29.1	29.10	none	
Tension capacity, gross yielding (kN)	4,564	4,564	none	
Tension capacity, rupture of critical section (kN)	5,398	5,398	none	
Ratio for tension	0	0	none	
Axial compression capacity about major axis (kN)	4,564	4,564	none	
Axial compression capacity about minor axis (kN)	4,360	4,360	none	
Ratio for compression	0.023	0.023	none	
Shear capacity in major axis (kN)	2,099	2,099	none	
Ratio for shear in major axis	0	0	none	
Shear capacity in minor axis (kN)	577.4	577.35	negligible	
Ratio for shear in minor axis	0.606	0.606	none	
Bending capacity about major axis (kN·m)	1,076	1,077	negligible	
Ratio for bending about major axis	0.325	0.325	none	
Bending capacity about minor axis (kN·m)	290.9	209.939	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio for bending about minor axis	0	0	none	
Ratio per Cl. 9.3.1.1	0.106	0.106	none	
Ratio per Eq.2 of Cl. 9.3.2.2	0.314	0.314	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - I Section - High Shear.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The default values of `KY` and `KZ` are used.
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The beam type is simply supported, specified by `CAN 2`
- The beam is laterally supported, specified by `LAT 1`
- The default values of `CMX`, `CMY`, and `CMZ` of 0.9 are used.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 23-Jun-19
END JOB INFORMATION
*****************************************************************************
*This problem has been created to validate the program calculated design
*results of laterally supported I section subjected to high shear force due
to
*applied axial compression and
*biaxial bending, designed using IS 800 2007 (LSD)
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TAPERED 0.55 0.008 0.55 0.4 0.02
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -350 1
1 CON GY -350 2
JOINT LOAD
2 FX -100
PERFORM ANALYSIS
SECTION 0.33 0.66
PRINT ANALYSIS RESULTS
PRINT MEMBER SECTION FORCES
PARAMETER 1
CODE IS800 LSD
CAN 2 ALL
FYLD 250000 ALL
TRACK 2 ALL
LAT 1 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                           |
 |  Member Section:       TAPERED               (AISC
SECTIONS)                   |
 |  Status: PASS  Ratio:   0.606  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Sec.
8.4                                                     |
 |  Critical Design Forces:  (Unit: KN
METE)                                |
 |      FX:     100.000E+00 C     FY:     350.000E+00      FZ:       0.000E
+00      |
 |      MX:       0.000E+00       MY:       0.000E+00      MZ:       0.000E
+00      |

|-----------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                         |
 |  AXX:     200.800E+00          IZZ:     121.257E+03          RZZ:
24.574E+00|
 |  AYY:      44.000E+00          IYY:      21.336E+03          RYY:
10.308E+00|
 |  AZZ:     160.000E+00          IXX:     222.037E+00           CW:
14.981E+06|
 |  ZEZ:       4.409E+03          ZPZ:       4.760E
```

```
+03                            |
 |  ZEY:       1.067E+03          ZPY:       1.608E
+03                            |

|-----------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                         |
 |  Actual Length:        3.000E
+00                                                   |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                      |
 |                 KZ:            1.000  KY:
1.000                      |
 |  Actual Ratio:  29.10 Allowable Ratio: 180.00  LOAD:
1                       |

|-----------------------------------------------------------------------------
-----|
 |  Section Class:      Compact; Flange Class:      Compact; Web Class:
Plastic|

|-----------------------------------------------------------------------------
-----|
      STAAD PLANE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                            |
 |  Member Section:       TAPERED              (AISC
SECTIONS)                    |

|-----------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                    |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                  |
 |                  NSF:            1.000    ALPHA:
0.800                  |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                       |
 |              Capacity:            4.564E+03     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                       |
 |              Capacity:            5.398E+03     As per:  Sec.
6.3               |

|-----------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                            |
 |  Buckling Class:   Major: a    Minor: b    As per:Cl.
```

```
7.1.2.2                         |
 |  Major Axis:   Design Force:      100.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            4.564E+03    As per:  Sec.
7.1.2             |
 |  Minor Axis:   Design Force:      100.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            4.360E+03    As per:  Sec.
7.1.2             |

|------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                         |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:            2.099E+03    As per:  Sec.
8.4               |
 |  Minor Axis:   Design Force:      350.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          577.350E+00    As per:  Sec.
8.4               |

|------------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                 |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   3.000E+00  Simp
Suppr  |
 |  Major Axis:   Design Force:     -350.000E+00    LC:     1  Loc:
1.000        |
 |                Capacity:            1.077E+03    As per:  Sec.
9.2.2(a           |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          290.939E+00    As per:  Sec.
8.2.1.2           |

|------------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                          |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900           |
 |  Section Strength:    Ratio:     0.106    As per:  Sec.
9.3.1.1                  |
 |                       LC:            1    Loc:
1.000                       |
 |  Overall Member Strength (Bending
+Compression):                                   |
 |  Equation 2:          Ratio:     0.314    As per:  Sec. 9.3.2.2   Ky :
1.0028  |
 |                       LC:            1    Loc:       1.000        Kz :
0.9986  |

|------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
```

```
Start( METE)      |

|
    |
 |  Tension                0.000      0                      0.000E
+00            |
 |  Compression            0.023      1                      0.000E
+00            |
 |  Shear Major            0.000      0                      0.000E
+00            |
 |  Shear Minor            0.606      1                      0.000E
+00            |
 |  Bend Major             0.325      1                      1.000E
+00            |
 |  Bend Minor             0.000      0                      0.000E
+00            |
 |  Sec. 9.3.1.1           0.106      1                      1.000E
+00            |
 |  Sec. 9.3.2.2(ii)       0.314      1                      1.000E
+00            |

|------------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - I Section - with LTB

Verify the design of an I section with lateral-torsional buckling per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

CISC. “Torsional Section Properties of steel Shapes”. Canadian Institute of Steel Construction. 2002. Willowdale, Ont.

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The member is a 3 m long cantilever subject to a 10 kN compressive force and uniform bending moments of 1 kN/m along both major and minor axes. The section used is a ISMB200 and is unbraced against lateral-torsional buckling. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, a modulus of elasticity of 205×(10)^3 MPa, and a Poisson's ratio, $\nu = 0.3$.

Validation

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Section Classification

$$\varepsilon = \sqrt{250 / f_y} = 1$$

$b = b_f/2 = 50$ mm

$b / t_f = 5 < 9.4\varepsilon = 9.4$

Since the section is symmetric about both axes, the neutral axis is at mid-depth.

$d_w / t_w = 27.72 < 84\varepsilon = 84$

Hence, from Table 2 of IS 800:2007, both the flange and web are plastic.

Therefore, the overall classification of the section is "plastic".

$h / b_f = 2 > 1.2$ and $t_f = 10\ mm < 40\ mm$

Hence, from Table 10 of IS 800:2007, the buckling class about the major axis is "a" and the same about the minor axis is "b".

Check Slenderness Ratio

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about the major axis, $k_z L / r_z = 36.20 < 180$

Slenderness ratio about the minor axis, $k_y L / r_y = 142.2 < 180$

So the slenderness is within the limit.

Axial Tension Capacity

Check for yielding of gross section:

$T_{dg} = A_g \times f_y / \gamma_{m0} = 700$ kN

Check for rupture of critical section:

$\alpha = 0.8$

$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 827.9$ kN

Block shear areas are not specified and therefore not checked.

No tension in the member, so critical ratio is 0.

Check Axial Compression

Axial compression capacity of major axis:

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\frac{f_y (k_z L / r_z)^2}{\pi^2 E}} = 0.402$

For buckling class "a", the imperfection factor, $\alpha = 0.21$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 0.602$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = 0.952$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\frac{\chi_z f_y}{\gamma_{m0}}, \frac{f_y}{\gamma_{m0}}\right) = 216.4\ MPa$

Therefore, axial compression capacity in the major axis: $P_{dz} = A_x f_{cdz} = 666.5\ kN$

The critical ratio for axial compression in major axis = $10 / 666.5 = 0.015$

Axial compression capacity of minor axis:

The non-dimensional slenderness ratio, $\lambda_y = \sqrt{\frac{f_y(k_y L / r_y)^2}{\pi^2 E}} = 1.581$

For buckling class "b", the imperfection factor, $\alpha = 0.34$

$$\phi_y = 0.5\left[1 + \alpha(\lambda_y - 0.2) + \lambda_y^2\right] = 1.985$$

$$\chi_y = \frac{1}{\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}} = 0.314$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdy} = min\left(\frac{\chi_y f_y}{\gamma_{m0}}, \frac{f_y}{\gamma_{m0}}\right) = 71.37\ MPa$

Therefore, axial compression capacity in the minor axis: $P_{dy} = A_x f_{cdy} = 219.8\ kN$, which governs.

The critical ratio for axial compression in minor axis = $10 / 219.8 = 0.045$

Check Shear Capacity

Major Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times b_f t_f = 2{,}000\ mm^2$

So, as per Cl. 8.4.1, $V_{pz} = \frac{A_{vz} f_{yw}}{\sqrt{3}} = 288.7\ kN$

Hence, shear capacity along major axis: $V_{dz} = \frac{V_{pz}}{\gamma_{m0}} = 262.4\ kN$

Minor Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vy} = h \times t_w = 1{,}140\ mm^2$

So, as per Cl. 8.4.1, $V_{py} = \frac{A_{vy} f_{yw}}{\sqrt{3}} = 164.5\ kN$

Hence, shear capacity along major axis: $V_{dy} = \frac{V_{py}}{\gamma_{m0}} = 149.6\ kN$

As per Cl. 8.4.2, $d_w / t_w = 27.72 < 67\varepsilon$, so resistance to shear buckling need not be checked.

Shear force along both axes = $3\ kN$.

Critical shear ratio along major axis: $3 / 149.6 = 0.020$

Critical shear ratio along minor axis: $3 / 262.4 = 0.011$

Check Bending Capacity

Since the section has been classified as "plastic", $\beta_b = 1$.

Major Axis Bending

The elastic lateral-torsional buckling moment:

$$M_{cr} = \sqrt{\frac{\pi^2 E I_y}{L_{LT}^2}\left(G I_t 6 + \frac{\pi^2 E I_w}{L_{LT}^2}\right)}$$

where

G	=	shear modulus, $= \frac{E}{2(1+\nu)} = 78{,}850\ MPa$
I_t	=	torsional constant, $I_x = 10.6\ cm^4$
d'	=	$h - t_f = 190\ mm$
I_w	=	warping constant, as taken from the CISC document “Torsional Section Properties of Steel Shapes” (see reference), $= \frac{(d')^2 b_f^3 t_f}{24} = 15{,}042\ cm^6$

$M_{cr} = 60.13$ kN·m

$$\lambda_{LT} = \min \begin{cases} \sqrt{\frac{\beta_b Z_{pz} f_y}{M_{cr}}} = 0.999 \\ \sqrt{\frac{1.2 Z_{cz} f_y}{M_{cr}}} = 1.027 \end{cases}$$

$$\phi_{LT} = 0.5\left[1 + 0.21(\lambda_{LT} - 0.2) + \lambda_{LT}^2\right] = 1.803$$

$$\chi_{LT} = \frac{1}{\phi_{LT} + \sqrt{\phi_{LT}^2 - \lambda_{LT}^2}} = 0.666$$

$$f_{bd} = \frac{\chi_{LT} f_y}{\gamma_{m0}} = 151.4 \text{ MPa}$$

So, the bending strength with respect to the major axis is $M_{dz} = \beta_b Z_{pz} f_{bd} = 36.35\ kN{\cdot}m$

Actual bending moment about the major axis is $M_z = 4.5\ kN{\cdot}m$.

The critical bending ratio in the major axis is $4.5\ /\ 36.35 = 0.124$

Minor Axis Bending

Actual shear force in both axes, $V = 3\ kN < 0.6V_d = 89.75\ kN$. Hence, the section is subjected to low shear.

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 10.46 \text{ kN·m}$$

$$\frac{1.2 Z_{ey} f_y}{\gamma_{m0}} = 9.341 \text{ kN·m}$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 9.341\ kN{\cdot}m$

Actual bending moment about the minor axis is $M_z = 4.5\ kN{\cdot}m$.

The critical bending ratio in the minor axis is $4.5\ /\ 9.341 = 0.482$

<u>Check Combined Interaction of Axial Compression and Bending</u>

Section strength calculation as per Cl. 9.3.1:

$$n = N / N_d = 0.014 < 0.2$$

Hence, from Table 17:

$$\alpha_1 = \max(5n\ ,\ 1) = 1$$

$$\alpha_2 = 2$$

$$M_{ndy} = M_{dy} = 9.341 \text{ kN·m} \qquad \text{(Cl. 9.3.1.2(c))}$$

$$1.11 \cdot M_{dz} \cdot (1 - n) = 39.78 \text{ kN} \cdot \text{m} > M_{dz}$$

$$M_{ndz} = M_{dz} = 36.35 \text{ kN} \cdot \text{m}$$

$$\left(\frac{M_y}{M_{ndy}}\right)^{\alpha_1} + \left(\frac{M_z}{M_{ndz}}\right)^{\alpha_2} = \left(\frac{4.5}{9.341}\right)^1 + \left(\frac{4.5}{36.35}\right)^2 = 0.497$$

Overall member strength in bending and axial compression

$$n_y = F_x / P_{dy} = 0.045$$

$$n_z = F_x / P_{dz} = 0.015$$

$$K_y = \min \left| \begin{array}{l} 1 + (\lambda_y - 0.2)n_y = 1 + (1.581 - 0.2) \cdot 0.045 = 1.063 \\ 1 + 0.8n_y = 1 + 0.8 \cdot 0.045 = 1.036 \end{array} \right.$$

$$K_z = \min \left| \begin{array}{l} 1 + (\lambda_z - 0.2)n_z = 1 + (0.402 - 0.2) \cdot 0.015 = 1.003 \\ 1 + 0.8n_z = 1 + 0.8 \cdot 0.015 = 1.012 \end{array} \right.$$

$$K_{LT} = \max \left| \begin{array}{l} 1 - \frac{0.1\lambda_{LT} n_y}{C_{mLT} - 0.25} = 1 - \frac{0.1 \times 0.999 \times 0.045}{0.9 - 0.25} = 0.993 \\ 1 - \frac{0.1 n_y}{C_{mLT} - 0.25} = 1 - \frac{0.1 \times 0.045}{0.9 - 0.25} = 0.993 \end{array} \right.$$

$$\frac{P}{P_{dy}} + K_y \frac{C_{my} M_y}{M_{dy}} + K_{LT} \frac{M_z}{M_{dz}} = 0.618$$

$$\frac{P}{P_{dz}} + 0.6K_y \frac{C_{my} M_y}{M_{dy}} + K_z \frac{C_{mz} M_z}{M_{dz}} = 0.397$$

Results

Table 670:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	142.3	142.24	negligible	
Tension capacity, gross yielding (kN)	700	700	none	
Tension capacity, rupture of critical section (kN)	827.9	827.904	negligible	
Axial compression capacity about major axis (kN)	666.5	666.502	negligible	
Axial compression capacity about minor axis (kN)	219.8	219.804	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Critical ratio for axial compression	0.045	0.045	none	
Shear capacity in major axis (kN)	262.4	262.432	negligible	
Shear capacity in minor axis (kN)	149.6	149.586	negligible	
Ratio for shear in major axis	0.011	0.011	none	
Ratio for shear in minor axis	0.020	0.020	none	
Bending capacity about major axis (kN·m)	36.35	36.347	negligible	
Bending capacity about minor axis (kN·m)	9.341	9.341	none	
Ratio for bending about major axis	0.124	0.124	none	
Ratio for bending about minor axis	0.482	0.482	none	
Ratio per Cl. 9.3.1.1	0.497	0.497	none	
Ratio per Eq. 1 of Cl. 9.3.2.2	0.618	0.618	none	
Ratio per Eq.2 of Cl. 9.3.2.2	0.397	0.397	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - I Section - with LTB.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The default values of `KY` and `KZ` are used (`KX 1.0` is directly specified).
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The effective length for lateral-torsional buckling is specified as `LX 3` (m).
- The beam type is a cantilever, specified by `CAN 1`

- The beam is laterally unsupported, specified by `LAT 0`
- The default values of `CMX`, `CMY`, and `CMZ` of 0.9 are used (`CMZ 0.9` is directly specified).

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Jun-2020
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of laterally unsupported I section subjected to axial compression and
*biaxial bending, designed using IS 800 2007 (LSD)
****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISMB200
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT MEMBER PROPERTY
PARAMETER 1
CODE IS800 LSD
LX 3 ALL
KX 1 ALL
CAN 1 ALL
CMZ 0.9 ALL
LAT 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|------------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                            |
 |  Member Section:  ST   ISMB200               (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.790  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Slenderness
(Compression)                                 |
 |  Critical Design Forces:  (Unit: KN
METE)                                   |
 |      FX:      10.000E+00 C     FY:       3.000E+00      FZ:       3.000E
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:       4.500E
+00      |

|------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                          |
 |  AXX:      30.800E+00          IZZ:       2.115E+03          RZZ:
8.287E+00|
 |  AYY:      11.400E+00          IYY:     137.000E+00          RYY:
2.109E+00|
 |  AZZ:      20.000E+00          IXX:      10.600E+00           CW:
15.042E+03|
 |  ZEZ:     211.500E+00          ZPZ:     240.000E
+00                             |
 |  ZEY:      27.400E+00          ZPY:      46.000E
+00                             |

|------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                   |
 |  Actual Length:        3.000E
+00                                          |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                     |
 |                  KZ:           1.000  KY:
1.000                     |
 |  Actual Ratio: 142.24 Allowable Ratio: 180.00  LOAD:
1                       |

|------------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                  -- PAGE NO.
```

```
5
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                           |
 |  Member Section:  ST   ISMB200               (INDIAN
SECTIONS)                     |

|------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                 |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                |
 |                   NSF:            1.000    ALPHA:
0.800                |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                      |
 |                Capacity:          700.000E+00     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                      |
 |                Capacity:          827.904E+00     As per:  Sec.
6.3               |

|------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                             |
 |  Buckling Class:   Major: a    Minor: b    As per:Cl.
7.1.2.2                      |
 |  Major Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          666.502E+00    As per:  Sec.
7.1.2              |
 |  Minor Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          219.804E+00    As per:  Sec.
7.1.2              |

|------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                           |
 |  Major Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          262.432E+00    As per:  Sec.
8.4                |
 |  Minor Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          149.586E+00    As per:  Sec.
8.4                |

|------------------------------------------------------------------------------
-----|
```

```
 |  Bending:    (Unit:  KN
METE)                                                   |
 |  Parameters:   Laterally Unsupported    KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major Axis:   Design Force:        4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:           36.347E+00    As per:  Sec.
8.2.2             |
 |  Minor Axis:   Design Force:       -4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:            9.341E+00    As per:  Sec.
8.2.1.2           |

|-------------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                            |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.497    As per:  Sec.
9.3.1.1                  |
 |                       LC:            1    Loc:
0.000                       |
 |  Overall Member Strength (Bending
+Compression):                                 |
 |  Equation 1:          Ratio:     0.618    As per:  Sec. 9.3.2.2   Ky :
1.0364  |
 |                       LC:            1    Loc:       0.000        KLT:
0.9930  |
 |  Equation 2:          Ratio:     0.396    As per:  Sec. 9.3.2.2   Ky :
1.0364  |
 |                       LC:            1    Loc:       0.000        Kz :
1.0030  |

|-------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                0.000      0                      0.000E
+00           |
 |  Compression            0.045      1                      0.000E
+00           |
 |  Shear Major            0.011      1                      0.000E
+00           |
 |  Shear Minor            0.020      1                      0.000E
+00           |
 |  Bend Major             0.124      1                      0.000E
+00           |
 |  Bend Minor             0.482      1                      0.000E
+00           |
 |  Sec. 9.3.1.1           0.497      1                      0.000E
+00           |
 |  Sec. 9.3.2.2 (i)       0.618      1                      0.000E
+00           |
 |  Sec. 9.3.2.2(ii)       0.396      1                      0.000E
```

```
+00                |

|---------------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - I Section - without LTB

Verify the design of an I section subject to axial force and biaxial bending per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The member is a 3 m long cantilever subject to a 10 kN compressive force and uniform bending moments of 1 kN/m along both major and minor axes. The section used is a ISMB200 and is braced to prevent lateral-torsional buckling. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, and a modulus of elasticity of 205×(10)^3 MPa.

Validation

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Section Classification

$$\varepsilon = \sqrt{250 / f_y} = 1$$

$$b = b_f/2 = 50 \text{ mm}$$

$$b / t_f = 5 < 9.4\varepsilon = 9.4$$

Since the section is symmetric about both axes, the neutral axis is at mid-depth.

$$d_w / t_w = 27.72 < 84\varepsilon = 84$$

Hence, from Table 2 of IS 800:2007, both the flange and web are plastic.

Therefore, the overall classification of the section is "plastic".

$h / b_f = 2 > 1.2$ and $t_f = 10 \text{ mm} < 40 \text{ mm}$

Hence, from Table 10 of IS 800:2007, the buckling class about the major axis is "a" and the same about the minor axis is "b".

Check Slenderness Ratio

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about the major axis, $k_z L / r_z = 36.20 < 180$

Slenderness ratio about the minor axis, $k_y L \ / \ r_y = 142.2 < 180$

So the slenderness is within the limit.

Axial Tension Capacity

Check for yielding of gross section:

$$T_{dg} = A_g \times f_y / \gamma_{m0} = 700 \text{ kN}$$

Check for rupture of critical section:

$$\alpha = 0.8$$

$$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 827.9 \text{ kN}$$

Block shear areas are not specified and therefore not checked.

No tension in the member, so critical ratio is 0.

Check Axial Compression

Axial compression capacity of major axis:

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\dfrac{f_y (k_z L \ / r_z)^2}{\pi^2 E}} = 0.402$

For buckling class "a", the imperfection factor, $\alpha = 0.21$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 0.602$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = 0.952$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\dfrac{\chi_z f_y}{\gamma_{m0}}, \dfrac{f_y}{\gamma_{m0}}\right) = 216.4 \ MPa$

Therefore, axial compression capacity in the major axis: $P_{dz} = A_x f_{cdz} = 666.5 \ kN$

The critical ratio for axial compression in major axis = $10 \ / \ 666.5 = 0.015$

Axial compression capacity of minor axis:

The non-dimensional slenderness ratio, $\lambda_y = \sqrt{\dfrac{f_y (k_y L \ / r_y)^2}{\pi^2 E}} = 1.581$

For buckling class "b", the imperfection factor, $\alpha = 0.34$

$$\phi_y = 0.5\left[1 + \alpha(\lambda_y - 0.2) + \lambda_y^2\right] = 1.985$$

$$\chi_y = \frac{1}{\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}} = 0.314$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdy} = min\left(\dfrac{\chi_y f_y}{\gamma_{m0}}, \dfrac{f_y}{\gamma_{m0}}\right) = 71.37 \ MPa$

Therefore, axial compression capacity in the minor axis: $P_{dy} = A_x f_{cdy} = 219.8 \ kN$, which governs.

The critical ratio for axial compression in minor axis = $10 \ / \ 219.8 = 0.045$

Check Shear Capacity

Major Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times b_f t_f = 2{,}000\ mm^2$

So, as per Cl. 8.4.1, $V_{pz} = \frac{A_{vz} f_{yw}}{\sqrt{3}} = 288.7\ kN$

Hence, shear capacity along major axis: $V_{dz} = \frac{V_{pz}}{\gamma_{m0}} = 262.4\ kN$

Minor Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vy} = h \times t_w = 1{,}140\ mm^2$

So, as per Cl. 8.4.1, $V_{py} = \frac{A_{vy} f_{yw}}{\sqrt{3}} = 164.5\ kN$

Hence, shear capacity along major axis: $V_{dy} = \frac{V_{py}}{\gamma_{m0}} = 149.6\ kN$

As per Cl. 8.4.2, $d_w\ /\ t_w = 27.72 < 67\varepsilon$, so resistance to shear buckling need not be checked.

Shear force along both axes = $3\ kN$.

Critical shear ratio along major axis: $3\ /\ 149.6 = 0.020$

Critical shear ratio along minor axis: $3\ /\ 262.4 = 0.011$

Check Bending Capacity

Since the section has been classified as "plastic", $\beta_b = 1$.

Actual shear force in both axes, $V = 3\ kN < 0.6V_d = 89.75\ kN$. Hence, the section is subjected to low shear.

Major Axis Bending

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 54.55 \text{ kN}\cdot\text{m}$$

$$\frac{1.2 Z_{ey} f_y}{\gamma_{m0}} = 72.10 \text{ kN}\cdot\text{m}$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 54.55\ kN \cdot m$

Actual bending moment about the major axis is $M_z = 4.5\ kN \cdot m$.

The critical bending ratio in the major axis is $4.5\ /\ 54.55 = 0.083$

Minor Axis Bending

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 10.46 \text{ kN}\cdot\text{m}$$

$$\frac{1.2 Z_{ey} f_y}{\gamma_{m0}} = 9.341 \text{ kN}\cdot\text{m}$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 9.341\ kN \cdot m$

Actual bending moment about the minor axis is $M_z = 4.5\ kN \cdot m$.

The critical bending ratio in the minor axis is $4.5\ /\ 9.341 = 0.482$

Check Combined Interaction of Axial Compression and Bending

Section strength calculation as per Cl. 9.3.1:

$$n = N / N_d = 0.014 < 0.2$$

Hence, from Table 17:

$$\alpha_1 = \max(5n , 1) = 1$$

$$\alpha_2 = 2$$

$$M_{ndy} = M_{dy} = 9.341 \text{ kN·m} \quad \text{(Cl. 9.3.1.2(c))}$$

$$1.11 \cdot M_{dz} \cdot (1 - n) = 59.70 \text{ kN·m} > M_{dz}$$

$$M_{ndz} = M_{dz} = 54.55 \text{ kN·m}$$

$$\left(\frac{M_y}{M_{ndy}}\right)^{\alpha_1} + \left(\frac{M_z}{M_{ndz}}\right)^{\alpha_2} = \left(\frac{4.5}{9.341}\right)^1 + \left(\frac{4.5}{54.55}\right)^2 = 0.489$$

Overall member strength in bending and axial compression

$$n_y = F_x / P_{dy} = 0.045$$

$$n_z = F_x / P_{dz} = 0.015$$

$$K_y = \min \left| \begin{array}{l} 1 + (\lambda_y - 0.2)n_y = 1 + (1.581 - 0.2) \cdot 0.045 = 1.063 \\ 1 + 0.8n_y = 1 + 0.8 \cdot 0.045 = 1.036 \end{array} \right.$$

$$K_z = \min \left| \begin{array}{l} 1 + (\lambda_z - 0.2)n_z = 1 + (0.402 - 0.2) \cdot 0.015 = 1.003 \\ 1 + 0.8n_z = 1 + 0.8 \cdot 0.015 = 1.012 \end{array} \right.$$

$$\frac{P}{P_{dz}} + 0.6K_y \frac{C_{my} M_y}{M_{dy}} + K_z \frac{C_{mz} M_z}{M_{dz}} = 0.359$$

Results

Table 671:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	142.3	142.24	negligible	
Tension capacity, gross yielding (kN)	700	700	none	
Tension capacity, rupture of critical section (kN)	827.9	827.904	negligible	
Axial compression capacity about major axis (kN)	666.5	666.502	negligible	
Axial compression capacity about minor axis (kN)	219.8	219.804	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Critical ratio for axial compression	0.045	0.045	none	
Shear capacity in major axis (kN)	262.4	262.432	negligible	
Shear capacity in minor axis (kN)	149.6	149.586	negligible	
Ratio for shear in major axis	0.011	0.011	none	
Ratio for shear in minor axis	0.020	0.020	none	
Bending capacity about major axis (kN·m)	54.55	54.545	negligible	
Bending capacity about minor axis (kN·m)	9.341	9.341	none	
Ratio for bending about major axis	0.082	0.082	none	
Ratio for bending about minor axis	0.482	0.482	none	
Ratio per Cl. 9.3.1.1	0.489	0.489	none	
Ratio per Eq.2 of Cl. 9.3.2.2	0.359	0.359	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - I Section - without LTB.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The default values of `KY` and `KZ` are used (`KX 1.0` is directly specified).
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The effective length for lateral-torsional buckling is specified as `LX 3` (m).
- The beam type is a cantilever, specified by `CAN 1`
- The beam is laterally supported, specified by `LAT 1`

- The default values of CMX, CMY, and CMZ of 0.9 are used (`CMZ 0.9` is directly specified).

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Jun-2020
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of laterally supported I section subjected to axial compression and
*biaxial bending, designed using IS 800 2007 (LSD)
****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST ISMB200
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT MEMBER PROPERTY
PARAMETER 1
CODE IS800 LSD
LX 3 ALL
KX 1 ALL
CAN 1 ALL
CMZ 0.9 ALL
LAT 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************
```

```
|--------------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                          |
 |  Member Section:  ST   ISMB200              (INDIAN
SECTIONS)                   |
 |  Status: PASS  Ratio:   0.790  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Slenderness
(Compression)                                  |
 |  Critical Design Forces:  (Unit: KN
METE)                                    |
 |      FX:      10.000E+00 C     FY:       3.000E+00      FZ:       3.000E
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:       4.500E
+00      |

|--------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                           |
 |  AXX:      30.800E+00          IZZ:       2.115E+03          RZZ:
8.287E+00|
 |  AYY:      11.400E+00          IYY:     137.000E+00          RYY:
2.109E+00|
 |  AZZ:      20.000E+00          IXX:      10.600E+00           CW:
15.042E+03|
 |  ZEZ:     211.500E+00          ZPZ:     240.000E
+00                             |
 |  ZEY:      27.400E+00          ZPY:      46.000E
+00                             |

|--------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                       |
 |  Actual Length:        3.000E
+00                                             |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                     |
 |                  KZ:           1.000  KY:
1.000                     |
 |  Actual Ratio: 142.24 Allowable Ratio: 180.00  LOAD:
1                       |

|--------------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|--------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************
```

```
|--------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                             |
 |  Member Section:  ST   ISMB200               (INDIAN
SECTIONS)                    |

|--------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                   |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                |
 |                   NSF:            1.000    ALPHA:
0.800                |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                       |
 |                Capacity:          700.000E+00     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                       |
 |                Capacity:          827.904E+00     As per:  Sec.
6.3               |

|--------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                               |
 |  Buckling Class:   Major: a    Minor: b    As per:Cl.
7.1.2.2                    |
 |  Major Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          666.502E+00    As per:  Sec.
7.1.2            |
 |  Minor Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          219.804E+00    As per:  Sec.
7.1.2            |

|--------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                           |
 |  Major Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          262.432E+00    As per:  Sec.
8.4              |
 |  Minor Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:          149.586E+00    As per:  Sec.
8.4              |

|--------------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                   |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   3.000E+00
Cantilever  |
```

```
 |  Major Axis:   Design Force:        4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:           54.545E+00    As per:  Sec.
8.2.1.2          |
 |  Minor Axis:   Design Force:       -4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:            9.341E+00    As per:  Sec.
8.2.1.2          |

|-----------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                           |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.489    As per:  Sec.
9.3.1.1                 |
 |                       LC:            1    Loc:
0.000                      |
 |  Overall Member Strength (Bending
+Compression):                                 |
 |  Equation 2:          Ratio:     0.359    As per:  Sec. 9.3.2.2   Ky :
1.0364  |
 |                       LC:            1    Loc:       0.000        Kz :
1.0030  |

|-----------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                0.000      0                       0.000E
+00          |
 |  Compression            0.045      1                       0.000E
+00          |
 |  Shear Major            0.011      1                       0.000E
+00          |
 |  Shear Minor            0.020      1                       0.000E
+00          |
 |  Bend Major             0.082      1                       0.000E
+00          |
 |  Bend Minor             0.482      1                       0.000E
+00          |
 |  Sec. 9.3.1.1           0.489      1                       0.000E
+00          |
 |  Sec. 9.3.2.2(ii)       0.359      1                       0.000E
+00          |

|-----------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - I Section with Cover Plate

Verify the design of an I section with cover plates subject to axial force and biaxial bending per the IS 800:2007 limit state method.

Reference

BIS. *IS 800: General Construction in Steel - Code of Practice*. 2007. Bureau of Indian Standards. New Delhi.

BIS. *IS 808: Dimensions for Hot Rolled Steel Beam, Column, Channel and Angle Sections*. 1989. Bureau of Indian Standards. New Delhi.

Related Links

- *D8.A.2 Design Process* (on page 1608)

Details

The structure consists of a 5 m long column with fixed based and a 5 m cantilever beam at the top. The resulting loads on the column to be designed are:

a 55.98 kN·m moment about the major axis
a 10 kN·m moment about the minor axis
a 23.984 kN compressive force
a 2 kN shear force along the major axis
a 1.2 kN shear force along the minor axis

The section used is an ISMB500 with a 250 mm × 10 mm top cover plate and a 200 mm × 10 mm bottom cover plate. The section is braced to prevent lateral-torsional buckling. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, and a modulus of elasticity of $205\times(10)^3$ MPa.

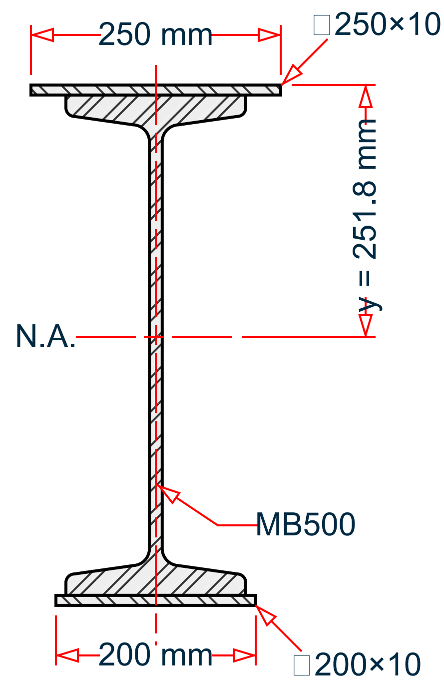


Figure 498: I section with cover plates

Validation

<u>Section Properties</u>

Overall depth of the whole section, $h = D + t_{pl1} + t_{pl2} = 500 + 10 + 10 = 520\ mm$

Depth of web of IS MB500, $d_w = D - 2 \times t_f = 500 - 2 \times 17.2 = 465.6\ mm$

Cross-section area of top cover plate, $A_{top} = 25\ cm \times 1\ cm = 25\ cm^2$

Cross-section area of bottom cover plate, $A_{bottom} = 20\ cm \times 1\ cm = 20\ cm^2$

Cross-section area, $A_x = 111\ cm^2 + 25\ cm^2 + 20\ cm^2 = 156\ cm^2$

Distance of CG from top of overall section: $y = \dfrac{A_{top}\frac{t_{pl1}}{2} + A_x'\left(t_{pl1} + \frac{D}{2}\right) + A_{bottom}\left(t_{pl1} + D + \frac{t_{pl2}}{2}\right)}{A_x} = 251.8\ mm$

Overall moment of inertia about the major axis:

$$I_z = I_z' + A_x'\left(\frac{D}{2} + t_{pl1} - y\right)^2 + \frac{b_{pl1}t_{pl1}^3}{12} + A_{top}\left(\frac{t_{pl1}}{2} - y\right)^2 + \frac{b_{pl2}t_{pl2}^3}{12} + A_{bottom}\left(t_{pl1} + D + \frac{t_{pl2}}{2} - y\right)^2$$

$$I_z = 45{,}200 \times (10)^4 + 111 \times (10)^2\left(\frac{500}{2} + 10 - 251.8\right)^2 + \frac{250 \times 10^3}{12}$$

$$+2{,}500\left(\frac{10}{2} - 251.8\right)^2 + \frac{200 \times 10^3}{12} + 2{,}000\left(10 + 500 + \frac{5}{2} - 251.8\right)^2$$

$$= 743.6(10)^6\ \mathrm{mm}^4 = 74{,}360\ \mathrm{cm}^4$$

Overall moment of inertia about the minor axis (section is symmetric about this axis):

$$I_y = I_y' + \frac{t_{p1} b_{p1}^3}{12} + \frac{t_{p2} b_{p2}^3}{12} = 1{,}370 \times (10)^4 + \frac{10 \times 250^3}{12} + \frac{10 \times 200^3}{12} = 33.39(10)^6 \text{ mm}^4 = 3{,}339 \text{ cm}^4$$

Radius of gyration about the major axis, $r_z = \sqrt{I_z / A_x} = \sqrt{74{,}360 / 156} = 21.8\ cm$

Radius of gyration about the minor axis, $r_y = \sqrt{I_y / A_x} = \sqrt{3{,}339 / 156} = 4.63\ cm$

Elastic section modulus about the major axis, $Z_{ez} = \frac{I_z}{h - y} = \frac{74{,}360}{52 - 25.18} = 2{,}773\ cm^3$

Elastic section modulus about the minor axis, $Z_{ey} = \frac{I_y}{b_{p1} / 2} = \frac{3{,}339}{25 / 2} = 267.1\ cm^3$

Distance to equal area axis from top, $c_p = \frac{(D + 2t_{p1} - t_f)t_w + b_{p2}t_{p2} - b_{p1}t_{p1}}{2t_W} = 226.9\ mm$

Plastic section modulus about the major axis,

$$Z_{pz} = b_{b1}t_{p1}\left(c_p - \frac{t_{p1}}{2}\right) + b_f t_f\left(c_p - t_{p1} - \frac{t_f}{2}\right) + t_w\frac{(c_p - t_{p1} - t_f)^2}{2} + t_w\frac{(D + t_{p1} - c_p - t_f)^2}{2}$$
$$+ b_f t_f\left(D + t_{p1} - c_p - \frac{t_f}{2}\right) + b_{b2}t_{p2}\left(D + t_{p1} + \frac{t_{p2}}{2} - c_p\right) = 3{,}190 \text{ cm}^3$$

Plastic section modulus about the minor axis, $Z_{py} = \frac{2t_f b_f^2}{4} + \frac{d_w t_w^2}{4} + \frac{t_{p1}b_{p1}^2}{4} + \frac{t_{p2}b_{p2}^2}{4} = 547\ cm^3$

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Section Classification

$$\varepsilon = \sqrt{250 / f_y} = 1$$

Section classification of different portions of the section must be checked separately. Checking is performed per Table 2 of IS 800:2007 as indicated in the following figure:

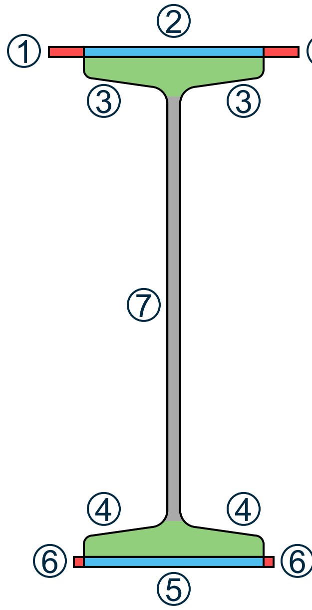


Figure 499: Different portions of the cross section to be checked for section classification

1. Outstanding element of top cover plate: $\frac{b_{p1} - b_f}{2t_{p1}} = 3.5 < 9.4\varepsilon = 9.4$ hence, plastic.
2. Internal element of top cover plate: $\frac{b_f}{t_{p1}} = 18 < 29.3\varepsilon = 29.3$ hence, plastic.
3. Internal element of top flange $\frac{b_f}{2t_f} = 5.23 < 29.3\varepsilon = 29.3$ hence, plastic.
4. Internal element of bottom flange $\frac{b_f}{2t_f} = 5.23 < 29.3\varepsilon = 29.3$ hence, plastic.
5. Internal element of bottom cover plate $\frac{b_f}{t_{p2}} = 18 < 29.3\varepsilon = 29.3$ hence, plastic.
6. Outstanding element of bottom cover plate $\frac{b_{p2} - b_f}{2t_{p2}} = 1 < 9.4\varepsilon = 9.4$ hence, plastic.
7. Web of ISMB section $\frac{d_2}{t_w} = 45.65 < 84\varepsilon = 84$ hence, plastic.

Overall classification of the section is based on the most critical class among the different classes identified above. Here, the overall section is classified as “plastic”.

Since $t_f = 17.2\ mm < 40\ mm$

Hence, from Table 10, as per the requirements of welded I section, the buckling class about major axis buckling is “b” and the same about minor axis buckling is “c”.

<u>Check Slenderness Ratio</u>

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about the major axis, $k_zL\ /\ r_z = 1.0 \times 500\ /\ 21.8 = 22.9 < 180$

Slenderness ratio about the minor axis, $k_y L \ / \ r_y = 1.0 \times 500 \ / \ 4.63 = 108.0 < 180$

So the slenderness is within the limit.

Axial Tension Capacity

Check for yielding of gross section:

$$T_{dg} = A_g \times f_y / \gamma_{m0} = 3{,}545 \text{ kN}$$

Check for rupture of critical section:

$$\alpha = 0.8$$

$$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 4{,}193 \text{ kN}$$

Block shear areas are not specified and therefore not checked.

No tension in the member, so critical ratio is 0.

Check Axial Compression

Axial compression capacity of major axis:

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\dfrac{f_y (k_z L \ / \ r_z)^2}{\pi^2 E}} = 0.255$

For buckling class “b”, the imperfection factor, $\alpha = 0.34$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 0.542$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = 0.981$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\dfrac{\chi_z f_y}{\gamma_{m0}}, \ \dfrac{f_y}{\gamma_{m0}}\right) = 222.9 \ MPa$

Therefore, axial compression capacity in the major axis: $P_{dz} = A_x f_{cdz} = 3{,}477 \ kN$

Axial compression capacity of minor axis:

The non-dimensional slenderness ratio, $\lambda_y = \sqrt{\dfrac{f_y (k_y L \ / \ r_y)^2}{\pi^2 E}} = 1.201$

For buckling class “c”, the imperfection factor, $\alpha = 0.49$

$$\phi_y = 0.5\left[1 + \alpha(\lambda_y - 0.2) + \lambda_y^2\right] = 1.467$$

$$\chi_y = \frac{1}{\phi_y + \sqrt{\phi_y^2 - \lambda_y^2}} = 0.433$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdy} = min\left(\dfrac{\chi_y f_y}{\gamma_{m0}}, \ \dfrac{f_y}{\gamma_{m0}}\right) = 98.43 \ MPa$

Therefore, axial compression capacity in the minor axis: $P_{dy} = A_x f_{cdy} = 1{,}536 \ kN$, which governs.

The critical ratio for axial compression in minor axis = $23.98 \ / \ 1{,}536 = 0.007$

Check Shear Capacity

Major Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times b_f t_f + b_{p1} t_{p1} + b_{p2} t_{p2} = 10,692\ mm^2$

So, as per Cl. 8.4.1, $V_{pz} = \dfrac{A_{vz} f_{yw}}{\sqrt{3}} = 1,543\ kN$

Hence, shear capacity along major axis: $V_{dz} = \dfrac{V_{pz}}{\gamma_{m0}} = 1,403\ kN$

Minor Axis

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vy} = d_w \times t_w = 4,749\ mm^2$

So, as per Cl. 8.4.1, $V_{py} = \dfrac{A_{vy} f_{yw}}{\sqrt{3}} = 685.5\ kN$

Hence, shear capacity along major axis: $V_{dy} = \dfrac{V_{py}}{\gamma_{m0}} = 623.2\ kN$

As per Cl. 8.4.2, $d_w / t_w = 45.65 < 67\varepsilon$, so resistance to shear buckling need not be checked.

Shear force along major axis = $2\ kN$.

Critical shear ratio along major axis: $2 / 1,403 = 0.001$

Shear force along minor axis = $1.2\ kN$.

Critical shear ratio along minor axis: $1.2 / 623.2 = 0.002$

<u>Check Bending Capacity</u>

Since the section has been classified as “plastic”, $\beta_b = 1$.

The section is low shear in both axes.

Major Axis Bending

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 724.9\ \mathrm{kN \cdot m}$$

Actual bending moment about the major axis is $M_z = 55.98\ kN \cdot m$.

The critical bending ratio in the major axis is $55.98 / 724.9 = 0.077$

Minor Axis Bending

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 124.3\ \mathrm{kN \cdot m}$$

Actual bending moment about the minor axis is $M_z = 10\ kN \cdot m$.

The critical bending ratio in the minor axis is $10 / 124.3 = 0.080$

<u>Check Combined Interaction of Axial Compression and Bending</u>

Section strength calculation as per Cl. 9.3.1:

$$n = N / N_d = 0.007 < 0.2$$

Hence, from Table 17:

$$\alpha_1 = \max(5n , 1) = 1$$

$$\alpha_2 = 2$$

As per 9.3.1.2(b),

$$a = \frac{A - 2b_f t_f}{A} = 0.603$$

$$n_1 = \max(n,a) = 0.603$$

$$M_{ndy} = \min \begin{vmatrix} M_{dy} = 124.3 \text{ kN}\cdot\text{m} \\ M_{dy}\left[1 - \left(\frac{n_1 - a}{1 - a}\right)^2\right] = 124.3 \text{ kN}\cdot\text{m} \end{vmatrix}$$

$$M_{ndz} = \min \begin{vmatrix} M_{dz} = 724.9 \text{ kN}\cdot\text{m} \\ M_{dz}\left(\frac{1 - n}{1 - 0.5a}\right) = 724.2 \text{ kN}\cdot\text{m} \end{vmatrix}$$

Therefore:

$$\left(\frac{M_y}{M_{ndy}}\right)^{a_1} + \left(\frac{M_z}{M_{ndz}}\right)^{a_2} = \left(\frac{10}{124.3}\right)^1 + \left(\frac{55.98}{724.2}\right)^2 = 0.086$$

Overall member strength in bending and axial compression

$$n_y = F_x / P_{dy} = 0.016$$

$$n_z = F_x / P_{dz} = 0.007$$

$$K_y = \min \begin{vmatrix} 1 + (\lambda_y - 0.2)n_y = 1 + (1.201 - 0.2) \times 0.016 = 1.016 \\ 1 + 0.8n_y = 1 + 0.8 \times 0.016 = 1.013 \end{vmatrix}$$

$$K_z = \min \begin{vmatrix} 1 + (\lambda_z - 0.2)n_z = 1 + (0.255 - 0.2) \times 0.007 = 1.000 \\ 1 + 0.8n_z = 1 + 0.8 \times 0.007 = 1.006 \end{vmatrix}$$

$$\frac{P}{P_{dz}} + 0.6K_y\frac{C_{my}M_y}{M_{dy}} + K_z\frac{C_{mz}M_z}{M_{dz}} = 0.120$$

Results

Table 672:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	108.0	108.08	negligible	
Tension capacity, gross yielding (kN)	3,545	3,545	none	
Tension capacity, rupture of critical section (kN)	4,193	4,163	negligible	
Axial compression capacity about major axis (kN)	3,477	3,477	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Axial compression capacity about minor axis (kN)	1,536	1,536	none	
Critical ratio for axial compression	0.016	0.016	none	
Shear capacity in major axis (kN)	1,403	1,403	none	
Shear capacity in minor axis (kN)	623.2	623.16	negligible	
Ratio for shear in major axis	0.001	0.001	none	
Ratio for shear in minor axis	0.002	0.002	none	
Bending capacity about major axis (kN·m)	724.9	724.745	negligible	
Bending capacity about minor axis (kN·m)	124.3	124.317	negligible	
Ratio for bending about major axis	0.077	0.077	none	
Ratio for bending about minor axis	0.080	0.080	none	
Ratio per Cl. 9.3.1.1	0.086	0.086	none	
Ratio per Eq.2 of Cl. 9.3.2.2	0.120	0.120	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - I Section with Cover Plate.std` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The default values of `KY` and `KZ` are used.
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The beam type is a general member, specified by `CAN 0`

- The beam is laterally supported, specified by `LAT 1`
- The default values of `CMX`, `CMY`, and `CMZ` of 0.9 are used.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Oct-08
END JOB INFORMATION
INPUT WIDTH 79
******************************************************************************
*
*THIS EXAMPLE DEMONSTRATES THE DESIGN OF A WIDE FLANGE SECTION WITH
*ADDITIONAL TOP AND BOTTOM COVER PLATES USING PROVISIONS OF IS 800:2007 LSD
******************************************************************************
*
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 2 TABLE TB ISMB500 WP 0.25 TH 0.01 BW 0.2 BT 0.01
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
3 FY -2
MEMBER LOAD
2 UNI GY -2
JOINT LOAD
2 FX 1.2
2 FZ -2
SELFWEIGHT Y -1 LIST ALL
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
PARAMETER 1
CODE IS800 LSD
CAN 0 MEMB 1
LAT 1 ALL
TRACK 2 MEMB 1
CHECK CODE MEMB 1
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-------------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                           |
 |  Member Section:  TB   ISMB500               (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.600  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Slenderness
(Compression)                                |
 |  Critical Design Forces:  (Unit: KN
METE)                                  |
 |      FX:      23.984E+00 C     FY:       1.200E+00      FZ:       2.000E
+00      |
 |      MX:       0.000E+00       MY:     -10.000E+00      MZ:      55.980E
+00      |

|-------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                           |
 |  AXX:     156.000E+00          IZZ:      74.381E+03          RZZ:
21.836E+00|
 |  AYY:      47.491E+00          IYY:       3.339E+03          RYY:
4.626E+00|
 |  AZZ:     106.920E+00          IXX:     117.000E+00           CW:
2.057E+06|
 |  ZEZ:       2.773E+03          ZPZ:       3.189E
+03                           |
 |  ZEY:     267.100E+00          ZPY:     547.000E
+00                           |

|-------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                  |
 |  Actual Length:        5.000E
+00                                         |
 |  Parameters:     LZ:        5.000E+00  LY:        5.000E
+00                     |
 |                 KZ:           1.000  KY:
1.000                      |
 |  Actual Ratio: 108.08 Allowable Ratio: 180.00  LOAD:
1                        |

|-------------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|-------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                   -- PAGE NO.
```

```
6
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                           |
 |  Member Section:  TB   ISMB500               (INDIAN
SECTIONS)                  |

|------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                 |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                 |
 |                    NSF:            1.000    ALPHA:
0.800                 |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                        |
 |                Capacity:            3.545E+03     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                        |
 |                Capacity:            4.193E+03     As per:  Sec.
6.3               |

|------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                             |
 |  Buckling Class:   Major: b    Minor: c    As per:Cl.
7.1.2.2                      |
 |  Major Axis:   Design Force:       23.984E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            3.477E+03    As per:  Sec.
7.1.2               |
 |  Minor Axis:   Design Force:       23.984E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            1.536E+03    As per:  Sec.
7.1.2               |

|------------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                          |
 |  Major Axis:   Design Force:        2.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            1.403E+03    As per:  Sec.
8.4                 |
 |  Minor Axis:   Design Force:        1.200E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          623.160E+00    As per:  Sec.
8.4                 |

|------------------------------------------------------------------------------
-----|
```

```
 |  Bending:    (Unit:  KN
METE)                                                      |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   5.000E+00
General     |
 |  Major Axis:   Design Force:       55.980E+00    LC:     1  Loc:
0.000       |
 |              Capacity:          724.754E+00    As per:  Sec.
8.2.1.2           |
 |  Minor Axis:   Design Force:      -10.000E+00    LC:     1  Loc:
0.000       |
 |              Capacity:          124.318E+00    As per:  Sec.
8.2.1.2           |

|------------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                             |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.086    As per:  Sec.
9.3.1.1                 |
 |                    LC:            1    Loc:
0.000                    |
 |  Overall Member Strength (Bending
+Compression):                                 |
 |  Equation 2:          Ratio:     0.120    As per:  Sec. 9.3.2.2   Ky :
1.0125  |
 |                    LC:            1    Loc:       0.000        Kz :
1.0004  |

|------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension               0.000       0                      0.000E
+00           |
 |  Compression           0.016       1                      0.000E
+00           |
 |  Shear Major           0.001       1                      0.000E
+00           |
 |  Shear Minor           0.002       1                      0.000E
+00           |
 |  Bend Major            0.077       1                      0.000E
+00           |
 |  Bend Minor            0.080       1                      0.000E
+00           |
 |  Sec. 9.3.1.1          0.086       1                      0.000E
+00           |
 |  Sec. 9.3.2.2(ii)      0.120       1                      0.000E
+00           |

|------------------------------------------------------------------------------
-----|
```

V. IS 800 2007 LSD - Pipe - Tension and Bending

Verify the design of a pipe section subject to axial tension and biaxial bending per the IS 800:2007 limit state method.

Details

The member is a 3 m long cantilever subject to a 10 kN tensile force and uniform bending moments of 1 kN/m along both major and minor axes. The section used is a 90 mm nominal bore with a wall thickness of 3.6 mm (STAAD.Pro section designation PIP10106L) and is braced to prevent lateral-torsional buckling. The steel has a yield strength of 250 MPa, an ultimate tensile strength of 420 MPa, and a modulus of elasticity of 205×(10)^3 MPa.

Validation

Section Properties

Outer diameter, $OD = 101.6\ mm$

Wall thickness, $t = 3.6\ mm$

Inner diameter, $ID = 101.6 - 2 \times 3.6 = 94.4\ mm$

Cross-section area, $A_x = \frac{\pi}{4}\left(101.6^2 - 94.4^2\right) = 11.08\ cm^2$

Moment of inertia about both axes, $I = \frac{\pi}{64}\left(101.6^4 - 94.4^4\right) = 133.2\ cm^4$

Radius of gyration about both axes, $r = \sqrt{I / A_x} = 3.47\ cm$

Elastic section modulus about both axes, $Z_e = \frac{I}{OD / 2} = 26.23\ cm^3$

Plastic section modulus about both axes, $Z_p = \frac{1}{6}\left(101.6^3 - 94.4^3\right) = 34.6\ cm^3$

Constants

$\gamma_{m0} = 1.1$
$\gamma_{m1} = 1.25$

Section Classification

$$\varepsilon = \sqrt{250 / f_y} = 1$$

For circular, hollow tubes subjected to moment, $OD/t = 28.22 < 42\varepsilon^2 = 42$

Therefore, the classification of the section is “plastic”.

Hence, from Table 10 of IS 800:2007, the buckling class about both axes is “a”.

Check Slenderness Ratio

$k_y = k_z = 1$

The maximum allowable slenderness ratio is 180.

Slenderness ratio about both axes, $k_z L\ /\ r_z = 86.67 < 180$

So the slenderness is within the limit.

Axial Tension Capacity

Check for yielding of gross section per Cl. 6.2:

$$T_{dg} = A_g \times f_y / \gamma_{m0} = 11.08 \times 250 / 1.1 = 251.8 \text{ kN}$$

Check for rupture of critical section per Cl. 6.3:

$$\alpha = 0.8$$

$$T_{dn} = \alpha \times A_{net} \times f_u / \gamma_{m1} = 0.8 \times 11.08 \times 420 / 1.25 = 297.8 \text{ kN}$$

Block shear areas are not specified and therefore not checked.

The critical tension ratio $10 / 251.8 = 0.040$

Check Axial Compression

The non-dimensional slenderness ratio, $\lambda_z = \sqrt{\frac{f_y (k_z L / r_z)^2}{\pi^2 E}} = 0.961$

For buckling class "a", the imperfection factor, $\alpha = 0.21$

$$\phi_z = 0.5\left[1 + \alpha(\lambda_z - 0.2) + \lambda_z^2\right] = 1.042$$

$$\chi_z = \frac{1}{\phi_z + \sqrt{\phi_z^2 - \lambda_z^2}} = 0.692$$

So, as per Cl. 7.1.2.1 of IS 800:2007, $f_{cdz} = min\left(\frac{\chi_z f_y}{\gamma_{m0}}, \frac{f_y}{\gamma_{m0}}\right) = 157.3\ MPa$

Therefore, axial compression capacity for both axes: $P_{dz} = A_x f_{cdz} = 147.3\ kN$

The critical ratio for axial compression is 0 (i.e., no compression).

Check Shear Capacity

As per Cl. 8.4.1.1 of IS 800:2007, the shear area, $A_{vz} = 2 \times A_x / \pi = 7.054\ cm^2$

So, as per Cl. 8.4.1, $V_{pz} = \frac{A_{vz} f_{yw}}{\sqrt{3}} = 101.8\ kN$

Hence, shear capacity along major axis: $V_{dz} = \frac{V_{pz}}{\gamma_{m0}} = 92.56\ kN$

Shear force along both axes = $3\ kN$.

Critical shear ratio along both axes: $3 / 92.56 = 0.032$

Check Bending Capacity

Since the section has been classified as "plastic", $\beta_b = 1$.

Actual shear force in both axes, $V = 3\ kN < 0.6V_d = 55.53\ kN$. Hence, the section is subjected to low shear.

$$\frac{\beta_b Z_{py} f_y}{\gamma_{m0}} = 7.861 \text{ kN} \cdot \text{m}$$

$$\frac{1.5 Z_e f_y}{\gamma_{m0}} = 8.925 \text{ kN} \cdot \text{m}$$

So, the governing bending strength with respect to the minor axis is $M_{dy} = 7.861\ kN \cdot m$

Actual bending moment about the minor axis is $M_z = 4.5\ kN \cdot m$.

The critical bending ratio in the minor axis is $4.5\ /\ 7.861 = 0.572$

Check Combined Interaction of Axial Compression and Bending

Section strength calculation as per Cl. 9.3.1:

$$n = N / N_d = 0.004$$

Hence, from Table 17:

$$\alpha_1 = 2$$

$$\alpha_2 = 2$$

$$M_{ndy} = M_{dy} = 9.341 \text{ kN} \cdot \text{m} \qquad \text{(Cl. 9.3.1.2(c))}$$

$$1.04 \cdot M_{dz} \cdot (1 - n^{1.7}) = 8.175 \text{ kN} \cdot \text{m} > M_{dz}$$

$$M_{ndz} = M_{dz} = 7.861 \text{ kN} \cdot \text{m}$$

$$\left(\frac{M_y}{M_{ndy}}\right)^{\alpha_1} + \left(\frac{M_{yz}}{M_{ndz}}\right)^{\alpha_2} = \left(\frac{4.5}{7.861}\right)^2 + \left(\frac{4.5}{7.861}\right)^2 = 0.655$$

Overall member strength in bending and axial compression

$$M_{\text{eff}} = M_z - \frac{\psi F_x Z_{ez}}{A_x} = 4.5 - \frac{1.0 \times 10 \times 26.23}{11.08 \times 10^2} = 4.264 \text{ kN} \cdot \text{m}$$

$$M_{\text{eff}} / M_{dz} = 4.263 / 7.861 = 0.542$$

The loading and section is the same in both axes.

Results

Table 673:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	86.46	86.67	negligible	
Tension capacity, gross yielding (kN)	251.8	252.273	negligible	
Tension capacity, rupture of critical section (kN)	297.8	298.368	negligible	
Axial compression capacity about both axes (kN)	174.3	174.338	negligible	
Shear capacity in both axes (kN)	92.56	92.723	negligible	
Ratio for shear	0.032	0.032	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Bending capacity about both axes (kN·m)	7.861	7.864	negligible	
Ratio for bending about both axes	0.572	0.572	none	
Ratio per Cl. 9.3.1.1	0.655	0.655	none	
Ratio for both axes per Cl. 9.3.2.1	0.542	0.542	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - Pipe - Tension and Bending.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The default value of `ALPHA` is used.
- The default values of `KY` and `KZ` are used.
- The default value of the maximum allowable slenderness ratio, `MAIN`, of 180 is used.
- The default value of Ψ.
- The beam type is a cantilever, specified by `CAN 1`
- The beam is laterally supported, specified by `LAT 1`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jul-20
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of laterally supported pipe section subjected to axial tension and
*biaxial bending, designed using IS 800 2007 (LSD)
****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY INDIAN
1 TABLE ST PIP1016L
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE IS800 LSD
CAN 1 ALL
LAT 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |   Member Number:
1                                                           |
 |  Member Section:  ST   PIP1016L              (INDIAN
SECTIONS)                    |
 |  Status: PASS  Ratio:   0.655  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Sec.
9.3.1.1                                                 |
 |  Critical Design Forces:  (Unit: KN
METE)                                     |
 |      FX:     -10.000E+00 T     FY:       3.000E+00      FZ:       3.000E
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:       4.500E
+00      |

|-----------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                             |
 |  AXX:      11.100E+00          IZZ:     133.000E+00          RZZ:
3.462E+00|
 |  AYY:       7.066E+00          IYY:     133.000E+00          RYY:
3.462E+00|
 |  AZZ:       7.066E+00          IXX:     266.000E+00           CW:
0.000E+00|
 |  ZEZ:      26.181E+00          ZPZ:      34.600E
+00                              |
```

```
 |  ZEY:      26.181E+00          ZPY:      34.600E
+00                             |

|--------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                         |
 |  Actual Length:        3.000E
+00                                                   |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                     |
 |                  KZ:            1.000  KY:
1.000                     |
 |  Actual Ratio:  86.67 Allowable Ratio: 400.00  LOAD:
1                      |

|--------------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|--------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                  -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|--------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                             |
 |  Member Section:  ST   PIP1016L              (INDIAN
SECTIONS)                  |

|--------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                  |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                 |
 |                   NSF:            1.000    ALPHA:
0.800                |
 |  Yielding   :  Design Force:       10.000E+00     LC:
1                      |
 |              Capacity:          252.273E+00     As per:  Sec.
6.2              |
 |  Rupture    :  Design Force:       10.000E+00     LC:
1                      |
 |              Capacity:          298.368E+00     As per:  Sec.
6.3              |

|--------------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                              |
 |  Buckling Class:   Major: a    Minor: a    As per:Cl.
7.1.2.2                     |
```

```
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |                Capacity:          174.338E+00    As per:  Sec.
7.1.2             |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000       |
 |                Capacity:          174.338E+00    As per:  Sec.
7.1.2             |

|-----------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                           |
 |  Major Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:           92.723E+00    As per:  Sec.
8.4               |
 |  Minor Axis:   Design Force:        3.000E+00    LC:     1  Loc:
0.000       |
 |                Capacity:           92.723E+00    As per:  Sec.
8.4               |

|-----------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                   |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major Axis:   Design Force:        4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:            7.864E+00    As per:  Sec.
8.2.1.2           |
 |  Minor Axis:   Design Force:       -4.500E+00    LC:     1  Loc:
0.000       |
 |                Capacity:            7.864E+00    As per:  Sec.
8.2.1.2           |

|-----------------------------------------------------------------------------
-----|
 |  Combined
Interaction:                                                              |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     0.655    As per:  Sec.
9.3.1.1                |
 |                       LC:           1    Loc:
0.000                     |
 |  Overall Member Strength (Bending
+Tension):                                  |
 |  Equation 1(Z):       Ratio:     0.542    As per:  Sec.
9.3.2.1                |
 |                       LC:           1    Loc:
0.000                     |
 |  Equation 2(Y):       Ratio:     0.542    As per:  Sec.
9.3.2.1                |
 |                       LC:           1    Loc:
0.000                     |
```

```
|--------------------------------------------------------------------------------
-----|
 |  Checks                 Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                 0.040       1                        0.000E
+00            |
 |  Compression             0.000       0                        0.000E
+00            |
 |  Shear Major             0.032       1                        0.000E
+00            |
 |  Shear Minor             0.032       1                        0.000E
+00            |
 |  Bend Major              0.572       1                        0.000E
+00            |
 |  Bend Minor              0.572       1                        0.000E
+00            |
 |  Sec. 9.3.1.1            0.655       1                        0.000E
+00            |
 |  Sec. 9.3.2.1 (Z)        0.542       1                        0.000E
+00            |
 |  Sec. 9.3.2.1 (Y)        0.542       1                        0.000E
+00            |

|--------------------------------------------------------------------------------
-----|
```

Related Links

- *D8.A.2 Design Process* (on page 1608)

V. IS 800 2007 - LSD Rod Compression and Bending

Verify the program-calculated design results for a 3 m long cantilever beam, subjected to axial compression and biaxial bending. The beam section used is a 50 mm diameter steel rod and is designed as per IS 800:2007.

Details

The 3 m cantilever section is subject to a 10 kN axial compression load at the free end as well as 1 kN/m uniform loads along the length in both the “major” and “minor” axes. The cantilever is assumed to be laterally supported.

The steel yield stress is $f_y = 250\ MPa$, the specified minimum tensile strength is $F_u = 420\ MPa$, and the modulus of elasticity is $E = 205(10)^3\ MPa$

Validation

<u>Section Properties</u>

Gross area:

$$A_g = \frac{\pi(5\text{ cm})^2}{4} = 19.6\text{ cm}^2$$

Moment of inertia:

$$I_y = I_z = \frac{\pi (5\text{ cm})^4}{64} = 30.7\text{ cm}^4$$

Elastic section modulus:

$$Z_e = \frac{I}{D/2} = 12.28\text{ cm}^3$$

Plastic section modulus:

$$Z_p = \frac{(5\text{ cm})^3}{6} = 20.83\text{ cm}^3$$

Radius of gyration:

$$r_{yy} = r_{zz} = \sqrt{\frac{I}{A_g}} = 1.252\text{ cm}$$

Partial Safety Factors

Partial safety factor for failure in tension by yielding (per Table 5 of IS:800 2007), $\gamma_{m0} = 1.1$.

Partial safety factor for failure at ultimate stress (per Table 5 of IS:800 2007), $\gamma_{m1} = 1.25$.

Forces

Axial force, $F_x = 10\ kN$

Shear forces, $F_y = F_z = 1\ kN/m \times 3\ m = 3\ kN$

Moments, $M_y = M_z = \frac{(1\ kN/m) \times (3\ m)^2}{2} = 4.5\ kN \cdot m$

Section Classification

Although Table 2 of IS800:2007 does not specify the classification for solid rod sections, the width to thickness ratio for a solid rod is 1.0. Hence, it is a Class 1 section (Plastic).

Check Slenderness Ratio

$K_y = K_z = 1$

The allowable slenderness ratio is 180.

$$\frac{k_y L}{r_{yy}} = \frac{k_z L}{r_{zz}} = \frac{(1.0) \times 300\text{ cm}}{1.252\text{ cm}} = 239.7 > 180$$

Section fails in slenderness.

Check Axial Tension

The design strength in tension (or compression) due to yielding (per Section 6),

$$T_{dg} = N_d = \frac{A_g \times F_y}{\gamma_{m0}} = \frac{19.6 \times (10)^{-4}\ m^2 \times 250 \times (10)^3 kN/m^2}{1.1} = 445.5\ kN$$

Assume that $A_{net} = A_g$ and $a = 0.8$, the design strength in rupture of critical section,

$$T_{dn} = a \times A_{net} \frac{F_u}{\gamma_{m1}} = 0.8 \times 19.6 \times (10)^{-4}\ m^2 \times \frac{420 \times (10)^3 kN/m^2}{1.25} = 526.8\ kN$$

Check Axial Compression

Per Section 7 of IS: 800-2007

Euler buckling stress, $f_{cc} = \frac{\pi^2 E}{\left(KL/r\right)^2} = \frac{\pi^2 \times 205(10)^3\ MPa}{(239.7)^2} = 35.21\ MPa$

Non-dimensional effective slenderness ratio, $\lambda = \sqrt{\frac{f_y}{f_{cc}}} = \sqrt{\frac{250\ MPa}{35.21\ MPa}} = 2.665$

The imperfection factor given in Table 7; for buckling class (as per Table 10 for solid sections), $a = 0.49$.

$$\phi = 0.5\left[1 + a(\lambda - 0.2) + \lambda^2\right] = 4.654$$

$$\chi = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}} = 0.118$$

The design compressive stress,

$$f_{cd} = \chi \frac{f_y}{\gamma_{m0}} = 0.118 \frac{250}{1.1} = 26.84 \text{ MPa} < \frac{f_y}{\gamma_{m0}} = \frac{250}{1.1} = 227.3 \text{ MPa}$$

The design compressive strength,

$$P_d = A_g f_{cd} = 19.6 \times (10)^{-4} \text{ m}^2 \times 26.84 \times (10)^3 \text{kN}/\text{m}^2 = 52.6 \text{ kN}$$

Design ratio in compression: $\frac{F_x}{P_d} = \frac{10}{52.6} = 0.19$ (for both local z and y axes)

Check for Shear

Per Cl 8.4 and 8.4.1 of IS: 800-2007

Per Cl. 8.4.1.1, the shear area is taken as the gross area of the solid round section; $A_v = A_g = 19.6\ cm^2$.

The shear force used for design is taken as the resultant SRSS combination of the “major” and “minor” axes:

$$V_u = \sqrt{(F_y)^2 + (F_z)^2} = \sqrt{(3 \text{ kN})^2 + (3 \text{ kN})^2} = 4.243 \text{ kN}$$

The plastic under pure shear is taken as the nominal shear strength,

$$V_p = A_v \frac{f_{yw}}{\sqrt{3}} = 19.6 \times (10)^{-4} \text{ m}^2 \times \frac{250 \times (10)^3}{\sqrt{3}} = 282.9 \text{ kN}$$

$$V_d = \frac{V_p}{\gamma_{m0}} = \frac{282.9 \text{ kN}}{1.1} = 257.2 \text{ kN}$$

Design ratio in shear: $\frac{V_u}{V_d} = \frac{4.654}{257.2} = 0.016$

Check for Bending

Per Section 8 of IS: 800-2007

The bending moment used for design is taken as the resultant SRSS combination of the “major” and “minor” axes:

$$M' = \sqrt{(M_y)^2 + (M_z)^2} = \sqrt{(4.5 \text{ kN}\cdot\text{m})^2 + (4.5 \text{ kN}\cdot\text{m})^2} = 6.364 \text{ kN}\cdot\text{m}$$

This is used as the “major” axis bending moment and the “minor” axis bending moment is taken as zero.

When $V < 0.6V_d$ on a cantilever beam, then the design bending strength is taken as:

$$V_u = 4.343 \text{ kN} < 0.6(257.2 \text{ kN}) = 154.3 \text{ kN}$$

$$M_d = \frac{\beta_b \times Z_p \times F_y}{\gamma_{m0}} < \frac{1.5 \times Z_e \times F_y}{\gamma_{m0}}$$

where

β_b = 1

$$M_d = \min \left| \begin{array}{l} \frac{1.0(20.83 \times 10^{-6} \text{ m}^3)(250 \times 10^3 \text{kN/m}^2)}{1.1} = 4.734 \text{ kN·m} \\ \frac{1.5(12.28 \times 10^{-6} \text{ m}^3)(250 \times 10^3 \text{kN/m}^2)}{1.1} = 4.186 \text{ kN·m} \end{array} \right.$$

Design ratio in bending: $\frac{M'}{M_d} = \frac{6.364}{4.186} = 1.52$

Check for Combined Axial Force

Check the reduced flexural strength under combined axial force and the respective uniaxial moment acting alone (Cl. 9.3.1.2).

The ratio for the compression yielding is $n = \frac{N}{N_d} = 0.022$

$$M_{nd,max} = 1.04 \times M_d(1 - n^{1.7}) = 1.04 \times 4.186 \text{ kN·m}(1 - 0.022^{1.7}) = 4.347 \text{ kN·m}$$

Use $M_d = 4.186$ kN·m.

The combined bending ratio is taken as:

$$\left(\frac{M_y'}{M_{nd,y}}\right)^{a_1} + \left(\frac{M_z'}{M_{nd,z}}\right)^{a_2}$$

where

a_1 = 1

a_2 = 1

$$\left(\frac{0}{4.186}\right)^{1.0} + \left(\frac{6.364}{4.186}\right)^{1.0} = 1.52$$

Check for Combined Axial Force and Bending Moment

Per Cl 9.3.2.2 of IS: 800-2007

$$K_y = K_z = \min \left| \begin{array}{l} 1 + (\lambda - 0.2)n = 1 + (2.665 - 0.2)0.19 = 1.468 \\ 1 + 0.8n = 1 + 0.8 \times 0.19 = 1.152 \end{array} \right.$$

$$K_{LT} = 1.0$$

$$\frac{P}{P_{dy}} + K_y\left(\frac{C_{my} \times M_y}{M_{dy}}\right) + K_{LT}\frac{M_z}{M_{dz}} = 0.19 + 1.152\left(\frac{0.9 \times 4.5}{4.186}\right) + 1.0\frac{4.5}{4.186} = 2.380$$

$$\frac{P}{P_{dy}} + K_y\left(\frac{C_{my}\sqrt{M_y^2 + M_z^2}}{M_{dy}}\right) + 0 = 0.19 + 1.152\left(\frac{0.9\sqrt{4.5^2 + 4.5^2}}{4.186}\right) + 0 = 1.766$$

$$\frac{P}{P_{dz}} + 0.6K_y\left(\frac{C_{my} \times M_y}{M_{dy}}\right) + K_z\left(\frac{C_{mz} \times M_z}{M_{dz}}\right) = 0.19 + 0.6 \times 1.152\left(\frac{0.9 \times 4.5}{4.186}\right) + 1.152\left(\frac{0.9 \times 4.5}{4.186}\right) = 1.973$$

Results

Table 674: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	239.7	239.71	negligible	
Yielding tension capacity (kN)	445.5	445.455	negligible	
Rupture tension capacity (kN)	526.8	526.848	negligible	
Axial compression capacity (kN)	52.6	52.598	negligible	
Critical ratio for axial compression	0.19	0.190	none	
Shear capacity (kN)	257.2	257.183	negligible	
Shear ratio	0.016	0.016	none	
Bending capacity (kN·m)	4.186	4.186	none	
Bending ratio	1.52	1.520	none	
Interaction ratio for Cl. 9.3.1.1	1.52	1.520	none	
Interaction ratio for Cl. 9.3.2.2	1.766	1.766	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 LSD - ROD - Compression and Bending.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The lateral support condition is specified by `LAT 1`
- The cantilever member is specified by `CAN 1`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Jan-2021
ENGINEER NAME TK
END JOB INFORMATION
*****************************************************************************
*This problem has been created to validate the program calculated design
*results of rod section subjected to axial Compression and biaxial bending
```

```
*using IS 800 2007 (LSD)
*****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY DUTCH
1 TABLE ST RD50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE IS800 LSD
LAT 1 ALL
CAN 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|-------------------------------------------------------------------------------
-----|
 |*  Member Number:
1                                                           |
 |  Member Section:  ST   RD50                  (DUTCH
SECTIONS)                   |
 |  Status: FAIL  Ratio:   1.766  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Sec.
9.3.2.2                                                 |
 |  Critical Design Forces:  (Unit: KN
```

```
METE)                                        |
 |      FX:      10.000E+00 C     FY:       3.000E+00      FZ:       3.000E
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:       4.500E
+00      |

|------------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                               |
 |  AXX:      19.600E+00          IZZ:      30.700E+00          RZZ:
1.252E+00|
 |  AYY:      19.600E+00          IYY:      30.700E+00          RYY:
1.252E+00|
 |  AZZ:      19.600E+00          IXX:      61.400E+00           CW:
0.000E+00|
 |  ZEZ:      12.280E+00          ZPZ:      20.833E
+00                             |
 |  ZEY:      12.280E+00          ZPY:      20.833E
+00                             |

|------------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                          |
 |  Actual Length:        3.000E
+00                                                 |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                     |
 |                  KZ:            1.000  KY:
1.000                     |
 |  Actual Ratio: 239.71 Allowable Ratio: 180.00  LOAD:
1                       |

|------------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|------------------------------------------------------------------------------
-----|
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - IS-800-2007-LSD (V2.2)
                     ****************************************************

|------------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                             |
 |  Member Section:  ST   RD50                  (DUTCH
SECTIONS)                    |

|------------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
```

```
+03                  |
 |                     NSF:            1.000    ALPHA:
0.800                |
 |  Yielding   :  Design Force:        0.000E+00     LC:
0                       |
 |                Capacity:          445.455E+00     As per:  Sec.
6.2               |
 |  Rupture    :  Design Force:        0.000E+00     LC:
0                       |
 |                Capacity:          526.848E+00     As per:  Sec.
6.3               |

|-----------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                            |
 |  Buckling Class:   Major: c    Minor: c    As per:Cl.
7.1.2.2                      |
 |  Major Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:           52.598E+00    As per:  Sec.
7.1.2              |
 |  Minor Axis:   Design Force:       10.000E+00    LC:     1  Loc:
0.000        |
 |                Capacity:           52.598E+00    As per:  Sec.
7.1.2              |

|-----------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:
KN  )                                                      |
 |  Major Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:          257.183E+00    As per:  Sec.
8.4                |
 |  Minor Axis:   Design Force:        4.243E+00    LC:     1  Loc:
0.000        |
 |                Capacity:          257.183E+00    As per:  Sec.
8.4                |

|-----------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major Axis:   Design Force:       -6.364E+00    LC:     1  Loc:
0.000        |
 |                Capacity:            4.186E+00    As per:  Sec.
8.2.1.2            |
 |  Minor Axis:   Design Force:        0.000E+00    LC:     0  Loc:
0.000        |
 |                Capacity:            4.186E+00    As per:  Sec.
8.2.1.2            |

|-----------------------------------------------------------------------------
-----|
 |  Combined
```

```
Interaction:                                                           |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Section Strength:    Ratio:     1.520    As per:  Sec.
9.3.1.1                  |
 |                       LC:            1    Loc:
0.000                       |
 |  Overall Member Strength (Bending
+Compression):                                       |
 |  Equation 2:          Ratio:     1.766    As per:  Sec. 9.3.2.2   Ky :
1.1521  |
 |                       LC:            1    Loc:       0.000       Kz :
1.1521  |

|------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension                0.000      0                       0.000E
+00            |
 |  Compression            0.190      1                       0.000E
+00            |
 |  Shear Major            0.000      0                       0.000E
+00            |
 |  Shear Minor            0.016      1                       0.000E
+00            |
 |  Bend Major             1.520      1                       0.000E
+00            |
 |  Bend Minor             0.000      0                       0.000E
+00            |
 |  Sec. 9.3.1.1           1.520      1                       0.000E
+00            |
 |  Sec. 9.3.2.2(ii)       1.766      1                       0.000E
+00            |

|------------------------------------------------------------------------------
-----|
```

Related Links

- *D8.A.2 Design Process* (on page 1608)

V. IS 800 2007 - WSD Rod Compression and Bending

Verify the program-calculated design results for a 3 m long cantilever beam, subjected to axial compression and biaxial bending. The beam section used is a 50 mm diameter steel rod and is designed as per IS 800:2007 working stress design method.

Details

The 3 m cantilever section is subject to a 10 kN axial compression load at the free end as well as 1 kN/m uniform loads along the length in both the “major” and “minor” axes. The cantilever is assumed to be laterally supported.

The steel yield stress is $f_y = 250\ MPa$, the specified minimum tensile strength is $F_u = 420\ MPa$, and the modulus of elasticity is $E = 205(10)^3\ MPa$

Validation

Section Properties

Gross area:

$$A_g = \frac{\pi(5\text{ cm})^2}{4} = 19.6\text{ cm}^2$$

Moment of inertia:

$$I_y = I_z = \frac{\pi(5\text{ cm})^4}{64} = 30.7\text{ cm}^4$$

Elastic section modulus:

$$Z_e = \frac{I}{D/2} = 12.28\text{ cm}^3$$

Plastic section modulus:

$$Z_p = \frac{(5\text{ cm})^3}{6} = 20.83\text{ cm}^3$$

Radius of gyration:

$$r_{yy} = r_{zz} = \sqrt{\frac{I}{A_g}} = 1.252\text{ cm}$$

Partial Safety Factors

Partial safety factor for failure in tension by yielding (per Table 5 of IS:800 2007), $\gamma_{m0} = 1.1$.

Partial safety factor for failure at ultimate stress (per Table 5 of IS:800 2007), $\gamma_{m1} = 1.25$.

Forces

Axial force, $F_x = 10\ kN$

Shear forces, $F_y = F_z = 1\ kN/m \times 3\ m = 3\ kN$

Moments, $M_y = M_z = \frac{(1\ kN/m) \times (3\ m)^2}{2} = 4.5\ kN{\cdot}m$

Section Classification

Although Table 2 of IS800:2007 does not specify the classification for solid rod sections, the width to thickness ratio for a solid rod is 1.0. Hence, it is a Class 1 section (Plastic).

Check Slenderness Ratio

$K_y = K_z = 1$

The allowable slenderness ratio is 180.

$$\frac{k_y L}{r_{yy}} = \frac{k_z L}{r_{zz}} = \frac{(1.0) \times 300\text{ cm}}{1.252\text{ cm}} = 239.7 > 180$$

Section fails in slenderness.

Check Axial Tension

The permissible stress in compression/tension due to yielding (per Section 11.2.1(a)),
$f_{atg} = 0.6 f_y = 0.6 \times 250\ MPa = 150\ MPa$

Assume that $A_{net} = A_g$ and $a = 0.8$, the design strength in rupture of critical section,

$$T_{dn} = a \times A_{net} \frac{F_u}{\gamma_{m1}} = 0.8 \times 19.6 \times (10)^{-4}\ m^2 \times \frac{420 \times (10)^3 kN / m^2}{1.25} = 526.8\ kN$$

The permissible stress governed by rupture of the net section,

$$f_{att} = 0.69 \frac{T_{dn}}{A_g} = 0.69 \frac{526.8\ kN (10)^3}{19.6 \times (10)^{-4}\ m^2} = 185.5\ MPa$$

<u>Check Axial Compression</u>

Per section 11.3 of IS:800 2007.

The actual compression under service load, $f_c = \frac{P_s}{A_e} = \frac{F_x}{A_g} = \frac{10\ kN \times (10)^3}{19.6 \times (10)^{-4}\ m^2} = 5.102\ MPa$

Euler buckling stress, $f_{cc} = \frac{\pi^2 E}{\left(KL / r \right)^2} = \frac{\pi^2 \times 205 (10)^3\ MPa}{(239.7)^2} = 35.21\ MPa$

Non-dimensional effective slenderness ratio, $\lambda = \sqrt{\frac{f_y}{f_{cc}}} = \sqrt{\frac{250\ MPa}{35.21\ MPa}} = 2.665$

The imperfection factor given in Table 7; for buckling class (as per Table 10 for solid sections), $a = 0.49$.

$$\phi = 0.5 \left[1 + a(\lambda - 0.2) + \lambda^2 \right] = 4.654$$

$$\chi = \frac{1}{\phi + \sqrt{\phi^2 - \lambda^2}} = 0.118$$

The design compressive stress,

$$f_{cd} = \chi \frac{f_y}{\gamma_{m0}} = 0.118 \frac{250}{1.1} = 26.84 \text{ MPa} < \frac{f_y}{\gamma_{m0}} = \frac{250}{1.1} = 227.3 \text{ MPa}$$

$$f_{ac} = 0.6 \times f_{cd} = 0.6 \times 26.84 \text{ MPa} = 16.10 \text{ MPa}$$

Design ratio in compression: $\frac{f_c}{f_{ac}} = \frac{5.102}{16.10} = 0.317$ (for both local z and y axes)

<u>Check for Shear</u>

Per Cl 11.42 of IS: 800-2007

Per Cl. 8.4.1.1, the shear area is taken as the gross area of the solid round section; $A_v = A_g = 19.6\ cm^2$.

The shear force used for design is taken as the resultant SRSS combination of the “major” and “minor” axes:

$$V_u = \sqrt{(F_y)^2 + (F_z)^2} = \sqrt{(3 \text{ kN})^2 + (3 \text{ kN})^2} = 4.243 \text{ kN}$$

The plastic under pure shear is taken as the nominal shear strength,

$$\tau_b = \frac{V_s}{A_v} = \times \frac{4.243 \text{ kN}}{19.6 \times (10)^{-4} \text{ m}^2} = 2.165 \text{ MPa}$$

The permissible shear stress at working load, $\tau_{ab} = 0.4 f_y = 0.4 \times 250\ MPa = 100\ MPa$

Design ratio in shear: $\frac{\tau_s}{\tau_{ab}} = \frac{2.165}{100} = 0.022$

<u>Check for Bending</u>

Per Section 11.4 of IS: 800-2007

The bending moment used for design is taken as the resultant SRSS combination of the “major” and “minor” axes:

$$M' = \sqrt{(M_y)^2 + (M_z)^2} = \sqrt{(4.5\ \text{kN}\cdot\text{m})^2 + (4.5\ \text{kN}\cdot\text{m})^2} = 6.364\ \text{kN}\cdot\text{m}$$

This is used as the “major” axis bending moment and the “minor” axis bending moment is taken as zero.

The actual bending stress, $f_b = \frac{M_s}{Z_e} = \frac{6.364 kN\cdot m \times 10^3}{12.28\ cm^3} = 518.2\ MPa$

The permissible bending stress at working load, $f_{ab} = 0.66 f_y = 0.66 \times 250\ MPa = 165\ MPa$

Design ratio in bending: $\frac{f_b}{f_{ab}} = \frac{518.2}{165} = 3.141$

<u>Check for Combined Axial Force and Bending</u>

As per Cl. 11.5.2(a) of IS: 800 2007.

$$C_{mLT} = 0.9$$

$$K_y = K_z = \min \begin{cases} 1 + (\lambda - 0.2)(f_c / f_{ac}) = 1 + (2.665 - 0.2)0.317 = 1.781 \\ 1 + 0.8(f_c / f_{ac}) = 1 + 0.8(0.317) = 1.254 \end{cases}$$

$$K_{LT} = -\frac{0.1(f_c / f_{ac})}{C_{mLT} - 0.25} = -\frac{0.1(0.317)}{0.9 - 0.25} = -0.049$$

$$\frac{f_c}{f_{ac}} + 0.6 K_y C_{my} \frac{f_{bcy}}{f_{abcy}} + K_{LT} \frac{f_{bc}}{f_{abcz}} = 0.317 + 0.6(1.254)(0.9)(0) - 0.049(3.141) = 0.163$$

$$\frac{f_c}{f_{ac}} + 0.6 K_y C_{my} \frac{f_{bcy}}{f_{abcy}} + K_z C_{mz} \frac{f_{bc}}{f_{abcz}} = 0.317 + 0.6(1.254)(0.9)(0) + 1.254(0.9)(3.141) = 3.862$$

As per Cl. 11.5.2(b) of IS: 800 2007.

$$\frac{f_c}{0.6 f_y} + \frac{f_{bcy}}{f_{abcy}} + \frac{f_{bcz}}{f_{abcz}} = \frac{5.102}{0.6(250)} + \frac{0}{165} + 3.141 = 3.175$$

Results

Table 675:

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	239.7	239.71	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable stress in tension yielding on gross section (MPa)	150	150	none	
Allowable stress in tension rupture on critical section (MPa)	185.5	185.472	negligible	
Allowable stress in compression (MPa)	16.10	16.101	negligible	
Critical ratio for axial compression	0.317	0.317	none	
Allowable stress in shear (MPa)	100	100	none	
Shear stress ratio	0.022	0.022	none	
Allowable stress in bending (MPa)	165	165	none	
Bending stress ratio	3.141	3.141	none	
Interaction ratio for Cl. 11.5.2(a)(ii)	3.862	3.860	negligible	
Interaction ratio for Cl. 11.5.2(b)	3.175	3.175	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 800 2007 WSD - ROD - Compression and Bending.STD` is typically installed with the program.

The following D8.A.4 Design Parameters (on page 1617) are used:

- The lateral support condition is specified by `LAT 1`
- The cantilever member is specified by `CAN 1`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Jan-2021
ENGINEER NAME TK
END JOB INFORMATION
*****************************************************************************
*This problem has been created to validate the program calculated design
*results of rod section subjected to axial Compression and biaxial bending
*using IS 800 2007 (WSD)
*****************************************************************************
```

```
INPUT WIDTH 79
SET SHEAR
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY DUTCH
1 TABLE ST RD50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GY -1
1 UNI GZ -1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE IS800 WSD
LAT 1 ALL
CAN 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - IS-800-2007-WSD (V2.1)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |*  Member Number:
1                                                           |
 |  Member Section:  ST   RD50                  (DUTCH
SECTIONS)                |
 |  Status: FAIL  Ratio:   3.860  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: Sec.
11.5.2(a)                                               |
 |  Critical Design Forces:  (Unit: KN
METE)                                |
 |      FX:      10.000E+00 C     FY:       3.000E+00      FZ:       3.000E
```

```
+00      |
 |      MX:       0.000E+00       MY:      -4.500E+00      MZ:       4.500E
+00      |

|-----------------------------------------------------------------------------
-----|
 |  Section Properties:    (Unit:
CM  )                                           |
 |  AXX:      19.600E+00          IZZ:      30.700E+00          RZZ:
1.252E+00|
 |  AYY:      19.600E+00          IYY:      30.700E+00          RYY:
1.252E+00|
 |  AZZ:      19.600E+00          IXX:      61.400E+00           CW:
0.000E+00|
 |  ZEZ:      12.280E+00          ZPZ:      20.833E
+00                               |
 |  ZEY:      12.280E+00          ZPY:      20.833E
+00                               |

|-----------------------------------------------------------------------------
-----|
 |  Slenderness Check:    (Unit:  KN
METE)                                       |
 |  Actual Length:        3.000E
+00                                                 |
 |  Parameters:     LZ:        3.000E+00  LY:        3.000E
+00                       |
 |                  KZ:           1.000  KY:
1.000                       |
 |  Actual Ratio: 239.71 Allowable Ratio: 180.00  LOAD:
1                       |

|-----------------------------------------------------------------------------
-----|
 |  Section Class:      Plastic; Flange Class:      Plastic; Web Class:
Plastic|

|-----------------------------------------------------------------------------
-----|
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - IS-800-2007-WSD (V2.1)
                     ****************************************************

|-----------------------------------------------------------------------------
-----|
 |  Member Number:
1                                                           |
 |  Member Section:  ST   RD50                 (DUTCH
SECTIONS)                    |

|-----------------------------------------------------------------------------
-----|
 |  Tension:    (Unit:  KN
METE)                                                  |
 |  Parameters:     FYLD:      250.000E+03       FU:      420.000E
+03                  |
 |                   NSF:            1.000    ALPHA:
```

```
0.800               |
 |  Yielding   :  Actual Stress:           0.000E+00    LC:
0               |
 |                Allowable Stress:      150.000E+03    As per:  Sec.
11.2.1(a)     |
 |  Rupture    :  Actual Stress:           0.000E+00    LC:
0               |
 |                Allowable Stress:      185.472E+03    As per:  Sec.
11.2.1(b)     |

|-----------------------------------------------------------------------------
-----|
 |  Compression:    (Unit:  KN
METE)                                              |
 |  Buckling Class:   Major: c    Minor: c    As per:Cl.
7.1.2.2                     |
 |  Major Axis:   Actual Stress:           5.102E+03    LC:     1  Loc:
0.000   |
 |                Allowable Stress:       16.101E+03    As per:  Sec.
11.3.1        |
 |  Minor Axis:   Actual Stress:           5.102E+03    LC:     1  Loc:
0.000   |
 |                Allowable Stress:       16.101E+03    As per:  Sec.
11.3.1        |

|-----------------------------------------------------------------------------
-----|
 |  Shear:    (Unit:  KN
METE)                                                    |
 |  Major Axis:   Actual Stress:           0.000E+00    LC:     0  Loc:
0.000   |
 |                Allowable Stress:      100.000E+03    As per:  Sec.
11.4.2        |
 |  Minor Axis:   Actual Stress:           2.165E+03    LC:     1  Loc:
0.000   |
 |                Allowable Stress:      100.000E+03    As per:  Sec.
11.4.2        |

|-----------------------------------------------------------------------------
-----|
 |  Bending:    (Unit:  KN
METE)                                                  |
 |  Parameters:   Laterally Supported      KX:   1.00  LX:   3.000E+00
Cantilever  |
 |  Major
Axis:                                                                 |
 |  Actual Stress:  fbc:    518.238E+03  fbt:    518.238E+03  LC:     1
Loc:     0.|00
 |  Allowable Stress:  Fbc/Fbt:    165.000E+03    As per:  Sec.
11.4.1(a)           |
 |  Minor
Axis:                                                                 |
 |  Actual Stress:  fbc:      0.000E+00  fbt:      0.000E+00  LC:     0
Loc:     0.|00
 |  Allowable Stress:  Fbc/Fbt:    165.000E+03    As per:  Sec.
11.4.1(a)           |

|-----------------------------------------------------------------------------
```

```
-----|
 |  Combined
Interaction:                                                         |
 |  Parameters:  PSI: 1.00   CMX:    0.900   CMY:    0.900   CMZ:
0.900          |
 |  Member Stability (Bending
+Compression):                                         |
 |  Equation 2:        Ratio:     3.860    As per:  Sec. 11.5.2(a)   Ky :
1.2535  |
 |                     LC:            1    Loc:       0.000          Kz :
1.2535  |
 |  Member Strength (Bending
+Compression):                                          |
 |                     Ratio:     3.175    As per:Sec.
11.5.2(b)                   |
 |                     LC:            1    Loc:
0.000                        |

|------------------------------------------------------------------------------
-----|
 |  Checks               Ratio   Load Case No.      Location from
Start( METE)      |

|
     |
 |  Tension              0.000      0                       0.000E
+00          |
 |  Compression          0.317      1                       0.000E
+00          |
 |  Shear Major          0.000      0                       0.000E
+00          |
 |  Shear Minor          0.022      1                       0.000E
+00          |
 |  Bend Ten Major       3.141      1                       0.000E
+00          |
 |  Bend Comp Major      3.141      1                       0.000E
+00          |
 |  Bend Ten Minor       0.000      0                       0.000E
+00          |
 |  Bend Comp Minor      0.000      0                       0.000E
+00          |
 |              (i)      0.000      0                       0.000E
+00          |
 |  Sec. 11.5.2(a)(ii)   3.860      1                       0.000E
+00          |
 |  Sec. 11.5.2(b)       3.175      1                       0.000E
+00          |

|------------------------------------------------------------------------------
-----|
```

Related Links

- *D8.A.2 Design Process* (on page 1608)

V. IS 801-Beam with axial and major axis bending

Verification example for a cold-formed beam subject to axial compression and bending moment according to IS: 801-1975.

Details

Verifies the calculations for an IS 60CU40x4 (Channel without lips) beam that is 2 m long and subject to axial compression and major axis bending moment. This example checks for compression, shear, bending and compression and bending interaction as per IS 801.

Material properties:

$E = 2{,}074{,}000\ kgf/cm^2 = 203{,}400\ MPa$
$F_{yi} = 350.0\ MPa = 3{,}569\ kgf/cm^2$
$F_u = 450\ MPa$
$G = 795{,}000\ kgf/cm^2 = 77{,}963\ MPa$

Design forces:

$P = 0.5\ kN$
$M_z = 0.748\ kN{\cdot}m$
$V_y = 1.874\ kN$

Verification

Section Dimension Checks

Check for flat width ratio:

$$w = b - (r + t) = r - (0.4 + 0.6) = 3\text{ mm}$$

$$w/t = 3 / 0.4 = 7.5 < 60$$

Hence, OK (ref. cl 5.2.3(a))

$$h_w/t = (d - 2\times t_f - 2\times \text{ root radius}) / t = (6 - 2\times0.4 - 2\times0.6)/0.4 = 10 < 500$$

Hence, OK (ref. cl 5.2.3(b))

Check for web height to thickness ratio:

$$h/t = (6 - 2\times0.4) / 0.4 = 13 < 150$$

Hence, OK (ref. cl 5.2.4(a))

Check for limiting slenderness

$$k\frac{L_y}{r_y} = 1.0\frac{200\ mm}{1.263\ mm} = 158.3\ < 200$$

Hence, OK (ref. cl 6.3.3)

Calculation of Allowable Compressive Stress

For calculation of axially loaded member, " Q " is an important factor. The definition and method of calculation for value of Q is provided in Clause no. 6.6.1.1 (a). Channel section without lips is a combination of stiffened & unstiffened elements.

As per clause no 6.1.1.1 of IS801, the increase of steel strength happens due to cold work of forming.

Total corner area, $A_{corner} = 2(45.2\ mm^2) = 91.2\ mm^2 = 0.912\ cm^2$

Total area of flanges, $A_{flange} = 2 \times b \times t = 2(0.4)(0.4) = 3.2\ cm^2$

$$C = A_{corner} / A_{flange} = 0.931 / 3.2 = 0.285$$

Effective depth, $h_e = h = 4\ cm$

Therefore, effective area, $A_e = A = 4.91\ cm^2$

$$B_e = 3.69\frac{F_u}{F_y} - 0.819\left(\frac{F_u}{F_y}\right)^2 - 1.79 = 1.6004$$

$$m = 0.192\frac{F_u}{F_y} - 0.068 = 0.1789$$

Tensile yield point of corner, $F_{yc} = \dfrac{B_e \times F_y}{\left(\frac{r}{t}\right)^m} = 521.0\ MPa$

Tensile yield point of flat portions, $F_{yt} = F_y = 350.0\ MPa$

Average yield point of cold-forming for tension/compression members, $F_{ya(compression)} = (C \times F_{yc}) + (1 - C) \times F_{yc} = 381.8\ MPa\ (= 3{,}893\ kgf/cm^2)$

Average yield point of cold-forming for flexural members, $F_{ya(bending)} = (C \times F_{yc}) + (1 - C) \times F_{yc} = 398.8\ MPa$

$$w = 3\text{ cm}$$

$$w / t = 7.5$$

As per cl. 6.2 of IS 801, compressive stress:

$$F_c = 0.6 \times F_{ya} = 0.6 \times 381.8 = 229.1\text{ MPa}$$

$$Q_s = \frac{F_c}{0.6F_{ya}} = \frac{229.1}{229.1} = 1$$

$$h = d - 2(r + t) = 4\text{ cm}$$

$$\frac{h}{t} = 13 < \frac{1435}{\sqrt{Fc}} = 29.69$$

$$Q_a = \frac{A_e}{A} = 1$$

$$Q = Q_s \times Q_a = 1$$

Allowable compression stress, F_{a1} , for members braced against twisting (ref. cl 6.6.1.1)

$$C_e = \sqrt{\frac{2\pi E}{F_{ya}}} = 102.5$$

$$\frac{C_e}{\sqrt{Q}} = 102.5$$

Slenderness ratio $KL/r = 158.3$

$$F_{a1} = \frac{12}{23}Q \times F_{ya} - \frac{3(Q \times F_{ya})}{23\pi^2 E}\left(\frac{KL}{r}\right)^2 = 41.06\ MPa$$

Maximum allowable compressive stress (Fa2) for flexural-torsional buckling (ref. cl 6.6.1.2 of IS 801)

$$r_0 = \sqrt{(r_x{}^2) + (r_y{}^2) + (x_0{}^2)} = 3.732\ cm$$

$$\sigma_x = \pi^2 \times \frac{E}{\left(K_x \frac{L_x}{r_x}\right)^2} = 274.8\ MPa$$

$$\beta = 1 - \left(\frac{x_0}{r_x}\right)^2 = 0.5071$$

$$\sigma_t = \frac{1}{A \times r_o{}^2} \times \frac{(G \times J) + (\pi^2 \times E \times C_w)}{(K_x \times L_x)^2} = 324.2\ MPa$$

$$\sigma_{TF0} = \frac{1}{2\beta}\left[(\sigma_{ex} + \sigma_t) - \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4\beta \times \sigma_{ex} \times \sigma_t}\right] = 174.5 \text{ MPa}$$

$$F_{qy} = F_y \times Q = 381.8 \text{ MPa}$$

$$F_{a2} = 0.522 \times \sigma_{TF0} = 91.09 \text{ MPa}$$

The allowable compressive stress, F_a is the minimum of F_{a1} and F_{a2} :

$$F_a = 41.06 \text{ MPa}$$

Calculation of Allowable Bending Stress

As per clause number 6.1, maximum allowable stress on extreme fiber is:

$$F = 0.6 \times F_{ya(compression)} = 0.6 \times 381.8 \text{ MPa} = 229.1 \text{ MPa}$$

As the section is channel without lips, the flanges are unstiffened. So, as per clause 6.2 we need to check allowable compressive stress on the unstiffened element.

$$\frac{w}{t} = 7.5 < \frac{530}{\sqrt{F_{ya}}} = \frac{530}{\sqrt{3{,}893\ kgf / cm^2}} = 8.494$$

Also, the yield strength of steel, $F_y > 2{,}230\ kgf/cm^2$ (= 227.5 MPa).

$$F_c = 0.6 \times F_{ya(compression)} = 229.1 \text{ MPa}$$

For the major axis bending, the allowable compressive stress, F_{bc} , is the minimum of F and F_c

$$F_{bc} = 229.1 \text{ MPa}$$

Similarly, for major axis bending, the allowable tensile stress, $F_{bt} = 0.6 \times F_{ya(compression)} = 229.1\ MPa$

Calculate the allowable bending stress for laterally unbraced beams:

Allowable bending stress for laterally unbraced beams has been calculated as per clause 6.3 (a).

Unsupported length, $L = 2\ m$ (the `UNL` parameter can be used for this).

$$S_{xc} = Z_{xx} = 8.93 \text{ cm}^3$$

$C_b = 1.0$ for a member under compression and bending.

$$\frac{L^2 \times S_{xc}}{d \times I_{yc}} = 14{,}140 > 1.8\pi^2 E \frac{C_b}{F_{ya(compression)}} = 9{,}464$$

$$F_b = 0.6\pi^2 E \times C_b \frac{d \times I_{yc}}{L^2 \times S_{xc}} = 85.18\ MPa$$

Allowable bending stress in the web:

$$F_{bw1} = \frac{36{,}560{,}000}{\left(h/t\right)^2} = 216{,}330 \frac{kgf}{cm^2} = 21{,}215\ MPa$$

Per cl. 6.4.2, F_{bw} is the minimum of F_{bw1} and $0.6 \times F_{ya(bending)} = 239.2\ MPa$

$$F_{bw} = 239.2 \text{ MPa}$$

<u>Calculation of Allowable Shear Stress</u>

Per cl. 6.4:

Clear distance between flanges = $h = d - 2t = 52\ mm$

$$F_{v1} = \frac{1275 \times \sqrt{F_y}}{h/t} = 577.1042\ MPa \qquad \text{(cl. 6.4.1(a))}$$

$$F_{v2} = \frac{585000}{\left(h/t\right)^2} = 3{,}395\ MPa \qquad \text{(cl. 6.4.1(b))}$$

$$\frac{h}{t} = 13 < \frac{4{,}590}{\sqrt{F_y}} = 76.8$$

Allowable shear stress, F_v is the minimum of F_{v1} or $0.4 \times F_y = 140.1\ MPa$

$$F_v = 140.1 \text{ MPa}$$

Allowable combined bending and shear stress:

As $h/t < 4590/\sqrt{F_y}$, $F_{vc} = F_{vc1} = 577.1\ MPa$

<u>Actual Stresses</u>

Compression

$$f_a = P/A = 0.5 \text{ kN} / 4.91 \text{ mm}^2 = 1.018 \text{ MPa}$$

Bending

$$f_b = M / Z_{xx} = 0.748 \text{ kN·m} / 8.93 \text{ cm}^3 = 83.76 \text{ MPa}$$

Bending in Web

Actual bending stress in the web is calculated by interpolation of bending stress diagram:

$$f_{bw} = f_b \times \left(1 - t/_{0.5 \times d}\right) = 72.59\ MPa$$

Shear

$$f_v = \frac{V_y}{N_w t(d - 2t)} = \frac{1.874(10)^3}{(1)4(60 - 2 \times 4)} = 9.01\ MPa$$

<u>Stress Ratio</u>

Compression

$$f_a / F_a = 1.018 / 41.06 = 0.024$$

Bending

for bending compression: f_b / F_{bc} = *83.76 / 229.1 = 0.366*
for bending tension: f_b / F_{bt} = *83.76 / 229.1 = 0.366*
for unbraced bending: f_b / F_b = *83.76 / 85.18 = 0.983*
for web bending: f_{bw} / F_{bw} = *72.59 / 239.2 = 0.303*

Shear

$$f_v / F_v = 9.01 / 140.1 = 0.064$$

Combined bending and shear (ref. cl 6.4.3 of IS 801):

$$\sqrt{\left(\frac{f_{bw}}{F_{bw1}}\right)^2 + \left(\frac{f_{vy}}{F_{vy1}}\right)^2} = \sqrt{\left(\frac{75.59}{21,215}\right)^2 + \left(\frac{9.01}{577.1}\right)^2} = 0.016$$

Interaction ratio for axial and bending

As $Q = 1.0$, F_{a0} can be calculated using cl. 6.6.1.1(b) with $L = 0$:

$$F_{a0} = \frac{12}{23} Q \times F_{ya} - \frac{3(Q \times F_{ya})^2}{23\pi^2 E}\left(\frac{K^*L}{r}\right)^2 = 182.6\ MPa$$

$$\frac{f_a}{F_{a0}} + \frac{f_{b1}}{F_{b1}} = \frac{1.018}{182.6} + \frac{83.76}{229.1} = 0.371 \quad \text{(6.7.2(a) - 2nd eq)}$$

Results

Table 676: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Compression stress ratio	0.024	0.024	none	
Bending Z (compressive) stress ratio	0.366	0.365	negligible	
Bending Z (tensile) stress ratio	0.366	0.365	negligible	
Bending unbraced	0.983	0.983	none	
Bending at web/ flange junction stress ratio	0.303	0.303	none	
Shear Y stress ratio	0.064	0.064	none	
Compression + Bending interaction	0.371	0.371	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Bending + Shear interaction	0.016	0.016	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801-Beam with axial and major axis bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED INDIAN
1 TABLE ST 60CU40X4
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -1.5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
1 CON GX -1
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
LOAD LIST 4
PARAMETER 1
CODE IS801
CWY 1 ALL
FU 450000 ALL
FYLD 350000 ALL
RATIO 1 ALL
TRACK 2 ALL
CHECK CODE ALL
```

```
STEEL TAKE OFF ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - ( IS:801 )   v3.0
                        ***********************
   ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1  SECTION: 60CU40X4             LEN:    2.000   LOC:
0.000   |
| STATUS: PASS    RATIO:   0.983               REF: 6.3 LTB     LC:      4
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Fx:(C)        0.500       Fy:         1.874          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.748
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:                                           (Unit:   CM)
|
|  Ag:         4.91000       Az:     3.20000            Ay:     2.08000
|
|  Cz:         1.38000       Cy:     3.00000            Z0:     2.62000
|
|  Iz:        26.80000       Iy:     7.84000             J:     0.25500
|
|  Sz:         8.93000       Sy:     2.99000
|
|  Rz:         2.33629       Ry:     1.26362            Cw:    45.60001
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:                                                 (Unit: MPa)
|
|  Fy:   350.025        Fu:   450.032      E: 203404.356       G:  77968.401
|
|  Fya(compression):   381.800             Fya(bending):   398.781
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:     2.000      Lz:     2.000 Ly:     2.000 Lb:     2.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000     Ky:   1.000    NSF:  1.000   Cb:   0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
```

```
|
|  Actual:     158.275      Allowable:     200.000      Ratio:   0.791
|
|-----------------------------------------------------------------------------
|
| CHECKS:                              |      Stresses       |
|
|              | Loc. | Demand |  L/C |  Actual  |  Allow   |Ratio | Ref CL
|
|              |(MET) |(KN-MET)|      | (MPa)    | (MPa)    |      |
|
|--------------|------|--------|------|----------|----------|------|----------
|
| Tension      | 1.167|   -0.50|     4|    1.018 |  229.080 | 0.004| 6.1
|
| Compression  | 0.000|    0.50|     4|    1.018 |   41.835 | 0.024| 6.6.1.1
|
| BendZComp    | 0.000|    0.75|     4|   83.688 |  229.080 | 0.365| 6.3
|
| BendZTens    | 0.000|    0.75|     4|   83.688 |  229.080 | 0.365| 6.3
|
| BendUnbraced | 0.000|    0.75|     4|   83.688 |   85.160 | 0.983| 6.3 LTB
|
| BendYComp    |  -   |   -    |  -   |    -     | 239.268 |  -   | 6.3
|
| BendYTens    |  -   |   -    |  -   |    -     | 229.080 |  -   | 6.3
|
| Bend Web     | 0.000|    0.75|     4|   72.529 |  239.268 | 0.303| 6.4.2
|
| Shear Z      |  -   |   -    |  -   |    -     | 140.010 |  -   | 6.4.1
|
| Shear Y      | 0.000|    1.87|     4|    9.009 |  140.010 | 0.064| 6.4.1
|
| Axial+Bend   | 0.000|   -    |     4|    -     |     -    | 0.371|
6.7.2(a)2|
| Bend+Shear   | 0.000|   -    |     4|    -     |     -    | 0.016| 6.4.3
|
|-----------------------------------------------------------------------------
|
| Effective Section Properties:(cm)
|
|  Ae:   4.910 SzTop:   8.933 SzBot:   8.933 SyLeft:   5.681 SyRight:   2.992
|
| Intermediate Results:  Cb =  1.000
|
|-----------------------------------------------------------------------------
|
 NOTE: Torsion and deflections have not been considered in the design.
      STAAD SPACE                                                -- PAGE NO.
5
```

Related Links

- *D8.C.2 Design Procedure* (on page 1640)

V. IS 801-Column with axial and major axis bending

Verification example for a cold-formed column subject to axial compression and bending moment according to IS:801-1975.

Details

Verifies the calculations for an IS 60CS40x4 (channel with lips) column that is 2 m long and subject to axial compression and major axis bending moment. This example checks for compression, shear, bending, and compression and bending interaction as per IS 801.

Material properties:

$E = 2{,}074{,}000\ kgf/cm^2 = 203{,}400\ MPa$
$F_y = 3{,}569\ kgf/cm^2 = 353.0\ MPa$
$F_u = 4{,}588\ kgf/cm^2 = 450\ MPa$
$G = 795{,}000\ kgf/cm^2 = 77{,}963\ MPa$

Design forces:

$P = 10\ kN$
$M_z = 4.0\ kN{\cdot}m$
$V_y = 2\ kN$

Verification

Section Dimension Checks

Flanges (ref. cl 5.2.3):

$$w = 4 - 2\times 0.4 - 2\times 0.6 = 2$$

$w/t = 2\ /\ 0.4 = 5 < 60$, Hence OK

Web (ref. cl 5.2.4):

$W/t = (6 - 2\times 0.4 - 2\times 0.6)\ /\ 0.4 = 10 < 500$ and

$h/t = (6 - 2\times 0.4)\ /\ 0.4 = 13 < 150$, Hence OK

Check for limiting slenderness (ref. cl 6.3.3):

$$K\times L_y/r_y = 1.0\times 200\text{ mm} / 1.453\text{ mm} = 137.6 < 200$$

Hence, OK

Allowable Compression Strength

Maximum allowable compressive stress for flexural buckling Y (ref. cl 6.6.1.1 of IS 801)

Design stress, $F = 0.6\times f_y = 2{,}160\ kgf/cm^2 = 211.8\ MPa$

Effective width of flange, B_{eff}

$$W = 4 - 2\times 0.4 - 2\times 0.6 = 2\text{ cm}$$

$$\frac{w}{t} = 5 < \frac{1{,}435}{\sqrt{F}} = 30.87$$

B_{eff} = w = 2 cm

Effective width of web, B_{eff}

W/t < 30.87

B_{eff} = w = 10 cm

A_{eff} = 5.8 cm^2

Factor, $Q = A_{eff} / A = 1.0$=

$$C_e = \sqrt{\frac{2\pi^2 E}{F_y}} = 106.6$$

Slenderness limit $= \frac{C_e}{\sqrt{Q}} = 106.6 < 137.6$

$$F_{a1} = \frac{10{,}680{,}000}{(kL\ /r)^2} = 564.1\ kgf \Big/ cm^2$$

Maximum allowable compressive stress for flexural torsional buckling (ref. cl 6.6.1.2 of IS 801)

Polar radius of gyration, $r_0 = \sqrt{r_x{}^2 + r_y{}^2 + x_0{}^2} = 5.181\ cm$

$$\beta = 1\ -\ (x_0 / r_0)^2 = 0.259$$

σ_t = 2,036 kgf/cm^2

σ_{ex} = 2,480 kgf/cm^2

$$\sigma_{TF0} = \frac{1}{2\beta}\left[(\sigma_{ex} + \sigma_t)\ -\ \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4\beta \times \sigma_{ex} \times \sigma_t}\right] = 1{,}201\ \text{kgfcm}^2 = 117.8\ \text{MPa}$$

1,201 kgf/cm^2 < 0.5×F = 1,800 kgf/cm^2

$F_{a2} = 0.522 \times \sigma_{TF0}$ = 626.9 kgf/cm^2

The allowable compressive stress, F_a is the minimum of F_{a1} and F_{a1} .

F_a = 546.1 kgf/cm^2 = 55.32 MPa

<u>Allowable Bending Strength</u>

$C = 1.0$ for member under combined axial and bending.

$$F_b = \frac{2}{3}F_y\ -\ \frac{F_y{}^2}{54\pi^2 E\ \cdot\ C_b}\left(\frac{L^2 S_{xc}}{d \times I_{yc}}\right) = 1{,}412\ kgf \Big/ cm^2 = 138.5\ MPa \qquad \text{(ref. cl 6.3)}$$

Maximum allowable compressive stress:

$F_{b1} = 0.6 \times F_y$ = 2,160 kgf/cm^2 = 211.8 MPa

<u>Allowable Shear Strength</u>

$$\frac{h}{t} = 13\ <\ \frac{4{,}590}{\sqrt{F_y}} = 76.5$$

$$F_{v1} = \frac{1{,}275\sqrt{F_y}}{(h\ /t)} = 5{,}884\ kgf \Big/ cm^2$$

$$F_{v2} = 0.4F_y = 1{,}440\ kgf \big/ cm^2$$

The allowable shear stress, F_v is the minimum of F_{v1} and F_{v2} :

F_v = 1,440 kgf/cm^2 = 141.3 MPa

<u>Axial and Bending Interaction</u>

f_a = P / A = 10 kN / 5.8 cm^2 = 1,019.7 kgf / 5.8 cm^2 = 175.8 kgf/cm^2

f_{b1} = M / Z = 4.0 kN·m / 9.4 cm^3 = 40,789 kgf·cm / 9.4 cm^3 = 4,339 kgf/cm^2

C_{mx} = 0.6

F'_e = 1,294 kgf/cm^2

$$\frac{f_a}{F_{a1}} + \frac{f_{b1}C_m}{F_{b1}\left[1 - \frac{f_a}{F'_e}\right]} = \frac{175.8}{564.1} + \frac{4{,}339(0.6)}{2{,}160\left[1 - \frac{175.8}{1{,}294}\right]} = 0.312 + 1.395 = 1.705$$

Hence, not OK

F_{a0} = 1,879 kgf/cm^3

$$\frac{f_a}{F_{a0}} + \frac{f_{b1}}{F_{b1}} = \frac{175.8}{1{,}879} + \frac{4339}{2{,}160} = 0.094 + 2.009 = 2.103$$

Results

Table 677: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	55.32	55.371	negligible	
Allowable bending stress (MPa)	211.8	211.854	negligible	
Allowable unbraced bending stress (MPa)	138.5	138.583	negligible	
Allowable shear stress along Y (MPa)	141.3	141.236	negligible	
Axial and Bending interaction	2.103	2.102	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801-Column with axial and major axis bending.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 2 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED INDIAN
1 TABLE ST 60CS40X4
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 2 FY -10
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE IS801
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - ( IS:801 )   v3.0
                        ***********************
   ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1  SECTION: 60CS40X4              LEN:    2.000   LOC:
0.000   |
| STATUS: FAIL    RATIO:   3.071               REF: 6.3 LTB     LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Fx:(C)       10.000       Fy:         2.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       4.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:                                            (Unit:   CM)
```

```
|
|  Ag:          5.82000       Az:      3.20000            Ay:      3.36000
|
|  Cz:          1.75000       Cy:      3.00000            Z0:      4.46000
|
|  Iz:         28.20000       Iy:     12.30000             J:      0.29600
|
|  Sz:          9.40000       Sy:      5.46000
|
|  Rz:          2.20122       Ry:      1.45375            Cw:    162.00003
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:                                                (Unit: MPa)
|
|  Fy:   353.090        Fu:   449.993      E: 203404.356       G:  77968.401
|
|  Fya(compression):   353.090             Fya(bending):   353.090
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:     2.000      Lz:     2.000 Ly:     2.000 Lb:     2.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|  Actual:     137.575      Allowable:     200.000      Ratio:   0.688
|
|-----------------------------------------------------------------------------
|
| CHECKS:                              |      Stresses       |
|
|             | Loc. | Demand |  L/C |  Actual  |  Allow   |Ratio | Ref CL
|
|             |(MET) |(KN-MET)|      | (MPa)    | (MPa)    |      |
|
|-------------|------|--------|------|----------|----------|------|----------
|
| Tension      |  -   |   -    |  -   |    -     |  211.854 |  -   | 6.1
|
| Compression  | 0.000|   10.00|     1|   17.183 |   55.371 | 0.310| 6.6.1.1
|
| BendZComp    | 0.000|    4.00|     1|  425.562 |  211.854 | 2.009| 6.3
|
| BendZTens    | 0.000|    4.00|     1|  425.562 |  211.854 | 2.009| 6.3
|
| BendUnbraced | 0.000|    4.00|     1|  425.562 |  138.583 | 3.071| 6.3 LTB
|
| BendYComp    |  -   |   -    |  -   |    -     |  211.854 |  -   | 6.3
|
| BendYTens    |  -   |   -    |  -   |    -     |  211.854 |  -   | 6.3
```

```
|
| Bend Web     | 0.000|    4.00|      1|  368.820 |  211.854 | 1.741| 6.4.2
|
| Shear Z      |  -   |   -    |  -   |     -    | 141.236 |  -   | 6.4.1
|
| Shear Y      | 0.000|    2.00|      1|    5.953 |  141.236 | 0.042| 6.4.1
|
| Axial+Bend   | 0.000|   -    |      1|    -     |     -    | 2.102|
6.7.2(a)2|
| Bend+Shear   | 0.000|   -    |      1|    -     |     -    | 0.020| 6.4.3
|
|-----------------------------------------------------------------------------
|
| Effective Section Properties:(cm)
|
|  Ae:    5.820 SzTop:   9.400 SzBot:   9.400 SyLeft:   7.029 SyRight:   5.467
|
| Intermediate Results:  Cb =   1.000
|
|-----------------------------------------------------------------------------
|
 NOTE: Torsion and deflections have not been considered in the design.
      STAAD SPACE                                                -- PAGE NO.
5
```

Related Links

- *D8.C.2 Design Procedure* (on page 1640)

V. IS 801-Pipe subject to axial compression and bending

Verification example for a cold-formed beam subject to axial compression and bending moment according to IS: 801-1975.

Details

The section used is an ST 101.6x2.6CHS (pipe) section. The beam is a 6 m span subject to axial compression and bending moments. Both ends of the span are fixed.

Material properties:

$E = 203{,}400\ MPa$
$F_y = 353.0\ MPa$

Design forces:

$P = \pm 2.5\ kN$
$M_z = 3.88\ kN{\cdot}m$
$V_y = 3.88\ kN$

Section properties:

Outer diameter, $OD = 101.6\ mm$
Thickness, $t = 2.6\ mm$
Area, $A = 8.09\ cm^2$
Moment of inertia about the major axis, $I_{zz} = 99.1\ cm^3$

Section modulus about the major axis, $Z_{xx} = 19.51\ cm^3$

Verification

Compressive Stress

Radius of gyration, $r_x = \sqrt{\frac{I_{zz}}{A}} = \sqrt{\frac{99.1}{8.09}} = 3.50\ cm$

Allowable compression stress:

$$C_e = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 \times 203{,}404}{353.1}} = 106.6$$

$$Q = 1$$

$$\frac{C_e}{\sqrt{Q}} = \frac{106.6}{\sqrt{1.0}} = 106.6 < {KL}/{r} = \frac{1.0(600)}{3.50} = 171.4$$

Therefore, the allowable compressive stress:

$$F_c = \frac{12\pi^2 E}{23\left({KL}/{r}\right)^2} = \frac{12\pi^2 \times 203{,}404}{23(171.4)^2} = 35.65\text{ MPa}$$

Actual stress in compression

$$f_c = \frac{P}{A} = \frac{2.5 \times 10^3}{8.09 \times 10^2} = 3.09\text{ MPa}$$

Stress ratio in compression: $3.09\ /\ 35.65 = 0.087$

Bending Stress

Avg. diameter = 101.6 mm - 2.6 mm = 99.0 mm

$$\frac{D}{t} = \frac{99.0}{2.6} = 38.1 < \frac{232{,}000}{F_y} = \frac{232{,}000}{353.1} = 657.0$$

Per Cl. 6.8, the maximum allowable stress in bending:

$$F = 0.6 \times F_y = 0.6 \times 353.1 = 211.9\text{ MPa}$$

Actual bending stress:

$$f_b = \frac{M}{Z_{xx}} = \frac{3.88 \times 10^3}{19.51} = 198.9\text{ MPa}$$

Stress ratio in bending: $198.9\ /\ 211.9 = 0.939$

Tensile Stress

Allowable tension stress:

$$F = 0.6 \times F_y = 0.6 \times 353.1 = 211.9\text{ MPa}$$

Actual stress in tension

$$f_t = \frac{P}{A} = \frac{2.5 \times 10^3}{8.09 \times 10^2} = 3.09\text{ MPa}$$

Stress ratio in tension: $3.09\ /\ 211.9 = 0.015$

Calculation of Allowable Shear Stress

Allowable shear stress:

$$F_v = 0.4 \times F_y = 0.4 \times 353.1 = 141.2 \text{ MPa}$$

Shear area:

$$A_z = 0.5 \times 2\pi\left(10.16/2\right)0.26 = 4.15 \text{ cm}^2$$

Actual shear stress:

$$f_v = \frac{V_y}{A_z} = \frac{3.88(10)}{4.15} = 9.349 \text{ MPa}$$

Stress ration in shear: $9.35 \ / \ 141.2 = 0.066$

Results

Table 678: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compression stress (MPa)	35.65	35.660	negligible	
Actual compression stress (MPa)	3.09	3.090	none	
Compression stress ratio	0.087	0.087	none	
Allowable bending stress (MPa)	211.9	211.854	negligible	
Actual bending stress (MPa)	198.9	198.857	negligible	
Allowable tensile stress (MPa)	211.9	211.854	negligible	
Actual tensile stress (MPa)	3.09	3.090	none	
Tension stress ratio	0.015	0.015	none	
Allowable shear stress (MPa)	141.2	141.236	negligible	
Actual shear stress (MPa)	9.349	9.349	none	
Shear stress ratio	0.066	0.066	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801-Pipe subject to axial compression and bending.STD` is typically installed with the program.

The following D8.C.4 Design Parameters (on page 1647) are used:

- The effect of cold work of forming strengthening is not considered by specifying the parameter `CWY 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AUSTRALIAN
1 TABLE ST 101.6x2.6CHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1(DL)
MEMBER LOAD
1 UNI GY -0.169
1 CON GX -5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -1.01
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
1 UNI GY 1.893
LOAD 4 LOADTYPE None  TITLE LOAD CASE 4
MEMBER LOAD
1 UNI GY -0.158
LOAD COMB 5 COMBINATION LOAD CASE 5
1 1.0 2 1.0
LOAD COMB 6 COMBINATION LOAD CASE 6
1 0.75 3 0.75
LOAD COMB 7 COMBINATION LOAD CASE 7
1 0.75 2 0.75 4 0.75
PERFORM ANALYSIS
LOAD LIST 5 TO 7
```

```
PARAMETER 1
CODE IS801
TRACK 2 ALL
CWY 0 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.Pro CODE CHECKING - ( IS:801 )   v3.0
                       ***********************
   ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1  SECTION: 101.6X2.6CHS          LEN:    6.000   LOC:
0.000   |
| STATUS: PASS    RATIO:   0.951               REF: 6.7.1b      LC:      6
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Fx:(C)        1.875       Fy:        -3.879          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:      -3.879
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:                                           (Unit:   CM)
|
|  Ag:         8.09000       Az:     4.14956            Ay:     4.14956
|
|  Cz:         5.08000       Cy:     5.08000            Z0:     0.00000
|
|  Iz:        99.10001       Iy:    99.10001             J:   198.20002
|
|  Sz:        19.50788       Sy:    19.50788
|
|  Rz:         3.49996       Ry:     3.49996            Cw:   656.18642
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:                                                 (Unit: MPa)
|
|  Fy:   353.090        Fu:   449.993      E: 203404.356       G:  77968.401
|
|  Fya(compression):   353.090             Fya(bending):   353.090
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:     6.000      Lz:     6.000 Ly:     6.000 Lb:     6.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000
```

```
|
|--------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|  Actual:     171.431      Allowable:     200.000      Ratio:   0.857
|
|--------------------------------------------------------------------------
|
| CHECKS:                            |      Stresses       |
|
|              | Loc. | Demand |  L/C |  Actual  |  Allow   |Ratio | Ref CL
|
|              |(MET) |(KN-MET)|      | (MPa)    | (MPa)    |      |
|
|--------------|------|--------|------|----------|----------|------|----------
|
| Tension      | 3.500|   -2.50|     5|    3.090 |  211.854 | 0.015| 6.1
|
| Compression  | 0.000|    2.50|     5|    3.090 |   35.660 | 0.087| Cl.6.8
|
| BendZComp    | 0.000|    3.88|     6|  198.857 |  211.854 | 0.939| 6.3
|
| BendZTens    | 0.000|    3.88|     6|  198.857 |  211.854 | 0.939| 6.3
|
| BendUnbraced |  -   |   -    |  -   |    -     |    0.000 |  -   |
|
| BendYComp    |  -   |   -    |  -   |    -     |  211.854 |  -   | 6.3
|
| BendYTens    |  -   |   -    |  -   |    -     |  211.854 |  -   | 6.3
|
| Shear Z      |  -   |   -    |  -   |    -     |  141.236 |  -   | 6.4.1
|
| Shear Y      | 0.000|   -3.88|     6|    9.349 |  141.236 | 0.066| 6.4.1
|
| Axial+Bend   | 0.000|   -    |     6|    -     |     -    | 0.951| 6.7.1b
|
| Bend+Shear   | 0.000|   -    |     6|    -     |     -    | 0.066| 6.4.3
|
|--------------------------------------------------------------------------
|
| Effective Section Properties:(cm)
|
|  Ae:   8.090 SzTop:  19.508 SzBot:  19.508 SyLeft:  19.508 SyRight:  19.508
|
| Intermediate Results:  Cb =  0.000
|
|--------------------------------------------------------------------------
|
NOTE: Torsion and deflections have not been considered in the design.
```

V. IS 801-Zee with lips having axial compression and bending

Verification example for a cold-formed beam subject to axial compression and bending moment according to IS: 801-1975.

Details

The section used is a IS 125ZS45x2.55 section. The beam is a 2 m span subject to axial compression and major axis bending moments. The span is a propped cantilever (one end fixed, the other pinned).

Material properties:

$E = 203{,}400\ MPa = 2{,}074{,}000\ kgf/cm^2$
$F_{yi} = 350\ MPa = 3{,}569\ kgf/cm^2$
$G = 77{,}968\ MPa = 795{,}000\ kgf/cm^2$

Design forces:

$P = 25\ kN$
$M_z = 2.71\ kN{\cdot}m$
$V_y = 6.85\ kN$

Section properties:

Depth of section, $d = 125\ mm$
Width of section, $b = 45\ mm$
Thickness, $t = 2.55\ mm$
Length of lips, $c = 20\ mm$
Fillet radius, $r = 3.825\ mm$
Area, $A = 5.94\ cm^2$
Moment of inertia about the major axis, $I_{zz} = 135\ cm^3$
Moment of inertia about the minor axis, $I_{yy} = 27.5\ cm^3$
Section modulus about the major axis, $Z_{xx} = 21.6\ cm^3$
Section modulus about the minor axis, $Z_{yy} = 6.3\ cm^3$
Torsion constant, $J = 0.125\ cm^4$
Warping constant, $C_w = 834\ cm^6$
Number of corners, $N_c = 4$
Area of corner, $A_c = 74\ mm^2$

Verification

Section Dimension Checks

Check flat width ratio per Cl. 5.2.3:

$$w = b - 2 \times (r + t) = 4.5 - 2 \times (0.3825 + 0.255) = 3.225 \text{ cm}$$

$$\frac{w}{t} = \frac{3.225}{0.255} = 12.65 < 60$$

Hence, OK.

Check web height to thickness ratio per Cl. 5.2.4:

$$h = D - 2(t + r) = 125 - 2\times(2.55 + 3.825) = 112.3 \text{ mm}$$

$$\frac{h}{t} = \frac{11.23}{0.255} = 44.02 < 150$$

Hence, OK.

Check slenderness ratio limits per Cl. 6.3.3:

Radius of gyration, about major axis $r_x = \sqrt{\frac{I_{zz}}{A}} = \sqrt{\frac{135}{5.94}} = 4.77\ cm$

$$KL/r_x = \frac{1.0(200)}{4.77} = 41.95$$

Radius of gyration, about minor axis $r_y = \sqrt{\frac{I_{yy}}{A}} = \sqrt{\frac{27.5}{5.94}} = 2.15\ cm$

$$KL/r_y = \frac{1.0(200)}{2.15} = 92.95 < 200$$

Hence, OK.

Compressive Stress Actual stress in compression

$$f_c = \frac{P}{A} = \frac{25 \times 10^3}{5.94 \times 10^2} = 42.1 \text{ MPa} = 429 \text{ kgf/cm}^2$$

Calculate the factor, Q, per Cl. 6.6.1.1(a):

$$F_c = 0.6 \times F_y = 0.6 \times 350 = 210 \text{ MPa} = 2{,}141 \text{ kgf/cm}^2$$

$$\frac{w}{t} = 12.65 < \frac{1{,}435}{\sqrt{F_c}} = \frac{1{,}435}{\sqrt{2{,}141}} = 31.0$$

Therefore, $b_{eff} = w = 3.225\ cm$ and $A_{lost,f} = (w - b_{eff}) \times t = 0\ cm^2$

Compression stress:

$$Q_s = \frac{F_c}{0.6 \times F_y} = 1.0$$

$$\frac{h}{t} = 44.02 > \frac{1{,}435}{\sqrt{F_c}} = 31.0$$

Calculate the effective depth of the section by re-arranging the flange ratio:

$$h_{\text{eff}} = t \times \frac{2{,}120}{\sqrt{f}}\left[1 - \frac{465}{(h/t)\sqrt{f}}\right] = 0.255 \times \frac{2{,}120}{\sqrt{2{,}141}}\left[1 - \frac{465}{(44.02)\sqrt{2{,}141}}\right] = 9.015 \text{ cm} \quad (5.2.1.1)$$

and $A_{lost,d} = (h - h_{eff}) \times t = (11.23 - 9.015) \times 0.255 = 0.565\ cm^2$

Therefore the effective area, $A_{eff} = A - A_{lost,f} - A_{lost,d} = 5.94 - 0 - 0.565 = 5.38\ cm^2$

$$Q_a = \frac{A_{\text{eff}}}{A} = \frac{5.38}{5.94} = 0.905$$

$$Q = Q_s \times Q_a = 0.905$$

Allowable compression stress:

$$C_e = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 \times 203{,}404}{350}} = 107.1$$

$$\frac{C_e}{\sqrt{Q}} = \frac{106.6}{\sqrt{1.0}} = 107.1 < KL/r = \frac{1.0(200)}{2.15} = 92.95$$

Allowable compression stress for members braced against twisting (Ref Cl. 6.6.1.1):

$$F_{a1} = \frac{12}{23}QF_y - \frac{3(QF_y)^2}{23\pi^2 E}\left(\frac{KL}{r}\right)^2 = \frac{12}{23} \times 0.905 \times 3{,}569 - \frac{3(0.905 \times 3{,}569)^2}{23\pi^2(2{,}074{,}000)}(92.95)^2 = 1{,}111 \text{ kgf/cm}^2 = 108.9$$

Maximum allowable compressive stress for flexural torsional buckling (Ref Cl.6.6.1.2). The section is symmetric, so the distance between the geometric and shear center, $x_0 = 0$.

$$r_0 = \sqrt{r_x{}^2 + r_x{}^2 + x_0{}^2} = \sqrt{(4.77)^2 + (2.15)^2 + (0)^2} = 5.23 \text{ cm}$$

$$\sigma_x = \frac{\pi^2 E}{(KL\ / r)_x{}^2} = \frac{\pi^2(203{,}400)}{(41.95)^2} = 274.8 \text{ MPa}$$

$$\beta = 1 - (x_0 / r_0)^2 = 1.0$$

$$\sigma_t = \frac{1}{A \times r_0{}^2} \times \frac{GJ + \pi^2 EC_w}{(KL\ )^2} = \frac{1}{(594) \times (52.3)^2} \times \frac{77{,}968(1{,}260) + \pi^2(203{,}400)(834 \times 10^6)}{(2{,}000)^2} = 318.2 \text{ MPa}$$

$$\sigma_{TF0} = \frac{1}{2\beta}\left[\left(\sigma_{ex} + \sigma_t\right) - \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4\beta\sigma_{ex}\sigma_t}\right]$$

$$= \frac{1}{2}\left[\left(274.8 + 318.2\right) - \sqrt{(274.8 + 318.2)^2 - 4(274.8 \times 318.2)}\right] = 274.8 \text{ MPa}$$

$$> 0.5 \times Q \times F_y = 0.5 \times 0.905 \times 350 = 158.4 \text{ MPa}$$

$$F_{a2} = 0.522 \times Q \times F_y - \frac{(Q \times F_y)^2}{7.67 \times \sigma_{TF0}} = 165.3 - \frac{(0.905 \times 350)^2}{7.67 \times 274.8} = 118.3 \text{ MPa}$$

Therefore, the allowable compressive stress:

$$F_c = \min \left| \begin{array}{l} F_{a1} = 108.9 \text{ MPa} \\ F_{a2} = 118.3 \text{ MPa} \end{array} \right.$$

Stress ratio in compression: *42.1 / 108.9 = 0.387*

<u>Bending Stress</u>

Actual bending stress:

$$f_b = \frac{M}{Z_{xx}} = \frac{2.71 \times 10^3}{21.60} = 125.5 \text{ MPa}$$

Per Cl. 6.8, the maximum allowable stress on the extreme fiber: $F_c = 0.6 \times F_y = 210 \text{ MPa}$

Per Cl. 6.3(b) , the allowable bending stress for laterally unbraced beams:

$$C_b = 1.0$$

$$\frac{L^2 S_{sc}}{d \times I_{yc}} = \frac{(2{,}000)^2(21.6 \times 10^3)}{125(15.14 \times 10^4)} = 4{,}565$$

$$\frac{0.18\pi^2 E \times C_b}{F_y} = \frac{0.18\pi^2(203{,}400)1.0}{350} = 1{,}032$$

$$\frac{0.9\pi^2 E \times C_b}{F_y} = \frac{0.9\pi^2(203{,}400)1.0}{350} = 5{,}162$$

Since $\frac{0.18\pi^2 E \times C_b}{F_y} < \frac{L^2 S_{sc}}{d \times I_{yc}} < \frac{0.9\pi^2 E \times C_b}{F_y}$,

$$F_b = \frac{2}{3}\pi^2 E \times C_b \frac{d \times I_{\mathrm{yc}}}{L^2 S_{\mathrm{xc}}} = 130.1 \text{ MPa}$$

Stress ratio in bending: *125.5 / 130.1 = 0.963*

Shear Stress

Shear area:

$$A_z = 2Bt = 2.295 \text{ cm}^2$$

$$A_y = t\left[\left(D - 2t\right) + 2\left(\mathrm{c} - t\right)\right] = 3.947 \text{ cm}^2$$

Clear distance between flanges, $h = d - 2t = 12.5 - 2\times 0.255 = 11.99\ cm$

$$\frac{h}{t} = \frac{11.99}{0.255} = 47.02 < \frac{4{,}590}{\sqrt{F_y}} = 99.20$$

$$F_v = \min \left| \begin{array}{l} \frac{1{,}275\sqrt{F_y}}{h\ /t} = \frac{1{,}275\sqrt{3{,}600}}{47.02} = 1{,}627 \text{ kgf/cm}^2 = 159.6 \text{ MPa} \\ 0.4F_y = 0.4 \text{ x } 350 = 140 \text{ MPa} \end{array} \right.$$

Actual shear stress:

$$f_v = \frac{V_y}{A_z} = \frac{6.85(10)}{3.947} = 17.36 \text{ MPa}$$

Stress ratio in shear: *17.36 / 140 = 0.124*

Bending in Web Stress

The actual bending stress in the web is calculated by interpolating from the bending stress diagram:

$$f_{\mathrm{bw}} = f_b\left(1 - \frac{t}{0.5d}\right) = 125.5\left(1 - \frac{0.255}{0.5 \times 12.5}\right) = 120.4 \text{ MPa}$$

The allowable bending stress in the web:

$$F_{\mathrm{bw}} = \frac{36{,}560{,}000}{\left(h/t\right)^2} = 16{,}536 \text{ kgf/cm}^2 = 1622 \text{ MPa} > 0.6F_y = 211.9 \text{ MPa} \qquad (6.4.2)$$

Stress ratio in web bending: *120.4 / 211.9 = 0.568*

Combined Bending and Shear Stress in Web

Per Cl. 6.4.3, use $F_{bw} = \frac{36{,}560{,}000}{\left(h/t\right)^2} = 16{,}536\ kgf / cm^2 = 1622\ MPa$ and $F_v = 159.6\ MPa$

Stress ratio in combined web bending and shear: $\sqrt{\left(\frac{f_{bw}}{F_{bw}}\right)^2 + \left(\frac{f_v}{F_{vc}}\right)^2} = \sqrt{\left(\frac{120.3}{1{,}622}\right)^2 + \left(\frac{17.35}{159.6}\right)^2} = 0.132$

Results

Table 679: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compression stress (MPa)	108.9	109.052	negligible	
Actual compression stress (MPa)	42.1	42.090	negligible	
Compression stress ratio	0.387	0.386	negligible	
Allowable bending stress (MPa)	130.1	130.131	negligible	
Actual bending stress (MPa)	125.5	125.339	negligible	
Bending stress ratio	0.963	0.963	none	
Allowable shear stress (MPa)	140	140.01	negligible	
Actual shear stress (MPa)	17.36	17.353	negligible	
Shear stress ratio	0.124	0.124	none	
Web bending stress ratio	0.568	0.572	negligible	
Combined Bending and Shear Stress Ratio	0.132	0.132	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\India\IS 801-Zee with lips having axial compression and bending.STD` is typically installed with the program.

The following D8.C.4 Design Parameters (on page 1647) are used:

- The effect of cold work of forming strengthening is not considered by specifying the parameter `CWY 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Mar-19
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED INDIAN
1 TABLE ST 125ZS45X2.55
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -5.5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
1 CON GX -25 0
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
LOAD LIST 4
PARAMETER 1
CODE IS801
FU 450000 ALL
FYLD 350000 ALL
RATIO 1 ALL
TRACK 2 ALL
CWY 0 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - ( IS:801 )   v3.0
                        ***********************
   ALL UNITS ARE IN - METE  KN   (U.N.O.)
|--------------------------------------------------------------------------------
|
|  MEMBER:     1  SECTION: 125ZS45X2.55          LEN:    2.000   LOC:
0.000   |
| STATUS: PASS    RATIO:   0.963               REF: 6.3 LTB     LC:      4
|
|--------------------------------------------------------------------------------
|
```

```
| DESIGN FORCES:
|
|  Fx:(C)       25.000       Fy:          6.854          Fz:        0.000
|
|  Mx:           0.000       My:          0.000          Mz:        2.707
|
|---------------------------------------------------------------------------
|
| SECTION PROPERTIES:                                          (Unit:   CM)
|
|  Ag:          5.94000       Az:     2.29500            Ay:     3.94740
|
|  Cz:          4.37250       Cy:     6.25000            Z0:     0.00000
|
|  Iz:        135.00002       Iy:    27.50000             J:     0.12600
|
|  Sz:         21.60000       Sy:     6.30000
|
|  Rz:          4.76731       Ry:     2.15166            Cw:   834.00017
|
|---------------------------------------------------------------------------
|
| MATERIAL INFO:                                               (Unit: MPa)
|
|  Fy:   350.025        Fu:   450.032      E: 203404.356       G:  77968.401
|
|  Fya(compression):   350.025             Fya(bending):   350.025
|
|---------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:     2.000      Lz:     2.000 Ly:     2.000 Lb:     2.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000
|
|---------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|  Actual:      92.952      Allowable:     200.000      Ratio:   0.465
|
|---------------------------------------------------------------------------
|
| CHECKS:                            |      Stresses       |
|
|             | Loc. | Demand |  L/C |  Actual  |  Allow   |Ratio | Ref CL
|
|             |(MET) |(KN-MET)|      | (MPa)    | (MPa)    |      |
|
|-------------|------|--------|------|----------|----------|------|----------
|
| Tension     | 2.000|   -0.00|     4|    0.000 |  210.015 | 0.000| 6.1
|
| Compression | 0.000|   25.00|     4|   42.090 |  109.052 | 0.386| 6.6.1.1
|
```

```
| BendZComp    | 0.000|    2.71|     4|  125.339 |  210.015 | 0.597| 6.3
|
| BendZTens    | 0.000|    2.71|     4|  125.339 |  210.015 | 0.597| 6.3
|
| BendUnbraced | 0.000|    2.71|     4|  125.339 |  130.131 | 0.963| 6.3 LTB
|
| BendYComp    |  -   |   -    |  -   |    -     |  210.015 |  -   | 6.3
|
| BendYTens    |  -   |   -    |  -   |    -     |  210.015 |  -   | 6.3
|
| Bend Web     | 0.000|    2.71|     4|  120.225 |  210.015 | 0.572| 6.4.2
|
| Shear Z      |  -   |   -    |  -   |    -     |  140.010 |  -   | 6.4.1
|
| Shear Y      | 0.000|    6.85|     4|   17.363 |  140.010 | 0.124| 6.4.1
|
| Axial+Bend   | 0.000|   -    |     4|    -     |     -    | 0.851|
6.7.2(a)2|
| Bend+Shear   | 0.000|   -    |     4|    -     |     -    | 0.132| 6.4.3
|
|-----------------------------------------------------------------------------
|
| Effective Section Properties:(cm)
|
|  Ae:   5.377 SzTop:  21.600 SzBot:  21.600 SyLeft:   6.275 SyRight:   6.275
|
| Intermediate Results:  Cb =  1.000
|
|-----------------------------------------------------------------------------
|
 NOTE: Torsion and deflections have not been considered in the design.
```

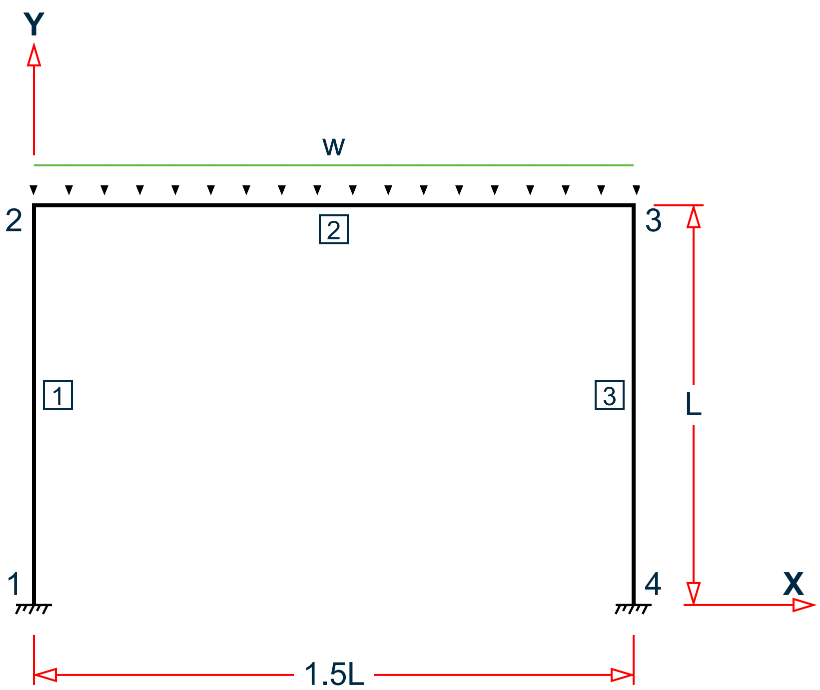
V. Japan

V.AIJ 2002 Check for MBG parameter

Verify the effect of the `MBG` parameter of the AIJ 2002 code on a doubly symmetric I shaped beam of span 6m, that is subjected to UDL of -20 KN/m on its span.

Details

Japanese H Shape Doubly Symmetric I Section H400X200X12X22 used in the following frame, E = 205,000 MPa, F_y = 235 MPa



where w = -20 kN/m and L = 4 m

Equation SSB-1.10 of JSME is used to evaluate Young's modulus, E, for equation 5.8 (`YNG 1`).

d = 400 mm, b_f = 200 mm, t_w = 12 mm, t_f = 22 mm

Verification

Maximum Moment, $MZ1 = 43.714\ kN{\cdot}m$

Minimum Moment, $MZ2 = 43.714\ kN{\cdot}m$

Span Moment = $46.286\ kN{\cdot}m$

Unbraced length of compression flange, $l_b = 6{,}000\ mm$

Note: Only the flanges will be considered (i.e., the web is ignored for calculating the section modulus). (`MBG 1`)

Area of compression flange = $A_f = b_f \times t_f = 4{,}400\ mm^2$

$$C_g = \frac{b_f \times t_f \times \left(d - \frac{t_f}{2}\right) + b_f \times t_f \times \frac{t_f}{2}}{\left(b_f \times t_f\right) + \left(b_f \times t_f\right)} = 200\ mm$$

$$I_{zz} = 2 \times \left(b_f \times \frac{t_f{}^3}{12}\right) + 2 \times b_f \times t_f \times \left(c_g - \frac{t_f}{2}\right)^2 = 3.147 \times (10)^8\ mm^4$$

Elastic Section Modulus about Z axis on compression side $Z_{zc} = I_{zz}/c_g = 1.573{\times}10^6\ mm^3$

$I_{yc} = t_f{\times}b_f^3/12 = 1.467{\times}107\ mm^4$

$A_C = t_f{\times}b_f = 4{,}400\ mm^2$

$r_i = \sqrt{(I_{yc}/A_C)} = 57.735\ mm$

Bending Moment Factor C = 1.75 – 1.05×(-MZ2/MZ1) + 0.3×(-MZ2/MZ1)2 = 3.198, but not more than 2.3

As Span moment > MZ1, C will be 1

Allowable slenderness ratio $\Lambda = \sqrt{\frac{\pi^2 * E}{0.6 * FYLD}} = 119.789$

$$f_{b1} = 1.5 \times \left[1 - 0.4 \times \frac{\left(\frac{l_b}{r_i}\right)^2}{C \times \Lambda^2} \right] \times \frac{f_t}{\Upsilon} = 109.501\ N/mm^2$$

As per Eq. SSB-1.10 for YNG = 1:

$$f_{b2} = 1.5 \times \frac{0.433 \times E}{\frac{l_b \times d}{A_f} \times \Upsilon} = 162.736\ N/mm^2$$

$$f_{bz1} = \max(f_{b1}, f_{b2}) = 162.736\ \text{N/mm}^2$$

Allowable tensile bending stress, $f_{bzt} = f_t = 156.67\ N/mm^2$

Actual tensile bending stress about Z axis = $46.286/Z_{ZC} = 29.43\ N/mm^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.43/156.67 = 0.188$

Allowable compressive bending stress, $f_{bzc} = min(f_{bz1}, f_t) = 156.67\ N/mm^2$

Actual compressive bending stress about Z axis = $46.286/Z_{ZC} = 29.43\ N/mm^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.43/156.67 = 0.188$

Results

Table 680: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Actual bending tensile stress (N/mm^2)	29.43	29.42	negligible	
Allowable bending tensile stress (N/mm^2)	156.67	156.67	none	
Ratio	0.188	0.188	none	
C	1	1	none	
Actual bending compressive stress (N/mm^2)	29.43	29.42	negligible	
Allowable bending compressive stress (N/mm^2)	156.67	156.67	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio	0.188	0.188	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2002 Check for MBG parameter.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 28-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 6 4 0; 4 6 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT METER KN
ANGLE
ANGLE
0.2 0.2 0.018 0.0394179 0.0024 0.0024
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TO 3 TABLE ST H400X200X12X22
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -20
PERFORM ANALYSIS
DEFINE ENVELOPE
1 ENVELOPE 1 TYPE STRENGTH
END DEFINE ENVELOPE
LOAD LIST ENV 1
PARAMETER 1
CODE JAPANESE 2002
TRACK 2 MEMB 2
MAIN 1 MEMB 2
MISES 1 MEMB 2
YNG 1 MEMB 2
```

```
MBG 1 MEMB 2
CB 1 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
      STAAD SPACE                                              -- PAGE NO.
3
                        STAAD.Pro CODE CHECKING - (AIJ-2002)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:       2        Profile:ST  H400X200X12X22   (JAPANESE
SECTIONS)  |
|  Ratio:    0.231  (PASS)  Reference: Eq.5.16          Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    0.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)       15.749       Vy:        60.000          Vz:       0.000
|
|  Tx:           0.000       My:         0.000          Mz:      43.714
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:    0.000 MM             (UNIT: MM)
|
|  Ax:     1.32200E+04       Az: 5.86652E+03            Ay: 4.80000E+03
|
|  Iz:     3.64000E+08       Iy: 2.94000E+07             J: 1.64000E+06
|
|  Zz:     1.82000E+06       Zy: 2.94000E+05            Zx: 7.45455E+04
|
|  iZ:     1.65934E+02       iY: 4.71583E+01            Iw: 1.05020E+12
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   204999.987          G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     6.000      Lz:     6.000 Ly:     6.000  UNL:      6.000
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   1.000
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
```

```
 |
 |--------------------------------------------------------------------------
 |
 |            Compression              |             Bending
 |
 |--------------------------------------------------------------------------
 |
 |  LC#   |     Actual    |   Allowable  |   LC#   |     Actual    |   Allowable
 |
 |--------------------------------------------------------------------------
 |
 |      1 |    27.5000    |     47.9461  |      1  |    27.5000    |     71.7561
 |
 |--------------------------------------------------------------------------
 |
 | CHECKS                            |Env|     Stresses    |
 |
 |              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
 |
 |              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
 |
 |--------------|------|--------|------|---|--------|--------|------|----------
 |
 | Tension      |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
 |
 | Compression  | 0.000|   15.75|     1| P |    1.19|   57.70| 0.021| Eq.5.4
 |
 | Bending Z (T)| 3.000|  -46.29|     1| P |   29.42|  156.67| 0.188| Eq.5.1
 |
 | Bending Z (C)| 3.000|  -46.29|     1| P |   29.42|  156.67| 0.188| Eq.5.1
 |
 | Bending Y (T)|  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
 |
 | Bending Y (C)|  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
 |
 | Shear Z      |  -   |   -    |  -   | - |   -    |  90.45|  -   | Eq.5.2
 |
 | Shear Y      | 0.000|   60.00|     1| P |   12.50|   90.45| 0.138| Eq.5.2
 |
 | Comp+ Bend C | 3.000|   -    |     1| P |   -    |    -   | 0.208| Eq.6.1
 |
 | Comp+ Bend T | 3.000|   -    |     1| P |   -    |    -   | 0.180| Eq.6.2
 |
 | Ten + Bend T | 3.000|   -    |     1| P |   -    |    -   | 0.188| Eq.6.3
 |
 | Ten + Bend C | 3.000|   -    |     1| P |   -    |    -   | 0.188| Eq.6.4
 |
 | Von-Mises    | 0.000|   -    |     1| P |   36.17|  156.67| 0.231| Eq.5.16
 |
 |
 |
 | INTERMEDIATE RESULTS (Bending)
 |
 |  C:  1.000
 |
 |
 |
 | INTERMEDIATE RESULTS (Von-Mises)
```

```
|
|  Sigmax:    28.972 N/mm2   Tou:    12.500 N/mm2  fm:    36.168 N/mm2
|
|--------------------------------------------------------------------------------
|
```

Related Links

- *D9.C.8 Code Checking* (on page 1741)

V.AIJ 2002 Check for MISES parameter

Verify the von Mises stress calculations per the AIJ 2002 code using a cantilever beam.

Details

Japanese Angle section L250X250X35, E = 205,000 MPa, F_y = 200 MPa

The member is a 5 m cantilever with the following applied loads at the end:

Fx = 10 kN
Vz = 5 kN
Vy = 5 kN
T = 5 kN·m

Verification

The maximum of the left-hand side of the von Mises stress equation apparently occurs at the fixed support of the beam. Section forces at the fixed end are calculated as below.

FX = -10 KN
FY = -5 KN
FZ = -5 KN
MX = -5 KN-m
MY = 25 KN-m
MZ = -25 KN-m

The standard AIJ calculation is used (`MISES 1`). From these section forces, σ_x and τ_{xy} at the section of the fixed end are calculated:

$$\sigma_x = \left|\frac{F_x}{A_x}\right| + \left|\frac{M_y}{Z_y}\right| + \left|\frac{M_z}{Z_z}\right|$$

$$= \left|\frac{-10{,}000}{16{,}260}\right| + \left|\frac{25{,}000{,}000}{838{,}660}\right| + \left|\frac{-25{,}000{,}000}{355{,}900}\right|$$

$$=0.615 + 29.809 + 70.244 = 100.668 \text{ N/mm}^2$$

$$\tau_{xy} = \left|\frac{Mx}{Zx}\right| + \sqrt{\left|\frac{Fy}{Ay}\right|^2 + \left|\frac{Fz}{Az}\right|^2}$$

$$= \left|\frac{5{,}000{,}000}{189{,}700}\right| + \sqrt{\left|\frac{5{,}000}{5{,}833}\right|^2 + \left|\frac{5{,}000}{5{,}833}\right|^2}$$

$$= 27.57 \text{ N/mm}^2$$

From σ_x and τ_{xy} , f_m is calculated:

$$f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} = 111.42 \ N / mm^2$$

For permanent loading, $k = 1$ and $f_t = f_y / 1.5 = 133.33 \ N/mm^2$

Therefore, the ratio = $f_m / k \times f_t = 111.419/(1 \times 133.33) = 0.836$

Results

Table 681: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
f_m (N/mm^2)	111.42	111.42	none	
f_t (N/mm^2)	133.33	133.33	none	
σ_x (N/mm^2)	100.668	100.669	negligible	
τ_{xy} (N/mm^2)	27.569	27.57	negligible	
Ratio	0.836	0.836	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2002 Check for MISES parameter.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Apr-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TABLE ST L250X250X35
*1 TABLE ST H100X50X5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
```

```
2 FX 10 FY 5 FZ 5 MX 5
PERFORM ANALYSIS PRINT STATICS LOAD
PRINT MEMBER PROPERTIES ALL
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE JAPANESE 2002
*TMP 1 ALL
*KZ 1.5 ALL
FYLD 200000 ALL
MISES 1 ALL
*MBG -1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2002)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-------------------------------------------------------------------------------
|
|   Member No:       1        Profile:ST  L250X250X35      (JAPANESE
SECTIONS)  |
|  Ratio:    0.836  (PASS)  Reference: Eq.5.16          Loadcase:       1
|
|-------------------------------------------------------------------------------
|
|  Location:    0.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(T)      -10.000       Vy:        -5.000          Vz:      -5.000
|
|  Tx:          -5.000       My:        25.000          Mz:     -25.000
|
|-------------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:    0.000 MM              (UNIT: MM)
|
|  Ax:     1.62600E+04       Az: 5.83333E+03            Ay: 5.83333E+03
|
|  Iz:     3.79328E+07       Iy: 1.48256E+08             J: 6.63950E+06
|
|  Zz:     3.55901E+05       Zy: 8.38661E+05            Zx: 1.89700E+05
|
|  iZ:     4.83000E+01       iY: 9.54873E+01            Iw: 2.99365E+10
|
|-------------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         200.000      E :   205000.000          G :     79000.000
|
|-------------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     5.000      Lz:     5.000 Ly:     5.000  UNL:      5.000
```

```
 |
 | DESIGN PARAMETERS
 |
 |  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000
 |
 |-----------------------------------------------------------------------------
 |
 | CRITICAL SLENDERNESS (Tension)
 |
 |  Actual :    103.520      Allowable :    400.000      Ratio :  0.259
 |
 |-----------------------------------------------------------------------------
 |
 | Section Classification Results :    Width to Thickness Ratios
 |
 |-----------------------------------------------------------------------------
 |
 |           Compression               |             Bending
 |
 |-----------------------------------------------------------------------------
 |
 |  LC#   |    Actual   |   Allowable  |   LC#   |    Actual   |   Allowable
 |
 |-----------------------------------------------------------------------------
 |
 |   -    |      -      |       -      |      1  |     7.1429  |     14.1421
 |
 |-----------------------------------------------------------------------------
 |
 | CHECKS                              |Env|     Stresses    |
 |
 |             | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
 |
 |             |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
 |
 |-------------|------|--------|------|---|--------|--------|------|----------
 |
 | Tension     | 0.000|  -10.00|     1| P |    0.62|  133.33| 0.005| Eq.5.1
 |
 | Compression |  -   |    -   |  -   | - |   -    |   77.53|  -   | Eq.5.3
 |
 | Bending Z (T)| 0.000|  -25.00|     1| P |   70.24|  133.33| 0.527| Eq.5.1
 |
 | Bending Z (C)| 0.000|  -25.00|     1| P |   70.24|  133.33| 0.527| Eq.5.1
 |
 | Bending Y (T)| 0.000|   25.00|     1| P |   29.81|  133.33| 0.224| Eq.5.1
 |
 | Bending Y (C)| 0.000|   25.00|     1| P |   29.81|  133.33| 0.224| Eq.5.1
 |
 | Shear Z     | 0.000|   -5.00|     1| P |    0.86|   76.98| 0.011| Eq.5.2
 |
 | Shear Y     | 0.000|   -5.00|     1| P |    0.86|   76.98| 0.011| Eq.5.2
 |
 | Comp+ Bend C | 0.000|   -    |     1| P |   -    |    -   | 0.750| Eq.6.1
 |
 | Comp+ Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.750| Eq.6.2
 |
 | Ten + Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.755| Eq.6.3
```

```
|
| Ten + Bend C | 0.000|    -    |      1| P |   -    |    -   | 0.746| Eq.6.4
|
| Von-Mises    | 0.000|    -    |      1| P |  111.42|  133.33| 0.836| Eq.5.16
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  C:  0.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:   100.669 N/mm2   Tou:    27.570 N/mm2  fm:   111.420 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

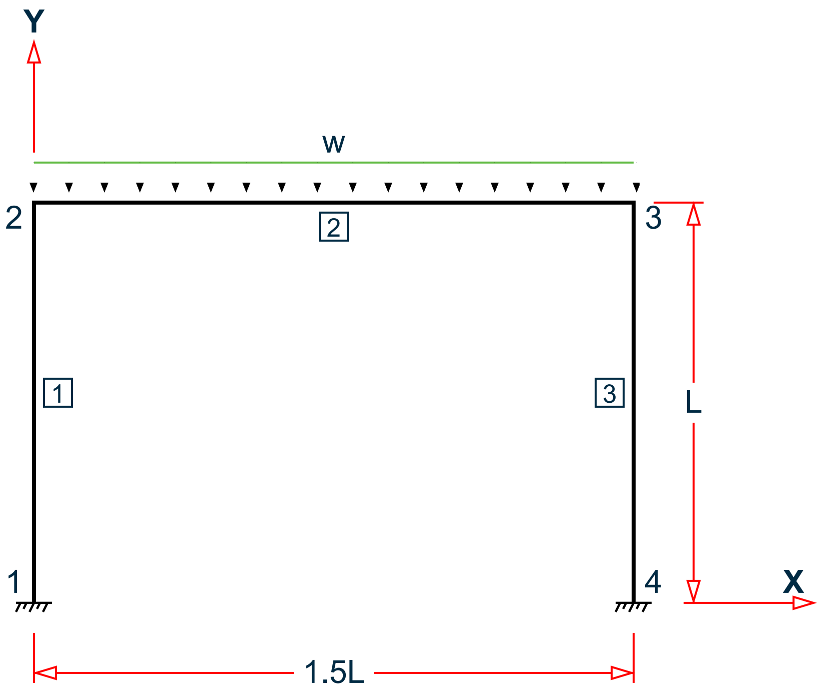
- *D9.C.10 Von Mises Stresses Check* (on page 1742)
- *D9.C.10 Von Mises Stresses Check* (on page 1742)

V.AIJ 2005 Check for MBG parameter

Verify the effect of the `MBG` parameter of the AIJ 2005 code on a doubly symmetric I shaped beam of span 6m, that is subjected to UDL of -20 KN/m on its span.

Details

Japanese H Shape Doubly Symmetric I Section H400X200X12X22 used in the following frame, E = 205,000 MPa, F_y = 235 MPa



where w = -20 kN/m and L = 4 m

Equation SSB-1.10 of JSME is used to evaluate Young's modulus, E, for equation 5.8 (`YNG 1`).

d = 400 mm, b_f = 200 mm, t_w = 12 mm, t_f = 22 mm

Verification

Maximum Moment, $MZ1 = 43.714\ kN{\cdot}m$

Minimum Moment, $MZ2 = 43.714\ kN{\cdot}m$

Span Moment = $46.286\ kN{\cdot}m$

Unbraced length of compression flange, $l_b = 6{,}000\ mm$

Note: Only the flanges will be considered (i.e., the web is ignored for calculating the section modulus). (`MBG 1`)

Area of compression flange = $A_f = b_f \times t_f = 4{,}400\ mm^2$

$$C_g = \frac{b_f \times t_f \times \left(d - \frac{t_f}{2}\right) + b_f \times t_f \times \frac{t_f}{2}}{\left(b_f \times t_f\right) + \left(b_f \times t_f\right)} = 200\ mm$$

$$I_{zz} = 2 \times \left(b_f \times \frac{t_f{}^3}{12}\right) + 2 \times b_f \times t_f \times \left(c_g - \frac{t_f}{2}\right)^2 = 3.147 \times (10)^8\ mm^4$$

Elastic Section Modulus about Z axis on compression side $Z_{zc} = I_{zz}/c_g = 1.573{\times}10^6\ mm^3$

$$I_{yc} = t_f{\times}b_f{}^3/12 = 1.467{\times}107\ mm^4$$

$$A_C = t_f{\times}b_f = 4{,}400\ mm^2$$

$$r_i = \sqrt{(I_{yc}/A_C)} = 57.735\ mm$$

Bending Moment Factor C = 1.75 – 1.05×(-MZ2/MZ1) + 0.3×(-MZ2/MZ1)2 = 3.198, but not more than 2.3

As Span moment > MZ1, C will be 1. (Eq. 5.15)

Elastic critical slenderness ratio ${}_e\lambda_b = 1/\sqrt{0.6} = 1.291$ (Eq. 5.11)

Plastic slenderness ratio = ${}_p\lambda_b = 0.3$ (Eq. 5.14)

$$M_y = f_y \times Z_Z = 427.7 \text{ k}\cdot\text{m}$$

$M_e = 558.21\ kN\cdot m$
(Eq. 5.16)

So, $\lambda_b = \sqrt{(M_y/M_e)} = 0.8753$ (Eq. 5.10)

So, here ${}_p\lambda_b < \lambda_b < {}_e\lambda_b$

So, we calculate Allowable bending stress as per Eq. 5.8

$$v = 3/2 + 2/3\times(\lambda_b/{}_e\lambda_b)^2 = 1.8065$$

$$f_b = \frac{\left[1 - 0.4 \times \frac{\lambda_b - {}_p\lambda_b}{{}_e\lambda_b - {}_p\lambda_b}\right] \times f_y}{v} = 99.88\ N/mm^2$$

Allowable tensile bending stress, $f_{bzt} = f_t = 156.667\ N/mm^2$

Actual tensile bending stress about Z axis = $46.286/ZZC = 29.425\ N/mm^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.425/156.667 = 0.1878$

Allowable compressive bending stress $f_{bzc} = min(f_b, f_t) = 99.88\ N/mm^2$

Actual compressive bending stress about Z axis = $46.286/ZZC = 29.425\ N/mm^2$

Utilization ratio = Actual bending stress/Allowable bending stress = $29.425/99.88 = 0.295$

Results

Table 682: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Actual bending tensile stress (N/mm^2)	29.425	29.42	Negligible	
Allowable bending tensile stress (N/mm^2)	156.667	156.67	Negligible	
Ratio	0.1878	0.188	Negligible	
My (kN·m)	427.7	427.7	None	
Me (kN·m)	558.21	558.21	None	

Result Type	Reference	STAAD.Pro	Difference	Comments
${}_{p}\lambda_{b}$	0.3	0.3	None	
C	1	1	None	
Actual bending compressive stress (N/mm^2)	29.425	29.42	Negligible	
Allowable bending compressive stress (N/mm^2)	99.88	99.88	None	
Ratio	0.295	0.295	None	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 Check for MBG parameter.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 28-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 6 4 0; 4 6 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT METER KN
ANGLE
ANGLE
0.2 0.2 0.018 0.0394179 0.0024 0.0024
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TO 3 TABLE ST H400X200X12X22
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
```

```
MEMBER LOAD
2 UNI GY -20
PERFORM ANALYSIS
DEFINE ENVELOPE
1 ENVELOPE 1 TYPE STRENGTH
END DEFINE ENVELOPE
LOAD LIST ENV 1
PARAMETER 1
CODE JAPANESE 2005
TRACK 2 MEMB 2
MAIN 1 MEMB 2
MISES 1 MEMB 2
MBG 1 MEMB 2
CB 1 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|----------------------------------------------------------------------------
|
|   Member No:       2        Profile:ST  H400X200X12X22   (JAPANESE
SECTIONS)  |
|  Ratio:    0.315  (PASS)  Reference: Eq.6.1           Loadcase:      1
|
|----------------------------------------------------------------------------
|
|  Location:    3.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)       15.749       Vy:         0.000          Vz:       0.000
|
|  Tx:           0.000       My:         0.000          Mz:     -46.286
|
|----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE: 3000.000 MM             (UNIT: MM)
|
|  Ax:     1.32200E+04       Az: 5.86652E+03            Ay: 4.80000E+03
|
|  Iz:     3.64000E+08       Iy: 2.94000E+07             J: 1.64000E+06
|
|  Zz:     1.82000E+06       Zy: 2.94000E+05            Zx: 7.45455E+04
|
|  iZ:     1.65934E+02       iY: 4.71583E+01            Iw: 1.05020E+12
|
|----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   205000.000          G :     79000.000
|
|----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
```

```
|
| Member Length:     6.000      Lz:     6.000 Ly:     6.000  UNL:      6.000
|
| DESIGN PARAMETERS
|
| Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   1.000    pλb:  0.000
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|-----------------------------------------------------------------------------
|
|           Compression                |            Bending
|
|-----------------------------------------------------------------------------
|
| LC#   |    Actual   |   Allowable  |   LC#   |    Actual   |   Allowable
|
|-----------------------------------------------------------------------------
|
|     1 |    27.5000  |     47.2567  |     1   |    27.5000  |     70.8850
|
|-----------------------------------------------------------------------------
|
| CHECKS                             |Env|     Stresses    |
|
|             | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|             |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|-------------|------|--------|------|---|--------|--------|------|----------
|
| Tension      |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Compression  | 0.000|   15.75|     1| P |    1.19|   57.70| 0.021| Eq.5.4
|
| Bending Z (T)| 3.000|  -46.29|     1| P |   29.42|  156.67| 0.188| Eq.5.1
|
| Bending Z (C)| 3.000|  -46.29|     1| P |   29.42|   99.88| 0.295| Eq.5.8
|
| Bending Y (T)|  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Bending Y (C)|  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Shear Z      |  -   |   -    |  -   | - |   -    |   90.45|  -   | Eq.5.2
|
| Shear Y      | 0.000|   60.00|     1| P |   12.50|   90.45| 0.138| Eq.5.2
|
| Comp+ Bend C | 3.000|   -    |     1| P |   -    |    -   | 0.315| Eq.6.1
|
| Comp+ Bend T | 3.000|   -    |     1| P |   -    |    -   | 0.180| Eq.6.2
|
| Ten + Bend T | 3.000|   -    |     1| P |   -    |    -   | 0.188| Eq.6.3
|
| Ten + Bend C | 3.000|   -    |     1| P |   -    |    -   | 0.295| Eq.6.4
|
| Von-Mises    | 0.000|   -    |     1| P |   36.17|  156.67| 0.231| Eq.5.24
```

```
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 5.5821E+02 KN-MET  My: 4.2770E+02 KN-MET   pλb:   0.300  C:  1.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:    28.972 N/mm2   Tou:    12.500 N/mm2  fm:    36.168 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V.AIJ 2005 Check for MISES parameter

Verify the Von Mises stresses per the AIJ 2005 code of a beam in a frame.

Details

A cantilever beam is subject to the following forces at the free end. The cantilever member is a pair of C12X30, back-to-back.

$F_x = 5$ kN

$F_y = -15$ kN

$F_z = 5$ kN

$M_x = 10$ kN·m

$M_y = -6.667$ kN·m

The member is steel with the following material properties:

E = 205,000 MPa

$F_y = 200$ MPa

The section forces at the fixed-end of the left cantilever are taken from the STAAD.Pro analysis:

$F_x = -5$ kN

$F_y = -15$ kN

$F_z = -5$ kN

$M_x = -10$ kN·m

$M_y = 15$ kN·m

$M_z = 25$ kN·m

Validation

From the section forces, σ_x and τ_{xy} at the cross-section at the fixed end are calculated:

$$\sigma_x = \left| \frac{F_x}{A_x} \right| + \left| \frac{M_y}{Z_y} \right| + \left| \frac{M_z}{Z_z} \right| = \left| \frac{\text{-5,000}}{11{,}368} \right| + \left| \frac{1{,}5000{,}000}{425{,}030} \right| + \left| \frac{25{,}000{,}000}{884{,}900} \right|$$

$$= 0.440 + 35.292 + 28.25 = 63.98 \ \ \text{N/mm}^2$$

$$\tau_{xy} = \left| \frac{Mx}{Zx} \right| + \sqrt{\left| \frac{Fy}{Ay} \right|^2 + \left| \frac{Fz}{Az} \right|^2} = \frac{10{,}000{,}000}{51{,}605} + \sqrt{\left(\frac{15{,}000}{7{,}897}\right)^2 + \left(\frac{5{,}000}{2{,}732}\right)^2} = 196.4 \ \ \text{N/mm}^2$$

From these, the Von Mises stress, f_m, is calculated as:

$$f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} = \sqrt{(63.98)^2 + 3(196.5)^2} = 346.17 \ \ \text{N/mm}^2$$

For permanent loading conditions, $k = 1$ and $f_t = F_y/1.5 = 133.3\ N/mm^2$

The ratio = $f_m\ /\ (k \times f_t\ ) = 346.17\ /\ (1 \times 133.3) = 2.596$

Results

Table 683: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
σ_x (N/mm^2)	63.98	63.983	negligible	
τ_{xy} (N/mm^2)	196.4	196.415	negligible	
f_m (N/mm^2)	346.17	346.16	negligible	
f_t (N/mm^2)	133.3	133.3	negligible	
Ratio	2.596	2.596	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 Check for MISES parameter.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 28-Apr-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1.667 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
```

```
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE D C12X30 SP 0.1
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 5 FY -15 FZ 5 MX 10 MY -6.6667
PERFORM ANALYSIS PRINT STATICS LOAD
PRINT ANALYSIS RESULTS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE JAPANESE 2005
MISES 1 MEMB 1
FYLD 200000 ALL
TRACK 2 MEMB 1
CHECK CODE MEMB 1
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:       1        Profile:D   C12X30           (AISC
SECTIONS)      |
|  Ratio:    2.596  (FAIL)  Reference: Eq.5.24           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    0.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(T)       -5.000       Vy:        15.000          Vz:      -5.000
|
|  Tx:         -10.000       My:        15.002          Mz:      25.005
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:    0.000 MM              (UNIT: MM)
|
|  Ax:     1.13677E+04       Az: 2.73233E+03            Ay: 7.89676E+03
|
|  Iz:     1.34859E+08       Iy: 5.54743E+07             J: 6.56703E+05
|
|  Zz:     8.84901E+05       Zy: 4.25031E+05            Zx: 5.16057E+04
|
|  iZ:     1.08919E+02       iY: 6.98569E+01            Iw: 8.08857E+10
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
```

```
||
||  Fy:         200.000      E :   205000.000         G :     79000.000
||
||-----------------------------------------------------------------------------
||
|| DESIGN PROPERTIES
||
||  Member Length:     1.667      Lz:     1.667 Ly:     1.667  UNL:      1.667
||
|| DESIGN PARAMETERS
||
||  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
||
||-----------------------------------------------------------------------------
||
|| CRITICAL SLENDERNESS (Tension)
||
||  Actual :     23.863      Allowable :    400.000      Ratio :  0.060
||
||-----------------------------------------------------------------------------
||
|| Section Classification Results :    Width to Thickness Ratios
||
||-----------------------------------------------------------------------------
||
||           Compression              |             Bending
||
||-----------------------------------------------------------------------------
||
||  LC#   |    Actual    |   Allowable |   LC#   |    Actual    |   Allowable
||
||-----------------------------------------------------------------------------
||
||   -    |      -       |      -      |      1  |    10.7824   |     76.8375
||
||-----------------------------------------------------------------------------
||
|| CHECKS                             |Env|     Stresses     |
||
||              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
||
||              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
||
||--------------|------|--------|------|---|--------|--------|------|----------
||
|| Tension      | 0.000|   -5.00|     1| P |    0.44|  133.33| 0.003| Eq.5.1
||
|| Compression  |  -   |   -    |  -   | - |   -    |  129.59|  -   | Eq.5.3
||
|| Bending Z (T)| 0.000|   25.00|     1| P |   28.26|  133.33| 0.212| Eq.5.1
||
|| Bending Z (C)| 0.000|   25.00|     1| P |   28.26|  131.65| 0.215| Eq.5.7
||
|| Bending Y (T)| 0.000|   15.00|     1| P |   35.30|  133.33| 0.265| Eq.5.1
||
|| Bending Y (C)| 0.000|   15.00|     1| P |   35.30|  133.33| 0.265| Eq.5.1
||
|| Shear Z      | 0.000|   -5.00|     1| P |    1.83|   76.98| 0.024| Eq.5.2
```

```
|
| Shear Y      | 0.000|   15.00|      1| P |    1.90|   76.98| 0.025| Eq.5.2
|
| Comp+ Bend C | 0.000|   -    |      1| P |   -    |    -   | 0.479| Eq.6.1
|
| Comp+ Bend T | 0.000|   -    |      1| P |   -    |    -   | 0.477| Eq.6.2
|
| Ten + Bend T | 0.000|   -    |      1| P |   -    |    -   | 0.480| Eq.6.3
|
| Ten + Bend C | 0.000|   -    |      1| P |   -    |    -   | 0.476| Eq.6.4
|
| Von-Mises    | 0.000|   -    |      1| P |  346.17|  133.33| 2.596| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 3.7016E+03 KN-MET  My: 1.7698E+02 KN-MET   pλb:   0.600  C:  1.750
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:    63.993 N/mm2   Tou:   196.415 N/mm2  fm:   346.167 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.4 Von Mises Stresses Check* (on page 1727)

V.AIJ 2005 Check for MISES parameter 2

A slender, cantilever beam subjected to a load at the end.

Given

A cantilever beam of length 0.3 meters is subjected to a permanent joint load of 3 kN in the Y direction and 2 kN in the Z direction as well as a 0.008 kN·m torque applied at the end. Axial tension of 10 kN is also applied to the member. An H100x50x5 section is used from the Japanese steel tables.

Section properties

$D = 100$ mm, $B = 50$ mm, $t_f = 7$ mm, $t_w = 5$ mm
$I_x = 15{,}000$ mm^4
$A_x = 1185$ mm^2, $A_y = 500$ mm^2, $A_z = 467$ mm^2
$Z_x = I_x/t_{max} = 15{,}000/7 = 2{,}143$ mm^3, $Z_y = 5{,}920$ mm^3, $Z_z = 37{,}400$ mm^3

The maximum of the left hand side of the von Mises stress equation apparently occurs at the fixed end of the beam. Section forces at the fixed end are as follows:

-10.0 kN (Tension)
0.6 kN·m (Bending-Y)
0.9 kN·m (Bending-Z)
-3.0 kN (Shear-Y)

-2.0 kN (Shear-Z)
-0.008 kN·m (Torsion

Material

FYLD = 300 MPa
E = 2.05E+05 MPa
G = E/2.6 MPa

Solution

From these section forces, σ_x and τ_{xy} at the section of the fixed end are calculated as follows:

$$\sigma_x = \left| \frac{F_x}{A_x} \right| + \left| \frac{M_y}{Z_y} \right| + \left| \frac{M_z}{Z_z} \right|$$
$$= \left| \frac{-10,000}{1,185} \right| + \left| \frac{600,000}{5,920} \right| + \left| \frac{-900,000}{37,400} \right| = 8.44 + 101.35 + 24.06 = 133.85 \; N/mm^2$$

$$\tau_{xy} = \left| \frac{M_x}{Z_x} \right| + \sqrt{\left| \frac{F_y}{A_y} \right|^2 + \left| \frac{F_z}{A_z} \right|^2}$$
$$= \left| \frac{-8,000}{2,143} \right| + \sqrt{\left| \frac{-3,000}{500} \right|^2 + \left| \frac{-2,000}{467} \right|^2} = 3.73 + \sqrt{6^2 + 4.28^2} = 11.10 \; N/mm^2$$

From σ_x and τ_{xy}, f_m is calculated:

$$f_m = \sqrt{\sigma_x^2 + 3\tau_{xy}^2} = \sqrt{(133.85)^2 + 3(11.10)^2} = 135.22 N / mm^2$$

Since $f_t = FYLD/1.5 = 300.0\ MPa/15 = 200.0\ N/mm^2$ and k = 1 for permanent loading,

Ratio = $135.22/(200.0 \cdot 1) = 0.676 < 1$, So OK.

Comparison

Table 684: Comparison of results

	Hand Calculation	STAAD.Pro Result	Comments
von Mises Stress, f_m (N/mm^2)	135.22	135.21	negligible

STAAD Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD SPACE VERIFICATION FOR VON MISES STRESSES IN AIJ 2005
START JOB INFORMATION
ENGINEER DATE 18-AUG-10
END JOB INFORMATION
UNIT MMS KN
JOINT COORDINATES
1 0 0 0; 2 300 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
UNIT METER KN
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
1 TABLE ST H100x50x5x7
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
UNIT METER KN
LOAD 1 LC1
JOINT LOAD
2 FX 10 FY 3 FZ 2 MX 0.008
PERFORM ANALYSIS
LOAD LIST 1
PRINT MEMBER FORCES LIST ALL
PARAMETER 1
CODE JAPANESE 2005
UNL 0.002 ALL
MISES 1 ALL
TRACK 2 ALL
FYLD 300000 ALL
CHECK CODE ALL
FINISH
```

Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:       1        Profile:ST  H100X50X5X7      (JAPANESE
SECTIONS)  |
|  Ratio:    0.676  (PASS)  Reference: Eq.5.24          Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    0.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(T)      -10.000       Vy:        -3.000          Vz:      -2.000
|
|  Tx:          -0.008       My:         0.600          Mz:      -0.900
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:    0.000 MM              (UNIT: MM)
|
```

```
|  Ax:     1.18500E+03       Az: 4.66655E+02            Ay: 5.00000E+02
|
|  Iz:     1.87000E+06       Iy: 1.48000E+05             J: 1.53000E+04
|
|  Zz:     3.74000E+04       Zy: 5.92000E+03            Zx: 2.18571E+03
|
|  iZ:     3.97248E+01       iY: 1.11756E+01            Iw: 3.20013E+08
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         300.000      E :   205000.000          G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     0.300      Lz:     0.300 Ly:     0.300  UNL:      0.002
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Tension)
|
|  Actual :     26.844      Allowable :    400.000      Ratio :  0.067
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|-----------------------------------------------------------------------------
|
|           Compression             |             Bending
|
|-----------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable  |   LC#   |    Actual    |   Allowable
|
|-----------------------------------------------------------------------------
|
|   -    |      -       |      -       |     1   |    14.0000   |     62.7376
|
|-----------------------------------------------------------------------------
|
| CHECKS                            |Env|     Stresses     |
|
|             | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|             |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|-------------|------|--------|------|---|--------|--------|------|----------
|
| Tension     | 0.000|  -10.00|     1| P |    8.44|  200.00| 0.042| Eq.5.1
|
```

```
| Compression  |  -   |   -    |  -   | - |   -    |  189.47|  -   | Eq.5.3
|
| Bending Z (T)| 0.000|   -0.90|     1| P |   24.06|  200.00| 0.120| Eq.5.1
|
| Bending Z (C)| 0.000|   -0.90|     1| P |   24.06|  200.00| 0.120| Eq.5.7
|
| Bending Y (T)| 0.000|    0.60|     1| P |  101.35|  200.00| 0.507| Eq.5.1
|
| Bending Y (C)| 0.000|    0.60|     1| P |  101.35|  200.00| 0.507| Eq.5.1
|
| Shear Z      | 0.000|   -2.00|     1| P |    4.29|  115.47| 0.037| Eq.5.2
|
| Shear Y      | 0.000|   -3.00|     1| P |    6.00|  115.47| 0.052| Eq.5.2
|
| Comp+ Bend C | 0.000|   -    |     1| P |   -    |    -   | 0.627| Eq.6.1
|
| Comp+ Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.627| Eq.6.2
|
| Ten + Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.669| Eq.6.3
|
| Ten + Bend C | 0.000|   -    |     1| P |   -    |    -   | 0.585| Eq.6.4
|
| Von-Mises    | 0.000|   -    |     1| P |  135.21|  200.00| 0.676| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 6.0918E+06 KN-MET  My: 1.1220E+01 KN-MET   pλb:   0.600  C:  1.750
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:   133.854 N/mm2   Tou:    11.034 N/mm2  fm:   135.212 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.4 Von Mises Stresses Check* (on page 1727)

V.AIJ 2005 UPT Channel

Verify the bending capacity and compression capacity of a channel section beam.

Details

The member is a 5 m long simply-supported beam. The section used is a C380X100X10.5. The steel has a yield strength of 235 MPa, a modulus of elasticity of $205\times(10)^3$ MPa, and a shear modulus of $79\times(10)^3$ MPa.

The member is subject to the following loads:

a 10 kN·m concentrated moment at the left end
a -3 kN·m concentrated moment at the right end

Validation

Section Properties

Dimensions:

Depth, $D = 380\ mm$
Width, $B = 100\ mm$
Thickness of web, $t_w = 10.5\ mm$
Thickness of flange, $t_f = 16\ mm$

Height of web, $d = D - 2t_f = 348\ mm$

Area, $A_x = 2B \times t_f + d \times t_w = 6{,}854\ mm^2$

Distance from heel to centroid, $c_z = \dfrac{t_f B^2 + \frac{1}{2} t_w{}^2 d}{A} = \dfrac{16(100)^2 + \frac{1}{2}(10.5)^2(348)}{6{,}854} = 26.14\ mm$

Moment of inertia about major axis, $I_z = \dfrac{d^3 t_w}{12} + 2\left[\dfrac{t_f{}^3 B}{12} + t_f B\left(\dfrac{t_f}{2} + \dfrac{d}{2}\right)^2\right] = 142.9(10)^6\ mm^4$

Moment of inertia about minor axis, $I_y = \dfrac{t_w{}^3 d}{12} + t_w d\left(c_z - \dfrac{t_w}{2}\right)^2 + 2\left[\dfrac{t_f B^3}{12} + t_f B\left(\dfrac{B}{2} - c_z\right)^2\right] = 6.117(10)^6\ mm^4$

Major axis section modulus, $Z_z = \dfrac{I_z}{D/2} = 752.3(10)^3\ mm^3$

Minor axis section modulus, $Z_y = \dfrac{I_y}{B - c_z} = 82.82(10)^3\ mm^3$

Torsional constant, $J = \frac{1}{3}\left(d t_w{}^3 + 2B t_f{}^3\right) = 407.4(10)^3\ mm^4$

Warping constant, $C_W = 144.3(10)^9\ mm^6$

Radius of gyration about major axis, $r_z = \sqrt{I_z / A} = 144.4\ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y / A} = 29.87\ mm$

Section Classification

For webs of beams (Cl. 8.5):

$$\frac{d}{t_w} = 33.14 < 2.4\sqrt{\frac{E}{F}} = 2.4\sqrt{\frac{205{,}000}{235}} = 70.89$$

Depth to thickness ratio is “OK”.

Allowable Compressive Stress

The slenderness ratio of compression members (Cl. 11.1):

$$\lambda = \frac{K_y \times L_y}{r_y} = \frac{1 \times 5{,}000}{29.87} = 167.4$$

Slenderness ratio dividing elastic and inelastic buckling (Cl no 5.5):

$$\Lambda = \sqrt{\frac{\pi^2 \times E}{0.6 \times F}} = \sqrt{\frac{\pi^2 \times 205{,}000}{0.6 \times 235}} = 119.8$$

$$v = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda}{\Lambda}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{167.4}{119.8}\right)^2 = 2.802$$

$\lambda > \Lambda$, so the allowable compressive stress is given by Cl. 5.4:

$$f_c = \frac{0.277 \times F}{\left(\frac{\lambda}{\Lambda}\right)^2} = \frac{0.277 \times 235}{\left(\frac{167.4}{119.8}\right)^2} = 33.35 \text{ MPa}$$

Allowable Flexural Stress

Distance between braced points of the compression flange, $l_b = 5{,}000\ mm$.

The yield moment, $M_y = F \times Z_z = \frac{235 \times 752.3(10)^3}{10^6} = 176.8\ kN{\cdot}m$

The modification factor for allowable flexural stress (Cl. 5.13):

$$C = 1.75 - 1.05 \times \left(\frac{M_2}{M_1}\right) + 0.3 \times \left(\frac{M_2}{M_1}\right)^2 = 1.75 - 1.05 \times \left(\frac{3}{10}\right) + 0.3 \times \left(\frac{3}{10}\right)^2 = 1.462$$

The elastic lateral-torsional buckling moment (Cl 5.12):

$$M_e = C \times \sqrt{\frac{\pi^4 \times E \times I_y \times E \times I_w}{{l_b}^4} + \frac{\pi^2 \times E \times I_y \times G \times J}{{l_b}^2}}$$

$$\frac{\pi^4 \times E \times I_y \times E \times I_w}{{l_b}^4} = \frac{\pi^4 \times 205{,}000 \times 6.117 \times 10^6 \times 205{,}000 \times 144.3 \times 10^9}{5{,}000^4} = 5.791 \times 10^{15}$$

$$\frac{\pi^2 \times E \times I_y \times G \times J}{{l_b}^2} = \frac{\pi^2 \times 205{,}000 \times 6.117 \times 10^6 \times 79{,}000 \times 407.4 \times 10^3}{5{,}000^2} = 15.93 \times 10^{15}$$

$$= 1.462 \times \sqrt{5.791 \times 10^{15} + 15.93 \times 10^{15}} = 215.4 \text{ kN·m}$$

The maximum slenderness ratio to achieve the plastic strength (Cl. 5.12):

$$p\lambda_b = 0.6 + 0.3 \times \left(\frac{-M_2}{M_1}\right) = 0.6 + 0.3 \times \left(\frac{-3}{10}\right) = 0.51$$

The elastic slenderness limit:

$$e\lambda_b = \frac{1}{\sqrt{0.6}} = 1.291$$

Slenderness ratio of the flexural member, $\lambda_b = \sqrt{\frac{M_y}{M_e}} = 0.906$

Here, $p\lambda_b < \lambda_b \leq e\lambda_b$, so:

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda_b}{e\lambda_b}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{0.483}{1.291}\right)^2 = 1.828$$

$$f_b = \frac{\left(1 - 0.4 \times \frac{\lambda_b - p\lambda_b}{e\lambda_b - p\lambda_b}\right) \times F}{\nu} = \frac{\left(1 - 0.4 \times \frac{0.906 - 0.3}{1.291 - 0.3}\right) \times 235}{1.828} = 102.5 \text{ MPa}$$

Results

Table 685:

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	33.35	33.34	negligible	
Allowable flexural stress (MPa)	102.5	102.48	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 UPT Channel.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Oct-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
31 19 0 0; 32 24 0 0;
MEMBER INCIDENCES
16 31 32;
START USER TABLE
TABLE 1
UNIT METER KN
GENERAL
GE_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.000480667 -
6.77333e-05 0.00195 0.0018 0 0 1.07545e-07 0.256
*0.004678 0.3 0 0.15 0 7.21e-05 5.08e-06 9.95e-08 0.000480667 6.77333e-05 -
*0.00195 0.0018 0 0 1.07545e-07 0.256
TABLE 2
UNIT METER KN
WIDE FLANGE
WF_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 0.0018
WF2_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 -
0.0018 0.15 0.009
WF_H300X100-200X7X9-12
0.005253 0.3 0.007 0.1 0.009 7.29942e-05 8.75797e-06 1.71399e-07 0.0021 -
0.0022 0.2 0.012
TABLE 3
UNIT METER KN
ISECTION
IS_I300X150X8
0.3 0.008 0.3 0.15 0.013 0.15 0.013 0.0024 0.0026 2.69e-07
TABLE 4
UNIT METER KN
```

```
CHANNEL
CH_C380X100X10.5
0.006854 0.38 0.0105 0.1 0.016 0.000142897 6.11657e-06 4.07351e-07 -
0.0261429 0.00399 0.00213333
TABLE 5
UNIT METER KN
ANGLE
AG_L175X175X15
0.175 0.175 0.015 0.0342 0.00175 0.00175
TABLE 6
UNIT METER KN
TUBE
TB_TUB1501506
0.003363 0.15 0.15 0.006 1.15e-05 1.15e-05 1.79e-05 0.0018 0.0018
TB_RHS200X100X9
0.004867 0.2 0.1 0.009 2.35e-05 7.82e-06 1.93e-05 0.0036 0.0018
TABLE 7
UNIT METER KN
PIPE
PP_PIP165.2X7.1
0.1652 0.151 0.001763 0.001763
TABLE 8
UNIT METER KN
DOUBLE ANGLE
DA_L100X100X13LD
0.1 0.1 0.013 0 4.48744e-06 8.79409e-06 2.73893e-07 0.0294 0.00173333 -
0.00173333 1.94
TABLE 9
UNIT METER KN
TEE
TE_CT100X200X8X12
0.003177 0.1 0.2 0.012 0.008 1.84e-06 8.01e-06 1.3e-07 0.0173 0.000533333 -
0.0016
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
16 UPTABLE 4 CH_C380X100X10.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
31 PINNED
32 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
31 MZ 10
32 MZ -3
*148 FX 0.01 FY 0.01 MX 0.8
PERFORM ANALYSIS
PARAMETER 1
```

```
CODE JAPANESE 2005
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:      16        Profile  :
CH_C380X100X10(UPT)                    |
|  Ratio:    0.130  (PASS)  Reference: Eq.5.8           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    0.000  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)        0.000       Vy:         1.400          Vz:       0.000
|
|  Tx:           0.000       My:         0.000          Mz:      10.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:    0.000 MM              (UNIT: MM)
|
|  Ax:     6.85400E+03       Az: 2.13333E+03            Ay: 3.99000E+03
|
|  Iz:     1.42897E+08       Iy: 6.11657E+06             J: 4.07351E+05
|
|  Zz:     7.52090E+05       Zy: 8.28163E+04            Zx: 2.54594E+04
|
|  iZ:     1.44391E+02       iY: 2.98732E+01            Iw: 1.44256E+11
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   205000.000          G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     5.000      Lz:     5.000 Ly:     5.000  UNL:      5.000
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Tension)
|
```

```
|  Actual :    167.374      Allowable :    400.000      Ratio :  0.418
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|-----------------------------------------------------------------------------
|
|           Compression            |              Bending
|
|-----------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable  |   LC#   |    Actual    |   Allowable
|
|-----------------------------------------------------------------------------
|
|   -    |      -       |      -       |     1   |    33.1429   |     70.8850
|
|-----------------------------------------------------------------------------
|
| CHECKS                             |Env|     Stresses     |
|
|              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|--------------|------|--------|------|---|--------|--------|------|----------
|
| Tension      |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Compression  |  -   |   -    |  -   | - |   -    |  33.34|  -   | Eq.5.4
|
| Bending Z (T)| 0.000|   10.00|     1| P |   13.30|  156.67| 0.085| Eq.5.1
|
| Bending Z (C)| 0.000|   10.00|     1| P |   13.30|  102.48| 0.130| Eq.5.8
|
| Bending Y (T)|  -   |   -    |  -   | - |   -    |  156.67|  -   | Eq.5.1
|
| Bending Y (C)|  -   |   -    |  -   | - |   -    |  156.67|  -   | Eq.5.1
|
| Shear Z      |  -   |   -    |  -   | - |   -    |   90.45|  -   | Eq.5.2
|
| Shear Y      | 0.000|    1.40|     1| P |    0.35|   90.45| 0.004| Eq.5.2
|
| Comp+ Bend C | 0.000|   -    |     1| P |   -    |    -   | 0.130| Eq.6.1
|
| Comp+ Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.085| Eq.6.2
|
| Ten + Bend T | 0.000|   -    |     1| P |   -    |    -   | 0.085| Eq.6.3
|
| Ten + Bend C | 0.000|   -    |     1| P |   -    |    -   | 0.130| Eq.6.4
|
| Von-Mises    | 0.000|   -    |     1| P |   13.31|  156.67| 0.085| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
```

```
|  Me: 2.1541E+02 KN-MET  My: 1.7674E+02 KN-MET   pλb:   0.510  C:  1.462
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:    13.296 N/mm2   Tou:     0.351 N/mm2  fm:    13.310 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V.AIJ 2005 UPT Double Angle

Verify the bending capacity and compression capacity of a double angle section beam.

Details

The member is a 5 m long simply-supported beam. The section used is a 2x L100x100x13, with no spacing. The steel has a yield strength of 235 MPa, a modulus of elasticity of $205 \times (10)^3$ MPa, and a shear modulus of $79 \times (10)^3$ MPa.

The member is subject to the following loads:

a -5 kN concentrated force at mid-span in the global Y direction
a 3 kN concentrated force at mid-span in the global Z direction
a -40 kN axial force
a -0.2 kN·m concentrated moment at the left end

Validation

<u>Section Properties</u>

Dimensions:

Leg, $D = 100\ mm$
Width, $B = 200\ mm$
Thickness, $t = 13\ mm$

Outstanding leg length, $d = D - t = 87\ mm$

Area, $A_x = B \times t + 2d \times t = 4{,}862\ mm^2$

Distance from heel to centroid, $c_y = \dfrac{\frac{1}{2} B t^2 + 2dt\left(t + \frac{d}{2}\right)}{A} = \dfrac{\frac{1}{2} 200(13)^2 + 2\left(87\right)\left(13\right)\left(13 + \frac{87}{2}\right)}{4{,}862} = 29.76\ mm$

Moment of inertia about major axis, $I_z = \dfrac{Bt^3}{12} + Bt\left(c_y - \frac{t}{2}\right) + 2\left[\dfrac{td^3}{12} + td\left(t + \frac{d}{2} - c_y\right)^2\right] = 4.487(10)^6\ mm^4$

Moment of inertia about minor axis, $I_y = \dfrac{tB^3}{12} + \frac{2}{3} d t^3 = 8.676(10)^6\ mm^4$

Major axis section modulus, $Z_z = \frac{I_z}{D - c_y} = 63.81(10)^3\ mm^3$

Minor axis section modulus, $Z_y = \frac{I_y}{D} = 86.76(10)^3\ mm^3$

Torsional constant, $J = 273.9(10)^3\ mm^4$

Warping constant, $C_W = 199.5(10)^9\ mm^6$

Radius of gyration about major axis, $r_z = \sqrt{I_z/A} = 30.38\ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y/A} = 42.53\ mm$

<u>Section Classification</u>

For webs of beams (Cl. 8.1):

$$\frac{d}{t_w} = 7.69 < 0.44\sqrt{\frac{E}{F}} = 0.44\sqrt{\frac{205{,}000}{235}} = 13.0$$

Depth to thickness ratio is “OK”.

<u>Allowable Compressive Stress</u>

The slenderness ratio of compression members (Cl. 11.1):

$$\lambda_z = \frac{K_z \times L_z}{r_z} = \frac{1 \times 5{,}000}{30.38} = 164.6$$

$$\lambda_y = \frac{K_y \times L_y}{r_y} = \frac{1 \times 5{,}000}{42.53} = 117.6$$

Therefore, $\lambda = 164.6$.

Slenderness ratio dividing elastic and inelastic buckling (Cl no 5.5):

$$\Lambda = \sqrt{\frac{\pi^2 \times E}{0.6 \times F}} = \sqrt{\frac{\pi^2 \times 205{,}000}{0.6 \times 235}} = 119.8$$

$$\text{v} = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda}{\Lambda}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{164.6}{119.8}\right)^2 = 2.758$$

$\lambda > \Lambda$, so the allowable compressive stress is given by Cl. 5.4:

$$f_c = \frac{0.277 \times F}{\left(\frac{\lambda}{\Lambda}\right)^2} = \frac{0.277 \times 235}{\left(\frac{164.6}{119.8}\right)^2} = 34.49\ \text{MPa}$$

<u>Allowable Flexural Stress</u>

Distance between braced points of the compression flange, $l_b = 5{,}000\ mm$.

The yield moment, $M_y = F \times Z_z = \frac{235 \times 87.94(10)^3}{10^6} = 20.67\ kN \cdot m$

The modification factor for allowable flexural stress (Cl. 5.13):

$$C = 1$$

The elastic lateral-torsional buckling moment (Cl 5.12):

$$M_e = C \times \sqrt{\frac{\pi^4 \times E \times I_{zz} \times E \times I_w}{l_b^4} + \frac{\pi^2 \times E \times I_{zz} \times G \times J}{l_b^2}}$$

$$\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b^4} = \frac{\pi^4 \times 205{,}000 \times 4.847 \times 10^6 \times 205{,}000 \times 199.5 \times 10^6}{5{,}000^4} = 6.333 \times 10^{12}$$

$$\frac{\pi^2 \times E \times I_y \times G \times J}{l_b^2} = \frac{\pi^2 \times 205{,}000 \times 4.847 \times 10^6 \times 79{,}000 \times 273.9 \times 10^3}{5{,}000^2} = 8.488 \times 10^{15}$$

$$= 1.0 \times \sqrt{6.333 \times 10^{12} + 8.488 \times 10^{15}} = 88.59 \text{ kN·m}$$

The maximum slenderness ratio to achieve the plastic strength (Cl. 5.12):

$$p\lambda_b = 0.3$$

The elastic slenderness limit:

$$e\lambda_b = \frac{1}{\sqrt{0.6}} = 1.291$$

Slenderness ratio of the flexural member, $\lambda_b = \sqrt{\frac{M_y}{M_e}} = 0.483$

Here, $p\lambda_b < \lambda_b \leq e\lambda_b$, so:

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda_b}{e\lambda_b}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{0.483}{1.291}\right)^2 = 1.593$$

$$f_b = \frac{\left(1 - 0.4 \times \frac{\lambda_b - p\lambda_b}{e\lambda_b - p\lambda_b}\right) \times F}{\nu} = \frac{\left(1 - 0.4 \times \frac{0.483 - 0.3}{1.291 - 0.3}\right) \times 235}{1.593} = 136.6 \text{ MPa}$$

Results

Table 686:

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	34.49	34.48	negligible	
Allowable flexural stress (MPa)	136.6	136.62	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\Japan\AIJ 2005 UPT Double Angle.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 22-Oct-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
29 12 0 8; 30 17 0 8;
MEMBER INCIDENCES
15 29 30;
START USER TABLE
TABLE 1
UNIT METER KN
GENERAL
GE_H300X150X6.5X9
0.004678 0.3 0 0.15 0 7.21e-05 5.08e-06 9.95e-08 0.000480667 6.77333e-05 -
0.00195 0.0018 0 0 1.07545e-07 0.256
TABLE 2
UNIT METER KN
WIDE FLANGE
WF_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 0.0018
WF2_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 -
0.0018 0.15 0.009
TABLE 3
UNIT METER KN
ISECTION
IS_I300X150X8
0.3 0.008 0.3 0.15 0.013 0.15 0.013 0.0024 0.0026 2.69e-07
TABLE 4
UNIT METER KN
CHANNEL
CH_C380X100X10.5
0.006939 0.38 0.0105 0.1 0.016 0.000145 5.35e-06 3.99e-07 0.0241 0.00399 -
0.00213333
TABLE 5
UNIT METER KN
ANGLE
AG_L175X175X15
0.175 0.175 0.015 0.0342 0.00175 0.00175
TABLE 6
UNIT METER KN
TUBE
TB_TUB1501506
0.003363 0.15 0.15 0.006 1.15e-05 1.15e-05 1.79e-05 0.0018 0.0018
TB_RHS200X100X9
0.004867 0.2 0.1 0.009 2.35e-05 7.82e-06 1.93e-05 0.0036 0.0018
TABLE 7
UNIT METER KN
PIPE
PP_PIP165.2X7.1
0.1652 0.151 0.001763 0.001763
TABLE 8
UNIT METER KN
DOUBLE ANGLE
L100X100X13_LD
0.1 0.1 0.013 0 4.48744e-06 8.79409e-06 2.73893e-07 0.029762 0.00173333 -
0.00173333 0
END
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
15 UPTABLE 8 L100X100X13_LD
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
29 PINNED
30 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
15 CON GY -5
15 CON GZ 3
JOINT LOAD
30 FX -40
29 MX 0.2
PERFORM ANALYSIS
PARAMETER 1
CODE JAPANESE 2005
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:      15        Profile  :
L100X100X13_LD(UPT)                    |
|  Ratio:    0.923  (PASS)  Reference: Eq.6.3           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    2.500  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)       40.000       Vy:         2.500          Vz:      -1.500
|
|  Tx:           0.200       My:        -3.750          Mz:      -6.250
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE: 2500.000 MM              (UNIT: MM)
|
|  Ax:     4.86200E+03       Az: 1.73333E+03            Ay: 1.73333E+03
|
```

```
|  Iz:     4.48744E+06       Iy: 8.79409E+06            J: 2.73893E+05
|
|  Zz:     6.12519E+04       Zy: 8.79409E+04           Zx: 2.10687E+04
|
|  iZ:     3.03803E+01       iY: 4.25293E+01           Iw: 1.99536E+08
|
|----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   205000.000         G :     79000.000
|
|----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     5.000      Lz:     5.000 Ly:     5.000  UNL:      5.000
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
|
|----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Compression)
|
|  Actual :    164.580      Allowable :    200.000      Ratio :  0.823
|
|----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|----------------------------------------------------------------------------
|
|           Compression               |             Bending
|
|----------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable  |   LC#   |    Actual    |   Allowable
|
|----------------------------------------------------------------------------
|
|      1 |     7.6923   |     12.9956  |      1  |     7.6923   |     12.9956
|
|----------------------------------------------------------------------------
|
| CHECKS                              |Env|     Stresses     |
|
|             | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|             |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|-------------|------|--------|------|---|--------|--------|------|----------
|
| Tension     |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Compression | 0.000|   40.00|     1| P |    8.23|   34.48| 0.239| Eq.5.4
|
```

```
| Bending Z (T)| 2.500|   -6.25|      1| P |  102.04|  156.67| 0.651| Eq.5.1
|
| Bending Z (C)| 2.500|   -6.25|      1| P |   37.24|  156.67| 0.238| Eq.5.1
|
| Bending Y (T)| 2.500|   -3.75|      1| P |   42.64|  156.67| 0.272| Eq.5.1
|
| Bending Y (C)| 2.500|   -3.75|      1| P |   42.64|  136.62| 0.312| Eq.5.8
|
| Shear Z      | 0.000|   -1.50|      1| P |    0.87|   90.45| 0.010| Eq.5.2
|
| Shear Y      | 0.000|    2.50|      1| P |    1.44|   90.45| 0.016| Eq.5.2
|
| Comp+ Bend C | 2.500|   -    |      1| P |   -    |    -   | 0.788| Eq.6.1
|
| Comp+ Bend T | 2.500|   -    |      1| P |   -    |    -   | 0.871| Eq.6.2
|
| Ten + Bend T | 2.500|   -    |      1| P |   -    |    -   | 0.923| Eq.6.3
|
| Ten + Bend C | 2.500|   -    |      1| P |   -    |    -   | 0.550| Eq.6.4
|
| Von-Mises    | 2.500|   -    |      1| P |  137.82|  156.67| 0.880| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 8.8679E+01 KN-MET   My: 2.0666E+01 KN-MET   pλb:   0.300  C:  1.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:   136.453 N/mm2   Tou:    11.175 N/mm2  fm:   136.819 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V.AIJ 2005 UPT General

Verify the bending capacity and compression capacity of an H section beam.

Details

The member is a 5 m long simply-supported beam. The section used is an H300x150x6.5x9. The steel has a yield strength of 235 MPa, a modulus of elasticity of 205×(10)^3 MPa, and a shear modulus of 79×(10)^3 MPa.

The member is subject to the following loads:

a -10 kN concentrated force at mid-span in the global Y direction
a 3 kN concentrated force at mid-span in the global Z direction
a -40 kN axial force
a -0.2 kN·m concentrated moment at the left end

Validation

<u>Section Properties</u>

Dimensions:

Leg, $D = 300\ mm$
Width, $B = 150\ mm$
Thickness of web, $t_w = 6.5\ mm$
Thickness of flange, $t_f = 9\ mm$

Depth of web, $d = D - 2t_f = 282\ mm$

Area, $A_x = 4{,}678\ mm^2$

Moment of inertia about major axis, $I_z = 72.1(10)^6\ mm^4$

Moment of inertia about minor axis, $I_y = 0.508(10)^6\ mm^4$

Major axis section modulus, $Z_z = 480.7(10)^3\ mm^3$

Minor axis section modulus, $Z_y = 67.73(10)^3\ mm^3$

Torsional constant, $J = 99.5(10)^3\ mm^4$

Warping constant, $C_W = 107.5(10)^9\ mm^6$

Radius of gyration about major axis, $r_z = \sqrt{I_z / A} = 124.1\ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y / A} = 32.95\ mm$

<u>Section Classification</u>

For webs of beams in compression (Cl. 8.1):

$$\frac{d}{t_w} = \frac{282}{8} = 43.39 < 1.6\sqrt{\frac{E}{F}} = 1.6\sqrt{\frac{205{,}000}{235}} = 47.26$$

Depth to thickness ratio is “OK”.

For webs of beams in bending (Cl. 8.1):

$$\frac{d}{t_w} = 43.39 < 2.4\sqrt{\frac{E}{F}} = 2.4\sqrt{\frac{205{,}000}{235}} = 70.89$$

Depth to thickness ratio is “OK”.

<u>Allowable Compressive Stress</u>

The slenderness ratio of compression members (Cl. 11.1):

$$\lambda_z = \frac{K_z \times L_z}{r_z} = \frac{1 \times 5{,}000}{124.1} = 40.27$$

$$\lambda_y = \frac{K_y \times L_y}{r_y} = \frac{1 \times 5{,}000}{32.95} = 151.7$$

Therefore, $\lambda = 151.7$.

Slenderness ratio dividing elastic and inelastic buckling (Cl no 5.5):

$$\Lambda = \sqrt{\frac{\pi^2 \times E}{0.6 \times F}} = \sqrt{\frac{\pi^2 \times 205{,}000}{0.6 \times 235}} = 119.8$$

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda}{\Lambda}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{151.7}{119.8}\right)^2 = 2.570$$

$\lambda > \Lambda$, so the allowable compressive stress is given by Cl. 5.4:

$$f_c = \frac{0.277 \times F}{\left(\frac{\lambda}{\Lambda}\right)^2} = \frac{0.277 \times 235}{\left(\frac{151.7}{119.8}\right)^2} = 40.57 \text{ MPa}$$

<u>Allowable Flexural Stress</u>

Distance between braced points of the compression flange, $l_b = 5{,}000\ mm$.

The yield moment, $M_y = F \times Z_z = \frac{235 \times 480.7(10)^3}{10^6} = 113.0\ kN \cdot m$

The modification factor for allowable flexural stress (Cl. 5.13):

$$C = 1$$

The elastic lateral-torsional buckling moment (Cl 5.12):

$$M_e = C \times \sqrt{\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b^4} + \frac{\pi^2 \times E \times I_y \times G \times J}{l_b^2}}$$

$$\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b^4} = \frac{\pi^4 \times 205{,}000 \times 50.8 \times 10^6 \times 205{,}000 \times 107.5 \times 10^9}{5{,}000^4} = 35.77 \times 10^{15}$$

$$\frac{\pi^2 \times E \times I_y \times G \times J}{l_b^2} = \frac{\pi^2 \times 205{,}000 \times 50.8 \times 10^6 \times 79{,}000 \times 99.54 \times 10^3}{5{,}000^2} = 32.32 \times 10^{15}$$

$$= 1.0 \times \sqrt{35.77 \times 10^{15} + 32.32 \times 10^{15}} = 82.49 \text{ kN} \cdot \text{m}$$

The maximum slenderness ratio to achieve the plastic strength (Cl. 5.12):

$$p\lambda_b = 0.3$$

The elastic slenderness limit:

$$e\lambda_b = \frac{1}{\sqrt{0.6}} = 1.291$$

Slenderness ratio of the flexural member, $\lambda_b = \sqrt{\frac{M_y}{M_e}} = 1.17$

Here, $p\lambda_b < \lambda_b \le e\lambda_b$, so:

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda_b}{e\lambda_b}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{1.17}{1.291}\right)^2 = 2.048$$

$$f_b = \frac{\left(1 - 0.4 \times \frac{\lambda_b - p\lambda_b}{e\lambda_b - p\lambda_b}\right) \times F}{\nu} = \frac{\left(1 - 0.4 \times \frac{1.17 - 0.3}{1.291 - 0.3}\right) \times 235}{2.048} = 74.45 \text{ MPa}$$

Results

Table 687:

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	40.57	40.57	none	
Allowable flexural stress (MPa)	74.45	74.47	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 UPT General.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Oct-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
107 26 0 0; 108 31 0 0;
MEMBER INCIDENCES
54 107 108;
START USER TABLE
TABLE 1
UNIT METER KN
GENERAL
UT_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.000480667 -
6.77333e-05 0.00195 0.0018 0 0 1.07545e-07 0.256
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
54 UPTABLE 1 UT_H300X150X6.5X9
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
107 PINNED
108 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
54 CON GY -10
54 CON GZ 3
JOINT LOAD
108 FX -40
107 MX 0.2
PERFORM ANALYSIS
PARAMETER 1
CODE JAPANESE 2005
SLF 1 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:      54        Profile  : UT_H300X150X6.
(UPT)                   |
|  Ratio:    0.913  (PASS)  Reference: Eq.6.1           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    2.500  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)      40.000      Vy:         5.000          Vz:      -1.500
|
|  Tx:          0.200      My:        -3.750          Mz:     -12.500
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE: 2499.999 MM             (UNIT: MM)
|
|  Ax:     4.67800E+03      Az: 1.80000E+03           Ay: 1.95000E+03
|
|  Iz:     7.21000E+07      Iy: 5.08000E+06            J: 9.95000E+04
|
|  Zz:     4.80667E+05      Zy: 6.77333E+04           Zx: 1.10556E+04
|
|  iZ:     1.24147E+02      iY: 3.29535E+01           Iw: 1.07545E+11
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000     E :   205000.000         G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     5.000      Lz:     5.000 Ly:     5.000  UNL:      5.000
|
```

```
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Compression)
|
|  Actual :    151.729      Allowable :    200.000      Ratio :  0.759
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|-----------------------------------------------------------------------------
|
|           Compression                |             Bending
|
|-----------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable  |   LC#   |    Actual    |   Allowable
|
|-----------------------------------------------------------------------------
|
|      1 |    43.3846   |     47.2567  |      1  |    43.3846   |     70.8850
|
|-----------------------------------------------------------------------------
|
| CHECKS                              |Env|     Stresses    |
|
|              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|--------------|------|--------|------|---|--------|--------|------|----------
|
| Tension      |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Compression  | 0.000|   40.00|     1| P |    8.55|   40.57| 0.211| Eq.5.4
|
| Bending Z (T)| 2.500|  -12.50|     1| P |   26.01|  156.67| 0.166| Eq.5.1
|
| Bending Z (C)| 2.500|  -12.50|     1| P |   26.01|   74.47| 0.349| Eq.5.8
|
| Bending Y (T)| 2.500|   -3.75|     1| P |   55.36|  156.67| 0.353| Eq.5.1
|
| Bending Y (C)| 2.500|   -3.75|     1| P |   55.36|  156.67| 0.353| Eq.5.1
|
| Shear Z      | 0.000|   -1.50|     1| P |    0.83|   90.45| 0.009| Eq.5.2
|
| Shear Y      | 0.000|    5.00|     1| P |    2.56|   90.45| 0.028| Eq.5.2
|
| Comp+ Bend C | 2.500|   -    |     1| P |   -    |    -   | 0.913| Eq.6.1
|
| Comp+ Bend T | 2.500|   -    |     1| P |   -    |    -   | 0.465| Eq.6.2
|
| Ten + Bend T | 2.500|   -    |     1| P |   -    |    -   | 0.519| Eq.6.3
|
```

```
| Ten + Bend C | 2.500|   -    |     1| P |   -    |    -   | 0.703| Eq.6.4
|
| Von-Mises    | 2.500|   -    |     1| P |   96.86|  156.67| 0.618| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 8.2523E+01 KN-MET  My: 1.1296E+02 KN-MET   pλb:   0.300  C:  1.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:    89.920 N/mm2   Tou:    20.787 N/mm2  fm:    96.860 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V.AIJ 2005 UPT I

Verify the bending capacity and compression capacity of an I section beam.

Details

The member is a 5 m long simply-supported beam. The section used is an I300X150X8. The steel has a yield strength of 235 MPa, a modulus of elasticity of $205\times(10)^3$ MPa, and a shear modulus of $79\times(10)^3$ MPa.

The member is subject to the following loads:

a -10 kN concentrated force at mid-span in the global Y direction
a 3 kN concentrated force at mid-span in the global Z direction
a -40 kN axial force
a -0.2 kN·m concentrated moment at the left end

Validation

<u>Section Properties</u>

Dimensions:

Leg, $D = 300\ mm$
Width, $B = 150\ mm$
Thickness of web, $t_w = 8\ mm$
Thickness of flange, $t_f = 13\ mm$

Depth of web, $d = D - 2t_f = 274\ mm$

Area, $A_x = 6{,}092\ mm^2$

Moment of inertia about major axis, $I_z = 94.08(10)^6\ mm^4$

Moment of inertia about minor axis, $I_y = 7.324(10)^6\ mm^4$

Major axis section modulus, $Z_z = 627.2(10)^3\ mm^3$

Minor axis section modulus, $Z_y = 97.66(10)^3\ mm^3$

Torsional constant, $J = 26.87(10)^3\ mm^4$

Warping constant, $C_W = 150.6(10)^9\ mm^6$

Radius of gyration about major axis, $r_z = \sqrt{I_z/A} = 124.3\ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y/A} = 34.67\ mm$

<u>Section Classification</u>

For webs of beams in bending (Cl. 8.5):

$$\frac{d}{t_w} = \frac{274}{8} = 34.25 < 2.4\sqrt{\frac{E}{F}} = 2.4\sqrt{\frac{205,000}{235}} = 70.89$$

Depth to thickness ratio is “OK”.

For flanges of beams in bending (Cl. 8.3):

$$\frac{B}{t_f} = \frac{150}{13} = 11.54 < 1.6\sqrt{\frac{E}{F}} = 1.6\sqrt{\frac{205,000}{235}} = 47.26$$

Depth to thickness ratio is “OK”.

<u>Allowable Compressive Stress</u>

The slenderness ratio of compression members (Cl. 11.1):

$$\lambda_y = \frac{K_y \times L_y}{r_y} = \frac{1 \times 5,000}{34.67} = 144.2$$

Slenderness ratio dividing elastic and inelastic buckling (Cl no 5.5):

$$\Lambda = \sqrt{\frac{\pi^2 \times E}{0.6 \times F}} = \sqrt{\frac{\pi^2 \times 205,000}{0.6 \times 235}} = 119.8$$

$$v = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda}{\Lambda}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{144.2}{119.8}\right)^2 = 2.466$$

$\lambda > \Lambda$, so the allowable compressive stress is given by Cl. 5.4:

$$f_c = \frac{0.277 \times F}{\left(\frac{\lambda}{\Lambda}\right)^2} = \frac{0.277 \times 235}{\left(\frac{144.2}{119.8}\right)^2} = 44.92\ \text{MPa}$$

<u>Allowable Flexural Stress</u>

Distance between braced points of the compression flange, $l_b = 5,000\ mm$.

The yield moment, $M_y = F \times Z_z = \frac{235 \times 627.2(10)^3}{10^6} = 147.4\ kN \cdot m$

The modification factor for allowable flexural stress (Cl. 5.13):

$$C = 1$$

The elastic lateral-torsional buckling moment (Cl 5.12):

$$M_e = C \times \sqrt{\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b{}^4} + \frac{\pi^2 \times E \times I_y \times G \times J}{l_b{}^2}}$$

$$\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b{}^4} = \frac{\pi^4 \times 205{,}000 \times 7.324 \times 10^6 \times 205{,}000 \times 150.6 \times 10^9}{5{,}000^4} = 7.224 \times 10^{15}$$

$$\frac{\pi^2 \times E \times I_y \times G \times J}{l_b{}^2} = \frac{\pi^2 \times 205{,}000 \times 7.324 \times 10^6 \times 79{,}000 \times 26.87 \times 10^3}{5{,}000^2} = 1.258 \times 10^{15}$$

$$= 1.0 \times \sqrt{7.224 \times 10^{15} + 1.258 \times 10^{15}} = 140.6 \text{ kN} \cdot \text{m}$$

The maximum slenderness ratio to achieve the plastic strength (Cl. 5.12):

$$p\lambda_b = 0.3$$

The elastic slenderness limit:

$$e\lambda_b = \frac{1}{\sqrt{0.6}} = 1.291$$

Slenderness ratio of the flexural member, $\lambda_b = \sqrt{\frac{M_y}{M_e}} = 1.024$

Here, $p\lambda_b < \lambda_b \le e\lambda_b$, so:

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda_b}{e\lambda_b}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{1.024}{1.291}\right)^2 = 1.919$$

$$f_b = \frac{\left(1 - 0.4 \times \frac{\lambda_b - p\lambda_b}{e\lambda_b - p\lambda_b}\right) \times F}{\nu} = \frac{\left(1 - 0.4 \times \frac{1.024 - 0.3}{1.291 - 0.3}\right) \times 235}{1.919} = 86.68 \text{ MPa}$$

Results

Table 688:

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	44.92	44.92	none	
Allowable flexural stress (MPa)	86.68	86.74	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 UPT I.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Oct-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
3 1 0 2; 4 6 0 2;
MEMBER INCIDENCES
2 3 4;
START USER TABLE
TABLE 1
UNIT METER KN
GENERAL
GE_H300X150X6.5X9
0.004678 0.3 0 0.15 0 7.21e-05 5.08e-06 9.95e-08 0.000480667 6.77333e-05 -
0.00195 0.0018 0 0 1.07545e-07 0.256
TABLE 2
UNIT METER KN
WIDE FLANGE
WF_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 0.0018
WF2_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 -
0.0018 0.15 0.009
TABLE 3
UNIT METER KN
ISECTION
IS_I300X150X8
0.3 0.008 0.3 0.15 0.013 0.15 0.013 0.0024 0.0026 2.69e-07
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
2 UPTABLE 3 IS_I300X150X8
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
3 PINNED
4 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 CON GY -10
2 CON GZ 3
```

```
JOINT LOAD
4 FX -40
3 MX 0.2
PERFORM ANALYSIS
PARAMETER 1
CODE JAPANESE 2005
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - METE  KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:       2        Profile  : IS_I300X150X8
(UPT)                    |
|  Ratio:    0.621  (PASS)  Reference: Eq.6.1           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:    2.500  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)       40.000       Vy:         5.000          Vz:      -1.500
|
|  Tx:           0.200       My:        -3.750          Mz:     -12.500
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE: 2500.000 MM              (UNIT: MM)
|
|  Ax:     6.09200E+03       Az: 2.60000E+03            Ay: 2.40000E+03
|
|  Iz:     9.40786E+07       Iy: 7.32419E+06             J: 2.69000E+05
|
|  Zz:     6.27191E+05       Zy: 9.76559E+04            Zx: 2.06923E+04
|
|  iZ:     1.24270E+02       iY: 3.46737E+01            Iw: 1.50822E+11
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   205000.000          G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:     5.000      Lz:     5.000 Ly:     5.000  UNL:      5.000
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
```

```
|
|--------------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Compression)
|
|  Actual :    144.202      Allowable :    200.000      Ratio :  0.721
|
|--------------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|--------------------------------------------------------------------------------
|
|            Compression               |             Bending
|
|--------------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable |   LC#   |    Actual    |   Allowable
|
|--------------------------------------------------------------------------------
|
|     1 |    34.2500  |     47.2567  |      1  |    34.2500  |     70.8850
|
|--------------------------------------------------------------------------------
|
| CHECKS                            |Env|     Stresses     |
|
|              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|--------------|------|--------|------|---|--------|--------|------|----------
|
| Tension      |  -   |   -    |  -   | - |   -    | 156.67|  -   | Eq.5.1
|
| Compression  | 0.000|   40.00|     1| P |    6.57|   44.92| 0.146| Eq.5.4
|
| Bending Z (T)| 2.500|  -12.50|     1| P |   19.93|  156.67| 0.127| Eq.5.1
|
| Bending Z (C)| 2.500|  -12.50|     1| P |   19.93|   86.74| 0.230| Eq.5.8
|
| Bending Y (T)| 2.500|   -3.75|     1| P |   38.40|  156.67| 0.245| Eq.5.1
|
| Bending Y (C)| 2.500|   -3.75|     1| P |   38.40|  156.67| 0.245| Eq.5.1
|
| Shear Z      | 0.000|   -1.50|     1| P |    0.58|   90.45| 0.006| Eq.5.2
|
| Shear Y      | 0.000|    5.00|     1| P |    2.08|   90.45| 0.023| Eq.5.2
|
| Comp+ Bend C | 2.500|   -    |     1| P |   -    |    -   | 0.621| Eq.6.1
|
| Comp+ Bend T | 2.500|   -    |     1| P |   -    |    -   | 0.330| Eq.6.2
|
| Ten + Bend T | 2.500|   -    |     1| P |   -    |    -   | 0.372| Eq.6.3
|
| Ten + Bend C | 2.500|   -    |     1| P |   -    |    -   | 0.475| Eq.6.4
|
| Von-Mises    | 2.500|   -    |     1| P |   68.05|  156.67| 0.434| Eq.5.24
```

```
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 1.4083E+02 KN-MET  My: 1.4739E+02 KN-MET   pλb:   0.300  C:  1.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:    64.896 N/mm2   Tou:    11.827 N/mm2  fm:    68.053 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V.AIJ 2005 UPT Tee

Verify the bending capacity and compression capacity of a tee section beam.

Details

The member is a 2 m long simply-supported beam. The section used is a CT100x200x8x12. The steel has a yield strength of 235 MPa, a modulus of elasticity of $205\times(10)^3$ MPa, and a shear modulus of $79\times(10)^3$ MPa.

The member is subject to the following loads:

a -10 kN concentrated force at mid-span in the global Y direction
a -2 kN·m concentrated moment at the left end
a 2 kN·m concentrated moment at the right end

Validation

Section Properties

Dimensions:

Leg, $D = 100\ mm$
Width, $B = 200\ mm$
Thickness of web, $t_w = 8\ mm$
Thickness of flange, $t_f = 12\ mm$

Depth of web, $d = D - t_f = 88\ mm$

Area, $A_x = 3{,}104\ mm^2$

Moment of inertia about major axis, $I_z = 1.844(10)^6\ mm^4$

Moment of inertia about minor axis, $I_y = 8.004(10)^6\ mm^4$

Major axis section modulus, $Z_z = 22.31(10)^3\ mm^3$

Minor axis section modulus, $Z_y = 80.04(10)^3\ mm^3$

Torsional constant, $J = 130(10)^3\ mm^4$

Warping constant, $C_W = 107.8(10)^9\ mm^6$

Radius of gyration about major axis, $r_z = \sqrt{I_z/A} = 24.07\ mm$

Radius of gyration about minor axis, $r_y = \sqrt{I_y/A} = 50.21\ mm$

Section Classification

For webs of beams in bending (Cl. 8.5):

$$\frac{d}{t_w} = \frac{88}{8} = 11 < 0.53\sqrt{\frac{E}{F}} = 0.53\sqrt{\frac{205{,}000}{235}} = 15.65$$

Depth to thickness ratio is “OK”.

Allowable Compressive Stress

The slenderness ratio of compression members (Cl. 11.1):

$$\lambda_z = \frac{K_z \times L_z}{r_z} = \frac{1 \times 2{,}000}{24.07} = 83.11$$

Slenderness ratio dividing elastic and inelastic buckling (Cl no 5.5):

$$\Lambda = \sqrt{\frac{\pi^2 \times E}{0.6 \times F}} = \sqrt{\frac{\pi^2 \times 205{,}000}{0.6 \times 235}} = 119.8$$

$$\nu = \frac{3}{2} + \frac{2}{3}\times\left(\frac{\lambda}{\Lambda}\right)^2 = \frac{3}{2} + \frac{2}{3}\times\left(\frac{83.11}{119.8}\right)^2 = 1.821$$

$\lambda < \Lambda$, so the allowable compressive stress is given by Cl. 5.3:

$$f_c = \frac{\left[1 - 0.4\left(\frac{\lambda}{\Lambda}\right)^2\right] \times F}{\nu} = \frac{\left[1 - 0.4\left(\frac{83.11}{119.8}\right)^2\right] \times 235}{1.821} = 104.2\ \mathrm{MPa}$$

Allowable Flexural Stress

Distance between braced points of the compression flange, $l_b = 2{,}000\ mm$.

The yield moment, $M_y = F \times Z_z = \frac{235 \times 80.04(10)^3}{10^6} = 18.8\ kN{\cdot}m$

The modification factor for allowable flexural stress (Cl. 5.13):

$$C = 1$$

The elastic lateral-torsional buckling moment (Cl 5.12):

$$M_e = C \times \sqrt{\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b^4} + \frac{\pi^2 \times E \times I_y \times G \times J}{l_b^2}}$$

$$\frac{\pi^4 \times E \times I_y \times E \times I_w}{l_b^4} = \frac{\pi^4 \times 205{,}000 \times 1.844 \times 10^6 \times 205{,}000 \times 107.8 \times 10^9}{2{,}000^4} = 50.86 \times 10^{15}$$

$$\frac{\pi^2 \times E \times I_y \times G \times J}{l_b^2} = \frac{\pi^2 \times 205{,}000 \times 1.844 \times 10^6 \times 79{,}000 \times 13 \times 10^3}{2{,}000^2} = 0.958 \times 10^{15}$$

$$= 1.0 \times \sqrt{50.86 \times 10^{15} + 0.958 \times 10^{15}} = 98.03 \text{ kN} \cdot \text{m}$$

The maximum slenderness ratio to achieve the plastic strength (Cl. 5.12):

$$p\lambda_b = 0.6$$

The elastic slenderness limit:

$$e\lambda_b = \frac{1}{\sqrt{0.6}} = 1.291$$

Slenderness ratio of the flexural member, $\lambda_b = \sqrt{\frac{M_y}{M_e}} = 0.43$

Here, $\lambda_b < p\lambda_b$, so:

$$\nu = \frac{3}{2} + \frac{2}{3} \times \left(\frac{\lambda_b}{e\lambda_b}\right)^2 = \frac{3}{2} + \frac{2}{3} \times \left(\frac{0.43}{1.291}\right)^2 = 1.573$$

$$f_b = \frac{F}{\nu} = \frac{235}{1.573} = 149.4 \text{ MPa}$$

Results

Table 689:

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable compressive stress (MPa)	104.2	104.24	negligible	
Allowable flexural stress (MPa)	149.4	149.04	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Japan\AIJ 2005 UPT Tee.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Oct-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
127 26 0 20; 128 28 0 20;
MEMBER INCIDENCES
64 127 128;
START USER TABLE
TABLE 1
```

```
UNIT METER KN
GENERAL
GE_H300X150X6.5X9
0.004678 0.3 0 0.15 0 7.21e-05 5.08e-06 9.95e-08 0.000480667 6.77333e-05 -
0.00195 0.0018 0 0 1.07545e-07 0.256
TABLE 2
UNIT METER KN
WIDE FLANGE
WF_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 0.0018
WF2_H300X150X6.5X9
0.004678 0.3 0.0065 0.15 0.009 7.21e-05 5.08e-06 9.95e-08 0.00195 -
0.0018 0.15 0.009
TABLE 3
UNIT METER KN
ISECTION
IS_I300X150X8
0.3 0.008 0.3 0.15 0.013 0.15 0.013 0.0024 0.0026 2.69e-07
TABLE 4
UNIT METER KN
CHANNEL
CH_C380X100X10.5
0.006939 0.38 0.0105 0.1 0.016 0.000145 5.35e-06 3.99e-07 0.0241 0.00399 -
0.00213333
TABLE 5
UNIT METER KN
ANGLE
AG_L175X175X15
0.175 0.175 0.015 0.0342 0.00175 0.00175
TABLE 6
UNIT METER KN
TUBE
TB_TUB1501506
0.003363 0.15 0.15 0.006 1.15e-05 1.15e-05 1.79e-05 0.0018 0.0018
TB_RHS200X100X9
0.004867 0.2 0.1 0.009 2.35e-05 7.82e-06 1.93e-05 0.0036 0.0018
TABLE 7
UNIT METER KN
PIPE
PP_PIP165.2X7.1
0.1652 0.151 0.001763 0.001763
TABLE 8
UNIT METER KN
DOUBLE ANGLE
DA_L100X100X13LD
0.1 0.1 0.013 0 4.48744e-06 8.79409e-06 2.73893e-07 0.0294 0.00173333 -
0.00173333 1.94
TABLE 9
UNIT METER KN
TEE
TE_CT100X200X8X12
0.003177 0.1 0.2 0.012 0.008 1.84e-06 8.01e-06 1.3e-07 0.0173 0.000533333 -
0.0016
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
```

```
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY JAPANESE
64 UPTABLE 9 TE_CT100X200X8X12
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
127 PINNED
128 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
64 CON GY -10
JOINT LOAD
127 MZ 2
128 MZ -2
PERFORM ANALYSIS
UNIT MMS KN
PARAMETER 1
CODE JAPANESE 2005
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (AIJ-2005)   V2.1
                        ***********************
ALL UNITS ARE IN - MMS   KN   (U.N.O.)
|-----------------------------------------------------------------------------
|
|   Member No:      64        Profile  :
TE_CT100X200X8(UPT)                     |
|  Ratio:    0.867  (PASS)  Reference: Eq.5.24           Loadcase:       1
|
|-----------------------------------------------------------------------------
|
|  Location:  999.999  Criteria:     Stress      Load Case:     1
(Permanent)|
|  Px:(C)        0.000       Vy:         5.000           Vz:       0.000
|
|  Tx:       0.000E+00       My:     0.000E+00           Mz:  -3.000E+03
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES AT DISTANCE:  999.999 MM              (UNIT: MM)
|
|  Ax:     3.17700E+03       Az: 1.60000E+03            Ay: 5.33333E+02
|
|  Iz:     1.84000E+06       Iy: 8.01000E+06             J: 1.30000E+05
|
|  Zz:     2.22491E+04       Zy: 8.01000E+04            Zx: 1.08333E+04
|
```

```
|  iZ:     2.40658E+01       iY: 5.02120E+01            Iw: 1.07813E+08
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES   (Unit: N/mm2)
|
|  Fy:         235.000      E :   205000.000          G :     79000.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES
|
|  Member Length:  1999.999      Lz:  1999.999 Ly:  1999.999  UNL:   1999.999
|
| DESIGN PARAMETERS
|
|  Kz:   1.000     Ky:   1.000   NSF:  1.000   Cb:   0.000    pλb:  0.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS (Tension)
|
|  Actual :     83.105      Allowable :    400.000      Ratio :  0.208
|
|-----------------------------------------------------------------------------
|
| Section Classification Results :    Width to Thickness Ratios
|
|-----------------------------------------------------------------------------
|
|           Compression              |             Bending
|
|-----------------------------------------------------------------------------
|
|  LC#   |    Actual    |   Allowable  |   LC#   |    Actual    |   Allowable
|
|-----------------------------------------------------------------------------
|
|   -    |      -       |      -       |     1   |    11.0000   |     15.6538
|
|-----------------------------------------------------------------------------
|
| CHECKS                             |Env|     Stresses    |
|
|              | Loc. | Demand |  L/C |Typ| Actual | Allow  |Ratio |   Ref.
|
|              |(MET) |(KN-MET)|      |   |(N/mm2) |(N/mm2) |      |
|
|--------------|------|--------|------|---|--------|--------|------|----------
|
| Tension      |  -   |   -    |  -   | - |   -    |  156.67|  -   | Eq.5.1
|
| Compression  |  -   |   -    |  -   | - |   -    |  104.21|  -   | Eq.5.3
|
| Bending Z (T)| 1.000|   -3.00|     1| P |  134.84|  156.67| 0.861| Eq.5.1
|
| Bending Z (C)| 0.000|    2.00|     1| P |   89.89|  156.67| 0.574| Eq.5.1
|
```

```
| Bending Y (T)|   -   |   -    |  -   | - |    -    |  156.67|   -   | Eq.5.1
|
| Bending Y (C)|   -   |   -    |  -   | - |    -    |  149.04|   -   | Eq.5.7
|
| Shear Z      |   -   |   -    |  -   | - |    -    |   90.45|   -   | Eq.5.2
|
| Shear Y      | 0.000|    5.00|     1| P |    9.38|   90.45| 0.104| Eq.5.2
|
| Comp+ Bend C | 0.000|   -    |     1| P |    -    |    -   | 0.574| Eq.6.1
|
| Comp+ Bend T | 1.000|   -    |     1| P |    -    |    -   | 0.861| Eq.6.2
|
| Ten + Bend T | 1.000|   -    |     1| P |    -    |    -   | 0.861| Eq.6.3
|
| Ten + Bend C | 0.000|   -    |     1| P |    -    |    -   | 0.574| Eq.6.4
|
| Von-Mises    | 1.000|   -    |     1| P |  135.81|  156.67| 0.867| Eq.5.24
|
|
|
| INTERMEDIATE RESULTS (Bending)
|
|  Me: 9.8026E+01 KN-MET  My: 1.8824E+01 KN-MET   pλb:   0.600  C:  1.000
|
|
|
| INTERMEDIATE RESULTS (Von-Mises)
|
|  Sigmax:   134.837 N/mm2   Tou:     9.375 N/mm2  fm:   135.811 N/mm2
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D9.B.2 Member Capacities* (on page 1720)

V. New Zealand

V.NZS3404 1997-Angle section compact

Verify the design capacity of compact angle section as per NZ3404 1997.

Details

The member is an A150X150X19 section used in a 5 m cantilever member. The cantilever is loaded with a 2 kN point load at the free end. Steel grade is 280 MPa.

Validation

<u>Section Classification</u>

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{150}{19}\sqrt{\frac{280}{250}} = 7.297$$

Section classification is compact.

<u>Section Bending Capacity About Z-Axis</u>

Effective Section Modulus, $Z_{ez} = 110{,}960\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$M_{sz} = 280\times 110{,}960/10^6 = 31.07$ kN·m

$\phi M_{sz} = 0.9\times 31.07 = 27.962$ kN·m

<u>Section Bending Capacity About Y-Axis</u>

Effective Section Modulus, $Z_{ey} = 248{,}967\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$M_{sy} = 280\times 248{,}967/10^6 = 69.71$ kN·m

$\phi M_{sy} = 0.9\times 69.71 = 62.74$ kN·m

<u>Member Bending Capacity</u>

End restraint arrangement = FU

A twist restraint factor, $K_t = 1.0$

Minor axis rotation restraints = Both

Lateral rotation restraint factor, $K_r = 1.0$

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×0.7×9,000 = 6,300 mm

$$\alpha_m = \frac{1.7M_m^*}{\sqrt{\left(M_2^*\right)^2 + \left(M_3^*\right)^2 + \left(M_4^*\right)^2}} \leq 2.5$$

$\alpha_m = 1.25$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]} = 137.62\ kN \cdot m$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sy}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sy}}{M_{oa}}\right)\right\} = 0.780$$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sy} \leq M_{sy}$

$M_{bz} = 1.25 \times 0.780 \times 69.71 = 67.98$ kN·m $\leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9\times 67.98 = 61.17$ kN·m

<u>Check for Shear</u>

Shear Area of the section, $A_y = d\times t_w = 150\times 19 = 2{,}850\ mm^2$

Section Shear Capacity (Along Y axis), $V_y = 0.6\times f_y\times A_y = 0.6\times 280\times 2{,}850 = 478\ kN$

$V_{vn} = 2\times 478/(0.9 + 1.2) = 456$ kN [Ref : Clause 5.11.2]

$\phi V_y = 0.9\times 456 = 410.4$ kN

Shear Area of the section, $A_Z = b_f\times t_f = 150\times 19 = 2{,}850\ mm^2$

Section Shear Capacity (Along z axis), $V_z = 0.6 \times f_y \times A_z = 0.6 \times 280 \times 2{,}850 = 478\ kN$

$V_{vn} = 2 \times 478/(0.9 + 1.2) = 456$ kN

$\phi V_z = 456 \times 0.9 = 410.4$ kN

Check for Axial Compression

Section Compression Capacity:

Gross Area, $A_g = 5{,}360\ mm^2$

Net Area, $A_n = 5{,}360\ mm^2$

Form factor, $K_f = A_e / A_g = 1$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1 \times 5{,}360 \times 280 = 1500.8$ kN

$\phi N_s = 0.9 \times 1500.8 = 1{,}350.7$ kN [Ref : Clause 6.2.1]

Member Compression Capacity

Length of the member, $L = 5{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.00$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.00$

Effective Length of member, $L_{ez} = 5{,}000\ mm$

Effective Length of member, $L_{ey} = 5{,}000\ mm$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 5{,}000\ /\ 29.3 = 170.648$

Geometrical Slenderness Ratio = $L_{ey}/r_y = 5{,}000\ /\ 57.362 = 87.166$

Member slenderness,

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 170.648\sqrt{1}\sqrt{\frac{280}{250}} = 180.097$$ [Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 87.166\sqrt{1}\sqrt{\frac{280}{250}} = 92.248$$ [Ref : Clause 6.3.3]

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 10.999$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 18.077$

$\alpha_b = 0.5$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az} \times \alpha_z = 186.097$

$\lambda_y = \lambda_{ny} + \alpha_{ay} \times \alpha_b = 101.286$

$\eta = 0.563$

$\eta = 0.286$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2 \times (\lambda_z/90)^2) = 0.683$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2 \times (\lambda_y/90)^2) = 1.008$

$\alpha_{cz} = 0.201$

$\alpha_{cy} = 0.532$

The nominal member capacity,

$N_{cz}= \alpha_{cz}\times N_s = 0.201\times 1{,}500.8 = 301.388$ kN [Ref : Clause 6.3.3]

$\phi N_{cz} = 271.3$ kN

The nominal member capacity,

$N_{cy}= \alpha_{cy}\times N_s = 0.532\times 1{,}500.8 = 798.946$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = 719.1$ kN

Nominal Section tension Capacity

[Ref : Clause 7.1]

$K_{te} = 1.00$

$N_{t1} = A_g\times f_y = 1{,}500.8$ kN

$N_{t2} = 0.85\times K_{te}\times A_n\times f_u = 2{,}004.6$ kN

$\phi N_t = 1{,}350.7$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 690: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	27.962	27.9620	none	
ϕM_{sy} (kN·m)	62.74	62.7397	negligible	
ϕM_{bz} (kN·m)	61.17	61.0794	negligible	
ϕV_y (kN)	410.4	410.4	none	
ϕV_z (kN)	410.4	410.4	none	
ϕN_s (kN)	1,350.7	1,350.7	none	
ϕN_{cz} (kN)	271.3	271.2	negligible	
ϕN_{cy} (kN)	719.1	719.1	none	
ϕN_t (kN)	1,350.7	1,350.7	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Angle section compact.std` is typically installed with the program.

The following D11.A.6 Design Parameters (on page 1776) are used:

- The twist restraint factor, K_t = 1 is specified by `SKT 1.0` (default value; K_t is calculated)

- The lateral rotation restraint factor, K_r = 1.0 is specified by `SKR 1.0` (default value; K_r is calculated)

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Angle_Section_Compact.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  EQUAL ANGLE SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 13-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
*
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST A150X150X19
*
CONSTANTS
BETA 45 ALL
MATERIAL STEEL ALL
*
SUPPORTS
1 FIXED
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FZ 2
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
TRACK 2 PMEMB 1
IST 2 PMEMB 1
SGR 0 PMEMB 1
```

```
PBRACE TOP 0 FR 1 U PMEMB 1
PBRACE BOTTOM 0 FR 1 U PMEMB 1
DUCT 2 PMEMB 1
GLD 1 PMEMB 1
CHECK CODE PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ST ANGLE SECTION:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             X/x          Major principal axis of section
      Z/z             Y/y          Minor Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   A150X150X19              (AISC SECTIONS)
   Governing Load Case:     1*
   Governing
Criteria:
   Governing Ratio:   0.811  (PASS)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       150.0000 mm    b:       150.0000 mm
    t:        19.0000 mm
   Ag:      5360.0000 mm2   J:   644.9868E+03 mm4             Iw:     1.0569E
+09 mm6
   Iz:     4.6015E+06 mm4  Sz:   134.6263E+03 mm3 (plastic)   Zz:   165.9782E
+03 mm3 (elastic)
   rz:    29.3000E+00 mm
   Iy:    17.6365E+06 mm4  Sy:   265.6150E+03 mm3 (plastic)   Zy:    73.9736E
+03 mm3 (elastic)
   ry:    57.3620E+00 mm
      STAAD SPACE                                             -- PAGE NO.
4
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
```

```
   G            :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    280.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    280.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     170.648  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.253
   Critical Location  :   0.000 m from Start.
   Mz* =    -7.0711E+00 KNm
   Section Slenderness: Compact
   Zez =   110.9604E+03 mm3
   φMsz =    27.9620E+00 KNm               [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.113
   Critical Location  :   0.000 m from Start.
   My* =    -7.0711E+00 KNm
   Section Slenderness: Compact
   Zey =   248.9673E+03 mm3
   φMsy =    62.7397E+00 KNm               [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.116
   Critical Location  :   0.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(FR)-  5.00 m(U )
   Mz* =     7.0711E+00 KNm
   kt   =      1.00                        [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                        [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                        [NZS3404 Table 5.6.3(3)]
   le   =      5.00 m                      [NZS3404 5.6.3]
   αm   =     1.250                        [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =   137.6171E+00 KNm               [NZS3404 5.6.1.1.1(d)]
   αsy  =     0.779                        [NZS3404 5.6.1.1.1(c)]
   φMby =    61.0794E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD SPACE                                             -- PAGE NO.
5
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.003
   Critical Location  :   0.000 m from Start.
   Vy*   =     1.4142E+00 KN
   φVvmy =   410.4000E+00 KN               [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.003
   Critical Location  :   0.000 m from Start.
   Vz*   =     1.4142E+00 KN
   φVvmz =   410.4000E+00 KN               [NZS3404 5.12.2]
      STAAD SPACE                                             -- PAGE NO.
```

```
6
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =     5.3600E+03 mm2              [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                       [AS 4100 6.2.2]
   An    =     5.3600E+03 mm2
   φNs   =     1.3507E+03 KN               [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  5.00 m(U )
   Lez   =      5.00 m
   αb    =      0.50                       [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =   180.597                       [NZS3404 6.3.3]
   λ,z   =   186.097                       [NZS3404 6.3.3]
   ε,z   =     0.683                       [NZS3404 6.3.3]
   αc,z  =     0.201                       [NZS3404 6.3.3]
   φNcz  = 0.2712E+3 KN                    [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  5.00 m(U )
   Ley   =      5.00 m
   λn,y  =    92.248                       [NZS3404 6.3.3]
   λ,y   =   101.286                       [NZS3404 6.3.3]
   ε,y   =     1.008                       [NZS3404 6.3.3]
   αc,y  =     0.532                       [NZS3404 6.3.3]
   φNcy  = 0.7191E+3 KN                    [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =     5.3600E+03 mm2
   φNt   =     1.3507E+03 KN               [NZS3404 7.2]
      STAAD SPACE                                                  -- PAGE NO.
7
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.253
   Critical Location  :   0.000 m from Start.
   φMrz  =    27.9620E+00 KNm              [NZS3404 8.3.2]
```

```
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.113
   Critical Location  :   0.000 m from Start.
   φMry  =    62.7397E+00 KNm              [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.366
   Critical Location  :   0.000 m from Start.
   γ     =     1.400                        [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.253
   Critical Location  :   0.000 m from Start.
   φMiz  =    27.9620E+00 KNm              [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.113
   Critical Location  :   0.000 m from Start.
   φMiy  =    62.7397E+00 KNm              [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoy  =     0.0000E+00 KN               [NZS3404 8.4.4.1.2]
   φMoy,t=     0.0000E+00 KNm              [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoy,c=     0.0000E+00 KNm              [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
      STAAD SPACE                                             -- PAGE NO.
8
    *
   SEISMIC PROVISIONS
   ------------------
   Section Slenderness (Bending about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.811
   Critical Location  :   5.000 m from Start.
   λsz             =      7.30             [NZS3404 12.5.1.1]
   λez             =      9.00             [NZS3404 Table 12.5]
   Section Slenderness (Bending about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.811
   Critical Location  :   0.000 m from Start.
   λsy             =      7.30             [NZS3404 12.5.1.1]
   λey             =      9.00             [NZS3404 Table 12.5]
   Max Specific Yield Stress
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.778
   Critical Location  :   0.000 m from Start.
   Fy,actual        =    280.00
   Fy,limit         =    360.00             [NZS3404 Table 12.4(1)]
   Max Actual Yield Ratio (Fy/Fu)
   Critical Load Case :     1*
   Critical Ratio     :   0.795
   Critical Location  :   0.000 m from Start.
   Fy/Fu,actual     =      0.64
   Fy/Fu,limit      =      0.80             [NZS3404 Table 12.4(3)]
   Fabrication Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.4.1.2]
   Section Symmetry Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.5.2]
   Min Web Thickness Requirement for Beam
   Critical Load Case :     1*
   Critical Ratio     :   0.089
   Critical Location  :   0.000 m from Start.
   tw,actual        =     19.00
   tw,min           =      1.69             [NZS3404 12.7.2]
   Max Axial Force Limit for Column (a)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.70             [NZS3404 Table 12.8.1]
   Max Axial Force Limit for Column (b)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
    b m               =      0.00
   NoL              =   372.4033E+00 KN
   λEYC             =      2.01
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.05             [NZS3404 12.8.3.1(b)]
   Max Axial Force Limit for Column (c)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Ng*/φNs - actual =      0.00
   Ng*/φNs - limit  =      1.00             [NZS3404 12.8.3.1(c)]
   Shear-Y + Bend-Z Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.253
   Critical Location  :   0.000 m from Start.
   Mz*  =    -7.0711E+00 KN
   φMsvz=    27.9620E+00 KN                 [NZS3404 12.10.3.1]
   Shear-Z + Bend-Y Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.113
   Critical Location  :   0.000 m from Start.
```

```
   My*  =     7.0711E+00 KN
   φMsvy=    62.7397E+00 KN                 [NZS3404 12.10.3.1]

*******************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-Angle section Non compact

Verify the design capacity of equal leg, non-compact angle section as per NZ3404 1997.

Details

The member is an A150X150X12 section used in a 5 m cantilever member. The cantilever is loaded with a 2 kN point load at the mid-point. Steel grade is 300 MPa.

Validation

Section Classification

Evaluate the slenderness effects of the beam flanges:

$$\lambda_{ef} = \frac{B}{t_f}\sqrt{\frac{f_y}{250}} = \frac{138}{12}\sqrt{\frac{300}{250}} = 12.60$$

Section flange classification is non-compact.

Evaluate the slenderness effects of the beam web:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{138}{12}\sqrt{\frac{300}{250}} = 12.60$$

Section web classification is non-compact

Section Bending Capacity About Z-Axis

Effective Section Modulus, $Z_{ez} = 72{,}560\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 300\times 72{,}560/10^6 = 21.77\ kN\cdot m$$

$$\phi M_{sz} = 0.9\times 21.77 = 19.59\ kN\cdot m$$

Section Bending Capacity About Y-Axis

Effective Section Modulus, $Z_{ey} = 155{,}980\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 300\times 155{,}980/10^6 = 46.79\ kN\cdot m$$

$$\phi M_{sy} = 0.9\times 46.79 = 42.11\ kN\cdot m$$

Member Bending Capacity

End restraint arrangement = FU

A twist restraint factor, $K_t = 1.00$

Lateral rotation restraint factor, K_r = 1.0

Load Height factor, Kl, = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1×5,000 = 5,000 mm

$$\alpha_m = \frac{1.7M_m^*}{\sqrt{(M_2^*)^2 + (M_3^*)^2 + (M_4^*)^2}} \leq 2.5$$

$\alpha_m = 1.25$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]} = 57.42\ kN \cdot m$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sy}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sy}}{M_{oa}}\right)\right\} = 0.660$$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sy} \leq M_{sy}$

$M_{bz} = 1.25 \times 0.660 \times 46.79 = 38.61\ kN \cdot m \leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 38.61 = 34.74\ kN \cdot m$

<u>Check for Shear</u>

Shear Area of the section, $A_y = d \times t_w = 150 \times 12 = 1{,}800\ mm^2$

Section Shear Capacity (Along Y axis), $V_y = 0.6 \times f_y \times A_y = 0.6 \times 300 \times 1{,}800 = 324\ kN$

$V_{vn} = 2 \times 324/(0.9 + 1.2) = 308.6\ kN$ [Ref : Clause 5.11.2]

$\phi V_y = 0.9 \times 308.6 = 277.7\ kN$

Shear Area of the section, $A_Z = b_f \times t_f = 150 \times 12 = 1{,}800\ mm^2$

Section Shear Capacity (Along z axis), $V_z = 0.6 \times f_y \times A_z = 0.6 \times 300 \times 1{,}800 = 324\ kN$

$V_{vn} = 2 \times 324/(0.9 + 1.2) = 308.6\ kN$

$\phi V_z = 0.9 \times 308.6 = 277.7\ kN$

<u>Check for Axial Compression</u>

Section Compression Capacity:

Gross Area, $A_g = 3{,}480\ mm^2$

Net Area, $A_n = 3{,}480\ mm^2$

Form factor, $K_f = A_e/A_g = 1$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1 \times 3{,}480 \times 300 = 1{,}044\ kN$ [Ref : Clause 6.2.1]

$\phi N_s = 0.9 \times 1{,}044 = 939.6\ kN$

<u>Member Compression Capacity</u>

Length of the member, $L = 5{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 2.2$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 2.2$

Effective Length of member, $L_{ez} = 2.2 \times 5{,}000\ mm = 11{,}000\ mm$

Effective Length of member, $L_{ey} = 2.2 \times 5{,}000\ mm = 11{,}000\ mm$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 11{,}000\ /\ 29.6 = 371.62$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 11{,}000\ /\ 58.609 = 187.7$

Member slenderness,

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 371.62\sqrt{1}\sqrt{\frac{300}{250}} = 407.1$$ [Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 187.7\sqrt{1}\sqrt{\frac{300}{250}} = 205.6$$ [Ref : Clause 6.3.3]

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 5.116$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 9.797$

$\alpha_b = 0.5$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az} \times \alpha_z = 409.65$

$\lambda_y = \lambda_{ny} + \alpha_{ay} \times \alpha_b = 210.50$

$\eta = 1.29$

$\eta = 0.64$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2 \times (\lambda_z/90)^2) = 0.56$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2 \times (\lambda_y/90)^2) = 0.65$

$\alpha_{cz} = 0.045$

$\alpha_{cy} = 0.160$

The nominal member capacity,

$N_{cz} = \alpha_{cz} \times N_s = 0.045 \times 1{,}044 = 46.98$ kN [Ref : Clause 6.3.3]

$\phi N_{cz} = 42.28$ kN

The nominal member capacity,

$N_{cy} = \alpha_{cy} \times N_s = 0.160 \times 1{,}044 = 167.0$ kN

$\phi N_{cy} = 150.3$ kN

Nominal Section tension Capacity *[Ref : Clause 7.1]*

$K_{te} = 1.00$

$N_{t1} = A_g \times f_y = 1{,}044$ kN

$N_{t2} = 0.85 \times K_{te} \times A_n \times f_u = 1{,}301.5$ kN

$\phi N_t = 0.9 \times 1{,}044 = 939.6$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 691: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	19.59	19.5911	negligible	
ϕM_{sy} (kN·m)	42.11	42.1144	negligible	
ϕM_{bz} (kN·m)	34.74	34.6143	negligible	
ϕV_y (kN)	277.7	277.7143	negligible	
ϕV_z (kN)	277.7	277.7143	negligible	
ϕN_s (kN)	939.6	939.6	none	
ϕN_{cz} (kN)	42.28	42.57	negligible	
ϕN_{cy} (kN)	150.3	150.7	negligible	
ϕN_t (kN)	939.6	939.6	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Angle section Non compact.std` is typically installed with the program.

The following D11.A.6 Design Parameters (on page 1776) are used:

- The twist restraint factor, K_t = 1 is specified by `SKT 1.0` (default value; K_t is calculated)
- The lateral rotation restraint factor, K_r = 1.0 is specified by `SKR 1.0` (default value; K_r is calculated)

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Angle_Section_Non_Compact.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  EQUAL ANGLE SHAPE  PER
*             THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 13-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
*
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST A150X150X12
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 FIXED
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FZ 2
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
TRACK 2 PMEMB 1
PBRACE TOP 0 U 1 F PMEMB 1
PBRACE BOTTOM 0 U 1 F PMEMB 1
PBCRES ZZ 0 T 1 U PMEMB 1
PBCRES YY 0 T 1 U PMEMB 1
DUCT 1 PMEMB 1
GLD 1 PMEMB 1
CHECK CODE PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                      **************************************************
   AXIS NOTATION FOR ST ANGLE SECTION:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             X/x          Major principal axis of section
      Z/z             Y/y          Minor Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
```

```
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   A150X150X12              (AISC SECTIONS)
   Governing Load Case:     1*
   Governing
Criteria:
   Governing Ratio:   1.400 *(FAIL)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       150.0000 mm    b:       150.0000 mm
    t:        12.0000 mm
   Ag:      3480.0000 mm2   J:   167.0400E+03 mm4             Iw:   286.6545E
+06 mm6
   Iz:     3.0490E+06 mm4  Sz:    88.4845E+03 mm3 (plastic)   Zz:   112.4115E
+03 mm3 (elastic)
   rz:    29.6000E+00 mm
   Iy:    11.9540E+06 mm4  Sy:   176.0526E+03 mm3 (plastic)   Zy:    52.2925E
+03 mm3 (elastic)
   ry:    58.6094E+00 mm
      STAAD SPACE                                               -- PAGE NO.
4
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    300.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    300.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     168.919  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Mz* =     0.0000E+00 KNm
   Section Slenderness: Noncompact
   Zez =    72.5597E+03 mm3
   φMsz =    19.5911E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.237
   Critical Location  :   0.000 m from Start.
   My* =   -10.0000E+00 KNm
   Section Slenderness: Noncompact
```

```
   Zey =   155.9794E+03 mm3
   φMsy =    42.1144E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.289
   Critical Location  :   0.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(U )-  5.00 m(F )
   Mz* =    10.0000E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =      5.00 m                       [NZS3404 5.6.3]
   αm   =     1.250                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =    57.0084E+00 KNm                [NZS3404 5.6.1.1.1(d)]
   αsy  =     0.658                         [NZS3404 5.6.1.1.1(c)]
   φMby =    34.6143E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD SPACE                                              -- PAGE NO.
5
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vy*   =     0.0000E+00 KN
   φVvmy =   277.7143E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   Vz*   =     2.0000E+00 KN
   φVvmz =   277.7143E+00 KN                [NZS3404 5.12.2]
      STAAD SPACE                                              -- PAGE NO.
6
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =     3.4800E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =     3.4800E+03 mm2
   φNs   =   939.6000E+00 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(T )-  5.00 m(U )
   Lez   =     11.00 m
   αb    =      0.50                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =   407.091                        [NZS3404 6.3.3]
```

```
   λ,z   =   409.649                        [NZS3404 6.3.3]
   ε,z   =     0.555                        [NZS3404 6.3.3]
   αc,z  =     0.045                        [NZS3404 6.3.3]
   φNcz  = 0.4257E+2 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(T )-  5.00 m(U )
   Ley   =     11.00 m
   λn,y  =   205.597                        [NZS3404 6.3.3]
   λ,y   =   210.495                        [NZS3404 6.3.3]
   ε,y   =     0.650                        [NZS3404 6.3.3]
   αc,y  =     0.160                        [NZS3404 6.3.3]
   φNcy  = 0.1507E+3 KN                     [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =     3.4800E+03 mm2
   φNt   =   939.6000E+00 KN               [NZS3404 7.2]
      STAAD SPACE                                              -- PAGE NO.
7
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMrz  =    19.5911E+00 KNm               [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.237
   Critical Location  :   0.000 m from Start.
   φMry  =    42.1144E+00 KNm               [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.237
   Critical Location  :   0.000 m from Start.
   γ     =     1.400                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiz  =    19.5911E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.237
   Critical Location  :   0.000 m from Start.
   φMiy  =    42.1144E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
```

```
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoy  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoy,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoy,c=     0.0000E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
      STAAD SPACE                                                   -- PAGE NO.
8
    *
   SEISMIC PROVISIONS
   ------------------
   Section Slenderness (Bending about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.400
   Critical Location  :   0.000 m from Start.
   λsz              =     12.60             [NZS3404 12.5.1.1]
   λez              =      9.00             [NZS3404 Table 12.5]
   Section Slenderness (Bending about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.400
   Critical Location  :   0.000 m from Start.
   λsy              =     12.60             [NZS3404 12.5.1.1]
   λey              =      9.00             [NZS3404 Table 12.5]
   Max Specific Yield Stress
   Critical Load Case :     1*
   Critical Ratio     :   0.833
   Critical Location  :   0.000 m from Start.
   Fy,actual        =    300.00
   Fy,limit         =    360.00             [NZS3404 Table 12.4(1)]
   Max Actual Yield Ratio (Fy/Fu)
   Critical Load Case :     1*
   Critical Ratio     :   0.852
   Critical Location  :   0.000 m from Start.
   Fy/Fu,actual     =      0.68
   Fy/Fu,limit      =      0.80             [NZS3404 Table 12.4(3)]
   Fabrication Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.4.1.2]
   Section Symmetry Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.5.2]
   Min Web Thickness Requirement for Beam
   Critical Load Case :     1*
```

```
   Critical Ratio     :   0.154
   Critical Location  :   0.000 m from Start.
   tw,actual        =     12.00
   tw,min           =      1.84              [NZS3404 12.7.2]
   Max Axial Force Limit for Column (a)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.50              [NZS3404 Table 12.8.1]
   Max Axial Force Limit for Column (b)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
    b m               =      0.50
   NoL              =   967.4472E+00 KN
   λEYC             =      1.04
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.32              [NZS3404 12.8.3.1(b)]
   Max Axial Force Limit for Column (c)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Ng*/φNs - actual =      0.00
   Ng*/φNs - limit  =      1.00              [NZS3404 12.8.3.1(c)]
   Shear-Y + Bend-Z Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Mz*  =     0.0000E+00 KN
   φMsvz=    19.5911E+00 KN                  [NZS3404 12.10.3.1]
   Shear-Z + Bend-Y Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.237
   Critical Location  :   0.000 m from Start.
   My*  =    10.0000E+00 KN
   φMsvy=    42.1144E+00 KN                  [NZS3404 12.10.3.1]

******************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-Channel Section

Verify the design capacity of a channel section per NZS 3404 1997.

Details

The member is a PFC250 section used in a 9 m simply supported span. The beam is loaded with a vertical load of 8.4 kN point load at 5m from start end, a 10 kN axial load at the mid-point, and a distributed load that decreases from 0.5 kN/m at the start down to 0.2 kN/m at the end. Steel grade is 300 MPa.

Validation

Section Classification

Evaluate the slenderness effects of the beam flanges:

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{90}{15}\sqrt{\frac{300}{250}} = 5.988$$

Section flange classification is compact.

Evaluate the slenderness effects of the beam web:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{220}{8}\sqrt{\frac{300}{250}} = 30.124 < 89$$

Section web classification is compact

Section Bending Capacity About Z-Axis

Effective Section Modulus, $Z_{ez} = 421{,}000\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 300\times 421{,}000/10^6 = 126.3\ kN{\cdot}m$$

$$\phi M_{sz} = 0.9\times 126.3 = 113.67\ kN{\cdot}m$$

Section Bending Capacity About Y-Axis

Effective Section Modulus, $Z_{ey} = 88{,}925\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 300\times 88{,}925/10^6 = 26.68\ kN{\cdot}m$$

$$\phi M_{sy} = 0.9\times 26.68 = 24.01\ kN{\cdot}m$$

Member Bending Capacity

End restraint arrangement = FF

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 0.70

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1×9,000 = 9,000 mm

$$\alpha_m = \frac{1.7M_m^*}{\sqrt{\left(M_2^*\right)^2 + \left(M_3^*\right)^2 + \left(M_4^*\right)^2}} = \frac{1.7 \times 22.416}{\sqrt{(-12.087)^2 + (-22.416)^2 + (-12.087)^2}} = 1.352 \leq 2.25$$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]} = 42.25\ kN{\cdot}m$$

[Ref : Clause 5.6.1.1 (c)]

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\} = 0.281$$

$M_{bx} = \alpha_m \alpha_s M_{sx} \leq M_{sx}$

$M_{bz} = 1.352 \times 0.281 \times 126.3 = 48.0$ kN·m $\leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 48.0 = 43.19$ kN·m

Check for Shear

Shear Area of the section, $A_y = d \times t_w = 250 \times 8 = 2{,}000\ mm^2$

Section Shear Capacity (Along Y axis), $V_y = 0.6 \times f_y \times A_y = 0.6 \times 300 \times 2{,}000 = 384\ kN$

$V_{vn} = 2 \times 324/(0.9 + 1.2) = 308.6$ kN [Ref : Clause 5.11.2]

$\phi V_y = 0.9 \times 384 = 345.6$ kN

Shear Area of the section, $A_Z = 2 \times b_f \times t_f = 2 \times 90 \times 15 = 2{,}700\ mm^2$

Section Shear Capacity (Along z axis), $V_z = 0.6 \times f_y \times A_z = 0.6 \times 300 \times 2{,}700 = 486\ kN$

$\phi V_z = 0.9 \times 486 = 437.4$ kN

Check for Axial Compression

Section Compression Capacity:

The flange slenderness, $\lambda_{eb} = 5.9884$ [Ref : Cl - 6.2.3.1]

Yield slender for flange, $\lambda eby = 14$ [Ref : Table 6.2.4]

The web slenderness, $\lambda_{ew} = 30.125$

Gross Area, $A_g = 4{,}520\ mm^2$

Net Area, $A_n = 4{,}520\ mm^2$

Form factor, $K_f = A_e / A_g = 1$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1 \times 4{,}520 \times 300 = 1{,}356$ kN [Ref : Clause 6.2.1]

$\phi N_s = 0.9 \times 1{,}356 = 1{,}220.4$ kN

Member Compression Capacity

Length of the member, $L = 9{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.0$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.0$

Effective Length of member, $L_{ez} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

Effective Length of member, $L_{ey} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 9{,}000\ /\ 99.89 = 90.10$

Geometrical Slenderness Ratio = $L_{ey}/r_y = 9{,}000\ /\ 28.37 = 317.2$

Member slenderness,

[Ref : Clause 6.3.3]

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 90.10\sqrt{1}\sqrt{\frac{300}{250}} = 98.70$$

[Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 317.2\sqrt{1}\sqrt{\frac{300}{250}} = 347.5$$

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 17.402$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 5.970$

$\alpha_b = 0.5$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az} \times \alpha_z = 107.4$

$\lambda_y = \lambda_{ny} + \alpha_{ay} \times \alpha_b = 350.5$

$\eta = 0.31$

$\eta = 1.10$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2 \times (\lambda_z/90)^2) = 0.96$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2 \times (\lambda_y/90)^2) = 0.57$

$\alpha_{cz} = 0.493$

$\alpha_{cy} = 0.061$

The nominal member capacity,

$N_{cz} = \alpha_{cz} \times N_s = 0.493 \times 1{,}356 = 668.7$ kN [Ref : Clause 6.3.3]

$\phi N_{cz} = 601.8$ kN

The nominal member capacity,

$N_{cy} = \alpha_{cy} \times N_s = 0.061 \times 1{,}356 = 82.72$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = 74.44$ kN

Nominal Section tension Capacity

[Ref : Clause 7.1]

$K_{te} = 1.00$

$N_{t1} = A_g \times f_y = 1{,}356$ kN

$N_{t2} = 0.85 \times K_{te} \times A_n \times f_u = 1{,}690.5$ kN

$\phi N_t = 0.9 \times 1{,}356 = 1{,}220.4$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 692: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	113.67	113.67	none	
ϕM_{sy} (kN·m)	24.01	24.0098	negligible	
ϕM_{bz} (kN·m)	43.19	42.9925	negligible	
ϕV_{vy} (kN)	345.6	345.600	none	
ϕN_s (kN)	1,220.4	1,220.4	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕN_{cz} (kN)	601.8	601.8	none	
ϕN_{cy} (kN)	74.44	74.74	negligible	
ϕN_t (kN)	1,220.4	1,220.4	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Channel Section.std` is typically installed with the program.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Channel_Section_Compact.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF CHANNEL SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 03-Jan-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 4 9 0 0;
*
MEMBER INCIDENCES
1 1 4;
DEFINE PMEMBER
1 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST PFC250
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
```

```
1 PINNED
4 FIXED BUT FX MY MZ
PRINT ALL
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -8.4
SELFWEIGHT Y -1
MEMBER LOAD
1 UNI X -10
1 TRAP Z -0.5 -2
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
TRACK 2 PMEMB 1
CHECK CODE PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   PFC250                   (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
8.4.5.1
   Governing Ratio:  46.122 *(FAIL)
   Governing Location:   2.250 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       250.0000 mm   bf:        90.0000 mm
   tf:        15.0000 mm   tw:         8.0000 mm
   Ag:      4520.0000 mm2   J:   238.0000E+03 mm4             Iw:    34.8250E
+09 mm6
   Iz:    45.1000E+06 mm4  Sz:   421.0000E+03 mm3 (plastic)   Zz:   360.8000E
+03 mm3 (elastic)
   rz:    99.8893E+00 mm
```

```
   Iy:     3.6400E+06 mm4  Sy:   107.0000E+03 mm3 (plastic)   Zy:    59.2834E
+03 mm3 (elastic)
   ry:    28.3780E+00 mm
      STAAD SPACE                                            -- PAGE NO.
10
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    300.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     317.147  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  180.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.205
   Critical Location  :   4.500 m from Start.
   Mz* =   -23.3483E+00 KNm
   Section Slenderness: Compact
   Zez =   421.0000E+03 mm3
   φMsz =   113.6700E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.449
   Critical Location  :   1.500 m from Start.
   My* =    34.7977E+00 KNm
   Section Slenderness: Compact
   Zey =    88.9251E+03 mm3
   φMsy =    24.0098E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.544
   Critical Location  :   4.500 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )-  9.00 m(F )
   Mz* =   -23.3483E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =      9.00 m                       [NZS3404 5.6.3]
   αm   =     1.352                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =    42.2526E+00 KNm                [NZS3404 5.6.1.1.1(d)]
   αsz  =     0.279                         [NZS3404 5.6.1.1.1(c)]
   φMbz =    42.9225E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD SPACE                                            -- PAGE NO.
11
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.017
   Critical Location  :   0.000 m from Start.
   Vy*   =     5.7625E+00 KN
   φVvmy =   345.6000E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.016
   Critical Location  :   0.750 m from Start.
   Vz*   =     4.0781E+00 KN
   φVvmz =   262.4400E+00 KN                [NZS3404 5.12.2]
      STAAD SPACE                                              -- PAGE NO.
12
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.074
   Critical Location  :   0.000 m from Start.
   N*    =    90.0000E+00 KN
   Ae    =     4.5200E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =     4.5200E+03 mm2
   φNs   =     1.2204E+03 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.150
   Critical Location  :   0.000 m from Start.
   N*    =    90.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Lez   =      9.00 m
   αb    =      0.50                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    98.699                        [NZS3404 6.3.3]
   λ,z   =   107.400                        [NZS3404 6.3.3]
   ε,z   =     0.959                        [NZS3404 6.3.3]
   αc,z  =     0.493                        [NZS3404 6.3.3]
   φNcz  = 0.6018E+3 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.204
   Critical Location  :   0.000 m from Start.
   N*    =    90.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Ley   =      9.00 m
   λn,y  =   347.417                        [NZS3404 6.3.3]
   λ,y   =   350.403                        [NZS3404 6.3.3]
   ε,y   =     0.569                        [NZS3404 6.3.3]
   αc,y  =     0.061                        [NZS3404 6.3.3]
   φNcy  = 0.7474E+2 KN                     [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                        [User defined]
```

```
   An    =     4.5200E+03 mm2
   φNt   =     1.2204E+03 KN                [NZS3404 7.2]
      STAAD SPACE                                             -- PAGE NO.
13
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.213
   Critical Location  :   4.500 m from Start.
   φMrz  =   109.4786E+00 KNm               [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.554
   Critical Location  :   0.750 m from Start.
   φMry  =    22.3867E+00 KNm               [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   1.689
   Critical Location  :   1.500 m from Start.
   γ     =     1.461                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.222
   Critical Location  :   4.500 m from Start.
   φMiz  =   105.1701E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :  14.256
   Critical Location  :   2.250 m from Start.
   φMiy  =     2.3263E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   3.092
   Critical Location  :   2.250 m from Start.
   φMoz,c=     4.1587E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :  46.122
   Critical Location  :   2.250 m from Start.

******************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-I section

Verify the Design Capacity of I Section as per NZS3404 1997.

Details

Verify the Section bending capacity of an UB530X92.4. Span of the member = 9 m. Both ends of the member are simply supported.

Loads:

Concentrated in the Y direction: 84.5 kN (at a distance of 5m from the left support)
Concentrated in the Z direction: 10 kN (at mid-span)
Concentrated in the X direction: 10 kN (at mid-span)

Fy = 300 MPa

Validation

<u>Section Classification</u>

Evaluate the slenderness effects of the beam flanges:

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{198.8}{2(15.60)}\sqrt{\frac{300}{250}} = 6.98 < 8$$

Section flange classification is compact.

Evaluate the slenderness effects of the beam web:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{501.8}{10.20}\sqrt{\frac{300}{250}} = 56.89 < 89$$

Section web classification is compact

<u>Section Bending Capacity About Z-Axis</u>

Effective Section Modulus, $Z_{ez} = 2.37 \times 10^6\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 300\times 2.37 = 711 \text{ kN}\cdot\text{m}$$

$$\phi M_{sz} = 0.9\times 700 = 639.9 \text{ kN}\cdot\text{m}$$

<u>Section Bending Capacity About Y-Axis</u>

Effective Section Modulus, $Z_{ey} = 341.6 \times 10^3\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 300\times 341.6\times 10^3/10^6 = 102.5 \text{ kN}\cdot\text{m}$$

$$\phi M_{sy} = 0.9\times 102.5 = 92.23 \text{ kN}\cdot\text{m}$$

<u>Member Bending Capacity</u>

End restraint arrangement = FF

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 0.70

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×0.7×9,000 = 6,300 mm

$$\alpha_m = \frac{1.7M_m^*}{\sqrt{(M_2^*)^2 + (M_3^*)^2 + (M_4^*)^2}} = \frac{1.7 \times 109.8}{\sqrt{(3.383)^2 + (-77.73)^2 + (-7.246)^2}} = 2.389 \leq 2.5$$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]} = 416.5 \text{ kN·m}$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\} = 0.435$$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sx} \leq M_{sx}$

$M_{bz} = 2.389 \times 0.435 \times 416.5 = 711.0 \text{ kN·m} \leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 711.0 = 639.9 \text{ kN·m}$

Check for Shear

Shear Area of the section, $A_y = d \times t_w = 533 \times 10.2 = 5{,}437\ mm^2$

Section Shear Capacity (Along Y axis), $V_y = 0.6 \times f_y \times A_y = 0.6 \times 300 \times 5{,}437 = 1{,}044\ kN$

$\phi V_y = 0.9 \times 1{,}044 = 939.4$ kN

Shear Area of the section, $A_Z = 2 \times b_f \times t_f = 2 \times 209 \times 15.6 = 6{,}521\ mm^2$

Section Shear Capacity (Along z axis), $V_z = 0.6 \times f_y \times A_z = 0.6 \times 300 \times 6{,}521 = 1{,}174\ kN$

$\phi V_z = 0.9 \times 1{,}174 = 1{,}056$ kN

Check for Axial Compression

Section Compression Capacity:

The flange slenderness, $\lambda_{eb} = 6.98$ [Ref : Cl - 6.2.3.1]

Yield slender for flange, $\lambda_{eby} = 16$ [Ref : Table 6.2.4]

The web slenderness, $\lambda_{ew} = 55.66$

Gross Area, $A_g = 11{,}800\ mm^2$

Net Area, $A_n = 11{,}800\ mm^2$

Form factor, $K_f = A_e/A_g = 0.92$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 0.92 \times 11{,}800 \times 300 = 3{,}257$ kN [Ref : Clause 6.2.1]

$\phi N_s = 0.9 \times 3{,}257 = 2{,}931$ kN

Member Compression Capacity

Length of the member, $L = 9{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.0$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.0$

Effective Length of member, $L_{ez} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

Effective Length of member, $L_{ey} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

$r_z = \sqrt{(554 \times 10^6 / 11{,}800)} = 216.7$

$r_y = \sqrt{(23.8 \times 10^6 / 11{,}800)} = 44.91$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 9{,}000\ /\ 216.7 = 41.54$

Geometrical Slenderness Ratio = $L_{ey}/r_y = 9{,}000\ /\ 44.91 = 200.4$

Member slenderness,

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 41.54\sqrt{1}\sqrt{\frac{300}{250}} = 43.57$$ [Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 200.4\sqrt{1}\sqrt{\frac{300}{250}} = 210.1$$ [Ref : Clause 6.3.3]

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 19.24$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 9.60$

$\alpha_b = 0.0$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az} \times \alpha_z = 43.57$

$\lambda_y = \lambda_{ny} + \alpha_{ay} \times \alpha_b = 210.1$

$\eta = 0.10$

$\eta = 0.64$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2 \times (\lambda_z/90)^2) = 2.84$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2 \times (\lambda_y/90)^2) = 0.65$

$\alpha_{cz} = 0.89$

$\alpha_{cy} = 0.16$

The nominal member capacity,

$N_{cz} = \alpha_{cz} \times N_s = 0.89 \times 3{,}246 = 2{,}888$ kN [Ref : Clause 6.3.3]

$\phi N_{cz} = 2{,}600$ kN

The nominal member capacity,

$N_{cy} = \alpha_{cy} \times N_s = 0.16 \times 3{,}246 = 521.9$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = 470$ kN

<u>Combined Bending and Axial</u>

Section Combined Capacity (About Z-axis) ≤ M_{sz}

$M_{rz} = 1.18 \times M_{sz} \times (1 - N/\phi N_{sz}) = 837.54$ kN·m

$M_{rz} = 1.18 \times M_{sz} \times (1 - N/\phi N_{sz}) = 837.54$ kN·m

$\phi M_{rz} = 639.90$ kN·m

Section Combined Capacity (About y-axis) ≤ M_{sy}

$M_{rz} = 1.19 \times M_{sy} \times (1-(N/\phi N_{sy})^2) = 121.96$ kN·m

$\phi M_{ry} = 92.239$ kN·m

Section Combined Capacity (Biaxial) [Ref : Cl -8.1.5]

$\lambda = 1.4+(N/\phi Ns) = 1.402$

$(M_z/\phi M_{rz})\lambda + (M_y/\phi M_{ry})\lambda = 0.14$

Member in-plane Capacity (Z-axis) $M_{iz} \leq M_{rz}$

$M_{iz} = M_{sz}\{[1-(1+ßm/2)3](1-N/\phi N_{cz}) + 1.18(1+ßm/2)3\sqrt{}(1-N/\phi N_{cz})\} = 709.63$ kN·m

$\phi M_{iz} = 638.67$ kN·m

Member in-plane Capacity (y-axis) $M_{iy} \leq M_{ry}$ *[Ref : Cl - 8.4.4.2.2]*

$M_{sy}\{[1-(1+ßm/2)3](1-Ny/\phi N_{cy}) + 1.18(1+ßm/2)3\sqrt{}(1-Ny/\phi N_{cy})\} = 101.40$ kN·m

$\phi M_{iy} = 91.26$ kN·m

Member Out- of- plane Capacity (Tension) *[Ref : Cl - 8.4.4.2]*

$M_{ozt} = M_{bz}(1 + N/\phi N_t) = 712.12$ kN·m

Nominal out-plane member moment capacity,M_{ozt} = 711.00 kN·m

$\phi M_{ozt} = 639.90$ kN·m

Member Out-of-plane Capacity(Compression) $M_{oz} \leq M_{rz}$ *[Ref : Cl - 8.4.4.1]*

$M_{oz} = M_{bz}(1 - N/\phi N_{cy}) = 703.43$ kN·m

$\phi M_{ozc} = 633.09$ kN·m

Member Biaxial Capacity (Compression)

$M_{cz} = 709.63$ kN·m

$\phi M_{cz} = 638.67$ kN·m

$(M_z/\phi M_{cz})1.4+(M_y/\phi M_{iy})1.4 = 0.139$

Member Biaxial Capacity (tension)

$M_{tx} = 711.00$ kN·m

$\phi M_{tx} = 639.90$ kN·m

$(M_z/\phi M_{tz})1.4+(M_y/\phi M_{ry})1.4 = 0.116$

Results

Table 693: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	639.9	639.9001	Negligible	
ϕM_{sy} (kN·m)	92.23	92.2392	Negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{bz} (kN·m)	639.9	639.900	None	
ϕV_y (kN)	939.4	939.4	None	
ϕV_z (kN)	1,056	1,056.4	Negligible	
ϕN_s (kN)	2,931	2,921.3	Negligible	
ϕN_{cz} (kN)	2,600	2,599	None	
ϕN_{cy} (kN)	470	469.7	None	
ϕM_{rz} (kN·m)	639.900	639.900	None	
ϕM_{ry} (kN·m)	92.239	92.239	None	
ϕM_{iz} (kN·m)	638.7849	638.7849	None	
ϕM_{iy} (kN·m)	91.2744	91.2744	None	
ϕM_{ozc} (kN·m)	633.09	633.09	None	
ϕM_{ozt} (kN·m)	639.900	639.900	None	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-I section.std` is typically installed with the program.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_I_Section.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF A UB SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 03-Jan-17
END JOB INFORMATION
*
INPUT WIDTH 79
UNIT METER KN
*
JOINT COORDINATES
1 0 0 0; 4 9 0 0;
*
MEMBER INCIDENCES
1 1 4;
*
DEFINE PMEMBER
```

```
1 PMEMBER 1
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST UB530X92.4
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 4 FIXED
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -84 5
SELFWEIGHT Y -1
MEMBER LOAD
1 CON GX 10
1 CON GZ 10
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
MAIN 1 PMEMB 1
TRACK 1 PMEMB 1
DUCT 1 PMEMB 1
GLD 1 PMEMB 1
PBCRES ZZ 0 T 1 T PMEMB 1
PBCRES YY 0 T 1 T PMEMB 1
PBRACE TOP 0 FR 1 FR PMEMB 1
PBRACE BOTTOM 0 FR 1 FR PMEMB 1
CHECK CODE PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
    *
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
```

```
|--------------------------------------------------------------------------------
-----|
 |  PMember Number:
1                                                               |
 |  Member Section:  ST   UB530X92.4      (AISC
SECTIONS)                                    |
 |  Status: PASS  Ratio:   0.852  Critical Load Case:     1  Location:
0.00   |
 |  Critical Condition: T.
12.4                                                           |
 |  Critical Design Forces:  (Unit: KN
METE)                                  |
 |      FX:      -5.000E+00 T     FY:      39.108E+00      FZ:      -5.000E
+00      |
 |      MX:       0.000E+00       MY:      11.250E+00      MZ:      89.082E
+00      |

|--------------------------------------------------------------------------------
-----|
 | φMsz  =     639.900E+00 KNm     φMsy  =      92.239E+00 KNm     [Cl.
5.1    ]     |
 | φMbz  =     639.900E+00 KNm                                     [Cl.
5.1    ]     |
 | φVvmy =     939.444E+00 KNm     φVvmz =       1.056E+03 KNm     [Cl.
5.12.2 ]     |
 | φNs   =       2.921E+03 KN                                      [Cl.
6.1    ]     |
 | φNcz  =       2.599E+03 KN      φNcy  =     469.708E+00 KN      [Cl.
6.1    ]     |
 | φNt   =       3.186E+03 KN                                      [Cl.
7.1    ]     |
 | φMrz  =     639.900E+00 KNm     φMry  =      92.239E+00 KNm     [Cl.
8.3.2.2]     |
 | φMiz  =     638.785E+00 KNm     φMiy  =      91.274E+00 KNm     [Cl.
5.3.2.4]     |
 | φMozc =     633.088E+00 KNm     φMozt =     639.900E+00 KNm     [Cl.
8.4.4.1]     |
 | φMcz  =     633.088E+00 KNm                                     [Cl.
8.4.5.1]     |
 | φMtz  =     639.900E+00 KNm                                     [Cl.
8.4.5.1]     |

|--------------------------------------------------------------------------------
-----|
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-RHS Section

Verify the design capacity of a rectangular hollow section as per NZS3404 1997.

Details

Verify the section bending capacity of an HSST20X12X75 spanning 6 m. Steel grade is 250 MPa.

Note: This is the left column in the STAAD.Pro model (i.e., physical member no. 2).

Validation

Section Classification

Evaluate the slenderness effects of the Y axis:

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{269.3}{2(17.73)}\sqrt{\frac{250}{250}} = 15.19$$

Section classification is compact.

Evaluate the slenderness effects of the Z axis:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{472.5}{17.73}\sqrt{\frac{250}{250}} = 26.65$$

Section classification is compact

Section Bending Capacity About Z-Axis

Effective Section Modulus, $Z_{ez} = 4.425 \times 10^6\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 250 \times 4.425 \times 10^6 / 10^6 = 1{,}106 \text{ kN·m}$$

$$\phi M_{sz} = 0.9 \times 1{,}106 = 995.6 \text{ kN·m}$$

Section Bending Capacity About Y-Axis

Effective Section Modulus, $Z_{ey} = 3.114 \times 10^6\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 250 \times 3.114 \times 10^6 / 10^6 = 778.5 \text{ kN·m}$$

$$\phi M_{sy} = 0.9 \times 778.5 = 700.7 \text{ kN·m}$$

Member Bending Capacity

End restraint arrangement = LL

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 1.0

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1×6,000 = 6,000 mm

$\alpha_m = 1$ [Ref: Cl no -5.6.1.1 (b)]

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 E I_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 E I_w}{L_e^2}\right)\right]} = 41{,}334 \text{ kN·m}$$

$\alpha_s = 1$ [Ref : Clause 5.6.1.1 (c)]

$$M_{bx} = \alpha_m \alpha_s M_{sx} \le M_{sx}$$

$M_{bz} = 1 \times 1 \times 1{,}106 = 1{,}106\ kN{\cdot}m \le (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 1{,}106 = 995.6\ kN{\cdot}m$

Check for Axial Compression

Section Compression Capacity:

Gross Area, $A_g = 26{,}774\ mm^2$

Net Area, $A_n = 26{,}774\ mm^2$

Form factor, $K_f = A_e / A_g = 1.0$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1.0 \times 26{,}774 \times 250 = 6{,}694\ kN$ [Ref : Clause 6.2.1]

$\phi N_s = 0.9 \times 6{,}694 = 6{,}024\ kN$

Member Compression Capacity

Length of the member, $L = 6{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.0$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.0$

Effective Length of member, $L_{ez} = 1.0 \times 6{,}000\ mm = 6{,}000\ mm$

Effective Length of member, $L_{ey} = 1.0 \times 6{,}000\ mm = 6{,}000\ mm$

$r_z = \sqrt{(911.5 \times 10^6 / 26{,}774)} = 184.5$

$r_y = \sqrt{411.2 \times 10^6 / 26{,}774)} = 123.9$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 6{,}000\ /\ 184.5 = 32.5$

Geometrical Slenderness Ratio = $L_{ey}/r_y = 6{,}000\ /\ 123.9 = 48.4$

Member slenderness,

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 32.5\sqrt{1}\sqrt{\frac{250}{250}} = 32.5$$ [Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 48.4\sqrt{1}\sqrt{\frac{250}{250}} = 48.4$$ [Ref : Clause 6.3.3]

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 15.30$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 20.07$

$\alpha_b = -1.0$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az} \times \alpha_z = 17.22$

$\lambda_y = \lambda_{ny} + \alpha_{ay} \times \alpha_b = 28.34$

$\eta = 0.012$

$\eta = 0.048$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2 \times (\lambda_z/90)^2) = 14.33$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2 \times (\lambda_y/90)^2) = 5.785$

$\alpha_{cz} = 0.988$

α_{cy}= 0.949

The nominal member capacity,

N_{cz}= α_{cz}×N_s =0.988×6,694 = 6,611 kN [Ref : Clause 6.3.3]

ϕN_{cz} = 5,950 kN

The nominal member capacity,

N_{cy}= α_{cy}×N_s =0.949×6,694 = 6,353 kN [Ref : Clause 6.3.3]

ϕN_{cy} = 5,717 kN

Nominal Section tension Capacity

[Ref : Clause 7.1]

K_{te} = 1.00

N_{t1} = A_g×f_y = 6,024 kN

N_{t2} = 0.85×K_{te}×A_n×f_u = 6,694 kN

ϕN_t = 0.9×6,694 = 6,024 kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 694: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	995.6	995.5142	negligible	
ϕM_{sy} (kN·m)	700.7	700.547	negligible	
ϕM_{bz} (kN·m)	995.6	995.5142	negligible	
ϕN_{cz} (kN)	5,950	5,949	negligible	
ϕN_{cy} (kN)	5,717	5,719	negligible	
ϕN_s (kN)	6,024	6,024.2	negligible	
ϕN_t (kN)	6,024	6,024.2	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-RHS Section.std` is typically installed with the program.

```
STAAD PLANE
*
*  INPUT FILE: NZS3404_Frame 2.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  TUBE,HSST SHAPE  PER
```

```
*               THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 21-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0; 3 8 6 0; 4 8 0 0;
*
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE PMEMBER
2 PMEMBER 1
1 PMEMBER 2
3 PMEMBER 3
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY INDIAN
2 TABLE ST TUB1251256
*
MEMBER PROPERTY AMERICAN
1 3 TABLE ST HSST20X12X.75
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 FIXED
4 PINNED
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
PMEMBER LOAD
1 UNI Y -10
2 UMOM GZ 25
3 TRAP GZ 4 6
3 CON Y 10 0.1 0
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
IST 2 PMEMB 1 TO 3
TRACK 2 PMEMB 1 TO 3
SGR 0 PMEMB 1 TO 3
CHECK CODE PMEMB 2 3
```

```
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     2
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   HSST20X12X.75            (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
8.4.4.1
   Governing Ratio:   0.090  (PASS)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       508.0000 mm    b:       304.8000 mm
    t:        17.7292 mm
   Ag:     26774.1387 mm2   J:   924.0338E+06 mm4            Iw:     0.0000E
+00 mm6
   Iz:   911.5469E+06 mm4  Sz:     4.4245E+06 mm3 (plastic)   Zz:     3.5888E
+06 mm3 (elastic)
   rz:   184.5150E+00 mm
   Iy:   411.2367E+06 mm4  Sy:     3.1135E+06 mm3 (plastic)   Zy:     2.6984E
+06 mm3 (elastic)
   ry:   123.9333E+00 mm
      STAAD PLANE                                                -- PAGE NO.
4
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS 1163
   Nominal Grade            :  250
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    250.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    250.000 MPa         [NZS3404 Table 2.1]
   fu          :    320.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      48.413  LOAD:     1   LOC.
```

```
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  180.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.090
   Critical Location  :   0.000 m from Start.
   Mz* =   -89.2106E+00 KNm
   Section Slenderness: Compact
   Zez =     4.4245E+06 mm3
   φMsz =   995.5142E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My* =     0.0000E+00 KNm
   Section Slenderness: Compact
   Zey =     3.1135E+06 mm3
   φMsy =   700.5470E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.090
   Critical Location  :   0.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )-  6.00 m(F )
   Mz* =   -89.2106E+00 KNm
   kt   =      1.00                        [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                        [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                        [NZS3404 Table 5.6.3(3)]
   le   =      6.00 m                      [NZS3404 5.6.3]
   αm   =     1.000                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =    41.0351E+03 KNm                [NZS3404 5.6.1.1.1(d)]
   αsz  =     1.000                         [NZS3404 5.6.1.1.1(c)]
   φMbz =   995.5142E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD PLANE                                          -- PAGE NO.
5
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.001
   Critical Location  :   0.000 m from Start.
   Vy*   =     1.1125E+00 KN
   φVvmy =     2.1765E+03 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vz*   =     0.0000E+00 KN
   φVvmz =     1.2893E+03 KN                [NZS3404 5.12.2]
      STAAD PLANE                                          -- PAGE NO.
6
    *
   AXIAL
   -----
   Section Compression Capacity
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    40.2237E+00 KN
   Ae    =    26.7741E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =    26.7741E+03 mm2
   φNs   =     6.0242E+03 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    40.2237E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  6.00 m(U )
   Lez   =      6.00 m
   αb    =     -1.00                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    32.518                        [NZS3404 6.3.3]
   λ,z   =    17.215                        [NZS3404 6.3.3]
   ε,z   =    14.331                        [NZS3404 6.3.3]
   αc,z  =     0.988                        [NZS3404 6.3.3]
   φNcz  = 0.5949E+4 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    40.2237E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  6.00 m(U )
   Ley   =      6.00 m
   λn,y  =    48.413                        [NZS3404 6.3.3]
   λ,y   =    28.343                        [NZS3404 6.3.3]
   ε,y   =     5.785                        [NZS3404 6.3.3]
   αc,y  =     0.949                        [NZS3404 6.3.3]
   φNcy  = 0.5719E+4 KN                     [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                        [User defined]
   An    =    26.7741E+03 mm2
   φNt   =     6.0242E+03 KN                [NZS3404 7.2]
      STAAD PLANE                                              -- PAGE NO.
7
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.090
   Critical Location  :   0.000 m from Start.
   φMrz  =   995.5142E+00 KNm               [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMry  =   700.5470E+00 KNm               [NZS3404 8.3.3]
```

```
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.034
   Critical Location  :   0.000 m from Start.
   γ     =     1.407                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.090
   Critical Location  :   0.000 m from Start.
   φMiz  =   995.5142E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiy  =   695.6196E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.090
   Critical Location  :   0.000 m from Start.
   φMoz,c=   988.5120E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.

*****************************************************************************
**
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
      STAAD PLANE                                          -- PAGE NO.
8
    *
   MEMBER DESIGN OUTPUT FOR PMEMBER     3
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
```

```
   --------------
   Designation: ST   HSST20X12X.75            (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
8.4.4.1
   Governing Ratio:   0.053  (PASS)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       508.0000 mm    b:       304.8000 mm
    t:        17.7292 mm
   Ag:     26774.1387 mm2   J:   924.0338E+06 mm4             Iw:     0.0000E
+00 mm6
   Iz:   911.5469E+06 mm4  Sz:     4.4245E+06 mm3 (plastic)   Zz:     3.5888E
+06 mm3 (elastic)
   rz:   184.5150E+00 mm
   Iy:   411.2367E+06 mm4  Sy:     3.1135E+06 mm3 (plastic)   Zy:     2.6984E
+06 mm3 (elastic)
   ry:   123.9333E+00 mm
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS 1163
   Nominal Grade            :  250
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    250.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    250.000 MPa         [NZS3404 Table 2.1]
   fu          :    320.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      48.413  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  180.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
      STAAD PLANE                                              -- PAGE NO.
9
    *
   Critical Load Case :     1*
   Critical Ratio     :   0.053
   Critical Location  :   0.000 m from Start.
   Mz* =    52.3655E+00 KNm
   Section Slenderness: Compact
   Zez =     4.4245E+06 mm3
   φMsz =   995.5142E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.022
   Critical Location  :   6.000 m from Start.
   My* =    15.6295E+00 KNm
   Section Slenderness: Compact
   Zey =     3.1135E+06 mm3
   φMsy =   700.5470E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.053
   Critical Location  :   0.000 m from Start.
   Crtiical Flange Segment:
```

```
   Location (Type):   0.00 m(F )-  6.00 m(F )
   Mz* =    52.3655E+00 KNm
   kt   =      1.00                        [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                        [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                        [NZS3404 Table 5.6.3(3)]
   le   =      6.00 m                      [NZS3404 5.6.3]
   αm   =     1.000                        [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =    41.0351E+03 KNm               [NZS3404 5.6.1.1.1(d)]
   αsz  =     1.000                        [NZS3404 5.6.1.1.1(c)]
   φMbz =   995.5142E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.004
   Critical Location  :   0.500 m from Start.
   Vy*   =     8.8875E+00 KN
   φVvmy =     2.1765E+03 KN               [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.013
   Critical Location  :   6.000 m from Start.
   Vz*   =    16.2009E+00 KN
   φVvmz =     1.2893E+03 KN               [NZS3404 5.12.2]
      STAAD PLANE                                          -- PAGE NO.
10
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    39.7763E+00 KN
   Ae    =    26.7741E+03 mm2              [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                       [AS 4100 6.2.2]
   An    =    26.7741E+03 mm2
   φNs   =     6.0242E+03 KN               [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    39.7763E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  6.00 m(U )
   Lez   =      6.00 m
   αb    =     -1.00                       [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    32.518                       [NZS3404 6.3.3]
   λ,z   =    17.215                       [NZS3404 6.3.3]
   ε,z   =    14.331                       [NZS3404 6.3.3]
   αc,z  =     0.988                       [NZS3404 6.3.3]
   φNcz  = 0.5949E+4 KN                    [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   N*    =    39.7763E+00 KN
   Unbraced Segment:
```

```
   Location (Type):   0.00 m(U )-  6.00 m(U )
   Ley   =      6.00 m
   λn,y  =    48.413                       [NZS3404 6.3.3]
   λ,y   =    28.343                       [NZS3404 6.3.3]
   ε,y   =     5.785                       [NZS3404 6.3.3]
   αc,y  =     0.949                       [NZS3404 6.3.3]
   φNcy  = 0.5719E+4 KN                    [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =    26.7741E+03 mm2
   φNt   =     6.0242E+03 KN               [NZS3404 7.2]
      STAAD PLANE                                            -- PAGE NO.
11
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.053
   Critical Location  :   0.000 m from Start.
   φMrz  =   995.5142E+00 KNm              [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.022
   Critical Location  :   6.000 m from Start.
   φMry  =   700.5470E+00 KNm              [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.020
   Critical Location  :   0.000 m from Start.
   γ     =     1.407                        [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.053
   Critical Location  :   0.000 m from Start.
   φMiz  =   989.0403E+00 KNm              [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.022
   Critical Location  :   6.000 m from Start.
   φMiy  =   695.8025E+00 KNm              [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN               [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm              [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.053
   Critical Location  :   0.000 m from Start.
   φMoz,c=   988.5899E+00 KNm              [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.021
   Critical Location  :   0.000 m from Start.
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-Simply Supported Beam With Overhang

Verify the design strength of an I section per the NZS3404 1997 code.

Reference

Kirke, Brian, and Iyad Hassan Al-Jamel. *Steel Structures Design Manual to AS4100.* 2004. p.124

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

Details

From the reference:

> The 250UC89.5 beam in Grade 300 steel shown below is continuous over the supports at B and D and is free at A and E. The beam section is restrained against lateral deflection at B and D, fully restrained against twist rotation at B and D, and is unrestrained at A and E. A downward concentrated load of 5P* acts at C and a downward concentrated load of P* acts at A and E. These loads act at the top flange, and are free to deflect laterally with the beam. Determine the maximum value of P*.

Validation

Section Classification

Evaluate the slenderness effects of the flange:

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{256}{2(17.3)}\sqrt{\frac{280}{250}} = 7.83$$

Section classification of the flange is compact.

Evaluate the slenderness effects of the web:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{225.4}{10.5}\sqrt{\frac{280}{250}} = 22.72$$

Section classification of the web is compact

Section Bending Capacity About Z-Axis

Effective Section Modulus, $Z_{ez} = 1.23\times10^6\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 280 \times 1.23 \times 10^{-6} = 344.4 \text{ kN}\cdot\text{m}$$

$$\phi M_{sz} = 0.9 \times 344.4 = 309.96 \text{ kN}\cdot\text{m}$$

Section Bending Capacity About Y-Axis

Effective Section Modulus, $Z_{ey} = 575 \times 10^3 \ mm^3$

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 280 \times 575 \times 10^{-3} = 161 \text{ kN}\cdot\text{m}$$

$$\phi M_{sy} = 0.9 \times 161 = 144.9 \text{ kN}\cdot\text{m}$$

Member Bending Capacity of Cantilevers

End restraint arrangement = FU

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 1.0

Load Height factor, Kl, (LHT) = 2 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×2×4,000 = 8,000 mm

The alpha value for a cantilever is taken from Table 5.6.2 in the code.

$$\alpha_m = 1.25 \qquad \text{[Ref. Table 5.6.2]}$$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 E I_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 E I_w}{L_e^2}\right)\right]}$$

$$M_o = \sqrt{\frac{\pi^2(205)(4.84 \times 10^6)}{(8{,}000)^2}\left[80\left(1.04 \times 10^6\right) + \frac{\pi^2(205)(713 \times 10^9)}{(8{,}000)^2}\right]}10^{-3} = 399 \text{ kN}\cdot\text{m}$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\}$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{344.4}{399}\right)^2 + 3\right]} - \left(\frac{344.4}{399}\right)\right\} = 0.643 \qquad \text{[Ref : Clause 5.6.1.1 (c)]}$$

$$M_{bx} = \alpha_m \alpha_s M_{sx} \le M_{sx}$$

$$M_{bz} = 1.25 \times 0.643 \times 344.4 = 276.8 \text{ kN}\cdot\text{m} \le (M_{sz}, M_{sy})_{Max.} \qquad \text{[Ref : Clause 5.6.1.1.1(a)]}$$

$$\phi M_{bz} = 0.9 \times 276.8 = 249.1 \text{ kN}\cdot\text{m}$$

Member Bending Capacity of Main Span

End restraint arrangement = FF

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 1.0

Load Height factor, Kl, (LHT) = 1.4 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1.4×8,000 = 11,200 mm

$$a_m = \frac{1.7M_m^{\times}}{\sqrt{\left(M_1^*\right)^2 + \left(M_2^*\right)^2 + \left(M_3^*\right)^2}} \leq 2.5$$

$$a_m = \frac{1.7 \times 6P^*}{\sqrt{(P^*)^2 + (6P^*)^2 + (P^*)^2}} = 1.655$$

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]}$$

$$M_o = \sqrt{\frac{\pi^2(205)(4.84 \times 10^6)}{(11{,}200)^2}\left[80\left(1.04 \times 10^6\right) + \frac{\pi^2(205)(713 \times 10^9)}{(11{,}200)^2}\right]}10^{-3} = 271.9 \text{ kN·m}$$

$$a_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\}$$

$$a_s = 0.6\left\{\sqrt{\left[\left(\frac{344.4}{271.9}\right)^2 + 3\right]} - \left(\frac{344.4}{271.9}\right)\right\} = 0.528$$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sx} \leq M_{sx}$

$M_{bz} = 1.655 \times 0.528 \times 344.4 = 300.95 \text{ kN·m} \leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 300.95 = 270.9 \text{ kN·m}$

<u>Maximum Load Determination</u>

The moments over the supports due to the cantilevers, $M_1 = M_2 = 4 \times P^*$

The maximum moment in the middle span occurs at the 5P load:

$$M_{max} = \frac{(5P^*)8}{4} - (M_1 + M_2) = 10P^* - 4P^* = 6P^*$$

So the maximum value of P* is derived from setting $M_{max} = \phi M_{bz}$ and solving for P*:

$P^* = \phi M_{bz}/6 = 270.9 / 6 = 45.1 \text{ kN}$

Results

Table 695: Comparison of results

Result Type	Calculations	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	309.96	309.96	none	
ϕM_{sy} (kN·m)	144.9	142.9313	1.4%	
ϕM_{bz} (kN·m) - Cantilever	249.1	249.4725	negligible	
ϕM_{bz} (kN·m) - Main span	270.9	269.5	negligible	

Result Type	Calculations	STAAD.Pro	Difference	Comments
P* (kN)	45.1†	44.9‡	0.5%	

†The reference gives a P* value of 44.33 kN (though to be noted this is also calculated for the AS 4100 code as opposed to the NZS 34043 1997 code).

‡For STAAD.Pro, the value of 44.9 kips gives a critical ratio of 1.00 for the member bending capacity.

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Simply Supported Beam With Overhang.std` is typically installed with the program.

```
STAAD PLANE
*
*  INPUT FILE: NZS3404_Simply_Supported_Beam_with_Overhang.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF UC SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 17-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 12 0 0; 4 16 0 0;
*
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE PMEMBER
1 PMEMBER 1
2 PMEMBER 2
3 PMEMBER 3
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TO 3 TABLE ST UC250X89.5
*
CONSTANTS
MATERIAL STEEL ALL
```

```
*
SUPPORTS
2 PINNED
3 FIXED BUT FX MY MZ
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
*SELFWEIGHT Y -1
MEMBER LOAD
2 CON GY -224.5
JOINT LOAD
1 4 FY -44.9
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
TRACK 2 PMEMB 1 TO 3
SGR 2 PMEMB 1 TO 3
LHT 1 PMEMB 1 TO 3
PBRACE TOP 0 U 1 F PMEMB 1
PBRACE BOTTOM 0 U 1 F PMEMB 1
PBCRES ZZ 0 U 1 T PMEMB 1
PBCRES YY 0 U 1 T PMEMB 1
PBRACE TOP 0 F 1 U PMEMB 3
PBRACE BOTTOM 0 F 1 U PMEMB 3
PBCRES ZZ 0 T 1 U PMEMB 3
PBCRES YY 0 T 1 U PMEMB 3
CHECK CODE PMEMB 1 2 3
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   UC250X89.5               (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
5.1
   Governing Ratio:   0.720  (PASS)
```

```
   Governing Location:   4.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       260.0000 mm   bf:       256.0000 mm
   tf:        17.3000 mm   tw:        10.5000 mm
   Ag:     11400.0000 mm2   J:     1.0400E+06 mm4             Iw:   712.3514E
+09 mm6
   Iz:   143.0000E+06 mm4  Sz:     1.2300E+06 mm3 (plastic)   Zz:     1.1000E
+06 mm3 (elastic)
   rz:   111.9994E+00 mm
   Iy:    48.4000E+06 mm4  Sy:   575.0001E+03 mm3 (plastic)   Zy:   378.1251E
+03 mm3 (elastic)
   ry:    65.1584E+00 mm
      STAAD PLANE                                              -- PAGE NO.
4
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    280.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      61.389  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.579
   Critical Location  :   4.000 m from Start.
   Mz* =   179.6000E+00 KNm
   Section Slenderness: Compact
   Zez =     1.2300E+06 mm3
   φMsz =   309.9600E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My* =     0.0000E+00 KNm
   Section Slenderness: Compact
   Zey =   567.1876E+03 mm3
   φMsy =   142.9313E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.720
   Critical Location  :   4.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(U )-  4.00 m(F )
   Mz* =   179.6000E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      2.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =      8.00 m                       [NZS3404 5.6.3]
```

```
   αm    =     1.250                          [NZS3404 5.6.1.1.1(b)(iii)]
   Mo    =   399.9072E+00 KNm                 [NZS3404 5.6.1.1.1(d)]
   αsz  =     0.644                           [NZS3404 5.6.1.1.1(c)]
   φMbz =   249.4725E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD PLANE                                               -- PAGE NO.
5
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.095
   Critical Location  :   0.000 m from Start.
   Vy*   =   -44.9000E+00 KN
   φVvmy =   471.7440E+00 KN                  [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vz*   =     0.0000E+00 KN
   φVvmz =     1.3393E+03 KN                  [NZS3404 5.12.2]
      STAAD PLANE                                               -- PAGE NO.
6
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =    11.4000E+03 mm2                 [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                          [AS 4100 6.2.2]
   An    =    11.4000E+03 mm2
   φNs   =     2.8728E+03 KN                  [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  4.00 m(T )
   Lez   =      8.80 m
   αb    =      0.00                          [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    83.153                          [NZS3404 6.3.3]
   λ,z   =    83.153                          [NZS3404 6.3.3]
   ε,z   =     1.219                          [NZS3404 6.3.3]
   αc,z  =     0.659                          [NZS3404 6.3.3]
   φNcz  = 0.1892E+4 KN                       [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  4.00 m(T )
   Ley   =      8.80 m
   λn,y  =   142.929                          [NZS3404 6.3.3]
```

```
   λ,y   =   142.929                        [NZS3404 6.3.3]
   ε,y   =     0.782                        [NZS3404 6.3.3]
   αc,y  =     0.318                        [NZS3404 6.3.3]
   φNcy  = 0.9146E+3 KN                     [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                        [User defined]
   An    =    11.4000E+03 mm2
   φNt   =     2.8728E+03 KN                [NZS3404 7.2]
      STAAD PLANE                                              -- PAGE NO.
7
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.579
   Critical Location  :   4.000 m from Start.
   φMrz  =   309.9600E+00 KNm               [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMry  =   142.9313E+00 KNm               [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.466
   Critical Location  :   4.000 m from Start.
   γ     =     1.400                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.579
   Critical Location  :   4.000 m from Start.
   φMiz  =   309.9600E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiy  =   142.9313E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoz,c=     0.0000E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
```

```
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.

*****************************************************************************
**
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
      STAAD PLANE                                            -- PAGE NO.
8
    *
   MEMBER DESIGN OUTPUT FOR PMEMBER     2
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   UC250X89.5               (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
5.1
   Governing Ratio:   1.000  (PASS)
   Governing Location:   4.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       260.0000 mm   bf:       256.0000 mm
   tf:        17.3000 mm   tw:        10.5000 mm
   Ag:     11400.0000 mm2   J:     1.0400E+06 mm4             Iw:   712.3514E
+09 mm6
   Iz:   143.0000E+06 mm4  Sz:     1.2300E+06 mm3 (plastic)   Zz:     1.1000E
+06 mm3 (elastic)
   rz:   111.9994E+00 mm
   Iy:    48.4000E+06 mm4  Sy:   575.0001E+03 mm3 (plastic)   Zy:   378.1251E
+03 mm3 (elastic)
   ry:    65.1584E+00 mm
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    280.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     122.778  LOAD:     1   LOC.
(MET):   0.000
```

```
                  ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
      STAAD PLANE                                              -- PAGE NO.
9
    *
   Critical Load Case :     1*
   Critical Ratio     :   0.869
   Critical Location  :   4.000 m from Start.
   Mz* =  -269.4000E+00 KNm
   Section Slenderness: Compact
   Zez =     1.2300E+06 mm3
   φMsz =   309.9600E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My* =     0.0000E+00 KNm
   Section Slenderness: Compact
   Zey =   567.1876E+03 mm3
   φMsy =   142.9313E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   1.000
   Critical Location  :   4.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )-  8.00 m(F )
   Mz* =  -269.4000E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      1.40                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =     11.20 m                       [NZS3404 5.6.3]
   αm   =     1.655                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =   270.1554E+00 KNm                [NZS3404 5.6.1.1.1(d)]
   αsz  =     0.525                         [NZS3404 5.6.1.1.1(c)]
   φMbz =   269.5068E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.294
   Critical Location  :   4.000 m from Start.
   Vy*   =   112.2500E+00 KN
   φVvmy =   381.8150E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vz*   =     0.0000E+00 KN
   φVvmz =     1.3393E+03 KN                [NZS3404 5.12.2]
      STAAD PLANE                                              -- PAGE NO.
10
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
```

```
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =    11.4000E+03 mm2              [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                       [AS 4100 6.2.2]
   An    =    11.4000E+03 mm2
   φNs   =     2.8728E+03 KN               [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  8.00 m(U )
   Lez   =      8.00 m
   αb    =      0.00                       [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    75.593                       [NZS3404 6.3.3]
   λ,z   =    75.593                       [NZS3404 6.3.3]
   ε,z   =     1.352                       [NZS3404 6.3.3]
   αc,z  =     0.711                       [NZS3404 6.3.3]
   φNcz  = 0.2043E+4 KN                    [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  8.00 m(U )
   Ley   =      8.00 m
   λn,y  =   129.936                       [NZS3404 6.3.3]
   λ,y   =   129.936                       [NZS3404 6.3.3]
   ε,y   =     0.831                       [NZS3404 6.3.3]
   αc,y  =     0.372                       [NZS3404 6.3.3]
   φNcy  = 0.1068E+4 KN                    [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =    11.4000E+03 mm2
   φNt   =     2.8728E+03 KN               [NZS3404 7.2]
      STAAD PLANE                                        -- PAGE NO.
11
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.869
   Critical Location  :   4.000 m from Start.
   φMrz  =   309.9600E+00 KNm              [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMry  =   142.9313E+00 KNm              [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.822
   Critical Location  :   4.000 m from Start.
   γ     =     1.400                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.869
   Critical Location  :   4.000 m from Start.
   φMiz  =   309.9600E+00 KNm              [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiy  =   142.9313E+00 KNm              [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN               [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm              [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoz,c=     0.0000E+00 KNm              [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
```

V.NZS3404 1997-Tube Section Compact

Verify the design capacity of a PIPX80 section per the NZS3404 1997 code.

Details

Verify the bending capacity of a PIPX80 member. Steel grade = 250 MPa. The simply supported span is 9 m with a 8.4 kN load at 5 m from the left support.

Validation

Section Classification

Evaluate the slenderness effects of the pipe member:

$$\lambda_{ew} = \frac{\mathrm{OD}}{t}\sqrt{\frac{f_y}{250}} = \frac{219.1}{11.8}\sqrt{\frac{250}{250}} = 18.3$$

Section classification is compact

Section Bending Capacity About Z-Axis

Effective Section Modulus, $Z_{ez} = 508 \times 10^3\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$M_{sz} = 250\times 508\times10^3/10^6 = 127$ kN·m

$\phi M_{sz} = 0.9\times127 = 114.3$ kN·m

The pipe is symmetric, thus $\phi M_{sy} = \phi M_{sz}$

Member Bending Capacity

End restraint arrangement = LL

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 1.0

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1×9,000 = 9,000 mm

$\alpha_m = 1$ [Ref: Cl no -5.6.1.1 (b)]

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 EI_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 EI_w}{L_e^2}\right)\right]} = 2{,}641 \text{ kN·m}$$

$\alpha_s = 1$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sx} \leq M_{sx}$

$M_{bz} = 1 \times 1 \times 127 = 127$ kN·m $\leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9\times127 = 114.3$ kN·m

Check for Shear

Section Shear Capacity (Along Y axis), $V_y = 0.36\times f_y \times A_g = 0.36\times250\times7{,}677 = 691\ kN$

$\phi V_y = 0.9\times691 = 622$ kN

The pipe is symmetric, thus $\phi V_z = \phi V_y$

Check for Axial Compression

Section Compression Capacity:

Gross Area, $A_g = 7{,}677\ mm^2$

Net Area, $A_n = 7{,}677\ mm^2$

Form factor, $K_f = A_e/A_g = 1.0$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1.0\times7{,}677\times250 = 1{,}919$ kN [Ref : Clause 6.2.1]

$\phi N_s = 0.9\times1{,}919 = 1{,}727$ kN

Member Compression Capacity

Length of the member, $L = 9{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.0$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.0$

Effective Length of member, L_{ez} = *1.0×9,000 mm = 9,000 mm*

Effective Length of member, L_{ey} = *1.0×9,000 mm = 9,000 mm*

$r_y = r_z = \sqrt{(41.62\times10^6 / 7,677)} = 73.63$

Geometrical Slenderness Ratio = $L_{ey}/r_y = L_{ez}/r_z$ = *9,000 / 73.63 = 122.2*

Member slenderness,

$$\lambda_n = \frac{L_e}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 122.2\sqrt{1}\sqrt{\frac{250}{250}} = 122.2$$ [Ref : Clause 6.3.3]

$\alpha_a = 2,100\times(\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2,050) = 15.10$

$\alpha_b = -1.0$ [Ref : Table 6.3.3(2)]

$\lambda = \lambda_n + \alpha_a\times\alpha_b = 107.1$

$\eta = 0.3$

$\xi_y = \xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2\times(\lambda_z/90)^2) = 1$

$\alpha_{cy} = \alpha_{cz} = 0.495$

The nominal member capacity,

$N_{cy} = N_{cz} = \alpha_{cz}\times N_s = 0.495\times1,919 = 950$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = \phi N_{cz} = 855$ kN

<u>Nominal Section tension Capacity</u>

[Ref : Clause 7.1]

$K_{te} = 1.00$

$N_{t1} = A_g\times f_y = 7,677 \times 250\times10^{-3} = 1,919$ kN

$N_{t2} = 0.85\times K_{te}\times A_n\times f_u = 0.85(1.0)(7,677)(250\times10^{-3}) = 1,631$ kN

$\phi N_t = 0.9\times1,919 = 1,727$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 696: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	114.3	114.2998	negligible	
ϕM_{bz} (kN·m)	114.3	114.2998	negligible	
ϕV_y (kN)	622	622.9	negligible	
ϕN_s (kN)	1,727	1,727.4	negligible	
ϕN_{cy} (kN)	855	854.7	negligible	
ϕN_t (kN)	1,727	1,727.4	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Tube Section Compact.std` is typically installed with the program.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Tube_Section_Compact.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  TUBE SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 03-Jan-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 4 9 0 0; 5 3 0 0; 6 6 0 0;
*
MEMBER INCIDENCES
1 1 5; 2 5 6; 3 6 4;
DEFINE PMEMBER
1 TO 3 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST PIPX80
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 PINNED
4 FIXED BUT FX MY MZ
PRINT ALL
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 CON GY -8.4
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
*
```

```
PARAMETER 1
CODE NZS3404 1997
BEAM 1 PMEMB 1
IST 2  PMEMB 1
SGR 6  PMEMB 1
SKL 1  PMEMB 1
SKR 1  PMEMB 1
SKT 1  PMEMB 1
TRACK 2  PMEMB 1
CHECK CODE  PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   PIPX80                   (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
5.1
   Governing Ratio:   0.165  (PASS)
   Governing Location:   4.500 m from Start.
   SECTION PROPERTIES
   ------------------
   OD:       219.0750 mm    t:        11.8110 mm
   Ag:      7677.4033 mm2   J:    83.2463E+06 mm4             Iw:     0.0000E
+00 mm6
   Iz:    41.6231E+06 mm4  Sz:   507.9990E+03 mm3 (plastic)   Zz:   379.9899E
+03 mm3 (elastic)
   rz:    73.6309E+00 mm
   Iy:    41.6231E+06 mm4  Sy:   507.9990E+03 mm3 (plastic)   Zy:   379.9899E
+03 mm3 (elastic)
   ry:    73.6309E+00 mm
      STAAD SPACE                                               -- PAGE NO.
13
    *
   MATERIAL PROPERTIES
   -------------------
```

```
   Material Standard        :  AS 1163
   Nominal Grade            :  250
   Residual Stress Category :  HR (Hot-rolled)
   E (#)        : 204999.984 MPa         [NZS3404 1.4]
   G            :  80000.000 MPa         [NZS3404 1.4]
   fy, flange   :    250.000 MPa         [NZS3404 Table 2.1]
   fy, web      :    250.000 MPa         [NZS3404 Table 2.1]
   fu           :    320.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     122.231  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.165
   Critical Location  :   4.500 m from Start.
   Mz* =   -18.9000E+00 KNm
   Section Slenderness: Compact
   Zez =   507.9990E+03 mm3
   φMsz =   114.2998E+00 KNm                 [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My* =     0.0000E+00 KNm
   Section Slenderness: Compact
   Zey =   507.9990E+03 mm3
   φMsy =   114.2998E+00 KNm                 [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.165
   Critical Location  :   4.500 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )-  9.00 m(F )
   Mz* =   -18.9000E+00 KNm
   kt   =      1.00                          [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                          [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                          [NZS3404 Table 5.6.3(3)]
   le   =      9.00 m                        [NZS3404 5.6.3]
   αm   =     1.000                          [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =     2.6123E+03 KNm                 [NZS3404 5.6.1.1.1(d)]
   αsz  =     1.000                          [NZS3404 5.6.1.1.1(c)]
   φMbz =   114.2998E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD SPACE                                              -- PAGE NO.
14
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.007
   Critical Location  :   0.000 m from Start.
   Vy*   =      4.2000E+00 KN
   φVvmy =   622.9389E+00 KN                 [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
```

```
   Critical Location  :   0.000 m from Start.
   Vz*   =     0.0000E+00 KN
   φVvmz =   622.9389E+00 KN                [NZS3404 5.12.2]
      STAAD SPACE                                                   -- PAGE NO.
15
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =     7.6774E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                        [AS 4100 6.2.2]
   An    =     7.6774E+03 mm2
   φNs   =     1.7274E+03 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Lez   =      9.00 m
   αb    =     -1.00                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =   122.231                        [NZS3404 6.3.3]
   λ,z   =   107.130                        [NZS3404 6.3.3]
   ε,z   =     0.961                        [NZS3404 6.3.3]
   αc,z  =     0.495                        [NZS3404 6.3.3]
   φNcz  = 0.8547E+3 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Ley   =      9.00 m
   λn,y  =   122.231                        [NZS3404 6.3.3]
   λ,y   =   107.130                        [NZS3404 6.3.3]
   ε,y   =     0.961                        [NZS3404 6.3.3]
   αc,y  =     0.495                        [NZS3404 6.3.3]
   φNcy  = 0.8547E+3 KN                     [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                        [User defined]
   An    =     7.6774E+03 mm2
   φNt   =     1.7274E+03 KN                [NZS3404 7.2]
      STAAD SPACE                                                   -- PAGE NO.
16
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
```

```
   Critical Load Case :     1*
   Critical Ratio     :   0.165
   Critical Location  :   4.500 m from Start.
   φMrz  =   114.2998E+00 KNm               [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMry  =   114.2998E+00 KNm               [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.165
   Critical Location  :   4.500 m from Start.
   γ     =     1.400                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.165
   Critical Location  :   4.500 m from Start.
   φMiz  =   114.2998E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiy  =   114.2998E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoz,c=     0.0000E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.

*****************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-Tube Section Non Compact

Verify the design capacity of a pipe section per the NZS3404 1997 code.

Details

Verify the capacity of a PIP3239H member. Steel grade = 250 MPa. The simply supported span is 9 m with a 60 kN load at mid-span.

Validation

Section Classification

Evaluate the slenderness effects of the pipe member:

$$\lambda_{ef} = \frac{\mathrm{OD}}{t}\sqrt{\frac{f_y}{250}} = \frac{323.9}{6.3}\sqrt{\frac{250}{250}} = 51.41$$

Section classification is non-compact ($\lambda_{ey} = 120 > \lambda_{ef} > \lambda_{ep} = 50$)

Section Bending Capacity About Z-Axis

Effective Section Modulus for a compact section:

$$\min\left| \begin{array}{l} S_{xx} = 636 \times 10^3 \mathrm{mm}^3 \blacktriangleleft \\ 1.5 \times Z_{xx} = 1.5(489.6 \times 10^3)\mathrm{mm}^3 = 734.4 \times 10^3 \mathrm{mm}^3 \end{array}\right.$$

Effective section modulus for a non-compact section:

$$Z_e = Z + \frac{\lambda_{ey} - \lambda_{ef}}{\lambda_{ey} - \lambda_{ep}}(Z_c - Z_{xx}) = 489.6 + \frac{120 - 51.41}{120 - 50}(636 - 489.6) = 633.1 \times 10^3 \ \mathrm{mm}^3$$

The nominal section capacity in bending about Z axis, $\phi M_s = \phi f_y \times Z_e$

$$M_s = 250 \times 633.1 \times 10^3/10^6 = 158.3 \text{ kN·m}$$

$$\phi M_{sz} = 0.9 \times 158.3 = 142.4 \text{ kN·m}$$

Member Bending Capacity

End restraint arrangement = LL

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Both

Lateral rotation restraint factor, K_r (SKR) = 1.0

Load Height factor, Kl, (LHT) = 1.00 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×1×9,000 = 9,000 mm

$\alpha_m = 1$ [Ref: Cl no -5.6.1.1 (b)]

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 E I_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 E I_w}{L_e^2}\right)\right]}$$

$$= \sqrt{\frac{\pi^2 \times 250 \times 10^3 79.29 \times 10^6}{9,\ 000^2}\left[80,\ 000 \times 158.58 \times 10^6 + \left(\frac{\pi^2 \times 250 \times 10^3 79.29 \times 10^6}{9,\ 000^2}\right)\right]} = 5,\ 013 \text{ kN·m}$$

$\alpha_s = 1$ [Ref : Clause 5.6.1.1 (c)]

Member bending capacity, M_{bz}

$M_{bz} = \alpha_m \alpha_s M_{sz} \leq M_{sz}$

$M_{bz} = 1 \times 1 \times 158.2 = 158.3$ kN·m $\leq (M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

$\phi M_{bz} = 0.9 \times 158.3 = 142.4$ kN·m

Check for Shear

Nominal shear capacity, V_z, of a circular hollow section: *[Ref. Cl no 5.11.4.2]*

$V_y = 0.36 \times f_y \times A_e = 0.36 \times 250 \times 6{,}290 \times 10^{-3} = 566.1$ kN

$\phi V_y = 0.9 \times 566.1 = 509.5$ kN

Nominal shear capacity, V_y, of a circular hollow section: *[Ref. Cl no 5.12.2]*

Design bending moment, $M = 135\ kN \cdot m$

$0.75 \times \phi \times M_s \leq M \leq \phi \times M_s$

$0.75 \times 142.4 = 106.8$ kN·m

$$V_y = V_v\left[2.2 - \left(\frac{1.6 \times M}{\phi M_S}\right)\right] = 566.1 \times \left[2.2 - \left(\frac{1.6 \times 135}{142.4}\right)\right] = 386.9 \text{ kN}$$

$\phi V_y = 0.9 \times 386.9 = 348.3$ kN

Check for Axial Capacity

Section Compression Capacity:

Gross Area, $A_g = 6{,}290\ mm^2$

The effective outside diameter is the minimum of d_{e1}, d_{e2}, and d_o:

$$d_{e1} = d_o\sqrt{\frac{\lambda_{ey}}{\lambda_e}} = 409.1 \text{ mm}$$

$$d_{e2} = d_o\left(\frac{3 \times \lambda_{ey}}{\lambda_e}\right)^2 = 7,\ 415 \text{ mm}$$

The effective outside diameter = $d_o = 323.9\ mm$

Net Area, $A_n = 6{,}290\ mm^2$

Form factor, $K_f = A_e/A_g = 1.0$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1.0 \times 6{,}290 \times 250 = 1{,}573$ kN [Ref : Clause 6.2.1]

$\phi N_s = 0.9 \times 1{,}573 = 1{,}415$ kN

Member Compression Capacity

Length of the member, $L = 9{,}000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 1.0$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 1.0$

Effective Length of member, $L_{ez} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

Effective Length of member, $L_{ey} = 1.0 \times 9{,}000\ mm = 9{,}000\ mm$

$$r_y = r_z = \sqrt{79.29 \times 10^6 / 6,\ 290} = 112.3 \text{ mm}$$

Geometrical Slenderness Ratio = $L_{ey}/r_y = L_{ez}/r_z = 9{,}000 \ / \ 112.3 = 80.13$

Member slenderness,

$$\lambda_n = \frac{L_e}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 80.13\sqrt{1}\sqrt{\frac{250}{250}} = 80.13$$ [Ref : Clause 6.3.3]

$$\alpha_a = 2,\ 100\frac{\lambda_{nz} - 13.5}{\lambda_{nz}^2 - 15.3\lambda_{nz} + 2,\ 050} = 19.31$$

$\alpha_b = -0.5$ [Ref : Table 6.3.3(2)]

$\lambda = \lambda_n + \alpha_a \times \alpha_b = 70.48$

$\eta = 0.19$

$$\xi_y = \xi_z = \frac{(\lambda_z / 90)^2 + 1 + \eta}{2 \times (\lambda_z / 90)^2} = \frac{(70.48 / 90)^2 + 1 + 0.19}{2 \times (70.48 / 90)^2} = 1.47$$

$$\alpha_{cz} = \xi_z\left[1 + \sqrt{1 - \left(\frac{90}{\xi_z \times \lambda_z}\right)^2}\right] = 0.74$$

The nominal member capacity,

$N_{cy} = N_{cz} = \alpha_{cz} \times N_s = 0.74 \times 1{,}573 = 1{,}171$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = \phi N_{cz} = 1{,}054$ kN

<u>Nominal Section tension Capacity</u>

[Ref : Clause 7.1]

$K_{te} = 1.00$

$N_{t1} = A_g \times f_y = 6{,}290 \times 250 \times 10^{-3} = 1{,}573$ kN

$N_{t2} = 0.85 \times K_{te} \times A_n \times f_u = 0.85(1.0)(6{,}290)(250 \times 10^{-3}) = 1{,}711$ kN

$\phi N_t = 0.9 \times 1{,}573 = 1{,}415$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 697: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕM_{sz} (kN·m)	142.4	142.4352	negligible	
ϕM_{sy} (kN·m)	142.4	142.4352	negligible	
ϕM_{bz} (kN·m)	142.4	142.4352	negligible	
ϕV_y (kN)	348.3	348.022	negligible	
ϕV_z (kN)	509.5	509.16	negligible	
ϕN_s (kN)	1,415	1,414.3	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕN_{cz} (kN)	1,054	1,054	none	
ϕN_{cy} (kN)	1,054	1,054	none	
ϕN_t (kN)	1,415	1,415.3	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Tube Section Non Compact.std` is typically installed with the program.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Tube_Section_Non_Compact.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF TUBE SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 03-Jan-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 4 9 0 0; 5 3 0 0; 6 6 0 0;
*
MEMBER INCIDENCES
1 1 5; 2 5 6; 3 6 4;
DEFINE PMEMBER
1 TO 3 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
UNIT MMS NEWTON
MEMBER PROPERTY INDIAN
1 TO 3 TABLE ST PIP3239H
*
UNIT METER KN
CONSTANTS
MATERIAL STEEL ALL
```

```
*
SUPPORTS
1 PINNED
4 FIXED BUT FX MY MZ
PRINT ALL
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 CON GY -60
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
*
PARAMETER 1
CODE NZS3404 1997
BEAM 1 PMEMB 1
IST 2 PMEMB 1
SGR 6 PMEMB 1
SKL 1 PMEMB 1
SKR 1 PMEMB 1
SKT 1 PMEMB 1
TRACK 2 PMEMB 1
DUCT 3 PMEMB 1
GLD 1 PMEMB 1
CHECK CODE PMEMB ALL
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   PIP3239H                 (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: T.
12.4
   Governing Ratio:   0.977  (PASS)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
```

```
   OD:       323.9000 mm    t:         6.3000 mm
   Ag:      6290.0005 mm2   J:   158.5800E+06 mm4             Iw:     0.0000E
+00 mm6
   Iz:    79.2900E+06 mm4  Sz:   636.0001E+03 mm3 (plastic)   Zz:   489.5957E
+03 mm3 (elastic)
   rz:   112.2752E+00 mm
   Iy:    79.2900E+06 mm4  Sy:   636.0001E+03 mm3 (plastic)   Zy:   489.5957E
+03 mm3 (elastic)
   ry:   112.2752E+00 mm
      STAAD SPACE                                              -- PAGE NO.
13
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS 1163
   Nominal Grade            :  250
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    250.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    250.000 MPa         [NZS3404 Table 2.1]
   fu          :    320.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:      80.160  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   Mz* =  -135.0000E+00 KNm
   Section Slenderness: Noncompact
   Zez =   633.0454E+03 mm3
   φMsz =   142.4352E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My* =     0.0000E+00 KNm
   Section Slenderness: Noncompact
   Zey =   633.0454E+03 mm3
   φMsy =   142.4352E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )-  9.00 m(F )
   Mz* =  -135.0000E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =      9.00 m                       [NZS3404 5.6.3]
   αm   =     1.000                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =     4.9763E+03 KNm                [NZS3404 5.6.1.1.1(d)]
   αsz  =     1.000                         [NZS3404 5.6.1.1.1(c)]
   φMbz =   142.4352E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
```

```
      STAAD SPACE                                               -- PAGE NO.
14
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.086
   Critical Location  :   4.500 m from Start.
   Vy*   =    30.0000E+00 KN
   φVvmy =   348.0229E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vz*   =     0.0000E+00 KN
   φVvmz =   509.1619E+00 KN                [NZS3404 5.12.2]
      STAAD SPACE                                               -- PAGE NO.
15
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Ae    =     6.2859E+03 mm2               [NZS3404 6.2.3 / 6.2.4]
   kf    =     0.999                        [AS 4100 6.2.2]
   An    =     6.2900E+03 mm2
   φNs   =     1.4143E+03 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Lez   =      9.00 m
   αb    =     -0.50                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    80.134                        [NZS3404 6.3.3]
   λ,z   =    70.478                        [NZS3404 6.3.3]
   ε,z   =     1.467                        [NZS3404 6.3.3]
   αc,z  =     0.745                        [NZS3404 6.3.3]
   φNcz  = 0.1054E+4 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )-  9.00 m(U )
   Ley   =      9.00 m
   λn,y  =    80.134                        [NZS3404 6.3.3]
   λ,y   =    70.478                        [NZS3404 6.3.3]
   ε,y   =     1.467                        [NZS3404 6.3.3]
   αc,y  =     0.745                        [NZS3404 6.3.3]
   φNcy  = 0.1054E+4 KN                     [NZS3404 6.3.3]
```

```
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                      [User defined]
   An    =     6.2900E+03 mm2
   φNt   =     1.4153E+03 KN              [NZS3404 7.2]
      STAAD SPACE                                            -- PAGE NO.
16
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   φMrz  =   142.4352E+00 KNm             [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMry  =   142.4352E+00 KNm             [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   γ     =     1.400                       [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   φMiz  =   142.4352E+00 KNm             [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiy  =   142.4352E+00 KNm             [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN              [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm             [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoz,c=     0.0000E+00 KNm             [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
```

```
      STAAD SPACE                                                -- PAGE NO.
17
    *
   SEISMIC PROVISIONS
   ------------------
   Section Slenderness (Bending about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.791
   Critical Location  :   0.000 m from Start.
   λsz              =     51.41             [NZS3404 12.5.1.1]
   λez              =     65.00             [NZS3404 Table 12.5]
   Section Slenderness (Bending about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.791
   Critical Location  :   0.000 m from Start.
   λsy              =     51.41             [NZS3404 12.5.1.1]
   λey              =     65.00             [NZS3404 Table 12.5]
   Max Specific Yield Stress
   Critical Load Case :     1*
   Critical Ratio     :   0.694
   Critical Location  :   0.000 m from Start.
   Fy,actual        =    250.00
   Fy,limit         =    360.00             [NZS3404 Table 12.4(1)]
   Max Actual Yield Ratio (Fy/Fu)
   Critical Load Case :     1*
   Critical Ratio     :   0.977
   Critical Location  :   0.000 m from Start.
   Fy/Fu,actual     =      0.78
   Fy/Fu,limit      =      0.80             [NZS3404 Table 12.4(3)]
   Fabrication Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.4.1.2]
   Section Symmetry Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed              [NZS3404 12.5.2]
   Min Web Thickness Requirement for Beam
   Critical Load Case :     1*
   Critical Ratio     :   0.489
   Critical Location  :   0.000 m from Start.
   tw,actual        =      6.30
   tw,min           =      3.08             [NZS3404 12.7.2]
   Max Axial Force Limit for Column (a)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.80             [NZS3404 Table 12.8.1]
   Max Axial Force Limit for Column (b)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
    b m               =      0.00
   NoL              =     1.9806E+03 KN
   λEYC             =      0.89
```

```
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.26             [NZS3404 12.8.3.1(b)]
   Max Axial Force Limit for Column (c)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Ng*/φNs - actual =      0.00
   Ng*/φNs - limit  =      0.78             [NZS3404 12.8.3.1(c)]
   Shear-Y + Bend-Z Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.948
   Critical Location  :   4.500 m from Start.
   Mz*  =   135.0000E+00 KN
   φMsvz=   142.4352E+00 KN                 [NZS3404 12.10.3.1]
   Shear-Z + Bend-Y Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   My*  =     0.0000E+00 KN
   φMsvy=   142.4352E+00 KN                 [NZS3404 12.10.3.1]

*****************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V. NZS3404 1997-UB Section

Verify the Design Capacity of I Section as per NZS3404 1997.

Details

Verify the section bending capacity of an UB530X92.4. The member is used in a 15 m simply supported span..

Validation

<u>Section Classification</u>

Evaluate the slenderness the beam flanges:

$$\lambda_{ef} = (B/2t_f) \times \sqrt{(f_y/250)} = 198.8/ (2\times15.60)\times\sqrt{(300/250)} = 6.98 < 8$$

Flange is compact.

Evaluate the slenderness of the beam web:

$$\lambda_{ew} = (d/t_w) \times\sqrt{(f_y/250)} = 501.80/10.20\times\sqrt{(300/250)} = 53.89 < 89$$

Web is compact.

Bending Capacity

<u>Section Bending Capacity About Strong Axis</u>

Effective section modulus, $Z_{ez} = 2.37(10)^6\ mm^3$

The nominal section capacity in bending about the Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

$$M_{sz} = 0.9 \times 300 \times 2.37 = 639.9 \text{ kN·m}$$

Section Bending Capacity About Weak Axis

Effective section modulus, $Z_{ey} = 341.6(10)^3\ mm^3$

The nominal section capacity in bending about the Y axis, $M_{sy} = \phi f_y \times Z_{ey}$

$$M_{sy} = 0.9 \times 300 \times 341.6\ (10)^{-3} = 92.24 \text{ kN·m}$$

Member Bending Capacity

End restraint arrangement = FF

A twist restraint factor, $K_t(SKT) = 1.00$

Minor axis rotation restraints = Both

Lateral rotation restraint factor, $K_r(SKR) = 1$

Load Height factor, $K_l,(LHT) = 1.00$ (Table 5.6.3(2) of NZS3404:1997)

Effective length = $1 \times 1 \times 1 \times 15{,}000 = 15{,}000\ mm$

Design Bending Moment, $Mm = 160.8\ kN{\cdot}m$

Quarter Point Moment of segment, $M_2 = -72.443\ kN{\cdot}m$

Mid-Point Moment of segment, $M_3 = -2.267\ kN{\cdot}m$

Quarter Point Moment of segment, $M_4 = 72.499\ kN{\cdot}m$

$$\alpha_m = \frac{1.7M_m^{\times}}{\sqrt{\left(M_1^*\right)^2 + \left(M_2^*\right)^2 + \left(M_3^*\right)^2}} \leq 2.5$$

$$\alpha_m = \frac{1.7 \times 160.8}{\sqrt{(-72.44)^2 + (-2.267)^2 + (72.5)^2}} = 2.66$$

Therefore, $\alpha_m = 2.5$

Reference Buckling Moment, M_o

$$M_o = \sqrt{\left(\frac{\pi^2 EI_y}{L_e^2}\right)\left[GJ + \left(\frac{\pi^2 EI_y}{L_e^2}\right)\right]} = 127.0 \text{ kN·m}$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\} = 0.157$$ [Ref. cl 5.6.1.1(c)]

$$M_{bz} = \alpha_m \alpha_s M_{sx} \leq M_{sx} = 2.5 \times 0.157 \times 71 = 279.1 \text{ kN·m}$$

$$\phi M_{bz} = 251.2 \text{ kN·m}$$

Check for Shear

Shear Area of the section, $A_y = d \times t_w = 533.0 \times 10.2 = 5{,}436.6\ mm^2$

Section Shear Capacity (Along Y axis), $V_y = 0.6 \times f_y \times A_y = 0.6 \times 320 \times 5{,}436.6 = 1{,}044\ kN$

$ZV_y = 0.9 \times 1{,}044 = 939.4$ kN

Shear Area of the section, $A_Z = 2 \times b_f \times t_f = 2 \times 209 \times 15.6 = 6{,}520.8\ mm^2$

Section Shear Capacity (Along z axis), $Vz = 0.6 \times fy \times Az = 0.6 \times 300 \times 6{,}520.8 = 1{,}174\ kN$

ϕV_z = 1,056 kN

Check for Axial Compression

<u>Section Compression Capacity</u>

The flange slenderness, $\lambda_{eb} = 6.98$ [Ref : Cl no - 6.2.3.1]

Yield slender for flange, $\lambda_{eby} = 16.00$ [Ref : Table 6.2.4]

The web slenderness, $\lambda_{ew} = 55.66$

Yield slender for web, $\lambda_{eby} = 45$

Effective Width, b_e=$209\ mm$

Effective Depth, DE=$405.7\ mm$

Gross Area, $A_g = 11{,}800\ mm^2$

Net Area, $A_n = 10{,}820\ mm^2$

Form factor, $K_f = A_e / A_g = 0.917$

The nominal member section capacity for axial compression ,

$N_s = K_f \times A_n \times f_y = 0.917 \times 10{,}819.8 \times 300 = 3{,}246$ kN [Ref : Cl no - 6.2.1]

$\phi N_s = 0.9 \times 3{,}246 = 2{,}920$ kN [Ref : Cl no - 6.2.1]

<u>Member Compression Capacity</u>

Effective length factor for slenderness & buckling about minor Y- axis, $Ky = 1.00$

Effective length factor for slenderness & buckling about minor Z- axis, $Kz = 1.00$

Effective Length of member, $L_{ez} = 15{,}000\ mm$

Effective Length of member, $L_{ey} = 15{,}000\ mm$

Geometrical Slenderness Ratio = 69.23

Geometrical Slenderness Ratio = 334.0

Member slenderness, $\lambda_{nz} = (L_e / r) \times \sqrt{(k_f)} \times \sqrt{(f_y / 250)}$ [Ref : Cl no - 6.3.3.]

$\lambda nz = 69.23 \times \sqrt{1} \times \sqrt{(300/250)} = 72.62$

Member slenderness, $\lambda_{ny} = (L_{ey} / r) \times \sqrt{(k_f)} \times \sqrt{(f_y / 250)}$ [Ref : Cl no - 6.3.3]

$\lambda_{ny} = 334 \times \sqrt{1} \times \sqrt{(300/250)} = 350.35$

$\alpha_{az} = 2{,}100 \times (\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2{,}050) = 19.984$

$\alpha_{ay} = 2{,}100 \times (\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2{,}050) = 5.923$

$\alpha_b = 0.00$ [Ref : table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{ax}\alpha_b = 72.62$

$\lambda_y = \lambda_{ny} + \alpha_{ay}\alpha_b = 350.35$

$\eta = 0.19$

$\eta = 1.10$

$\xi_z = ((\lambda_z/90)2+1+\eta)/(2 \times (\lambda_z/90)2) = 1.42$

$\xi_y = ((\lambda_y/90)2+1+\eta)/(2\times(\lambda_y/90)2) = 0.57$

α_{cz}=0.731 [Ref : Cl no - 6.3.3]

α_{cy}=0.061 [Ref : Cl no - 6.3.3]

The nominal member capacity, $N_{cz}= \alpha_{cz} \times N_s$ [Ref : Cl no - 6.3.3]

$N_{cz}= \alpha_{cz} \times N_s$ =0.731×3,246 =2,373 kN

ϕNcz= 2,136 kN

The nominal member capacity, $N_{cy}= \alpha_{cy} \times N_s$ [Ref : Cl no - 6.3.3]

$N_{cy}= \alpha_{cy} \times N_s$ =0.061×3,246 = 198.9 kN

ϕN_{cy}= 178.96 KN

Nominal Section Tension Capacity

Ref. Cl 7.1

K_{te}= 1.00

Nt1 = $A_g \times f_y$ = 3,540 kN

Nt2 = $0.85\times K_{te}\times A_n\times f_u$ = 4,413 kN

ϕNt = 3,186 kN [Ref : Cl no -5.6.1.1.1(a)]

Results

Table 698: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕMsz (kN·m)	639.9	639.9001	negligible	
ϕMsy (kN·m)	92.24	92.2392	negligible	
ϕMbz (kN·m)	251.2	251.3427	negligible	
ϕVy (kN)	939.4	939.4444	negligible	
ϕVz (kN)	1,056	1,056.4	negligible	
ϕNs (kN)	2,920	2,921.3	negligible	
ϕNcz (kN)	2,136	2,136	none	
ϕNcy (kN)	178.96	179.0	negligible	
ϕNt (kN)	3,186	3,186.0	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\New Zealand\NZS3404 1997-UB Section.std` is typically installed with the program.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Frame.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  UB,UC SHAPE  PER
*              THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 16-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 5 0; 4 5 0 0; 5 10 5 0; 6 10 0 0; 7 0 10 0; 8 5 10 0;
9 10 10 0; 10 0 15 0; 11 5 15 0; 12 10 15 0; 13 0 0 5; 14 0 5 5; 15 5 5 5;
16 5 0 5; 17 10 5 5; 18 10 0 5; 19 0 10 5; 20 5 10 5; 21 10 10 5; 22 0 15 5;
23 5 15 5; 24 10 15 5; 25 0 0 10; 26 0 5 10; 27 5 5 10; 28 5 0 10; 29 10 5 10;
30 10 0 10; 31 0 10 10; 32 5 10 10; 33 10 10 10; 34 0 15 10; 35 5 15 10;
36 10 15 10; 37 0 0 15; 38 0 5 15; 39 5 5 15; 40 5 0 15; 41 10 5 15;
42 10 0 15; 43 0 10 15; 44 5 10 15; 45 10 10 15; 46 0 15 15; 47 5 15 15;
48 10 15 15; 49 5 10 18;
*
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 3 5; 5 5 6; 6 2 7; 7 3 8; 8 5 9; 9 7 8; 10 8 9; 11 7
10;
12 8 11; 13 9 12; 14 10 11; 15 11 12; 16 2 14; 17 3 15; 18 5 17; 19 7 19;
20 8 20; 21 9 21; 22 10 22; 23 11 23; 24 12 24; 25 13 14; 26 14 15; 27 15 16;
28 15 17; 29 17 18; 30 14 19; 31 15 20; 32 17 21; 33 19 20; 34 20 21; 35 19
22;
36 20 23; 37 21 24; 38 22 23; 39 23 24; 40 14 26; 41 15 27; 42 17 29; 43 19
31;
44 20 32; 45 21 33; 46 22 34; 47 23 35; 48 24 36; 49 25 26; 50 26 27; 51 27
28;
52 27 29; 53 29 30; 54 26 31; 55 27 32; 56 29 33; 57 31 32; 58 32 33; 59 31
34;
60 32 35; 61 33 36; 62 34 35; 63 35 36; 64 26 38; 65 27 39; 66 29 41; 67 31
43;
68 32 44; 69 33 45; 70 34 46; 71 35 47; 72 36 48; 73 37 38; 74 38 39; 75 39
40;
76 39 41; 77 41 42; 78 38 43; 79 39 44; 80 41 45; 81 43 44; 82 44 45; 83 43
46;
84 44 47; 85 45 48; 86 46 47; 87 47 48; 88 44 49;
DEFINE PMEMBER
22 46 70 PMEMBER 1
19 43 67 PMEMBER 2
16 40 64 PMEMBER 3
20 44 68 PMEMBER 4
17 41 65 PMEMBER 5
24 48 72 PMEMBER 6
```

```
21 45 69 PMEMBER 7
18 42 66 PMEMBER 8
1 6 11 PMEMBER 9
25 30 35 PMEMBER 10
49 54 59 PMEMBER 11
73 78 83 PMEMBER 12
88 PMEMBER 13
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
2 4 9 10 14 TO 24 26 28 33 34 38 TO 48 50 52 57 58 62 TO 72 74 76 81 82 86 -
87 TO 88 TABLE ST UB530X92.4
1 3 5 TO 8 11 TO 13 25 27 29 TO 32 35 TO 37 49 51 53 TO 56 59 TO 61 73 75 -
77 TO 80 83 TO 85 TABLE ST UC310X158
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 4 6 13 16 18 25 28 30 37 40 42 FIXED
*
MEMBER RELEASE
2 4 9 10 14 TO 24 26 28 33 34 38 39 50 52 57 58 62 63 74 76 81 82 86 -
87 START MY MZ
2 4 9 10 14 15 26 28 33 34 38 39 50 52 57 58 62 TO 66 68 69 71 72 74 76 81 -
82 86 87 END MY MZ
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1
JOINT LOAD
23 49 FY -200
46 49 FX 200
36 49 FZ -200
MEMBER LOAD
20 44 68 88 UNI Y -50
26 88 CMOM GZ -30
88 CON GZ 20
*
PERFORM ANALYSIS
*
PRINT ANALYSIS RESULTS
*
PRINT MEMBER FORCES
*
PARAMETER 1
CODE NZS3404 1997
BEAM 1 PMEMB 5 13
```

```
DMAX 1.5 PMEMB 5 13
DMIN 0.4 PMEMB 5 13
IST 2 PMEMB 5 13
LHT 1 PMEMB 5 13
NSC 1 PMEMB 5 13
NSF 1 PMEMB 5 13
RATIO 1 PMEMB 5 13
SGR 0 PMEMB 5 13
SKL 1 PMEMB 5 13
SKR 1 PMEMB 5 13
SKT 1 PMEMB 5 13
TMAIN 400 PMEMB 5 13
TRACK 2 PMEMB 5 13
TSP 0 PMEMB 5 13
DUCT 1 PMEMB 5
GLD 1 PMEMB 5
CHECK CODE PMEMB 5
*
FINISH
```

STAAD.Pro Output

```
 STEEL DESIGN
   NOTE : SGR NOT SPECIFIED OR "DEFAULT" SPECIFIED FOR PMEMBER NO.      5.
   NOTE : BY DEFAULT "AS/NZS 3679.1 300" WILL BE USED FOR ROLLED SECTIONS.
      STAAD SPACE                                              -- PAGE NO.
21
    *
                     STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                   **************************************************
   AXIS NOTATION FOR ANY SECTION OTHER THAN ST ANGLE:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ----------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             Y/y          Minor principal axis of section
      Z/z             X/x          Major Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     5
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   UB530X92.4               (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
12.8.3.1.1
   Governing Ratio:  14.910 *(FAIL)
   Governing Location:  10.015 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       532.9999 mm   bf:       209.0000 mm
```

```
   tf:        15.6000 mm   tw:        10.2000 mm
   Ag:     11800.0000 mm2   J:   775.0001E+03 mm4             Iw:     1.5886E
+12 mm6
   Iz:   554.0001E+06 mm4  Sz:     2.3700E+06 mm3 (plastic)   Zz:     2.0788E
+06 mm3 (elastic)
   rz:   216.6776E+00 mm
   Iy:    23.8000E+06 mm4  Sy:   355.0000E+03 mm3 (plastic)   Zy:   227.7512E
+03 mm3 (elastic)
   ry:    44.9105E+00 mm
      STAAD SPACE                                              -- PAGE NO.
22
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    300.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     333.998  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  180.000
   BENDING
   -------
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.257
   Critical Location  :   5.015 m from Start.
   Mz* =   164.5103E+00 KNm
   Section Slenderness: Compact
   Zez =     2.3700E+06 mm3
   φMsz =   639.9001E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.291
   Critical Location  :  10.015 m from Start.
   My* =   -26.8162E+00 KNm
   Section Slenderness: Compact
   Zey =   341.6269E+03 mm3
   φMsy =    92.2392E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.655
   Critical Location  :   5.015 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(F )- 15.00 m(F )
   Mz* =   164.5103E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      1.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =     15.00 m                       [NZS3404 5.6.3]
   αm   =     2.500                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =   127.0231E+00 KNm                [NZS3404 5.6.1.1.1(d)]
   αsz  =     0.157                         [NZS3404 5.6.1.1.1(c)]
   φMbz =   251.3427E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
```

```
      STAAD SPACE                                              -- PAGE NO.
23
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.071
   Critical Location  :   5.015 m from Start.
   Vy*   =    66.8261E+00 KN
   φVvmy =   939.4444E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.002
   Critical Location  :  10.015 m from Start.
   Vz*   =     2.4943E+00 KN
   φVvmz =     1.0564E+03 KN                [NZS3404 5.12.2]
      STAAD SPACE                                              -- PAGE NO.
24
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.039
   Critical Location  :  10.015 m from Start.
   N*    =   114.4841E+00 KN
   Ae    =    10.8198E+03 mm2              [NZS3404 6.2.3 / 6.2.4]
   kf    =     0.917                       [AS 4100 6.2.2]
   An    =    11.8000E+03 mm2
   φNs   =     2.9213E+03 KN                [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.054
   Critical Location  :  10.015 m from Start.
   N*    =   114.4841E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )- 15.00 m(U )
   Lez   =     15.00 m
   αb    =      0.00                        [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =    72.617                        [NZS3404 6.3.3]
   λ,z   =    72.617                        [NZS3404 6.3.3]
   ε,z   =     1.416                        [NZS3404 6.3.3]
   αc,z  =     0.731                        [NZS3404 6.3.3]
   φNcz  = 0.2136E+4 KN                     [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.640
   Critical Location  :  10.015 m from Start.
   N*    =   114.4841E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(U )- 15.00 m(U )
   Ley   =     15.00 m
   λn,y  =   350.351                        [NZS3404 6.3.3]
   λ,y   =   350.351                        [NZS3404 6.3.3]
   ε,y   =     0.569                        [NZS3404 6.3.3]
   αc,y  =     0.061                        [NZS3404 6.3.3]
   φNcy  = 0.1790E+3 KN                     [NZS3404 6.3.3]
```

```
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                       [User defined]
   An    =    11.8000E+03 mm2
   φNt   =     3.1860E+03 KN               [NZS3404 7.2]
      STAAD SPACE                                            -- PAGE NO.
25
    *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.257
   Critical Location  :   5.015 m from Start.
   φMrz  =   639.9001E+00 KNm              [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.291
   Critical Location  :  10.015 m from Start.
   φMry  =    92.2392E+00 KNm              [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.271
   Critical Location  :   9.985 m from Start.
   γ     =     1.426                        [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.267
   Critical Location  :   5.015 m from Start.
   φMiz  =   616.9343E+00 KNm              [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.806
   Critical Location  :  10.015 m from Start.
   φMiy  =    33.2658E+00 KNm              [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoz  =     0.0000E+00 KN               [NZS3404 8.4.4.1.2]
   φMoz,t=     0.0000E+00 KNm              [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   1.150
   Critical Location  :   5.015 m from Start.
   φMoz,c=   143.1029E+00 KNm              [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   1.918
   Critical Location  :  10.015 m from Start.
```

```
      STAAD SPACE                                                   -- PAGE NO.
26
    *
   SEISMIC PROVISIONS
   ------------------
   Section Slenderness (Bending about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.776
   Critical Location  :   0.000 m from Start.
   λsz             =      6.98             [NZS3404 12.5.1.1]
   λez             =      9.00             [NZS3404 Table 12.5]
   Section Slenderness (Bending about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.776
   Critical Location  :   0.000 m from Start.
   λsy             =      6.98             [NZS3404 12.5.1.1]
   λey             =      9.00             [NZS3404 Table 12.5]
   Max Specific Yield Stress
   Critical Load Case :     1*
   Critical Ratio     :   0.833
   Critical Location  :   0.000 m from Start.
   Fy,actual        =    300.00
   Fy,limit         =    360.00             [NZS3404 Table 12.4(1)]
   Max Actual Yield Ratio (Fy/Fu)
   Critical Load Case :     1*
   Critical Ratio     :   0.852
   Critical Location  :   0.000 m from Start.
   Fy/Fu,actual     =      0.68
   Fy/Fu,limit      =      0.80             [NZS3404 Table 12.4(3)]
   Fabrication Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed             [NZS3404 12.4.1.2]
   Section Symmetry Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed             [NZS3404 12.5.2]
   Min Web Thickness Requirement for Beam
   Critical Load Case :     1*
   Critical Ratio     :   0.679
   Critical Location  :   0.000 m from Start.
   tw,actual        =     10.20
   tw,min           =      6.92             [NZS3404 12.7.2]
   Max Axial Force Limit for Column (a)
   Critical Load Case :     1*
   Critical Ratio     :   0.078
   Critical Location  :  10.015 m from Start.
   N*/φNs - actual  =      0.04
   N*/φNs - limit   =      0.50             [NZS3404 Table 12.8.1]
   Max Axial Force Limit for Column (b)
   Critical Load Case :     1*
   Critical Ratio     :  14.910
   Critical Location  :  10.015 m from Start.
    b m               =      0.00
   NoL              =   214.0169E+00 KN
   λEYC             =      3.89
```

```
   N*/φNs - actual  =      0.04
   N*/φNs - limit   =      0.00             [NZS3404 12.8.3.1(b)]
   Max Axial Force Limit for Column (c)
   Critical Load Case :     1*
   Critical Ratio     :   0.202
   Critical Location  :  10.015 m from Start.
   Ng*/φNs - actual =      0.04
   Ng*/φNs - limit  =      0.19             [NZS3404 12.8.3.1(c)]
   Shear-Y + Bend-Z Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.253
   Critical Location  :   5.015 m from Start.
   Mz*  =  -161.9652E+00 KN
   φMsvz=   639.9001E+00 KN                 [NZS3404 12.10.3.1]
   Shear-Z + Bend-Y Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.151
   Critical Location  :   9.985 m from Start.
   My*  =   -13.9665E+00 KN
   φMsvy=    92.2392E+00 KN                 [NZS3404 12.10.3.1]

******************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V.NZS3404 1997-Unequal Angle Section

Verify the design capacity of an A125x75x8 section as per the NZS3404 1997 code.

Details

Verify the section capacity of an A125x75x8 section used for a 5 m cantilever span. Steel grade = 320 MPa.

Validation

<u>Section Classification</u>

Evaluate the slenderness effects of the beam flanges:

$$\lambda_{ef} = \frac{B}{2t_f}\sqrt{\frac{f_y}{250}} = \frac{117.2}{7.8}\sqrt{\frac{320}{250}} = 17.0$$

Section flange classification is compact.

Evaluate the slenderness effects of the beam web:

$$\lambda_{ew} = \frac{d}{t_w}\sqrt{\frac{f_y}{250}} = \frac{67.2}{7.8}\sqrt{\frac{320}{250}} = 9.75$$

Section web classification is compact

<u>Section Bending Capacity About Z-Axis</u>

Effective Section Modulus, $Z_{ez} = 12{,}720\ mm^3$

The nominal section capacity in bending about Z axis, $M_{sz} = \phi f_y \times Z_{ez}$

M_{sz} = 320× 12,720 ×10^{-6} = 4.07 kN·m

ϕM_{sz} = 0.9×4.07 = 3.66 kN·m

<u>Section Bending Capacity About Y-Axis</u>

Effective Section Modulus, Z_{ey} = *40,550 mm*^{3}

The nominal section capacity in bending about Z axis, $M_{sy} = \phi f_y \times Z_{ey}$

M_{sy} = 320× 40,550×10^{-6} = 12.98 kN·m

ϕM_{sy} = 0.9×12.98 = 11.68 kN·m

<u>Member Bending Capacity</u>

End restraint arrangement = FU

A twist restraint factor, K_t (SKT) = 1.00

Minor axis rotation restraints = Fu

Lateral rotation restraint factor, K_r (SKR) = 0.70

Load Height factor, Kl, = 2.0 *[Ref : Table 5.6.3(2)]*

Effective length = 1×1×2×5,000 = 10,000 mm

α_m = 1.25

Reference buckling moment, M_o

$$M_o = \sqrt{\frac{\pi^2 E I_y}{L_e^2}\left[GJ + \left(\frac{\pi^2 E I_w}{L_e^2}\right)\right]} = 4.43 \text{ kN·m}$$

$$\alpha_s = 0.6\left\{\sqrt{\left[\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{sx}}{M_{oa}}\right)\right\} = 0.284$$ [Ref : Clause 5.6.1.1 (c)]

$M_{bx} = \alpha_m \alpha_s M_{sx} \le M_{sx}$

M_{bz} = 1.25 × 0.284 × 12.98 = 4.61 kN·m ≤ $(M_{sz}, M_{sy})_{Max.}$ [Ref : Clause 5.6.1.1.1(a)]

ϕM_{bz} = 0.9×4.61 = 4.15 kN·m

<u>Check for Shear</u>

Shear Area of the section, $A_y = d \times t$ = *125×7.8 = 975 mm*^{2}

Section Shear Capacity (Along Y axis), $V_y = 0.6 \times f_y \times A_y$ = *0.6×320×975 = 187 kN*

V_{vn} = 2×187/(0.9 + 1.2) = 178 kN [Ref : Clause 5.11.2]

ϕV_y = 0.9×178 = 133.2 kN

Shear Area of the section, $A_Z = b \times t$ = *75×7.8 = 585 mm*^{2}

Section Shear Capacity (Along z axis), $V_z = 0.6 \times f_y \times A_z$ = *0.6×320×585 = 112.3 kN*

V_{vn} = 2×112.3/(0.9 + 1.2) = 107 kN

ϕV_z = 0.9×107 = 96.3 kN

<u>Check for Axial Compression</u>

Section Compression Capacity:

Gross Area, A_g = *1,500 mm*^{2}

Net Area, $A_n = 1,500\ mm^2$

Form factor, $K_f = A_e / A_g = 1.0$

The nominal member section capacity for axial compression,

$N_s = K_f \times A_n \times f_y = 1.0\times1,500\times320 = 480$ kN [Ref : Clause 6.2.1]

$\phi N_s = 0.9\times480 = 432$ kN

Member Compression Capacity

Length of the member, $L = 5,000\ mm$

Effective length factor for slenderness & buckling about minor Y- axis, $K_y = 2.2$

Effective length factor for slenderness & buckling about minor Z- axis, $K_z = 2.2$

Effective Length of member, $L_{ez} = 2.2\times5,000\ mm = 11,000\ mm$

Effective Length of member, $L_{ey} = 2.2\times5,000\ mm = 11,000\ mm$

$r_y = \sqrt{}(2.72\times10^6 / 1,500) = 42.6$

$r_z = \sqrt{}(398\times10^3 / 1,500) = 16.3$

Geometrical Slenderness Ratio = $L_{ez}/r_z = 11,000\ /\ 16.3 = 674.9$

Geometrical Slenderness Ratio = $L_{ey}/r_y = 11,000\ /\ 42.6 = 258.3$

Member slenderness,

$$\lambda_{nz} = \frac{L_{ez}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 674.9\sqrt{1}\sqrt{\frac{320}{250}} = 763.5$$ [Ref : Clause 6.3.3]

$$\lambda_{ny} = \frac{L_{ey}}{r}\sqrt{k_f}\sqrt{\frac{f_y}{250}} = 258.3\sqrt{1}\sqrt{\frac{320}{250}} = 291.9$$ [Ref : Clause 6.3.3]

$\alpha_{az} = 2,100\times(\lambda_{nz} - 13.5)/(\lambda_{nz}^2 - 15.3\lambda_{nz} + 2,050) = 2.747$

$\alpha_{ay} = 2,100\times(\lambda_{ny} - 13.5)/(\lambda_{ny}^2 - 15.3\lambda_{ny} + 2,050) = 7.061$

$\alpha_b = 0.5$ [Ref : Table 6.3.3(2)]

$\lambda_z = \lambda_{nz} + \alpha_{az}\times\alpha_z = 764.9$

$\lambda_y = \lambda_{ny} + \alpha_{ay}\times\alpha_b = 295.5$

$\eta = 2.45$

$\eta = 0.92$

$\xi_z = ((\lambda_z/90)^2 + 1 + \eta)/(2\times(\lambda_z/90)^2) = 0.52$

$\xi_y = ((\lambda_y/90)^2 + 1 + \eta)/(2\times(\lambda_y/90)^2) = 0.59$

$\alpha_{cz} = 0.013$

$\alpha_{cy} = 0.085$

The nominal member capacity,

$N_{cz} = \alpha_{cz}\times N_s = 0.013\times480 = 6.42$ kN [Ref : Clause 6.3.3]

$\phi N_{cz} = 5.78$ kN

The nominal member capacity,

$N_{cy} = \alpha_{cy} \times N_s = 0.085 \times 480 = 40.7$ kN [Ref : Clause 6.3.3]

$\phi N_{cy} = 36.66$ kN

Nominal Section tension Capacity

[Ref : Clause 7.1]

$K_{te} = 1.00$

$N_{t1} = A_g \times f_y = 480$ kN

$N_{t2} = 0.85 \times K_{te} \times A_n \times f_u = 516$ kN

$\phi N_t = 0.9 \times 480 = 432$ kN [Ref : Clause 5.6.1.1.1(a)]

Results

Table 699: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕMsz(KN·m)	3.66	3.6625	negligible	
ϕMsy(KN·m)	11.68	11.6789	negligible	
ϕMbz (KN-m)	4.15	4.1237	negligible	
ϕVz (KN)	133.2	133.18	negligible	
ϕVy(KN)	96.3	96.2743	negligible	
ϕNs(KN)	432	432	none	
ϕNcz (KN)	5.78	5.78	none	
ϕNcy (KN)	36.66	36.66	none	
ϕNt (KN)	432	432	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\New Zealand\NZS3404 1997-Unequal Angle Section.std` is typically installed with the program.

The following D11.A.6 Design Parameters (on page 1776) are used in this example:

- The load height position is at the top flange: `LHT 1`.

```
STAAD SPACE
*
*  INPUT FILE: NZS3404_Unequal_Angle_section.STD
*
* REFERENCE : Hand Calculation
*
*  OBJECTIVE : TO DETERMINE THE ADEQUACY OF  UNEQUAL ANGLE  SHAPE  PER
```

```
*               THE NZS3404-1997 CODE
*
START JOB INFORMATION
ENGINEER DATE 13-Feb-17
END JOB INFORMATION
INPUT WIDTH 79
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
*
MEMBER INCIDENCES
1 1 2;
DEFINE PMEMBER
1 PMEMBER 1
*
*
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
*
MEMBER PROPERTY AUSTRALIAN
1 TABLE ST A125X75X8
*
CONSTANTS
MATERIAL STEEL ALL
*
SUPPORTS
1 FIXED
*
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FZ 2
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE NZS3404 1997
LHT 1 PMEMB 1
TRACK 2 PMEMB 1
PBCRES ZZ 0 T 1 U PMEMB 1
PBCRES YY 0 T 1 U PMEMB 1
PBRACE TOP 0 FR 1 U PMEMB 1
PBRACE BOTTOM 0 FR 1 U PMEMB 1
DUCT 1 PMEMB 1
GLD 1 PMEMB 1
CHECK CODE PMEMB 1
*
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - NZS-3404-1997 (v1.0)
                     **************************************************
   AXIS NOTATION FOR ST ANGLE SECTION:-
   STAAD.Pro     NZS3404 Spec.     Description
   ---------     -------------     ---------------
      X/x             Z/z          Longitudinal axis of section
      Y/y             X/x          Major principal axis of section
      Z/z             Y/y          Minor Principal axis of section
   MEMBER DESIGN OUTPUT FOR PMEMBER     1
   DESIGN Notes
   ------------
   1. (*) next to a Load Case number signifies that a P-Delta analysis has
not been performed for
      that particular Load Case; i.e. analysis does not include second-order
effects.
   2. φ = 0.9 for all the calculations [NZS3404 Table 3.4]
   3. (#) next to Young's modulus E indicates that its value is not 200000
MPa as per NZS3404 1.4.
   DESIGN SUMMARY
   --------------
   Designation: ST   A125X75X8                (AISC SECTIONS)
   Governing Load Case:     1*
   Governing Criteria: Cl.
5.1
   Governing Ratio:   2.425 *(FAIL)
   Governing Location:   0.000 m from Start.
   SECTION PROPERTIES
   ------------------
    d:       125.0000 mm    b:        75.0000 mm
    t:         7.8000 mm
   Ag:      1500.0000 mm2   J:    30.4200E+03 mm4             Iw:    28.1486E
+06 mm6
   Iz:   398.5350E+03 mm4  Sz:    20.2467E+03 mm3 (plastic)   Zz:    32.4411E
+03 mm3 (elastic)
   rz:    16.3000E+00 mm
   Iy:     2.7259E+06 mm4  Sy:    55.9491E+03 mm3 (plastic)   Zy:    13.3505E
+03 mm3 (elastic)
   ry:    42.6290E+00 mm
      STAAD SPACE                                               -- PAGE NO.
4
    *
   MATERIAL PROPERTIES
   -------------------
   Material Standard        :  AS/NZS 3679.1
   Nominal Grade            :  300
   Residual Stress Category :  HR (Hot-rolled)
   E (#)       : 204999.984 MPa         [NZS3404 1.4]
   G           :  80000.000 MPa         [NZS3404 1.4]
   fy, flange  :    320.000 MPa         [NZS3404 Table 2.1]
   fy, web     :    320.000 MPa         [NZS3404 Table 2.1]
   fu          :    440.000 MPa         [NZS3404 Table 2.1]
  SLENDERNESS:   ACTUAL SLENDERNESS RATIO:     306.748  LOAD:     1   LOC.
(MET):   0.000
                 ALLOWABLE SLENDERNESS RATIO:  400.000
   BENDING
   -------
```

```
   Section Bending Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Mz* =     0.0000E+00 KNm
   Section Slenderness: Noncompact
   Zez =    12.7170E+03 mm3
   φMsz =     3.6625E+00 KNm                [NZS3404 Cl.5.1    ]
   Section Bending Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.856
   Critical Location  :   0.000 m from Start.
   My* =   -10.0000E+00 KNm
   Section Slenderness: Noncompact
   Zey =    40.5518E+03 mm3
   φMsy =    11.6789E+00 KNm                [NZS3404 Cl.5.1    ]
   Member Bending Capacity
   Critical Load Case :     1*
   Critical Ratio     :   2.425
   Critical Location  :   0.000 m from Start.
   Crtiical Flange Segment:
   Location (Type):   0.00 m(FR)-  5.00 m(U )
   Mz* =    10.0000E+00 KNm
   kt   =      1.00                         [NZS3404 Table 5.6.3(1)]
   kl   =      2.00                         [NZS3404 Table 5.6.3(2)]
   kr   =      1.00                         [NZS3404 Table 5.6.3(3)]
   le   =     10.00 m                       [NZS3404 5.6.3]
   αm   =     1.250                         [NZS3404 5.6.1.1.1(b)(iii)]
   Mo   =     4.3977E+00 KNm                [NZS3404 5.6.1.1.1(d)]
   αsy  =     0.282                         [NZS3404 5.6.1.1.1(c)]
   φMby =     4.1237E+00 KNm (&lt;= φMsz)      [NZS3404 5.6.1.1.1(a)]
      STAAD SPACE                                           -- PAGE NO.
5
    *
   SHEAR
   -----
   Section Shear Capacity (along Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Vy*   =     0.0000E+00 KN
   φVvmy =    96.2743E+00 KN                [NZS3404 5.12.2]
   Section Shear Capacity (along Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.015
   Critical Location  :   0.000 m from Start.
   Vz*   =     2.0000E+00 KN
   φVvmz =   133.1808E+00 KN                [NZS3404 5.12.2]
      STAAD SPACE                                           -- PAGE NO.
6
    *
   AXIAL
   -----
   Section Compression Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
```

```
   Ae    =     1.5000E+03 mm2                [NZS3404 6.2.3 / 6.2.4]
   kf    =     1.000                         [AS 4100 6.2.2]
   An    =     1.5000E+03 mm2
   φNs   =   432.0000E+00 KN                 [NZS3404 6.2.1]
   Member Compression Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(T )-  5.00 m(U )
   Lez   =     11.00 m
   αb    =      0.50                         [NZS3404 Table 6.3.3(1)/6.3.3(2)]
   λn,z  =   763.502                         [NZS3404 6.3.3]
   λ,z   =   764.875                         [NZS3404 6.3.3]
   ε,z   =     0.524                         [NZS3404 6.3.3]
   αc,z  =     0.013                         [NZS3404 6.3.3]
   φNcz  = 0.5782E+1 KN                      [NZS3404 6.3.3]
   Member Compression Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   Unbraced Segment:
   Location (Type):   0.00 m(T )-  5.00 m(U )
   Ley   =     11.00 m
   λn,y  =   291.939                         [NZS3404 6.3.3]
   λ,y   =   295.469                         [NZS3404 6.3.3]
   ε,y   =     0.589                         [NZS3404 6.3.3]
   αc,y  =     0.085                         [NZS3404 6.3.3]
   φNcy  = 0.3666E+2 KN                      [NZS3404 6.3.3]
   Section Tension Capacity
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*    =     0.0000E+00 KN
   kt    =      1.00                         [User defined]
   An    =     1.5000E+03 mm2
   φNt   =   432.0000E+00 KN                 [NZS3404 7.2]
      STAAD SPACE                                                   -- PAGE NO.
7
  *
   COMBINED BENDING AND AXIAL
   ------------------------
   Section Combined Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMrz  =     3.6625E+00 KNm                [NZS3404 8.3.2]
   Section Combined Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.856
   Critical Location  :   0.000 m from Start.
   φMry  =    11.6789E+00 KNm                [NZS3404 8.3.3]
   Section Combined Capacity (Biaxial)
   Critical Load Case :     1*
   Critical Ratio     :   0.856
   Critical Location  :   0.000 m from Start.
```

```
   γ     =     1.400                         [NZS3404 8.3.4]
   Member In-plane Capacity (about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMiz  =     3.6625E+00 KNm               [NZS3404 8.4.2]
   Member In-plane Capacity (about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   0.856
   Critical Location  :   0.000 m from Start.
   φMiy  =    11.6789E+00 KNm               [NZS3404 8.4.2]
   Member Out-of-plane Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   αbc   =      0.00
   φNoy  =     0.0000E+00 KN                [NZS3404 8.4.4.1.2]
   φMoy,t=     0.0000E+00 KNm               [NZS3404 8.4.4.1]
   Member Out-of-plane Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   φMoy,c=     0.0000E+00 KNm               [NZS3404 8.4.4.2]
   Member Biaxial Capacity (Tension)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Member Biaxial Capacity (Compression)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
      STAAD SPACE                                             -- PAGE NO.
8
    *
   SEISMIC PROVISIONS
   ------------------
   Section Slenderness (Bending about Z-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.889
   Critical Location  :   0.000 m from Start.
   λsz             =     17.00              [NZS3404 12.5.1.1]
   λez             =      9.00              [NZS3404 Table 12.5]
   Section Slenderness (Bending about Y-axis)
   Critical Load Case :     1*
   Critical Ratio     :   1.083
   Critical Location  :   0.000 m from Start.
   λsy             =     17.00              [NZS3404 12.5.1.1]
   λey             =      9.00              [NZS3404 Table 12.5]
   Max Specific Yield Stress
   Critical Load Case :     1*
   Critical Ratio     :   0.889
   Critical Location  :   0.000 m from Start.
   Fy,actual       =    320.00
   Fy,limit        =    360.00              [NZS3404 Table 12.4(1)]
   Max Actual Yield Ratio (Fy/Fu)
   Critical Load Case :     1*
   Critical Ratio     :   0.909
   Critical Location  :   0.000 m from Start.
```

```
   Fy/Fu,actual     =      0.73
   Fy/Fu,limit      =      0.80              [NZS3404 Table 12.4(3)]
   Fabrication Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed               [NZS3404 12.4.1.2]
   Section Symmetry Requirement
   Critical Load Case : N/A
   Critical Ratio     : N/A
   Critical Location  : N/A
   Status           =   Passed               [NZS3404 12.5.2]
   Min Web Thickness Requirement for Beam
   Critical Load Case :     1*
   Critical Ratio     :   0.207
   Critical Location  :   0.000 m from Start.
   tw,actual        =      7.80
   tw,min           =      1.62              [NZS3404 12.7.2]
   Max Axial Force Limit for Column (a)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.50              [NZS3404 Table 12.8.1]
   Max Axial Force Limit for Column (b)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
    b m               =      0.50
   NoL              =   220.6053E+00 KN
   λEYC             =      1.48
   N*/φNs - actual  =      0.00
   N*/φNs - limit   =      0.20              [NZS3404 12.8.3.1(b)]
   Max Axial Force Limit for Column (c)
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Ng*/φNs - actual =      0.00
   Ng*/φNs - limit  =      1.00              [NZS3404 12.8.3.1(c)]
   Shear-Y + Bend-Z Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.000
   Critical Location  :   0.000 m from Start.
   Mz*  =     0.0000E+00 KN
   φMsvz=     3.6625E+00 KN                  [NZS3404 12.10.3.1]
   Shear-Z + Bend-Y Interaction
   Critical Load Case :     1*
   Critical Ratio     :   0.856
   Critical Location  :   0.000 m from Start.
   My*  =    10.0000E+00 KN
   φMsvy=    11.6789E+00 KN                  [NZS3404 12.10.3.1]

******************************************************************************
**
```

Related Links

- *D11.A.5.1 Analytical Member Design* (on page 1769)

V. Russia

V.SNiP SP16 2011 - I Section with Axial Load

Verify the design capaciy of an I section used in a column per the SP16.13330.2011 code.

Details

The column is 7.5 m tall with a pinned base and a roller (free to move vertically and rotate) at the top. The column has a 3,500 kN axial load. The section used is an HD320X127. Steel grade is S235.

$E = 205{,}000\ MPa$
$R_y = 235\ MPa$ (yield strength)
$\gamma_{c1} = 1, \gamma_{c2} = 1$

Validation

Check for Axial Force

$$\frac{N}{A_n \times R_y \times \gamma_c} \leq 1 \qquad \text{[Cl. 7.1.1, Eqn. 5]}$$

$$\frac{3{,}500}{161\left(10^{-4}\right) \times 235\left(10^{3}\right) \times 1.0} = 0.925 \leq 1 \text{ (OK)}$$

$$\frac{N}{\phi \times A \times R_y \times \gamma_c} \leq 1 \qquad \text{[Cl. 7.1.3, Eqn. 7]}$$

where

A = cross sectional area = $161\ cm^2$

Slenderness ratios:

$$K_y = K_x = 0.75$$

$$\lambda_x = \frac{K_x L}{r_x} = \frac{0.75 \times 7.5}{0.138} = 40.76$$

$$\lambda_y = \frac{K_y L}{r_y} = \frac{0.75 \times 7.5}{0.0757} = 74.3$$

$$\bar{\lambda}_x = \lambda_x \sqrt{\frac{R_y}{E}} = 40.76\sqrt{\frac{235}{205,\ 000}} = 1.38$$

$$\bar{\lambda}_y = \lambda_y \sqrt{\frac{R_y}{E}} = 74.3\sqrt{\frac{235}{205,\ 000}} = 2.516$$

$$\phi = 0.5\frac{\partial - \sqrt{\partial^2 + 39.48 \times \bar{\lambda}^2}}{\bar{\lambda}^2} \qquad \text{[Eqn. 8]}$$

$$\partial = 9.87 \times \left(1 - \alpha + \beta \times \bar{\lambda}\right) + \bar{\lambda}^2 \qquad \text{[Eqn. 9]}$$

where

α = 0.04 (From Table 7)

β = 0.09 (From Table 7)

$\bar{\lambda}$ = 2.516 (maximum slenderness ratio)

$$\partial = 9.87 \times \left[1 - 0.04 + 0.09 \times (2.516)\right] + (2.516)^2 = 18.04$$

$$\phi = 0.5 \frac{18.04 - \sqrt{(18.04)^2 + 39.48 \times (2.516)^2}}{(2.516)^2} = 0.7385 < \frac{7.6}{\lambda^2} = 1.2006 \text{ (OK)}$$

$$\frac{N}{\phi \times A \times R_y \times \gamma_c} = \frac{3,\ 500}{0.7385 \times 0.0161 \times 235\left(10^3\right) \times 1.0} = 1.25 > 1$$

Check Web Slenderness

$$h_{ef} = D - 4T_f = 320 - 4 \times 20.5 = 238 \text{ mm}$$

$$\bar{\lambda}_w = \frac{h_{ef}}{T_w}\sqrt{\frac{R_y}{E}} = \frac{238}{11.5}\sqrt{\frac{235}{205,\ 000}} = 0.700$$

$$\bar{\lambda}_{uw} = 1.2 + 0.35\bar{\lambda} = 1.2 + 0.35(2.516) = 2.08 \qquad \text{[Table 9; Eqn. 24]}$$

$$\bar{\lambda}_w < \bar{\lambda}_{uw} \text{ (OK)}$$

Check Flange Slenderness

$$b_{ef} = \frac{1}{2}\left(B - 2T_w\right) = \frac{1}{2}\left(300 - 4 \times 11.5\right) = 127 \text{ mm}$$

$$\bar{\lambda}_f = \frac{b_{ef}}{T_f}\sqrt{\frac{R_y}{E}} = \frac{127}{20.5}\sqrt{\frac{235}{205,\ 000}} = 0.210$$

$$\bar{\lambda}_{uf} = 0.36 + 0.1\bar{\lambda} = 0.36 + 0.1(2.516) = 0.612 \qquad \text{[Table 10; Eqn. 37]}$$

$$\bar{\lambda}_f < \bar{\lambda}_{uf} \text{ (OK)}$$

Results

Table 700: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Cl. 7.1.1	0.925	0.92	negligible	
Cl. 7.1.3	1.25	1.25	none	
$\bar{\lambda}_w$	0.700	0.699	negligible	
$\bar{\lambda}_{uw}$	2.08	2.08	none	
$\bar{\lambda}_f$	0.210	0.209	negligible	
$\bar{\lambda}_{uf}$	0.612	0.611	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2011 - I Section with Axial Load.std` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 7.5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -3500
PERFORM ANALYSIS
PARAMETER 1
CODE RUSSIAN 2011
KY 0.75 ALL
KZ 0.75 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2011)   V2.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
*     1  I      HD320X127     FAIL      SP cl.7.1.3      1.25         1
                           3.500E+03 C    0.000E+00    0.000E+00   0.000E+00
```

```
      1  I      HD320X127     PASS      SP cl.7.1.1      0.92         1
                          3.500E+03 C    0.000E+00    0.000E+00   0.000E+00
   MATERIAL DATA
      Steel                           = S235        EN10025-2
      Modulus of elasticity           = 206.E+06 kPa
      Design Strength (Ry)            = 235.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                   = 7.50E+00
      Gross Area                      = 1.61E-02
      Net Area                        = 1.61E-02
                                           x-axis      y-axis
      Moment of inertia (I)           :   308.E-06    924.E-07
      Section modulus (W)             :   193.E-05    616.E-06
      First moment of area (S)        :   107.E-05    470.E-06
      Radius of gyration (i)          :   138.E-03    757.E-04
      Effective Length                :   5.63E+00    5.63E+00
      Slenderness                     :   407.E-01    743.E-01
   DESIGN DATA (units -kN,m) SP16.13330.2011
      Axial force                     :   350.0E+01
                                           x-axis      y-axis
      Moments                         :   0.000E+00    0.000E+00
      Shear force                     :   0.000E+00    0.000E+00
      Bi-moment                       :   0.000E+00 Value of Bi-moment not
being entered!!!
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(7)  N/(FI*A*Ry*GammaC)= 350.0E+01/( 7.40E-01* 1.613E-02* 235.0E+03*
1.00E+00)= 1.25E+00>1
      7.3 LAMBDA_w&amp;lt;=LAMBDA_uw= 6.99E-01&amp;lt;= 2.08E+00 OK
          LAMBDA_f&amp;lt;=LAMBDA_uf= 2.09E-01&amp;lt;= 6.11E-01 OK
      F.(5)  N/(An*Ry*GammaC)= 350.0E+01/( 1.61E-02* 235.0E+03* 1.00E+00)=
9.23E-01=&lt;1
```

Related Links

- *D13.C.2 Member Capacities* (on page 1906)

V.SNiP SP16 2011 - I Section with UDL

Verify the design capacity per SP16.13330.2011 of an I section subject to a uniformly distributed load.

Details

Uniformly distributed load of 100 kN/m along the entire span. Member is simply supported span 5m in length. Section used is an HD320x127.

Grade of Steel: S235 - $E = 206{,}000\ MPa, f_y = 235\ MPa, \gamma_{c1} = 1.1, \gamma_{c2} = 1.1$

Validation

Check for Flexure

$$\frac{M}{W_{n,min} R_y \gamma_c} \le 1$$ [Eqn. 41 of SNiP SP16.13330.2011]

where

M	=	312.5
$W_{n,min}$	=	$W_x = 1{,}926.5\ cm^3$
R_y	=	235 MPa

$$\frac{M}{W_{n,min}R_y\gamma_c} = 0.628 \le 1 \text{ (OK)}$$

Check for Shear

$$\frac{QS}{It_w R_s\gamma_c} \le 1 \qquad \text{[Eqn. 42 of SNiP SP16.13330.2011]}$$

where

Q	=	$q.l/2 = 250\ kN$
S	=	1,070 cm^3
I	=	30,820 cm^4
R_s	=	0.58×235 MPa
t_w	=	11.5 mm

$$\frac{QS}{It_w R_s\gamma_c} = 0.5034 \le 1 \text{ (OK)}$$

Check for Lateral Torsional Buckling

$$\frac{M_x}{\phi_b W_{cx} R_y\gamma_c} \le 1 \qquad \text{[Eqn. 43 of SNiP SP16.13330.2011]}$$

where

α = $1.54\frac{I_t}{I_y}\left(\frac{l_{ef}}{h}\right)^2 = 9.1603$ [From Eqn. G.4]

$0.1 \le \alpha \le 40$ so from Table G.1:

$\psi = 1.6 + 0.08\alpha = 2.333$

ϕ_1 = $\psi\frac{I_y}{I_x}\left(\frac{h}{l_{ef}}\right)^2\frac{E}{R_y} = 0.00078 < 0.85$

from Table G.3: $\phi_b = 0.68 + 0.21\phi_1 = 1.2073 \ge 1.0$ so use $\phi_b = 1.0$

$$\frac{M_x}{\phi_b W_{cx} R_y\gamma_c} = \frac{312.5}{1.0 \times 1{,}926.5\left(10^{-6}\right) \times 235\left(10^{3}\right) \times 1.1} = 0.628 \le 1 \text{ (OK)}$$

Check for Combined Shear & Flexure

$$\frac{0.87}{R_y\gamma_c}\sqrt{\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2} \le 1 \qquad \text{[Eqn. 44 of SNiP SP16.13330.2011]}$$

where

σ_x	=	$M / W_x = 312.5\times10^3 / 1{,}926.5 = 162.2\ MPa$
σ_y	=	0 (No weak axis bending)
τ_{xy}	=	$QS / (I \times t_w) = 0\ MPa$

Note: Moment is maximum at mid-span, where shear is zero.

$$\frac{0.87}{235 \times 1.1}\sqrt{162.2^2} = 0.55 \le 1 \text{ (OK)}$$

From Table 11 of SP16 code:

$\lambda_{ub} = 0.35 + 0.0032\times b/t + (0.76\text{-}0.02\times b/t)\times b/h = 0.35 + 0.0032\times300/20.5 + (0.76\text{-}0.02\times300/20.5)\times1.0016 = 0.865$ [Eqn. 73]

$$\lambda_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}} = 0.5629$$

$\lambda_b \leq \lambda_{ub}$, OK

Check for Deflection

$$f_{\max} = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{100 \times 5^4}{206{,}000 \times 10^3 \times 30{,}820 \times 10^{-8}} = 0.0128 \text{ m}$$

Maximum member deflect = $L/200 = 0.025\ m$

Ratio = $0.0128\ /\ 0.025 = 0.512$

Results

Table 701: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Cl. 8.2.1 (41)	0.628	0.63	Negligible	
Cl. 8.2.1 (44)	0.55	0.55	None	
F.69	0.628	0.628	None	
Max deflection	0.025	0.025	None	
Deflection	0.0128	0.0128	None	
Deflection ratio	0.512	0.512	None	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2011 - I Section with UDL.std` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
```

```
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -100
PERFORM ANALYSIS
PRINT FORCE ENVELOPE NSECTION 2 ALL
PARAMETER 1
CODE RUSSIAN 2011
BEAM 1 ALL
ENSGR 1 ALL
ENMAIN 1 ALL
GAMC2 1.1 ALL
GAMC1 1.1 ALL
TB 1 ALL
DFF 200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2011)   V2.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
      1  I      HD320X127     PASS     SP cl.8.2.1(41)    0.63         1
                            0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127     PASS     SP cl.8.2.1(42)    0.00         1
                            0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127     PASS     SP cl.8.2.1(44)    0.55         1
                            0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127      PASS      SP cl.8.4.1      0.63         1
                            0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127      PASS      SP cl.DISPL      0.51         1
                            0.000E+00      3.125E+02    0.000E+00   2.500E+00
   MATERIAL DATA
      Steel                         = S235        EN10025-2
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 235.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 5.00E+00
      Gross Area                    = 1.61E-02
      Net Area                      = 1.61E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   308.E-06    924.E-07
      Section modulus (W)           :   193.E-05    616.E-06
      First moment of area (S)      :   107.E-05    470.E-06
      Radius of gyration (i)        :   138.E-03    757.E-04
      Effective Length              :   5.00E+00    5.00E+00
```

```
      Slenderness                     :   0.00E+00    0.00E+00
   DESIGN DATA (units -kN,m) SP16.13330.2011
      Axial force                     :   0.000E+00
                                           x-axis      y-axis
      Moments                         :   312.5E+00    0.000E+00
      Shear force                     :   0.000E+00    0.000E+00
      Bi-moment                       :   0.000E+00 Value of Bi-moment not 
being entered!!!
      Class of beam cross-section     :   1 class
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(41)  M/(Wn,min*Ry*GammaC)= 312.5E+00/( 1.93E-03* 235.0E+03* 1.10E
+00= 6.28E-01=&lt;1
      F.(44)  0.87/(Ry*GammaC)*SQRT(SIGMx^2+3*TAUxy^2)=
              0.87/( 235.0E+03* 1.10E+00)*SQRT(-162.2E+03^2+3* 0.000E+00)=
              5.46E-01=&lt;1
              TAUxy/(Rs*GammaC)= 0.000E+00/( 136.3E+03* 1.10E+00)= 0.00E
+00=&lt;1
      F.(69)  M/(FIb*Wcx*Ry*GammaC)=
              312.5E+00/( 1.00E+00* 1.93E-03* 235.0E+03* 1.10E+00)=
6.28E-01=&lt;1
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.025 M)
          SPAN/DEFLECTION = 390.4E+00 (DEFLECTION=  1.281E-02M)
          LOAD=    1     RATIO=    0.512
```

Related Links

- *D13.C.2 Member Capacities* (on page 1906)

V. SNiP SP16 2017 - Channel section with UDL

Design an channel section subjected to a uniform distributed load per the SP 16.13330.2017 code.

Details

A 5m long, simply supported beam has an American C15X50 section. The beam is subjected to a uniform distributed load of 100 kN/m in the Y direction. The steel used has a modulus of elasticity of 206,000 MPa and a $R_{yn} = 235$ MPa. $\gamma_{c1} = 1.1$, $\gamma_{c2} = 1.1$

Validation

$$R_y = R_{yn} / \gamma_m = 223.8 \text{ MPa}$$

$$R_s = 0.58 \times R_y / \gamma_m = 129.8 \text{ MPa}$$

Check for Flexure

Need to satisfy the following equation:

$$\frac{M}{W_{n,\min} R_y \gamma_c} \leq 1 \quad \text{(Eq. 41)}$$

where

M = $100 (5)^2 / 8 = 312.5$ kN·m
$W_{n,min}$ = $W_x = 883$ cm^3

Thus, the ratio is $\frac{312.5}{883(10)^{-3} \times 235 \times 1.1} = 1.37 > 1$

<u>Check for Shear</u>

Need to satisfy the following equation:

$$\frac{QS}{It_w R_s \gamma_c} \le 1 \qquad \text{(Eq. 42)}$$

where

Q = $0\ kN$

Thus, the ratio is $0.0 < 1$

<u>Check for Combined Flexure & Shear</u>

Need to satisfy the following equation:

$$\frac{0.87}{R_y \gamma_c}\sqrt{{\sigma_x}^2 - \sigma_x \sigma_y + {\sigma_y}^2 + 3{\tau_{xy}}^2} \le 1 \qquad \text{(Eq. 44)}$$

where

σ_x = $M / W_x = 312.5 (10)^3 / 883 = 353.9\ MPa$

σ_y = $0\ MPa$

τ_{xy} = $0\ MPa$

Thus, the ratio is $\frac{0.87}{235 \times 1.1}\sqrt{(353.9)^2} = 1.19 > 1$

<u>Check for Stability</u>

Check per Cl. 8.4.4. From Table 11 of SP 16.13330-2017:

$$\bar{\lambda}_{ub} = 0.35 + 0.0032\frac{b}{t} + \left(0.76 - 0.02\frac{b}{t}\right)\frac{b}{h} \qquad \text{(Eq. 73)}$$

$$\bar{\lambda}_{ub} = 0.35 + 0.0032\frac{94.49}{16.51} + \left(0.76 - 0.02\frac{94.49}{16.51}\right)0.2593 = 0.536$$

$$\bar{\lambda}_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}} = 1.787 > \bar{\lambda}_{ub}$$

So, the stability of the beam is *not* ensured per Cl. 8.4.4.b. Therefore a check per Cl 8.4.1 needs to be made.

<u>Check for Lateral-Torsional Buckling</u>

Need to satisfy the following equation:

$$\frac{M_x}{\phi_b W_{cx} R_y \gamma_c} \le 1 \qquad \text{(Eq. 69)}$$

$$\alpha = 1.54\frac{I_t}{I_y}\left(\frac{l_{ef}}{h}\right)^2 = 63.80 \qquad \text{(Eq. G.4)}$$

Since $\alpha > 40$, from Table G.1:

$$\psi = 3.15 + 0.04\alpha - 2.7(10)^{-5}\alpha^2 = 5.592$$

$$\phi_1 = \psi\frac{I_y}{I_x}\left(\frac{h}{l_{ef}}\right)^2\frac{E}{R_y} = 0.775$$

$$\phi_b = 0.7\phi_1 = 0.7(0.775) = 0.543$$

Thus, the ratio is $\frac{312.5}{0.543 \times 883(10)^{-3} \times 235 \times 1.1} = 2.52 > 1$

<u>Check for Deflection</u>

$$f_{\max} = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{100 \times 5^4}{206(10)^6 16,\ 816(10)^{-8}} = 0.0235 \text{ m}$$

The maximum member deflection is limited to $l\ /\ 200 = 0.025\ m$

Thus, the ratio is $0.0235\ /\ 0.025 = 0.94$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio of Flexure (Eq. 41)	1.37	1.37	none	
Ratio of Shear (Eq. 42)	0	0	none	
Ratio of LTB (Eq. 69)	2.52	2.52	none	
Ratio of Combined Shear & Flexure (Eq. 44)	1.19	1.19	none	
Deflection (m)	0.0235	0.02348	negligible	
Deflection Ratio	0.94	0.94	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - Channel section with UDL.std` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 01-Sep-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -100
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE RUSSIAN
CB 1 ALL
PY 235000 ALL
GAMC2 1.1 ALL
GAMC1 1.1 ALL
TB 1 ALL
DFF 200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
*     1  [      C15X50        FAIL     SP cl.8.2.1(41)    1.37         1
                           0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  [      C15X50        PASS     SP cl.8.2.1(42)    0.00         1
                           0.000E+00      3.125E+02    0.000E+00   2.500E+00
*     1  [      C15X50        FAIL     SP cl.8.2.1(44)    1.19         1
                           0.000E+00      3.125E+02    0.000E+00   2.500E+00
*     1  [      C15X50         FAIL      SP cl.8.4.1      2.52         1
                           0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  [      C15X50         PASS         DISPL         0.94         1
                           0.000E+00      3.125E+02    0.000E+00   2.500E+00
   MATERIAL DATA
      Steel                            = User
      Modulus of elasticity            = 206.E+06 kPa
      Design Strength (Ry)             = 235.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                    = 5.00E+00
      Gross Area                       = 9.48E-03
      Net Area                         = 9.48E-03
                                            x-axis      y-axis
      Moment of inertia (I)            :   168.E-06    458.E-08
      Section modulus (W)              :   883.E-06    617.E-07
      First moment of area (S)         :   561.E-06    667.E-07
      Radius of gyration (i)           :   133.E-03    220.E-04
      Effective Length                 :   5.00E+00    5.00E+00
      Slenderness                      :   0.00E+00    0.00E+00
   DESIGN DATA (units -kN,m) SP16.13330.2017
```

```
      Axial force                     :   0.000E+00
                                           x-axis      y-axis
      Moments                         :   312.5E+00    0.000E+00
      Shear force                     :   0.000E+00    0.000E+00
      Bi-moment                       :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    1
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(41)  M/(Wn,min*Ry*GammaC)= 312.5E+00/( 8.83E-04* 235.0E+03* 1.10E
+00= 1.37E+00>1
      F.(44)  0.87/(Ry*GammaC)*SQRT(SIGMx^2+3*TAUxy^2)=
              0.87/( 235.0E+03* 1.10E+00)*SQRT(-354.0E+03^2+3* 0.000E+00^2)=
              1.19E+00>1
              TAUxy/(Rs*GammaC)= 0.000E+00/( 136.3E+03* 1.10E+00)= 0.00E
+00=&lt;1
      F.(69)  Mx/(FIb*Wcx*Ry*GammaC)=
              312.5E+00/( 5.43E-01* 8.83E-04* 235.0E+03* 1.10E+00)= 2.52E
+00>1
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.025 M)
          SPAN/DEFLECTION = 213.0E+00 (DEFLECTION=  2.348E-02M)
          LOAD=    1     RATIO=    0.939     LOCATION=    2.500
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - CLASS 2 Rolled I Section with Bi-Moment

Verify a rolled I section subjected to uniform distributed loads per the SP 16.13330.2017 code.

Details

A 6m long, simply supported beam has a UPT HE500A section. The beam is subjected to a uniform distributed load of 132.2 kN/m in the Y direction and a bi-moment of 10 kN·. The steel used has a modulus of elasticity of 206,000 MPa and a $R_{yn} = 235\ MPa$. $\gamma_m = 1.025$, $\gamma_c = 1$

Section Properties

$D = 490$ mm

$B = 300$ mm

$t_f = 23$ mm

$t_w = 12$ mm

$I_x = 84{,}054\ cm^4$

$I_y = 10{,}356\ cm^4$

$W_x = 3{,}431\ cm^3$

$W_y = 690.4\ cm^3$

Validation

$R_y = R_{yn}/\ \gamma_m = 229.3$ MPa

$R_s = 0.58 \times R_y/\ \gamma_m = 133.0$ MPa

Check for Flexure

As this is a Class 2 section (elasto-plastic), it must satisfy equation 53 from Cl. 8.2.3:

$$\frac{M_x}{c_x W_{x,\min} R_y \gamma_c} + \frac{B}{c_w W_{y,\min} R_y \gamma_c} \le 1 \qquad \text{(Eq. 53)}$$

where

M_x	=	$132.2\ (6)^2 / 8 = 594.9\ kN{\cdot}m$
β	=	1 per Eq. 52 (Cl. 8.2.3), as $Q_x = Q_y = 0$, and therefore $\tau_x = \tau_y = 0$
c_x	=	1.063 (from Table E.1); for $A_f / A_w = 69 / 53.28 = 1.295$
c_w	=	2.915 (from Table 10a)

Thus, the ratio is $\frac{594.9(10)^3}{1.063(3{,}431)(229.3)(1.0)} + \frac{10(10)^3}{2.915(690.4)(229.3)(1.0)} = 0.711 + 0.022 = 0.733$

Check for Stability

Check per Cl. 8.4.4. From Table 11 of SP 16.13330-2017:

$$\sigma_x = \frac{M_x}{W_c \times \gamma_c} = \frac{594.9}{3{,}431 \times (10)^{-3} \times 1} = 173.4 \text{ MPa}$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{b}{t} + \left(0.76 - 0.02\frac{b}{t}\right)\frac{b}{h}\right]\delta\sqrt{\frac{R_y}{\phi_x}}$$

Per Cl. 8.4.6, this value is multiplied by δ as indicated above, where:

$$\delta = 1 - 0.6\frac{(c_{1x} - 1)}{(c_x - 1)} \qquad \text{(Eq. 76)}$$

where

$$c_{1x} = \max \left| \begin{array}{l} M_x / \left(W_{xn} \times R_y \times \gamma_c\right) = \frac{594.9(10)^3}{3{,}431(229.3)(1.0)} = 0.756 \\ \beta c_x = 1.0 \times 1.063 = 1.063 \end{array} \right.$$

Therefore, since $c_{1x} = c_x$, $\delta = 1 - 0.6 = 0.4$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{300}{23} + \left(0.76 - 0.02\frac{300}{23}\right)\frac{300}{490}\right]0.4\sqrt{\frac{229.3}{173.4}} = 0.328$$

$$\bar{\lambda}_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}}\frac{6{,}000}{300}\sqrt{\frac{229.3}{206{,}000}} = 0.667 > \bar{\lambda}_{ub}$$

So, the stability of the beam is *not* ensured per Cl. 8.4.4.b.

Check for Deflection

$$f_y = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{132 \times 6^4}{206(10)^6 84{,}054(10)^{-8}} = 0.0129 \text{ m}$$

The maximum member deflection is limited to $l / 200 = 0.03\ m$

Thus, the ratio is $0.0129 / 0.03 = 0.429$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio of Flexure (Eq. 53)	0.733	0.73	negligible	
C_x	1.063	1.06	negligible	
C_w	2.915	2.92	negligible	
σ_x (MPa)	173.4	173.4	none	
λ_b	0.667	0.6672	negligible	
λ_{ub}	0.328	0.3277	negligible	
Deflection (m)	0.0129	0.01287	negligible	
Deflection Ratio	0.429	0.429	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - CLASS 2 Rolled I Section with Bi-Moment.std` is typically installed with the program.

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER KN
WIDE FLANGE
UTP-HE-500-A
0.019128 0.49 0.012 0.3 0.023 0.000840544 0.000103564 2.68914e-06 -
0.00588 0.0092 0.3 0.023
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY RUSSIAN
1 UPTABLE 1 UTP-HE-500-A
CONSTANTS
```

```
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -132.2
PERFORM ANALYSIS
PARAMETER 1
CODE RUSSIAN
BMT 10 ALL
GAMM 1 ALL
SGR 9 ALL
GAMC2 1 ALL
GAMC1 1 ALL
GAMM 1 ALL
STP 2 ALL
TB 2 ALL
DFF 200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
      1  B I    UTP-HE-500-A  PASS      SP cl.8.2.3      0.73         1
                           0.000E+00      5.949E+02    0.000E+00   3.000E+00
      1  B I    UTP-HE-500-A  PASS         DISPL         0.43         1
                           0.000E+00      5.949E+02    0.000E+00   3.000E+00
   MATERIAL DATA
      Steel                         = C255       SP16.13330
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 229.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 6.00E+00
      Gross Area                    = 1.91E-02
      Net Area                      = 1.91E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   841.E-06    104.E-06
      Section modulus (W)           :   343.E-05    690.E-06
      First moment of area (S)      :   191.E-05    525.E-06
      Radius of gyration (i)        :   210.E-03    736.E-04
      Effective Length              :   6.00E+00    6.00E+00
      Slenderness                   :   0.00E+00    0.00E+00
      Sectorial moment of inertia (Iw)   :    564.E-08 [m^6]
      Sectorial section modulus (Ww)     :    226.E-07 [m^4]
      Sectorial area (coordinate) (w)    :    250.E-03 [m^2]
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   0.000E+00
                                         x-axis      y-axis
```

```
      Moments                       :   594.9E+00    0.000E+00
      Shear force                   :   0.000E+00    0.000E+00
      Bi-moment                     :   100.0E-01 [kNm^2]
      Stress-strain state checked as: Class    2
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(53)  Mx/(Cx*beta*Wxn,min*Ry*GammaC)+B/(Cw*Wyn,min*Ry*GammaC)=
               594.9E+00/( 1.06E+00* 1.00E+00* 3.43E-03* 229.3E+03* 1.00E+00)+
               100.0E-01/( 2.92E+00* 6.90E-04* 229.3E+03* 1.00E+00)=
               7.33E-01=&lt;1
              TAUx=Qy/Aw= 0.000E+00/ 532.8E-05= 0.00E+00 =&lt; 0,5*RS= 664.9E
+02
      LAMBDA_b=(Lef/b)*SQRT(Ry/E)=
              ( 600.0E-02/( 3.000E-01))*SQRT( 229.3E+03/ 206.0E+06)= 6.672E-01
      SIGMA_x=Mx/(Wc*GammaC)= 594.9E+00/( 343.1E-05* 100.0E-02)= 1.734E+05 kPa
      LAMBDA_ub=(0.35+0.0032*b/t+(0.76-0.02*b/t)*b/h)*delta*SQRT(Ry/SIGMA_x)=
              =(0.35+0.0032* 1.304E+01+(0.76-0.02* 1.304E+01)* 6.424E-01)*
4.000E-01* 1.150E+00
              = 3.277E-01&amp;lt; LAMBDA_b= 6.672E-01
      **Warning- Stability of the beam is not ensured according to cl. 8.4.4
b)
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.030 M)
          SPAN/DEFLECTION = 466.0E+00 (DEFLECTION=  1.287E-02M)
          LOAD=    1     RATIO=    0.429     LOCATION=    3.000
```

V. SNiP SP16 2017 - CLASS 2 UPT I section

Check the capacity of a user-provided table I section subject to biaxial moment per the SP 16.13330.2017 code.

Details

A 6m, simply supported beam uses a welded, built-up I section. The section is defined as Class 2 (elasto-plastic). The flanges are 300 mm × 23 mm; the total section depth is 490 mm; the web thickness is 12 mm. The steel used has a modulus of elasticity of 206,000 MPa and a $R_{yn} = 235\ MPa$. $\gamma_m = 1.025$. $\gamma_c = 1.0$

The member is subjected to uniformly distributed loads of 150 kN/m in the Y direction and 30 kN/m in the Z direction.

Validation

$$R_y = R_{yn}/\ \gamma_m = 229.3 \text{ MPa}$$

$$R_s = 0.58 \times R_y/\ \gamma_m = 133.0 \text{ MPa}$$

<u>Section Properties</u>

$$I_x = \frac{12(490 - 2 \times 23)^3}{12} + 2\left[\frac{300(23)^3}{12} + 300(23)\left(\frac{490}{2} - \frac{23}{2}\right)^2\right] = 84,\ 054 \text{ cm}^4$$

$$I_y = \frac{(490 - 2 \times 23)12^3}{12} + 2\left[\frac{23(300)^3}{12}\right] = 10,\ 356 \text{ cm}^4$$

$$W_x = \frac{84,\ 054(2)}{49} = 3,\ 431 \text{ cm}^3$$

$$W_x = \frac{10,\ 356(2)}{30} = 690.4 \text{ cm}^3$$

Check for Flexure

Ratio of area of the flange to the area of the web:

$$A_f / A_w = \frac{300 \times 23}{(490 - 2 \times 23)12} = 1.295$$

Need to satisfy the following equation from Cl. 8.2.3:

$$\frac{M_x}{c_x \beta W_{xn,\min} R_y \gamma_c} + \frac{M_y}{c_y \beta W_{yn,\min} R_y \gamma_c} \le 1 \qquad \text{(Eq. 51)}$$

where

M_x	=	$150\ (6)^2\ /\ 8 = 675\ kN{\cdot}m$
M_y	=	$30\ (6)^2\ /\ 8 = 135\ kN{\cdot}m$
β	=	1 per Eq. 52, as $Q_x = Q_y = 0$, and therefore $\tau_x = \tau_y = 0$
c_x	=	1.0611 (from Table E.1)
c_y	=	1.47 > 1.15, thus use 1.15

Thus, the ratio is $\frac{675}{1.061 \times 1 \times 3,\ 431 \times (10)^{-3} \times 229.3 \times 1} + \frac{135}{1.15 \times 1 \times 690.4 \times (10)^{-3} \times 229.3 \times 1} = 1.55 > 1$

Check for Stability

$$\phi_x = \frac{M_x}{W_c \times \gamma_c} = \frac{675}{3,\ 431 \times (10)^{-3} \times 1} = 196.7 \text{ MPa}$$

$$\bar{\lambda}_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}} = \frac{6}{0.3}\sqrt{\frac{229.3}{206,\ 000}} = 0.667$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{b}{t} + \left(0.76 - 0.02\frac{b}{t}\right)\frac{b}{h}\right] \times \delta\sqrt{\frac{R_y}{\phi_x}} \qquad \text{(Eq. 76)}$$

Per Cl. 8.4.6, the value $\bar{\lambda}_{ub}$ is multiplied by δ.

where

δ	=	$1 - 0.6(c_{1x} - 1)\ /\ (c_x - 1)$
c_{1x}	=	is the larger of
		$\frac{M_x}{W_{xn} R_y \gamma_c} = \frac{675}{3,\ 431 \times (10)^{-3} 229.3 \times 1} = 0.858$, or $\beta c_x = 1.061$, so =1.061

$$\delta = 1 - 0.6\frac{1}{1} = 0.4$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{300}{23} + \left(0.76 - 0.02\frac{300}{23}\right)\frac{300}{490 - 23}\right] \times 0.4\sqrt{\frac{229.3}{196.7}} = 0.308 < \bar{\lambda}_b$$

So, the stability of the beam is *not* ensured per Cl. 8.4.4.b.

Check for Deflection

$$f_y = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{150 \times 6^4}{206(10)^6 84,\ 054(10)^{-8}} = 0.0237 \text{ m}$$

$$f_z = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{30 \times 6^4}{206(10)^6 10,\ 356(10)^{-8}} = 0.0146 \text{ m}$$

$$f_{\max} = \sqrt{(0.0237)^2 + (0.0146)^2} = 0.0279 \text{ m}$$

The maximum member deflection is limited to $l / 200 = 0.03 m$

Thus, the ratio is $0.0279 / 0.03 = 0.93$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio for flexure (Eq. 51)	1.55	1.55	none	
Deflection (m)	0.0279	0.02785	negligible	
Deflection ratio	0.93	0.93	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - CLASS 2 UPT I section with biaxial moment.std` is typically installed with the program.

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT METER KN
WIDE FLANGE
I_500
0.019128 0.49 0.012 0.3 0.023 0.000840544 0.000103564 2.68914e-06 -
0.00588 0.0092
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
****************************************
MEMBER PROPERTY
1 UPTABLE 1 I_500
****************************************
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
```

```
MEMBER LOAD
1 UNI GY -150
1 UNI GZ 30
PERFORM ANALYSIS
********************************
PARAMETER 1
CODE RUSSIAN
TRACK 2 ALL
DFF 200 ALL
TB 2 ALL
STP 2 ALL
SGR 9 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ============================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ============================================================================
*     1  B I    I_500         FAIL      SP cl.8.2.3      1.55         1
                           0.000E+00      6.750E+02    1.350E+02   3.000E+00
      1  B I    I_500         PASS         DISPL         0.93         1
                           0.000E+00      6.750E+02    1.350E+02   3.000E+00
   MATERIAL DATA
      Steel                         = C255       SP16.13330
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 229.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 6.00E+00
      Gross Area                    = 1.91E-02
      Net Area                      = 1.91E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   841.E-06    104.E-06
      Section modulus (W)           :   343.E-05    690.E-06
      First moment of area (S)      :   191.E-05    525.E-06
      Radius of gyration (i)        :   210.E-03    736.E-04
      Effective Length              :   6.00E+00    6.00E+00
      Slenderness                   :   0.00E+00    0.00E+00
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   0.000E+00
                                         x-axis      y-axis
      Moments                       :   675.0E+00    135.0E+00
      Shear force                   :   0.000E+00    0.000E+00
      Bi-moment                     :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    2
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(51)  Mx/(Cx*beta*Wxn,min*Ry*GammaC)+My/(Cy*Wyn,min*Ry*GammaC)=
               675.0E+00/( 1.06E+00* 1.00E+00* 3.43E-03* 229.3E+03* 1.00E+00)+
               135.0E+00/( 1.15E+00* 6.90E-04* 229.3E+03* 1.00E+00)=
               1.55E+00>1
              TAUx=Qy/Aw= 0.000E+00/ 532.8E-05= 0.00E+00 &lt;= 0,9*RS= 119.7E
```

```
+03
              TAUy=Qx/Af= 0.000E+00/ 138.0E-04= 0.00E+00 &lt;= 0,5*RS= 664.9E
+02
      LAMBDA_b=(Lef/b)*SQRT(Ry/E)=
              ( 600.0E-02/( 3.000E-01))*SQRT( 229.3E+03/ 206.0E+06)= 6.672E-01
      SIGMA_x=Mx/(Wc*GammaC)= 675.0E+00/( 343.1E-05* 100.0E-02)= 1.967E+05 kPa
      LAMBDA_ub=(0.35+0.0032*b/t+(0.76-0.02*b/t)*b/h)*delta*SQRT(Ry/SIGMA_x)=
              =(0.35+0.0032* 1.304E+01+(0.76-0.02* 1.304E+01)* 6.424E-01)*
4.000E-01* 1.079E+00
              = 3.076E-01&lt; LAMBDA_b= 6.672E-01
      **Warning- Stability of the beam is not ensured according to cl. 8.4.4
b)
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.030 M)
          SPAN/DEFLECTION = 215.4E+00 (DEFLECTION=  2.785E-02M)
          LOAD=    1      RATIO=    0.928     LOCATION=    3.000
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - Column in compression

Design a column per the SP 16.13330.2017 code.

Details

A 6.78 m tall, simply supported column has a Russian K2-30 section. The column is subject to a 1,500 kN axial load. The steel used has a modulus of elasticity of 206,000 MPa and a $R_y = 239\ MPa$. $\gamma_{c1} = 1$, $\gamma_{c2} = 1$

Section Properties

D = 300 mm

B = 300 mm

t_f = 15 mm

t_w = 10 mm

A = 119.78 cm^2

I_x = 20,410 cm^4

I_y = 6,755 cm^4

r_x = 13.05 cm

r_y = 7.51 cm

J = 76.5 cm^4

Validation

Check for Axial Force

Need to satisfy the following equation per Cl. 7.1.1:

$$\frac{N}{A_n R_y \gamma_c} \leq 1 \qquad \text{(Eq. 5)}$$

Thus, the ratio is $\frac{1,\ 500}{119.78 \times 10^{-1} \times 239 \times 1} = 0.524 < 1$

Check the slenderness per Cl. 10.4.1. Limit the slenderness to 120.

$$\lambda_x = \frac{k_x l}{r_x} = \frac{1.0 \times 6.78}{0.1305} = 51.94 < 120$$

$$\lambda_y = \frac{k_y l}{r_y} = \frac{1.0 \times 6.78}{0.0751} = 90.28 < 120$$

Thus, the ratio is $90.28\ /\ 120 = 0.75$

Check Cl. 7.1.3

$$\bar{\lambda}_x = \lambda_x \sqrt{\frac{R_y}{E}} = 51.94\sqrt{\frac{239}{206,\ 000}} = 1.769$$

$$\bar{\lambda}_y = \lambda_y \sqrt{\frac{R_y}{E}} = 90.28\sqrt{\frac{239}{206,\ 000}} = 3.075$$

Therefore, $\bar{\lambda} = 3.075$

Need to satisfy the following equation:

$$\frac{N}{\phi A R_y \gamma_c} \le 1 \qquad \text{(Eq. 7)(}$$

where

ϕ = $0.5\frac{\delta - \sqrt{\delta^2 - 39.48 \times \bar{\lambda}^2}}{\bar{\lambda}^2}$, per Eq. 8

δ = $9.87(1 - \alpha + \beta\bar{\lambda}) + \bar{\lambda}^2$, per Eq. 9

From Table 7, note 2, $\alpha = 0.04$ and $\beta = 0.14$.

$$\delta = 9.87[1 - 0.04 + 0.14(3.075)] + (3.075)^2 = 23.18$$

$$\phi = 0.5\frac{23.18 - \sqrt{(23.18)^2 - 39.48 \times (3.075)^2}}{(3.075)^2} = 0.549 < \frac{7.6}{\bar{\lambda}^2} = \frac{7.6}{(3.075)^2} = 0.804$$

Thus, the ratio is $\frac{1,\ 500}{0.549 \times 119.78 \times 10^{-1} \times 239 \times 1} = 0.955 < 1$

Check Cl. 7.3.2

Design height of the cross-section wall, $h_{ef} = 240\ mm$ (Cl. 7.3.1)

$$\bar{\lambda}_w = \frac{h_{ef}}{t_w}\sqrt{\frac{R_y}{E}} = \frac{240}{10}\sqrt{\frac{239}{206,\ 000}} = 0.818$$

Since $\bar{\lambda} = 3.075 > 2, \bar{\lambda}_{uw} = 1.20 + 0.35\bar{\lambda} = 2.276 < 2.3$ (per F.(24), Table 9)

$\bar{\lambda}_w < \bar{\lambda}_{uw}$, OK

Check Cl. 7.3.8

Design width of the cross-section wall, $b_{ef} = b/2 - 2 \times t_w = 130\ mm$

$$\bar{\lambda}_f = \frac{b_{ef}}{t_f}\sqrt{\frac{R_y}{E}} = \frac{130}{15}\sqrt{\frac{239}{206,\,000}} = 0.295$$

Since $\bar{\lambda} = 3.075 > 2, \bar{\lambda}_{uf} = 0.36 + 0.10\bar{\lambda} = 0.668$ (per F.(37), Table 10)

$\bar{\lambda}_f < \bar{\lambda}_{uf}$, OK

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio per Cl. 7.1.3	0.955	0.96	negligible	
Ratio per Cl. 7.1.1	0.524	0.52	negligible	
Ratio per Cl. 10.4.1C	0.75	0.75	none	
$\bar{\lambda}_w$	0.818	0.818	none	
$\bar{\lambda}_{uw}$	2.276	2.28	negligible	
$\bar{\lambda}_f$	0.295	0.295	none	
$\bar{\lambda}_{uf}$	0.668	0.668	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - Column in compression.STD` is typically installed with the program.

The following design parameters are used:

- The slenderness for compression member is limited to 120 by the `CMN 2` parameter.
- The steel grade of S255B-1 (for K type sections) is specified by `SGR 20`.
- The stress strain state is indicated by `TB 1`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08/31/2020
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6.78 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
```

```
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
****************************************
MEMBER PROPERTY RUSSIAN
1 TABLE ST K2-30
****************************************
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
****************************************
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1500
********************************
PERFORM ANALYSIS
PRINT MEMBER PROP ALL
*********************************
PARAMETER 1
CODE RUSSIAN
TRACK 2 ALL
TB 1 ALL
CMN 2 ALL
SGR 20 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx             My      LOCATION
   ========================================================================
      1  I      K2-30          PASS      SP cl.7.1.3      0.96         1
                           1.500E+03 C    0.000E+00    0.000E+00   0.000E+00
      1  I      K2-30          PASS      SP cl.7.1.1      0.52         1
                           1.500E+03 C    0.000E+00    0.000E+00   0.000E+00
      1  I      K2-30          PASS      SP cl.10.4.1C    0.75         1
                           1.500E+03 C    0.000E+00    0.000E+00   0.000E+00
   MATERIAL DATA
      Steel                          = C255B      SP16.13330
      Modulus of elasticity          = 206.E+06 kPa
      Design Strength (Ry)           = 239.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                  = 6.78E+00
      Gross Area                     = 1.20E-02
      Net Area                       = 1.20E-02
                                          x-axis      y-axis
      Moment of inertia (I)          :   204.E-06    675.E-07
      Section modulus (W)            :   136.E-05    450.E-06
      First moment of area (S)       :   751.E-06    342.E-06
```

```
      Radius of gyration (i)        :   131.E-03    751.E-04
      Effective Length              :   6.78E+00    6.78E+00
      Slenderness                   :   519.E-01    903.E-01
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   150.0E+01
                                         x-axis      y-axis
      Moments                       :   0.000E+00    0.000E+00
      Shear force                   :   0.000E+00    0.000E+00
      Bi-moment                     :   0.000E+00 Value of Bi-moment not
being entered!!!
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(7)  N/(FI*A*Ry*GammaC)= 150.0E+01/( 5.48E-01* 1.198E-02* 239.0E+03*
1.00E+00)= 9.55E-01=&lt;1
      7.3 cl. LAMBDA_w&amp;lt;=LAMBDA_uw= 8.18E-01&amp;lt;= 2.28E+00 OK
              LAMBDA_f&amp;lt;=LAMBDA_uf= 2.95E-01&amp;lt;= 6.68E-01 OK
      F.(5)  N/(An*Ry*GammaC)= 150.0E+01/( 1.20E-02* 239.0E+03* 1.00E+00)=
5.24E-01=&lt;1
      ULTIM. SLENDERNESS >= Lambda_y 120.0E+00 >= 902.8E-01
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - Eccentrically Compressed Tube Section

Design a column subject to axial compressive force and biaxial moment per the SP 16.13330.2017 code.

Details

A 5 m tall, simply supported column has a TUB200X160X8 section. The column is subject to a 80 kN axial load along with a uniformly distributed load of 30 kN/m in the local X axis. The steel used has a modulus of elasticity of 206,000 MPa and a $R_y = 562\ MPa$. $\gamma_c = 1, \gamma_m = 1.05$

<u>Section Properties</u>

D = 200 mm

B = 160 mm

t = 8 mm

A = 52.84 cm^2

I_x = 2,975 cm^4

I_y = 2,110 cm^4

I_T = 4,083 cm^4

r_x = 7.50 cm

r_y = 6.32 cm

Validation

$R_y = R_{yn}/\ \gamma_m$ = 561.9 MPa

$R_s = 0.58 \times R_y/\ \gamma_m$ = 325.9 MPa

Bending moment:

$M_x = q_x \times L^2 / 8 = 30\ (5)^2 / 8 = 93.75$ kN·m

Design for Strength (Cl. 9.1.1)

$$\sigma = \frac{N}{A_n} = \frac{80}{52.84(10)^{-1}} = 14.54 < 0.1R_y = 56.19$$

$R_{yn} \le 440$ N/mm^2

$\tau = 0$; i.e., $< 0.5 \times R_s$

So, as per Cl. 9.1.1, F.105 should not be checked. Rather F.106 needs to be checked.

$$\frac{\frac{N}{A_n} \pm \frac{M_x y}{I_{xn}} \pm \frac{M_y x}{I_{yn}} \pm \frac{B_\omega}{I_{\omega n}}}{R_y \gamma_c} \le 1 \qquad \text{(F.(106))}$$

where

y	=	D / 2 = 100 mm
B_ω	=	0

So, the ratio is $\dfrac{14.54 + \dfrac{93.75(100)}{2{,}975 \times 10^{-2}}}{561.9 \times 1} = 0.588 < 1$

Design for Stability (Cl. 9.2.2)

To satisfy F.109, $m_{ef} \le 20$, where:

$$m_{ef} = \eta \times m \qquad \text{(F.110)}$$

where

e	=	$M / N = 93.75 / 80 = 1.172$ m
m	=	$e \times {}^{A}\!/_{W_c} = 1.172 \times 0.00055 / 0.000319 = 20.20$

Thus, $m > 20$, so must review Section 8 for further checks.

Design for Stability for a Box Section (Cl. 9.2.10)

Check the stability of box bars with constant cross-section subject to compression on one or two main planes:

$$\frac{N}{\phi_{ey} \times A \times R_y \times \gamma_c} + \frac{M_x}{c_x \times \delta_x W_{x,\min} \times R_y \times \gamma_c} \qquad \text{(F.120)}$$

As per this clause, for uniaxial bending in the plane of maximum stiffness (i.e., $I_x > I_y$, $M_y = 0$), ϕ_{ey} should be replaced by ϕ_y.

$\lambda_x = K_x \times L / r_x = 1.0\ (500) / 7.50 = 66.34$ (Cl 10.4.1)

$\lambda_y = K_y \times L / r_y = 1.0\ (500) / 6.32 = 79.12$

$$\bar{\lambda}_x = \lambda_x \sqrt{\frac{R_y}{E}} = 66.64\sqrt{\frac{561.9}{206{,}000}} = 3.480$$

$$\bar{\lambda}_y = \lambda_y \sqrt{\frac{R_y}{E}} = 79.12\sqrt{\frac{561.9}{206{,}000}} = 4.132$$

The conditional slenderness, $\bar{\lambda}$, is the larger of $\bar{\lambda}_x$ and $\bar{\lambda}_y$. Thus, $\bar{\lambda} = 4.132$

From Table 7, $a = 0.03$ and $\beta = 0.06$ for a tube cross-section.

$$\delta = 9.87\left(1 - a + \beta \times \bar{\lambda}\right) + \bar{\lambda}^2 = 9.87\left(1 - 0.03 + 0.06 \times 4.132\right) + (4.132)^2 = 29.098 \qquad \text{(F.(9))}$$

$$\phi = 0.5\frac{\delta - \sqrt{\delta^2 - 39.48 \times \bar{\lambda}^2}}{\bar{\lambda}^2} = 0.476$$

Note that per Cl. 7.1.3, for section type *a*, the maximum value of ϕ is given as $7.6/\bar{\lambda}^2 = 7.6/(4.132)^2 = 0.445$ for conventional flexibility greater than 3.8. Thus, take $\phi = 0.445$.

$c_x = 1.14$

$$\delta_x = 1 + \frac{0.1N \times \bar{\lambda}x^2}{A \times R_y} = 1 + \frac{0.1 \times 80 \times (3.480)^2(10)}{52.84 \times 561.9} = 1.033 \qquad \text{(F.122)}$$

So the ratio for stability about x axis is:

$$\frac{80(10)^1}{0.445(52.84)(561.9)(1.0)} + \frac{93.75(10)^3}{1.14(1.03)(297.5)(561.9)(1.0)} = 0.061 + 0.478 = 0.537$$

<u>Check for Flexure</u>

$$\frac{M}{W_{n,min} R_y \gamma_c} \le 1 \qquad \text{[Eqn. 41 of SNiP SP16.13330.2011]}$$

where

$W_{n,min}$ = $W_x = 3{,}191.2\ cm^3 = 319.6(10)^{-6}\ m^3$

$$\frac{93.75}{297.5(10)^{-6} \times 561.9 \times 1.0} = 0.561 \le 1 \text{ (OK)}$$

<u>Check for Shear</u>

By inspection, the shear at the maximum moment location is zero under a uniformly distributed load.

$$\frac{QS}{It_w R_s \gamma_c} \le 1 \qquad \text{[Eqn. 42 of SNiP SP16.13330.2011]}$$

where

Q = $q.l/2 = 0\ kN$

$$\frac{QS}{It_w R_s \gamma_c} = 0 \le 1 \text{ (OK)}$$

<u>Check for Combined Shear & Flexure</u>

$$\frac{0.87}{R_y \gamma_c}\sqrt{\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2} \le 1$$

where

σ_x = $M / W_x = 93.75 \times 10^3 / 297.5 = 315.1\ MPa$

σ_y = 0 (No weak axis bending)

τ_{xy} = $QS / (I \times t_w) = 0\ MPa$

$$\frac{0.87}{561.9 \times 1.0}\sqrt{315.1^2} = 0.488 \le 1 \text{ (OK)}$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio per Cl. 9.1.1	0.588	0.588	none	
Ratio per Cl. 9.2.10	0.537	0.536	negligible	
ϕ_y	0.445	0.445	none	
δ_x	1.03	1.03	none	
Ratio per Cl. 8.2.1 (41)	0.561	0.561	none	
Ratio per Cl. 8.2.1 (42)	0	0	none	
Ratio per Cl. 8.2.1 (44)	0.488	0.488	none	
σ_x (MPa)	315.1	315.1	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - Eccentrically Compressed Tube Section.std` is typically installed with the program.

The following design parameters are used:

- The partial coefficient, $\gamma_m = 1.05$, is specified by `GAMMA 2`.
- The steel grade of S590 (for tube sections) is specified by `SGR 18`.
- The stress strain state is indicated by `TB 1`.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Jan-21
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
****************************************
```

```
MEMBER PROPERTY RUSSIAN
1 TABLE ST TUB200X160X8
****************************************
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
****************************************
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -80
MEMBER LOAD
1 UNI GX 30
*1 UNI GZ 2
********************************
PERFORM ANALYSIS
PRINT MEMBER PROP ALL
***********************
PARAMETER 1
CODE RUSSIAN
TB 1 ALL
GAMM 2 ALL
SGR 18 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
      1  TUB    TUB200X160X8  PASS      SP cl.9.1.1      0.59         1
                          8.000E+01 C    9.375E+01    0.000E+00   2.500E+00
      1  TUB    TUB200X160X8  PASS      SP cl.9.2.10     0.54         1
                          8.000E+01 C    9.375E+01    0.000E+00   2.500E+00
      1  TUB    TUB200X160X8 PASS     SP cl.8.2.1(41)    0.56         1
                          8.000E+01 C    9.375E+01    0.000E+00   2.500E+00
      1  TUB    TUB200X160X8 PASS     SP cl.8.2.1(42)    0.00         1
                          8.000E+01 C    9.375E+01    0.000E+00   2.500E+00
      1  TUB    TUB200X160X8 PASS     SP cl.8.2.1(44)    0.49         1
                          8.000E+01 C    9.375E+01    0.000E+00   2.500E+00
   MATERIAL DATA
      Steel                         = C590       SP16.13330
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 562.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 5.00E+00
      Gross Area                    = 5.28E-03
      Net Area                      = 5.28E-03
                                         x-axis      y-axis
      Moment of inertia (I)         :   298.E-07    211.E-07
```

```
      Section modulus (W)            :   298.E-06    264.E-06
      First moment of area (S)       :   149.E-06    132.E-06
      Radius of gyration (i)         :   750.E-04    632.E-04
      Effective Length               :   5.00E+00    5.00E+00
      Slenderness                    :   666.E-01    791.E-01
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                    :   800.0E-01
                                          x-axis      y-axis
      Moments                        :   937.5E-01    0.000E+00
      Shear force                    :   0.000E+00    0.000E+00
      Bi-moment                      :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    1
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(106) (N/A+Mx*y/Ix+My*x/Iy+B*w/Iw)/(Ry*GammaC)=
              ( 800.0E-01/ 5.3E-03+ 937.5E-01* 1.00E-01/ 2.98E-05+ 0.000E+00*
8.00E-02/
               2.11E-05+ 0.000E+00* 0.00E+00/ 0.00E+00)/( 561.9E+03* 1.00E+00)
               = 5.88E-01=&lt;1
      F.(120)  N/(FIy*A*Ry*GammaC)+Mx/(cx*DELx*Wx,min*Ry*GammaC)=-800.0E-01/
( 4.45E-01* 5.28E-03* 561.9E+03*1.00E+00)+-937.5E-01/( 1.14E+00* 1.03E+00*
297.5E-06* 561.9E+03*1.00E+00)=5.36E-01=&amp;lt;1
      m_x =20.8E+00>20.
      F.(41)  M/(Wn,min*Ry*GammaC)= 937.5E-01/( 2.98E-04* 561.9E+03* 1.00E
+00= 5.61E-01=&lt;1
      F.(44)  0.87/(Ry*GammaC)*SQRT(SIGMx^2+3*TAUxy^2)=
              0.87/( 561.9E+03* 1.00E+00)*SQRT(-315.1E+03^2+3* 0.000E+00^2)=
              4.88E-01=&lt;1
              TAUxy/(Rs*GammaC)= 0.000E+00/( 325.9E+03* 1.00E+00)= 0.00E
+00=&lt;1
```

V. SNiP SP16 2017 - Interaction check of a column

Design a column per the SP 16.13330.2017 code.

Details

A 6.78m tall, simply supported column has a European HE600B section. The column is subject to a 310 kN axial load along with a uniformly distributed load of 50 kN/m in the local Y axis. The steel used has a modulus of elasticity of 206,000 MPa and a $R_y = 239\ MPa$. $\gamma_c = 1$, $\gamma_m = 1.05$

Validation

$$R_y = R_{yn} / \gamma_m = 223.8 \text{ MPa}$$

$$R_s = 0.58 \times R_y / \gamma_m = 129.8 \text{ MPa}$$

Shear force at support:

$$Q = q_x \times L / 2 = 171.8 \text{ kN}$$

$$M_x = q_x \times L^2 / 8 = 295.0 \text{ kN} \cdot \text{m}$$

Bending moment:

$$M_x = q_x \times L^2 / 8 = 171.8\ (6.78)^2 / 8 = 295.0 \text{ kN} \cdot \text{m}$$

<u>Design for Strength (Cl. 9.1.1)</u>

$$\sigma = \frac{N}{A_n} = \frac{310}{270(10)^{-1}} = 11.48 < 0.1R_y = 22.38$$

$R_{yn} \leq 440$ N/mm^2

$\tau = 0$; i.e., $> 0.5 \times R_s$

So, as per Cl. 9.1.1, F.105 should not be checked. Rather F.106 needs to be checked.

$$m_{ef} = \eta \times m$$

where

m = $e \times A\ /\ W_c = 4.51$ ($e = M/N = 0.9516$)

$$\lambda_x = \frac{k_x l}{r_x} = \frac{1.0 \times 6.87}{0.2517} = 27.3$$

$$\lambda_y = \frac{k_y l}{r_y} = \frac{1.0 \times 6.87}{0.0708} = 97.0$$

$$\bar{\lambda}_x = \lambda_x \sqrt{\frac{R_y}{E}} = 27.3\sqrt{\frac{223.8}{206,\ 000}} = 0.900$$

$$\bar{\lambda}_y = \lambda_y \sqrt{\frac{R_y}{E}} = 97.0\sqrt{\frac{223.8}{206,\ 000}} = 3.198$$

Therefore, $\bar{\lambda} = min\ (\bar{\lambda}_x,\ \bar{\lambda}_y) = 0.900$

$$\eta = (1.90 - 0.1m) - 0.02(6 - m)\bar{\lambda} = (1.90 - 0.1 \times 4.51) - 0.02(6 - 4.51)0.900 = 1.422$$

So, $m_{ef} = 1.422 \times 4.51 = 6.41 < 20$ [As per F.(110)]

$$A_f / A_w = 90 / 90 = 1$$

$$\frac{\frac{N}{A_n} \pm \frac{M_x y}{I_{xn}} \pm \frac{M_y x}{I_{yn}} \pm \frac{B_\omega}{I_{\omega n}}}{R_y \gamma_c} \leq 1 \qquad \text{(F.(106))}$$

where

x = 150 mm
y = 300 mm
B_ω = 0

So, the ratio is $\dfrac{11.48 + \dfrac{295.0(300)}{171,\ 000 \times 10^{-2}}}{223.8 \times 1} = 0.28 < 1$

<u>Design for Stability (Cl. 9.2.2)</u>

From Table E.3, depending on conditional slenderness and reduced relative eccentricity:

$\phi_e = 0.2144$

$$\frac{N}{\phi_e \times A \times R_y \times \gamma_c} = \frac{310}{0.2144 \times 270(10)^{-1} \times 223.8 \times 1} = 0.24 < 1 \qquad \text{(F.(109))}$$

<u>Design for Stability (Cl. 9.2.4)</u>

Calculate the stability of eccentrically compressed elements of constant cross-section, out-of-plane bending moment in the plan of maximum stiffness ($I_x > I_y$), coinciding with the plane of symmetry:

$$\frac{N}{c \times \phi_y \times A \times R_y \times \gamma_c} \leq 1 \qquad \text{(F.(111))}$$

where

ϕ_y	=	$\frac{0.5(\delta - \sqrt{\delta^2 - 39.48\bar{\lambda}^2}}{\bar{\lambda}^2}$
$\bar{\lambda}$	=	conditional slenderness $= max(\bar{\lambda_x}, \bar{\lambda_y}) = 3.198$
δ	=	$9.87(1 - \alpha + \beta\bar{\lambda}) + \bar{\lambda}^2$, per Eq. 9

From Table 7, $\alpha = 0.03$ and $\beta = 0.06$.

$$\delta = 9.87[1 - 0.03 + 0.06(3.198)] + (3.198)^2 = 21.70$$

$$\phi_y = 0.5\frac{21.70 - \sqrt{(21.70)^2 - 39.48 \times (3.198)^2}}{(3.198)^2} = 0.660$$

$$c = \frac{\beta}{1 + \alpha \times m_x} \qquad \text{(Cl. 9.2.5)}$$

where

α	=	$0.65 + 0.05 \times m_x = 0.875$
$\bar{\lambda_y}$	=	3.14
δ_c	=	$9.87[1 - 0.03 + 0.06(3.14)] + (3.14)^2 = 21.29$
ϕ_c	=	$0.5\frac{21.29 - \sqrt{(21.29)^2 - 39.48 \times (3.14)^2}}{(3.14)^2} = 0.674$
β	=	$\sqrt{\frac{\phi_c}{\phi_y}} = \sqrt{\frac{0.674}{0.661}} = 1.009$

Therefore, $c = \frac{1.009}{1 + 0.875 \times 4.51} = 0.204 < 1$

So, the ratio is $\frac{310}{0.204 \times 0.66 \times 270(10)^{-1} \times 223.9 \times 1} = 0.38 < 1$

Calculate C_{max} Per Annex E.1

$$c_{max} = \frac{2}{1 + \delta\beta + \sqrt{(1 + \delta\beta)^2 + \frac{16}{\mu}\left(\alpha - \frac{e_x}{h}\right)^2}} \qquad \text{(F.(E.1))}$$

where

δ	=	$4 \times \rho / \mu$
ρ	=	$\frac{I_x + I_y}{Ah^2} + \alpha^2$
μ	=	$8\omega + \frac{0.156 \times I_t \lambda_y^2}{Ah^2}$
ω	=	$\frac{I_\omega}{I_y h_c^2} = \frac{h_c^2 \times b^3 \times t_f / 24}{I_y h_c^2} = \frac{b^3 \times t_f}{24 I_y} = \frac{(30\ cm)^3 \times 3\ cm}{24 \times 13,\ 530\ cm^4} = 0.249$
I_t	=	$\frac{k}{3}\Sigma b_i t_i^3 = \frac{1.29}{3}\left[2(30cm)(3cm)^3 + (60cm - 2 \times 3cm)(1.55cm)^3\right] = 783.1cm^4$ ($k = 1.29$)
e_x	=	$M / N = 295 / 310 = 0.952$
B	=	$1 + 2(\beta / \rho)(e_x / h_c) = 1$

As per Table E.6, $\alpha = 0, \beta = 0$

$$\mu = 8(0.249) + \frac{0.156 \times (783.1)(97.0)^2}{270 \times (60)^2} = 3.17$$

$$\rho = \frac{171,\ 000 + 13,\ 530}{270 \times 60^2} + 0 = 0.190$$

$$\delta = \frac{4 \times 0.19}{3.17} = 0.24$$

$$c_{\text{max}} = \frac{2}{1 + 0.24(0) + \sqrt{(1 + 0.24 \times 0)^2 + \frac{16}{3.17}\left(0 - \frac{0.952}{0.6}\right)^2}} = 0.43$$

Results

Result Type	**Reference**	**STAAD.Pro**	**Difference**	**Comments**
Ratio per Cl. 9.1.1	0.28	0.28	none	
Ratio per Cl. 9.2.2	0.24	0.24	none	
Ratio per Cl. 9.2.4	0.38	0.38	none	
m_{ef}	6.41	6.41		
m_x	4.51	4.51	none	
C	0.204	0.204	none	
C_{max}	0.43	0.40	negligible	
Φ_y	0.66	0.66	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - Interaction check of a column.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Sep-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6.87 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
```

```
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
****************************************
MEMBER PROPERTY EUROPEAN
1 TABLE ST HE600B
****************************************
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
****************************************
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -310
MEMBER LOAD
1 UNI GX 50
********************************
PERFORM ANALYSIS
*********************************
PARAMETER 1
CODE RUSSIAN
TB 1 ALL
ENSGR 1 ALL
GAMM 2 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
      1  I      HE600B        PASS      SP cl.9.1.1      0.28         1
                          3.100E+02 C    2.950E+02    0.000E+00   3.435E+00
      1  I      HE600B        PASS      SP cl.9.2.2      0.24         1
                          3.100E+02 C    2.950E+02    0.000E+00   3.435E+00
      1  I      HE600B        PASS      SP cl.9.2.4      0.38         1
                          3.100E+02 C    2.950E+02    0.000E+00   3.435E+00
   MATERIAL DATA
      Steel                         = S235       EN10025-2
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 224.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 6.87E+00
      Gross Area                    = 2.70E-02
      Net Area                      = 2.70E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   171.E-05    135.E-06
```

```
      Section modulus (W)            :   570.E-05    902.E-06
      First moment of area (S)       :   321.E-05    696.E-06
      Radius of gyration (i)         :   252.E-03    708.E-04
      Effective Length               :   6.87E+00    6.87E+00
      Slenderness                    :   273.E-01    970.E-01
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                    :   310.0E+00
                                          x-axis      y-axis
      Moments                        :   295.0E+00    0.000E+00
      Shear force                    :   0.000E+00    0.000E+00
      Bi-moment                      :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    1
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(106) (N/A+Mx*y/Ix+My*x/Iy+B*w/Iw)/(Ry*GammaC)=
              ( 310.0E+00/ 2.7E-02+ 295.0E+00* 3.00E-01/ 1.71E-03+ 0.000E+00*
1.50E-01/
               1.35E-04+ 0.000E+00* 2.49E-01/ 1.10E-05)/( 223.8E+03* 1.00E+00)
               = 2.83E-01=&lt;1
      cl.9.2.2  m_ef=eta*mx= 1.42E+00* 4.51E+00= 6.41E+00
      F.(109)  N/(FIe*A*Ry*GammaC)= 310.0E+00/( 2.14E-01* 2.70E-02* 223.8E
+03* 1.00E+00)
               = 2.39E-01=&lt;1
      F.(112)  c=beta/(1+alfa*mx)= 1.01E+00/(1+8.75E-01* 4.51E+00)= 2.04E-01
                    c_max= 4.00E-01
      F.(111)  N/(c*FIy*A*Ry*GammaC)
               = 0.31E+03/( 0.20E+00* 0.66E+00* 0.27E-01* 223.8E+03* 1.00E+00)
               = 3.80E-01=&lt;1
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - I section with biaxial moment

Design an I section subjected to uniform distributed loads in both major in minor axes per the SP 16.13330.2017 code.

Details

A 6m long, simply supported beam has a European HE500A section. The beam is subjected to a uniform distributed load of 132.2 kN/m in the Y direction and 30 kN/m in the Z direction. The steel used has a modulus of elasticity of 206,000 MPa and a $R_{yn} = 235\ MPa.\ \gamma_m = 1.05, \gamma_c = 1$

Validation

$$R_y = R_{yn}/\ \gamma_m = 223.8\ \text{MPa}$$

$$R_s = 0.58 \times R_y/\ \gamma_m = 129.8\ \text{MPa}$$

<u>Check for Flexure</u>

Need to satisfy the following equation from Cl. 8.2.1:

$$\frac{M_x}{I_{xn} R_y \gamma_c} y \pm \frac{M_y}{I_{yn} R_y \gamma_c} s \pm \frac{B\omega}{I_{\omega n} R_y \gamma_c} \leq 1 \qquad \text{(Eq. 43)}$$

where

M_x = $132.2\ (6)^2\ /\ 8 = 594.9\ kN{\cdot}m$
M_y = $30\ (6)^2\ /\ 8 = 135\ kN{\cdot}m$
B_x = 0

Thus, the ratio is $\frac{M_x}{I_{xn}R_y\gamma_c}y \pm \frac{M_y}{I_{yn}R_y\gamma_c}s \pm \frac{B\omega}{I_{\omega n}R_y\gamma_c} = 1.62 > 1$

Check for Shear

Need to satisfy the following equation:

$$\frac{QS}{It_w R_s \gamma_c} \le 1 \qquad \text{(Eq. 42)}$$

where

Q = $0\ kN$

Thus, the ratio is $0.0 < 1$

Check for Combined Flexure & Shear

Need to satisfy the following equation:

$$\frac{0.87}{R_y\gamma_c}\sqrt{{\sigma_x}^2 - \sigma_x\sigma_y + {\sigma_y}^2 + 3{\tau_{xy}}^2} \le 1 \qquad \text{(Eq. 44)}$$

where

σ_x = $M_x\ /\ W_x = 594.9\ (10)^3\ /\ 3{,}550 = 167.6\ MPa$
σ_y = $M_y\ /\ W_y = 135\ (10)^3\ /\ 691.1 = 195.3\ MPa$
τ_{xy} = $0\ MPa$

Thus, the ratio is $\frac{0.87}{223.8 \times 1.}\sqrt{(167.6)^2 -\ 167.6\ \times\ 195.3\ + (195.3)^2} = 0.71 < 1$

Check for Stability

Check per Cl. 8.4.4. From Table 11 of SP 16.13330-2017:

$$\phi_x = \frac{M_x}{W_c \times \gamma_c} = \frac{594.9}{3{,}550 \times (10)^{-3} \times 1} = 167.6 \text{ MPa}$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{b}{t} + \left(0.76 - 0.02\frac{b}{t}\right)\frac{b}{h}\right]\sqrt{\frac{R_y}{\phi_x}}$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{300}{23} + \left(0.76 - 0.02\frac{300}{23}\right)\frac{300}{490}\right]\sqrt{\frac{235}{167.6}} = 0.823$$

$$\bar{\lambda}_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}}\frac{6}{0.3}\sqrt{\frac{235}{206{,}000}} = 0.659 < \bar{\lambda}_{ub}$$

So, the stability of the beam is ensured per Cl. 8.4.4.b. Check per Cl 8.4.1 is not required.

Check for Deflection

$$f_y = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{132 \times 6^4}{206(10)^6 86{,}970(10)^{-8}} = 0.0124 \text{ m}$$

$$f_z = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{30 \times 6^4}{206(10)^6 10,\ 356(10)^{-8}} = 0.0237 \text{ m}$$

$$f_{\max} = \sqrt{(0.0124)^2 + (0.0237)^2} = 0.0267 \text{ m}$$

The maximum member deflection is limited to $l / 200 = 0.03\ m$

Thus, the ratio is $0.0267 / 0.03 = 0.89$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio of Flexure (Eq. 43)	1.62	1.62	none	
Ratio of Shear (Eq. 42)	0	0	none	
Ratio of Combined Shear & Flexure (Eq. 44)	0.71	0.71	none	
Deflection (m)	0.0267	0.02675	negligible	
Deflection Ratio	0.89	0.89	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - I section with biaxial moment.std` is typically installed with the program.

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
****************************************
MEMBER PROPERTY EUROPEAN
1 TABLE ST HE500A
****************************************
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
*2 FIXED BUT FX MZ
2 FIXED BUT FX MY MZ
****************************************
```

```
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -132.2
1 UNI GZ 30
********************************
PERFORM ANALYSIS
***********************
PARAMETER 1
CODE RUSSIAN
ENSGR 1 ALL
GAMM 2 ALL
DFF 200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
*     1  I      HE500A       FAIL     SP cl.8.2.1(43)    1.62         1
                           0.000E+00      5.949E+02   1.350E+02   3.000E+00
      1  I      HE500A       PASS     SP cl.8.2.1(42)    0.00         1
                           0.000E+00      5.949E+02   1.350E+02   3.000E+00
      1  I      HE500A       PASS     SP cl.8.2.1(44)    0.71         1
                           0.000E+00      5.949E+02   1.350E+02   3.000E+00
      1  I      HE500A        PASS         DISPL         0.89         1
                           0.000E+00      5.949E+02   1.350E+02   3.000E+00
   MATERIAL DATA
      Steel                         = S235       EN10025-2
      Modulus of elasticity         = 206.E+06 kPa
      Design Strength (Ry)          = 224.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                 = 6.00E+00
      Gross Area                    = 1.98E-02
      Net Area                      = 1.98E-02
                                         x-axis      y-axis
      Moment of inertia (I)         :   870.E-06    104.E-06
      Section modulus (W)           :   355.E-05    691.E-06
      First moment of area (S)      :   197.E-05    530.E-06
      Radius of gyration (i)        :   210.E-03    724.E-04
      Effective Length              :   6.00E+00    6.00E+00
      Slenderness                   :   0.00E+00    0.00E+00
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   0.000E+00
                                         x-axis      y-axis
      Moments                       :   594.9E+00    135.0E+00
      Shear force                   :   0.000E+00    0.000E+00
      Bi-moment                     :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    1
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
```

```
      F.(43)  (Mx*y)/(Ixn*Ry*GammaC)+(My*x)/(Iyn*Ry*GammaC)+
              +(B*w)/(Iwn*Ry*GammaC)=
              ( 594.9E+00* 2.45E-01)/( 8.70E-04* 223.8E+03* 1.00E+00)+
              ( 135.0E+00* 1.50E-01/( 1.04E-04* 223.8E+03* 1.00E+00)+
              ( 0.000E+00* 2.50E-01)/( 5.64E-06* 223.8E+03* 1.00E+00)=
              = 1.62E+00>1
      F.(44)  0.87/(Ry*GammaC)*SQRT(SIGMx^2-SIGMx*SIGMy+SIGMy^2+3*TAUxy^2)=
              0.87/( 223.8E+03* 1.00E+00)*SQRT(-167.6E+03^2--167.6E+03
              *-195.3E+03+-195.3E+03^2+3* 0.000E+00^2)= 7.11E-01=&lt;1
              TAUxy/(Rs*GammaC)= 0.000E+00/( 129.8E+03* 1.00E+00)= 0.00E
+00=&lt;1
      LAMBDA_b=(Lef/b)*SQRT(Ry/E)=
              ( 600.0E-02/( 3.000E-01))*SQRT( 223.8E+03/ 206.0E+06)= 6.592E-01
      SIGMA_x=Mx/(Wc*GammaC)= 594.9E+00/( 355.0E-05* 100.0E-02)= 1.676E+05 kPa
      LAMBDA_ub=(0.35+0.0032*b/t+(0.76-0.02*b/t)*b/h)*delta*SQRT(Ry/SIGMA_x)=
              =(0.35+0.0032* 1.304E+01+(0.76-0.02* 1.304E+01)* 6.424E-01)*
1.000E+00* 1.156E+00
              = 8.232E-01>=LAMBDA_b= 6.592E-01
      Stability of the beam is ensured according to cl. 8.4.4 b)
      Check according to cl. 8.4.1 is not required
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.030 M)
          SPAN/DEFLECTION = 224.3E+00 (DEFLECTION=  2.675E-02M)
          LOAD=    1     RATIO=    0.892     LOCATION=    3.000
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - I section with UDL

Design an I section subjected to a uniform distributed load per the SP 16.13330.2017 code.

Details

A 5m long, simply supported beam has a European HD320X127 section. The beam is subjected to a uniform distributed load of 100 kN/m in the Y direction. The steel used has a modulus of elasticity of 206,000 MPa and a $R_{yn} = 235$ *MPa*. $\gamma_{c1} = 1.1$, $\gamma_{c2} = 1.1$

Validation

$$R_y = R_{yn} / \gamma_m = 223.8 \text{ MPa}$$

$$R_s = 0.58 \times R_y / \gamma_m = 129.8 \text{ MPa}$$

<u>Check for Flexure</u>

Need to satisfy the following equation:

$$\frac{M}{W_{n,\min} R_y \gamma_c} \leq 1 \qquad \text{(Eq. 41)}$$

where

$$M = 100\,(5)^2 \,/\, 8 = 312.5 \text{ kN·m}$$

Thus, the ratio is $\frac{312.5}{1{,}926.5 \times 235(10)^{-3} \times 1.1} = 0.63 < 1$

<u>Check for Shear</u>

Need to satisfy the following equation:

$$\frac{QS}{I t_w R_s \gamma_c} \le 1 \qquad \text{(Eq. 42)}$$

where

Q = $0\ kN$

Thus, the ratio is $0.0 < 1$

Check for Stability

$$\phi_x = \frac{M_x}{W_c \times \gamma_c} = \frac{312.5}{1{,}926.5 \times (10)^{-3} \times 1.1} = 147.5 \text{ MPa}$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{b}{t} + \left(0.76 - 0.02\frac{b}{t}\right)\frac{b}{h}\right]\sqrt{\frac{R_y}{\phi_x}}$$

$$\bar{\lambda}_{ub} = \left[0.35 + 0.0032\frac{300}{20.5} + \left(0.76 - 0.02\frac{300}{20.5}\right)\frac{300}{320}\right]\sqrt{\frac{235}{147.5}} = 1.054$$

$$\bar{\lambda}_b = \frac{l_{ef}}{b}\sqrt{\frac{R_y}{E}}\frac{10}{0.3}\sqrt{\frac{235}{206{,}000}} = 1.126 > \bar{\lambda}_{ub}$$

So, the stability of the beam is *not* ensured per Cl. 8.4.4.b. Check per Cl 8.4.1 needs to be performed.

Check for Lateral-Torsional Buckling

Use an effective length of 10 m.

Need to satisfy the following equation:

$$\frac{M_x}{\phi_b W_{cx} R_y \gamma_c} \le 1 \qquad \text{(Eq. 69)}$$

$$\alpha = 1.54\frac{I_t}{I_y}\left(\frac{l_{ef}}{h}\right)^2 = 1.54\frac{225.1}{9{,}239}\left(\frac{10}{0.32}\right)^2 = 36.64 \qquad \text{(Eq. G.4)}$$

Since $0.1 < \alpha < 40$, from Table G.1:

$$\psi = 1.6 + 0.08\alpha = 4.531$$

$$\phi_1 = \psi\frac{I_y}{I_x}\left(\frac{h}{l_{ef}}\right)^2\frac{E}{R_y} = 1.219$$

$$\phi_b = 0.68 + 0.21\phi_1 = 0.68 + 0.21(1.219) = 0.936$$

Thus, the ratio is $\frac{312.5}{0.936 \times 1{,}926.5(10)^{-3} \times 235 \times 1.1} = 0.67 < 1$

Check for Combined Flexure & Shear

Need to satisfy the following equation:

$$\frac{0.87}{R_y \gamma_c}\sqrt{{\sigma_x}^2 - \sigma_x\sigma_y + {\sigma_y}^2 + 3{\tau_{xy}}^2} \le 1 \qquad \text{(Eq. 44)}$$

where

σ_x = $M_x / W_x = 312.5\ (10)^3 / 1{,}926.5 = 162.2\ MPa$
σ_y = $M_y / W_y = 0\ MPa$
τ_{xy} = $0\ MPa$

Thus, the ratio is $\frac{0.87}{235 \times 1.1}\sqrt{(162.2)^2} = 0.55 < 1$

Check for Deflection

$$f_{max} = \frac{5}{384}\frac{ql^4}{EI_x} = \frac{5}{384}\frac{100 \times 5^4}{206(10)^6 30{,}820(10)^{-8}} = 0.0128 \text{ m}$$

The maximum member deflection is limited to $l\ /\ 200 = 0.025\ m$

Thus, the ratio is $0.0128\ /\ 0.025 = 0.51$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ratio of Flexure (Eq. 41)	0.63	0.63	none	
Ratio of Shear (Eq. 42)	0	0	none	
Ratio of LTB (Eq. 69)	0.67	0.67	none	
Ratio of Combined Shear & Flexure (Eq. 44)	0.55	0.55	none	
Deflection (m)	0.0128	0.01281	negligible	
Deflection Ratio	0.51	0.51	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - I section with UDL.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 1-Sep-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HD320X127
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -100
PERFORM ANALYSIS
PARAMETER 1
CODE RUSSIAN
CMN 8 ALL
CMM 6 ALL
GAMM 2 ALL
LY 10 ALL
LZ 10 ALL
PY 235000 ALL
GAMC2 1.1 ALL
GAMC1 1.1 ALL
TB 1 ALL
DFF 200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ============================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ============================================================================
      1  I      HD320X127    PASS     SP cl.8.2.1(41)    0.63         1
                          0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127    PASS     SP cl.8.2.1(42)    0.00         1
                          0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127    PASS     SP cl.8.2.1(44)    0.55         1
                          0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127     PASS      SP cl.8.4.1      0.67         1
                          0.000E+00      3.125E+02    0.000E+00   2.500E+00
      1  I      HD320X127     PASS        DISPL          0.51         1
                          0.000E+00      3.125E+02    0.000E+00   2.500E+00
   MATERIAL DATA
      Steel                          = User
      Modulus of elasticity          = 206.E+06 kPa
      Design Strength (Ry)           = 235.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                  = 5.00E+00
      Gross Area                     = 1.61E-02
      Net Area                       = 1.61E-02
                                          x-axis      y-axis
```

```
      Moment of inertia (I)         :   308.E-06    924.E-07
      Section modulus (W)           :   193.E-05    616.E-06
      First moment of area (S)      :   107.E-05    470.E-06
      Radius of gyration (i)        :   138.E-03    757.E-04
      Effective Length              :   1.00E+01    1.00E+01
      Slenderness                   :   0.00E+00    0.00E+00
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                   :   0.000E+00
                                         x-axis      y-axis
      Moments                       :   312.5E+00    0.000E+00
      Shear force                   :   0.000E+00    0.000E+00
      Bi-moment                     :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    1
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(41)  M/(Wn,min*Ry*GammaC)= 312.5E+00/( 1.93E-03* 235.0E+03* 1.10E
+00= 6.28E-01=&lt;1
      F.(44)  0.87/(Ry*GammaC)*SQRT(SIGMx^2+3*TAUxy^2)=
              0.87/( 235.0E+03* 1.10E+00)*SQRT(-162.2E+03^2+3* 0.000E+00^2)=
              5.46E-01=&lt;1
              TAUxy/(Rs*GammaC)= 0.000E+00/( 136.3E+03* 1.10E+00)= 0.00E
+00=&lt;1
      LAMBDA_b=(Lef/b)*SQRT(Ry/E)=
              ( 100.0E-01/( 3.000E-01))*SQRT( 235.0E+03/ 206.0E+06)= 1.126E+00
      SIGMA_x=Mx/(Wc*GammaC)= 312.5E+00/( 192.6E-05* 110.0E-02)= 1.475E+05 kPa
      LAMBDA_ub=(0.35+0.0032*b/t+(0.76-0.02*b/t)*b/h)*delta*SQRT(Ry/SIGMA_x)=
              =(0.35+0.0032* 1.463E+01+(0.76-0.02* 1.463E+01)* 1.002E+00)*
1.000E+00* 1.262E+00
              = 1.092E+00&lt; LAMBDA_b= 1.126E+00
      **Warning- Stability of the beam is not ensured according to cl. 8.4.4
b)
      F.(69)  Mx/(FIb*Wcx*Ry*GammaC)=
              312.5E+00/( 9.36E-01* 1.93E-03* 235.0E+03* 1.10E+00)=
6.70E-01=&lt;1
          LIMIT SPAN/DEFLECTION (DFF) =    200.00   (DEFLECTION LIMIT=
0.025 M)
          SPAN/DEFLECTION = 390.4E+00 (DEFLECTION=  1.281E-02M)
          LOAD=    1     RATIO=    0.512     LOCATION=    2.500
```

Related Links

- *D13.E.2 Member Capacities* (on page 1933)

V. SNiP SP16 2017 - I section with axial force and Bi-moment

Design a column subject to axial compressive force and biaxial moment per the SP 16.13330.2017 code.

Details

A 5 m tall, simply supported column has an HE650A section. The column is subject to a 80 kN axial load along with a uniformly distributed load of 30 kN/m in the local X axis and a uniformly distributed load of 2 kN/m in the local Y axis. The steel used has a modulus of elasticity of 206,000 MPa and a $R_y = 235\ MPa$. $\gamma_c = 1$, $\gamma_m = 1.05$

Section Properties

D = 640 mm

B = 300 mm

t_f = 26 mm

t_w = 13.5 mm

A = 241.6 cm^2

I_x = 175,200 cm^4

I_y = 11,720 cm^4

I_T = 458 cm^4

r_x = 26.93 cm

r_y = 6.97 cm

Validation

$R_y = R_{yn}/\ \gamma_m = 223.8$ MPa

$R_s = 0.58 \times R_y/\ \gamma_m = 129.8$ MPa

Shear force at support:

$Q_x = q_x \times L / 2 = 75$ kN

Bending moment:

$M_x = q_x \times L^2 / 8 = 30\ (5)^2 / 8 = 93.75$ kN·m

$M_y = q_y \times L^2 / 8 = 2\ (5)^2 / 8 = 6.25$ kN·m

Design for Strength (Cl. 9.1.1)

$$\sigma = \frac{N}{A_n} = \frac{80}{241.6(10)^{-1}} = 3.311 < 0.1R_y = 22.38$$

$R_{yn} \leq 440$ N/mm^2

$\tau = 0$; i.e., $< 0.5 \times R_s$

So, as per Cl. 9.1.1, F.105 should not be checked. Rather F.106 needs to be checked.

$$m_{ef} = \eta \times m$$

where

$m = e \times A\ /\ W_c = 5.172$ ($e = M/N = 93.75\ /\ 80 = 1.172$)

$$\lambda_x = \frac{k_x l}{r_x} = \frac{1.0 \times 500}{26.93} = 18.57$$

$$\lambda_y = \frac{k_y l}{r_y} = \frac{1.0 \times 500}{6.97} = 71.74$$

$$\bar{\lambda}_x = \lambda_x \sqrt{\frac{R_y}{E}} = 18.58\sqrt{\frac{223.8}{206{,}000}} = 0.612$$

$$\bar{\lambda}_y = \lambda_y \sqrt{\frac{R_y}{E}} = 71.74\sqrt{\frac{223.8}{206{,}000}} = 2.364$$

Therefore, $\bar{\lambda} = min\ (\bar{\lambda}_x,\ \bar{\lambda}_y) = 0.612$

Evaluate η, the coefficient of the shape of cross section vertical element when $5 < m \leq 20, A_f / A_w \geq 1$, and $0 \leq \lambda \leq 5$:

$$\eta = 1.4 - 0.02\bar{\lambda} = (1.4 - 0.02 \times 0.612) = 1.388$$ (Table Д.2, Note 1)

So, $m_{ef} = 1.388 \times 5.172 = 7.18 < 20$ [As per F.(110)]

$$\frac{\frac{N}{A_n} \pm \frac{M_x y}{I_{xn}} \pm \frac{M_y x}{I_{yn}} \pm \frac{B_\omega}{I_{\omega n}}}{R_y \gamma_c} \leq 1$$ (F.(106))

where

x	=	150 mm
y	=	320 mm
B_ω	=	0

So, the ratio is $\frac{3.311 + \frac{93.75(320)}{175{,}200 \times 10^{-2}} + \frac{6.25(150)}{11{,}720 \times 10^{-2}}}{223.8 \times 1} = 0.127 < 1$

Design for Stability (Cl. 9.2.2)

From Table E.3, depending on conditional slenderness and reduced relative eccentricity:

$\phi_e = 0.203$

$$\frac{N}{\phi_e \times A \times R_y \times \gamma_c} = \frac{80}{0.203 \times 241.6(10)^{-1} \times 223.8 \times 1} = 0.073 < 1$$ (F.(109))

Design for Stability (Cl. 9.2.4)

Calculate the stability of eccentrically compressed elements of constant cross-section, out-of-plane bending moment in the plan of maximum stiffness ($I_x > I_y$), coinciding with the plane of symmetry:

$$\frac{N}{c \times \phi_y \times A \times R_y \times \gamma_c} \leq 1$$ (F.111)

where

ϕ_y	=	$\frac{0.5(\delta - \sqrt{\delta^2 - 39.48\bar{\lambda}^2}}{\bar{\lambda}^2}$
$\bar{\lambda}$	=	conditional slenderness $= max\,(\bar{\lambda}_x, \bar{\lambda}_y) = 2.365$
δ	=	$9.87\left(1 - \alpha + \beta\bar{\lambda}\right) + \bar{\lambda}^2$, per Eq. 9 where $\alpha = 0.03$ and $\beta = 0.06$ from Table 7.

$= 9.87[1 - 0.03 + 0.06(2.365)] + (2.365)^2 = 16.57$

$$\phi_y = 0.5\frac{16.57 - \sqrt{(16.57)^2 - 39.48 \times (2.365)^2}}{(2.365)^2} = 0.826$$

$$c_{max} = c_5(2 - 0.2m_x) + c_{10}(0.2m_x - 1)$$ (F.114)

where

α	=	$0.65 + 0.05 \times m_x = 0.909$
β	=	1.0 for $\lambda_y < 3.14$ per Table 21
m_x	=	relative eccentricity = $m = 5.172$
c_5	=	$\frac{\beta}{1 + \alpha \times m_x} = \frac{1}{1 + 0.909 \times 5.172} = 0.175 < 1.0$ per F.112

ψ = $2.25 + 0.07a = 2.507$ where:

$$a = 1.54\frac{I_t}{I_y}\left(\frac{l_{ef}}{h}\right)^2 = 1.54\frac{458}{11{,}720}\left(\frac{5{,}000}{640}\right)^2 = 3.673 \quad \text{(Eq. G. 4)}$$

ϕ_1 = $\psi\frac{I_y}{I_x}\left(\frac{h}{l_{ef}}\right)^2\frac{E}{R_y} = 2.507\frac{11{,}720}{175{,}200}\left(\frac{640}{5{,}000}\right)^2\frac{206{,}000}{223.8} = 2.529$

ϕ_b = $0.68 + 0.21 \times \phi_1 = 1.211 > 1.0$, take $\phi_b = 1.0$

c_{10} = $\frac{1}{1 + m_x(\phi_1 / \phi_b)} = \frac{1}{1 + 5.172 \times (0.826 / 1)} = 0.190$ per F.113

Therefore, $c_{max} = 0.175(2 - 0.2 \times 5.172) + 0.190(0.2 \times 5.172 - 1) = 0.176$

So, the ratio is $\frac{80}{0.176 \times 0.826 \times 242(10)^{-1} \times 223.9 \times 1} = 0.102 < 1$

<u>Design for Stability (Cl. 9.2.9)</u>

$$\frac{N}{\phi_{exy} \times A \times R_y \times \gamma_c} \le 1 \quad \text{(F.111)}$$

where

ϕ_{ey} = 0.333 (per Table Д.3)

ϕ_{exy} = $\phi_{ey}\left(0.6\sqrt[3]{c} + 0.4\sqrt[4]{c}\right) = 0.333\left(0.6\sqrt[3]{0.176} + 0.4\sqrt[4]{0.176}\right) = 0.198$

So, the ratio is $\frac{80}{0.198 \times 242(10)^{-1} \times 223.9 \times 1} = 0.075 < 1$

Results

Result Type	**Reference**	**STAAD.Pro**	**Difference**	**Comments**
Ratio per Cl. 9.1.1	0.127	0.127	none	
Ratio per Cl. 9.2.2	0.073	0.0729	negligible	
Ratio per Cl. 9.2.4	0.102	0.099	negligible	
Ratio per Cl. 9.2.9	0.075	0.074	negligible	
m_{ef}	7.18	7.16	negligible	
m_x	5.172	5.18	negligible	
C	0.176	0.179	negligible	
ϕ_{ey}	0.333	0.33	negligible	
ϕ_{exy}	0.198	0.199	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\Russia\SNiP SP16 2017 - I section with axial force and Bi-moment.std` is typically installed with the program.

```
STAAD SPACE
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY EUROPEAN
1 TABLE ST HE650A
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -80
MEMBER LOAD
1 UNI GX 30
1 UNI GZ 2
PERFORM ANALYSIS
PARAMETER 1
CODE RUSSIAN
TB 2 ALL
GAMM 2 ALL
ENSGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (SP 16.13330.2017)   V1.0
                       ********************************************
   ALL UNITS ARE - KN METRE
   ========================================================================
   MEMBER     CROSS          RESULT/   CRITICAL COND/    RATIO/    LOADING/
              SECTION NO.      N             Mx            My      LOCATION
   ========================================================================
      1  I      HE650A         PASS      SP cl.9.1.1      0.13         1
                            8.000E+01 C    9.375E+01    6.250E+00   2.500E+00
      1  I      HE650A         PASS      SP cl.9.2.2      0.07         1
```

```
                          8.000E+01 C    9.375E+01    6.250E+00   2.500E+00
      1  I      HE650A        PASS      SP cl.9.2.4      0.10         1
                          8.000E+01 C    9.375E+01    6.250E+00   2.500E+00
      1  I      HE650A        PASS      SP cl.9.2.9      0.07         1
                          8.000E+01 C    9.375E+01    6.250E+00   2.500E+00
   MATERIAL DATA
      Steel                          = S235       EN10025-2
      Modulus of elasticity          = 206.E+06 kPa
      Design Strength (Ry)           = 224.E+03 kPa
   SECTION PROPERTIES (units - m, m^2, m^3, m^4)
      Member Length                  = 5.00E+00
      Gross Area                     = 2.42E-02
      Net Area                       = 2.42E-02
                                         x-axis      y-axis
      Moment of inertia (I)          :   175.E-05    117.E-06
      Section modulus (W)            :   548.E-05    781.E-06
      First moment of area (S)       :   307.E-05    603.E-06
      Radius of gyration (i)         :   269.E-03    696.E-04
      Effective Length               :   5.00E+00    5.00E+00
      Slenderness                    :   186.E-01    718.E-01
   DESIGN DATA (units -kN,m) SP16.13330.2017
      Axial force                    :   800.0E-01
                                         x-axis      y-axis
      Moments                        :   937.5E-01    625.0E-02
      Shear force                    :   0.000E+00    0.000E+00
      Bi-moment                      :   0.000E+00 Value of Bi-moment not
being entered!!!
      Stress-strain state checked as:   Class    2
   CRITICAL CONDITIONS FOR EACH CLAUSE CHECK
      F.(106) (N/A+Mx*y/Ix+My*x/Iy+B*w/Iw)/(Ry*GammaC)=
              ( 800.0E-01/ 2.4E-02+ 937.5E-01* 3.20E-01/ 1.75E-03+ 625.0E-02*
1.50E-01/
               1.17E-04+ 0.000E+00* 2.50E-01/ 1.10E-05)/( 223.8E+03* 1.00E+00)
               = 1.27E-01=&lt;1
      cl.9.2.2  m_ef=eta*mx= 1.38E+00* 5.18E+00= 7.16E+00
      F.(109)  N/(FIe*A*Ry*GammaC)= 800.0E-01/( 2.03E-01* 2.42E-02* 223.8E
+03* 1.00E+00)
               = 7.29E-02=&lt;1
      F.(114)  c=c5(2-0.2*mx)+c10*(0.2*mx-1.0)=
                1.82E-01*(2-0.2* 5.18E+00)+ 1.08E-01*(0.2* 5.18E+00-1.0)=
1.79E-01
            c_max= 3.20E-01
      F.(111)  N/(c*FIy*A*Ry*GammaC)= 8.00E+01/(1.79E-01*8.26E-01* 2.42E-02*
223.8E+03* 100.E-02)
               = 998.E-04=&lt;1
      F.(117)  FIexy=FIey*(0.6*c**(1/3)+0.4*c**(1/4))=
               3.33E-01*(0.6*1.79E-01**(1/3)+0.4*1.79E-01**(1/4))= 1.99E-01
      F.(116) N/(FIexy*A*Ry*GammaC)= 800.0E-01/( 1.99E-01* 2.42E-02* 223.8E
+03*1.00E+00)
               =7.41E-02=&lt;1
```

V. South Africa

V.I Section in Bending

Determine the capacity of a South African I-section beam in bending per South African steel design code (SANS10162-1:1993). The beam has torsional and simple lateral rotational restraint at the supports, and the applied point load provides effective lateral restraint at the point of application is braced at its ends for both axes.

Reference

Example 4.5, page 4.37, *Structural Steel Design to SABS 0162-1 1993 (Limit States Design)*. Greg Parrott, 1st edition, Shades Technical Publications. 1993

Related Links

- *D14.B.6 Member Resistances* (on page 1958)

Given

FYLD = 300 Mpa

Comparison

Table 702: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Major Axis Bending Resistance (kN)	353.4	364.5	3%

Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
*ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 7 0 0
MEMBER INCIDENCES
1 1 3; 2 3 2
MEMBER PROPERTY SAFRICAN
1 2 TABLE ST 406X67UB
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2.00E+008
POISSON 3
DENSITY 977
ISOTROPIC STEEL
E 2.00E+008
POISSON 3
DENSITY 8195
```

```
ALPHA 2E-005
DAMP 03
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL MEMB 1 2
UNIT METER KN
SUPPORTS
1 3 PINNED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -104 4
1 UNI GY -4
2 UNI GY -2
PERFORM ANALYSIS
PARAMETER
CODE SANS10162-1: 1993
CB 0 ALL
UNL 4 MEMB 1
FU 450000 ALL
BEAM 1 ALL
NSF 85 ALL
FYLD 300000 ALL
TRACK 2 ALL
CHECK CODE MEMB 1
FINISH
```

Output

```
                **************************************
                       STAAD.PRO CODE CHECKING
                (SOUTHAFRICAN STEEL/SANS10162-01:1993 )
                **************************************
  PROGRAM CODE REVISION V1.0_SANS10162-1: 1993
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
=======================================================================
       1 ST   406X67UB                 (SOUTHAFRICAN SECTIONS)
                         PASS       SAN-13.8           0.526         1
                         0.00           0.00       -191.90        4.08
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   SECTION CLASS =   1
   CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  7.00E+02
   IZ =  2.43E+04   SZ =  1.19E+03   PZ =  1.35E+03
   IY =  1.36E+03   SY =  1.52E+02   PY =  2.37E+02
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 450.0
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
   CRY =  4.532E+02   CRZ =  2.016E+03
   CTORFLX =  4.532E+02
   TENSILE CAPACITY     =  2.308E+03   COMPRESSIVE CAPACITY =  4.532E+02
   FACTORED MOMENT RESISTANCE : MRY =  6.399E+01   MRZ =  3.645E+02
   FACTORED SHEAR RESISTANCE  : VRY =  6.420E+02   VRZ =  6.075E+02
```

```
  MISCELLANEOUS INFORMATION
  --------------------------
  NET SECTION FACTOR FOR TENSION = 85.000
  UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  4.000
  OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.75
  SHEAR FORCE (KNS) : Y AXIS = -6.305E+01   Z AXIS =  0.000E+00
  SLENDERNESS RATIO OF WEB (H/W) =  4.33E+01
  KL/RY =  175.514   KL/RZ =   41.522   ALLOWABLE KL/R =  300.000
```

V. I Section in Compression

Determine the capacity of a South African I-section column in axial compression per South African steel design code (SANS10162-1:1993). Column is braced at its ends for both axes.

Reference

Example 4.3.4.1, page 4.18, *Structural Steel Design to SABS 0162-1 1993 (Limit States Design)*. Greg Parrott, 1st edition, Shades Technical Publications. 1993

Related Links

- *D14.B.6 Member Resistances* (on page 1958)

Given

FYLD = 300 Mpa

Length = 6,000 mm

Comparison

Table 703: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Axial Compressive Strength (kN)	1,516	1,516	none

Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
* ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY SAFRICAN
1 TABLE ST 356X67UB
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 1.99947e+008
POISSON 0.3
DENSITY 76.8191
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 248210 FU 399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KN
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1500
PERFORM ANALYSIS
PARAMETER 1
CODE SANS10162-1: 1993
LZ 6 ALL
LY 3 ALL
FU 450000 ALL
BEAM 1 ALL
NSF 0.85 ALL
TRACK 2 ALL
FYLD 300000 ALL
CHECK CODE ALL
FINISH
```

Output

```
                ***************************************
                       STAAD.PRO CODE CHECKING
                (SOUTHAFRICAN STEEL/SANS10162-01:1993 )
                ***************************************
  PROGRAM CODE REVISION V1.0_SANS10162-1: 1993
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
=======================================================================
       1 ST   356X67UB                 (SOUTHAFRICAN SECTIONS)
                         PASS       COMPRESSION        0.989         1
                      1500.00           0.00           0.00        0.00
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   SECTION CLASS =   1
   CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  6.00E+02
   IZ =  1.95E+04   SZ =  1.07E+03   PZ =  1.21E+03
   IY =  1.36E+03   SY =  1.57E+02   PY =  2.43E+02
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 450.0
   SECTION CAPACITIES (UNIT - KN,M)
   ---------------------------------
```

```
  CRY =  1.516E+03   CRZ =  2.038E+03
  CTORFLX =  1.516E+03
  TENSILE CAPACITY     =  2.308E+03   COMPRESSIVE CAPACITY =  1.516E+03
  FACTORED MOMENT RESISTANCE : MRY =  6.561E+01   MRZ =  3.267E+02
  FACTORED SHEAR RESISTANCE  : VRY =  5.903E+02   VRZ =  6.461E+02
  MISCELLANEOUS INFORMATION
  --------------------------
  NET SECTION FACTOR FOR TENSION =  0.850
  UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  6.000
  OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
  SHEAR FORCE (KNS) : Y AXIS =  0.000E+00   Z AXIS =  0.000E+00
  SLENDERNESS RATIO OF WEB (H/W) =  3.65E+01
  KL/RY =   75.220   KL/RZ =   39.730   ALLOWABLE KL/R =  200.000
```

V. I Section in Shear

Determine the elastic shear capacity per South African steel design code (SANS10162-1:1993) of a South African I-section which is simply supported over the span of 8 m.

Reference

Example 4.6.5, page 4.54, *Structural Steel Design to SABS 0162-1 1993 (Limit States Design)*. Greg Parrott, 1st edition, Shades Technical Publications. 1993

Related Links

- *D14.B.6 Member Resistances* (on page 1958)

Given

FYLD = 300 Mpa

Comparison

Table 704: SAB 0162-1:1993 Verification Problem 3 comparison

Criteria	Reference	STAAD.Pro	Difference
Shear Capacity (kN)	687.1	687.1	none

Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
*ENGINEER DATE
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 8 0 0
MEMBER INCIDENCES
1 1 2
```

```
MEMBER PROPERTY SAFRICAN
1 TABLE ST 457X67UB
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 2E+008
POISSON 3
DENSITY 977
ISOTROPIC STEEL
E 2E+008
POISSON 3
DENSITY 8195
ALPHA 2E-005
DAMP 03
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL MEMB 1
UNIT METER KN
SUPPORTS
1 2 PINNED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -70
PERFORM ANALYSIS
PARAMETER
CODE SANS10162-1: 1993
FU 450000 ALL
BEAM 1 ALL
FYLD 300000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

Output

```
                ***************************************
                       STAAD.PRO CODE CHECKING
                (SOUTHAFRICAN STEEL/SANS10162-01:1993 )
                ***************************************
  PROGRAM CODE REVISION V1.0_SANS10162-1: 1993
 ALL UNITS ARE - KNS  MET  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
=======================================================================
*     1 ST   457X67UB                (SOUTHAFRICAN SECTIONS)
                         FAIL       SAN-13.8          1.411         1
                         0.00           0.00       -560.00        4.00
   MEMBER PROPERTIES (UNIT = CM)
   -----------------------------
   SECTION CLASS =   1
   CROSS SECTION AREA =  8.55E+01   MEMBER LENGTH =  8.00E+02
   IZ =  2.94E+04   SZ =  1.30E+03   PZ =  1.47E+03
   IY =  1.45E+03   SY =  1.53E+02   PY =  2.37E+02
   MATERIAL PROPERTIES (UNIT = MPA)
   --------------------------------
   FYLD = 300.0   FU = 450.0
   SECTION CAPACITIES (UNIT - KN,M)
```

```
--------------------------------
CRY =  3.738E+02   CRZ =  1.996E+03
CTORFLX =  3.738E+02
TENSILE CAPACITY     =  2.308E+03   COMPRESSIVE CAPACITY =  3.738E+02
FACTORED MOMENT RESISTANCE : MRY =  6.399E+01   MRZ =  3.969E+02
FACTORED SHEAR RESISTANCE  : VRY =  6.871E+02   VRZ =  5.730E+02
MISCELLANEOUS INFORMATION
--------------------------
NET SECTION FACTOR FOR TENSION =  1.000
UNSUPPORTED LENGTH OF THE COMPRESSION FLANGE (M) =  8.000
OMEGA-1 (Y-AXIS) = 1.00   OMEGA-1 (Z-AXIS) = 1.00   OMEGA-2 = 1.00
SHEAR FORCE (KNS) : Y AXIS =  1.697E-05   Z AXIS =  0.000E+00
SLENDERNESS RATIO OF WEB (H/W) =  5.04E+01
KL/RY =  194.263   KL/RZ =   43.142   ALLOWABLE KL/R =  300.000
```

V. United Kingdom

V. BS5950 2000 - Fully Restrained Simply Supported Beam

A 6.5 m, simply-supported beam is fully restrained along its length. The beam is designed in S275 steel for the loading described.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 2, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

The beam is designed in S275 steel for the loading described.

Dead Loads, $\gamma_{fd} = 1.4$:

- Distributed load (including s/w) $w_d = 15\ kN/m$
- Point Load $W_d = 40\ kN$

Imposed Loads, $\gamma_{fi} = 1.6$:

- Distributed load $W_i = 30\ kN/m$
- Point Load $W_i = 50\ kN$

Comparison

Table 705: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Shear capacity, P_v(kN)	888	888.4	none

Result Type	Theory	STAAD.Pro	Difference
Moment capacity, M_{cx} (kN-m)	649	649	none
Total deflection, δ (mm)	8.69	9.25[a]	6.4%
Deflection limit (mm)	18.1	18.056	none

a. STAAD.Pro includes effects of shear deformation, resulting in a higher calculated maximum deflection.

STAAD Input

`TRACK 2 ALL`	Maximum detail output
`UNI 0 ALL`	Identifies the beam as fully restrained
`DEF 360 ALL`	Limiting ratio of beam length to maximum deflection
`DJ1 1 ALL`	Identifies starting joint of "physical member" for deflection check
`DJ2 3 ALL`	Identifies ending joint of "physical member" for deflection check

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Fully Restrained Simply Supported Beam.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Fully Restrained Simply Supported Beam.STD` is typically installed with the program.


```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 2
JOB CLIENT The Steel Construction Institute
JOB COMMENT Simply supported restrained beam
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
*********************************************
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3.25 0 0; 3 6.5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 2 TABLE ST UB533X210X92
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 FIXED BUT MZ
*********************************************
* Loading
LOAD 1 DEAD
JOINT LOAD
2 FY -40
MEMBER LOAD
1 2 UNI GY -15
*********************************************
LOAD 2 LIVE
JOINT LOAD
2 FY -50
MEMBER LOAD
1 2 UNI GY -30
*********************************************
LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK
*********************************************
* First check the forces
PARAMETER 1
CODE BS5950
TRACK 2 ALL
UNL 0 ALL
CHECK CODE ALL
*********************************************
* Second check the displacements
UNIT MMS KN
LOAD LIST 2
PARAMETER 2
CODE BS5950
DFF 360 ALL
DJ1 1 ALL
DJ2 3 ALL
TRACK 4 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                              -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST UB533X210X92  PASS     BS-4.3.6           0.902      100
                        0.00            0.00        585.41        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
```

```
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    325.00
      Gross Area =  117.00  Net Area =  117.00  Eff. Area =  117.00
                                 z-z  axis       y-y  axis
      Moment of inertia        :    55200.004       2390.000
      Plastic modulus          :     2360.000        355.000
      Elastic modulus          :     2070.906        228.380
      Effective modulus        :     2360.000        355.000
      Shear Area               :       58.771         53.843
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
                                 z-z  axis       y-y  axis
      Moment Capacity          :        649.0           94.2
      Reduced Moment Capacity  :        649.0           94.2
      Shear Capacity           :        969.7          888.4
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   649.00,    0.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   1.00  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  585.41 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.329   100      -     292.3      -       -       -
   BS-4.3.6       0.902   100      -     292.3      -    585.4      -
    Torsion and deflections have not been considered in the design.
                         _________________________
      STAAD SPACE                                             -- PAGE NO.
6
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST UB533X210X92  PASS     BS-4.3.6           0.902      100
                        0.00           0.00         585.41       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    325.00
      Gross Area =  117.00  Net Area =  117.00  Eff. Area =  117.00
                                 z-z  axis       y-y  axis
      Moment of inertia        :    55200.004       2390.000
      Plastic modulus          :     2360.000        355.000
      Elastic modulus          :     2070.906        228.380
      Effective modulus        :     2360.000        355.000
      Shear Area               :       58.771         53.843
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
                                 z-z  axis       y-y  axis
      Moment Capacity          :        649.0           94.2
      Reduced Moment Capacity  :        649.0           94.2
      Shear Capacity           :        969.7          888.4
   BUCKLING CALCULATIONS (units - kN,m)
```

```
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   649.00,    0.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   1.00  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  585.41 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.329   100      -     292.3      -       -       -
   BS-4.3.6       0.902   100      -      68.0      -    585.4      -
    Torsion and deflections have not been considered in the design.
                            _________________________
      STAAD SPACE                                              -- PAGE NO.
7
   ************** END OF TABULATED RESULT OF DESIGN **************
    53. *********************************************
    54. * SECOND CHECK THE DISPLACEMENTS
    55. UNIT MMS KN
    56. LOAD LIST 2
    57. PARAMETER 2
    58. CODE BS5950
    59. DFF 360 ALL
    60. DJ1 1 ALL
    61. DJ2 3 ALL
    62. TRACK 4 ALL
    63. CHECK CODE ALL
 STEEL DESIGN
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                              -- PAGE NO.
8
 CLAUSE BS-2.5.1
CHECKS
 LENGTH UNITS -
MMS
 MEMBER     TABLE         RESULT   ACTUAL    DEFL.    DEFL.LEN/
LOAD/
                                   DEFL.     LIMIT       DFF
LOCATION

=======================================================================

       1 ST UB533X210X92   PASS      9.255    18.056  6500.000
2
                                                       360.000
3249.67
       2 ST UB533X210X92   PASS      9.255    18.056  6500.000
2
                                                       360.000
0.00
   ************** END OF TABULATED RESULT OF DESIGN **************
```

V. BS5950 2000 - Unrestrained Simply Supported Beam

Design of a 9 m, unrestrained beam with end moments.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 3, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

Intermediate point loads are applied to the bottom flange. These do not provide restraint against lateral-torsional buckling. The beam is designed in S275 steel.

Dead Loads, $\gamma_{fd} = 1.4$:

- Selfweight $w_s = 3\ kN/m$
- Point Load $W_{1d} = 40\ kN$
- Point Load $W_{2d} = 20\ kN$

Imposed Loads, $\gamma_{fi} = 1.6$:

- Point Load $W_{1i} = 60\ kN$
- Point Load $W_{2i} = 30\ kN$

Comparison

Table 706: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Shear capacity, P_v (kN)	636	635.9	none
Moment capacity, M_{cx} (kN-m)	404	404.5	none
Lateral-torsional buckling capacity, M_b (kN-m)	150	150.26	none

STAAD Input

TRACK 2 ALL	Maximum detail output
MLT 0.46ALL	Specifies the m_{LT} value to use in the lateral-torsional buckling calculations
UNI 6.3 ALL	Specifies the lateral-torsional buckling length to be used

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Unrestrained Simply Supported Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
JOB NAME Example no. 3
JOB CLIENT The Steel Construction Institute
JOB COMMENT Unrestrained beam with end moments
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TO 3 TABLE ST UB457X191X67
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
*************************************
LOAD 1 DEAD
JOINT LOAD
2 FY -40
MEMBER LOAD
1 TO 3 UNI GY -3
UNIT MMS KN
JOINT LOAD
3 FY -20
UNIT METER KN
LOAD 2 LIVE
JOINT LOAD
2 FY -60
UNIT MMS KN
JOINT LOAD
3 FY -30
UNIT METER KN
LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
TRACK 2 ALL
MLT 0.46 ALL
UNL 6.3 ALL
CHECK CODE ALL
FINISH
***************************************
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                              -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
       1 ST UB457X191X67  PASS     BS-4.3.6          0.861       100
                         0.00           0.00        280.96        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    300.00
      Gross Area =   85.50  Net Area =   85.50  Eff. Area =   85.50
                                   z-z  axis       y-y  axis
      Moment of inertia        :    29400.002        1450.000
      Plastic modulus          :     1470.000         237.000
      Elastic modulus          :     1296.868         152.712
      Effective modulus        :     1470.000         237.000
      Shear Area               :       43.411          38.539
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
                                   z-z  axis       y-y  axis
      Moment Capacity          :        404.3           63.0
      Reduced Moment Capacity  :        404.3           63.0
      Shear Capacity           :        716.3          635.9
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   150.11,    6.300
      LTB Coefficients & Associated Moments (kNm):
      mLT =   0.46  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  280.96 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.238   100      -     151.0      -       -       -
   BS-4.3.6       0.861   100      -     151.0      -    281.0      -
    Torsion and deflections have not been considered in the design.
                         _________________________
      STAAD SPACE                                              -- PAGE NO.
6
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
       2 ST UB457X191X67  PASS     BS-4.3.6          0.470       100
                         0.00           0.00        153.25        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
```

```
    SECTION PROPERTIES (units - cm)
       Member Length =    300.00
       Gross Area =   85.50  Net Area =   85.50  Eff. Area =   85.50
                                  z-z  axis       y-y  axis
       Moment of inertia        :    29400.002       1450.000
       Plastic modulus          :     1470.000        237.000
       Elastic modulus          :     1296.868        152.712
       Effective modulus        :     1470.000        237.000
       Shear Area               :       43.411         38.539
    DESIGN DATA (units - kN,m)   BS5950-1/2000
       Section Class            :   PLASTIC
                                  z-z  axis       y-y  axis
       Moment Capacity          :        404.3          63.0
       Reduced Moment Capacity  :        404.3          63.0
       Shear Capacity           :        716.3         635.9
    BUCKLING CALCULATIONS (units - kN,m)
    (axis nomenclature as per design code)
       LTB Moment Capacity (kNm) and LTB Length (m):   150.11,    6.300
       LTB Coefficients & Associated Moments (kNm):
       mLT =   0.46  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
       Mlt =  153.25 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
    CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
     CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
    BS-4.2.3-(Y)   0.041   100      -      26.2      -       -       -
    BS-4.3.6       0.470   100      -      13.6      -    153.2      -
     Torsion and deflections have not been considered in the design.
                        _________________________
       STAAD SPACE                                              -- PAGE NO.
7
  ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
  MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                           FX            MY             MZ       LOCATION
  =======================================================================
        3 ST UB457X191X67  PASS     BS-4.3.6           0.710       100
                         0.00           0.00         231.74        0.00
  =======================================================================
    MATERIAL DATA
       Grade of steel           =  S 275
       Modulus of elasticity    =  205 kN/mm2
       Design Strength  (py)    =  275  N/mm2
    SECTION PROPERTIES (units - cm)
       Member Length =    300.00
       Gross Area =   85.50  Net Area =   85.50  Eff. Area =   85.50
                                  z-z  axis       y-y  axis
       Moment of inertia        :    29400.002       1450.000
       Plastic modulus          :     1470.000        237.000
       Elastic modulus          :     1296.868        152.712
       Effective modulus        :     1470.000        237.000
       Shear Area               :       43.411         38.539
    DESIGN DATA (units - kN,m)   BS5950-1/2000
       Section Class            :   PLASTIC
                                  z-z  axis       y-y  axis
       Moment Capacity          :        404.3          63.0
       Reduced Moment Capacity  :        404.3          63.0
       Shear Capacity           :        716.3         635.9
    BUCKLING CALCULATIONS (units - kN,m)
    (axis nomenclature as per design code)
       LTB Moment Capacity (kNm) and LTB Length (m):   150.11,    6.300
```

```
   LTB Coefficients & Associated Moments (kNm):
   mLT =   0.46  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
   Mlt =  231.74 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
 CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
BS-4.2.3-(Y)   0.180  100      -      114.8      -       -       -
BS-4.3.6       0.710  100      -      102.2      -    231.7      -
 Torsion and deflections have not been considered in the design.
```

V. BS5950 2000 - Beam from UB Restrained at Loading

Design of a simply-supported, 9m beam, which is laterally restrained at the ends and at the points of load application only.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 4, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

Dead Loads, $\gamma_{fd} = 1.4$:

- Selfweight $w_s = 3\ kN/m$
- Point Load $W_{1d} = 40\ kN$
- Point Load $W_{2d} = 20\ kN$

Imposed Loads, $\gamma_{fi} = 1.6$:

- Point Load $W_{1i} = 60\ kN$
- Point Load $W_{2i} = 30\ kN$

Comparison

Table 707: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Shear capacity, P_v (kN)	751	751.4	none
Moment capacity, M_{cx} (kN-m)	503	503.5	none
Lateral-torsional bucking moment, M_b (kN-m)	397	396.6	none

STAAD Input

TRACK 2 ALL	Maximum detail output
MLT _MAIN J1 U2 U3 J4	Specifies that m_{LT} is to be calculated with the end joints fully restrained and the intermediate joints restraining the upper flange

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Beam from UB Restrained at Loading.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 4
JOB CLIENT The Steel Construction Institute
JOB COMMENT Simply supported beam with lateral restraint points
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
*************************************
START GROUP DEFINITION
MEMBER
_MAIN 1 TO 3
END GROUP DEFINITION
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TO 3 TABLE ST UB457X191X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED BUT MZ
*************************************
LOAD 1 DEAD
JOINT LOAD
2 FY -40
MEMBER LOAD
1 TO 3 UNI GY -3
UNIT MMS KN
JOINT LOAD
3 FY -20
UNIT METER KN
```

```
LOAD 2 LIVE
JOINT LOAD
2 FY -60
UNIT MMS KN
JOINT LOAD
3 FY -30
LOAD COMB 100 COMBINATION LOAD CASE 3
1 1.4 2 1.6
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
TRACK 2 ALL
MLT _MAIN J1 U2 U3 J4
CHECK CODE ALL
FINISH
***************************************
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                              -- PAGE NO.
5
 ALL UNITS ARE - KN   MMS  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST UB457X191X82  PASS     BS-4.3.6          0.830      100
                         0.00            0.00      417800.03       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    300.00
      Gross Area =  104.00  Net Area =  104.00  Eff. Area =  104.00
                                z-z  axis       y-y  axis
      Moment of inertia        :    37100.004       1870.000
      Plastic modulus          :     1830.000        304.000
      Elastic modulus          :     1613.044        195.504
      Effective modulus        :     1830.000        304.000
      Shear Area               :       55.094         45.540
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
                                z-z  axis       y-y  axis
      Moment Capacity          :        503.3          80.6
      Reduced Moment Capacity  :        503.3          80.6
      Shear Capacity           :        909.1         751.4
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   396.39,    3.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   0.61  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  417.80 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
```

```
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.194   100      -     145.6      -       -       -
   BS-4.3.6       0.830   100      -     145.6      -    417.8      -
    Torsion and deflections have not been considered in the design.
                          _________________________
      STAAD SPACE                                              -- PAGE NO.
6
 ALL UNITS ARE - KN   MMS  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST UB457X191X82  PASS     BS-4.3.6           0.986       100
                        0.00           0.00      417800.03        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  275  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    300.00
      Gross Area =  104.00  Net Area =  104.00  Eff. Area =  104.00
                                    z-z  axis       y-y  axis
      Moment of inertia        :    37100.004        1870.000
      Plastic modulus          :     1830.000         304.000
      Elastic modulus          :     1613.044         195.504
      Effective modulus        :     1830.000         304.000
      Shear Area               :       55.094          45.540
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   PLASTIC
                                    z-z  axis       y-y  axis
      Moment Capacity          :        503.3           80.6
      Reduced Moment Capacity  :        503.3           80.6
      Shear Capacity           :        909.1          751.4
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   396.39,    3.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   0.94  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  417.80 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.042   100      -      31.6      -       -       -
   BS-4.3.6       0.986   100      -      19.0      -    417.8      -
    Torsion and deflections have not been considered in the design.
                          _________________________
      STAAD SPACE                                              -- PAGE NO.
7
 ALL UNITS ARE - KN   MMS  (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       3 ST UB457X191X82  PASS     BS-4.3.6           0.679       100
                        0.00           0.00      341800.03        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
```

```
   Design Strength  (py)    =  275  N/mm2
SECTION PROPERTIES (units - cm)
   Member Length =    300.00
   Gross Area =  104.00  Net Area =  104.00  Eff. Area =  104.00
                              z-z  axis       y-y  axis
   Moment of inertia        :    37100.004       1870.000
   Plastic modulus          :     1830.000        304.000
   Elastic modulus          :     1613.044        195.504
   Effective modulus        :     1830.000        304.000
   Shear Area               :       55.094         45.540
DESIGN DATA (units - kN,m)   BS5950-1/2000
   Section Class            :   PLASTIC
                              z-z  axis       y-y  axis
   Moment Capacity          :        503.3          80.6
   Reduced Moment Capacity  :        503.3          80.6
   Shear Capacity           :        909.1         751.4
BUCKLING CALCULATIONS (units - kN,m)
(axis nomenclature as per design code)
   LTB Moment Capacity (kNm) and LTB Length (m):   396.39,    3.000
   LTB Coefficients & Associated Moments (kNm):
   mLT =   0.61  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
   Mlt =  341.80 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
 CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
BS-4.2.3-(Y)   0.160   100      -     120.2      -       -       -
BS-4.3.6       0.679   100      -     107.6      -    341.8      -
 Torsion and deflections have not been considered in the design.
```

V. BS5950 2000 - Beam from UC Restrained at Loading

Design of a simply-supported, 9m beam, which is laterally restrained at the ends and at the points of load application only.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 5, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

The beam is a universal column from S355 steel.

Loads:

- Selfweight $w_s' = 3.4\ kN/m$
- Point Load $W_1' = 122\ kN$
- Point Load $W_2' = 61\ kN$

Comparison

Table 708: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Shear capacity, P_v (kN)	465	465.5	none
Moment capacity, M_{cx} (kN-m)	350	350.5	none
Lateral-torsional buckling moment, M_b (kN-m)	333	332.17	none

STAAD Input

SGR 1 ALL	Specifies the use of Grade S355 steel
TRACK 2 ALL	Maximum detail output
MLT _MAIN J1 U2 U3 J4	Specifies that m_{LT} is to be calculated with the end joints fully restrained and the intermediate joints restraining the upper flange

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Beam from UC Restrained at Loading.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 5
JOB CLIENT The Steel Construction Institute
JOB COMMENT Simply supported beam with lateral restraint
JOB COMMENT at load application points using a class 3
JOB COMMENT UC.
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 6 0 0; 4 9 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
*************************************
START GROUP DEFINITION
MEMBER
_MAIN 1 TO 3
END GROUP DEFINITION
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
```

```
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TO 3 TABLE ST UC254X254X73
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED BUT MZ
*************************************
LOAD 100 FACTORED
JOINT LOAD
2 FY -122
3 FY -61
MEMBER LOAD
1 TO 3 UNI GY -3.4
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
SGR 1 ALL
TRACK 2 ALL
MLT _MAIN J1 U2 U3 J4
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                              -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST UC254X254X73  PASS     BS-4.3.6           0.958       100
                         0.00           0.00         335.60       0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 355
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  355  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    300.00
      Gross Area =   93.10  Net Area =   93.10  Eff. Area =   93.10
                                    z-z  axis       y-y  axis
      Moment of inertia        :    11400.002       3910.000
      Plastic modulus          :      992.000        465.000
      Elastic modulus          :      897.285        307.148
      Effective modulus        :      986.822        456.370
      Shear Area               :       65.076         21.853
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   SEMI-COMPACT
                                    z-z  axis       y-y  axis
      Moment Capacity          :        350.3          162.0
```

```
      Reduced Moment Capacity  :        350.3          162.0
      Shear Capacity           :       1386.1          465.5
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   332.21,    3.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   0.61  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  335.60 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.251   100      -     117.0      -       -       -
   BS-4.3.6       0.958   100      -     117.0      -    335.6      -
    Torsion and deflections have not been considered in the design.
                          _________________________
      STAAD SPACE                                               -- PAGE NO.
6
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
      2 ST UC254X254X73  PASS     BS-4.3.6           0.958       100
                        0.00           0.00         335.60        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 355
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  355  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    300.00
      Gross Area =   93.10  Net Area =   93.10  Eff. Area =   93.10
                                   z-z  axis       y-y  axis
      Moment of inertia        :    11400.002       3910.000
      Plastic modulus          :      992.000        465.000
      Elastic modulus          :      897.285        307.148
      Effective modulus        :      986.822        456.370
      Shear Area               :       65.076         21.853
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   SEMI-COMPACT
                                   z-z  axis       y-y  axis
      Moment Capacity          :        350.3          162.0
      Reduced Moment Capacity  :        350.3          162.0
      Shear Capacity           :       1386.1          465.5
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
      LTB Moment Capacity (kNm) and LTB Length (m):   332.21,    3.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   0.94  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =  335.60 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.2.3-(Y)   0.055   100      -      25.4      -       -       -
   BS-4.3.6       0.958   100      -      15.2      -    335.6      -
    Torsion and deflections have not been considered in the design.
                          _________________________
      STAAD SPACE                                               -- PAGE NO.
7
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
```

```
                         FX            MY              MZ       LOCATION
=======================================================================
      3 ST UC254X254X73  PASS     BS-4.3.6          0.784       100
                        0.00            0.00         274.60        0.00
=======================================================================
  MATERIAL DATA
     Grade of steel           =  S 355
     Modulus of elasticity    =  205 kN/mm2
     Design Strength  (py)    =  355  N/mm2
  SECTION PROPERTIES (units - cm)
     Member Length =    300.00
     Gross Area =   93.10  Net Area =   93.10  Eff. Area =   93.10
                                   z-z  axis       y-y  axis
     Moment of inertia        :    11400.002        3910.000
     Plastic modulus          :      992.000         465.000
     Elastic modulus          :      897.285         307.148
     Effective modulus        :      986.822         456.370
     Shear Area               :       65.076          21.853
  DESIGN DATA (units - kN,m)   BS5950-1/2000
     Section Class            :   SEMI-COMPACT
                                   z-z  axis       y-y  axis
     Moment Capacity          :        350.3          162.0
     Reduced Moment Capacity  :        350.3          162.0
     Shear Capacity           :       1386.1          465.5
  BUCKLING CALCULATIONS (units - kN,m)
  (axis nomenclature as per design code)
     LTB Moment Capacity (kNm) and LTB Length (m):   332.21,    3.000
     LTB Coefficients & Associated Moments (kNm):
     mLT =   0.61  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
     Mlt =  274.60 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
  CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
   CLAUSE         RATIO  LOAD     FX       VY      VZ      MZ      MY
  BS-4.2.3-(Y)   0.208   100      -      96.6      -       -       -
  BS-4.3.6       0.784   100      -      86.4      -    274.6      -
   Torsion and deflections have not been considered in the design.
                        _________________________
```

V. BS5950 2000 - Pinned Column Using Non-Slender UC

A 6.0 m column is pin ended about both axes and has no intermediate restraints.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 6, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

The column is designed in S275 steel for the factored loading.

Point load, $F_c = 2{,}500\ kN$

Comparison

Table 709: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Slenderness about strong axis, λ_x	38.5	38.299	none
Slenderness about weak axis, λ_y	63.6	63.569	none
Compression capacity, P_c (kN)	3,100	3,090	none

STAAD Input

`TRACK 2 ALL`	Maximum detail output
`SGR 0 ALL`	Identifies steel grade as S275

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Pinned Column Using Non-Slender UC.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 10
JOB CLIENT The Steel Construction Institute
JOB COMMENT Pinned column using a non-slender UC.
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST UC356X368X129
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
```

```
*************************************
LOAD 1 AXIAL LOAD
JOINT LOAD
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
SGR 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                               -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST UC356X368X129 PASS     BS-4.7   (C)       0.809         1
                      2500.00 C          0.00          0.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 275
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  265  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =  164.00  Net Area =  164.00  Eff. Area =  164.00
                                   z-z  axis       y-y  axis
      Moment of inertia        :    40200.008       14600.002
      Plastic modulus          :     2480.000        1200.000
      Elastic modulus          :     2260.968         792.187
      Effective modulus        :     2458.506        1159.980
      Shear Area               :      116.109          36.982
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   SEMI-COMPACT
      Squash Load              :   4346.00
      Axial force/Squash load  :     0.575
                                   z-z  axis       y-y  axis
      Compression Capacity     :       3992.2          3089.3
      Shear Capacity           :       1846.1           588.0
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
                                    x-x axis        y-y axis
      Slenderness              :       38.323          63.591
      Radius of gyration (cm)  :       15.656           9.435
      Effective Length         :        6.000           6.000
      LTB Moment Capacity (kNm) and LTB Length (m):   577.91,    6.000
      LTB Coefficients & Associated Moments (kNm):
      mLT =   1.00  :   mx =   0.00   :  my =   0.00   :  myx =   0.00
      Mlt =    0.00 :   Mx =    0.00  :  My =   0.00   :   My =   0.00
```

```
      STAAD SPACE                                                -- PAGE NO.
6
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD      FX        VY      VZ      MZ      MY
   BS-4.7   (C)   0.809     1  2500.0        -       -       -       -
    Torsion and deflections have not been considered in the design.
                        _________________________
```

V. BS5950 2000 - Pinned Column Using Non-Slender RHS

A 6.0m rectangular hollow section (RHS) column is pin ended about both axes and has no intermediate restraints.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 11, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

The column is designed in S355 steel for the factored loading.

Point load, $F_c = 2{,}500\ kN$

Comparison

Table 710: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Slenderness, λ	61.4	61.424	none
Compression capacity, P_c (kN-m)	2,790	2,794.6	none

STAAD Input

TRACK 2 ALL	Maximum detail output
SGR 1 ALL	Identifies steel grade as S355

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Pinned Column Using Non-Slender RHS.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 11
```

```
JOB CLIENT The Steel Construction Institute
JOB COMMENT Pinned column using a non-slender RHS.
ENGINEER DATE Jun 2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST TUB25025010.0
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
*************************************
LOAD 1 AXIAL LOAD
JOINT LOAD
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                               -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST TUB25025010.0 PASS     BS-4.7   (C)       0.895         1
                     2500.00 C          0.00          0.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 355
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  355  N/mm2
   SECTION PROPERTIES (units - cm)
```

```
    Member Length =    600.00
    Gross Area =   94.90  Net Area =   94.90  Eff. Area =   94.90
                                 z-z  axis       y-y  axis
    Moment of inertia        :     9055.001       9055.001
    Plastic modulus          :      851.000        851.000
    Elastic modulus          :      724.400        724.400
    Effective modulus        :      851.000        851.000
    Shear Area               :       47.450         47.450
 DESIGN DATA (units - kN,m)   BS5950-1/2000
    Section Class            :   SEMI-COMPACT
    Squash Load              :   3368.95
    Axial force/Squash load  :     0.742
                                 z-z  axis       y-y  axis
    Compression Capacity     :       2794.6         2794.6
    Shear Capacity           :       1010.7         1010.7
 BUCKLING CALCULATIONS (units - kN,m)
 (axis nomenclature as per design code)
                                  x-x axis        y-y axis
    Slenderness              :       61.424         61.424
    Radius of gyration (cm)  :        9.768          9.768
    Effective Length         :        6.000          6.000
    LTB check unnecessary for this section
 CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
  CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
 BS-4.7   (C)   0.895     1  2500.0       -       -       -       -
  Torsion and deflections have not been considered in the design.
```

V. BS5950 2000 - Pinned Column Using Slender CHS

A 6.0m circular hollow section (CHS) column is pin ended about both axes and has no intermediate restraints.

Reference

Steelwork Design Guide to BS 5950-1:2000, Volume 2, Worked Examples, SCI publication P326, Example 12, The Steel Construction Institute.

Related Links

- *D3.B.5 Member Capacities* (on page 1382)

Problem

The column is designed in S355 steel for the factored loading.

Point load, $F_c = 2{,}500\ kN$

Comparison

Table 711: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Effective area, A_{eff} (cm^2)	77.6	77.63	none

Result Type	Theory	STAAD.Pro	Difference
Slenderness, λ	42.2	42.413	none
Compression capacity, P_c (kN-m)	2,561	2,562.3	none

STAAD Input

TRACK 2 ALL	Maximum detail output
SGR 1 ALL	Identifies steel grade as S355

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\UK\BS5950 2000 - Pinned Column Using Slender CHS.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Example no. 12
JOB CLIENT The Steel Construction Institute
JOB COMMENT Pinned column using a slender CHS.
ENGINEER DATE Jun-2003
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
*************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+08
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-05
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
1 TABLE ST PIP4066.3
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
*************************************
LOAD 1 AXIAL FORCE
JOINT LOAD
2 FY -2500
PERFORM ANALYSIS PRINT STATICS CHECK
***************************************
PARAMETER 1
CODE BS5950
SGR 1 ALL
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (BSI )
                         ***********************
  PROGRAM CODE REVISION V2.13_5950-1_2000
      STAAD SPACE                                                 -- PAGE NO.
5
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST PIP4066.3     PASS     BS-4.7   (C)       0.976         1
                      2500.00 C          0.00           0.00        0.00
 =======================================================================
   MATERIAL DATA
      Grade of steel           =  S 355
      Modulus of elasticity    =  205 kN/mm2
      Design Strength  (py)    =  355  N/mm2
   SECTION PROPERTIES (units - cm)
      Member Length =    600.00
      Gross Area =   79.20  Net Area =   79.20  Eff. Area =   77.63
                                   z-z  axis       y-y  axis
      Moment of inertia        :    15850.002       15850.002
      Plastic modulus          :     1009.000        1009.000
      Elastic modulus          :      780.020         780.020
      Effective modulus        :      780.020         780.020
      Shear Area               :       47.520          47.520
   DESIGN DATA (units - kN,m)   BS5950-1/2000
      Section Class            :   SLENDER
      Squash Load              :   2811.60
      Axial force/Squash load  :     0.889
                                   z-z  axis       y-y  axis
      Compression Capacity     :       2562.3          2562.3
      Shear Capacity           :       1012.2          1012.2
   BUCKLING CALCULATIONS (units - kN,m)
   (axis nomenclature as per design code)
                                    x-x axis        y-y axis
      Slenderness              :       42.413          42.413
      Radius of gyration (cm)  :       14.147          14.147
      Effective Length         :        6.000           6.000
      LTB check unnecessary for this section
   CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):
    CLAUSE        RATIO  LOAD     FX       VY      VZ      MZ      MY
   BS-4.7   (C)   0.976     1  2500.0       -       -       -       -
    Torsion and deflections have not been considered in the design.
                         _________________________
```

V. United States

V. AASHTO

V. AASHTO 17th Ed ASD - Design Frame

The following compares the solution of a design performed using STAAD.Pro against a hand calculation.

Reference

Following step by step hand calculation of Allowable Stress Steel Design per the AASHTO *Standard Specifications for Highway Bridges*, 17th Edition (2002).

Related Links

- *D1.D.1 AASHTO (ASD)* (on page 1176)

Problem

Determine the allowable stresses (AASHTO code) for the members of the structure as shown in figure. Also, perform a code check for these members based on the results of the analysis.

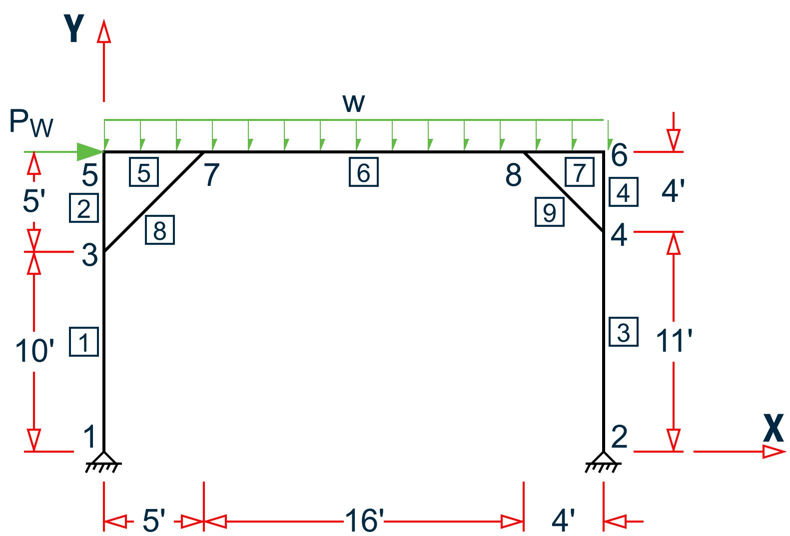


Figure 500: AASHTO ASD verification problem

Members 1, 2 = W12X26, Members 3, 4 = W14X43

Members 5, 6, 7 = W16X36, Memb8= L40404,

Member 9 = L50506

The frame is subject to the following load cases:

1. a uniform gravity load along the beam, $w = 2\ kips/ft$ (dead + live)
2. a lateral wind load, $F = 15\ kips$
3. a combination of 75% of load 1 and 75% of load 2

Solution

Only the AASHTO steel design elements are check here. No structural analysis calculations are included in these hand verifications.

Though the program does check shear per the AASHTO specifications, those calculations are not reflected here. Only the controlling stress ratios are presented.

As all members are grade 36 steel, the following critical slenderness parameter applies to each:

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = \sqrt{\frac{2\pi^2 29,\ 000}{36}} = 126.1$$

Member 1

Size W 12X26, L = 10 ft., a = 7.65 in^2, S_z = 33.39 in^3

From observation, Load case 1 will govern

- Fx = 25.0 kip (compression)
- Mz= 56.5 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$\text{FTY} = \text{FTZ} = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}}\left(\frac{I_{yc}}{l}\right)\sqrt{0.772\frac{J}{I_{yc}} + 9.87\left(\frac{d}{l}\right)^2} \le 0.55F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2)+0.3x(M1/M2)^2$$

M1 = 0, so $C_b = 1.75$

S_{zc} = Section modulus with respect to the compression flange =$204/(0.5 \cdot 12.22) = 33.38789\ in^3$

$$I_{YC} = t_b{}^3/12 = 0.38 \cdot 6.49^3/12 = 8.6564 \text{ in}^4$$

$$J = (2 \text{ x } 6.49 \cdot 0.38^3 + (12.22 - 2 \cdot 0.38) \cdot 0.23^3)/3 = 0.28389 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.75}{33.38789}\left(\frac{8.6564}{120}\right)\sqrt{0.772\frac{0.28389}{8.6564} + 9.87\left(\frac{11.46}{120}\right)^2}10^{-3} = 64.375ksi$$

which is larger than $0.55 \cdot F_Y = 19.8\ ksi$, so $F_{CZ} = 19.8\ ksi$

Axial Compression

Critical $(kL/r) = 1.0 \cdot 120/1.5038 = 79.7978$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL\ /r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(79.79)^2 36}{4\pi^2 29,\ 000}\right] = 13.58ksi$$

Actual Stress

Actual axial stress, $f_a = 25 / 7.65 = 3.26\ ksi$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 56.5 \cdot 12 / 33.4 = 1.692 \cdot 12 = 20.3\ ksi$

$$F_{ez} = \frac{\pi^2 E}{F.S.(kL / r)^2} = \frac{\pi^2 29,000}{2.12(120 / 5.17)^2} = 250.6 ksi$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{3.26}{13.58} + \frac{0.58 \cdot 20.3}{\left(1 - \frac{3.26}{250.6}\right) 19.8} + 0 = 1.122$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{3.26}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.217$$

The critical stress ratio is thus 1.217. The value calculated by STAAD is 1.218

Member 2

Size W 12X26, L = 5 ft., a = 7.65 in^2, Sz = 33.4 in^3

From observation Load case 1 will govern, Forces at the midspan are

- Fx = 8.71 kip (compression)
- Mz= 56.5 k-ft

Calculate the allowable stress as per Table 10.32.1A.

<u>Bending Minor Axis</u>

Allowable minor axis bending stress:

$$FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

<u>Bending Major Axis</u>

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}} \left(\frac{I_{yc}}{l}\right) \sqrt{0.772 \frac{J}{I_{yc}} + 9.87 \left(\frac{d}{l}\right)^2} \leq 0.55 F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2) + 0.3x(M1/M2)^2$$

M1 = 39.44 and M2 = 677.96, so $C_b = 1.69$

S_{zc} = Section modulus with respect to the compression flange =$204 / (0.5 \cdot 12.22) = 33.38789\ in^3$

$$I_{YC} = t_b{}^3/12 = 0.38 \cdot 6.49^3/12 = 8.6564 \text{ in}^4$$

$$J = (2 \text{ x } 6.49 \cdot 0.38^3 + (12.22 - 2 \cdot 0.38) \cdot 0.23^3)/3 = 0.28389 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.69}{33.38789} \left(\frac{8.6564}{60}\right) \sqrt{0.772 \frac{0.28389}{8.6564} + 9.87 \left(\frac{11.46}{60}\right)^2} 10^{-3} = 227.34 ksi$$

which is larger than $0.55 \cdot F_Y = 19.8\ ksi$, so $F_{CZ} = 19.8\ ksi$

Axial Compression

Critical $(kL/r) = 1.0 \cdot 60/1.504 = 39.92$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL\ /r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(39.92)^2 36}{4\pi^2 29,\ 000}\right] = 16.13 ksi$$

Actual Stress

Actual axial stress, $f_a = 8.71\ /\ 7.65 = 1.138\ ksi$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 56.5 \cdot 12/33.4 = 1.691 \cdot 12 = 20.3\ ksi$

$$(KL/r)_z = 1 \cdot 60/5.16 = 11.618$$

$$F_{ez} = \frac{\pi^2 E}{F.S.(kL\ /r)^2} = \frac{\pi^2 29,\ 000}{2.12(11.618)^2} = 998.5 ksi$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{1.138}{16.13} + \frac{0.58 \cdot 20.3}{\left(1 - \frac{1.138}{998.5}\right) 19.8} + 0 = 0.942$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{1.138}{0.472(36)} + \frac{20.3}{19.8} + 0 = 1.092$$

The critical stress ratio is thus 1.092. The value calculated by STAAD is 1.093.

Member 3

Size W 14X43, L = 11 ft., a = 12.6 in^2, Sz = 62.7 in^3

From observation Load case 3 will govern, Forces at the end are

- F_x = 25.5 kip (compression)
- M_z = 112.17 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$FTY = FTZ = 0.55 \cdot F_Y = 19.8\ ksi$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}}\left(\frac{I_{yc}}{l}\right)\sqrt{0.772\frac{J}{I_{yc}} + 9.87\left(\frac{d}{l}\right)^2} \le 0.55 F_y$$

Where:

C_b = 1.75 + 1.05(M1/M2)+0.3x(M1/M2)^2

M1 = 0, so $C_b = 1.75$

S_{zc} = Section modulus with respect to the compression flange = $428/(0.5 \cdot 13.66) = 62.7\ in^3$

$I_{YC} = tb^3/12 = 0.53 \cdot 8.0^3/12 = 22.61\text{in in}^4$

$J = (2 \cdot 8.0 \cdot 0.53^3 + (13.66 - 2 \cdot 0.53) \cdot 0.305^3)/3 = 0.913\ \text{in}^4$

$$F_{cz} = \frac{50(10)^6 1.75}{62.7}\left(\frac{22.61}{132}\right)\sqrt{0.772\frac{0.917}{22.61} + 9.87\left(\frac{12.6}{132}\right)^2}10^{-3} = 83.19ksi$$

which is larger than $0.55 \cdot F_Y = 19.8\ ksi$, so $F_{CZ} = 19.8\ ksi$

Axial Compression

Critical $(kL/r) = 1.0 \cdot 132/1.894 = 69.69$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL\ /r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(69.69)^2 36}{4\pi^2 29,\ 000}\right] = 14.39ksi$$

Actual Stress

Actual axial stress, $f_a = 25.5\ /\ 12.6 = 2.024\ ksi$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 112.17 \cdot 12/62.7 = 1.789 \cdot 12 = 21.467\ ksi$

$(KL/r)_z = 1 \cdot 132/5.828 = 22.649$

$$F_{ez} = \frac{\pi^2 E}{F.S.(kL\ /r)^2} = \frac{\pi^2 29,\ 000}{2.12(22.648)^2} = 263.18ksi$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right)F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right)F_{by}} = \frac{2.024}{14.39} + \frac{0.58 \cdot 21.467}{\left(1 - \frac{2.024}{263.21}\right)19.8} + 0 = 1.069$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.024}{0.472(36)} + \frac{21.467}{19.8} + 0 = 1.203$$

The critical stress ratio is thus 1.203. The value calculated by STAAD is 1.204.

Member 4

Size W 14X43, L = 4 ft., a = 12.6 in^2, Sz = 62.6 in^3

From observation Load case 3 will govern, Forces at the end are

- F_x = 8.75 kip (tension)

- M_z = 112.17 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}}\left(\frac{I_{yc}}{l}\right)\sqrt{0.772\frac{J}{I_{yc}} + 9.87\left(\frac{d}{l}\right)^2} \le 0.55F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2)+0.3 \cdot (M1/M2)^2$$

$$M1 = -191.36 \text{ Kip-in , } M2 = -1346.08 \text{ Kip-in so } C_b = 1.606$$

$$I_{YC} = tb^3/12 = 0.53 \cdot 8.0^3/12 = 22.61 \text{ in}^4$$

$$J = (2 \cdot 8.0 \cdot 0.53^3 + (13.66 - 2 \cdot 0.53) \cdot 0.305^3)/3 = 0.913 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.606}{62.6}\left(\frac{22.61}{48}\right)\sqrt{0.772\frac{0.911}{22.61} + 9.87\left(\frac{12.6}{48}\right)^2}10^{-3} = 508.8ksi$$

which is larger than $0.55 \cdot F_Y = 19.8\ ksi$, so $F_{CZ} = 19.8\ ksi$

Axial Tension

Note: No connection information is specified and no reduction of section is assumed.

$$F_t = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Actual Stress

Actual axial stress, $f_a = 8.75\ /\ 12.6 = 0.694\ ksi$

Actual bending stress = $f_{bz} = 112.17 \cdot 12/62.7 = 1.789 \cdot 12 = 21.47\ ksi$, which exceeds F_{CZ}.

$$f_{bz}/F_{CZ} = 21.47/19.8 = 1.084$$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.694}{0.472(36)} + \frac{21.467}{19.8} + 0 = 1.125$$

The critical stress ratio is thus 1.125. The value calculated by STAAD is 1.126.

Member 5

Size W 16X36, L = 5 ft., a = 10.6 in^2, Sz = 56.5 in^3

From observation Load case 3 will govern, Forces at the end are

- Fx = 14.02 kip (compression)
- Mz= 57.04 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}}\left(\frac{I_{yc}}{l}\right)\sqrt{0.772\frac{J}{I_{yc}} + 9.87\left(\frac{d}{l}\right)^2} \leq 0.55F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2)+0.3 \cdot (M1/M2)^2$$

$$M1 = 40.14,\ M2 = -684.4 \text{ so } C_b = 1.81$$

S_{zc} = Section modulus with respect to the compression flange = $448/(0.5 \cdot 15.86) = 56.5 \text{ in } in^3$

$$I_{YC} = tb^3/12 = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4$$

$$J = (2 \cdot 6.99 \cdot 0.43^3 + (15.86 - 2 \cdot 0.43) \cdot 0.29^3)/3 = 0.5 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.81}{56.5}\left(\frac{12.238}{60}\right)\sqrt{0.772\frac{0.5}{12.238} + 9.87\left(\frac{15}{60}\right)^2}10^{-3} = 263.1ksi$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

Critical $(kL/r) = 1.0 \cdot 60/1.52 = 69.69$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(39.474)^2 36}{4\pi^2 29,000}\right] = 16.15ksi$$

Actual Stress

Actual axial stress, $f_a = 14.02 / 10.6 = 1.323 \text{ ksi}$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 57.04 \cdot 12/56.5 = 1.001 \cdot 12 = 12.115 \text{ ksi}$

$$(KL/r)_z = 1 \cdot 60/6.5 = 9.231$$

$$F_{ez} = \frac{\pi^2 E}{F.S.(kL/r)^2} = \frac{\pi^2 29,000}{2.12(9.231)^2} = 1,584.4ksi$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right)F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right)F_{by}} = \frac{1.323}{16.149} + \frac{0.85 \cdot 12.115}{\left(1 - \frac{1.323}{1,584.4}\right)19.8} + 0 = 0.602$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{1.323}{0.472(36)} + \frac{12.115}{19.8} + 0 = 0.689$$

The critical stress ratio is thus 0.689. The value calculated by STAAD is 0.690.

Member 6

Size W 16X36, L = 16 ft., a = 10.6 in^2, Sz = 56.5 in^3

From observation Load case 3 will govern, Forces at the end are

- Fx = 10.2 kip (compression)
- Mz= 62.96 k-ft

Calculate the allowable stress as per Table 10.32.1A.

Bending Minor Axis

Allowable minor axis bending stress:

$$FTY = FTZ = 0.55 \cdot F_Y = 19.8 \text{ ksi}$$

Bending Major Axis

$$F_{cz} = \frac{50(10)^6 C_b}{S_{xc}}\left(\frac{I_{yc}}{l}\right)\sqrt{0.772\frac{J}{I_{yc}} + 9.87\left(\frac{d}{l}\right)^2} \leq 0.55F_y$$

Where:

$$C_b = 1.75 + 1.05(M1/M2)+0.3 \cdot (M1/M2)^2$$

$$M1 = 8.947 \; M2 = 183.05 \text{ so } C_b = 1.69$$

S_{zc} = Section modulus with respect to the compression flange = $448/(0.5 \cdot 15.86) = 56.5 \text{ in } in^3$

$$I_{YC} = tb^3/12 = 0.43 \cdot 6.99^3/12 = 12.238 \text{ in}^4$$

$$J = (2 \cdot 6.99 \cdot 0.43^3 + (15.86 - 2 \cdot 0.43) \cdot 0.29^3)/3 = 0.5 \text{ in}^4$$

$$F_{cz} = \frac{50(10)^6 1.81}{56.5}\left(\frac{12.238}{192}\right)\sqrt{0.772\frac{0.5}{12.238} + 9.87\left(\frac{15}{192}\right)^2}10^{-3} = 287.9ksi$$

which is larger than $0.55 \cdot F_Y = 19.8 \text{ ksi}$, so $F_{CZ} = 19.8 \text{ ksi}$

Axial Compression

Critical $(kL/r) = 1.0 \cdot 192/1.52 = 126.3$

As $(kL/r) > C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{\pi^2 E}{F.S.(kL/r)^2} = \frac{\pi^2 29,000}{2.12(126.3)^2} = 8.46ksi$$

Actual Stress

Actual axial stress, $f_a = 10.2 / 10.6 = 0.962 \text{ ksi}$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.42 in section 10-36 to calculate the design ratio.

Actual bending stress = $f_{bz} = 62.96 \cdot 12/56.5 = 1.114 \cdot 12 = 13.37 \text{ ksi}$

$(KL/r)_z = 1 \cdot 192/6.51 = 29.49$

$$F_{ez} = \frac{\pi^2 E}{F.S.(kL / r)^2} = \frac{\pi^2 29,000}{2.12(29.49)^2} = 155.2ksi$$

From Table 10-36A, $C_{mz} = 0.85$

Equation 10-42

$$\frac{f_a}{F_a} + \frac{C_{mz} f_{bz}}{\left(1 - \frac{f_a}{F'_{ez}}\right) F_{bz}} + \frac{C_{my} f_{by}}{\left(1 - \frac{f_a}{F'_{ey}}\right) F_{by}} = \frac{0.962}{8.46} + \frac{0.85 \cdot 13.37}{\left(1 - \frac{0.926}{155.2}\right) 19.8} + 0 = 0.691$$

For the end section, use Equation 10.43:

$$\frac{f_a}{0.472 F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{0.962}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.732$$

The critical stress ratio is thus 0.732. The value calculated by STAAD is 0.732

Note: The program gives this value when the slenderness check is suppressed (`MAIN 1.0` for member 6); otherwise the member fails as a compression member with a slenderness parameter greater than 120).

Member 7

Size W 16X36, L = 4 ft., a = 10.6 in^2, Sz = 56.5 in^3

From observation Load case 3 will govern, Forces at the midspan are

- Fx = 24.05 kip (tension)
- Mz= 62.96 k-ft

Calculate the allowable stress as per Table 10.32.1A

<u>Bending Minor Axis</u>

Allowable minor axis bending stress:

$FTY = FTZ = 0.55 \cdot F_Y = 19.8$ ksi

<u>Bending Major Axis</u>

$FCZ = 0.55 \cdot F_Y = 19.8$ ksi

<u>Axial Tension</u>

Note: No connection information is specified and no reduction of section is assumed.

$F_a = 0.55 \cdot F_Y = 19.8$ ksi

<u>Actual Stress</u>

Actual axial stress, $f_a = 24.05 / 10.6 = 2.268\ ksi$, hence, ok.

Actual bending stress = $f_{bz} = 62.96 \cdot 12/56.5 = 1.114 \cdot 12 = 13.37\ ksi$

So the combined ratio is

$f_a/F_a + f_{bz}/F_{TZ} + f_{by}/F_{TY} = 2.268/19.8 + 13.37/19.8 + 0 = 0.790$

The critical moment occurs at the end node of the beam. So we use the AASHTO equation 10.43 in section 10-36 to calculate the design ratio for the end section.

$$\frac{f_a}{0.472F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = \frac{2.268}{0.472(36)} + \frac{13.37}{19.8} + 0 = 0.809$$

The critical stress ratio is thus 0.809. The value calculated by STAAD is 0.809.

Member 8

Size L4x4x1/4, L = 7.07 ft., a = 1.938 in^2

From observation Load case 1 will govern, Forces

F_x = 23.04 kip (compression)

Calculate the allowable stress as per Table 10.32.1A

Axial Compression

Critical $(KL/r)_y = 1.0 \cdot 7.07 \cdot 12/0.795 = 106.7$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(106.7)^2 36}{4\pi^2 29,000}\right] = 10.89 ksi$$

Actual Stress

Actual axial stress, $f_a = 23.04/1.938 = 11.88$ ksi

$f_a/F_a = 11.88/10.89 = 1.091$

The value calculated by STAAD is 1.091.

Member 9

Size L5x5x3/8, L = 5.657 ft, A = 3.61 in^2

From observation Load case 1 will govern, Forces

F_x = 48.44 kip (compression)

Calculate the allowable stress as per Table 10.32.1A

Axial Compression

Critical $(KL/r)_y = 1.0 \cdot 5.567 \cdot 12/0.99 = 68.57$

As $(kL/r) < C_c$, the allow axial stress in compression is given by:

$$F_a = \frac{F_y}{F.S.}\left[1 - \frac{(kL/r)^2 F_y}{4\pi^2 E}\right] = \frac{36}{2.12}\left[1 - \frac{(68.57)^2 36}{4\pi^2 29,000}\right] = 14.47 ksi$$

Actual Stress

Actual axial stress, $f_a = 48.44/3.61 = 13.42$ ksi

$f_a/F_a = 13.42/14.47 = 0.927$

The value calculated by STAAD is 0.928.

Comparison

Table 712: Comparison of results

Member Number	STAAD.Pro Results	Hand Calculation	Difference
1	1.216	1.217	none
2	1.091	1.092	none
3	1.207	1.203	none
4	1.130	1.126	none
5	0.691	0.689	none
6	1.052	0.732	43.7% [1]
7	0.812	0.808	<1%
8	1.114	1.091	2.1%
9	0.920	0.927	<1%

Note: ([1]) The ratio is 0.832 when the slenderness check is suppressed, which results in a 13.7% difference.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AASHTO\AASHTO 17th Ed ASD - Design Frame.STD` is typically installed with the program.

```
STAAD PLANE VERIFICATION PROBLEM FOR AASHTO CODE
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
*
*  THIS DESIGN EXAMPLE IS VERIFIED BY HAND CALCULATION
*  FOLLOWING AASHTO ASD 97 CODE.
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0; 3 0 10 0; 4 25 11 0; 5 0 15 0; 6 25 15 0; 7 5 15 0;
8 21 15 0;
MEMBER INCIDENCES
1 1 3; 2 3 5; 3 2 4; 4 4 6; 5 5 7; 6 7 8; 7 8 6; 8 3 7; 9 4 8;
MEMBER PROPERTY AMERICAN
1 2 TABLE ST W12X26
3 4 TABLE ST W14X43
5 TO 7 TABLE ST W16X36
8 TABLE ST L40404
9 TABLE ST L50506
MEMBER TRUSS
8 9
DEFINE MATERIAL START
```

```
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
LOAD 1 DL + LL
MEMBER LOAD
5 TO 7 UNI Y -2
LOAD 2 WIND FROM LEFT
JOINT LOAD
5 FX 15
LOAD COMBINATION 3
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PARAMETER 1
CODE AASHTO
TRACK 0 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - ( AASHTO - ASD)   v1.0
                        ***********************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
*    1  ST   W12X26                    (AISC SECTIONS)
                           FAIL     AASHTO 10-43       1.216         1
                       25.00 C         0.00           56.50       10.00
*    2  ST   W12X26                    (AISC SECTIONS)
                           FAIL     AASHTO 10-43       1.091         1
                        8.72 C         0.00           56.50        0.00
*    3  ST   W14X43                    (AISC SECTIONS)
                           FAIL     AASHTO 10-43       1.207         3
                       25.50 C         0.00         -112.20       11.00
*    4  ST   W14X43                    (AISC SECTIONS)
                           FAIL     AASHTO 10-43       1.130         3
                        8.83 T         0.00         -112.20        0.00
     5  ST   W16X36                    (AISC SECTIONS)
                           PASS     AASHTO 10-43       0.691         3
                       14.02 C         0.00          -57.00        5.00
*    6  ST   W16X36                    (AISC SECTIONS)
                           FAIL     KL/R ratio         1.052         1
                        5.65 C         0.00          -15.25        0.00
     7  ST   W16X36                    (AISC SECTIONS)
                           PASS     AASHTO 10-43       0.812         3
                       24.13 T         0.00           63.00        0.00
*    8  ST   L40404                    (AISC SECTIONS)
                           FAIL     AASHTO 10-42       1.114         1
                       23.03 C         0.00            0.00        0.00
```

```
  **WARNING: For Memb #      8  SECTIONS WHICH ARE NOT I-SHAPED
           CANNOT BE DESIGNED FOR BENDING. AS PER THE AASHTO
           CODE 17th EDITION. PLEASE USE THE AISC 9th EDITION
           CODE FOR DESIGNING THE SECTION.
     9  ST  L50506                   (AISC SECTIONS)
                         PASS     AASHTO 10-42       0.920         3
                     48.55 C          0.00           0.00        0.00
  **WARNING: For Memb #      9  SECTIONS WHICH ARE NOT I-SHAPED
           CANNOT BE DESIGNED FOR BENDING. AS PER THE AASHTO
           CODE 17th EDITION. PLEASE USE THE AISC 9th EDITION
           CODE FOR DESIGNING THE SECTION.
```

V. AASHTO 2nd Ed LRFD - Design Beam

The following compares the solution of a design performed using STAAD.Pro against a hand calculation of steel design per the AASHTO (LRFD) code.

Details

Determine the allowable resistances (AASHTO LRFD, 1998 code) for the member of the structure as shown in figure. Also, perform a code check for the member based on the results of the analysis.

A 48 foot long, non-composite girder is assumed to be simply supported. The section is a plate girder with 16" x 1" plate flanges and a 34" x 3.6" plate web (36 in. total depth). All plates are grade 36 steel.

The beam is subject to a 9.111 kip/ft uniform load.

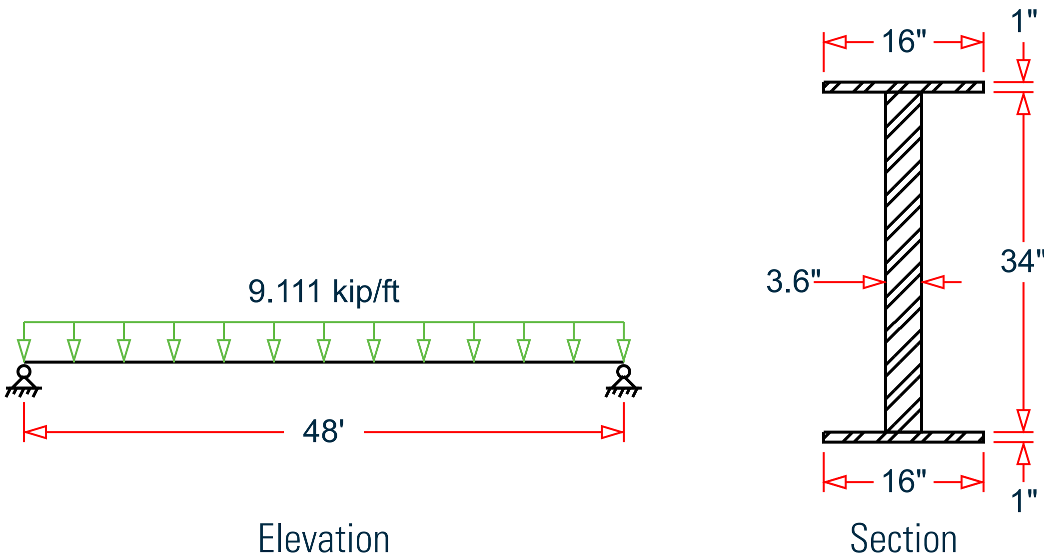


Figure 501: AASHTO LRFD verification example

Validation

$A = 154.4\ in^2$, $I_z = 21{,}594\ in^4$, $I_y = 814.9\ in^4$

$S_z = 21{,}594(2)/36 = 1{,}200\ in^3$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{814.9}{154.4}} = 2.30in$$

$$r_z = \sqrt{\frac{I_z}{A}} = \sqrt{\frac{21,\ 594}{154.4}} = 11.83in$$

From observation Load case 1 will govern,

M_z = 2,624 kip-ft

Axial Compression Capacity

Refer Clause 6.9.4 of the code.

$(kL/r)_y = 0.333 \times 576/ 2.297 = 83.50$

$(kL/r)_z = 1 \times 576/ 11.826 = 48.71$

$(kL/r)_{crit} = 83.50 < 120$, ok.

Calculation of Width/Thickness ratio for axial compression

Plate buckling coefficients taken from Table 6.9.4.2-1:

$k_w = 1.49$, $k_f = 0.56$

Slenderness ratio for the web:

$(d - 2 \cdot t_f)/t_w = (36 - 2 \cdot 1) / 3.6 = 9.444$

Slenderness ratio for the 1/2 flange:

$b_f/(2 \cdot t_f) = 16/(2 \cdot 1) = 8.0$

Critical ratio for web:

$$k_w\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,\ 000}{36}} = 42.29 > 9.444$$

Critical ratio for flange:

$$k_f\sqrt{\frac{E}{F_y}} = 0.56\sqrt{\frac{29,\ 000}{36}} = 15.89 > 8.0$$

Thus, **OK** [AASHTO LRFD Cl. 6.4.9.2]

Slenderness ratio about major and minor axis:

$$\lambda_z = \left(\frac{Kl}{r_z\pi}\right)^2\frac{F_y}{E} = \left(\frac{1.0 \cdot 576}{11.83\pi}\right)^2\frac{36}{29,\ 000} = 0.298$$

$$\lambda_y = \left(\frac{Kl}{r_y\pi}\right)^2\frac{F_y}{E} = \left(\frac{0.333 \cdot 576}{2.30\pi}\right)^2\frac{36}{29,\ 000} = 0.877$$

λ_y governs, thus $\lambda = 0.877$

$\lambda < 2.25$, so Equation 6.9.4.1-1 is used to determine the nominal compressive resistance

$P_n = 0.66^{\lambda}F_yA_s = 0.66^{0.877} \cdot 36 \cdot 154.4 = 3{,}861$ kips

The factored compressive resistance, $P_r = \varphi_c P_n = 0.9 \cdot 3{,}861 = 3{,}475\ kips$

Major Axis Bending Capacity

The compression flange moment of inertia:

$$I_{yc} = 1(16)^3/12 = 341.3 \text{ in}^4$$

$$I_{yc}/I_y = 341.3/814.9 = 0.419, > 0.1 \text{ and } < 0.9$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.2.1]

$$\frac{2D_o}{t_w} = \frac{2(17)}{3.6} = 9.445 < 6.77\sqrt{\frac{E}{F_y}} = 6.77\sqrt{\frac{29,\ 000}{36}} = 192.1$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.2.2]

Calculation of depth of the web in compression at the plastic moment, D_{cp}(Clause 6.10.3.3.2)

Area of Web, $A_w = (36 - 2 \times 1) \times 3.6 = 122.4 \ in^2$

Area of flange area in tension, A_{ft} = Area of flange in compression, $A_{fc} = 16 \times 1 = 16 \text{ in}^2$

$$D_{cp} = \left(\frac{D_w}{2A_w F_y}\right)Fy\left(A_{ft} + A_w - A_{fc}\right) = \left(\frac{34}{2 \cdot 122.4 \cdot 36}\right)36\left(16 + 122.4 - 16\right) = 17in$$

$$\frac{2D_{cp}}{t_w} = \frac{2(17)}{3.6} = 9.445 < 3.76\sqrt{\frac{E}{F_y}} = 3.76\sqrt{\frac{29,\ 000}{36}} = 108.1$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.4.1.2]

$$\frac{b_f}{t_f} = \frac{16.0}{2(1.0} = 8.0 < 0.382\sqrt{\frac{E}{F_y}} = 0.382\sqrt{\frac{29,\ 000}{36}} = 10.98$$

Thus, **OK** [AASHTO LRFD Cl. 6.10.4.3]

Check clause 6.10.4.1.6a

$$(B/t)_{flange} < 0.75 \times (B/t)_{flange\_limit} = 0.75 \times 10.978$$

$$(D/T)_{web} < 0.75 \times (D/T)_{web\_limit} = 0.75 \times 108.057$$

Check clause 6.10.4.1.7

$$M_{pz} = P_z \times F_y = 1{,}600 \text{ in}^3(36 \text{ ksi}) = 57{,}614 \text{ in-k}$$

$$L_b = \left[0.124 - 0.0759\left(M_l \big/ M_{pz}\right)\right] \times \left[\frac{r_y E}{F_y}\right]$$

$$= \left[0.124 - 0.0759\left(0 \big/ 57,\ 614\right)\right] \times \left[\frac{2.30(29,\ 000)}{36}\right] = 229.5 \text{ in.}$$

Unsupported length, L_u = 576 in > L_b.; section is non-compact.

Check clause 6.10.4.1.9

A notional section comprised of the compression flange and one-third of the depth of the web in compression, taken about the vertical axis.

$$A_{rt} = A_{cf} + \left(\frac{c - t_f}{3}\right)t_w = 16 + \left(\frac{18 - 1.0}{3}\right)3.6 = 36.4in^2$$

$$I_{rt} = t_f\left[\left(\frac{c - t_f}{3}\right)\frac{t_w^3}{12}\right] = 1\left[\left(\frac{18 - 1}{3}\right)\frac{3.6^3}{12}\right] = 363.4in^4$$

$$r_{rt} = \sqrt{\frac{363.4}{36.4}} = 3.149in$$

$$L_p = 1.76\left(3.159\right)\sqrt{\frac{29,\ 000}{36}} = 159.8in$$

Lb > Lp

<u>Clause 6.10.4.2.6</u>

$$L_r = 4.44\sqrt{\frac{I_{yc}d}{S_{xc}}\frac{E}{F_y}} = 4.44\sqrt{\frac{341.3(36)}{1,\ 200}\frac{29,\ 000}{36}} = 403.3in$$

Minimum radius of gyration of the compression flange taken about the vertical axis:

$$r_t = \sqrt{\frac{I_{yc}}{A_{cf}}} = \sqrt{\frac{341.3}{16}} = 4.619in$$

Hybrid factor, Rh = 1.0 (For Homogeneous sections, Hybrid factors shall be taken as 1.0 per clause 6.10.4.3.1)

As per clause 6.10.4.3.2, Load-shedding factor, Rb

If area of the compression flange, A_{cf} ≥ the area of the tension flange, A_{tf}

l_b= 5.76

Dc = 17.00

$$\frac{2D_c}{t_w} = \frac{2(17)}{3.6} = 9.445 < L_b\sqrt{\frac{E}{F_y}} = 5.76\sqrt{\frac{29,\ 000}{36}} = 163.5$$

Thus:

$R_{b,comp} = R_{b,ten} = 1.0$

Lr < Lb

C_b = 1.75 + 1.05(M1/M2)+0.3x(M1/M2)^2

Here M1 = 0, M2 = 0 so C_b = 1.75

$$M_{nz,comp} = C_{bz}R_{b,comp}R_h\frac{M_y}{2}\left(\frac{L_r}{L_b}\right)^2 = 1.75\left(1.0\right)\left(1.0\right)\left[\frac{36(1,\ 200)}{2}\right]\left(\frac{408.4}{576}\right)^2 = 19,\ 000kip\ \cdot in$$

$$M_{nz,ten} = R_{b,ten}R_hF_yZ = 1.0\left(1.0\right)\left(36\right)\left(1,\ 200\right) = 43,\ 188kip\ \cdot in$$

M_{nz} = 19,000 kip ·in

Resisting Moment

$M_r = Q_f \cdot M_n$= 1.0 (19,000) = 19,000 kip ·in

Actual Moment = 31,488 kip·in

Interaction ratio = 31,488 / 19,000 = 1.657

Comparison

Table 713: Comparison of results

Value	Hand Calculations	STAAD.Pro Results	Difference
Critical Interaction Ratio	1.657	1.654	negligible

STAAD Input

The following input is used in this verification example.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-May-11
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 38 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
AASHTOGIRDER
154.4 36 3.6 16 1 21593.8 814.86 539.4 129.6 32
END
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 AASHTOGIRDER
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
UNIT INCHES KIP
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.76
PERFORM ANALYSIS
PARAMETER 1
CODE AASHTO LRFD
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (   AASHTO - LRFD)   v1.0
                       ********************************************
*    1  ST  AASHTOGIRDER             (UPT)
                         FAIL      Slenderness       1.654        1
                        0.00          0.00          0.00       0.00
   Section Properties (in)
   -----------------------
   Ax =     154.40   Ay =     129.60   Az =      32.00
   Iz =   21593.80   Iy =     814.86
   Rz =      11.83   Ry =       2.30
   Sz =    1199.66   Sy =     101.86
   Input Parameters (Kip-in)
   --------------------------
   Fyld  =      36.00   Fu    =      58.00
   Lz    =     456.00   Ly    =     456.00
   Kz    =       1.00   Ky    =       1.00
   UNL   =       0.00   Ratio =       1.00
   Design Results (Kip-in)
   -----------------------
   Klrz =      38.56   Klry =     198.49
   CBy  =       0.00   CBz  =       0.00
   CAPACITIES (Kip-in)
   --------------------
   Compressive Capacity =       0.00
   Tensile Capacity     =       0.00
   Moment Capacity_y    =       0.00
   Moment Capacity_z    =       0.00
   Shear Capacity       =       0.00
   DESIGN FORCES (Kip-in)
   -----------------------
   Compressive Force    =       0.00
   Tensile Force        =       0.00
   Moment Y             =       0.00
   Moment Z             =       0.00
   Shear Force          =     173.28
```

Related Links

- *D1.D.2 AASHTO (LRFD)* (on page 1180)

V. AISC

V.AISC 360-16 Angle F.11A

Verify the available flexural strength of a single angle member per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017

2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.F-48 – F-51., Example F.11A

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a single L4x4x1/4. The member is used as a 6 ft simple span subject to a uniform dead load of 0.05 kip/ft and a uniform live load of 0.15 kip/ft. The vertical leg of the single angle is up with the toe in compression. The material is ASTM A36.

Validation

Note: When assigning the angle, it should be given in a beta angle as the geometrical axis and principal axis are different, and in STAAD.Pro the calculation is done with reference with the principal axis. To match the geometric axis used in AISC 360-16, a BETA angle is assigned to the member in STAAD.Pro.

The nominal flexural strength, M_n , shall be the lowest value obtained to the limit states of yielding (plastic moment), lateral-torsional buckling, and leg local buckling.

Ultimate load (for LRFD):

$$w_{ux} = 1.2(0.05) + 1.6(0.15) = 0.30 \text{ kip/ft}$$

$$M_{ux} = w_{ux}L^2/8 = (0.30)(6)^2/8 = 1.35 \text{ kip·ft}$$

Service level load (for ASD):

$$w_{ax} = 0.05 + 0.15 = 0.20 \text{ kip/ft}$$

$$M_{ax} = w_{ax}L^2/8 = (0.20)(6)^2/8 = 0.90 \text{ kip·ft}$$

Note: Only bending about the major axis is compared as bending due to loads is about the geometrix X-X axis.

Yielding

The nominal flexural strength in the limit state of flexural yielding is:

$$M_n = 1.5M_y = 1.5F_yS_x \qquad \text{Eq. F10-1}$$

$$M_n = 1.5(36 \text{ ksi})(1.03 \text{ in}^3) = 55.6 \text{ in·kip}$$

Lateral Torsional Buckling

For single angles bending about a geometric axis with no lateral-torsional restraint:

$$M_y = 0.8.F_yS_x \qquad \text{F10-2}$$

$$M_y = 0.8(36 \text{ ksi})(1.03 \text{ in}^3) = 29.7 \text{ in·kip}$$

For bending about one of the geometric axes of an equal-leg angle without axial compression, no lateral-torsional restraint, and with maximum compression in the toe:

$$M_{cr} = \frac{0.58Eb^4tC_b}{L_b^2}\left[\sqrt{1 + 0.88\left(\frac{L_bt}{b^2}\right)^2} - 1\right] \qquad \text{Eq. F10-5a}$$

where

C_b = *1.14* from AISC Table 3-1.Manual

$$M_{cr} = \frac{0.58(29{,}000ksi)(4.0\ in)^4(0.25\ in)(1.14)}{(6\ ft\ \times\ 12\ in/ft)^2}\left\{\sqrt{1+0.88\left[\frac{(6\ ft\ \times\ 12\ in/ft)(0.25\ in)}{(4.0\ in)^2}\right]^2}-1\right\} = 107\ in\cdot kips$$

$M_y\ /\ M_{cr} = 29.7\ /\ 107 = 0.278 < 1.0$; therefore, AISC eq. F 10-2 is applicable:

$$M_n = \left(1.92 - 1.17\sqrt{\frac{M_y}{M_{cr}}}\right)M_y \le 1.5M_y \qquad \text{Eq. F10-2}$$

$$M_n = \left(1.92 - 1.17\sqrt{\frac{29.7}{107}}\right)29.7 = 38.7\ in\cdot kip < 1.5(29.7)\ =\ 44.6\ in\cdot kip$$

Leg Loca Buckling

AISC section F10.3 applies when the toe of the leg is in compression. Check the slenderness of the leg in compression:

$$\lambda = b/t = 4.0 / 0.25 = 16.0$$

Determine the compact and noncompact slenderness ratios from Table B4.1b, Case 12:

$$\lambda_p = 0.54\sqrt{\frac{E}{F_y}} = 0.54\sqrt{\frac{29{,}000}{36}} = 15.3$$

$$\lambda_r = 0.91\sqrt{\frac{E}{F_y}} = 0.91\sqrt{\frac{29{,}000}{36}} = 25.8$$

$\lambda_p < \lambda < \lambda_r$, therefore the leg is noncompact in flexure.

$$S_c = 0.80S_x = 0.80\ (1.03\ in^3) = 0.824\ in^3$$

$$M_n = F_yS_c\left[2.43 - 1.72\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{E}}\right] \qquad \text{Eq. F10-6}$$

$$M_n = (36)(0.824)\left[2.43 - 1.72(16)\sqrt{\frac{36}{29{,}000}}\right] = 43.3\ in\cdot kip$$

Available Flexural Strength

The lateral-torsional buckling limit state governs.

$$M_n = 38.7\ in\cdot kip = 3.23\ ft\cdot kip$$

LRFD

$$\phi_b = 0.90$$

$$\phi_bM_n = 0.90(3.23) = 2.91\ ft\text{-}kip > 1.35\ kip\cdot ft$$

ASD

$$\Omega_b = 1.67$$

$$M_n/\Omega_b = 3.23/1.67 = 1.93\ ft\text{-}kip > 0.90\ kip\cdot ft$$

Results

Table 714: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Lateral-torsional buckling, about X axis (ft·kips) LRFD	2.91	2.940	1.0%	
Lateral-torsional buckling, about X axis (ft·kips) ASD	1.93	1.956	1.3%	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Angle F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40404
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GZ 0.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GZ 0.2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
```

```
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
CB 1.14 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
CB 1.14 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       U                Z          Axis typically parallel to the sections
principal major axis.
       V                Y          Axis typically parallel to the sections
principal minor axis.
       Z                X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz               FX         Axial force.
       Vy               FY         Shear force along minor axis.
       Vx               FZ         Shear force along major axis.
       Tz               MX         Torsional moment.
       My               MY         Bending moment about minor axis.
       Mx               MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
```

```
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  L40404             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.447       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.F10.2
|
|  Pz:       0.000     T     Vy:       -1.315           Vx:      1.315
|
|  Tz:       0.000           My:       -1.315           Mx:     -1.315
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     91.954
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.447(PASS)
|
|          Loc  :    0.00            Condition :    Cl.F10.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
```

```
 |
 | Ag  :   1.930E+00     Axx :   1.000E+00     Ayy :   1.000E+00
 |
 | Ixx :   3.039E+00     Iyy :   3.039E+00     J   :   4.021E-02
 |
 | Sxx+:   7.627E-01     Sxx-:   9.266E-01     Zxx :   1.373E+00
 |
 | Syy+:   1.731E+00     Syy-:   1.731E+00     Zyy :   3.115E+00
 |
 | Cw  :   5.051E-02     x0  :   9.720E-01     y0  :   9.720E-01
 |
 |-----------------------------------------------------------------------------
 |
 | MATERIAL PROPERTIES
 |
 | Fyld:        5184.000              Fu:        8351.999
 |
 |-----------------------------------------------------------------------------
 |
 | Actual Member Length:         6.000
 |
 | Design Parameters                                          (Rolled)
 |
 | Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      3 LOC:    72.00)
 |
 |                          λ         λp        λr       CASE
 |
 | Flange: Slender         16.00       N/A      12.77     Table.4.1a.Case3
 |
 | Web   : Slender         16.00       N/A      12.77     Table.4.1a.Case3
 |
 |
 |
 | FLEXURE CLASSIFICATION     (L/C:      3 LOC:    72.00)
 |
 |                          λ         λp        λr       CASE
 |
 | Flange: NonCompact      16.00      15.33     25.83     Table.4.1b.Case12
 |
 | Web   : NonCompact      16.00      15.33     25.83     Table.4.1b.Case12
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR AXIAL TENSION
 |
 |-----------------------------------------------------------------------------
```

```
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       62.53       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  69.480     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       83.95       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.13403E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  111.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       48.51       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  57.375                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12520.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4359.2     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  53.898     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|            0.000       48.51       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  57.375                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  12520.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4359.2     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  53.898     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING U
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       50.73       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcu/ru =  45.208                Cl.E2
|
|  Elastic Buckling Stress   : Feu    =  20167.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcru   =  4655.2     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnu    =  56.369     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING V
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       40.01       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcv/rv =  91.954                Cl.E2
|
|  Elastic Buckling Stress   : Fev    =  4874.4     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrv   =  3321.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnv    =  44.453     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
```

```
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              1.315       19.44       0.068     Cl.G1          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G3
 |
 |  Nom. Shear Along X        : Vnx    =  21.600     kip        Eq.G6-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              1.315       19.44       0.068     Cl.G1          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along Y    : Kv     =  1.2000                Cl.G3
 |
 |  Nom. Shear Along Y        : Vny    =  21.600     kip        Eq.G3-1
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                                -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
```

```
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -1.315       4.240       0.310     Cl.F10.1       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  4.7114     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             1.315       4.240       0.310     Cl.F10.1       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  4.7114     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (U)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -1.860       7.010       0.265     Cl.F10.1       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along U : Mnu    =  7.7886     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (V)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -.3701E-16   3.089       0.000     Cl.F10.1       3      2.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along V : Mnv    =  3.4322     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
```

```
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|           -1.315       2.940       0.447     Cl.F10.2       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap              : Mnx    =  3.2670     kip-ft     Eq.F10-2
|
|  Mom. Distr. factor         : CbX    =  1.1400                Custom
|
|  Crit. Elastic L-T-B Mom.   : McrX   =  8.9536     kip-ft     Eq.F10-5a
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT Y
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            1.315       3.392       0.388     Cl.F10.2       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About Y : Mny    =  3.7691     kip-ft     Eq.F10-2
|
|  Mod. factor Cb             : CbY    =  1.1400                Custom
|
|  Crit. Elas. L-T-B Moment   : McrY   =  48.408     kip-ft     Eq.F10-5b
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT U
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|           -1.860       5.883       0.316     Cl.F10.2       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About U : Mnu    =  6.5367     kip-ft     Eq.F10-2
|
|  modification factor Cb     : CbU    =  1.1400                Custom
|
|  Crit. Elast. L-T-B Moment : McrU   =  16.263     kip-ft     Eq.F10-4
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|           -1.315       3.303       0.398     Cl.F10.3       3      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  3.6695     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -.6576       3.303       0.199     Cl.F10.3       3      3.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  3.6695     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(U)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -1.860       6.825       0.273     Cl.F10.3       3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnu    =  7.5829     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(V)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -.3701E-16   3.653       0.000     Cl.F10.3       3      2.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnv    =  4.0594     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
11
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
 |-----------------------------------------------------------------------------
```

```
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H2
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.316      Eq.H2-1            3        0.00
|
|
|
| Intermediate Results :
|
|  Applied Stress           : frbz   = -1857.0     kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fca    =  4665.6     kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fcbz   =  5873.5     kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fcbw   =  5760.9     kip/ft2    Eq.H2-1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2016
    52. METHOD ASD
    53. CB 1.14 ALL
    54. SGR 1 ALL
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       U               Z          Axis typically parallel to the sections
principal major axis.
       V               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
```

```
 ------------    -----------     -------------
      Pz              FX         Axial force.
      Vy              FY         Shear force along minor axis.
      Vx              FZ         Shear force along major axis.
      Tz              MX         Torsional moment.
      My              MY         Bending moment about minor axis.
      Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
13
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |------------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  L40404            (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.488       Loadcase:       4
 |
 |  Location:      0.00       Ref:      Cl.F10.2
 |
 |  Pz:       0.000     T     Vy:       -.9546           Vx:     0.9546
 |
 |  Tz:       0.000           My:       -.9546           Mx:     -.9546
 |
 |------------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     91.954
 |
 | Allowable Slenderness Ratio :    300.000           LOC  :     0.00
```

```
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.488(PASS)
|
|          Loc  :    0.00            Condition :    Cl.F10.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.930E+00     Axx :   1.000E+00     Ayy :   1.000E+00
|
| Ixx :   3.039E+00     Iyy :   3.039E+00     J   :   4.021E-02
|
| Sxx+:   7.627E-01     Sxx-:   9.266E-01     Zxx :   1.373E+00
|
| Syy+:   1.731E+00     Syy-:   1.731E+00     Zyy :   3.115E+00
|
| Cw  :   5.051E-02     x0  :   9.720E-01     y0  :   9.720E-01
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        5184.000              Fu:        8351.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:         6.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:    72.00)
|
|                            λ          λp         λr       CASE
|
| Flange: Slender         16.00       N/A      12.77     Table.4.1a.Case3
|
| Web   : Slender         16.00       N/A      12.77     Table.4.1a.Case3
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:    72.00)
|
|                            λ          λp         λr       CASE
|
| Flange: NonCompact      16.00      15.33      25.83     Table.4.1b.Case12
|
| Web   : NonCompact      16.00      15.33      25.83     Table.4.1b.Case12
|
|-----------------------------------------------------------------------------
```

```
|
     STAAD SPACE                                              -- PAGE NO.
14
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       41.60       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  69.480     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       55.97       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.13403E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  111.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       32.27       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  57.375                Cl.E2
```

```
 Elastic Buckling Stress   : Fex    =  12520.     kip/ft2    Eq.E3-4
 Crit. Buckling Stress     : Fcrx   =  4359.2     kip/ft2    Eq.E3-2
 Nom. Flexural Buckling    : Pnx    =  53.898     kip        Eq.E7-1
-----------------------------------------------------------------------------
FLEXURAL BUCKLING Y
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       32.27       0.000     Cl.E3          4      0.00


Intermediate Results :
 Effective Slenderness     : Lcy/ry =  57.375                Cl.E2
 Elastic Buckling Stress   : Fey    =  12520.     kip/ft2    Eq.E3-4
 Crit. Buckling Stress     : Fcry   =  4359.2     kip/ft2    Eq.E3-2
 Nom. Flexural Buckling    : Pny    =  53.898     kip        Eq.E7-1
-----------------------------------------------------------------------------
FLEXURAL BUCKLING U
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       33.75       0.000     Cl.E3          4      0.00


Intermediate Results :
 Effective Slenderness     : Lcu/ru =  45.208                Cl.E2
 Elastic Buckling Stress   : Feu    =  20167.     kip/ft2    Eq.E3-4
 Crit. Buckling Stress     : Fcru   =  4655.2     kip/ft2    Eq.E3-2
 Nom. Flexural Buckling    : Pnu    =  56.369     kip        Eq.E7-1
-----------------------------------------------------------------------------
FLEXURAL BUCKLING V
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       26.62       0.000     Cl.E3          4      0.00


Intermediate Results :
```

```
|
|  Effective Slenderness     : Lcv/rv =  91.954                Cl.E2
|
|  Elastic Buckling Stress   : Fev    =  4874.4     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrv   =  3321.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnv    =  44.453     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
15
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                               - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.9546       12.93       0.074     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G3
|
|  Nom. Shear Along X        : Vnx    =  21.600     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.9546       12.93       0.074     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  1.2000                Cl.G3
|
|  Nom. Shear Along Y        : Vny    =  21.600     kip        Eq.G3-1
|
|-----------------------------------------------------------------------------
```

```
|
     STAAD SPACE                                              -- PAGE NO.
16
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -.9546       2.821       0.338     Cl.F10.1       4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  4.7114     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.9546       2.821       0.338     Cl.F10.1       4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  4.7114     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (U)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -1.350       4.664       0.289     Cl.F10.1       4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along U : Mnu    =  7.7886     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (V)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|            -.1480E-15   2.055       0.000     Cl.F10.1       4      0.00
|
|
| Intermediate Results :
|  Nom Flex Yielding Along V : Mnv    =  3.4322     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
| LAT TOR BUCK ABOUT X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|            -.9546       1.956       0.488     Cl.F10.2       4      0.00
|
|
| Intermediate Results :
|  Nom L-T-B Cap             : Mnx    =  3.2670     kip-ft     Eq.F10-2
|  Mom. Distr. factor        : CbX    =  1.1400                Custom
|  Crit. Elastic L-T-B Mom.  : McrX   =  8.9536     kip-ft     Eq.F10-5a
|
|-----------------------------------------------------------------------------
| LAT TOR BUCK ABOUT Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|            0.9546       2.257       0.423     Cl.F10.2       4      0.00
|
|
| Intermediate Results :
|  Nom Flex Yielding About Y : Mny    =  3.7691     kip-ft     Eq.F10-2
|  Mod. factor Cb            : CbY    =  1.1400                Custom
|  Crit. Elas. L-T-B Moment  : McrY   =  48.408     kip-ft     Eq.F10-5b
|
|-----------------------------------------------------------------------------
| LAT TOR BUCK ABOUT U
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|            -1.350       3.914       0.345     Cl.F10.2       4      0.00
|
|
| Intermediate Results :
|  Nom Flex Yielding About U : Mnu    =  6.5367     kip-ft     Eq.F10-2
```

```
|  modification factor Cb    : CbU    =  1.1400              Custom
|
|  Crit. Elast. L-T-B Moment : McrU   =  16.263     kip-ft     Eq.F10-4
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -.9546       2.197       0.434     Cl.F10.3       4      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  3.6695     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -.4773       2.197       0.217     Cl.F10.3       4      3.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  3.6695     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(U)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -1.350       4.541       0.297     Cl.F10.3       4      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnu    =  7.5829     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(V)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -.1480E-15   2.431       0.000     Cl.F10.3       4      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Nom F-L-B Cap            : Mnv    =  4.0594     kip-ft     Eq.F10-6
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
17
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H2
|
|                              RATIO      CRITERIA          L/C      LOC
|
|                              0.345      Eq.H2-1            4        0.00
|
|
|
| Intermediate Results :
|
|  Applied Stress           : frbz   = -1347.8     kip/ft2    Eq.H2-1
|
|  Applied Stress           : frbw   = -.27607E-12 kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fca    =  3104.2     kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fcbz   =  3907.9     kip/ft2    Eq.H2-1
|
|  Allowable Stress         : Fcbw   =  3832.9     kip/ft2    Eq.H2-1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 C Flex Mem F.2-1A

Verify the flexural yielding capacity of a continuously braced channel member subject to major axis bending per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.F-16 – F-17., Example F. 2-1A

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a C15x33.9. The member is used as a 25 ft simple span subject to a uniform dead load of 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The member is continuously braced. The material is ASTM A36.

Validation

Ultimate load (for LRFD):

$$w_u = 1.2(0.23) + 1.6(0.6) = 1.38 \text{ kip/ft}$$

$$M_u = w_u L^2/8 = (1.38)(25)^2/8 = 108 \text{ kip·ft}$$

Service level load (for ASD):

$$w_a = 0.23 + 0.39 = 0.92 \text{ kip/ft}$$

$$M_a = w_a L^2/8 = (0.92)(25)^2/8 = 71.9 \text{ kip·ft}$$

Note: Per the User Note in AISC *Specification* Section F2, all ASTM A36 channel shapes are compact.

Yielding Limit State

$$M_n = M_p = F_y Z_x \qquad \text{Eq. F2-1}$$

$$M_n = (36 \text{ ksi})(50.8 \text{ in}^3) = 1{,}830 \text{ in·kip} = 152.5 \text{ ft·kip}$$

Available Flexural Strength

A continuously braced channel section is governed by the yielding limit state.

LRFD

$$\phi_b = 0.90$$

$$\phi_b M_n = 0.90(152.5) = 137 \text{ ft-kip} > 108 \text{ kip·ft}$$

ASD

$$\Omega_b = 1.67$$

$$M_n/\Omega_b = 152.5/1.67 = 91.3 \text{ ft-kip} > 71.9 \text{ kip·ft}$$

Results

Table 715: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Flexural yielding capacity, about X axis (ft·kips) LRFD	137	137.2	negligible	
Flexural yielding capacity, about X axis (ft·kips) ASD	91.3	91.26	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 C Flex Mem F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD LRFD
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD ASD
```

```
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ========================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
```

```
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C15X33             (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         1.113       Loadcase:        3
|
|  Location:      0.00       Ref:      Slenderness (T)
|
|  Pz:       0.000     T     Vy:        17.25           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -.2084E-13
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    333.952
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        1.113(FAIL)
|
|          Loc  :    0.00            Condition :    Cl.D1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.000E+01     Axx :   4.420E+00     Ayy :   6.000E+00
|
| Ixx :   3.150E+02     Iyy :   8.070E+00     J   :   1.010E+00
|
| Sxx+:   4.200E+01     Sxx-:   4.200E+01     Zxx :   5.080E+01
|
| Syy+:   3.178E+00     Syy-:   9.373E+00     Zyy :   6.190E+00
|
| Cw  :   3.555E+02     x0  :  -1.762E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:       5184.000              Fu:        8351.999
```

```
|
|--------------------------------------------------------------------------
|
| Actual Member Length:        25.000
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|--------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   300.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       5.23       N/A      15.89     Table.4.1a.Case1
|
| Web   : NonSlender      34.25       N/A      42.29     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   300.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          5.23      10.79     28.38     Table.4.1b.Case10
|
| Web   : Compact         34.25     106.72    161.78     Table.4.1b.Case15
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|--------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|--------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       324.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  360.00     kip        Eq.D2-1
|
|--------------------------------------------------------------------------
|
| TENSILE RUPTURE
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       435.0       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     = 0.69444E-01 ft2       Eq.D3-1
|  Nom. Ten. Rpt Cap         : Pn     =  580.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       278.8       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  53.452                Cl.E2
|  Elastic Buckling Stress   : Fex    =  14425.     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  4460.1     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  309.73     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       20.26       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  333.95                Cl.E2
|  Elastic Buckling Stress   : Fey    =  369.57     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  324.11     kip/ft2    Eq.E3-3
|  Nom. Flexural Buckling    : Pny    =  22.508     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       207.0       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Constant                 : H      = 0.91231               Eq.E4-8
|
|  Euler Buckling Stress    : Fez    =  5059.6     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress     : Fe     =  4844.8     kip/ft2    Eq.E4-3
|
|  Crit. F-T-B Stress       : Fcr    =  3312.6     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling   : Pn     =  230.04     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       85.92       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X   : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X   : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X       : Vnx    =  95.472     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             17.25       116.6       0.148     Cl.G1          3      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  129.60     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -107.8       137.2       0.786     Cl.F2.1        3     12.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  152.40     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       13.73       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  15.256     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                                - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.786      Eq.H1-1b           3       12.50
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  324.00     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  137.16     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  13.731     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2016
    51. FLX 2 ALL
    52. METHOD ASD
    53. SGR 1 ALL
    54. TRACK 2 ALL
    55. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                 -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
```

```
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.    
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |---------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C15X33             (AISC 
SECTIONS)|
|  Status:        FAIL       Ratio:         1.113       Loadcase:       4    
|
|  Location:      0.00       Ref:      Slenderness (T)                       
|
|  Pz:       0.000     T     Vy:        11.50           Vx:      0.000       
|
|  Tz:       0.000           My:        0.000           Mx:     -.1421E-13   
|
|---------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    333.952
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
```

```
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        1.113(FAIL)
|
|          Loc  :    0.00            Condition :    Cl.D1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.000E+01     Axx :   4.420E+00     Ayy :   6.000E+00
|
| Ixx :   3.150E+02     Iyy :   8.070E+00     J   :   1.010E+00
|
| Sxx+:   4.200E+01     Sxx-:   4.200E+01     Zxx :   5.080E+01
|
| Syy+:   3.178E+00     Syy-:   9.373E+00     Zyy :   6.190E+00
|
| Cw  :   3.555E+02     x0  :  -1.762E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        5184.000             Fu:        8351.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        25.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   300.00)
|
|                            λ          λp         λr        CASE
|
| Flange: NonSlender       5.23        N/A      15.89     Table.4.1a.Case1
|
| Web   : NonSlender      34.25        N/A      42.29     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   300.00)
|
|                            λ          λp         λr        CASE
|
| Flange: Compact          5.23      10.79      28.38     Table.4.1b.Case10
|
| Web   : Compact         34.25     106.72     161.78     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
```

```
|
     STAAD SPACE                                              -- PAGE NO.
14
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       215.6       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  360.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       290.0       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.69444E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  580.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       185.5       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  53.452                Cl.E2
```

```
|
|  Elastic Buckling Stress   : Fex    =  14425.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4460.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  309.73     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       13.48       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  333.95                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  369.57     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  324.11     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  22.508     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       137.7       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Constant                 : H      = 0.91231                Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  5059.6     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  4844.8     kip/ft2    Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  3312.6     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  230.04     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
15
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
```

```
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       57.17       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000              Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  95.472     kip      Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             11.50       77.60       0.148     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  129.60     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                            -- PAGE NO.
16
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -71.88       91.26       0.788     Cl.F2.1        4     12.50
```

```
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  152.40     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       9.136       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  15.256     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
17
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.788      Eq.H1-1b           4       12.50
|
|
|
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  215.57     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  91.257     kip-ft     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  9.1356     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 C LTB Test F.2B

Verify the flexural capacity of a channel member with bracing at ends and fifth points per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.F-20 – F-21., Example F.2B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a C15x33.9. The member is used as a 25 ft simple span subject to a uniform dead load of 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The member is braced at the supports and at fifth points. The material is ASTM A36.

Validation

Ultimate load (for LRFD):

$$w_u = 1.2(0.23) + 1.6(0.6) = 1.38 \text{ kip/ft}$$

$$M_u = w_u L^2/8 = (1.38)(25)^2/8 = 108 \text{ kip·ft}$$

Service level load (for ASD):

$$w_a = 0.23 + 0.39 = 0.92 \text{ kip/ft}$$

$$M_a = w_a L^2/8 = (0.92)(25)^2/8 = 71.9 \text{ kip·ft}$$

Note: Per the User Note in AISC *Specification* Section F2, all ASTM A36 channel shapes are compact.

From the AISC *Manual* Table 3-8, for a C15x33.9:

$$L_p = 3.75 \text{ ft}$$

$$L_r = 14.5 \text{ ft}$$

The unbraced length of the member:

$$L = 25.0/5 = 5 \text{ ft}$$

For a compact channel with $L_p < L_b \leq L_r$, the lesser of the flexural yielding or inelastic lateral-torsional buckling limit states will govern the available flexural strength.

<u>Lateral-Torsional Buckling Limit State</u>

$$M_n = M_p = F_y Z_x \qquad \text{Eq. F2-1}$$

$$M_n = (36 \text{ ksi})(50.8 \text{ in}^3) = 1{,}830 \text{ in·kip} = 152 \text{ ft·kip}$$

The nominal flexural strength in the lateral-torsional buckling limit state is:

$$M_n = C_b\left[M_p - \left(M_p - 0.75F_y S_x\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \leq M_p \qquad \text{Eq. F2-2}$$

where

C_b = *1.0* for the center segment of a uniformly loaded beam braced at ends and fifth points.

$$M_n = 1.0\left\{1{,}830 - \left[1{,}830 - 0.75(36)(42.0)\right]\left(\frac{5.0 - 3.75}{14.5 - 3.75}\right)\right\} = 1{,}740\ in \cdot kip$$

M_n = 145 ft·kip

<u>Available Flexural Strength</u>

By observation, the section is governed by the lateral-torsional buckling limit state.

LRFD

$\phi_b = 0.90$

$\phi_b M_n = 0.90(145) = 131$ ft-kip > 108 kip·ft

ASD

$\Omega_b = 1.67$

$M_n/\Omega_b = 145/1.67 = 86.8$ ft-kip > 71.9 kip·ft

Results

Table 716: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Lateral-torsional buckling, about X axis (ft·kips) LRFD	131	130.4	negligible	
Lateral-torsional buckling, about X axis (ft·kips) ASD	86.8	86.75	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 C LTB Test F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
```

```
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
CB 1 ALL
FLX 1 ALL
METHOD LRFD
SGR 1 ALL
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 1
CODE AISC UNIFIED 2016
CB 1 ALL
FLX 1 ALL
METHOD ASD
SGR 1 ALL
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
```

```
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C15X33             (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         1.113      Loadcase:       3
```

```
 Location:      0.00       Ref:      Slenderness (T)

 Pz:       0.000     T     Vy:        17.25           Vx:      0.000

 Tz:       0.000           My:        0.000           Mx:     -.2084E-13

-----------------------------------------------------------------------------

TENSION SLENDERNESS

Actual Slenderness Ratio    :    333.952

Allowable Slenderness Ratio :    300.000            LOC  :     0.00

-----------------------------------------------------------------------------

STRENGTH CHECKS

Critical L/C  :      3             Ratio     :        1.113(FAIL)

         Loc  :    0.00            Condition :    Cl.D1

-----------------------------------------------------------------------------

SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )

Ag  :   1.000E+01     Axx :   4.420E+00     Ayy :   6.000E+00

Ixx :   3.150E+02     Iyy :   8.070E+00     J   :   1.010E+00

Sxx+:   4.200E+01     Sxx-:   4.200E+01     Zxx :   5.080E+01

Syy+:   3.178E+00     Syy-:   9.373E+00     Zyy :   6.190E+00

Cw  :   3.555E+02     x0  :  -1.762E+00     y0  :   0.000E+00

-----------------------------------------------------------------------------

MATERIAL PROPERTIES

Fyld:        5184.000              Fu:        8351.999

-----------------------------------------------------------------------------

Actual Member Length:        25.000

Design Parameters                                                (Rolled)

Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00

-----------------------------------------------------------------------------

COMPRESSION CLASSIFICATION (L/C:      3 LOC:   300.00)

                           λ         λp        λr       CASE

Flange: NonSlender       5.23       N/A      15.89     Table.4.1a.Case1
```

```
|
| Web   : NonSlender      34.25       N/A      42.29     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   300.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          5.23      10.79     28.38     Table.4.1b.Case10
|
| Web   : Compact         34.25     106.72    161.78     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       324.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  360.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       435.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.69444E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  580.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
```

```
|-------------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       278.8       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  53.452               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  14425.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4460.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  309.73     kip        Eq.E3-1
|
|-------------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       20.26       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  333.95               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  369.57     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  324.11     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  22.508     kip        Eq.E3-1
|
|-------------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       207.0       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Constant                 : H      = 0.91231                Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  5059.6     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  4844.8     kip/ft2    Eq.E4-3
```

```
|
|  Crit. F-T-B Stress        : Fcr    =  3312.6     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  230.04     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       85.92       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  95.472     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             17.25       116.6       0.148     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  129.60     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
```

```
                     ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -107.8       137.2       0.786     Cl.F2.1        3     12.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  152.40     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       13.73       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  15.256     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -107.8       130.4       0.827     Cl.F2.2        3     12.50
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  144.87     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  3.7395     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0811                Eq.F2-8b
|
|  Effective Rad. of Gyr.    : Rts    = 0.94105E-01 ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  14.489     ft         Eq.F2-6
```

```
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.827      Eq.H1-1b           3       12.50
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  324.00     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  130.38     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  13.731     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 1
    51. CODE AISC UNIFIED 2016
    52. CB 1 ALL
    53. FLX 1 ALL
    54. METHOD ASD
    55. SGR 1 ALL
    56. TRACK 2 ALL
    57. UNB 5 ALL
    58. UNT 5 ALL
    59. CHECK CODE ALL
 **WARNING-DUPLICATE PARAMETER BLOCK NUMBER.
 PLEASE PROVIDE VALID PARAMETER BLOCK NUMBER.
 PARAMETER 1
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ======================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
```

```
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C15X33              (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         1.113      Loadcase:       4
|
```

```
 Location:      0.00       Ref:      Slenderness (T)

 Pz:       0.000     T     Vy:        11.50          Vx:      0.000

 Tz:       0.000           My:        0.000          Mx:     -.1421E-13

|----------------------------------------------------------------------------

 TENSION SLENDERNESS

 Actual Slenderness Ratio    :    333.952

 Allowable Slenderness Ratio :    300.000           LOC  :     0.00

|----------------------------------------------------------------------------

 STRENGTH CHECKS

 Critical L/C  :      4             Ratio     :        1.113(FAIL)

          Loc  :    0.00            Condition :    Cl.D1

|----------------------------------------------------------------------------

 SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )

 Ag  :   1.000E+01     Axx :   4.420E+00     Ayy :   6.000E+00

 Ixx :   3.150E+02     Iyy :   8.070E+00     J   :   1.010E+00

 Sxx+:   4.200E+01     Sxx-:   4.200E+01     Zxx :   5.080E+01

 Syy+:   3.178E+00     Syy-:   9.373E+00     Zyy :   6.190E+00

 Cw  :   3.555E+02     x0  :  -1.762E+00     y0  :   0.000E+00

|----------------------------------------------------------------------------

 MATERIAL PROPERTIES

 Fyld:        5184.000              Fu:        8351.999

|----------------------------------------------------------------------------

 Actual Member Length:        25.000

 Design Parameters                                             (Rolled)

 Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00

|----------------------------------------------------------------------------

 COMPRESSION CLASSIFICATION (L/C:      4 LOC:   300.00)

                           λ         λp        λr       CASE

 Flange: NonSlender       5.23       N/A      15.89     Table.4.1a.Case1
```

```
| Web   : NonSlender      34.25       N/A      42.29     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   300.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          5.23      10.79     28.38     Table.4.1b.Case10
|
| Web   : Compact         34.25     106.72    161.78     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
11
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       215.6       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  360.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       290.0       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.69444E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  580.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
```

```
|---------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       185.5       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  53.452               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  14425.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4460.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  309.73     kip        Eq.E3-1
|
|---------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       13.48       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  333.95               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  369.57     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  324.11     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  22.508     kip        Eq.E3-1
|
|---------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       137.7       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Constant                 : H      = 0.91231               Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  5059.6     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  4844.8     kip/ft2    Eq.E4-3
|
```

```
|  Crit. F-T-B Stress        : Fcr    =  3312.6     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  230.04     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       57.17       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  95.472     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              11.50       77.60       0.148     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  129.60     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
```

```
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -71.88       91.26       0.788     Cl.F2.1        4     12.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  152.40     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       9.136       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  15.256     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -71.88       86.75       0.829     Cl.F2.2        4     12.50
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  144.87     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  3.7395     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0811                Eq.F2-8b
|
|  Effective Rad. of Gyr.    : Rts    = 0.94105E-01 ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  14.489     ft         Eq.F2-6
|
```

```
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                                    -- PAGE NO.
14
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                             - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|--------------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                              RATIO      CRITERIA          L/C      LOC
|
|                              0.829      Eq.H1-1b           4       12.50
|
|
|
| Intermediate Results :
|
|  Axial Capacity             : Pc     =  215.57     kip        Cl.H1.1
|
|  Moment Capacity            : Mcx    =  86.750     kip-ft     Cl.H1.1
|
|  Moment Capacity            : Mcy    =  9.1356     kip-ft     Cl.H1.1
|
|--------------------------------------------------------------------------------
|
```

V.AISC 360-16 HSST Compact Flange F.6

Verify the flexural strength of a square HSS flexural member with compact flanges per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.F-30 – F-31. Example F.6

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a HSST 3.5x3.5x1/8. The member is used as a 7.5 ft simple span subject to a uniform dead load of 0.145 kip/ft and a uniform live load of 0.435 kip/ft. The member is continuously braced. The material is ASTM A500 Grade C ($F_y = 50\ ksi$).

Validation

Ultimate load (for LRFD):

$w_u = 1.2(0.145) + 1.6(0.435) = 0.87$ kip/ft

$M_u = w_u L^2/8 = (0.87)(7.5)^2/8 = 6.12$ kip·ft

Service level load (for ASD):

$w_a = 0.145 + 0.435 = 0.58$ kip/ft

$M_a = w_a L^2/8 = (0.58)(7.5)^2/8 = 4.08$ kip·ft

From the AISC Manual Table 3-13:

LRFD

$\phi_b M_n = 7.21$ ft·k

ASD

$M_n/\Omega_b = 4.79$ ft·k

However, this does *not* account for using the design wall thickness for calculating the slenderness ratio for the flange local buckling as per section B4.2.

$b\backslash t = [3.5 - 3\times(0.93\times0.125)] / (0.93\times0.125) = 27.1$

$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{50}} = 26.97 < 27.1$$

Note: STAAD.Pro uses the design wall thickness for this calculation and thus considers the section as non-compact.

$M_p = Z\times F_y = 1.93\times50 = 96.5$ in·k = 8.04 ft·k

For non-compact sections:

$$M_n = M_p - \left(M_p - F_y S\right)\left(3.57\frac{b}{t_f}\sqrt{\frac{F_y}{E}} - 4.0\right) \leq M_p \quad \text{[Eqn. F7-2]}$$

$$M_n = 96.5 - \left(96.5 - 50 \times 1.66\right)\left(3.57 \times 27.1\sqrt{\frac{50}{29,\ 000}} - 4.0\right) = 96.27 \text{ in}\cdot\text{k} = 8.02 \text{ ft}\cdot\text{k}$$

LRFD

$\phi_b M_n = 7.22$ ft·k

ASD

$M_n/\Omega_b = 4.80$ ft·k

Results

Table 717: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Flange slenderness	compact	non-compact	n/a	The reference does not consider the design wall thickness, which increases the slenderness ratio just above the plastic limit.
$\phi_b M_n$ (ft·k)	7.22	7.209	negligible	
M_n/Ω_b (ft·k)	4.8	4.796	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 HSST Compact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST3.5X3.5X0.125
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.145
```

```
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.435
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD LRFD
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD ASD
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
```

```
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  HSST3.5X3.5X0.125   (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.849       Loadcase:        3
 |
 |  Location:      3.75       Ref:      Cl.F7.2
 |
 |  Pz:       0.000     T     Vy:       -.1907E-06       Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:     -6.117
 |
 |-----------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     65.585
 |
 | Allowable Slenderness Ratio :    300.000           LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      3             Ratio     :        0.849(PASS)
 |
```

```
|          Loc  :    3.75           Condition :    Cl.F7.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )
|
| Ag  :   1.540E+00     Axx :   7.313E-01     Ayy :   7.313E-01
|
| Ixx :   2.900E+00     Iyy :   2.900E+00     J   :   4.580E+00
|
| Sxx+:   1.657E+00     Sxx-:   1.657E+00     Zxx :   1.930E+00
|
| Syy+:   1.657E+00     Syy-:   1.657E+00     Zyy :   1.930E+00
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        8927.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:         7.500
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    90.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender      27.17       N/A      33.72     Table.4.1a.Case6
|
| Web   : NonSlender      27.17       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    90.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonCompact      27.17      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         27.17      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
```

```
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       69.30       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  77.000     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       71.61       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.10694E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  95.480     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       50.60       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  65.585                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  9581.9     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5257.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  56.222     kip        Eq.E3-1
|
```

```
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       50.60       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  65.585               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  9581.9     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  5257.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  56.222     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                              -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       19.74       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  5.0000               Cl.G4
|
|  Nom. Shear Along X        : Vnx    =  21.938     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             3.263       19.74       0.165     Cl.G1          3      7.50
|
```

```
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  21.938     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
7
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                                - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -6.117       7.238       0.845     Cl.F7.1        3      3.75
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  8.0417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       7.238       0.000     Cl.F7.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  8.0417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -6.117       7.209       0.849     Cl.F7.2        3      3.75
|
```

```
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  8.0099     kip-ft     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       7.209       0.000     Cl.F7.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  8.0099     kip-ft     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.849      Eq.H1-1b           3        3.75
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  69.300     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  7.2089     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  7.2089     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       5.964       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  4320.0     kip/ft2    Eq.H3-3
|
|  Nom. Strength             : Tn     =  6.6268     kip-ft     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2016
    50. FLX 2 ALL
    51. METHOD ASD
    52. SGR 5 ALL
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                             -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
```

```
       Pz                FX         Axial force.
       Vy                FY         Shear force along minor axis.
       Vx                FZ         Shear force along major axis.
       Tz                MX         Torsional moment.
       My                MY         Bending moment about minor axis.
       Mx                MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
11
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST3.5X3.5X0.125   (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.850       Loadcase:        4
|
|  Location:      3.75       Ref:      Cl.F7.2
|
|  Pz:       0.000     T     Vy:       -.1192E-06       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -4.078
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     65.585
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
```

```
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      4             Ratio     :        0.850(PASS)           |
|         Loc   :    3.75            Condition :    Cl.F7.2                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.540E+00     Axx :   7.313E-01     Ayy :   7.313E-01               |
| Ixx :   2.900E+00     Iyy :   2.900E+00     J   :   4.580E+00               |
| Sxx+:   1.657E+00     Sxx-:   1.657E+00     Zxx :   1.930E+00               |
| Syy+:   1.657E+00     Syy-:   1.657E+00     Zyy :   1.930E+00               |
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00               |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        7200.000             Fu:        8927.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         7.500                                         |
| Design Parameters                                                (Rolled)   |
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:    90.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: NonSlender      27.17       N/A      33.72     Table.4.1a.Case6     |
| Web   : NonSlender      27.17       N/A      33.72     Table.4.1a.Case6     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:    90.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: NonCompact      27.17      26.97     33.72     Table.4.1b.Case17    |
| Web   : Compact         27.17      58.28    137.27     Table.4.1b.Case19    |
|-----------------------------------------------------------------------------|
```

```
     STAAD SPACE                                              -- PAGE NO.
12
                   STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                 ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR AXIAL TENSION
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE YIELDING
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       46.11       0.000     Cl.D2          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Ten. Yld Cap         : Pn     =  77.000     kip        Eq.D2-1
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE RUPTURE
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       47.74       0.000     Cl.D2          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Effective area            : Ae     = 0.10694E-01 ft2        Eq.D3-1
 |
 |  Nom. Ten. Rpt Cap         : Pn     =  95.480     kip        Eq.D2-2
 |
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR AXIAL COMPRESSION
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL BUCKLING X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       33.67       0.000     Cl.E3          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Effective Slenderness     : Lcx/rx =  65.585                Cl.E2
 |
```

```
|  Elastic Buckling Stress   : Fex    =  9581.9     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5257.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  56.222     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       33.67       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  65.585                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  9581.9     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  5257.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  56.222     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                            - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       13.14       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along X        : Vnx    =  21.938     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
```

```
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              2.175       13.14       0.166     Cl.G1          4      7.50
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000              Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  21.938     kip       Eq.G4-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                             -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -4.078       4.815       0.847     Cl.F7.1        4      3.75
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  8.0417     kip-ft    Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       4.815       0.000     Cl.F7.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  8.0417     kip-ft    Eq.F7-1
|
|-----------------------------------------------------------------------------
|
```

```
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -4.078       4.796       0.850     Cl.F7.2        4      3.75
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx     =  8.0099     kip-ft     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       4.796       0.000     Cl.F7.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny     =  8.0099     kip-ft     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
15
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.850      Eq.H1-1b           4        3.75
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf     =  1.0000                 Eq.H1-2
|
|  Axial Capacity            : Pc      =  46.108     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx     =  4.7964     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy     =  4.7964     kip-ft     Cl.H1.1
|
```

```
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                                    -- PAGE NO.
16
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                        CHECKS FOR TORSION
|
|--------------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY     RATIO     REFERENCE      L/C     LOC
|
|             0.000       3.968        0.000     Cl.H3.1         4       0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  4320.0     kip/ft2     Eq.H3-3
|
|  Nom. Strength             : Tn     =  6.6268     kip-ft      Eq.H3-1
|
|--------------------------------------------------------------------------------
|
```

V.AISC 360-16 HSST NonCompact Flange F.7

Verify the available flexural strength of a Rectangular HSS flexural member with non-compact flanges per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-34 – F-36, Example F.7B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a HSST 10x6x3/16. The member is used as a 21 ft simple span subject to a uniform dead load of 0.15 kip/ft and a uniform live load of 0.4 kip/ft. The member is braced at end points only. The material is ASTM A500 Grade C (F_y = *50 ksi*).

Validation

Ultimate load (for LRFD):

$w_u = 1.2(0.15) + 1.6(0.4) = 0.82$ kip/ft

$M_u = w_u L^2/8 = (0.82)(21)^2/8 = 45.2$ kip·ft

Service level load (for ASD):

$w_a = 0.15 + 0.4 = 0.55$ kip/ft

$M_a = w_a L^2/8 = (0.55)(21)^2/8 = 30.3$ kip·ft

Flange slenderness ratio

$\lambda = b/t_f = 31.5$ (From Reference 1, Table 1-11)

$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29,\ 000}{50}} = 27.0$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29,\ 000}{50}} = 33.7$$

$\lambda_p < \lambda < \lambda_r$, therefore, the flange is non-compact .

Web slenderness ratio:

$\lambda = h/t_f = 54.5$ (From Reference 1, Table 1-11)

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29,\ 000}{50}} = 58.3$$

$\lambda < \lambda_p$, therefore, the web is compact (limit state of web local buckling need not be checked).

<u>Flange Local Buckling</u>

$M_p = F_y Z_x = 50 \times 18.0 = 900$ in·k = 75 ft·k

For non-compact sections:

$$M_n = M_p - \left(M_p - F_y S\right)\left(3.57\frac{b}{t_f}\sqrt{\frac{F_y}{E}} - 4.0\right) \le M_p \qquad \text{[Eqn. F7-2]}$$

$$M_n = 900 - \left(900 - 50 \times 14.9\right)\left(3.57 \times 31.5\sqrt{\frac{50}{29,\ 000}} - 4.0\right) = 796 \text{ in}\cdot\text{k} = 66.4 \text{ ft}\cdot\text{k}$$

<u>Yielding and Lateral-Torsional Buckling</u>

$L_b = 21$ ft = 252 in

$$L_p = 0.13Er_y\frac{\sqrt{JA_g}}{M_p} = 0.13(29,\ 000)(2.52)\frac{\sqrt{73.8 \times 5.37}}{900} = 210 \text{ in}$$

$$L_p = 2Er_y\frac{\sqrt{JA_g}}{M_p} = 2(29,\ 000)(2.52)\frac{\sqrt{73.8 \times 5.37}}{900} = 5,\ 580 \text{ in}$$

The moment at the quarter and three-quarter points on a simply supported beam under uniform distributed load (Table 3-23, Case 1 in Reference 1) is $3/32wl^2$, which is simply 3/4 of the maximum moment at midspan. In other words, $M_A = M_C = 3/4 \times M_B$. Therefore, the lateral-torsional buckling modification factor becomes:

$$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C} = \frac{12.5(1.0)}{2.5(1.0) = 3(0.75) + 4(1.0) = 3(0.75)} = 1.14 \qquad \text{[Eqn. F1-1]}$$

$L_p < L_b < L_r$, therefore the nominal moment in the lateral-torsional limit state is given by:

$$M_n = C_b\left[M_p - \left(M_p + 0.7F_yS_x\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le M_p$$ [Eqn. F7-10]

$$M_n = 1.14\left\{900 - \left[900 + 0.7(50)(14.9)\right]\left(\frac{252 - 210}{5,\ 580 - 210}\right)\right\} = 1,\ 020\ \text{in}\cdot\text{k} = 85\ \text{ft}\cdot\text{k}$$

Flange buckling controls, $M_n = 66.4\ ft\cdot k$

LRFD

$$\phi_bM_n = 59.8\ \text{ft}\cdot\text{k}$$

ASD

$$M_n/\Omega_b = 39.8\ \text{ft}\cdot\text{k}$$

Results

Table 718: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Flange classification	non-compact	non-compact	none	
ϕ_bM_n (ft·k)	59.8	59.80	none	
M_n/Ω_b (ft·k)	39.8	39.79	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 HSST NonCompact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST HSST10X6X0.188
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.15
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.4
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
CB 0 ALL
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
CB 0 ALL
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
```

```
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.   
4
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST10X6X0.188      (AISC 
SECTIONS)|
|  Status:        PASS       Ratio:         0.756       Loadcase:       3      
|
|  Location:     10.50       Ref:      Cl.F7.2                                 
|
|  Pz:       0.000     T     Vy:       0.1431E-06       Vx:      0.000         
|
|  Tz:       0.000           My:        0.000           Mx:     -45.20         
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS                                                          
|
| Actual Slenderness Ratio    :    100.002                                     
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00           
```

```
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.756(PASS)
|
|          Loc  :   10.50            Condition :    Cl.F7.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    10.50, PROPERTIES UNIT: IN  )
|
| Ag  :   5.370E+00     Axx :   1.906E+00     Ayy :   3.298E+00
|
| Ixx :   7.460E+01     Iyy :   3.410E+01     J   :   7.380E+01
|
| Sxx+:   1.492E+01     Sxx-:   1.492E+01     Zxx :   1.800E+01
|
| Syy+:   1.137E+01     Syy-:   1.137E+01     Zyy :   1.270E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        8927.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        21.000
|
| Design Parameters                                           (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   252.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender      31.48       N/A      33.72     Table.4.1a.Case6
|
| Web   : Slender         54.47       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   252.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonCompact      31.48      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         54.47      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
```

```
|
     STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       241.6       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap          : Pn     =  268.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       249.7       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area             : Ae     = 0.37292E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap          : Pn     =  332.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       152.4       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness      : Lcx/rx =  67.611                Cl.E2
```

```
|
| Elastic Buckling Stress   : Fex    =  9016.2     kip/ft2    Eq.E3-4
|
| Crit. Buckling Stress     : Fcrx   =  5154.4     kip/ft2    Eq.E3-2
|
| Nom. Flexural Buckling    : Pnx    =  169.38     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       111.2       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
| Effective Slenderness     : Lcy/ry =  100.00                Cl.E2
|
| Elastic Buckling Stress   : Fey    =  4121.3     kip/ft2    Eq.E3-4
|
| Crit. Buckling Stress     : Fcry   =  3465.6     kip/ft2    Eq.E3-2
|
| Nom. Flexural Buckling    : Pny    =  123.59     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       51.47       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
| Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
| Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
|
| Nom. Shear Along X        : Vnx    =  57.190     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
```

```
||
|| SHEAR ALONG Y
||
||              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||              8.610       89.06       0.097     Cl.G1          3      0.00
||
||
||
|| Intermediate Results :
||
||  Coefficient Cv Along Y    : Cv     =  1.0000               Eq.G2-9
||
||  Coefficient Kv Along Y    : Kv     =  5.0000               Cl.G4
||
||  Nom. Shear Along Y        : Vny    =  98.950     kip       Eq.G4-1
||
||-----------------------------------------------------------------------------
||
     STAAD SPACE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
||-----------------------------------------------------------------------------
||
||                        CHECKS FOR BENDING
||
||-----------------------------------------------------------------------------
||
|| FLEXURAL YIELDING (X)
||
||              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             -45.20       67.50       0.670     Cl.F7.1        3     10.50
||
||
||
|| Intermediate Results :
||
||  Nom Flex Yielding Along X : Mnx    =  75.000     kip-ft     Eq.F7-1
||
||-----------------------------------------------------------------------------
||
|| FLEXURAL YIELDING (Y)
||
||              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||              0.000       47.63       0.000     Cl.F7.1        3      0.00
||
||
||
|| Intermediate Results :
||
||  Nom Flex Yielding Along Y : Mny    =  52.917     kip-ft     Eq.F7-1
||
||-----------------------------------------------------------------------------
```

```
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -45.20       67.50       0.670     Cl.F7.4        3     10.50
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  75.000     kip-ft     Eq.F7-10
|
|  Mom. Distr. factor        : CbX    =  1.1364                Eq.F1-1
|
|  Limiting Unbraced Length  : LpX    =  17.511     ft         Eq.F7-12
|
|  Limiting Unbraced Length  : LrX    =  464.32     ft         Eq.F7-13
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -45.20       59.80       0.756     Cl.F7.2        3     10.50
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  66.442     kip-ft     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       33.73       0.000     Cl.F7.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  37.476     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
```

```
|
|--------------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.756      Eq.H1-1b           3       10.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000               Eq.H1-2
|
|  Axial Capacity            : Pc     =  241.65     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  59.797     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  33.728     kip-ft     Cl.H1.1
|
|--------------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                        CHECKS FOR TORSION
|
|--------------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       44.78       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  4320.0     kip/ft2    Eq.H3-3
|
|  Nom. Strength             : Tn     =  49.753     kip-ft     Eq.H3-1
|
|--------------------------------------------------------------------------------
|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2016
    50. METHOD ASD
    51. CB 0 ALL
    52. SGR 5 ALL
    53. TRACK 2 ALL
```

```
    54. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
```

```
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
11
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST10X6X0.188      (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.762       Loadcase:       4
|
|  Location:     10.50       Ref:      Cl.F7.2
|
|  Pz:       0.000     T     Vy:       0.1192E-06       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -30.32
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    100.002
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.762(PASS)
|
|          Loc  :   10.50            Condition :    Cl.F7.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    10.50, PROPERTIES UNIT: IN  )
|
| Ag  :   5.370E+00     Axx :   1.906E+00     Ayy :   3.298E+00
|
| Ixx :   7.460E+01     Iyy :   3.410E+01     J   :   7.380E+01
|
| Sxx+:   1.492E+01     Sxx-:   1.492E+01     Zxx :   1.800E+01
|
| Syy+:   1.137E+01     Syy-:   1.137E+01     Zyy :   1.270E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        8927.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        21.000
```

```
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender      31.48       N/A      33.72     Table.4.1a.Case6
|
| Web   : Slender         54.47       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonCompact      31.48      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         54.47      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       160.8       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  268.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       166.5       0.000     Cl.D2          4      0.00
```

```
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.37292E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  332.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              0.000       101.4       0.000     Cl.E3           4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  67.611                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  9016.2     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5154.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  169.38     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              0.000       74.01       0.000     Cl.E3           4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  100.00                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  4121.3     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  3465.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  123.59     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
```

```
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       34.25       0.000     Cl.G1          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
 |
 |  Nom. Shear Along X        : Vnx    =  57.190     kip        Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             5.775       59.25       0.097     Cl.G1          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
 |
 |  Nom. Shear Along Y        : Vny    =  98.950     kip        Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                               -- PAGE NO.
14
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (X)
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -30.32       44.91       0.675     Cl.F7.1        4     10.50
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  75.000     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       31.69       0.000     Cl.F7.1        4      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  52.917     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -30.32       44.91       0.675     Cl.F7.4        4     10.50
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  75.000     kip-ft     Eq.F7-10
|
|  Mom. Distr. factor        : CbX    =  1.1364                Eq.F1-1
|
|  Limiting Unbraced Length  : LpX    =  17.511     ft         Eq.F7-12
|
|  Limiting Unbraced Length  : LrX    =  464.32     ft         Eq.F7-13
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -30.32       39.79       0.762     Cl.F7.2        4     10.50
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  66.442     kip-ft     Eq.F7-2
```

```
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       22.44       0.000     Cl.F7.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  37.476     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
15
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.762      Eq.H1-1b           4       10.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  160.78     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  39.785     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  22.441     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
16
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR TORSION
```

```
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       29.79       0.000     Cl.H3.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr          : Fcr     =  4320.0     kip/ft2    Eq.H3-3
|
|  Nom. Strength              : Tn      =  49.753     kip-ft     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 HSST Slender Flange F.8

Verify the available flexural strength of a square HSS flexural member with slender flange per AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-39 – F-41, Example F.8B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a HSST 18x8x3/16. The member is used as a 21 ft simple span subject to a uniform dead load of 0.125 kip/ft and a uniform live load of 0.375 kip/ft. The member is continuously braced. The material is ASTM A500 Grade C ($F_y = 50\ ksi$).

Validation

Ultimate load (for LRFD):

$$w_u = 1.2(0.125) + 1.6(0.375) = 0.75 \text{ kip/ft}$$

$$M_u = w_u L^2/8 = (0.75)(21)^2/8 = 41.3 \text{ kip·ft}$$

Service level load (for ASD):

$$w_a = 0.125 + 0.375 = 0.50 \text{ kip/ft}$$

$$M_a = w_a L^2/8 = (0.50)(21)^2/8 = 27.6 \text{ kip·ft}$$

Flange slenderness ratio

$$\lambda = b/t_f = 43.0 \text{ (From Reference 1, Table 1-12)}$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29,000}{50}} = 33.7$$

$\lambda > \lambda_r$, therefore, the flange is slender.

Web slenderness ratio:

$\lambda = h/t_f = 54.4$ (From Reference 1, Table 1-12)

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29,000}{50}} = 58.3$$

$\lambda < \lambda_p$, therefore, the web is compact (limit state of web local buckling need not be checked).

Flange Local Buckling

$M_n = F_y S_e$ [Eqn. F7-3]

where

S_e = the effect section modulus $\frac{I_{eff}}{(H / 2)}$

I_{eff} = the effective moment of inertia. A simplified method as presented in Reference 2 is used, with the effective flange width:

$$I_{eff} \approx I_x - \left(\Sigma \frac{bt^3}{12} + \Sigma a d^2\right)$$

b_e = the effective flange width of the compression flange, per Section F7.2(c) of Reference 1:

$$= 1.92 t_f \sqrt{\frac{E}{F_y}}\left(1 - \frac{0.38}{b / t_f}\sqrt{\frac{E}{F_y}}\right) \leq b$$

b = usable flange width

= b - 3t = 8.0 - 3(0.174) = 7.48 in

$$b_e = 1.92(0.174)\sqrt{\frac{29,000}{50}}\left(1 - \frac{0.38}{43.0}\sqrt{\frac{29,000}{50}}\right) = 6.33 \text{ in}$$

The portion of the compression flange that is not effective: $b - b_e = 7.48 - 6.33 = 1.15$ *in*

$$I_{eff} = 54.4 - 2\left[\frac{1.15(0.174)^3}{12} + 1.15(0.174)\left(\frac{8.0 - 0.175}{2}\right)^2\right] = 48.3 \text{ in}^4$$

$$S_e = \frac{48.3}{8.0/2} = 12.1 \text{ in}^3$$

$M_n = 50 \times 12.1 = 605$ in·k = 50.4 ft·k

LRFD

$\phi_b M_n = 45.4$ ft·k

ASD

$M_n/\Omega_b = 30.2$ ft·k

Results

Table 719: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Flange slenderness	slender	slender	none	
$\phi_b M_n$ (ft·k)	45.4	46.28	1.9%	
M_n / Ω_b (ft·k)	30.2	30.79	2.0%	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 HSST Slender Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST8X8X0.188
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.125
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.375
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
```

```
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD LRFD
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD ASD
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
```

```
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.    
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST8X8X0.188       (AISC 
SECTIONS)|
|  Status:        PASS       Ratio:         0.893      Loadcase:       3      
|
|  Location:     10.50       Ref:      Cl.F7.2                                
|
|  Pz:       0.000     T     Vy:        0.000          Vx:      0.000         
|
|  Tz:       0.000           My:        0.000          Mx:     -41.34         
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS                                                         
|
| Actual Slenderness Ratio    :     79.175                                    
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS                                                             
|
| Critical L/C  :      3              Ratio     :        0.893(PASS)          
|
|          Loc  :   10.50            Condition :    Cl.F7.2                   
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    10.50, PROPERTIES UNIT: IN  )                  
|
| Ag  :   5.370E+00     Axx :   2.602E+00     Ayy :   2.602E+00               
|
```

```
| Ixx :   5.440E+01     Iyy :   5.440E+01     J   :   8.450E+01
|
| Sxx+:   1.360E+01     Sxx-:   1.360E+01     Zxx :   1.570E+01
|
| Syy+:   1.360E+01     Syy-:   1.360E+01     Zyy :   1.570E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        8927.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        21.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Slender         42.98       N/A      33.72     Table.4.1a.Case6
|
| Web   : Slender         42.98       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Slender         42.98      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         42.98      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
5
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       241.6       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  268.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
| TENSILE RUPTURE
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       249.7       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     = 0.37292E-01 ft2        Eq.D3-1
|  Nom. Ten. Rpt Cap         : Pn     =  332.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|                        CHECKS FOR AXIAL COMPRESSION
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       151.6       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  79.175                Cl.E2
|  Elastic Buckling Stress   : Fex    =  6574.8     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  4552.8     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  168.40     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       151.6       0.000     Cl.E3          3      0.00
|
```

```
 |
 |
 | Intermediate Results :
 |
 |  Effective Slenderness     : Lcy/ry =  79.175              Cl.E2
 |
 |  Elastic Buckling Stress   : Fey    =  6574.8     kip/ft2    Eq.E3-4
 |
 |  Crit. Buckling Stress     : Fcry   =  4552.8     kip/ft2    Eq.E3-2
 |
 |  Nom. Flexural Buckling    : Pny    =  168.40     kip        Eq.E7-1
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                               -- PAGE NO.
6
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                          CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       70.26       0.000     Cl.G1          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  5.0000              Cl.G4
 |
 |  Nom. Shear Along X        : Vnx    =  78.070     kip        Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             7.875       70.26       0.112     Cl.G1          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000              Eq.G2-9
 |
 |  Coefficient Kv Along Y    : Kv     =  5.0000              Cl.G4
 |
```

```
|  Nom. Shear Along Y        : Vny    =  78.070     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -41.34       58.88       0.702     Cl.F7.1        3     10.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  65.417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       58.88       0.000     Cl.F7.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  65.417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -41.34       46.28       0.893     Cl.F7.2        3     10.50
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  51.425     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
```

```
| WEB LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       46.28       0.000     Cl.F7.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  51.425     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.893      Eq.H1-1b           3       10.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  241.65     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  46.283     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  46.283     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
9
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
```

```
| TORSION CAPACITY
|
|              DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000        47.91       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  4320.0     kip/ft2    Eq.H3-3
|
|  Nom. Strength             : Tn     =  53.233     kip-ft     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2016
    50. FLX 2 ALL
    51. METHOD ASD
    52. SGR 5 ALL
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ========================
```

```
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST8X8X0.188       (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.895       Loadcase:       4
|
|  Location:     10.50       Ref:      Cl.F7.2
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -27.56
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     79.175
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.895(PASS)
|
|          Loc  :   10.50            Condition :    Cl.F7.2
|
```

```
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    10.50, PROPERTIES UNIT: IN  )
|
| Ag  :   5.370E+00     Axx :   2.602E+00     Ayy :   2.602E+00
|
| Ixx :   5.440E+01     Iyy :   5.440E+01     J   :   8.450E+01
|
| Sxx+:   1.360E+01     Sxx-:   1.360E+01     Zxx :   1.570E+01
|
| Syy+:   1.360E+01     Syy-:   1.360E+01     Zyy :   1.570E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        8927.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        21.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Slender         42.98       N/A      33.72     Table.4.1a.Case6
|
| Web   : Slender         42.98       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   252.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Slender         42.98      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         42.98      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                    - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
```

```
                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       160.8       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap          : Pn     =  268.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       166.5       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.37292E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap          : Pn     =  332.94     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       100.8       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  79.175                 Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  6574.8     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4552.8     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  168.40     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
```

```
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       100.8       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  79.175                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  6574.8     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4552.8     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  168.40     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
13
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       46.75       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along X        : Vnx    =  78.070     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              5.250       46.75       0.112     Cl.G1          4      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000               Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  78.070     kip       Eq.G4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
14
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -27.56       39.17       0.704     Cl.F7.1        4     10.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  65.417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       39.17       0.000     Cl.F7.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  65.417     kip-ft     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -27.56       30.79       0.895     Cl.F7.2        4     10.50
|
|
|
```

```
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  51.425     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       30.79       0.000     Cl.F7.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  51.425     kip-ft     Eq.F7-3
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
15
                  STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.895      Eq.H1-1b           4       10.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  160.78     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  30.793     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  30.793     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
16
                  STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
```

```
|--------------------------------------------------------------------------
|
|                        CHECKS FOR TORSION
|
|--------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       31.88       0.000     Cl.H3.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr          : Fcr     =  4320.0     kip/ft2     Eq.H3-3
|
|  Nom. Strength              : Tn      =  53.233     kip-ft      Eq.H3-1
|
|--------------------------------------------------------------------------
|
```

V.AISC 360-16 I Minor Axis Bending F.5

Check the minor axis bending capacity of an I-shaped flexural member per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-28 – F-29, Example F.5

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a W12x58 in bending about the weak axis The member is used as a 15 ft simple span subject to a uniform dead load of 0.667 kip/ft and a uniform live load of 2.0 kip/ft. The member is braced at end points only. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

Ultimate load (for LRFD):

$$w_u = 1.2(0.667) + 1.6(2.0) = 4.0 \text{ kip/ft}$$

$$M_u = w_u L^2/8 = (4.0)(15)^2/8 = 112.5 \text{ kip·ft}$$

Service level load (for ASD):

$$w_a = 0.667 + 2.0 = 2.67 \text{ kip/ft}$$

$$M_a = w_a L^2/8 = (2.67)(15)^2/8 = 75.1 \text{ kip·ft}$$

Per the User Note in F6.2, the W12x58 has compact flanges in minor axis bending. Therefore, flange local buckling does not control. Yielding

$$M_p = F_y Z_y \le 1.6 F_y S_y \quad \text{[Eqn. F6-1]}$$

$$M_p = 50 \times 32.5 = 1{,}625 \text{ in}\cdot\text{k} = 135.4 \text{ ft}\cdot\text{k}$$

$$1.6 \times 50 \times 21.4 = 1{,}712 \text{ in}\cdot\text{k} = 142.7 \text{ ft}\cdot\text{k}$$

$$M_p = 135.4 \text{ ft}\cdot\text{k}$$

LRFD

$$\phi_b M_n = 121.9 \text{ ft}\cdot\text{k}$$

ASD

$$M_n/\Omega_b = 81.2 \text{ ft}\cdot\text{k}$$

Results

Table 720: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_{ny}$ (ft·k)	121.9	121.9	none	
M_{ny}/Ω_b (ft·k)	81.2	81.09	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 I Minor Axis Bending F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 15 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X58
CONSTANTS
```

```
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MY
1 FIXED BUT MY
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GZ -0.667
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GZ -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
```

```
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  W12X58             (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.923       Loadcase:       3
 |
 |  Location:      7.50       Ref:      Cl.F6.1
 |
 |  Pz:       0.000     T     Vy:        0.000           Vx:     0.5722E-06
 |
 |  Tz:       0.000           My:        112.5           Mx:      0.000
 |
 |-----------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     71.747
 |
 | Allowable Slenderness Ratio :    300.000           LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
```

```
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.923(PASS)
|
|          Loc  :    7.50            Condition :    Cl.F6.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     7.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.700E+01     Axx :   1.280E+01     Ayy :   4.392E+00
|
| Ixx :   4.750E+02     Iyy :   1.070E+02     J   :   2.100E+00
|
| Sxx+:   7.787E+01     Sxx-:   7.787E+01     Zxx :   8.640E+01
|
| Syy+:   2.140E+01     Syy-:   2.140E+01     Zyy :   3.250E+01
|
| Cw  :   3.564E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        15.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   180.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       7.81       N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      27.00       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   180.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          7.81       9.15     24.08     Table.4.1b.Case13
|
| Web   : Compact         27.00     ******    ******     Table.4.1.
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
```

```
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       765.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  850.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       828.8       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.11806     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  1105.0     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       702.8       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  34.053                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  35543.     kip/ft2    Eq.E3-4
```

```
||
|| Crit. Buckling Stress     : Fcrx   =  6614.7     kip/ft2    Eq.E3-2
||
|| Nom. Flexural Buckling    : Pnx    =  780.90     kip        Eq.E3-1
||
||-----------------------------------------------------------------------------
||
|| FLEXURAL BUCKLING Y
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||             0.000       525.0       0.000     Cl.E3          3      0.00
||
||
|| Intermediate Results :
|| Effective Slenderness     : Lcy/ry =  71.747                Cl.E2
||
|| Elastic Buckling Stress   : Fey    =  8006.6     kip/ft2    Eq.E3-4
||
|| Crit. Buckling Stress     : Fcry   =  4941.6     kip/ft2    Eq.E3-2
||
|| Nom. Flexural Buckling    : Pny    =  583.39     kip        Eq.E3-1
||
||-----------------------------------------------------------------------------
||
|| FLEXURAL-TORSIONAL-BUCKLING
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||             0.000       613.0       0.000     Cl.E4          3      0.00
||
||
|| Intermediate Results :
|| Elastic F-T-B Stress      : Fe     =  13608.     kip/ft2    Eq.E4-2
||
|| Crit. F-T-B Stress        : Fcr    =  5769.8     kip/ft2    Eq.E3-2
||
|| Nom. Flex-tor Buckling    : Pn     =  681.15     kip        Eq.E4-1
||
||-----------------------------------------------------------------------------
||
      STAAD SPACE                                           -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
```

```
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             30.00       345.6       0.087     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  384.00     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       131.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  131.76     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       324.0       0.000     Cl.F2.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  360.00     kip-ft    Eq.F2-1
```

```
||
||---------------------------------------------------------------------------
||
|| FLEXURAL YIELDING (Y)
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||            -112.5       121.9       0.923     Cl.F6.1        3      7.50
||
||
||
|| Intermediate Results :
||
||  Nom Flex Yielding Along Y : Mny    =  135.42     kip-ft     Eq.F6-1
||
||---------------------------------------------------------------------------
||
|| LAT TOR BUCK ABOUT X
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             0.000       289.2       0.000     Cl.F2.2        3      0.00
||
||
||
|| Intermediate Results :
||
||  Nom L-T-B Cap             : Mnx    =  321.34     kip-ft     Eq.F2-2
||
||  Mom. Distr. factor        : CbX    =  1.0000                Custom
||
||  Limiting Unbraced Length  : LpX    =  8.8616     ft         Eq.F2-5
||
||  coefficient C             : Cx     =  1.0000                Eq.F2-8a
||
||  Effective Rad. of Gyr.    : Rts    = 0.23467     ft         Eq.F2-7
||
||  Limiting Unbraced Length  : LrX    =  29.958     ft         Eq.F2-6
||
||---------------------------------------------------------------------------
||
     STAAD SPACE                                           -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
||---------------------------------------------------------------------------
||
||                          CHECKS FOR AXIAL BEND INTERACTION
||
||---------------------------------------------------------------------------
||
|| COMBINED FORCES CLAUSE H1
||
||                            RATIO      CRITERIA          L/C      LOC
||
||                            0.923      Eq.H1-1b           3        7.50
```

```
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  765.00     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  289.20     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  121.88     kip-ft     Cl.H1.1
|
|------------------------------------------------------------------------------
|
    46. LOAD LIST 4
    47. PARAMETER 2
    48. CODE AISC UNIFIED 2016
    49. METHOD ASD
    50. SGR 28 ALL
    51. TRACK 2 ALL
    52. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
```

```
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W12X58             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.925       Loadcase:       4
|
|  Location:      7.50       Ref:      Cl.F6.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:     0.4768E-06
|
|  Tz:       0.000           My:        75.01           Mx:      0.000
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     71.747
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.925(PASS)
|
|          Loc  :    7.50            Condition :    Cl.F6.1
|
|------------------------------------------------------------------------------
|
```

```
| SECTION PROPERTIES  (LOC:     7.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.700E+01     Axx :   1.280E+01     Ayy :   4.392E+00
|
| Ixx :   4.750E+02     Iyy :   1.070E+02     J   :   2.100E+00
|
| Sxx+:   7.787E+01     Sxx-:   7.787E+01     Zxx :   8.640E+01
|
| Syy+:   2.140E+01     Syy-:   2.140E+01     Zyy :   3.250E+01
|
| Cw  :   3.564E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        15.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   180.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       7.81       N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      27.00       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   180.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          7.81       9.15     24.08     Table.4.1b.Case13
|
| Web   : Compact         27.00     ******    ******     Table.4.1.
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
11
                          STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                        ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
```

```
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       509.0       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  850.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       552.5       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.11806     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  1105.0     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       467.6       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  34.053                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  35543.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6614.7     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  780.90     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       349.3       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  71.747               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  8006.6     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4941.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  583.39     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       407.9       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  13608.     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  5769.8     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  681.15     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              20.00       229.9       0.087     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
```

```
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  384.00     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       87.84       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  131.76     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
13
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       215.6       0.000     Cl.F2.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  360.00     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -75.01       81.09       0.925     Cl.F6.1        4      7.50
|
```

```
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  135.42     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       192.4       0.000     Cl.F2.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap              : Mnx    =  321.34     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor         : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length   : LpX    =  8.8616     ft         Eq.F2-5
|
|  coefficient C              : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.     : Rts    = 0.23467     ft         Eq.F2-7
|
|  Limiting Unbraced Length   : LrX    =  29.958     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.925      Eq.H1-1b           4        7.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity             : Pc     =  508.98     kip        Cl.H1.1
|
```

```
|  Moment Capacity           : Mcx    =  192.42     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  81.088     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------
|
```

V.AISC 360-16 Pipe F.9

Verify the available flexural strength of a compact pipe member per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-43 – F-44, Example F.9B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of an 8" x-strong pipe. The member is used as a 16 ft simple span. The member is braced only at end points. The material is ASTM A53 Grade B ($F_y = 35\ ksi$).

Validation

Slenderness ratio

$$\lambda = D/t = 18.5 \text{ (From Reference 1, Table 1-14)}$$

$$\lambda_p = 0.07E/F_y = 0.07 \times (29{,}000 / 35) = 58$$

$\lambda < \lambda_p$, therefore, the section is compact (limit state of flange local buckling need not be checked).

$$\frac{0.45E}{F_y} = \frac{0.45(29{,}000)}{35} = 373 > 18.5$$

Therefore, section F8.1 (Round HSS) of Reference 1 applies:

$$M_n = M_p = F_y Z = 35 \times 31.0 = 1{,}085 \text{ in}\cdot\text{k} = 90.4 \text{ ft}\cdot\text{k}$$

LRFD

$$\phi_b M_n = 81.38 \text{ ft}\cdot\text{k}$$

ASD

$$M_n/\Omega_b = 54.14 \text{ ft}\cdot\text{k}$$

Results

Table 721: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k)	81.38	81.38	none	
M_n / Ω_b (ft·k)	54.14	54.14	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Pipe F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 16 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPX80
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.32
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.96
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
```

```
METHOD LRFD
SGR 2 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
SGR 2 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
```

```
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  PIPX80             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.755       Loadcase:        3
|
|  Location:      8.00       Ref:      Cl.F8.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -61.44
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     66.233
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.755(PASS)
|
|          Loc  :    8.00            Condition :    Cl.F8.1
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     8.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.190E+01     Axx :   0.000E+00     Ayy :   0.000E+00
|
| Ixx :   1.000E+02     Iyy :   1.000E+02     J   :   2.000E+02
|
| Sxx+:   2.319E+01     Sxx-:   2.319E+01     Zxx :   3.100E+01
|
| Syy+:   2.319E+01     Syy-:   2.319E+01     Zyy :   3.100E+01
```

```
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        5040.000             Fu:        8639.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        16.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   192.00)
|
|                           λ          λp         λr       CASE
|
| Flange: NonSlender      18.55       N/A      91.14     Table.4.1a.Case9
|
| Web   : NonSlender      18.55       N/A      91.14     Table.4.1a.Case9
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   192.00)
|
|                           λ          λp         λr       CASE
|
| Flange: Compact         18.55      58.00    256.86     Table.4.1b.Case20
|
| Web   : Compact         18.55      58.00    256.86     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       374.8       0.000     Cl.D2          3      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  416.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       535.5       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.82639E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  714.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       299.5       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  66.233                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  9395.3     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4026.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  332.74     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       299.5       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  66.233                Cl.E2
```

```
|
|  Elastic Buckling Stress   : Fey    =  9395.3     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4026.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  332.74     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                            -- PAGE NO.
6
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       112.5       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  124.95     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  3024.0     kip/ft2    Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              15.36       112.5       0.137     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  124.95     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  3024.0     kip/ft2    Eq.G5-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                            -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                         - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|--------------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -61.44       81.38       0.755     Cl.F8.1        3      8.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  90.417     kip-ft     Eq.F8-1
|
|--------------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       81.38       0.000     Cl.F8.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  90.417     kip-ft     Eq.F8-1
|
|--------------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|--------------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.755      Eq.H1-1b           3        8.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
```

```
|
|  Axial Capacity            : Pc     =  374.85     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  81.375     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  81.375     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       76.60       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  3024.0     kip/ft2    Cl.H3.1(a)
|
|  Nom. Strength             : Tn     =  85.112     kip-ft     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    45. LOAD LIST 4
    46. PARAMETER 2
    47. CODE AISC UNIFIED 2016
    48. METHOD ASD
    49. SGR 2 ALL
    50. TRACK 2 ALL
    51. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
```

```
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                 -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  PIPX80             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.757       Loadcase:       4
|
|  Location:      8.00       Ref:      Cl.F8.1
|
```

```
| Pz:       0.000     T     Vy:        0.000          Vx:      0.000
|
| Tz:       0.000           My:        0.000          Mx:     -40.96
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     66.233
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4            Ratio     :        0.757(PASS)
|
|          Loc  :    8.00           Condition :    Cl.F8.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     8.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.190E+01     Axx :   0.000E+00     Ayy :   0.000E+00
|
| Ixx :   1.000E+02     Iyy :   1.000E+02     J   :   2.000E+02
|
| Sxx+:   2.319E+01     Sxx-:   2.319E+01     Zxx :   3.100E+01
|
| Syy+:   2.319E+01     Syy-:   2.319E+01     Zyy :   3.100E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        5040.000              Fu:        8639.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        16.000
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   192.00)
|
|                           λ          λp         λr       CASE
|
| Flange: NonSlender      18.55       N/A      91.14     Table.4.1a.Case9
|
| Web   : NonSlender      18.55       N/A      91.14     Table.4.1a.Case9
|
```

```
||
||
|| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   192.00)
||
||                             λ         λp        λr       CASE
||
|| Flange: Compact         18.55      58.00    256.86     Table.4.1b.Case20
||
|| Web   : Compact         18.55      58.00    256.86     Table.4.1b.Case20
||
||--------------------------------------------------------------------------
||
     STAAD SPACE                                             -- PAGE NO.
12
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
||--------------------------------------------------------------------------
||
||                       CHECKS FOR AXIAL TENSION
||
||--------------------------------------------------------------------------
||
|| TENSILE YIELDING
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             0.000       249.4       0.000     Cl.D2          4      0.00
||
||
||
|| Intermediate Results :
||
||  Nom. Ten. Yld Cap         : Pn     =  416.50     kip        Eq.D2-1
||
||--------------------------------------------------------------------------
||
|| TENSILE RUPTURE
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             0.000       357.0       0.000     Cl.D2          4      0.00
||
||
||
|| Intermediate Results :
||
||  Effective area            : Ae     = 0.82639E-01 ft2        Eq.D3-1
||
||  Nom. Ten. Rpt Cap         : Pn     =  714.00     kip        Eq.D2-2
||
||--------------------------------------------------------------------------
||
||                       CHECKS FOR AXIAL COMPRESSION
||
||--------------------------------------------------------------------------
||
```

```
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       199.2       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  66.233                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  9395.3     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  4026.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  332.74     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       199.2       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  66.233                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  9395.3     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4026.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  332.74     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
13
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       74.82       0.000     Cl.G1          4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  124.95     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  3024.0     kip/ft2    Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             10.24       74.82       0.137     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  124.95     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  3024.0     kip/ft2    Eq.G5-2
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -40.96       54.14       0.757     Cl.F8.1        4      8.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  90.417     kip-ft     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       54.14       0.000     Cl.F8.1        4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  90.417     kip-ft     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
15
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.757      Eq.H1-1b           4        8.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  249.40     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  54.142     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  54.142     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
16
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       50.97       0.000     Cl.H3.1        4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  3024.0     kip/ft2     Cl.H3.1(a)
|
|  Nom. Strength             : Tn     =  85.112     kip-ft      Eq.H3-1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 W Flex Memb F.1-1A

Verify the available flexural strength of a continuously braced wide-flange member per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-8, Example F.1-1B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a W18x50. The member is used as a 35 ft simple span. The member is continuously braced. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

Per the User Note in F2, the W18x35 has compact flanges in major axis bending. Therefore, flange local buckling does not control. Further, as the member is continuously braced, lateral-torsional buckling does not control.

Yielding

$$M_n = M_p = F_y Z_x = 50 \times 101 = 5{,}050\ \mathrm{in \cdot k} = 420.8\ \mathrm{ft \cdot k}$$

LRFD

$$\phi_b M_n = 378.8\ \mathrm{ft \cdot k}$$

ASD

$$M_n / \Omega_b = 252.0\ \mathrm{ft \cdot k}$$

Results

Table 722: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k)	378.8	378.8	none	
M_n / Ω_b (ft·k)	252.0	252.0	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W Flex Memb F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
SGR 28 ALL
FLX 2 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
SGR 28 ALL
```

```
FLX 2 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
```

```
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W18X50              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.703       Loadcase:        3
|
|  Location:     17.50       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -266.4
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    254.294
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.703(PASS)
|
|          Loc  :   17.50            Condition :    Cl.F2.1
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
|
| Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
|
| Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
|
| Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
|
| Cw  :   3.044E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|------------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:       7200.000             Fu:       9359.999
```

```
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters                                                 (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   420.00)
|
|                           λ          λp         λr        CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         45.23       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   420.00)
|
|                           λ          λp         λr        CASE
|
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         45.23      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       661.5       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  735.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
```

```
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       716.6       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area             : Ae     = 0.10208     ft2        Eq.D3-1
|  Nom. Ten. Rpt Cap          : Pn     =  955.50     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       506.3       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness      : Lcx/rx =  56.933                Cl.E2
|  Elastic Buckling Stress    : Fex    =  12716.     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress      : Fcrx   =  5680.7     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling     : Pnx    =  562.50     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       51.36       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness      : Lcy/ry =  254.29                Cl.E2
|  Elastic Buckling Stress    : Fey    =  637.37     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress      : Fcry   =  558.97     kip/ft2    Eq.E3-3
|  Nom. Flexural Buckling     : Pny    =  57.061     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       260.0       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  3227.1     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  2829.9     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  288.89     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
9
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       230.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  256.50     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             30.45       191.7       0.159     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  191.70     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
10
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -266.4       378.8       0.703     Cl.F2.1        3     17.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  420.83     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       62.25       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  69.167     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
11
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
```

```
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.703      Eq.H1-1b           3       17.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000               Eq.H1-2
|
|  Axial Capacity            : Pc     =  661.50     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  378.75     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  62.250     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2016
    51. METHOD ASD
    52. SGR 28 ALL
    53. FLX 2 ALL
    54. TRACK 2 ALL
    55. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                  -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
```

```
      Vy              FY         Shear force along minor axis.
      Vx              FZ         Shear force along major axis.
      Tz              MX         Torsional moment.
      My              MY         Bending moment about minor axis.
      Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.   
13
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W18X50             (AISC 
SECTIONS)|
|  Status:        PASS       Ratio:         0.729       Loadcase:        4     
|
|  Location:     17.50       Ref:      Cl.F2.1                                 
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000         
|
|  Tz:       0.000           My:        0.000           Mx:     -183.8         
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS                                                         
|
| Actual Slenderness Ratio    :    254.294                                    
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00          
|
|-----------------------------------------------------------------------------
```

```
|
| STRENGTH CHECKS
|
| Critical L/C  :      4            Ratio     :       0.729(PASS)
|
|         Loc  :   17.50           Condition :    Cl.F2.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
|
| Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
|
| Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
|
| Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
|
| Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
|
| Cw  :   3.044E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   420.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         45.23       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   420.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         45.23      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
```

```
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       440.1       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  735.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       477.7       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.10208     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  955.50     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       336.8       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  56.933               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12716.     kip/ft2    Eq.E3-4
```

```
|
|  Crit. Buckling Stress     : Fcrx   =  5680.7     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  562.50     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       34.17       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  254.29                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  637.37     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  558.97     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  57.061     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       173.0       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  3227.1     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  2829.9     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  288.89     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
15
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
```

```
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       153.6       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000              Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  256.50     kip      Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              21.00       127.8       0.164     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  191.70     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
16
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -183.8       252.0       0.729     Cl.F2.1        4     17.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  420.83     kip-ft   Eq.F2-1
```

```
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       41.42       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  69.167     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
17
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.729      Eq.H1-1b           4       17.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  440.12     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  252.00     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  41.417     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V. AISC 360-16 W Flexural Check LRFD

Verify the flexural yielding strength and lateral torsional buckling of an I section using the LRFD method per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the flexural yielding strength and lateral torsional buckling of an I section (W12X90) gr. ASTM A992 material, with a member length of 41'-8", and a concentrated moment of 35,000 in-kip applied at midspan. The member is unbraced along its length.

Validation

Flexural yielding about X axis

As the section is compact, nominal flexural yielding strength about X axis is:

$$M_{nx} = f_y \times Z_{xx} = 65 \text{ ksi} \times 311 \text{ in}^3 = 20{,}215 \text{ in}\cdot\text{kip} \qquad \text{(F2-1, ref. 1)}$$

$$M_x = \phi_c \times M_{nx} = 0.9(20{,}215 \text{ in}\cdot\text{kip}) = 18{,}193.5 \text{ in}\cdot\text{kip}$$

Flexural yielding about Y axis

As this is the minor axis, nominal flexural yielding strength about Y axis is:

$$M_{ny} = f_y \times Z_{yy} = 65 \text{ ksi} \times 143 \text{ in}^3 = 9{,}295 \text{ in}\cdot\text{kip} \qquad \text{(F6-1, ref. 1)}$$

$$M_y = \phi_c \times M_{ny} = 0.9(9{,}295 \text{ in}\cdot\text{kip}) = 8{,}365.5 \text{ in}\cdot\text{kip}$$

Lateral torsional buckling about X axis

$$r_y = \sqrt{I_{yy} / A_g} = 3.243\text{in}^2$$

limiting laterally unbraced length for the yielding limit state:

$$L_p = 1.76 \times r_y \sqrt{\frac{E}{f_y}} = 120.56\text{in} \qquad \text{(F2-5, ref. 1)}$$

$$r_{ts} = \sqrt{\frac{\sqrt{I_{yy} \times C_w}}{S_{xx}}} = 3.769\text{in} \qquad \text{(F2-7, ref. 1)}$$

For an I section, C = 1 (per F2-28a in ref. 1). The limiting unbraced length for the inelastic lateral-torsional buckling limit state is:

$$h_0 = d - t_f = 12.66 \text{ in}$$

$$L_r = 1.95 \times r_{ts} \frac{E}{0.7F_y} \sqrt{\frac{Jc}{S_{xx} h_0} + \sqrt{\left(\frac{Jc}{S_{xx} h_0}\right)^2 + 6.76\left(\frac{0.7F_y}{E}\right)^2}} = 810.27\text{in}$$

$$L_p < L_b = 500 \text{ in} < L_r$$

$$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_c} = \frac{12.5(20,\ 000)}{2.5(20,\ 000) + 3(10,\ 000) + 4(20,\ 000) + 3(10,\ 000)} = 1.316$$

$$M_p = f_y \times Z_{xx} = 65 \text{ ksi} \times 311 \text{ in}^3 = 20{,}215 \text{ in}\cdot\text{kip}$$

Nominal LTB strength:

$$M_n = C_b\left[M_p - \left(M_p - 0.7F_yS_x\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le M_p$$

M_n = 20,611.4 in·kip > M_p

$\phi_c M_{nx}$ = 0.9(20,215 in·kip) = 18,193.5 in·kip

Results

Table 723: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Flexural yielding capacity, X axis (in·kip)	18,194	18,190	negligible	
Flexural buckling capacity, Y axis (in·kip)	8366	8366	none	
Lateral-torsional buckling about X axis (in·kip)	18,194	18,190	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W Flexural Check LRFD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 41.67 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X190
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 PINNED
2 FIXED BUT MY
UNIT INCHES KIP
LOAD 2 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CMOM GZ 35000 250
UNIT FEET KIP
PERFORM ANALYSIS
PARAMETER 1
CODE AISC UNIFIED 2016
LY 41.6667 ALL
LZ 41.6667 ALL
CB 0 ALL
FU 9360 ALL
FYLD 9360 ALL
METHOD LRFD
PROFILE W12 ALL
STP 1 ALL
TRACK 2 ALL
UNIT KIP INCHES
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ========================
```

```
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W12X190            (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.903      Loadcase:        2
|
|  Location:    208.35       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        78.85          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:     -.1643E+05
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    154.185
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      2             Ratio     :        0.903(PASS)
|
|         Loc   :  208.35            Condition :    Cl.F2.1
|
```

```
|--------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:   208.35, PROPERTIES UNIT: IN  )
|
| Ag  :   5.600E+01     Axx :   4.420E+01     Ayy :   1.526E+01
|
| Ixx :   1.890E+03     Iyy :   5.890E+02     J   :   4.880E+01
|
| Sxx+:   2.625E+02     Sxx-:   2.625E+02     Zxx :   3.110E+02
|
| Syy+:   9.276E+01     Syy-:   9.276E+01     Zyy :   1.430E+02
|
| Cw  :   2.380E+04     x0  :   0.000E+00     y0  :   8.882E-16
|
|--------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          65.000              Fu:          65.000
|
|--------------------------------------------------------------------------
|
| Actual Member Length:       500.040
|
| Design Parameters                                         (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|--------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:   500.04)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       3.65       N/A      11.83     Table.4.1a.Case1
|
| Web   : NonSlender       9.19       N/A      31.47     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   500.04)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          3.65       8.03     21.12     Table.4.1b.Case10
|
| Web   : Compact          9.19      79.42    120.40     Table.4.1b.Case15
|
|--------------------------------------------------------------------------
|
    STAAD SPACE                                             -- PAGE NO.
5
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|--------------------------------------------------------------------------
|
```

```
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       3276.       0.000     Cl.D2          2      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap          : Pn      =  3640.0     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2730.       0.000     Cl.D2          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective area             : Ae      =  56.000     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap          : Pn      =  3640.0     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1620.       0.000     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness      : Lcx/rx =  86.066                Cl.E2
|
|  Elastic Buckling Stress    : Fex     =  38.639     ksi        Eq.E3-4
|
|  Crit. Buckling Stress      : Fcrx    =  32.146     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling     : Pnx     =  1800.2     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
```

```
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       532.2       0.000     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  154.17              Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  12.042     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  10.560     ksi        Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  591.39     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2911.       0.000     Cl.E4          2      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  230.56     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  57.765     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  3234.9     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       1551.       0.000     Cl.G1          2      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000              Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  1723.6     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              78.85       595.3       0.132     Cl.G1          2      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  595.30     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                              -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -.1643E+05  0.1819E+05   0.903     Cl.F2.1        2    208.35
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  20215.     kip-in     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|             0.000       8366.       0.000     Cl.F6.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  9295.0     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -.1643E+05  0.1819E+05   0.903     Cl.F2.2        2    208.35
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  20215.     kip-in     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.3868                Eq.F1-1
|
|  Limiting Unbraced Length  : LpX    =  120.56     in         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    =  3.7768     in         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  812.00     in         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.903      Eq.H1-1b           2      208.35
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
```

```
|  Axial Capacity            : Pc     =  2730.0     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  18194.     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  8365.5     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------
|
```

V.AISC 360-16 W Local Buckling F.3A

Verify the available flexural local buckling capacity of a wide-flange section that is continuously braced with major axis bending per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-22 – F-23, Example F.3A

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a W21x48. The member is used as a 40 ft simple span. The member is continuously braced. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

Flange Slenderness

$$\lambda = b_f / 2t_f = 9.47 \text{ (from Table 1-1 of Reference 1)}$$

$$\lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29,000}{50}} = 9.15$$

$$\lambda_r = 1.0\sqrt{\frac{E}{F_y}} = 1.0\sqrt{\frac{29,000}{50}} = 24.1$$

$\lambda_p < \lambda < \lambda_r$, therefore, the flange is non-compact .

Flange Local Buckling

$$M_p = F_y Z_x = 50 \times 107 = 5{,}350 \text{ in}\cdot\text{k} = 445.8 \text{ ft}\cdot\text{k}$$

For non-compact sections:

$$M_n = M_p - \left(M_p - 0.7F_y S_x\right)\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right) \le M_p \quad \text{[Eqn. F3-1]}$$

$$M_n = 5,350 - \left[5,350 - 0.7(50)(93.0)\right]\left(\frac{9.47 - 9.15}{24.1 - 9.15}\right) = 5,303 \text{ in}\cdot\text{k} = 441.9 \text{ ft}\cdot\text{k}$$

Since the member is continuously braced, lateral torsional buckling does not control.

LRFD

$$\phi_b M_n = 397.7 \text{ ft}\cdot\text{k}$$

ASD

$$M_n/\Omega_b = 264.6 \text{ ft}\cdot\text{k}$$

Results

Table 724: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k)	397.7	398.0	negligible	
M_n/Ω_b (ft·k)	264.6	264.8	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W Local Buckling F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.05
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -18 13.333
1 CON GY -18 26.667
```

```
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD LRFD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD ASD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ========================
```

```
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.    
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W21X48             (AISC 
SECTIONS)|
|  Status:        PASS       Ratio:         0.995       Loadcase:       3     
|
|  Location:     20.00       Ref:      Cl.F3.1                                
|
|  Pz:       0.000     T     Vy:       -.1431E-06       Vx:      0.000        
|
|  Tz:       0.000           My:        0.000           Mx:     -396.0        
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS                                                         
|
| Actual Slenderness Ratio    :    289.731                                    
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00        
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS                                                             
|
| Critical L/C  :      3             Ratio     :        0.995(PASS)          
|
|         Loc  :   20.00           Condition :    Cl.F3.1                    
|
```

```
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    20.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.410E+01     Axx :   7.000E+00     Ayy :   7.210E+00
|
| Ixx :   9.590E+02     Iyy :   3.870E+01     J   :   8.030E-01
|
| Sxx+:   9.311E+01     Sxx-:   9.311E+01     Zxx :   1.070E+02
|
| Syy+:   9.509E+00     Syy-:   9.509E+00     Zyy :   1.490E+01
|
| Cw  :   3.931E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        40.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   480.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       9.47       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         53.54       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   480.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonCompact       9.47       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         53.54      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
```

```
                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       634.5       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  705.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       687.4       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.97917E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  916.50     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       452.7       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  58.202                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12167.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5620.4     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  502.95     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
```

```
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       37.95       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  289.73               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  490.99     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  430.60     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  42.162     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       154.8       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  2003.0     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  1756.6     kip/ft2    Eq.E3-3
|
|  Nom. Flex-tor Buckling    : Pn     =  172.00     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                                -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       189.0       0.000     Cl.G1          3      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000              Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  210.01     kip      Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              30.00       216.3       0.139     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  216.30     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                            -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       55.88       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  62.083     kip-ft   Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|             -396.0       398.0       0.995     Cl.F3.1        3     20.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  442.17     kip-ft     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       55.23       0.000     Cl.F6.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  61.362     kip-ft     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                            -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.995      Eq.H1-1b           3       20.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  634.50     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  397.96     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  55.226     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 1
```

```
    51. CODE AISC UNIFIED 2016
    52. FLX 2 ALL
    53. METHOD ASD
    54. SGR 28 ALL
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 **WARNING-DUPLICATE PARAMETER BLOCK NUMBER.
 PLEASE PROVIDE VALID PARAMETER BLOCK NUMBER.
 PARAMETER 1
      STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
```

```
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W21X48              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.944       Loadcase:        4
|
|  Location:     20.00       Ref:      Cl.F3.1
|
|  Pz:       0.000     T     Vy:        -.1192E-06       Vx:      0.000
|
|  Tz:       0.000           My:         0.000           Mx:     -250.0
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    289.731
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.944(PASS)
|
|          Loc  :   20.00            Condition :    Cl.F3.1
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    20.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.410E+01     Axx :   7.000E+00     Ayy :   7.210E+00
|
| Ixx :   9.590E+02     Iyy :   3.870E+01     J   :   8.030E-01
|
| Sxx+:   9.311E+01     Sxx-:   9.311E+01     Zxx :   1.070E+02
|
| Syy+:   9.509E+00     Syy-:   9.509E+00     Zyy :   1.490E+01
|
| Cw  :   3.931E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|------------------------------------------------------------------------------
|
```

```
|| MATERIAL PROPERTIES
||
|| Fyld:        7200.000              Fu:        9359.999
||
||--------------------------------------------------------------------------------
||
|| Actual Member Length:        40.000
||
|| Design Parameters                                                   (Rolled)
||
|| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
||
||--------------------------------------------------------------------------------
||
|| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   480.00)
||
||                            λ          λp         λr        CASE
||
|| Flange: NonSlender       9.47       N/A      13.49     Table.4.1a.Case1
||
|| Web   : Slender         53.54       N/A      35.88     Table.4.1a.Case5
||
||
||
|| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   480.00)
||
||                            λ          λp         λr        CASE
||
|| Flange: NonCompact       9.47       9.15     24.08     Table.4.1b.Case10
||
|| Web   : Compact         53.54      90.55    137.27     Table.4.1b.Case15
||
||--------------------------------------------------------------------------------
||
     STAAD SPACE                                                   -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
||--------------------------------------------------------------------------------
||
||                          CHECKS FOR AXIAL TENSION
||
||--------------------------------------------------------------------------------
||
|| TENSILE YIELDING
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             0.000       422.2       0.000     Cl.D2          4      0.00
||
||
||
|| Intermediate Results :
||
||  Nom. Ten. Yld Cap         : Pn     =  705.00     kip       Eq.D2-1
||
```

```
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       458.3       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.97917E-01 ft2       Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  916.50     kip       Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       301.2       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  58.202               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12167.     kip/ft2   Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5620.4     kip/ft2   Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  502.95     kip       Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       25.25       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  289.73               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  490.99     kip/ft2   Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  430.60     kip/ft2   Eq.E3-3
|
```

```
|  Nom. Flexural Buckling    : Pny    =  42.162     kip         Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       103.0       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  2003.0     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  1756.6     kip/ft2    Eq.E3-3
|
|  Nom. Flex-tor Buckling    : Pn     =  172.00     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
15
                   STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                 *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       125.8       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  210.01     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              19.00       144.2       0.132     Cl.G1          4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  216.30     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                            -- PAGE NO.
16
                   STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                 ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       37.18       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  62.083     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -250.0       264.8       0.944     Cl.F3.1        4     20.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  442.17     kip-ft     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       36.74       0.000     Cl.F6.2        4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  61.362     kip-ft     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
17
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.944      Eq.H1-1b           4       20.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                 Eq.H1-2
|
|  Axial Capacity            : Pc     =  422.16     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  264.78     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  36.744     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 W LTB Test F.1-2B

Verify the available lateral-torsional buckling capacity of a wide-flange section which is braced at third points with major axis bending per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-10 – F-11, Example F.1-2B

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a W18x50. The member is used as a 35 ft simple span subject to uniform loads. The member is braced at ends and at third points. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

<u>Nonuniform Moment Modification</u>

From Table 3-1 of Reference 1:

$C_b = 1.01$ for center segment
$C_b = 1.45$ for end segments

By inspection, the center segment will govern.

$L_b = 35 / 5 = 11.7$ ft

The limiting lengths for lateral-torsional buckling are:

$$L_p = 1.75 r_y \sqrt{\frac{E}{F_y}} = 1.75(1.65)\sqrt{\frac{29,000}{50}} = 69.9\text{in} = 5.83\text{ft} \quad \text{[Eqn. F2-5]}$$

$$L_r = 1.95 r_{ts} \frac{E}{0.7F_y}\sqrt{\frac{Jc}{S_x h_o} + \sqrt{\left(\frac{Jc}{S_x h_o}\right)^2 + 6.76\left(\frac{E}{0.7F_y}\right)^2}} \quad \text{[Eqn. F2-6]}$$

where

c = 1 for a doubly symmetric I-shape (Eqn. F2-8a)

$$L_r = 1.95(1.98)\frac{29,000}{0.7(50)}\sqrt{\frac{1.24(1.0)}{88.9(17.4)} + \sqrt{\left(\frac{1.24(1.0)}{88.9(17.4)}\right)^2 + 6.76\left(\frac{29,000}{0.7(50)}\right)^2}} = 203\text{in} = 16.9\text{ft}$$

$M_p = F_y Z_x = 50 \times 101 = 5{,}050$ in·k

$$M_n = C_b\left[M_p - \left(M_p - 0.7F_y S_x\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le M_p \quad \text{[Eqn. F3-1]}$$

$$M_n = 1.01\left\{5,050 - \left[5,050 - 0.7(50)(88.9)\right]\left(\frac{11.7 - 5.83}{16.9 - 5.83}\right)\right\} = 4,062\ \text{in}\cdot\text{k} = 338.5\ \text{ft}\cdot\text{k}$$

LRFD

$\phi_b M_n = 304.7$ ft·k

ASD

$M_n/\Omega_b = 202.7$ ft·k

Results

Table 726: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k)	304.7	305.5	negligible	
M_n / Ω_b (ft·k)	202.7	203.3	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W LTB Test F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
SGR 28 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
```

```
CODE AISC UNIFIED 2016
METHOD ASD
SGR 28 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
```

```
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  W18X50             (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.872       Loadcase:       3
 |
 |  Location:     17.50       Ref:      Cl.F2.2
 |
 |  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:     -266.4
 |
 |-----------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :    254.294
 |
 | Allowable Slenderness Ratio :    300.000            LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      3             Ratio     :        0.872(PASS)
 |
 |          Loc  :   17.50            Condition :    Cl.F2.2
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
 |
 | Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
 |
 | Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
 |
 | Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
 |
 | Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
 |
 | Cw  :   3.044E+03     x0  :   0.000E+00     y0  :   0.000E+00
```

```
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   420.00)
|
|                            λ          λp         λr        CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         45.23       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   420.00)
|
|                            λ          λp         λr        CASE
|
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         45.23      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       661.5       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
```

```
|  Nom. Ten. Yld Cap         : Pn     =  735.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       716.6       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.10208     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  955.50     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       506.3       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  56.933                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12716.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5680.7     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  562.50     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       51.36       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  254.29                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  637.37     kip/ft2    Eq.E3-4
```

```
|
|  Crit. Buckling Stress     : Fcry   =  558.97     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  57.061     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       478.8       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  9995.5     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  5325.9     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  531.99     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       230.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  256.50     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|              30.45       191.7       0.159     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  191.70     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
10
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                              - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                             CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -266.4       378.8       0.703     Cl.F2.1        3     17.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  420.83     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       62.25       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  69.167     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|             -266.4       305.5       0.872     Cl.F2.2        3     17.50
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  339.47     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0100                Custom
|
|  Limiting Unbraced Length  : LpX    =  5.8339     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    = 0.16521     ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  16.964     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.872      Eq.H1-1b           3       17.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  661.50     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  305.53     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  62.250     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    51. LOAD LIST 4
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2016
    54. METHOD ASD
    55. SGR 28 ALL
```

```
    56. CB 1.01 ALL
    57. LX 11.67 ALL
    58. UNB 11.67 ALL
    59. UNT 11.67 ALL
    60. TRACK 2 ALL
    61. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
```

```
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  W18X50              (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.904       Loadcase:        4
 |
 |  Location:     17.50       Ref:      Cl.F2.2
 |
 |  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:     -183.8
 |
 |-----------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :    254.294
 |
 | Allowable Slenderness Ratio :    300.000           LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      4             Ratio     :        0.904(PASS)
 |
 |          Loc  :   17.50            Condition :    Cl.F2.2
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:    17.50, PROPERTIES UNIT: IN  )
 |
 | Ag  :   1.470E+01     Axx :   8.550E+00     Ayy :   6.390E+00
 |
 | Ixx :   8.000E+02     Iyy :   4.010E+01     J   :   1.240E+00
 |
 | Sxx+:   8.889E+01     Sxx-:   8.889E+01     Zxx :   1.010E+02
 |
 | Syy+:   1.069E+01     Syy-:   1.069E+01     Zyy :   1.660E+01
 |
 | Cw  :   3.044E+03     x0  :   0.000E+00     y0  :   0.000E+00
 |
 |-----------------------------------------------------------------------------
 |
 | MATERIAL PROPERTIES
 |
```

```
| Fyld:        7200.000              Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        35.000
|
| Design Parameters                                           (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   420.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       6.58       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         45.23       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   420.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          6.58       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         45.23      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       440.1       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  735.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
```

```
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       477.7       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.10208     ft2         Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  955.50     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       336.8       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  56.933                 Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12716.     kip/ft2     Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5680.7     kip/ft2     Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  562.50     kip         Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       34.17       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  254.29                 Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  637.37     kip/ft2     Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  558.97     kip/ft2     Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  57.061     kip         Eq.E7-1
|
```

```
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       318.6       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  9995.5     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  5325.9     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  531.99     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
15
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       153.6       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  256.50     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             21.00       127.8       0.164     Cl.G1          4      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  191.70     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
16
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -183.8       252.0       0.729     Cl.F2.1        4     17.50
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  420.83     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       41.42       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  69.167     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -183.8       203.3       0.904     Cl.F2.2        4     17.50
|
|
|
```

```
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  339.47     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0100                Custom
|
|  Limiting Unbraced Length  : LpX    =  5.8339     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    = 0.16521     ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  16.964     ft         Eq.F2-6
|
|------------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
17
                                  STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                              ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                                 - Member :     1 Contd.
|------------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|------------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.904      Eq.H1-1b           4       17.50
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  440.12     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  203.28     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  41.417     kip-ft     Cl.H1.1
|
|------------------------------------------------------------------------------
|
```

V.AISC 360-16 W Member Selection F.4

Verify the selection of an optimized wide flange section for major axis bending per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017

2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pg F-26 – F-27, Example F.4

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Select a wide flange section based on moment of inertia required to limit deflection. The member is used as a 30 ft simple span which caries a uniform dead load of 0.8 kip/ft and a uniform live load of 2 kip/ft. The member is continuously braced. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

Ultimate load (for LRFD):

$$w_u = 1.2(0.8) + 1.6(2.0) = 4.16 \text{ kip/ft}$$

$$M_u = w_u L^2/8 = (4.16)(30)^2/8 = 468 \text{ kip}\cdot\text{ft}$$

Service level load (for ASD):

$$w_a = 0.8 + 2.0 = 2.8 \text{ kip/ft}$$

$$M_a = w_a L^2/8 = (2.8)(30)^2/8 = 315 \text{ kip}\cdot\text{ft}$$

Minimum Moment of Inertia

Assume a live load deflection limit of $L/360 = 30\times12/360 = 1.0\ in$

$$\Delta_{max} = \frac{5 w_L L^4}{384 EI}$$

$$I_{min} = \frac{5 w_L L^4}{384 E \Delta_{max}} = \frac{5(2.0)(30)^4(12)^3}{384(29,000)(1.0)} = 1,260 \text{ in}^4$$

Try a W24x55 ($I_x = 1,350\ in^4$)

The W24x55 is a compact section and the member is continuously braced.

$$M_n = M_p = F_y Z_x = 50\times134 = 6,700 \text{ in}\cdot\text{k}$$

LRFD

$$\phi_b M_n = 502.5 \text{ ft}\cdot\text{k} > 468 \text{ ft}\cdot\text{k (OK)}$$

ASD

$$M_n/\Omega_b = 334.3 \text{ ft}\cdot\text{k} > 315 \text{ ft}\cdot\text{k (OK)}$$

Results

Table 727: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Selected section, LRFD	W24x55	W24x55	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Selected section, ASD	W24x55	W25x55	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W Memb Selection F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X192
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.8
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
DEFINE ENVELOPE
3 ENVELOPE 1 TYPE STRENGTH
4 ENVELOPE 2 TYPE SERVICEABILITY
END DEFINE ENVELOPE
LOAD LIST ENV 1 2
PARAMETER 1
CODE AISC UNIFIED 2016
BEAM 1 ALL
```

```
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
SGR 28 ALL
TRACK 2 ALL
FLX 2 ALL
PROFILE W24 ALL
METHOD LRFD
SELECT OPTIMIZED
LOAD LIST ENV  2
PARAMETER 2
CODE AISC UNIFIED 2016
BEAM 1 ALL
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
SGR 28 ALL
TRACK 2 ALL
FLX 2 ALL
PROFILE W24 ALL
METHOD ASD
SELECT OPTIMIZED
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ========================
```

```
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X192             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.223       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:       0.1144E-05       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -468.0
|
|-----------------------------------------------------------------------------
|
|  Status:        PASS       Ratio:         0.223       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    117.541
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
```

```
| STRENGTH CHECKS
|
| Critical L/C  :      3            Ratio     :        0.223(PASS)
|
|         Loc   :   15.00           Condition :    Cl.F2.1
|
|-----------------------------------------------------------------------------
|
| SERVICEABILITY CHECKS                                                      |
| Critical L/C  :      4            Ratio     :        0.186(PASS)
|
|         Loc   :   15.00           Condition :    Deflection
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    15.00, PROPERTIES UNIT: IN  )
|
| Ag  :   5.650E+01     Axx :   3.796E+01     Ayy :   2.066E+01
|
| Ixx :   6.260E+03     Iyy :   5.300E+02     J   :   3.080E+01
|
| Sxx+:   4.910E+02     Sxx-:   4.910E+02     Zxx :   5.590E+02
|
| Syy+:   8.154E+01     Syy-:   8.154E+01     Zyy :   1.260E+02
|
| Cw  :   7.724E+04     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                              (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                           λ          λp         λr       CASE
|
| Flange: NonSlender       4.45        N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      26.64        N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                           λ          λp         λr       CASE
|
| Flange: Compact          4.45       9.15      24.08     Table.4.1b.Case10
```

```
|
| Web   : Compact         26.64      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
5
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2542.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  2825.0     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2754.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.39236     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  3672.5     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2334.       0.000     Cl.E3          3      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  34.201                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  35236.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6609.8     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  2593.4     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       923.9       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  117.54                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  2983.2     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  2616.3     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  1026.5     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1930.       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  10933.     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  5465.5     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  2144.4     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
```

```
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1025.       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000              Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000              Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  1138.8     kip      Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             62.40       619.7       0.101     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  619.65     kip      Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -468.0       2096.       0.223     Cl.F2.1        3     15.00
```

```
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  2329.2     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       472.5       0.000     Cl.F6.1        3      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  525.00     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                                -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.223      Eq.H1-1b           3       15.00
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  2542.5     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  2096.3     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  472.50     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR DEFLECTION
```

```
|
|--------------------------------------------------------------------------------
|
|                            RATIO           L/C          LOC
|
|                            0.186            4           15.00
|
| Actual Deflection    : 0.27953      in.    Actual Span/Defl. :  1287.9
|
| Allowable Deflection :  1.5000      in.    Allow Span/Defl.  :  240.00
|
|--------------------------------------------------------------------------------
|
 PARAMETER 1
      STAAD SPACE                                                        -- PAGE NO.
9
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
```

```
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X55              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.931       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:       0.1144E-05       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -468.0
|
|-------------------------------------------------------------------------------
|
|  Status:        PASS       Ratio:         0.931       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|-------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    268.605
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.931(PASS)
|
|          Loc  :   15.00            Condition :    Cl.F2.1
|
|-------------------------------------------------------------------------------
|
| SERVICEABILITY CHECKS                                                        |
| Critical L/C  :      4             Ratio     :        0.864(PASS)
|
```

```
|          Loc  :   15.00            Condition :    Deflection
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    15.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.620E+01     Axx :   7.080E+00     Ayy :   9.322E+00
|
| Ixx :   1.350E+03     Iyy :   2.910E+01     J   :   1.180E+00
|
| Sxx+:   1.144E+02     Sxx-:   1.144E+02     Zxx :   1.340E+02
|
| Syy+:   8.302E+00     Syy-:   8.302E+00     Zyy :   1.330E+01
|
| Cw  :   3.866E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       6.94       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         54.63       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          6.94       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         54.63      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
```

```
|--------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|--------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       729.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  810.00     kip        Eq.D2-1
|
|--------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       789.8       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.11250     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  1053.0     kip        Eq.D2-2
|
|--------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|--------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       568.6       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  39.436                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  26502.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6426.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  631.83     kip        Eq.E7-1
|
```

```
|--------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       50.73       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  268.60               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  571.26     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  500.99     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  56.362     kip        Eq.E7-1
|
|--------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       201.7       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  2271.5     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  1992.1     kip/ft2    Eq.E3-3
|
|  Nom. Flex-tor Buckling    : Pn     =  224.11     kip        Eq.E7-1
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                           -- PAGE NO.
12
                          STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                      *******************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                    - Member :     1 Contd.
|--------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|--------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       191.2       0.000     Cl.G1          3      0.00
|
```

```
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  212.40     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             62.40       251.7       0.248     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  279.66     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
13
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -468.0       502.5       0.931     Cl.F2.1        3     15.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  558.33     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       49.81       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  55.350     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
14
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.931      Eq.H1-1b           3       15.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                 Eq.H1-2
|
|  Axial Capacity            : Pc     =  729.00     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  502.50     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  49.815     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR DEFLECTION
|
|-----------------------------------------------------------------------------
|
|                           RATIO          L/C          LOC
|
|                           0.864           4           15.00
|
| Actual Deflection    :  1.2962      in.    Actual Span/Defl. :  277.73
|
| Allowable Deflection :  1.5000      in.    Allow Span/Defl.  :  240.00
|
```

```
|---------------------------------------------------------------------------------
|
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 PARAMETER 1
      STAAD SPACE                                              -- PAGE NO.
15
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
```

```
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
16
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X55              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.931       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:       0.1144E-05       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -468.0
|
|------------------------------------------------------------------------------
|
|  Status:        PASS       Ratio:         0.931       Loadcase:        3
|
|  Location:     15.00       Ref:      Cl.F2.1
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    268.605
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3              Ratio     :        0.931(PASS)
|
|          Loc  :   15.00             Condition :    Cl.F2.1
|
|------------------------------------------------------------------------------
|
| SERVICEABILITY CHECKS                                                        |
| Critical L/C  :      4              Ratio     :        0.864(PASS)
|
|          Loc  :   15.00             Condition :    Deflection
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    15.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.620E+01     Axx :   7.080E+00     Ayy :   9.322E+00
|
| Ixx :   1.350E+03     Iyy :   2.910E+01     J   :   1.180E+00
|
```

```
| Sxx+:   1.144E+02     Sxx-:   1.144E+02     Zxx :   1.340E+02
|
| Syy+:   8.302E+00     Syy-:   8.302E+00     Zyy :   1.330E+01
|
| Cw  :   3.866E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                           λ          λp         λr       CASE
|
| Flange: NonSlender       6.94       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         54.63       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                           λ          λp         λr       CASE
|
| Flange: Compact          6.94       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         54.63      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
17
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     *******************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
             0.000       729.0       0.000     Cl.D2          3      0.00


 Intermediate Results :
  Nom. Ten. Yld Cap         : Pn     =  810.00     kip        Eq.D2-1
-----------------------------------------------------------------------------
 TENSILE RUPTURE
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       789.8       0.000     Cl.D2          3      0.00


 Intermediate Results :
  Effective area            : Ae     = 0.11250     ft2        Eq.D3-1
  Nom. Ten. Rpt Cap         : Pn     =  1053.0     kip        Eq.D2-2
-----------------------------------------------------------------------------
                        CHECKS FOR AXIAL COMPRESSION
-----------------------------------------------------------------------------
 FLEXURAL BUCKLING X
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       568.6       0.000     Cl.E3          3      0.00


 Intermediate Results :
  Effective Slenderness     : Lcx/rx =  39.436                Cl.E2
  Elastic Buckling Stress   : Fex    =  26502.     kip/ft2    Eq.E3-4
  Crit. Buckling Stress     : Fcrx   =  6426.1     kip/ft2    Eq.E3-2
  Nom. Flexural Buckling    : Pnx    =  631.83     kip        Eq.E7-1
-----------------------------------------------------------------------------
 FLEXURAL BUCKLING Y
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
             0.000       50.73       0.000     Cl.E3          3      0.00
```

```
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  268.60               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  571.26     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  500.99     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  56.362     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       201.7       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  2271.5     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  1992.1     kip/ft2    Eq.E3-3
|
|  Nom. Flex-tor Buckling    : Pn     =  224.11     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
18
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       191.2       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  212.40     kip        Eq.G6-1
|
```

```
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              62.40       251.7       0.248     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  279.66     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
19
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -468.0       502.5       0.931     Cl.F2.1        3     15.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  558.33     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       49.81       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  55.350     kip-ft     Eq.F6-1
|
```

```
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                 -- PAGE NO.
20
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.931      Eq.H1-1b           3       15.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000               Eq.H1-2
|
|  Axial Capacity            : Pc     =  729.00     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  502.50     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  49.815     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR DEFLECTION
|
|-----------------------------------------------------------------------------
|
|                           RATIO          L/C          LOC
|
|                           0.864           4           15.00
|
| Actual Deflection    :  1.2962      in.    Actual Span/Defl. :  277.73
|
| Allowable Deflection :  1.5000      in.    Allow Span/Defl.  :  240.00
|
|-----------------------------------------------------------------------------
|
    56. LOAD LIST ENV  2
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2016
    59. BEAM 1 ALL
    60. DFF 240 ALL
    61. DJ1 1 ALL
    62. DJ2 2 ALL
    63. SGR 28 ALL
```

```
    64. TRACK 2 ALL
    65. FLX 2 ALL
    66. PROFILE W24 ALL
    67. METHOD ASD
    68. SELECT OPTIMIZED
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
21
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
```

```
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
22
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X55              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.864       Loadcase:        4
|
|  Location:     15.00       Ref:      Deflection
|
|-----------------------------------------------------------------------------
|
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR DEFLECTION
|
|-----------------------------------------------------------------------------
|
|                             RATIO           L/C          LOC
|
|                             0.864            4           15.00
|
| Actual Deflection    :  1.2962      in.    Actual Span/Defl. :  277.73
|
| Allowable Deflection :  1.5000      in.    Allow Span/Defl.  :  240.00
|
|-----------------------------------------------------------------------------
|
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
23
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 ASD (V1.2)
                     *******************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
```

```
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                -- PAGE NO.
24
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 ASD (V1.2)
                     *******************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X55              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.864       Loadcase:        4
|
|  Location:     15.00       Ref:      Deflection
|
|------------------------------------------------------------------------------
|
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
25
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 ASD (V1.2)
```

```
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
26
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 ASD (V1.2)
                     ********************************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X55              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.864       Loadcase:        4
|
|  Location:     15.00       Ref:      Deflection
|
|-----------------------------------------------------------------------------
|
    69. FINISH

*****************************************************************************
   **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
               IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
               ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
               TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
               FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
               FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
               COMMANDS.

*****************************************************************************
```

V.AISC 360-16 WT Shape F.10

Verify the available flexural strength of a WT (“split tee”) member per the AISC 60-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.F-45 – F-47. Example F.10

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the bending capacity of a WT5x6. The member is used as a 6.0 ft simple span. The member is continuously braced. The material is ASTM A992 ($F_y = 50\ ksi$).

Validation

The member is continuously braced, so lateral-torsional buckling does not control.

Yielding

$$M_n = F_y Z_x \leq 1.6 M_y \qquad \text{[Eqn. F9-2]}$$

where

$$M_y = F_y S_x = 50 \times 1.22 = 61\ in \cdot k$$

$$M_n = 50 \times 2.20 = 110\ in \cdot k > 1.6(61.0) = 97.6\ in \cdot k$$

Flange Local Buckling

$\lambda = b/2t_f = 9.43$ (from Table 1-8 of Reference 1)

$$\lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29{,}000}{50}} = 9.15$$

$$\lambda_p = 1.0\sqrt{\frac{E}{F_y}} = 1.0\sqrt{\frac{29{,}000}{50}} = 24.1$$

$\lambda_p < \lambda < \lambda_r$, therefore, the flange is non-compact.

For non-compact sections:

$$M_n = M_p - \left(M_p - 0.7F_yS_x\right)\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right) \leq 1.6M_y \quad \text{[Eqn. F9-14]}$$

$$M_n = 110 - \left[110 - 0.7(50)(3.20)\right]\left(\frac{9.43 - 9.15}{24.1 - 9.15}\right) = 110 \text{ in}\cdot\text{k} > 96.7 \text{ in}\cdot\text{k}$$

$$M_n = 97.6 \text{ in}\cdot\text{k} = 8.13 \text{ ft}\cdot\text{k}$$

LRFD

$$\phi_b M_n = 7.32 \text{ ft}\cdot\text{k}$$

ASD

$$M_n/\Omega_b = 4.87 \text{ ft}\cdot\text{k}$$

Results

Table 728: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k)	7.32	7.21	1.5%	
M_n/Ω_b (ft·k)	4.87	4.797	1.5%	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 WT Shape F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
```

```
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W10X12
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.08
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.24
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD LRFD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
FLX 2 ALL
METHOD ASD
SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
```

```
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  T   W10X12             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.300      Loadcase:        3
|
|  Location:      3.00       Ref:      Cl.F9.1
|
|  Pz:       0.000     T     Vy:       -.7749E-07      Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:     -2.160
```

```
|-------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     91.750
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.300(PASS)
|
|          Loc  :    3.00            Condition :    Cl.F9.1
|
|-------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     3.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.770E+00     Axx :   8.316E-01     Ayy :   9.376E-01
|
| Ixx :   4.303E+00     Iyy :   1.090E+00     J   :   2.735E-02
|
| Sxx+:   3.178E+00     Sxx-:   1.202E+00     Zxx :   2.202E+00
|
| Syy+:   5.505E-01     Syy-:   5.505E-01     Zyy :   8.700E-01
|
| Cw  :   2.546E-02     x0  :   0.000E+00     y0  :   1.249E+00
|
|-------------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000              Fu:        9359.999
|
|-------------------------------------------------------------------------------
|
| Actual Member Length:         6.000
|
| Design Parameters                                           (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-------------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    72.00)
|
|                            λ          λp         λr       CASE
|
| Flange: NonSlender       9.43       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         25.97       N/A      18.06     Table.4.1a.Case4
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    72.00)
```

```
|
|                          λ          λp         λr        CASE
|
| Flange: NonCompact       9.43       9.15      24.08     Table.4.1b.Case10
|
| Web   : NonCompact      25.97      20.23      36.61     Table.4.1b.Case14
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                               - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       79.65       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  88.500     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       86.29       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.12292E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  115.05     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|            0.000       62.46       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  46.178               Cl.E2
|  Elastic Buckling Stress   : Fex    =  19328.     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  6160.5     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  69.400     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|            0.000       42.37       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  91.750               Cl.E2
|  Elastic Buckling Stress   : Fey    =  4896.1     kip/ft2    Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  3890.6     kip/ft2    Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  47.083     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
| FLEXURAL-TORSIONAL-BUCKLING
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|            0.000       31.48       0.000     Cl.E4          3      0.00
|
|
| Intermediate Results :
|  Constant                  : H      = 0.66133               Eq.E4-8
|  Euler Buckling Stress     : Fez    =  5409.2     kip/ft2    Eq.E4-7
|  Elastic F-T-B Stress      : Fe     =  3246.2     kip/ft2    Eq.E4-3
|  Crit. F-T-B Stress        : Fcr    =  2845.5     kip/ft2    Eq.E3-2
|  Nom. Flex-tor Buckling    : Pn     =  34.976     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
```

```
 |
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       22.45       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  24.948     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             1.440       25.32       0.057     Cl.G1          3      6.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  1.2000               Cl.G3
|
|  Nom. Shear Along Y        : Vny    =  28.129     kip       Eq.G3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
```

```
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -2.160       7.210       0.300     Cl.F9.1        3      3.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  8.0109     kip-ft     Eq.F9-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       3.263       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  3.6250     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -2.160       7.210       0.300     Cl.F9.3        3      3.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  8.0109     kip-ft     Eq.F9-14
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       3.229       0.000     Cl.F6.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  3.5875     kip-ft     Eq.F6-2
|
|-----------------------------------------------------------------------------
```

```
|
| WEB LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.4441E-16   3.941       0.000     Cl.F9.4        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom W-L-B Cap             : Mnx    =  4.3788     kip-ft     Eq.F9-16
|
|  Critical Com. W-B Stress  : FcrX   =  6296.9     kip/ft2    Eq.F9-18
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.300      Eq.H1-1b           3        3.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  79.650     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  7.2098     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  3.2288     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    46. LOAD LIST 4
    47. PARAMETER 2
    48. CODE AISC UNIFIED 2016
    49. FLX 2 ALL
    50. METHOD ASD
    51. SGR 28 ALL
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 PARAMETER 2
     STAAD SPACE                                              -- PAGE NO.
9
```

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
```

```
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  T   W10X12             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.300       Loadcase:        4
|
|  Location:      3.00       Ref:      Cl.F9.1
|
|  Pz:       0.000     T     Vy:       -.4470E-07       Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -1.440
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     91.750
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.300(PASS)
|
|          Loc  :    3.00            Condition :    Cl.F9.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     3.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.770E+00     Axx :   8.316E-01     Ayy :   9.376E-01
|
| Ixx :   4.303E+00     Iyy :   1.090E+00     J   :   2.735E-02
|
| Sxx+:   3.178E+00     Sxx-:   1.202E+00     Zxx :   2.202E+00
|
| Syy+:   5.505E-01     Syy-:   5.505E-01     Zyy :   8.700E-01
|
| Cw  :   2.546E-02     x0  :   0.000E+00     y0  :   1.249E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:         6.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
```

```
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      4 LOC:    72.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonSlender       9.43       N/A      13.49     Table.4.1a.Case1
 |
 | Web   : Slender         25.97       N/A      18.06     Table.4.1a.Case4
 |
 |
 |
 | FLEXURE CLASSIFICATION     (L/C:      4 LOC:    72.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonCompact       9.43       9.15     24.08     Table.4.1b.Case10
 |
 | Web   : NonCompact      25.97      20.23     36.61     Table.4.1b.Case14
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                         CHECKS FOR AXIAL TENSION
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE YIELDING
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              0.000       52.99       0.000     Cl.D2          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Ten. Yld Cap         : Pn     =  88.500     kip        Eq.D2-1
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE RUPTURE
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              0.000       57.52       0.000     Cl.D2          4      0.00
 |
 |
 |
 | Intermediate Results :
```

```
|
| Effective area          : Ae     = 0.12292E-01 ft2        Eq.D3-1
|
| Nom. Ten. Rpt Cap       : Pn     =  115.05     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       41.56       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  46.178               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  19328.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6160.5     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  69.400     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       28.19       0.000     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  91.750               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  4896.1     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  3890.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  47.083     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       20.94       0.000     Cl.E4          4      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Constant                  : H      = 0.66133                Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  5409.2     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  3246.2     kip/ft2    Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  2845.5     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  34.976     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       14.94       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  24.948     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.9600       16.84       0.057     Cl.G1          4      6.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  1.2000                Cl.G3
```

```
|
|  Nom. Shear Along Y        : Vny    =  28.129     kip        Eq.G3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
13
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -1.440       4.797       0.300     Cl.F9.1        4      3.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  8.0109     kip-ft     Eq.F9-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2.171       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  3.6250     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -1.440       4.797       0.300     Cl.F9.3        4      3.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  8.0109     kip-ft     Eq.F9-14
|
|-----------------------------------------------------------------------------
```

```
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2.148       0.000     Cl.F6.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  3.5875     kip-ft     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.3701E-16   2.622       0.000     Cl.F9.4        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom W-L-B Cap             : Mnx    =  4.3788     kip-ft     Eq.F9-16
|
|  Critical Com. W-B Stress  : FcrX   =  6296.9     kip/ft2    Eq.F9-18
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
14
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.300      Eq.H1-1b           4        3.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  52.994     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  4.7969     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  2.1482     kip-ft     Cl.H1.1
```

```
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 Built Up Column E.2

Verify the available compressive strength of a built-up column with a slender web per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-14 – E-18., Example E.2

Details

From reference 2:

> Verify that a built-up, ASTM A572 Grade 50 column with PL1 in.×8 in. flanges and a PL4 in.×15 in. web… is sufficient to carry a dead load of 70 kips and live load of 210 kips in axial compression. The column's unbraced length is 15 ft and the ends are pinned in both axes.

ASTM A572 Grade 50: $F_y = 50\ ksi$

Validation

Refer to reference.

Results

Table 729: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	500	497.9	negligible	
P_n / Ω_c (kips)	332	331.3	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Built Up Column E.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0;
```

```
MEMBER INCIDENCES
1 1 2;
START USER TABLE
*TABLE 1
*UNIT INCHES KIP
*ISECTION
*I1
*17 0.25 17 8 1 8 1 0 0 0
TABLE 2
UNIT INCHES KIP
WIDE FLANGE
I
19.75 17 0.25 8 1 1095.65 85.3529 5.41146 4.25 10.6667 8 1
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 UPTABLE 2 I
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -70
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -210
UNIT INCHES KIP
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
STP 2 ALL
SGR 15 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
STP 2 ALL
```

```
SGR 15 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
```

```
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD PLANE                                              -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  I                   (UPT)
|
|  Status:        PASS       Ratio:         0.843       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       420.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     86.586
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.843(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.975E+01     Axx :   1.067E+01     Ayy :   4.250E+00
|
| Ixx :   1.096E+03     Iyy :   8.535E+01     J   :   5.411E+00
|
| Sxx+:   1.289E+02     Sxx-:   1.289E+02     Zxx :   1.421E+02
|
| Syy+:   2.134E+01     Syy-:   2.134E+01     Zyy :   3.223E+01
|
| Cw  :   5.461E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:         50.000             Fu:         65.000
```

```
 |
 |-----------------------------------------------------------------------------
 |
 | Actual Member Length:       180.000
 |
 | Design Parameters                                               (Built-Up)
 |
 | Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      3 LOC:   180.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonSlender       4.00       N/A      11.08     Table.4.1a.Case2
 |
 | Web   : Slender         60.00       N/A      35.88     Table.4.1a.Case5
 |
 |
 |
 | FLEXURE CLASSIFICATION     (L/C:      3 LOC:   180.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: Compact          4.00       9.15     19.65     Table.4.1b.Case11
 |
 | Web   : Compact         60.00      90.55    137.27     Table.4.1b.Case15
 |
 |-----------------------------------------------------------------------------
 |
     STAAD PLANE                                                -- PAGE NO.
5
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                            CHECKS FOR AXIAL TENSION
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE YIELDING
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              0.000       888.8       0.000     Cl.D2          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Ten. Yld Cap         : Pn     =  987.50     kip        Eq.D2-1
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE RUPTURE
```

```
|             DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000        962.8       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area             : Ae     =  19.750     in2         Eq.D3-1
|  Nom. Ten. Rpt Cap          : Pn     =  1283.8     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|             DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             420.0        800.5       0.525     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness      : Lcx/rx =  24.167                 Cl.E2
|  Elastic Buckling Stress    : Fex    =  490.07     ksi         Eq.E3-4
|  Crit. Buckling Stress      : Fcrx   =  47.910     ksi         Eq.E3-2
|  Nom. Flexural Buckling     : Pnx    =  889.47     kip         Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|             DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             420.0        497.9       0.843     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness      : Lcy/ry =  86.586                 Cl.E2
|  Elastic Buckling Stress    : Fey    =  38.177     ksi         Eq.E3-4
|  Crit. Buckling Stress      : Fcry   =  28.900     ksi         Eq.E3-2
|  Nom. Flexural Buckling     : Pny    =  553.26     kip         Eq.E7-1
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             420.0       673.0       0.624     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  92.170     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  39.844     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  747.78     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
    STAAD PLANE                                             -- PAGE NO.
6
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       432.0       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  480.00     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       114.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Coefficient Cv Along Y    : Cv     =  1.0000              -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400              Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  127.50     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD PLANE                                          -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       6393.       0.000     Cl.F2.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  7103.1     kip-in     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       1451.       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1611.7     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       5403.       0.000     Cl.F2.2        3      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Nom L-T-B Cap             : Mnx    =  6003.6     kip-in     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  88.115     in         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    =  2.3015     in         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  304.70     in         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
      STAAD PLANE                                              -- PAGE NO.
8
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.843      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  497.94     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  5403.3     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  1450.5     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    62. LOAD LIST 4
    63. PARAMETER 2
    64. CODE AISC UNIFIED 2016
    65. METHOD ASD
    66. STP 2 ALL
    67. SGR 15 ALL
    68. TRACK 2 ALL
    69. CHECK CODE ALL
 PARAMETER 2
      STAAD PLANE                                              -- PAGE NO.
9
                          STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                        ****************************************************
```

```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD PLANE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
```

```
- Member :     1
|--------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  I                  (UPT)
|
|  Status:        PASS       Ratio:         0.845       Loadcase:        4
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       280.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|--------------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     86.586
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|--------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.845(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|--------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.975E+01     Axx :   1.067E+01     Ayy :   4.250E+00
|
| Ixx :   1.096E+03     Iyy :   8.535E+01     J   :   5.411E+00
|
| Sxx+:   1.289E+02     Sxx-:   1.289E+02     Zxx :   1.421E+02
|
| Syy+:   2.134E+01     Syy-:   2.134E+01     Zyy :   3.223E+01
|
| Cw  :   5.461E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|--------------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          65.000
|
|--------------------------------------------------------------------------------
|
| Actual Member Length:       180.000
|
| Design Parameters                                                (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|--------------------------------------------------------------------------------
```

```
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   180.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       4.00       N/A      11.08     Table.4.1a.Case2
|
| Web   : Slender         60.00       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   180.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          4.00       9.15     19.65     Table.4.1b.Case11
|
| Web   : Compact         60.00      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD PLANE                                                 -- PAGE NO.
11
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       591.3       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  987.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       641.9       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  19.750     in2        Eq.D3-1
```

```
|  Nom. Ten. Rpt Cap         : Pn     =  1283.8     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             280.0       532.6       0.526     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  24.167              Cl.E2
|  Elastic Buckling Stress   : Fex    =  490.07     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  47.910     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  889.47     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             280.0       331.3       0.845     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  86.586              Cl.E2
|  Elastic Buckling Stress   : Fey    =  38.177     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  28.900     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  553.26     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             280.0       447.8       0.625     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Elastic F-T-B Stress      : Fe     =  92.170     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  39.844     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  747.78     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD PLANE                                          -- PAGE NO.
12
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       287.4       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  480.00     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       76.35       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  127.50     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD PLANE                                          -- PAGE NO.
```

```
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                            - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       4253.       0.000     Cl.F2.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  7103.1     kip-in     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       965.1       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1611.7     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       3595.       0.000     Cl.F2.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap              : Mnx    =  6003.6     kip-in     Eq.F2-2
|
|  Mom. Distr. factor         : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length   : LpX    =  88.115     in         Eq.F2-5
|
|  coefficient C              : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.     : Rts    =  2.3015     in         Eq.F2-7
```

```
|
|  Limiting Unbraced Length  : LrX    =  304.70     in         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
      STAAD PLANE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.845      Eq.H1-1a           4        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  331.29     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  3595.0     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  965.10     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 Double L E.5

Verify the available compressive strength of a double angle compression member without slender elements per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-30 – E-35., Example E.5

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference 2:

Verify the strength of a 2L4×3-1/2×3/8 LLBB (3/4-in. separation) strut, ASTM A36, with a length of 8 ft and pinned ends carrying an axial dead load of 20 kips and live load of 60 kips. Also, calculate the required number of pretensioned bolted or welded intermediate connectors required.

ASTM A536: $F_y = 36\ ksi$

Validation

Refer to reference.

Results

Table 730: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	128	127.2	negligible	
P_n / Ω_c (kips)	85	84.62	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Double L E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 8 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L40356 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -60
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
CSPACING 32 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
CSPACING 32 ALL
SGR 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
```

```
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  LD  L40356             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:        0.943       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       120.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     76.914
```

```
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|--------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.943(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|--------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   5.360E+00     Axx :   2.625E+00     Ayy :   3.000E+00
|
| Ixx :   8.350E+00     Iyy :   1.541E+01     J   :   2.512E-01
|
| Sxx+:   6.950E+00     Sxx-:   2.984E+00     Zxx :   5.417E+00
|
| Syy+:   3.978E+00     Syy-:   3.978E+00     Zyy :   7.107E+00
|
| Cw  :   7.750E-01     x0  :   0.000E+00     y0  :  -1.014E+00
|
|--------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          36.000              Fu:          58.000
|
|--------------------------------------------------------------------------
|
| Actual Member Length:        96.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   32.00
|
|--------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    96.00)
|
|                            λ          λp         λr       CASE
|
| Flange: NonSlender       9.33        N/A      12.77     Table.4.1a.Case3
|
| Web   : NonSlender      10.67        N/A      12.77     Table.4.1a.Case3
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    96.00)
|
|                            λ          λp         λr       CASE
|
| Flange: Compact          9.33      15.33      25.83     Table.4.1b.Case12
|
| Web   : Compact         10.67      15.33      25.83     Table.4.1b.Case12
```

```
|
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                                 -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|--------------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       173.7       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  192.96     kip        Eq.D2-1
|
|--------------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       233.2       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  5.3600     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  310.88     kip        Eq.D2-2
|
|--------------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|--------------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             120.0       127.2       0.943     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
```

```
|
|  Effective Slenderness     : Lcx/rx =  76.914              Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  48.382     ksi      Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  26.366     ksi      Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  141.32     kip      Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             120.0       146.7       0.818     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  56.612              Eq.E6-2a
|
|  Elastic Buckling Stress   : Fey    =  89.304     ksi      Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  30.411     ksi      Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  163.00     kip      Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             120.0       137.5       0.873     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Constant                  : H      = 0.81177              Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  96.130     ksi      Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  64.518     ksi      Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  28.502     ksi      Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  152.77     kip      Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
```

```
                                                                 - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       51.03       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  56.700     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       58.32       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  64.800     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                                 - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|             0.000       145.0       0.000     Cl.F9.1        3      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  161.12     kip-in     Eq.F9-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       206.2       0.000     Cl.F6.1        3      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  229.10     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       136.9       0.000     Cl.F9.2        3      0.00
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  152.12     kip-in     Eq.F10-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  84.707     in         Eq.F9-8
|
|  coefficient B             : Bx     = -.75059                Eq.F9-12
|
|  Limiting Unbraced Length  : LrX    =  1105.8     in         Eq.F9-9
|
|  Crit. Elastic L-T-B Mom.  : McrX   =  579.32     kip-in     Eq.F9-10
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       185.7       0.000     Cl.F9.2        3      0.00
|
|
| Intermediate Results :
```

```
|
|  Nom Flex Yielding About Y : Mny    =  206.33     kip-in     Eq.F10-2
|
|  Crit. Elas. L-T-B Moment  : McrY   =  854.25     kip-in     Eq.C-F9-3
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.943      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  127.19     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  136.91     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  185.70     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    53. LOAD LIST 4
    54. PARAMETER 2
    55. CODE AISC UNIFIED 2016
    56. METHOD ASD
    57. CSPACING 32 ALL
    58. SGR 1 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
```

```
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  LD  L40356              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.945      Loadcase:       4
|
|  Location:      0.00       Ref:      Cl.E3
```

```
|  Pz:       80.00     C     Vy:        0.000          Vx:      0.000
|  Tz:       0.000           My:        0.000          Mx:      0.000
|-----------------------------------------------------------------------------
| COMPRESSION SLENDERNESS
| Actual Slenderness Ratio    :     76.914
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|-----------------------------------------------------------------------------
| STRENGTH CHECKS
| Critical L/C  :      4             Ratio     :        0.945(PASS)
|          Loc  :    0.00            Condition :    Cl.E3
|-----------------------------------------------------------------------------
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
| Ag  :   5.360E+00     Axx :   2.625E+00     Ayy :   3.000E+00
| Ixx :   8.350E+00     Iyy :   1.541E+01     J   :   2.512E-01
| Sxx+:   6.950E+00     Sxx-:   2.984E+00     Zxx :   5.417E+00
| Syy+:   3.978E+00     Syy-:   3.978E+00     Zyy :   7.107E+00
| Cw  :   7.750E-01     x0  :   0.000E+00     y0  :  -1.014E+00
|-----------------------------------------------------------------------------
| MATERIAL PROPERTIES
| Fyld:          36.000              Fu:          58.000
|-----------------------------------------------------------------------------
| Actual Member Length:        96.000
| Design Parameters                                             (Rolled)
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   32.00
|-----------------------------------------------------------------------------
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:    96.00)
|                           λ         λp        λr       CASE
| Flange: NonSlender       9.33       N/A      12.77     Table.4.1a.Case3
| Web   : NonSlender      10.67       N/A      12.77     Table.4.1a.Case3
```

```
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:    96.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          9.33      15.33     25.83     Table.4.1b.Case12
|
| Web   : Compact         10.67      15.33     25.83     Table.4.1b.Case12
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.   
11
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       115.5       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  192.96     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       155.4       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  5.3600     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  310.88     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             80.00       84.62       0.945     Cl.E3          4      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  76.914               Cl.E2
|  Elastic Buckling Stress   : Fex    =  48.382     ksi       Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  26.366     ksi       Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  141.32     kip       Eq.E3-1
|
|--------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             80.00       97.61       0.820     Cl.E3          4      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  56.612               Eq.E6-2a
|  Elastic Buckling Stress   : Fey    =  89.304     ksi       Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  30.411     ksi       Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  163.00     kip       Eq.E3-1
|
|--------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             80.00       91.48       0.875     Cl.E4          4      0.00
|
|
| Intermediate Results :
|  Constant                  : H      = 0.81177               Eq.E4-8
|  Euler Buckling Stress     : Fez    =  96.130     ksi       Eq.E4-7
|  Elastic F-T-B Stress      : Fe     =  64.518     ksi       Eq.E4-3
|  Crit. F-T-B Stress        : Fcr    =  28.502     ksi       Eq.E3-2
```

```
|
|  Nom. Flex-tor Buckling    : Pn     =  152.77     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       33.95       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  56.700     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       38.80       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000               Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  64.800     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
13
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
```

```
                                                  - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       96.48       0.000     Cl.F9.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  161.12     kip-in     Eq.F9-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       137.2       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  229.10     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       91.09       0.000     Cl.F9.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  152.12     kip-in     Eq.F10-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  84.707     in         Eq.F9-8
|
|  coefficient B             : Bx     = -.75059                Eq.F9-12
|
|  Limiting Unbraced Length  : LrX    =  1105.8     in         Eq.F9-9
|
|  Crit. Elastic L-T-B Mom.  : McrX   =  579.32     kip-in     Eq.F9-10
|
|-----------------------------------------------------------------------------
```

```
|
| LAT TOR BUCK ABOUT Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       123.6       0.000     Cl.F9.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About Y : Mny    =  206.33     kip-in     Eq.F10-2
|
|  Crit. Elas. L-T-B Moment  : McrY   =  854.25     kip-in     Eq.C-F9-3
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.945      Eq.H1-1a           4        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  84.625     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  91.088     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  123.55     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V. AISC 360-16 I Compression LRFD

Verify the flexural buckling capacity of an I section under compressive load using the LRFD method per the ASIC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

Verify the Flexural Buckling strength of an I Section (W14x90) using the LRFD method

Steel Gr. ASTM A992, about X and Y axis with a member length of 30 ft with dead load 140 kips and Live Load 420 kips.

Validation

Strong axis unbraced length, $L_x = 360\ in$

Torsional axis unbraced length $L_y = 180\ in$

Ultimate load, $P_u = 1.2(180\ kips) + 1.6(420\ kips) = 840\ kips$

Calculation for major axis (X axis)

From Table C-A-7.1 of ref. 1, $K = 1.0$ for a pinned-pinned condition.

$\frac{b_f}{2t_f} = 10.211 < 0.56\sqrt{\frac{E}{F_y}} = 13.487$, thus the flange is non-slender.

Slenderness ratio about X axis $= \frac{KL_x}{r_x} = 58.632 < 4.71\sqrt{\frac{E}{F_y}} = 113.432$

Calculate the elastic critical buckling stress: $F_{eX} = \frac{\pi^2 E}{(KL_x / r_x)^2} = 83.259ksi$

Calculate the flexural buckling stress: $F_{crX} = 0.658^{F_y/F_{eX}} F_y = 38.887ksi$

Compressive strength for flexural buckling along X: $\phi_c P_{nFBX} = \phi_c F_{crX} A_g = 947.462kips$

Ratio for flexural buckling strength = 0.906

Calculation for minor axis (Y axis)

$$h = d - 2 \cdot t_f = 12.58\ in$$

$\frac{h}{t_w} = 28.591 < 35.884$, thus the flange is non-slender.

Slenderness ratio about Y axis $= \frac{KL_y}{r_y} = 48.703 < 4.71\sqrt{\frac{E}{F_y}} = 113.432$

Calculate the elastic critical buckling stress: $F_{eY} = \frac{\pi^2 E}{(KL_y / r_y)^2} = 120.668ksi$

Calculate the flexural buckling stress: $F_{crY} = 0.658^{F_y/F_{eY}} F_y = 42.039ksi$

Compressive strength for flexural buckling along Y: $\phi_c P_{nFBY} = \phi_c F_{crY} A_g = 1,\ 002.625kips$

Ratio for flexural buckling strength = 0.838

Results

Table 731: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Flexural Buckling Capacity, X Axis (kips)	927.5	927.5	none	
Ratio	0.906	0.906	none	
Flexural Buckling Capacity, Y Axis (kips)	1,003	1,003	none	
Ratio	0.838	0.838	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 I Compression LRFD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Jan-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
```

```
2 FY -140
LOAD 2 LOADTYPE Live REDUCIBLE TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
*CODE AISC UNIFIED 2016
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
KX 1 ALL
KY 1 ALL
KZ 1 ALL
LX 30 ALL
LY 15 ALL
LZ 30 ALL
METHOD LRFD
PROFILE W14 ALL
STP 1 ALL
*SGR 28 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
```

```
       Mx              MZ          Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W14X90             (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         1.004       Loadcase:       3
|
|  Location:      0.00       Ref:      Cl.E4
|
|  Pz:       840.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     58.633
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        1.004(FAIL)
```

```
|
|         Loc  :    0.00           Condition :    Cl.E4
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00
|
| Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00
|
| Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02
|
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01
|
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                             λ         λp        λr       CASE
|
| Flange: NonSlender      10.21       N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      25.86       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                             λ         λp        λr       CASE
|
| Flange: NonCompact      10.21       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         25.86      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1192.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  1325.0     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1292.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.18403     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  1722.5     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       927.5       0.906     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  58.633                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  11989.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  5599.7     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  1030.5     kip        Eq.E3-1
```

```
||
|------------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1003.       0.838     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  48.701               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  17377.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  6053.6     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  1114.0     kip        Eq.E3-1
|
|------------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       837.0       1.004     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  8513.5     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  5053.6     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  930.01     kip        Eq.E4-1
|
|------------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|------------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|------------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       555.9       0.000     Cl.G1          3      0.00
```

```
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  617.70     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       184.8       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  184.80     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       283.5       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  315.00     kip-ft    Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
```

```
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       465.8       0.000     Cl.F2.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  517.55     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  13.055     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    = 0.34183     ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  42.564     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       573.6       0.000     Cl.F3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  637.28     kip-ft     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       272.7       0.000     Cl.F6.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  302.98     kip-ft     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                         -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
 |-----------------------------------------------------------------------------
```

```
                          CHECKS FOR AXIAL BEND INTERACTION

 ----------------------------------------------------------------------------

 COMBINED FORCES CLAUSE H1
                           RATIO      CRITERIA          L/C      LOC
                           1.004      Eq.H1-1a           3        0.00


 Intermediate Results :
  Axial Capacity           : Pc     =  837.01     kip        Cl.H1.1
  Moment Capacity          : Mcx    =  465.79     kip-ft     Cl.H1.1
  Moment Capacity          : Mcy    =  272.68     kip-ft     Cl.H1.1
 ----------------------------------------------------------------------------

    55. FINISH
```

V.AISC 360-16 Pipe E.11

Verify the available compressive strength of a pipe section per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-61 – E-63., Example E.11

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference 2:

Select an ASTM A53 Grade B Pipe compression member with a length of 30 ft to support a dead load of 35 kips and live load of 105 kips in axial compression. The column is pin-connected at the ends in both axes and braced at the midpoint in the y-y direction.

ASTM A572 Grade 50: $F_y = 35\ ksi$

Validation

Refer to reference.

Results

Table 732: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	221	218.6	1.1%	
P_n / Ω_c (kips)	147	145.4	1.1%	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Pipe E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS100
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -35
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -105
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
```

```
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
LY 180 ALL
METHOD LRFD
SGR 2 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
LY 180 ALL
METHOD ASD
SGR 2 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
```

```
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  PIPS100            (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.961      Loadcase:       3
 |
 |  Location:      0.00       Ref:      Cl.E3
 |
 |  Pz:       210.0     C     Vy:        0.000          Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000          Mx:      0.000
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     99.349
 |
 | Allowable Slenderness Ratio :    200.000            LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      3             Ratio     :        0.961(PASS)
 |
 |          Loc  :    0.00            Condition :    Cl.E3
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   1.150E+01     Axx :   0.000E+00     Ayy :   0.000E+00
 |
```

```
| Ixx :   1.510E+02     Iyy :   1.510E+02     J   :   3.020E+02
|
| Sxx+:   2.809E+01     Sxx-:   2.809E+01     Zxx :   3.690E+01
|
| Syy+:   2.809E+01     Syy-:   2.809E+01     Zyy :   3.690E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          35.000              Fu:          60.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:       360.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                          λ         λp        λr       CASE
|
| Flange: NonSlender      31.62       N/A      91.14     Table.4.1a.Case9
|
| Web   : NonSlender      31.62       N/A      91.14     Table.4.1a.Case9
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                          λ         λp        λr       CASE
|
| Flange: Compact         31.62      58.00    256.86     Table.4.1b.Case20
|
| Web   : Compact         31.62      58.00    256.86     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                              -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
```

```
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       362.2       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap          : Pn     =  402.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
| TENSILE RUPTURE
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       517.5       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area             : Ae     =  11.500     in2        Eq.D3-1
|  Nom. Ten. Rpt Cap          : Pn     =  690.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|                        CHECKS FOR AXIAL COMPRESSION
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             210.0       218.6       0.961     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness      : Lcx/rx =  99.349                Cl.E2
|  Elastic Buckling Stress    : Fex    =  28.998     ksi        Eq.E3-4
|  Crit. Buckling Stress      : Fcrx   =  21.119     ksi        Eq.E3-2
|  Nom. Flexural Buckling     : Pnx    =  242.87     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             210.0       319.3       0.658     Cl.E3          3      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  49.674                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  115.99     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  30.847     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  354.75     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
9
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                                 - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       108.7       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  120.75     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  21.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       108.7       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  120.75     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  21.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
```

```
      STAAD SPACE                                              -- PAGE NO.    
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                    - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (X)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       1162.       0.000     Cl.F8.1        3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom Flex Yielding Along X : Mnx    =  1291.5     kip-in     Eq.F8-1
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (Y)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       1162.       0.000     Cl.F8.1        3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom Flex Yielding Along Y : Mny    =  1291.5     kip-in     Eq.F8-1
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                              -- PAGE NO.    
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                    - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR AXIAL BEND INTERACTION
 |
 |-----------------------------------------------------------------------------
 |
 | COMBINED FORCES CLAUSE H1
 |
 |                           RATIO      CRITERIA          L/C      LOC
 |
 |                           0.961      Eq.H1-1a           3        0.00
 |
```

```
|
|
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  218.58     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  1162.4     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  1162.4     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1094.       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr        : Fcr    =  21.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength            : Tn     =  1215.4     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2016
    52. LY 180 ALL
    53. METHOD ASD
    54. SGR 2 ALL
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
13
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
```

```
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  PIPS100             (AISC
SECTIONS)|
```

```
 Status:        PASS       Ratio:        0.963       Loadcase:        4

 Location:      0.00       Ref:      Cl.E3

 Pz:       140.0     C     Vy:        0.000          Vx:      0.000

 Tz:       0.000           My:        0.000          Mx:      0.000

-----------------------------------------------------------------------------

COMPRESSION SLENDERNESS

Actual Slenderness Ratio    :     99.349

Allowable Slenderness Ratio :    200.000            LOC  :     0.00

-----------------------------------------------------------------------------

STRENGTH CHECKS

Critical L/C  :      4             Ratio     :        0.963(PASS)

         Loc  :    0.00            Condition :    Cl.E3

-----------------------------------------------------------------------------

SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )

Ag  :   1.150E+01     Axx :   0.000E+00     Ayy :   0.000E+00

Ixx :   1.510E+02     Iyy :   1.510E+02     J   :   3.020E+02

Sxx+:   2.809E+01     Sxx-:   2.809E+01     Zxx :   3.690E+01

Syy+:   2.809E+01     Syy-:   2.809E+01     Zyy :   3.690E+01

Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00

-----------------------------------------------------------------------------

MATERIAL PROPERTIES

Fyld:         35.000             Fu:          60.000

-----------------------------------------------------------------------------

Actual Member Length:       360.000

Design Parameters                                               (Rolled)

Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00

-----------------------------------------------------------------------------

COMPRESSION CLASSIFICATION (L/C:      4 LOC:   360.00)

                           λ         λp        λr       CASE
```

```
| Flange: NonSlender      31.62       N/A      91.14     Table.4.1a.Case9
|
| Web   : NonSlender      31.62       N/A      91.14     Table.4.1a.Case9
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   360.00)
|
|                          λ         λp        λr       CASE
|
| Flange: Compact         31.62      58.00    256.86     Table.4.1b.Case20
|
| Web   : Compact         31.62      58.00    256.86     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
15
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       241.0       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  402.50     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       345.0       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  11.500     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  690.00     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
```

```
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             140.0       145.4       0.963     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  99.349              Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  28.998     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  21.119     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  242.87     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             140.0       212.4       0.659     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  49.674              Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  115.99     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  30.847     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  354.75     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
16
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       72.31       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  120.75     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  21.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       72.31       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  120.75     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  21.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
17
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       773.4       0.000     Cl.F8.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1291.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
```

```
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       773.4       0.000     Cl.F8.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1291.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                            -- PAGE NO.
18
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.963      Eq.H1-1a           4        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  145.43     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  773.35     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  773.35     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                            -- PAGE NO.
19
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|              0.000       727.8       0.000     Cl.H3.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr          : Fcr     =  21.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength              : Tn      =  1215.4     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 Rect HSS E.9

Verify the available compressive strength of a rectangular HSS member per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-53 – E-55., Example E.9

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference 2:

Select an ASTM A500 Grade C rectangular HSS compression member, with a length of 20 ft, to support a dead load of 85 kips and live load of 255 kips in axial compression. The base is fixed and the top is pinned.

ASTM A500 Grade C: $F_y = 50\ ksi$

Validation

Refer to reference.

Results

Table 733: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	556	555.3	negligible	
P_n / Ω_c (kips)	370	369.5	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Rect HSS E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 240 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST12X10X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -85
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -255
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
KX 0.8 ALL
KY 0.8 ALL
METHOD LRFD
SGR 5 ALL
TRACK 2 ALL
KZ 0.8 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
KX 0.8 ALL
```

```
KY 0.8 ALL
KZ 0.8 ALL
METHOD ASD
SGR 5 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
```

```
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST12X10X0.375     (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.918       Loadcase:       3
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       510.0     C     Vy:        0.000          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     47.959
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.918(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.460E+01     Axx :   6.249E+00     Ayy :   7.645E+00
|
| Ixx :   3.100E+02     Iyy :   2.340E+02     J   :   4.210E+02
|
| Sxx+:   5.167E+01     Sxx-:   5.167E+01     Zxx :   6.110E+01
|
| Syy+:   4.680E+01     Syy-:   4.680E+01     Zyy :   5.400E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
```

```
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          62.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:       240.000
|
| Design Parameters                                              (Rolled)
|
| Kx:    0.80  Ky:    0.80  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   240.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender      25.65       N/A      33.72     Table.4.1a.Case6
|
| Web   : NonSlender      31.38       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   240.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact         25.65      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         31.38      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       657.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  730.00     kip        Eq.D2-1
|
```

```
|---------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       678.9       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area             : Ae     =  14.600     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap          : Pn     =  905.20     kip        Eq.D2-2
|
|---------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|---------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             510.0       578.7       0.881     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness      : Lcx/rx =  41.667                Cl.E2
|
|  Elastic Buckling Stress    : Fex    =  164.86     ksi        Eq.E3-4
|
|  Crit. Buckling Stress      : Fcrx   =  44.039     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling     : Pnx    =  642.97     kip        Eq.E3-1
|
|---------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             510.0       555.3       0.918     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness      : Lcy/ry =  47.959                Cl.E2
|
|  Elastic Buckling Stress    : Fey    =  124.44     ksi        Eq.E3-4
|
|  Crit. Buckling Stress      : Fcry   =  42.260     ksi        Eq.E3-2
|
```

```
|  Nom. Flexural Buckling    : Pny    =  617.00     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
                             STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                           *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                              - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       168.7       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along X        : Vnx    =  187.48     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       206.4       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  229.36     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
7
                             STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                           *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                              - Member :     1 Contd.
```

```
|--------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|--------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2749.       0.000     Cl.F7.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  3055.0     kip-in     Eq.F7-1
|
|--------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2430.       0.000     Cl.F7.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  2700.0     kip-in     Eq.F7-1
|
|--------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2219.       0.000     Cl.F7.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  2465.2     kip-in     Eq.F7-2
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|--------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
```

```
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.918      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  555.30     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  2749.5     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  2218.7     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2115.       0.000     Cl.H3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr        : Fcr    =  30.000     ksi        Eq.H3-3
|
|  Nom. Strength            : Tn     =  2349.6     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2016
    52. KX 0.8 ALL
    53. KY 0.8 ALL
    54. KZ 0.8 ALL
    55. METHOD ASD
    56. SGR 5 ALL
    57. TRACK 2 ALL
    58. CHECK CODE ALL
```

```
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
```

```
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  HSST12X10X0.375     (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.920      Loadcase:       4
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       340.0     C     Vy:        0.000          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      0.000
|
|-------------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     47.959
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|-------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        0.920(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|-------------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.460E+01     Axx :   6.249E+00     Ayy :   7.645E+00
|
| Ixx :   3.100E+02     Iyy :   2.340E+02     J   :   4.210E+02
|
| Sxx+:   5.167E+01     Sxx-:   5.167E+01     Zxx :   6.110E+01
|
| Syy+:   4.680E+01     Syy-:   4.680E+01     Zyy :   5.400E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-------------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          62.000
|
|-------------------------------------------------------------------------------
|
| Actual Member Length:       240.000
|
```

```
| Design Parameters                                             (Rolled)
|
| Kx:    0.80  Ky:    0.80  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   240.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender      25.65       N/A      33.72     Table.4.1a.Case6
|
| Web   : NonSlender      31.38       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   240.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact         25.65      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         31.38      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       437.1       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  730.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       452.6       0.000     Cl.D2          4      0.00
|
```

```
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  14.600     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  905.20     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              340.0       385.0       0.883     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  41.667                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  164.86     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  44.039     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  642.97     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              340.0       369.5       0.920     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  47.959                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  124.44     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  42.260     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  617.00     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
13
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
```

```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                         CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       112.3       0.000     Cl.G1          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  5.0000               Cl.G4
 |
 |  Nom. Shear Along X        : Vnx    =  187.48     kip       Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       137.3       0.000     Cl.G1          4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000               Eq.G2-9
 |
 |  Coefficient Kv Along Y    : Kv     =  5.0000               Cl.G4
 |
 |  Nom. Shear Along Y        : Vny    =  229.36     kip       Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                                -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                         CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (X)
 |
```

```
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1829.       0.000     Cl.F7.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  3055.0     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1617.       0.000     Cl.F7.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  2700.0     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| WEB LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1476.       0.000     Cl.F7.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Web Local Buckling    : Mny    =  2465.2     kip-in     Eq.F7-2
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
15
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.920      Eq.H1-1a           4        0.00
|
```

```
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc      =  369.46     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx     =  1829.3     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy     =  1476.2     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
16
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1407.       0.000     Cl.H3.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr     =  30.000     ksi        Eq.H3-3
|
|  Nom. Strength             : Tn      =  2349.6     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 W E.1A

Verify the available compressive strength of a wide flange section per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-4 – E-5., Example E.1A

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

The section checked is a W14x120. From reference 2:

> The column is pinned top and bottom in both axes. Limit the column size to a nominal 14-in. shape. The column is verified for ... ASTM A913 Grade 65 material.

ASTM A572 Grade 65: $F_y = 65\ ksi$

Validation

Refer to reference.

Results

Table 734: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	865	857.8	negligible	
P_n / Ω_c (kips)	569	570.8	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
```

```
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
KX 1 ALL
KY 1 ALL
METHOD LRFD
PROFILE W14 ALL
SGR 26 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
SGR 26 ALL
KX 1 ALL
KY 1 ALL
METHOD ASD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
```

```
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W14X120             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.979       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       840.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|------------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
```

```
|
| Actual Slenderness Ratio    :     96.136
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.979(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   3.530E+01     Axx :   2.764E+01     Ayy :   8.555E+00
|
| Ixx :   1.380E+03     Iyy :   4.950E+02     J   :   9.370E+00
|
| Sxx+:   1.903E+02     Sxx-:   1.903E+02     Zxx :   2.120E+02
|
| Syy+:   6.735E+01     Syy-:   6.735E+01     Zyy :   1.020E+02
|
| Cw  :   2.288E+04     x0  :   0.000E+00     y0  :  -8.882E-16
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        9359.999             Fu:       11519.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       7.82       N/A      11.83     Table.4.1a.Case1
|
| Web   : NonSlender      19.36       N/A      31.47     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          7.82       8.03     21.12     Table.4.1b.Case10
```

```
|
| Web   : Compact         19.36      79.42    120.40     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2065.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  2294.5     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2118.       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.24514     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  2824.0     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1507.       0.557     Cl.E3          3      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  57.577              Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12433.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6830.1     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  1674.3     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       857.8       0.979     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  96.136              Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  4459.5     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  3888.2     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  953.16     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1487.       0.565     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  11940.     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  6741.8     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  1652.7     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
```

```
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       970.0       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  1077.8     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       333.6       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  333.64     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1034.       0.000     Cl.F2.1        3      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1148.3     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       497.3       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  552.50     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       797.4       0.000     Cl.F2.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  886.00     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  11.601     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    = 0.35039     ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  41.522     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
```

```
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.979      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  857.84     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  797.40     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  497.25     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    51. LOAD LIST 4
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2016
    54. SGR 26 ALL
    55. KX 1 ALL
    56. KY 1 ALL
    57. METHOD ASD
    58. PROFILE W14 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
```

```
       Tz               MX         Torsional moment.
       My               MY         Bending moment about minor axis.
       Mx               MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W14X120             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.981      Loadcase:        4
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       560.0     C     Vy:        0.000          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     96.136
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
```

```
|
| Critical L/C  :      4            Ratio     :        0.981(PASS)
|
|         Loc   :    0.00           Condition :    Cl.E3
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   3.530E+01     Axx :   2.764E+01     Ayy :   8.555E+00
|
| Ixx :   1.380E+03     Iyy :   4.950E+02     J   :   9.370E+00
|
| Sxx+:   1.903E+02     Sxx-:   1.903E+02     Zxx :   2.120E+02
|
| Syy+:   6.735E+01     Syy-:   6.735E+01     Zyy :   1.020E+02
|
| Cw  :   2.288E+04     x0  :   0.000E+00     y0  :  -8.882E-16
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        9359.999             Fu:       11519.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        30.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       7.82       N/A      11.83     Table.4.1a.Case1
|
| Web   : NonSlender      19.36       N/A      31.47     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          7.82       8.03     21.12     Table.4.1b.Case10
|
| Web   : Compact         19.36      79.42    120.40     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
```

```
                    ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       1374.       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn      =  2294.5     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       1412.       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae      = 0.24514     ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn      =  2824.0     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            560.0       1003.       0.559     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  57.577               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  12433.     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6830.1     kip/ft2    Eq.E3-2
```

```
|
|  Nom. Flexural Buckling    : Pnx    =  1674.3     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             560.0       570.8       0.981     Cl.E3          4      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  96.136                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  4459.5     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  3888.2     kip/ft2    Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  953.16     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             560.0       989.6       0.566     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  11940.     kip/ft2    Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  6741.8     kip/ft2    Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  1652.7     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
15
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|              0.000       645.4       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  1077.8     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       222.4       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  333.64     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
16
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       687.6       0.000     Cl.F2.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1148.3     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       330.8       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  552.50     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       530.5       0.000     Cl.F2.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  886.00     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  11.601     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    = 0.35039     ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  41.522     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
17
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.981      Eq.H1-1a           4        0.00
|
|
```

```
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  570.75     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  530.54     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  330.84     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 W E.1B

Verify the compressive strength of a wide flange section per the AISC 360-16 code.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp.E-4 – E-5., Example E.1A

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference 2:

> Verify a W14x90 is adequate to carry the loading... The column is pinned top and bottom in both axes and braced at the midpoint about the y-y axis and torsionally. The column is verified for ... ASTM A913 Grade 65 material.

ASTM A572 Grade 65: $F_y = 65\ ksi$

Validation

Refer to reference.

Results

Table 735: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips)	1,080	1,080	none	
P_n / Ω_c (kips)	719	718.4	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 04-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 360 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
LY 228 ALL
LX 228 ALL
METHOD LRFD
SGR 26 ALL
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
```

```
LY 228 ALL
LX 228 ALL
SGR 26 ALL
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W14X90             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.778       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.E3
|
|  Pz:       840.0     C     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     61.688
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.778(PASS)
|
|          Loc  :    0.00            Condition :    Cl.E3
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00
|
| Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00
|
| Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02
|
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01
|
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
```

```
 |
 | MATERIAL PROPERTIES
 |
 | Fyld:          65.000              Fu:          80.000
 |
 |-----------------------------------------------------------------------------
 |
 | Actual Member Length:       360.000
 |
 | Design Parameters                                                 (Rolled)
 |
 | Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      3 LOC:   360.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonSlender      10.21       N/A      11.83     Table.4.1a.Case1
 |
 | Web   : NonSlender      25.86       N/A      31.47     Table.4.1a.Case5
 |
 |
 |
 | FLEXURE CLASSIFICATION     (L/C:      3 LOC:   360.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonCompact      10.21       8.03     21.12     Table.4.1b.Case10
 |
 | Web   : Compact         25.86      79.42    120.40     Table.4.1b.Case15
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                          CHECKS FOR AXIAL TENSION
 |
 |-----------------------------------------------------------------------------
 |
 | TENSILE YIELDING
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       1550.       0.000     Cl.D2          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Ten. Yld Cap         : Pn     =  1722.5     kip        Eq.D2-1
```

```
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1590.       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  26.500     in2         Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  2120.0     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1118.       0.751     Cl.E3          3      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  58.633                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  83.255     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  46.881     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  1242.3     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1080.       0.778     Cl.E3          3      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  61.688                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  75.213     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  45.271     ksi         Eq.E3-2
```

```
|
| Nom. Flexural Buckling    : Pny    =  1199.7     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             840.0       1174.       0.716     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  97.853     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  49.223     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  1304.4     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                            -- PAGE NO.
6
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       722.7       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  803.01     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       240.2       0.000     Cl.G1          3      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  240.24     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
7
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       4423.       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  4914.0     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       6542.       0.000     Cl.F2.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  7268.8     kip-in     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  137.40     in         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    =  4.1020     in         Eq.F2-7
```

```
|
|  Limiting Unbraced Length  : LrX    =  418.78     in         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       8627.       0.000     Cl.F3.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  9585.8     kip-in     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       4026.       0.000     Cl.F6.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  4473.2     kip-in     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
8
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.778      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  1079.7     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  6541.9     kip-in     Cl.H1.1
```

```
|
|  Moment Capacity           : Mcy    =  4025.9     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2016
    52. METHOD ASD
    53. LY 228 ALL
    54. LX 228 ALL
    55. SGR 26 ALL
    56. STP 1 ALL
    57. TRACK 2 ALL
    58. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
```

```
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  W14X90              (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.780       Loadcase:        4
 |
 |  Location:      0.00       Ref:      Cl.E3
 |
 |  Pz:       560.0     C     Vy:        0.000           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:      0.000
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     61.688
 |
 | Allowable Slenderness Ratio :    200.000            LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      4             Ratio     :        0.780(PASS)
 |
 |          Loc  :    0.00            Condition :    Cl.E3
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00
 |
 | Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00
 |
 | Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02
```

```
|
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01
|
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          65.000             Fu:          80.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:       360.000
|
| Design Parameters                                            (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender      10.21       N/A      11.83     Table.4.1a.Case1
|
| Web   : NonSlender      25.86       N/A      31.47     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:   360.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonCompact      10.21       8.03     21.12     Table.4.1b.Case10
|
| Web   : Compact         25.86      79.42    120.40     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1031.       0.000     Cl.D2          4      0.00
```

```
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  1722.5     kip         Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1060.       0.000     Cl.D2          4      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     =  26.500     in2         Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  2120.0     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             560.0       743.9       0.753     Cl.E3          4      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  58.633                 Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  83.255     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  46.881     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  1242.3     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             560.0       718.4       0.780     Cl.E3          4      0.00
|
|
| Intermediate Results :
```

```
|
| Effective Slenderness     : Lcy/ry =  61.688               Cl.E2
|
| Elastic Buckling Stress   : Fey    =  75.213     ksi       Eq.E3-4
|
| Crit. Buckling Stress     : Fcry   =  45.271     ksi       Eq.E3-2
|
| Nom. Flexural Buckling    : Pny    =  1199.7     kip       Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             560.0       781.1       0.717     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
| Elastic F-T-B Stress      : Fe     =  97.853     ksi       Eq.E4-2
|
| Crit. F-T-B Stress        : Fcr    =  49.223     ksi       Eq.E3-2
|
| Nom. Flex-tor Buckling    : Pn     =  1304.4     kip       Eq.E4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       480.8       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
| Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
| Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
| Nom. Shear Along X        : Vnx    =  803.01     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
```

```
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       160.2       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  240.24     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
13
                  STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       2943.       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  4914.0     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       4353.       0.000     Cl.F2.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  7268.8     kip-in     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000               Custom
```

```
 |
 |  Limiting Unbraced Length  : LpX    =  137.40     in         Eq.F2-5
 |
 |  coefficient C             : Cx     =  1.0000                Eq.F2-8a
 |
 |  Effective Rad. of Gyr.    : Rts    =  4.1020     in         Eq.F2-7
 |
 |  Limiting Unbraced Length  : LrX    =  418.78     in         Eq.F2-6
 |
 |-----------------------------------------------------------------------------
 |
 | FLANGE LOCAL BUCK(X)
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              0.000       5740.       0.000     Cl.F3.1        4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom F-L-B Cap             : Mnx    =  9585.8     kip-in     Eq.F3-1
 |
 |-----------------------------------------------------------------------------
 |
 | FLANGE LOCAL BUCK(Y)
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              0.000       2679.       0.000     Cl.F6.2        4      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom F-L-B Cap             : Mny    =  4473.2     kip-in     Eq.F6-2
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                             -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                         CHECKS FOR AXIAL BEND INTERACTION
 |
 |-----------------------------------------------------------------------------
 |
 | COMBINED FORCES CLAUSE H1
 |
 |                            RATIO      CRITERIA          L/C      LOC
 |
 |                            0.780      Eq.H1-1a           4        0.00
 |
 |
```

```
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  718.37     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  4352.6     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  2678.6     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

V. AISC 360-16 Shear Strong Axis

Verify the shear strength of an I section loaded in the strong axis using the LRFD and ASD methods per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017. pp G-3 - G-4, Example G.1A

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From the reference:

> determine the available shear strength and adequacy of an ASTM A992 W24X62 with end shears of 48 kips from dead load and 145 kips from live load.

The shear capacity values are evaluated from AISC Manual Table 2-4 and the also calculated.

Validation

Area of the web used for shear is:

$$A_w = dt_w = 23.7 \times 0.430 = 10.2 \text{ in}^2$$

The nominal shear strength is:

$$V_n = 0.6F_yA_wC_{v1} = 0.6\,(50)(10.2)(1.0) = 306 \text{ kips}$$

LRFD

The shear strength is:

$$\phi_vV_n = 1.00(306) = 306 \text{ kips}$$

The ultimate shear demand is:

$$Vu = 1.2\,(48) + 1.6\,(145) = 289.6 \text{ kips}$$

ASD

The allowable shear is:

$$V_n/\Omega_v = 306 / 1.5 = 204 \text{ kips}$$

The shear demand is:

$$V_a = 48 + 145 = 193 \text{ kips}$$

Results

Table 736: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear capacity, $\phi_v V_n$, Y axis (kips) LRFD	306	305.7	negligible	
Shear demand, V_u, Y axis (kips) LRFD	289.6	289.6	none	The reference rounds this value to 290 kips.
Shear capacity, V_n/Ω_vm, Y axis (kips) ASD	204	203.8	negligible	
Shear demand, V_a, Y axis (kips) ASD	193	193.0	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Shear Strong Axis.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Nov-18
JOB NAME AISC Design Example v15.0 - G.1
END JOB INFORMATION
*********************************************
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
**********************************************
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
**********************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X62
*
CONSTANTS
MATERIAL STEEL_50_KSI ALL
*
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
**********************************************
UNIT INCHES KIP
LOAD 1 LOADTYPE Dead  TITLE Dead
MEMBER LOAD
1 CON GY -96
LOAD 2 LOADTYPE Live  TITLE Live
MEMBER LOAD
1 CON GY -290
*
UNIT FEET KIP
LOAD COMB 3 LRFD COMBINATION
1 1.2 2 1.6
*
LOAD COMB 4 ASD Combination
1 1.0 2 1.0
*********************************************
PERFORM ANALYSIS
UNIT INCHES KIP
PRINT SUPPORT REACTION ALL
*********************************************
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
*
TRACK 2 ALL
CHECK CODE ALL
*********************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
*
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
```

```
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC 
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections 
principal major axis.
       Y               Y          Axis typically parallel to the sections 
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the 
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.    
5
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X62             (AISC 
SECTIONS)|
```

```
|  Status:        FAIL       Ratio:         1.009       Loadcase:        3
|
|  Location:     24.00       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        289.6           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -6950.
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     34.863
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        1.009(FAIL)
|
|          Loc  :   24.00            Condition :    Cl.F2.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    24.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.820E+01     Axx :   8.307E+00     Ayy :   1.019E+01
|
| Ixx :   1.550E+03     Iyy :   3.450E+01     J   :   1.710E+00
|
| Sxx+:   1.308E+02     Sxx-:   1.308E+02     Zxx :   1.530E+02
|
| Syy+:   9.801E+00     Syy-:   9.801E+00     Zyy :   1.570E+01
|
| Cw  :   4.581E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          62.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        48.000
|
| Design Parameters                                              (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    48.00)
|
|                            λ         λp        λr       CASE
|
```

```
| Flange: NonSlender       5.97       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         50.05       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    48.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          5.97       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         50.05      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                 -- PAGE NO.
6
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                                   - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       819.0       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  910.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       846.3       0.000     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  18.200     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  1128.4     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
```

```
                        CHECKS FOR AXIAL COMPRESSION

|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             0.000       726.0       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  5.2013               Cl.E2
|  Elastic Buckling Stress   : Fex    =  10580.     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  49.901     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  806.66     kip        Eq.E7-1
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             0.000       676.0       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  34.863               Cl.E2
|  Elastic Buckling Stress   : Fey    =  235.48     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  45.748     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  751.08     kip        Eq.E7-1
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             0.000       694.3       0.000     Cl.E4          3      0.00
|
|
| Intermediate Results :
|  Elastic F-T-B Stress      : Fe     =  371.19     ksi        Eq.E4-2
|  Crit. F-T-B Stress        : Fcr    =  47.259     ksi        Eq.E3-2
```

```
|  Nom. Flex-tor Buckling    : Pn     =  771.39     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
7
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                              - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       224.3       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  249.22     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             289.6       305.7       0.947     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  305.73     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
8
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                              - Member :     1 Contd.
```

```
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -6950.       6885.       1.009     Cl.F2.1        3     24.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  7650.0     kip-in     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       705.7       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  784.09     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA           L/C      LOC
|
|                            1.009      Eq.H1-1b            3       24.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000               Eq.H1-2
|
```

```
|  Axial Capacity           : Pc     =  819.00     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  6885.0     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  705.68     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    61. *********************************************
    62. LOAD LIST 4
    63. PARAMETER 2
    64. CODE AISC UNIFIED 2016
    65. METHOD ASD
    66. *
    67. TRACK 2 ALL
    68. CHECK CODE ALL
 PARAMETER 2
      STAAD SPACE                                                -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
```

```
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W24X62              (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         1.011       Loadcase:        4
|
|  Location:     24.00       Ref:      Cl.F2.1
|
|  Pz:       0.000     T     Vy:        193.0           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:     -4632.
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     34.863
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        1.011(FAIL)
|
|          Loc  :   24.00            Condition :    Cl.F2.1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    24.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.820E+01     Axx :   8.307E+00     Ayy :   1.019E+01
|
| Ixx :   1.550E+03     Iyy :   3.450E+01     J   :   1.710E+00
|
```

```
| Sxx+:   1.308E+02     Sxx-:   1.308E+02     Zxx :   1.530E+02
|
| Syy+:   9.801E+00     Syy-:   9.801E+00     Zyy :   1.570E+01
|
| Cw  :   4.581E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:         50.000             Fu:         62.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        48.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:    48.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       5.97       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         50.05       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:    48.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          5.97       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         50.05      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|               0.000       544.9       0.000     Cl.D2          4      0.00
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  910.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
| TENSILE RUPTURE
|               DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|               0.000       564.2       0.000     Cl.D2          4      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     =  18.200     in2        Eq.D3-1
|  Nom. Ten. Rpt Cap         : Pn     =  1128.4     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|                          CHECKS FOR AXIAL COMPRESSION
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|               DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|               0.000       483.0       0.000     Cl.E3          4      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  5.2013                Cl.E2
|  Elastic Buckling Stress   : Fex    =  10580.     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  49.901     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  806.66     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|               DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|               0.000       449.7       0.000     Cl.E3          4      0.00
|
|
```

```
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  34.863               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  235.48     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  45.748     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  751.08     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       461.9       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  371.19     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  47.259     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  771.39     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       149.2       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  249.22     kip        Eq.G6-1
|
```

```
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             193.0       203.8       0.947     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  305.73     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
14
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -4632.       4581.       1.011     Cl.F2.1        4     24.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  7650.0     kip-in     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       469.5       0.000     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  784.09     kip-in     Eq.F6-1
|
```

```
|--------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
15
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
|--------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|--------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           1.011      Eq.H1-1b           4       24.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  544.91     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  4580.8     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  469.52     kip-in     Cl.H1.1
|
|--------------------------------------------------------------------------
|
```

V. AISC 360-16 Shear Weak Axis

Verify the shear strength of an I section loaded in the weak axis using the LRFD and ASD methods per AISC 360-16.

References

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Design Examples*, Version 15.0, American Institute of Steel Construction, 2017, pp G-14 - G-15, Example G.1B

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From the reference:

Verify the available shear strength and adequacy of an ASTM A992 W21X48 beam with end shears of 20.0 kips from dead load and 60.0 kips from live load in the weak direction.

Validation

$C_{v2} = 1.0$ for all ASTM A6 W shapes with Fy ≤ 70 ksi, per the User Note in section G6 of the AISC 360-16 specification.

The nominal shear strength is:

$$V_n = 2\times[0.6F_y b_f t_f C_{v2}] = 2\times[0.6\ (50)(8.14)(0.43)(1.0)] = 210.0 \text{ kips}$$

<u>LRFD</u>

The shear strength is:

$$\phi_v V_n = 0.90(210.0) = 189.0 \text{ kips}$$

The ultimate shear demand is:

$$Vu = 1.2\ (20) + 1.6\ (60) = 120 \text{ kips}$$

<u>ASD</u>

The allowable shear is:

$$V_n/\Omega_v = 306\ /\ 1.67 = 125.8 \text{ kips}$$

The shear demand is:

$$V_a = 20 + 60 = 80 \text{ kips}$$

Results

Table 737: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear capacity, $\phi_v V_n$, X axis (kips) LRFD	189	189.0	none	
Shear demand, V_u, X axis (kips) LRFD	120	120.0	none	
Shear capacity, V_n/Ω_vm, X axis (kips) ASD	125.8	125.8	none	The reference rounds this value to 126 kips.
Shear demand, V_a, X axis (kips) ASD	80	80.0	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Shear Weak Axis.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 27-Nov-18
JOB NAME AISC Desgin Example v15.0 - G.6
END JOB INFORMATION
*********************************************
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
**********************************************
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
**********************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
*
CONSTANTS
MATERIAL STEEL_50_KSI ALL
BETA 90.0 MEMBER 1
*
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
**********************************************
UNIT INCHES KIP
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -120
*
UNIT FEET KIP
LOAD COMB 3 LRFD COMBINATION
1 1.2 2 1.6
*
LOAD COMB 4 ASD Combination
1 1.0 2 1.0
*********************************************
PERFORM ANALYSIS
UNIT INCHES KIP
```

```
PRINT SUPPORT REACTION ALL
*********************************************
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
METHOD LRFD
*
TRACK 2 ALL
CHECK CODE ALL
*********************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2016
METHOD ASD
*
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
```

```
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W21X48             (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         4.346       Loadcase:       3
|
|  Location:     24.00       Ref:      Cl.F6.2
|
|  Pz:       0.000     T     Vy:        0.000           Vx:     -120.0
|
|  Tz:       0.000           My:       -2880.           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     28.973
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        4.346(FAIL)
|
|          Loc  :   24.00            Condition :    Cl.F6.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    24.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.410E+01     Axx :   7.000E+00     Ayy :   7.210E+00
```

```
|
| Ixx :   9.590E+02     Iyy :   3.870E+01     J   :   8.030E-01
|
| Sxx+:   9.311E+01     Sxx-:   9.311E+01     Zxx :   1.070E+02
|
| Syy+:   9.509E+00     Syy-:   9.509E+00     Zyy :   1.490E+01
|
| Cw  :   3.931E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          62.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        48.000
|
| Design Parameters                                             (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    48.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       9.47       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         53.54       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    48.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonCompact       9.47       9.15     24.08     Table.4.1b.Case13
|
| Web   : Compact         53.54     ******    ******     Table.4.1.
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
```

```
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       634.5       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  705.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       655.7       0.000     Cl.D2          3      0.00
|
|
| Intermediate Results :
|  Effective area            : Ae     =  14.100     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  874.20     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       556.1       0.000     Cl.E3          3      0.00
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  5.8202               Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  8449.2     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  49.876     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  617.87     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.000       529.2       0.000     Cl.E3          3      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  28.973               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  340.96     ksi       Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  47.023     ksi       Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  588.02     kip       Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       537.9       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  498.48     ksi       Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  47.944     ksi       Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  597.69     kip       Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             120.0       189.0       0.635     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
```

```
 |
 |  Nom. Shear Along X        : Vnx    =  210.01     kip        Eq.G6-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       216.3       0.000     Cl.G1          3      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000                -
 |
 |  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
 |
 |  Nom. Shear Along Y        : Vny    =  216.30     kip        Eq.G2-1
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                             -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (Y)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             2880.       670.5       4.295     Cl.F6.1        3     24.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom Flex Yielding Along Y : Mny    =  745.00     kip-in     Eq.F6-1
 |
 |-----------------------------------------------------------------------------
 |
 | FLANGE LOCAL BUCK(X)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       4775.       0.000     Cl.F3.1        3      0.00
 |
 |
 |
 | Intermediate Results :
```

```
|
|  Nom F-L-B Cap             : Mnx    =  5306.1     kip-in     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             2880.       662.7       4.346     Cl.F6.2        3     24.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  736.35     kip-in     Eq.F6-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           4.346      Eq.H1-1b           3       24.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000                Eq.H1-2
|
|  Axial Capacity            : Pc     =  634.50     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  4775.5     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  662.71     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    62. *********************************************
    63. LOAD LIST 4
    64. PARAMETER 2
    65. CODE AISC UNIFIED 2016
    66. METHOD ASD
    67. *
    68. TRACK 2 ALL
```

```
   69. CHECK CODE ALL
 PARAMETER 2
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
```

```
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W21X48             (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         4.354       Loadcase:       4
|
|  Location:     24.00       Ref:      Cl.F6.2
|
|  Pz:       0.000     T     Vy:         0.000           Vx:     -80.00
|
|  Tz:       0.000           My:       -1920.            Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     28.973
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      4             Ratio     :        4.354(FAIL)
|
|          Loc  :   24.00            Condition :    Cl.F6.2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    24.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.410E+01     Axx :   7.000E+00     Ayy :   7.210E+00
|
| Ixx :   9.590E+02     Iyy :   3.870E+01     J   :   8.030E-01
|
| Sxx+:   9.311E+01     Sxx-:   9.311E+01     Zxx :   1.070E+02
|
| Syy+:   9.509E+00     Syy-:   9.509E+00     Zyy :   1.490E+01
|
| Cw  :   3.931E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          62.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:       48.000
```

```
|
| Design Parameters                                              (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      4 LOC:    48.00)
|
|                            λ          λp         λr       CASE
|
| Flange: NonSlender       9.47       N/A      13.49     Table.4.1a.Case1
|
| Web   : Slender         53.54       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      4 LOC:    48.00)
|
|                            λ          λp         λr       CASE
|
| Flange: NonCompact       9.47       9.15     24.08     Table.4.1b.Case13
|
| Web   : Compact         53.54     ******    ******     Table.4.1.
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       422.2       0.000     Cl.D2          4      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  705.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       437.1       0.000     Cl.D2          4      0.00
```

```
|
| Intermediate Results :
|  Effective area            : Ae     =  14.100     in2        Eq.D3-1
|  Nom. Ten. Rpt Cap         : Pn     =  874.20     kip        Eq.D2-2
|-----------------------------------------------------------------------------
|                        CHECKS FOR AXIAL COMPRESSION
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             0.000       370.0       0.000     Cl.E3          4      0.00
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  5.8202                Cl.E2
|  Elastic Buckling Stress   : Fex    =  8449.2     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  49.876     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  617.87     kip        Eq.E7-1
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             0.000       352.1       0.000     Cl.E3          4      0.00
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  28.973                Cl.E2
|  Elastic Buckling Stress   : Fey    =  340.96     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  47.023     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  588.02     kip        Eq.E7-1
|-----------------------------------------------------------------------------
| FLEXURAL-TORSIONAL-BUCKLING
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|             0.000       357.9       0.000     Cl.E4          4      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe      =  498.48     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr     =  47.944     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn      =  597.69     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
13
                          STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                        ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                             CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             80.00       125.8       0.636     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv      =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv      =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx     =  210.01     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       144.2       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv      =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv      =  5.3400                Eq.G2-5
```

```
|
|  Nom. Shear Along Y        : Vny    =  216.30     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
14
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              1920.       446.1       4.304     Cl.F6.1        4     24.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  745.00     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       3177.       0.000     Cl.F3.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mnx    =  5306.1     kip-in     Eq.F3-1
|
|-----------------------------------------------------------------------------
|
| FLANGE LOCAL BUCK(Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              1920.       440.9       4.354     Cl.F6.2        4     24.00
|
|
|
| Intermediate Results :
|
|  Nom F-L-B Cap             : Mny    =  736.35     kip-in     Eq.F6-2
|
|-----------------------------------------------------------------------------
```

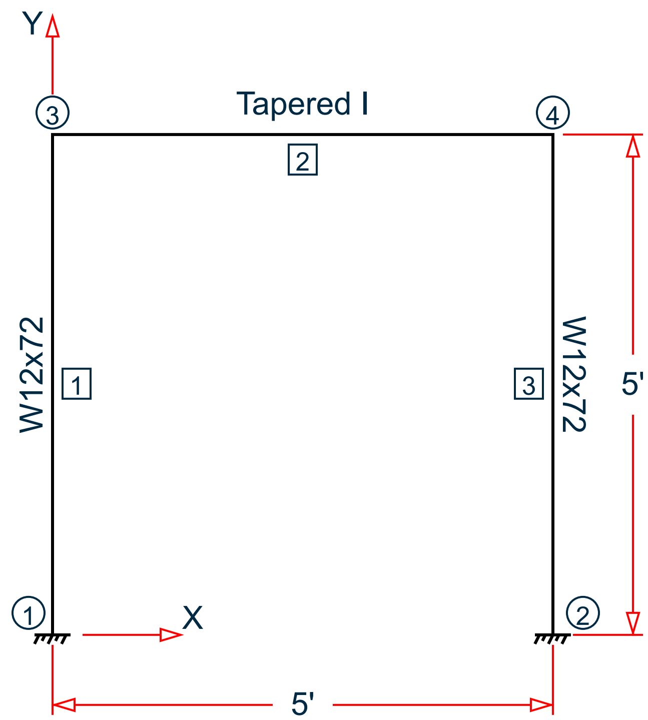
```
|
     STAAD SPACE                                              -- PAGE NO.
15
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           4.354      Eq.H1-1b           4       24.00
|
|
|
| Intermediate Results :
|
|  Modification Fact. for Cb : Cbf    =  1.0000               Eq.H1-2
|
|  Axial Capacity            : Pc     =  422.16     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  3177.3     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  440.93     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

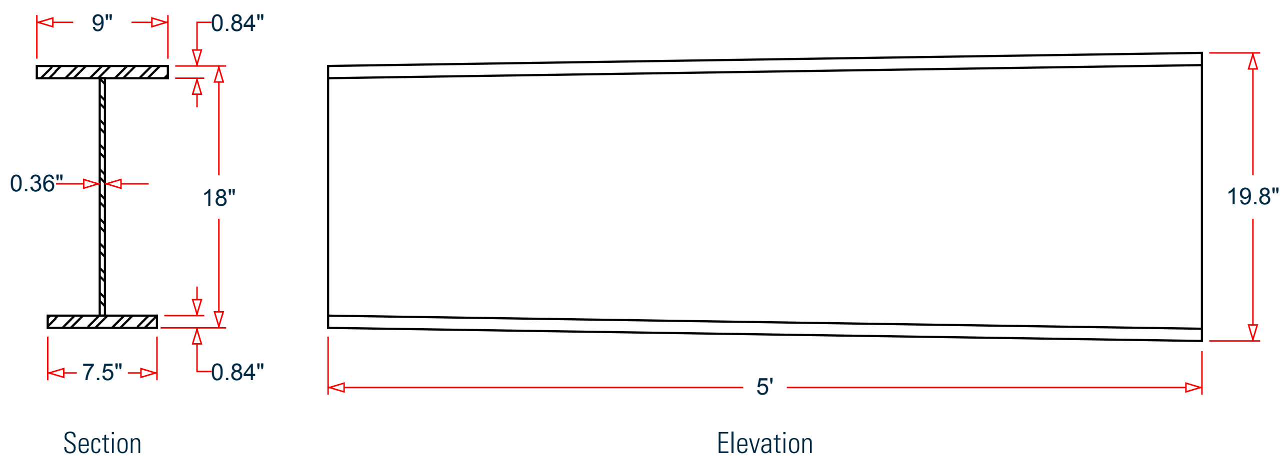
V. AISC 360-16 Tapered I Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered I section member per both the LRFD and ASD methods of the AISC 360-16 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





The following load cases are evaluated:

1. A uniformly distributed load of -3.5 k/ft along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. Lateral loads at the top of the right column of -50 kips in the global X direction and 45 kips in the global Z direction (out of plane).
4. A concreted load of -100 kips in the global Y direction and a concentrated torque of 9 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the left end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 32.26 in·kips, F_y = 8.755 kips
2. M_z = 677.6 in·kips, M_y = 2.942 in·kips, M_x = 5.106 in·kips, F_y = 22.49 kips, F_z = 0.098 kips, F_x = 24.70 kips
3. M_z = 275.9 in·kips, M_x = 4.497 in·kips, F_y = 50.23 kips, F_x = 5.473 kips
4. M_z = 654.5 in·kips, M_y = 5.298 in·kips, M_x = 9.192 in·kips, F_y = 22.50 kips, F_z = 0.177 kips, F_x = 24.35 kips

Section Properties

The section at the left end ($D = 18\ in.$) has the following properties:

Web height, $d_w = 18 - 2 \times 0.84 = 16.32\ in.$
Area, $A_g = (0.84)(9 + 7.5) + 16.32\ (0.36) = 19.74\ in^2$
Shear area in local y, $A_y = 16.32(0.36) = 5.875\ in^2$
Shear area in local x, $A_x = 0.84(9 + 7.5) = 13.86\ in^2$

Dist to centroid y, $c = \dfrac{7.5\left(0.84\right)\left(\frac{0.84}{2}\right) + 0.36\left(16.32\right)\left(0.84 + \frac{16.32}{2}\right) + 9\left(0.84\right)\left(18 - \frac{0.84}{2}\right)}{19.74} = 9.548\ in.$

Moment of inertia about major axis,

$$I_{zz} = \frac{7.5(0.84)^3}{12} + 7.5(0.84)\left(9.548 - \frac{0.84}{2}\right)^2 + \frac{0.36(16.32)^3}{12} + 0.36(16.32)\left(9.548 - \frac{16.32}{2} - 0.84\right)^2 + \frac{9(0.84)^3}{12} + 9(0.84)\left(18 - \frac{0.84}{2} - 9.548\right)^2 = 1,\ 147\ \text{in}^4$$

Moment of inertia about minor axis,

$$I_{yy} = \frac{0.84(9)^3}{12} + \frac{16.32(0.36)^3}{12} + \frac{0.84(7.5)^3}{12} = 80.62\ \text{in}^4$$

Elastic section modulus about major axis top flange fiber, $S_{xx,top} = \dfrac{I_{zz}}{D - c} = \dfrac{1,\ 146}{18 - 9.548} = 135.5\ in^3$

Elastic section modulus about major axis bottom flange fiber, $S_{xx,bottom} = \dfrac{I_{zz}}{c} = \dfrac{1,\ 146}{9.548} = 120.0\ in^3$

Elastic section modulus about minor axis, $S_{yy} = \dfrac{I_{yy}}{b_{f1}\ /\ 2} = \dfrac{80.62}{9\ /\ 2} = 17.92\ in^3$

Radius of gyration about major axis, $r_x = \sqrt{\dfrac{I_{zz}}{A_g}} = \sqrt{\dfrac{1,\ 147}{19.74}} = 7.619\ in$

Radius of gyration about minor axis, $r_y = \sqrt{\dfrac{I_{yy}}{A_g}} = \sqrt{\dfrac{80.62}{19.74}} = 2.021\ in$

Location of the plastic neutral axis, $y_{PNA} = 0.84 + \frac{19.74 - 2(7.5)(0.84)}{2 \times 0.36} = 10.75\ in.$

Plastic section modulus about major axis,

$$Z_{zt} = \frac{7.5(10.75)^2}{2} - \frac{(7.5 - 0.36)(10.75 - 0.84)^2}{2} + \frac{9(18 - 10.75)^2}{2} - \frac{(9 - 0.36)(18 - 0.84 - 10.75)^2}{2} = 141.8\ \text{in}^3$$

Plastic section modulus about minor axis, $Z_y = \frac{0.84(9)^2}{4} + \frac{16.32(0.36)^2}{4} + \frac{0.84(7.5)^2}{4} = 29.35\ in^3$

Torsional constant, $J = \frac{9(0.84)^3 + 16.32(0.36)^3 + 7.5(0.84)^3}{3} = 3.514\ in.^4$

<u>Section Classification</u>

Flange in compression

$$\lambda = \frac{b}{t} = \frac{1/2 \times 7.5}{0.84} = 4.464$$

$$k_c = \frac{4}{\sqrt{d_w / t_w}} = \frac{4}{\sqrt{\frac{16.32}{0.36}}} = 0.594$$

Per Table B4.1a, $\lambda < \lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.594(29,000)}{50}} = 11.88$, therefore flanges are non-slender for compression.

Web in compression

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table B4.1a, Case 5, $\lambda > \lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,000}{50}} = 35.88$, therefore the web is slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{b_f}{t_f} = \frac{9}{2(0.84)} = 5.36$$

Per Table 1.b, Case 11, $\lambda < \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29,000}{50}} = 9.15$, therefore flange is compact for bending.

Web in bending

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table 1.b, Case 15, $\lambda < \lambda_p = 3.76\sqrt{\frac{E}{F_y}} = 3.76\sqrt{\frac{29,000}{50}} = 90.55$, therefore web is compact for bending.

<u>Compression Capacity</u>

About X Axis:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{7.619} = 7.875 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,000}{50}} = 113.4$$

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,000}{(7.875)^2} = 4{,}615 \text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ex}} = \frac{50}{4{,}615} = 0.011 < 2.25$$

$$F_{crx} = 0.658^{F_y/F_{ex}} F_y = 0.658^{(0.011)} 50 = 49.77 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

Since the web is slender, use the effective area. The effective width imperfection adjustment factors (Table E7.1):

$c_1 = 0.18$

$c_2 = 1.31$

$$F_{el} = \left(c_2 \frac{\lambda_r}{\lambda}\right)^2 F_y = \left(1.31 \frac{35.88}{45.33}\right)^2 = 53.76 \text{ ksi} \qquad \text{(Eq. E7-5)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w \left(1 - c_1 \sqrt{\frac{F_{el}}{F_{crx}}}\right)\sqrt{\frac{F_{el}}{F_{crx}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{49.77}}\right)\sqrt{\frac{53.76}{49.77}} = 13.78 \text{ in.} \qquad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 13.78)0.36 = 18.84 \text{ in.}^2$$

$$P_{nx} = A_e \times F_{crx} = 18.84 \times 49.77 = 937.4 \text{ kips} \qquad \text{(Eq. E7-1)}$$

About Y Axis:

$$\frac{k_y L_y}{r_y} = \frac{1.0(60)}{2.021} = 29.69 < 4.71\sqrt{\frac{E}{F_y}} = 113.4$$

$$F_{ey} = \frac{\pi^2 E}{\left(\frac{k_y L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29,000}{(29.69)^2} = 324.7 \text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ey}} = \frac{50}{324.7} = 0.154 < 2.25$$

$$F_{cry} = 0.658^{(F_y/F_{ey})} F_y = 0.658^{(0.154)} 50 = 46.88 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w \left(1 - c_1 \sqrt{\frac{F_{el}}{F_{cry}}}\right)\sqrt{\frac{F_{el}}{F_{cry}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{46.88}}\right)\sqrt{\frac{53.76}{46.88}} = 14.11 \text{ in.} \qquad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 14.11)0.36 = 18.93 \text{ in.}^2$$

$$P_{ny} = A_e \times F_{cry} = 18.93 \times 46.88 = 887.4 \text{ kips} \quad \text{(Eq. E7-1)}$$

The Y axis governs; $P_n = P_{ny}$. The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 887.4 = 798.3\ kips$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 887.4\ /\ 1.67 = 531.4\ kips$

<u>Calculate Bending Capacity</u>

Bending capacity in plastic yielding about Y axis:

$$M_y = F_y \times Z_y = 50 \times 29.35 = 1{,}468 \text{ in·kips} > 1.6F_y \times S_y = 1.6\ (50)\ (17.92) = 1{,}434 \text{ in·kips}$$

Use $M_y = 1{,}434\ in \cdot kips$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}434 = 1{,}290\ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}434\ /\ 1.67 = 858.4\ in \cdot kips$

Bending capacity in plastic yielding about X axis:

Note: In the critical load case (LC4), the bending at the left end is *reverse* (i.e., “hogging”). Thus, the bottom flange is in compression; which has a lower elastic section modulus.

Calculate the bending strength reduction factor:

$$R_{pg} = 1 - \frac{a_w}{1,\ 200 + 300a_w}\left(\frac{h_c}{t_w} - 5.7\sqrt{\frac{E}{F_y}}\right) \le 1.0 \quad \text{(Eq. F5.6)}$$

where

h_c = Two times the distance from the center of gravity to the inside face of the compression flange. $2 \times (9.548 - 0.84) = 17.42\ in.$

a_w = $\frac{h_c t_w}{b_{fc} t_{fc}} = \frac{17.42 \times 0.36}{7.5 \times 0.84} = 0.995$

$$R_{pg} = 1 - \frac{0.995}{1,\ 200 + 300(0.995)}\left(\frac{17.42}{0.36} - 5.7\sqrt{\frac{29,\ 000}{50}}\right) = 1.01;\ \text{use } = 1.0$$

$$M_n = R_{pg} \times F_y \times S_{yc} = 1.0 \times 50 \times 120 = 6{,}000 \text{ in·kips}$$

The bending capacity for flexural yielding about the X axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 6{,}000 = 5{,}400\ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 5{,}000\ /\ 1.67 = 3{,}593\ in \cdot kips$

Lateral-torsional buckling about the X axis:

$$L_p = 1.1 r_t \sqrt{\frac{E}{F_y}} \quad \text{(Eq. F4-7)}$$

where

r_t = the effective radius of gyration for lateral-torsional buckling. For I-shapes with rectangular compression flanges:

$$= \frac{b_{fc}}{\sqrt{12\left(1 + \frac{1}{6}a_w\right)}}$$

Thus, $r_t = \frac{7.5}{\sqrt{12\left(1 + \frac{1}{6}0.995\right)}} = 2.005$ and $L_p = 1.1\left(2.005\right)\sqrt{\frac{29,000}{50}} = 53.12\ in.$. Therefore, lateral-torsional buckling must be checked.

$$L_r = 1.95 r_t \frac{E}{F_L}\sqrt{\frac{J}{S_{xc}h_o} + \sqrt{\left(\frac{J}{S_{xc}h_o}\right)^2 + 6.76\left(\frac{F_L}{E}\right)}} \qquad \text{(Eq. F4-8)}$$

where

F_L = $0.7F_y = 0.7\ (50) = 35\ ksi$ for $\frac{S_{xt}}{S_{xc}} = \frac{135.5}{120.0} = 1.13 > 0.7$

h_o = distance between flange centroids = $18 - 0.84 = 17.16\ in.$

$$L_r = 1.95(2.005)\frac{29,000}{35}\sqrt{\frac{3.514}{120 \times 17.16} + \sqrt{\left(\frac{3.514}{120 \times 17.16}\right)^2 + 6.76\left(\frac{35}{29,000}\right)^2}} = 235.4\text{ in.}$$

As $L_r < L_b = 60 < L_p$, the nominal moment capacity is:

$$M_n = C_b\left[R_{pc}M_{yc} - \left(R_{pc}M_{yc} - F_L\ S_{xc}\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le R_{pc}M_{yc} \qquad \text{(Eq. F4-2)}$$

where

C_b = 1.0
M_{yc} = $F_yS_{yc} = 50 \times 120 = 6,000\ ksi$
M_p = $F_yZ_x = 50 \times 141.8 = 7,090\ ksi \le 1.6\ F_yS_x = 9,600\ ksi$

$$M_n = 1.0\left\{1.182(6,000) - [1.182(6,000) - 35(120)]\left(\frac{60 - 53.12}{235.4 - 53.12}\right)\right\} = 6,983\text{ ksi}$$

The bending capacity for lateral-torsional buckling:

The ultimate bending capacity (LRFD) = $\phi_bM_y = 0.9 \times 6,983 = 6,285\ in \cdot kips$
The allowable bending capacity (ASD) = $M_y/\Omega_b = 6,983\ /\ 1.67 = 4,181\ in \cdot kips$

<u>Compression Flange Yielding</u>

$$M_n = R_{pc} \times M_{yc} \qquad \text{(Eq. F4-1)}$$

where

I_{yc} = moment of inertia of the compression flange about the Y axis $= \frac{0.84(7.5)^3}{12} = 29.53\ in.^4$

M_{yc} = $M_y \times S_{xc} = M_y \times S_{xx,top} = 50 \times 135.5 = 6,775\ in \cdot kips$

R_{pc} = the web plastification factor, $= \frac{M_p}{M_{yc}} = \frac{7,090}{6,775} = 1.046$ when $I_{yc}\ /\ I_y = {}^{29.53}\!/_{80.62} = 0.366 > 0.23$ and $\frac{h_c}{t_w} = \frac{17.42}{0.36} = 48.38 < \lambda_{pw} = 90.55$ (limiting slenderness for a compact web per Table 1b case 15)(Eq. F4-9a)

$$M_n = 1.046 \times 6,775 = 7,087\text{ in} \cdot \text{kips}$$

The bending capacity for compression flange yielding:

The ultimate bending capacity (LRFD) = $\phi_bM_n = 0.9 \times 7,087 = 6,378\ in \cdot kips$
The allowable bending capacity (ASD) = $M_n/\Omega_b = 7,087\ /\ 1.67 = 4,244\ in \cdot kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 2:

$$\frac{P_r}{P_c} = \frac{24.7}{798.3} = 0.031 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{24.7}{531.4} = 0.046 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.031}{2} + \left(\frac{677.6}{6{,}378} + \frac{2.942}{1,\ 321}\right) = 0.124 \text{ (LRFD)}$$

$$\frac{0.046}{2} + \left(\frac{677.6}{4{,}244} + \frac{2.942}{878.7}\right) = 0.188 \text{ (ASD)}$$

Results

Table 738:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	798.3	798.8	negligible	
	Major axis bending capacity (in·kips)	6,378	6,380	negligible	
	Minor axis bending capacity (in·kips)	1,290	1,290	none	
	Interaction ratio	0.124	0.124	none	
ASD	Compression capacity (kips)	531.4	531.48	negligible	
	Major axis bending capacity (in·kips)	4,244	4,245	negligible	
	Minor axis bending capacity (in·kips)	858.4	858.28	negligible	
	Interaction ratio	0.188	0.186	negligible	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Tapered I Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 0 0; 4 5 5 0;
MEMBER INCIDENCES
1 1 2; 2 2 4; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY
2 TAPERED 1.5 0.03 1.65 0.75 0.07 0.625 0.07
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -3.5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 4 LOADTYPE None  TITLE LOAD CASE 4
JOINT LOAD
4 FX -50 FZ 45
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CON GY -100
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2016
BEAM 1 MEMB 2
FYLD 50 ALL
FU 60 ALL
```

```
MAIN 200 MEMB 2
METHOD LRFD
STP 2 MEMB 2
TMAIN 300 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2016
BEAM 1 MEMB 2
FYLD 50 ALL
FU 60 ALL
MAIN 200 MEMB 2
METHOD ASD
STP 2 MEMB 2
TMAIN 300 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
 |-----------------------------------------------------------------------------
|
|  Member No:        2       Profile:      TAPERED            (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.315       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.G1
|
|  Pz:       5.473     C     Vy:        50.02           Vx:     -.5769E-06
|
|  Tz:      -4.497           My:       0.1731E-04       Mx:      275.9
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     29.685
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.315(PASS)
|
|          Loc  :    0.00            Condition :    Cl.G1
```

```
|
|--------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.974E+01     Axx :   1.386E+01     Ayy :   5.875E+00
|
| Ixx :   1.146E+03     Iyy :   8.062E+01     J   :   3.514E+00
|
| Sxx+:   1.355E+02     Sxx-:   1.200E+02     Zxx :   1.418E+02
|
| Syy+:   1.792E+01     Syy-:   1.792E+01     Zyy :   2.935E+01
|
| Cw  :   5.508E+03     x0  :   0.000E+00     y0  :   0.000E+00
|
|--------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          60.000
|
|--------------------------------------------------------------------------
|
| Actual Member Length:        60.000
|
| Design Parameters                                         (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|--------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:     0.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       5.36       N/A      11.88     Table.4.1a.Case2
|
| Web   : Slender         45.33       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:     0.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          5.36       9.15     21.08     Table.4.1b.Case11
|
| Web   : Compact         42.29     126.82    137.27     Table.4.1b.Case15
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
9
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
 |--------------------------------------------------------------------------
```

```
|                          CHECKS FOR AXIAL TENSION
|-----------------------------------------------------------------------------
| TENSILE YIELDING
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              0.000       888.1       0.000     Cl.D2          1      0.00
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  986.76     kip        Eq.D2-1
|-----------------------------------------------------------------------------
| TENSILE RUPTURE
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              0.000       888.1       0.000     Cl.D2          1      0.00
|
| Intermediate Results :
|  Effective area            : Ae     =  19.735     in2        Eq.D3-1
|  Nom. Ten. Rpt Cap         : Pn     =  1184.1     kip        Eq.D2-2
|-----------------------------------------------------------------------------
|                          CHECKS FOR AXIAL COMPRESSION
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|              24.70       843.0       0.029     Cl.E3          2      0.00
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  7.8750                Cl.E2
|  Elastic Buckling Stress   : Fex    =  4615.2     ksi        Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  49.774     ksi        Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  936.63     kip        Eq.E7-1
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.70       798.8       0.031     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  29.685               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  324.80     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  46.880     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  887.57     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.70       798.8       0.031     Cl.E4          2      0.00
|
|
|
| Intermediate Results :
|
|  Constant                 : H      =  1.0000               Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  389.10     ksi        Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  324.80     ksi        Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  46.880     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  887.57     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                         -- PAGE NO.
10
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
```

```
|
|            0.1766       374.2       0.000     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  415.80     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             50.02       158.6       0.315     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  176.26     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
11
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                    - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -5.298       1290.       0.004     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1433.3     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
```

```
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             654.5       6282.       0.104     Cl.F4.2        4      0.00
|
|
| Intermediate Results :
|  Nom L-T-B Cap              : Mnx    =  6980.3     kip-in     Eq.F4-2
|  Mom. Distr. factor         : CbX    =  1.0000                Custom
|  Limiting Unbraced Length   : LpX    =  53.119     in         Eq.F4-7
|  Limiting Unbraced Length   : LrX    =  235.38     in         Eq.F4-8
|  Aw Ratio                   : AwX    = 0.99518                Eq.F4-12
|  Nom. Compr. Flange Stress  : FLX    =  35.000     ksi        Eq.F4-6a
|  Compr. Flange Yield Mom.   : MycX   =  5999.4     kip-in     Eq.F4-4
|  Web Plastification Factor  : RpcX   =  1.1817                Eq.F4-9a
|
|-----------------------------------------------------------------------------
|
| COM FLANGE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -677.6       6380.       0.106     Cl.F4.1        2      0.00
|
|
| Intermediate Results :
|  Nom comp. Flange Yld       : Mny    =  7089.4     kip-in     Eq.F4-1
|  Compr Flange Yld Mom.      : Myc    =  6777.0     kip-in     Eq.F4-4
|  Web Plastification Factor  : Rpc    =  1.0461                Eq.F4-9a
|
|-----------------------------------------------------------------------------
|
| TEN FLANGE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            -677.6       6380.       0.106     Cl.F4.4        2      0.00
|
|
| Intermediate Results :
|  Nom Tens. Flange Yielding  : Mny    =  7089.4     kip-in     Eq.F4-15
```

```
|
|  Tens. Flange Yield Moment : Myt    =  5999.4     kip-in     Eq.F4-4
|
|  Web Plastification Factor : Rpt    =  1.1817                Eq.F4-16a
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.124      Eq.H1-1b           2        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  798.81     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  6380.4     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  1290.0     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
```

```
      Vy              FY          Shear force along minor axis.
      Vx              FZ          Shear force along major axis.
      Tz              MX          Torsional moment.
      My              MY          Bending moment about minor axis.
      Mx              MZ          Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
    56. PARAMETER 2
    57. CODE AISC UNIFIED 2016
    58. BEAM 1 MEMB 2
    59. FYLD 50 ALL
    60. FU 60 ALL
    61. MAIN 200 MEMB 2
    62. METHOD ASD
    63. STP 2 MEMB 2
    64. TMAIN 300 MEMB 2
    65. TRACK 2 MEMB 2
    66. CHECK CODE MEMB 2
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
      STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
 |------------------------------------------------------------------------------
 |
 |  Member No:        2       Profile:      TAPERED             (AISC
SECTIONS)|
```

```
 |  Status:        PASS       Ratio:        0.474       Loadcase:       3
 |
 |  Location:      0.00       Ref:      Cl.G1
 |
 |  Pz:       5.473     C     Vy:        50.02           Vx:     -.5769E-06
 |
 |  Tz:      -4.497           My:       0.1731E-04       Mx:      275.9
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     29.685
 |
 | Allowable Slenderness Ratio :    200.000            LOC  :     0.00
 |
 |-----------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      3             Ratio     :        0.474(PASS)
 |
 |          Loc  :    0.00            Condition :    Cl.G1
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   1.974E+01     Axx :   1.386E+01     Ayy :   5.875E+00
 |
 | Ixx :   1.146E+03     Iyy :   8.062E+01     J   :   3.514E+00
 |
 | Sxx+:   1.355E+02     Sxx-:   1.200E+02     Zxx :   1.418E+02
 |
 | Syy+:   1.792E+01     Syy-:   1.792E+01     Zyy :   2.935E+01
 |
 | Cw  :   5.508E+03     x0  :   0.000E+00     y0  :   0.000E+00
 |
 |-----------------------------------------------------------------------------
 |
 | MATERIAL PROPERTIES
 |
 | Fyld:          50.000              Fu:          60.000
 |
 |-----------------------------------------------------------------------------
 |
 | Actual Member Length:        60.000
 |
 | Design Parameters                                           (Built-Up)
 |
 | Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      2 LOC:     0.00)
 |
 |                            λ         λp        λr       CASE
 |
```

```
| Flange: NonSlender       5.36       N/A      11.88     Table.4.1a.Case2
|
| Web   : Slender         45.33       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:     0.00)
|
|                          λ         λp        λr       CASE
|
| Flange: Compact          5.36       9.15     21.08     Table.4.1b.Case11
|
| Web   : Compact         42.29     126.82    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                 -- PAGE NO.
15
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                             CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       590.9       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap          : Pn      =  986.76     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       592.1       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective area             : Ae      =  19.735     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap          : Pn      =  1184.1     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
```

```
                         CHECKS FOR AXIAL COMPRESSION

|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING X
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             24.70       560.9       0.044     Cl.E3          2      0.00
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  7.8750               Cl.E2
|  Elastic Buckling Stress   : Fex    =  4615.2     ksi       Eq.E3-4
|  Crit. Buckling Stress     : Fcrx   =  49.774     ksi       Eq.E3-2
|  Nom. Flexural Buckling    : Pnx    =  936.63     kip       Eq.E7-1
|-----------------------------------------------------------------------------
| FLEXURAL BUCKLING Y
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             24.70       531.5       0.046     Cl.E3          2      0.00
|
| Intermediate Results :
|  Effective Slenderness     : Lcy/ry =  29.685               Cl.E2
|  Elastic Buckling Stress   : Fey    =  324.80     ksi       Eq.E3-4
|  Crit. Buckling Stress     : Fcry   =  46.880     ksi       Eq.E3-2
|  Nom. Flexural Buckling    : Pny    =  887.57     kip       Eq.E7-1
|-----------------------------------------------------------------------------
| FLEXURAL-TORSIONAL-BUCKLING
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|             24.70       531.5       0.046     Cl.E4          2      0.00
|
| Intermediate Results :
|  Constant                  : H      =  1.0000               Eq.E4-8
|  Euler Buckling Stress     : Fez    =  389.10     ksi       Eq.E4-7
```

```
|  Elastic F-T-B Stress      : Fe     =  324.80     ksi        Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  46.880     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  887.57     kip        Eq.E7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
16
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                               - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.1766       249.0       0.001     Cl.G1          4      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  415.80     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              50.02       105.5       0.474     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400                Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  176.26     kip        Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
17
```

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                              - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -5.298       858.3       0.006     Cl.F6.1        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1433.3     kip-in     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              654.5       4180.       0.157     Cl.F4.2        4      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  6980.3     kip-in     Eq.F4-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  53.119     in         Eq.F4-7
|
|  Limiting Unbraced Length  : LrX    =  235.38     in         Eq.F4-8
|
|  Aw Ratio                  : AwX    = 0.99518                Eq.F4-12
|
|  Nom. Compr. Flange Stress : FLX    =  35.000     ksi        Eq.F4-6a
|
|  Compr. Flange Yield Mom.  : MycX   =  5999.4     kip-in     Eq.F4-4
|
|  Web Plastification Factor : RpcX   =  1.1817                Eq.F4-9a
|
|-----------------------------------------------------------------------------
|
| COM FLANGE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -677.6       4245.       0.160     Cl.F4.1        2      0.00
|
```

```
||
||
|| Intermediate Results :
||
||  Nom comp. Flange Yld      : Mny    =  7089.4     kip-in     Eq.F4-1
||
||  Compr Flange Yld Mom.     : Myc    =  6777.0     kip-in     Eq.F4-4
||
||  Web Plastification Factor : Rpc    =  1.0461                Eq.F4-9a
||
||-----------------------------------------------------------------------------
||
|| TEN FLANGE YIELDING
||
||              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             -677.6       4245.       0.160     Cl.F4.4        2      0.00
||
||
||
|| Intermediate Results :
||
||  Nom Tens. Flange Yielding : Mny    =  7089.4     kip-in     Eq.F4-15
||
||  Tens. Flange Yield Moment : Myt    =  5999.4     kip-in     Eq.F4-4
||
||  Web Plastification Factor : Rpt    =  1.1817                Eq.F4-16a
||
||-----------------------------------------------------------------------------
||
      STAAD SPACE                                                -- PAGE NO.
18
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
||-----------------------------------------------------------------------------
||
||                          CHECKS FOR AXIAL BEND INTERACTION
||
||-----------------------------------------------------------------------------
||
|| COMBINED FORCES CLAUSE H1
||
||                            RATIO      CRITERIA          L/C      LOC
||
||                            0.186      Eq.H1-1b           2        0.00
||
||
||
|| Intermediate Results :
||
||  Axial Capacity            : Pc     =  531.48     kip        Cl.H1.1
||
||  Moment Capacity           : Mcx    =  4245.1     kip-in     Cl.H1.1
||
||  Moment Capacity           : Mcy    =  858.28     kip-in     Cl.H1.1
||
```

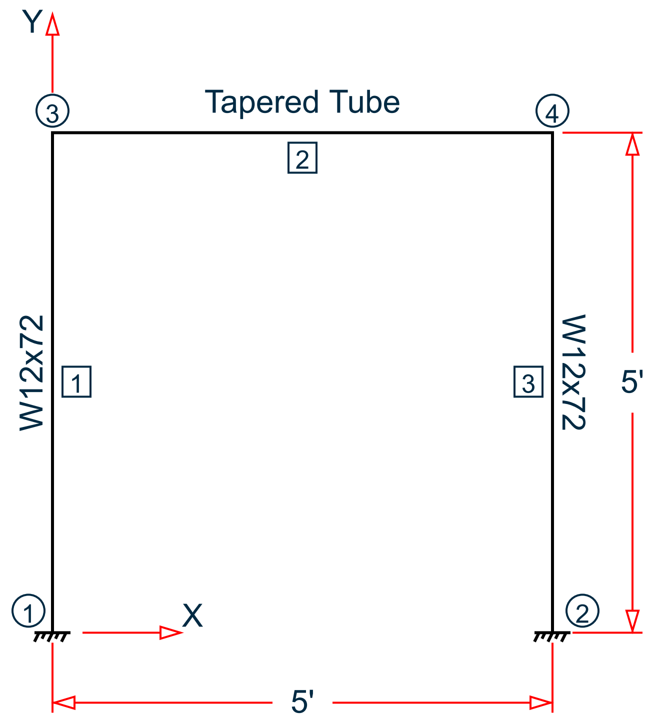
```
|-----------------------------------------------------------------------------
|
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
```

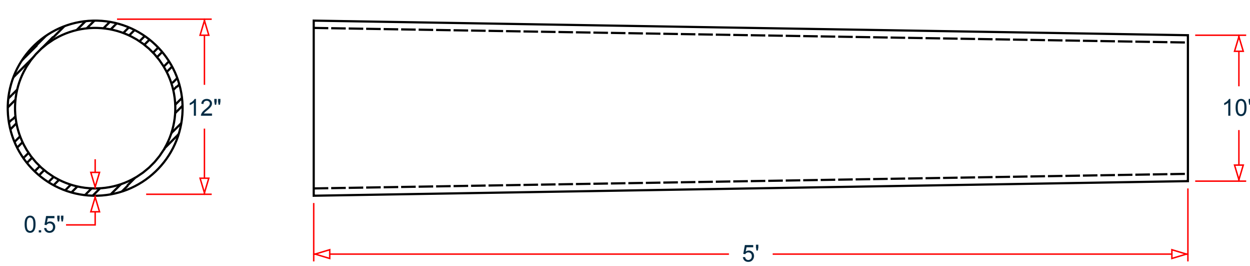
V. AISC 360-16 Tapered Tube Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered tube section member per both the LRFD and ASD methods of the AISC 360-16 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





Section

Elevation

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concreted torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the right end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

$M_z = 505.66\ in \cdot kips$
$F_y = 67.2\ kips$
$F_x = 10.23\ kips$

Section Properties

The section at the right end ($OD = 10\ in.$) has the following properties:

Inner diameter, ID = 9 in.
Area, $A_g = (\pi/4)(OD^2 - ID^2) = 14.92\ in^2$
Form factor = $0.49 + 0.8\ (t\ /\ OD) = 0.53$
Shear area, $A_y = 14.92(0.53) = 7.909\ in^2$
Moment of inertia,

$$I = \frac{\pi}{64}\left(\mathrm{OD}^4\text{-}\mathrm{ID}^4\right) = 168.8\ \mathrm{in}^4$$

Elastic section modulus, $S = \frac{\pi}{32}\left(\frac{OD^4\text{-}ID^4}{OD}\right) = 33.76\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{168.8}{14.92}} = 3.363\ in$

Plastic section modulus, $Z = \frac{OD^3\text{-}ID^3}{6} = 45.17\ in^3$

HSS Torsional constant, $C = \frac{\pi}{2}(OD\text{-}t)^2 t = 70.88\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{\mathrm{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1a, Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 63.8$, therefore flanges are non-slender for compression. The web is similarly non-slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{\mathrm{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1b, Case20, $\lambda < \lambda_r = 0.07\frac{E}{F_y} = 40.6$, therefore flange is compact for bending.

Compression Capacity

$$\frac{kL}{r} = \frac{1.0(60)}{3.363} = 17.84 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

$$F_e = \frac{\pi^2 E}{\left(\frac{kL}{r}\right)^2} = \frac{\pi^2 \times 29{,}000}{(17.84)^2} = 899.4 \text{ ksi} \quad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{899.4} = 0.056 < 2.25$$

$$F_{\text{crx}} = 0.658^{F_y / F_e} F_y = 0.658^{(0.056)} 50 = 48.85 \text{ ksi} \quad \text{(Eq. E3-2)}$$

$$P_n = A_g \times F_{cr} = 14.92 \times 48.85 = 729.0 \text{ kips} \quad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 729.0 = 656.1$ *kips*
The allowable compression capacity (ASD) = $P_n / \Omega_c = 729.0 / 1.67 = 436.5$ *kips*

Shear Capacity

The distance from the maximum shear force (ends) to zero shear force (mid-span), $L_v = L / 2 = 30$ *in.*

The critical shear stress, F_{cr}, *is the minimum of*:

$$F_{\text{cr1}} = \frac{1.60E}{\sqrt{\frac{L_v}{\text{OD}}} \left(\frac{\text{OD}}{t}\right)^{5/4}} = 633.4 \text{ ksi}$$

$$F_{\text{cr2}} = \frac{0.78E}{\left(\frac{\text{OD}}{t}\right)^{3/2}} = 252.9 \text{ ksi}$$

$$F_{\text{cr3}} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30$ *ksi*

Nominal shear capacity:

$$V_n = F_{\text{cr}} A_g / 2 = 223.8 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_c V_n = 0.9 \times 223.8 = 201.5$ *kips*
The allowable shear capacity (ASD) = $V_n / \Omega_c = 223.8 / 1.67 = 134.0$ *kips*

Calculate Bending Capacity

Bending capacity in plastic yielding:

$$M_p = F_y \times Z = 50 \times 45.17 = 2{,}259 \text{ in·kips} < 1.6F_y \times S = 1.6\,(50)\,(33.76) = 2{,}701 \text{ in·kips}$$

The bending capacity for flexural yielding:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 2{,}259 = 2{,}033$ *in·kips*
The allowable bending capacity (ASD) = $M_y / \Omega_b = 2{,}259 / 1.67 = 1{,}352$ *in·kips*

Torsional Capacity

The critical stress, F_{cr}, *is the larger of*:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{OD}}\left(\frac{OD}{t}\right)^{5/4}} = 344.3 \text{ ksi}$$

$$F_{cr2} = \frac{1.23E}{\left(\frac{OD}{t}\right)^{3/2}} = 194.5 \text{ ksi}$$

But shall not exceed:

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30\ ksi$

Nominal torsional capacity:

$$T_n = F_{cr}C = 2{,}126 \text{ in}\cdot\text{kips}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_c T_n = 0.9 \times 2{,}126 = 1{,}914\ kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_c = 2{,}126\ /\ 1.67 = 1{,}273\ kips$

<u>Interaction Ratio for Bending and Compression</u>

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{10.23}{656.1} = 0.016 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{10.23}{436.5} = 0.023 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.016}{2} + \left(\frac{505.7}{2{,}033} + \frac{0}{2{,}033}\right) = 0.257 \text{ (LRFD)}$$

$$\frac{0.023}{2} + \left(\frac{505.7}{1{,}352} + \frac{0}{1{,}352}\right) = 0.386 \text{ (ASD)}$$

Results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	656.1	656.1	none	
	Shear capacity (kips)	201.5	201.5	none	
	Bending capacity (in·kips)	2,033	2,033	none	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Torsional capacity (in·kips)	1,914	1,914	none	
	Interaction ratio	0.257	0.257	none	
ASD	Compression capacity (kips)	436.5	436.5	none	
	Shear capacity (kips)	134.0	134	none	
	Bending capacity (in·kips)	1,352	1,352	none	
	Torsional capacity (in·kips)	1,273	1,273	none	
	Interaction ratio	0.386	0.386	none	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 Tapered Tube Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 PRIS ROUND STA 12 END 10 THI 0.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
|-----------------------------------------------------------------------------
|
|  Member No:        2       Profile:      PRISMAT             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.334       Loadcase:        1
|
|  Location:     60.00       Ref:      Cl.G1
|
|  Pz:       10.23     C     Vy:        -67.19          Vx:      0.000
```

```
 Tz:       0.000           My:        0.000           Mx:      505.7
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     17.839
|
| Allowable Slenderness Ratio :    200.000           LOC  :    60.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      1             Ratio     :        0.334(PASS)
|
|          Loc  :   60.00            Condition :    Cl.G1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:    60.00, PROPERTIES UNIT: IN  )
|
| Ag  :   1.492E+01     Axx :   7.909E+00     Ayy :   7.909E+00
|
| Ixx :   1.688E+02     Iyy :   1.688E+02     J   :   3.376E+02
|
| Sxx+:   3.376E+01     Sxx-:   3.376E+01     Zxx :   4.517E+01
|
| Syy+:   3.376E+01     Syy-:   3.376E+01     Zyy :   4.517E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          60.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        60.000
|
| Design Parameters                                             (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
|
|                            λ          λp         λr        CASE
|
| Flange: NonSlender      20.00       N/A      63.80     Table.4.1a.Case9
|
| Web   : NonSlender      20.00       N/A      63.80     Table.4.1a.Case9
|
```

```
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
|
|                          λ         λp        λr       CASE
|
| Flange: Compact         20.00      40.60    179.80     Table.4.1b.Case20
|
| Web   : Compact         20.00      40.60    179.80     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       671.5       0.000     Cl.D2          1     60.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  746.13     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       537.2       0.000     Cl.D2          1     60.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  11.938     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  716.28     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
```

```
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              23.86       656.1       0.036     Cl.E3          2     60.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  17.839                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  899.40     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  48.850     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  728.97     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              23.86       656.1       0.036     Cl.E3          2     60.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  17.839                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  899.40     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  48.850     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  728.97     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
9
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.5390E-01   201.5       0.000     Cl.G1          2     60.00
|
|
```

```
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  223.84     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  30.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              67.19       201.5       0.334     Cl.G1          1     60.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  223.84     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  30.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
10
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              505.7       2033.       0.249     Cl.F8.1        1     60.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  2258.3     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             -1.617       2033.       0.001     Cl.F8.1        2     60.00
|
|
```

```
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  2258.3     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.257      Eq.H1-1b           1       60.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  656.07     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  2032.5     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  2032.5     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
12
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C     LOC
|
|              238.5       1914.       0.125     Cl.H3.1        2     60.00
|
|
|
| Intermediate Results :
```

```
|
|  Crit. Stress. Fcr         : Fcr    =  30.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength             : Tn     =  2126.5     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
```

```
    T-F-Y = Tension Flange Yielding
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2016
    54. SLF 0.8 MEMB 2
    55. FYLD 50 ALL
    56. FU 60 ALL
    57. METHOD ASD
    58. STP 2 MEMB 2
    59. TRACK 2 MEMB 2
    60. CHECK CODE MEMB 2
 PARAMETER 2
      STAAD SPACE                                                  -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
      STAAD SPACE                                                  -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
 |------------------------------------------------------------------------------
 |
 |  Member No:        2       Profile:      PRISMAT             (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.501       Loadcase:        1
 |
 |  Location:     60.00       Ref:      Cl.G1
 |
 |  Pz:       10.23     C     Vy:       -67.19           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:      505.7
 |
 |------------------------------------------------------------------------------
 |
 | COMPRESSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     17.839
 |
 | Allowable Slenderness Ratio :    200.000            LOC  :    60.00
 |
 |------------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      1             Ratio     :        0.501(PASS)
 |
 |         Loc  :   60.00            Condition :    Cl.G1
 |
 |------------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:    60.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   1.492E+01     Axx :   7.909E+00     Ayy :   7.909E+00
 |
 | Ixx :   1.688E+02     Iyy :   1.688E+02     J   :   3.376E+02
 |
```

```
| Sxx+:   3.376E+01     Sxx-:   3.376E+01     Zxx :   4.517E+01
|
| Syy+:   3.376E+01     Syy-:   3.376E+01     Zyy :   4.517E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          60.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        60.000
|
| Design Parameters                                          (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender      20.00       N/A      63.80     Table.4.1a.Case9
|
| Web   : NonSlender      20.00       N/A      63.80     Table.4.1a.Case9
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact         20.00      40.60    179.80     Table.4.1b.Case20
|
| Web   : Compact         20.00      40.60    179.80     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                           -- PAGE NO.
15
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                      - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|              0.000       446.8       0.000     Cl.D2           1     60.00
|
|
|
| Intermediate Results :
|  Nom. Ten. Yld Cap         : Pn     =  746.13     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              0.000       358.1       0.000     Cl.D2           1     60.00
|
|
|
| Intermediate Results :
|  Effective area            : Ae     =  11.938     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  716.28     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              23.86       436.5       0.055     Cl.E3           2     60.00
|
|
|
| Intermediate Results :
|  Effective Slenderness     : Lcx/rx =  17.839                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  899.40     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  48.850     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  728.97     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              23.86       436.5       0.055     Cl.E3           2     60.00
|
|
|
```

```
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  17.839                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  899.40     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  48.850     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  728.97     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
16
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.5390E-01   134.0       0.000     Cl.G1          2     60.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along X        : Vnx    =  223.84     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along X  : Fcrx   =  30.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             67.19       134.0       0.501     Cl.G1          1     60.00
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  223.84     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  30.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
17
```

```
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (X)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             505.7       1352.       0.374     Cl.F8.1        1     60.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom Flex Yielding Along X : Mnx    =  2258.3     kip-in     Eq.F8-1
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (Y)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |            -1.617       1352.       0.001     Cl.F8.1        2     60.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom Flex Yielding Along Y : Mny    =  2258.3     kip-in     Eq.F8-1
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                              -- PAGE NO.
18
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR AXIAL BEND INTERACTION
 |
 |-----------------------------------------------------------------------------
 |
 | COMBINED FORCES CLAUSE H1
 |
 |                           RATIO      CRITERIA          L/C      LOC
 |
 |                           0.386      Eq.H1-1b           1       60.00
 |
 |
 |
```

```
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  436.51     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  1352.3     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  1352.3     kip-in     Cl.H1.1
|
|-------------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
19
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
|-------------------------------------------------------------------------------
|
|                          CHECKS FOR TORSION
|
|-------------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             238.5       1273.       0.187     Cl.H3.1        2     60.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr        : Fcr    =  30.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength            : Tn     =  2126.5     kip-in     Eq.H3-1
|
|-------------------------------------------------------------------------------
|
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
```

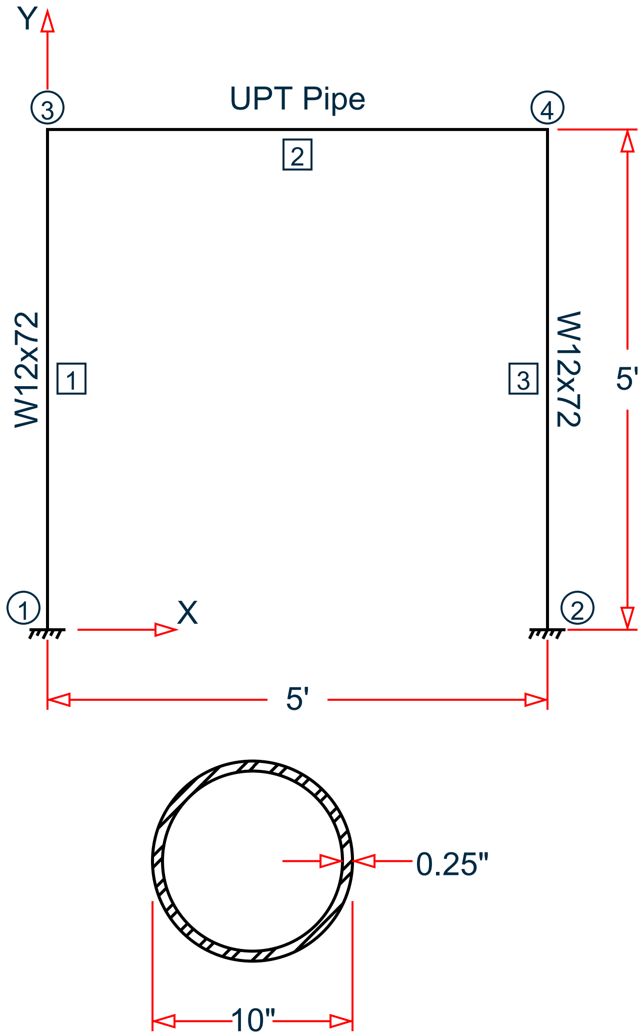
```
      Pz               FX         Axial force.
      Vy               FY         Shear force along minor axis.
      Vx               FZ         Shear force along major axis.
      Tz               MX         Torsional moment.
      My               MY         Bending moment about minor axis.
      Mx               MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase 
that
       produced the worst case design ratio for flexure/compression Capacity 
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the 
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), 
while
       the results for shear are reported as being for shear forces �along� 
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear 
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
```

V. AISC 360-16 UPT Pipe Section

Verify the axial compression capacity, shear capacity, flexure capacity, torsional capacity, and interaction ratio of a user-provided pipe section member per both the LRFD and ASD methods of the AISC 360-16 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT pipe member for the beam. The pipe outer diameter, D, is 10" and the thickness, t, is 0.25".



Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner diameter, $d = 10 - 2 \times 0.25 = 9.5$ in.
Area, $A_g = \frac{\pi}{4}\left(D^2 - d^2\right) = 7.66\ in^2$
Shear area, $A_y = A_x = A_g\ /\ 2 = 3.83\ in^2$
Moment of inertia, $I_x = I_y = \frac{\pi}{64}\left(D^4 - d^4\right) = 91.05\ in^4$
Elastic section modulus, $S_{xx} = \frac{I_{xx}}{D\ /\ 2} = \frac{91.05}{10\ /\ 2} = 18.21\ in^3$
Radius of gyration, $r_x = \sqrt{\frac{I_{xx}}{A_g}} = \sqrt{\frac{91.05}{7.66}} = 3.448\ in$
Plastic section modulus, $Z_x = \frac{1}{6}\left(D^3 - d^3\right) = 23.77\ in^3$
HSS Torsional constant, $C = \frac{\pi}{2}(D - t)^2 t = 37.33\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{D}{t} = \frac{10}{0.25} = 40$$

Per Table 4.1a Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 63.8$, therefore flanges are non-slender for compression.

Web in compression

Per Table 4.1a, Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 63.8$, therefore the web is non-slender for compression.

Flexure classification:

Per Table 4.1b, Case 20, $\lambda < \lambda_p = 0.07\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 40.6$, therefore flange is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.448} = 17.40 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(17.40)^2} = 945\ \text{ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{945} = 0.053 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y/F_e} F_y = 0.658^{(0.053)} 50 = 48.91 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 7.66 \times 48.91 = 374.5 \text{ kips} \quad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 374.5 = 337.0$ *kips*
The allowable compression capacity (ASD) = $P_n / \Omega_c = 374.5 / 1.67 = 224.3$ *kips*

Calculate Shear Capacity

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{D}}\left(\frac{D}{t}\right)^{\left(5/4\right)}} = 144.8 \text{ ksi}$$

$$F_{cr2} = \frac{0.78E}{\left(\frac{D}{t}\right)^{\left(3/2\right)}} = 89.4 \text{ ksi}$$

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

So, $F_{cr} = 30$ *ksi*.

The nominal shear strength:

$$V_n = F_{cr} \times A_x = 30 \times 3.83 = 114.9 \text{ kips}$$

The axial capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 114.9 = 103.4$ *kips*
The allowable shear capacity (ASD) = $V_n / \Omega_v = 114.9 / 1.67 = 68.78$ *kips*

Calculate Bending Capacity

Bending capacity in plastic yielding (compact section):

$$M_y = F_y \times Z_x = 50 \times 23.77 = 1{,}189 \text{ in} \cdot \text{kips} \quad \text{(Eq F8-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}189 = 1{,}070$ *in·kips*
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}189 / 1.67 = 711.7$ *in·kips*

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.75}{337.0} = 0.034 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.75}{224.3} = 0.052 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \le 1.0$.

$$\frac{0.034}{2} + \left(\frac{599.6}{1{,}070} + \frac{0}{1{,}070}\right) = 0.578 \text{ (LRFD)}$$

$$\frac{0.052}{2} + \left(\frac{599.6}{711.7} + \frac{0}{711.7}\right) = 0.869 \text{ (ASD)}$$

Calculate Torsion Capacity

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}. From shear calculations, $F_{cr} = 30\ ksi$.

The nominal shear strength:

$$T_n = F_{cr} \times C = 30 \times 37.33 = 1{,}120 \text{ in·kips}$$

The axial capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1{,}120 = 1{,}008\ in·kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 114.9\ /\ 1.67 = 670.6\ in·kips$

Results

Table 739:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	337	337	none	
	Bending capacity (in·kips)	1,070	1,070	none	
	Shear capacity (kips)	103.4	103.4	none	
	Interaction ratio	0.578	0.578	none	
	Torsional capacity (in·kips)	1,008	1,008	none	
ASD	Compression capacity (kips)	224.3	224.3	none	
	Bending capacity (in·kips)	711.7	711.7	none	
	Shear capacity (kips)	68.78	68.78	none	
	Interaction ratio	0.869	0.869	none	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Torsional capacity (in·kips)	670.6	670.6	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 UPT Pipe Section.STD` is typically installed with the program.

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded pipe member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
PIPE
PIPE
10 9.5 0 0
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 PIPE
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
```

```
       Tz               MX         Torsional moment.
       My               MY         Bending moment about minor axis.
       Mx               MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
 |------------------------------------------------------------------------------
 |
 |  Member No:        2       Profile:  ST  PIPE                (UPT)
 |
 |  Status:        PASS       Ratio:         0.653       Loadcase:       1
 |
 |  Location:      0.00       Ref:      Cl.G1
 |
 |  Pz:       11.75     C     Vy:        67.50           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:      599.6
 |
 |------------------------------------------------------------------------------
 |
 | COMPRESSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :     17.400
 |
 | Allowable Slenderness Ratio :    200.000           LOC  :     0.00
 |
 |------------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
```

```
 |
 | Critical L/C  :      1            Ratio     :        0.653(PASS)
 |
 |         Loc   :    0.00           Condition :    Cl.G1
 |
 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   7.658E+00     Axx :   3.829E+00     Ayy :   3.829E+00
 |
 | Ixx :   9.105E+01     Iyy :   9.105E+01     J   :   1.821E+02
 |
 | Sxx+:   1.821E+01     Sxx-:   1.821E+01     Zxx :   2.377E+01
 |
 | Syy+:   1.821E+01     Syy-:   1.821E+01     Zyy :   2.377E+01
 |
 | Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
 |
 |-----------------------------------------------------------------------------
 |
 | MATERIAL PROPERTIES
 |
 | Fyld:          50.000             Fu:          60.000
 |
 |-----------------------------------------------------------------------------
 |
 | Actual Member Length:        60.000
 |
 | Design Parameters                                              (Built-Up)
 |
 | Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
 |
 |-----------------------------------------------------------------------------
 |
 | COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: NonSlender      40.00       N/A      63.80     Table.4.1a.Case9
 |
 | Web   : NonSlender      40.00       N/A      63.80     Table.4.1a.Case9
 |
 |
 |
 | FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
 |
 |                           λ         λp        λr       CASE
 |
 | Flange: Compact         40.00      40.60    179.80     Table.4.1b.Case20
 |
 | Web   : Compact         40.00      40.60    179.80     Table.4.1b.Case20
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
```

```
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       344.6       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  382.88     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       275.7       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  6.1261     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  367.57     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.26       337.0       0.072     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  17.400                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  945.37     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  48.905     ksi        Eq.E3-2
```

```
||
||  Nom. Flexural Buckling    : Pnx    =  374.50     kip        Eq.E3-1
||
||-----------------------------------------------------------------------------
||
|| FLEXURAL BUCKLING Y
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             24.26       337.0       0.072     Cl.E3          2      0.00
||
||
||
|| Intermediate Results :
||
||  Effective Slenderness     : Lcy/ry =  17.400                Cl.E2
||
||  Elastic Buckling Stress   : Fey    =  945.37     ksi        Eq.E3-4
||
||  Crit. Buckling Stress     : Fcry   =  48.905     ksi        Eq.E3-2
||
||  Nom. Flexural Buckling    : Pny    =  374.50     kip        Eq.E3-1
||
||-----------------------------------------------------------------------------
||
      STAAD SPACE                                            -- PAGE NO.
9
                   STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                 *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                      - Member :     2 Contd.
||-----------------------------------------------------------------------------
||
||                        CHECKS FOR SHEAR
||
||-----------------------------------------------------------------------------
||
|| SHEAR ALONG X
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||            0.6958E-01   103.4       0.001     Cl.G1          2      0.00
||
||
||
|| Intermediate Results :
||
||  Nom. Shear Along X        : Vnx    =  114.86     kip        Eq.G5-1
||
||  Crit. Stress Fcr Along X  : Fcrx   =  30.000     ksi        Eq.G5-2
||
||-----------------------------------------------------------------------------
||
|| SHEAR ALONG Y
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             67.50       103.4       0.653     Cl.G1          1      0.00
```

```
|
|
|
| Intermediate Results :
|
|  Nom. Shear Along Y        : Vny    =  114.86     kip        Eq.G5-1
|
|  Crit. Stress Fcr Along Y  : Fcrx   =  30.000     ksi        Eq.G5-2
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
10
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             599.6       1070.       0.561     Cl.F8.1        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1188.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             2.087       1070.       0.002     Cl.F8.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1188.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
```

```
|
|                          CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.578      Eq.H1-1b           1        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity           : Pc     =  337.05     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  1069.7     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  1069.7     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             156.3       1008.       0.155     Cl.H3.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr        : Fcr    =  30.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength            : Tn     =  1119.9     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2016
    59. SLF 0.8 MEMB 2
    60. FYLD 50 ALL
    61. FU 60 ALL
    62. METHOD ASD
    63. STP 2 MEMB 2
```

```
    64. TRACK 2 MEMB 2
    65. CHECK CODE MEMB 2
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
```

```
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                -- PAGE NO.
14
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
|-----------------------------------------------------------------------------
|
|  Member No:        2       Profile:  ST  PIPE                (UPT)
|
|  Status:        PASS       Ratio:         0.981      Loadcase:        1
|
|  Location:      0.00       Ref:      Cl.G1
|
|  Pz:       11.75     C     Vy:        67.50          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      599.6
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     17.400
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      1             Ratio     :        0.981(PASS)
|
|          Loc  :    0.00            Condition :    Cl.G1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   7.658E+00     Axx :   3.829E+00     Ayy :   3.829E+00
|
| Ixx :   9.105E+01     Iyy :   9.105E+01     J   :   1.821E+02
|
| Sxx+:   1.821E+01     Sxx-:   1.821E+01     Zxx :   2.377E+01
|
| Syy+:   1.821E+01     Syy-:   1.821E+01     Zyy :   2.377E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          60.000
|
|-----------------------------------------------------------------------------
|
```

```
| Actual Member Length:        60.000
|
| Design Parameters                                               (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender      40.00       N/A      63.80     Table.4.1a.Case9
|
| Web   : NonSlender      40.00       N/A      63.80     Table.4.1a.Case9
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact         40.00      40.60    179.80     Table.4.1b.Case20
|
| Web   : Compact         40.00      40.60    179.80     Table.4.1b.Case20
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
15
                          STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                        ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                            - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       229.3       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  382.88     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|              0.000       183.8       0.000     Cl.D2           1      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  6.1261     in2         Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  367.57     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              24.26       224.3       0.108     Cl.E3           2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  17.400                 Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  945.37     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  48.905     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  374.50     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE      L/C    LOC
|
|              24.26       224.3       0.108     Cl.E3           2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  17.400                 Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  945.37     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  48.905     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  374.50     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                            -- PAGE NO.
16
```

```
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |            0.6958E-01   68.78       0.001     Cl.G1          2      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Shear Along X        : Vnx    =  114.86     kip        Eq.G5-1
 |
 |  Crit. Stress Fcr Along X  : Fcrx   =  30.000     ksi        Eq.G5-2
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             67.50       68.78       0.981     Cl.G1          1      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Nom. Shear Along Y        : Vny    =  114.86     kip        Eq.G5-1
 |
 |  Crit. Stress Fcr Along Y  : Fcrx   =  30.000     ksi        Eq.G5-2
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                                -- PAGE NO.
17
                    STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                  ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
 | FLEXURAL YIELDING (X)
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
```

```
|            599.6      711.7      0.843     Cl.F8.1       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1188.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|            DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            2.087       711.7       0.003     Cl.F8.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1188.5     kip-in     Eq.F8-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
18
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.869      Eq.H1-1b           1        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  224.25     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  711.68     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  711.68     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
19
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
```

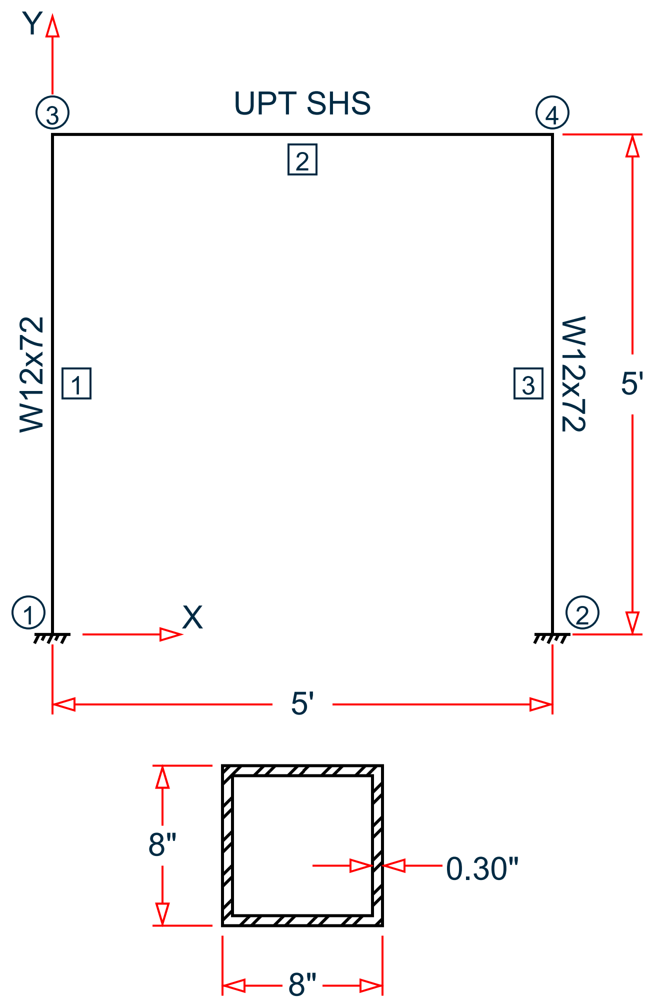
```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             156.3       670.6       0.233     Cl.H3.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  30.000     ksi        Cl.H3.1(a)
|
|  Nom. Strength             : Tn     =  1119.9     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H.3-X AXIS
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.775      Eq.H3-6            2        0.00
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H.3-Y AXIS
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.166      Eq.H3-6            2        0.00
|
|-----------------------------------------------------------------------------
|
```

V. AISC 360-16 UPT Square Hollow Section

Verify the axial compression capacity, shear capacity, flexure capacity, torsional capacity, and interaction ratio of a user-provided hollow square section member per both the LRFD and ASD methods of the AISC 360-16 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT SHS member for the beam. The section outer depth, D, is 8" and the thickness, t, is 0.30".



Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

<u>Design Forces</u>

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner depth, $d = 8 - 2\times0.30 = 7.40\ in.$

Area, $A_g = D^2 - d^2 = 9.24\ in^2$

Shear area, $A_w = 2\times D\times t_w = 4.80\ in^2$

Moment of inertia, $I = \frac{1}{12}\left(D^4 - d^4\right) = 91.45\ in^4$

Elastic section modulus, $S = \frac{I}{D/2} = \frac{91.45}{8/2} = 22.86\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{91.45}{9.24}} = 3.146\ in$

Plastic section modulus, $Z = \frac{1}{6}\left(D^3 - d^3\right) = 26.69\ in^3$

HSS Torsional constant, $C = 2\left(B - t_w\right)\left(D - t_w\right)t_w - 4.5\left(4 - \pi\right){t_w}^3 = 35.47\ in^3$

Section Classification

The design thickness, $t = 0.93\times t_w = 0.279\ in$

The design depth, $d = D - 3t = 7.163\ in$

Flange in compression

$$\lambda = \frac{d}{t} = \frac{7.163}{0.279} = 25.67$$

Per Table 4.1a Case 6, $\lambda < \lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29{,}000}{50}} = 33.72$, therefore flanges are non-slender for compression. Similar for the web of the square section.

Flexure classification of flange:

Per Table 4.1b, Case 17, $\lambda < \lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{50}} = 26.97$, therefore flange is compact for bending.

Flexure classification of web:

Per Table 4.1b, Case 19, $\lambda < \lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{50}} = 58.28$, therefore web is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.146} = 19.07 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(19.07)^2} = 787\ \text{ksi} \tag{Eq. E3-4}$$

$$\frac{F_y}{F_e} = \frac{50}{787} = 0.064 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y / F_e} F_y = 0.658^{(0.064)} 50 = 48.68 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 9.24 \times 48.68 = 449.9 \text{ kips} \qquad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 449.9 = 404.9 \; kips$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 449.9 \; / \; 1.67 = 269.4 \; kips$

Calculate Shear Capacity The nominal shear strength:

$$V_n = 0.6 \times F_y \times A_w \times C_{v2} \qquad \text{(Eq. G4-1)}$$

where

C_{v2} = *1.0* per Section G2.2 with $h/t = 25.67$ and $K_v = 5$.

$$V_n = 0.6\ (50)\ (4.80)\ (1.0) = 144 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 144 = 129.6 \; kips$
The allowable shear capacity (ASD) = $V_n / \Omega_v = 144 \; / \; 1.67 = 86.23 \; kips$

Calculate Bending Capacity

Bending capacity in plastic yielding (compact section):

$$M_p = F_y \times Z_x = 50 \times 26.69 = 1{,}335 \text{ in·kips} \qquad \text{(Eq F7-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}335 = 1{,}201 \; in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}335 \; / \; 1.67 = 799.2 \; in \cdot kips$

Torsional Capacity

$$\frac{h}{t} = 25.67 < 2.45\sqrt{\frac{E}{F_y}} = 59.0$$

$$F_{cr} = 0.6F_y = 30 \text{ ksi}$$

Nominal torsional capacity:

$$T_n = F_{cr} \times C = 1{,}064 \text{ ksi}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1{,}064 = 957.7 \; kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 1{,}064 \; / \; 1.67 = 637.2 \; kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.80}{404.9} = 0.029 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.80}{269.4} = 0.044 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0$.

$$\frac{0.029}{2} + \left(\frac{599.7}{1{,}201} + \frac{0}{1{,}201}\right) = 0.514 \text{ (LRFD)}$$

$$\frac{0.044}{2} + \left(\frac{599.7}{799.2} + \frac{0}{799.2}\right) = 0.765 \text{ (ASD)}$$

Results

Table 740:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	404.9	404.9	none	
	Bending capacity (in·kips)	1,201	1,201	none	
	Shear capacity (kips)	129.6	129.6	none	
	Torsional capacity (in·kips)	957.7	957.7	none	
	Interaction ratio	0.514	0.514	none	
ASD	Compression capacity (kips)	269.4	269.4	none	
	Bending capacity (in·kips)	799.2	799.2	none	
	Shear capacity (kips)	86.23	86.23	none	
	Torsional capacity (in·kips)	637.2	637.2	none	
	Interaction ratio	0.765	0.772	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 UPT Square Hollow Section.STD` is typically installed with the program.

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded SHS member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
SHS
9.24 8 8 0.3 91.4452 91.4452 136.96 4.8 4.8
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 SHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
```

```
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2016
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
|----------------------------------------------------------------------------
|
|  Member No:        2       Profile:  ST  SHS                 (UPT)
|
|  Status:        PASS       Ratio:         0.521       Loadcase:        1
|
|  Location:      0.00       Ref:      Cl.G1
|
|  Pz:       11.80     C     Vy:        67.50           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      599.7
|
|----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     19.072
|
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00
|
|----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      1             Ratio     :        0.521(PASS)
|
|          Loc  :    0.00            Condition :    Cl.G1
```

```
||
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   9.240E+00     Axx :   4.800E+00     Ayy :   4.800E+00
|
| Ixx :   9.145E+01     Iyy :   9.145E+01     J   :   1.370E+02
|
| Sxx+:   2.286E+01     Sxx-:   2.286E+01     Zxx :   2.669E+01
|
| Syy+:   2.286E+01     Syy-:   2.286E+01     Zyy :   2.669E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          60.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        60.000
|
| Design Parameters                                           (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender      25.45       N/A      33.72     Table.4.1a.Case6
|
| Web   : NonSlender      25.45       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact         25.45      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         25.45      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
8
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
 |-----------------------------------------------------------------------------
```

```
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       415.8       0.000     Cl.D2          1      0.00
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn      =  462.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       332.6       0.000     Cl.D2          1      0.00
|
|
| Intermediate Results :
|
|  Effective area            : Ae      =  7.3920     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn      =  443.52     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.39       404.9       0.060     Cl.E3          2      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  19.072               Cl.E2
|
|  Elastic Buckling Stress   : Fex     =  786.84     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx    =  48.688     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx     =  449.87     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
```

```
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.39       404.9       0.060     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  19.072               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  786.84     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  48.688     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  449.87     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                               -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|            0.7438E-01   129.6       0.001     Cl.G1          2      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  5.0000               Cl.G4
|
|  Nom. Shear Along X        : Vnx    =  144.00     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             67.50       129.6       0.521     Cl.G1          1      0.00
|
|
```

```
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
|
|  Nom. Shear Along Y        : Vny    =  144.00     kip        Eq.G4-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
10
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
 |-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             599.7       1201.       0.499     Cl.F7.1        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1334.7     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             2.231       1201.       0.002     Cl.F7.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1334.7     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     2 Contd.
 |-----------------------------------------------------------------------------
```

```
|
|                          CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA          L/C      LOC
|
|                            0.514      Eq.H1-1b           1        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc      =  404.89     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx     =  1201.2     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy     =  1201.2     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              131.1       957.7       0.137     Cl.H3.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr     =  30.000     ksi        Eq.H3-3
|
|  Nom. Strength             : Tn      =  1064.1     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
    59. PARAMETER 2
    60. CODE AISC UNIFIED 2016
    61. SLF 0.8 MEMB 2
    62. FYLD 50 ALL
    63. FU 60 ALL
    64. METHOD ASD
    65. STP 2 MEMB 2
```

```
    66. TRACK 2 MEMB 2
    67. CHECK CODE MEMB 2
 PARAMETER 2
      STAAD SPACE                                              -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
```

```
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     2
|-----------------------------------------------------------------------------
|
|  Member No:        2       Profile:  ST  SHS                (UPT)
|
|  Status:        PASS       Ratio:         0.783      Loadcase:        1
|
|  Location:      0.00       Ref:      Cl.G1
|
|  Pz:       11.80     C     Vy:        67.50          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      599.7
|
|-----------------------------------------------------------------------------
|
| COMPRESSION SLENDERNESS
|
| Actual Slenderness Ratio    :     19.072
|
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      1             Ratio     :        0.783(PASS)
|
|          Loc  :    0.00            Condition :    Cl.G1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   9.240E+00     Axx :   4.800E+00     Ayy :   4.800E+00
|
| Ixx :   9.145E+01     Iyy :   9.145E+01     J   :   1.370E+02
|
| Sxx+:   2.286E+01     Sxx-:   2.286E+01     Zxx :   2.669E+01
|
| Syy+:   2.286E+01     Syy-:   2.286E+01     Zyy :   2.669E+01
|
| Cw  :   0.000E+00     x0  :   0.000E+00     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          60.000
|
|-----------------------------------------------------------------------------
|
```

```
| Actual Member Length:        60.000
|
| Design Parameters                                            (Built-Up)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.80  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      2 LOC:    60.00)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender      25.45       N/A      33.72     Table.4.1a.Case6
|
| Web   : NonSlender      25.45       N/A      33.72     Table.4.1a.Case6
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      2 LOC:    60.00)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact         25.45      26.97     33.72     Table.4.1b.Case17
|
| Web   : Compact         25.45      58.28    137.27     Table.4.1b.Case19
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
15
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       276.6       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  462.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|             0.000       221.8       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  7.3920     in2         Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  443.52     kip         Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.39       269.4       0.091     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  19.072                 Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  786.84     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  48.688     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  449.87     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             24.39       269.4       0.091     Cl.E3          2      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  19.072                 Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  786.84     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  48.688     ksi         Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  449.87     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                          -- PAGE NO.
16
```

```
                       STAADPRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.7438E-01   86.23       0.001     Cl.G1          2      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  5.0000                Cl.G4
 |
 |  Nom. Shear Along X        : Vnx    =  144.00     kip        Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |              67.50       86.23       0.783     Cl.G1          1      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
 |
 |  Coefficient Kv Along Y    : Kv     =  5.0000                Cl.G4
 |
 |  Nom. Shear Along Y        : Vny    =  144.00     kip        Eq.G4-1
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                                -- PAGE NO.
17
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                           - Member :     2 Contd.
 |-----------------------------------------------------------------------------
 |
 |                        CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
 |
```

```
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             599.7       799.2       0.750     Cl.F7.1        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1334.7     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             2.231       799.2       0.003     Cl.F7.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  1334.7     kip-in     Eq.F7-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
18
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.772      Eq.H1-1b           1        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  269.39     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  799.22     kip-in     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  799.22     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
```

```
     STAAD SPACE                                              -- PAGE NO.
19
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                      - Member :     2 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION CAPACITY
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             131.1       637.2       0.206     Cl.H3.1        2      0.00
|
|
|
| Intermediate Results :
|
|  Crit. Stress. Fcr         : Fcr    =  30.000     ksi        Eq.H3-3
|
|  Nom. Strength             : Tn     =  1064.1     kip-in     Eq.H3-1
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H.3-X AXIS
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.614      Eq.H3-6            2        0.00
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H.3-Y AXIS
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.136      Eq.H3-6            2        0.00
|
|-----------------------------------------------------------------------------
|
```

V.AISC 360-16 C Tension ASD

Verify the tensile yield and tensile rupture strength of channel section with ASD method for AISC 360-16 code.

Details

The member is a C5x9 section. Steel grade is ASTM A992. Member is loaded with dead load of 15 kips and live load of 50 kips. The member is welded (i.e., no bolt holes) and the shear lag factor, U, is calculated as 0.908.

Validation

Service load: $P = DL + LL = 65\ kips$

Tensile Yielding

Nominal tensile yield strength:

$$P_{nY} = F_y A_g = 50 \times 2.64 = 132 \text{ kip} \qquad \text{[Eqn. D2-1]}$$

$$P_{nY}/\Omega_{tY} = 132 / 1.67 = 79.04 \text{ kip}$$

Ratio for tensile yield = 65 / 79.04 = 0.822

Tensile Rupture

Net area, $A_n = A_g = 2.64\ in^2$

Effective net area based on D3: $A_e = A_n \times U = 2.397\ in^2$

Nominal tensile rupture strength:

$$P_{nR} = F_u A_e = 65 \times 2.397 = 155.8 \text{ kip} \qquad \text{[Eqn. D2-2]}$$

$$P_{nR}/\Omega_{tR} = 155.8 / 2.00 = 77.9 \text{ kip}$$

Ratio for tensile yield = 65 / 77.9 = 0.834

Results

Table 741: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yield strength, P_{nY}/Ω_{tY} (kip)	79.04	79.04	none	
Ratio	0.822	0.822	none	
Tensile rupture strength, P_{nR}/Ω_{tR} (kip)	77.9	77.91	negligible	
Ratio	0.834	0.834	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 C Tension ASD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C5X9
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 15
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.0 2 1.0
PERFORM ANALYSIS
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
SLF 0.908 ALL
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
```

```
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C5X9                (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.834       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.D2
|
|  Pz:       65.00     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
```

```
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    246.826
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.834(PASS)
|
|          Loc  :    0.00            Condition :    Cl.D2
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   2.640E+00     Axx :   1.210E+00     Ayy :   1.625E+00
|
| Ixx :   8.890E+00     Iyy :   6.240E-01     J   :   1.090E-01
|
| Sxx+:   3.556E+00     Sxx-:   3.556E+00     Zxx :   4.390E+00
|
| Syy+:   4.555E-01     Syy-:   1.200E+00     Zyy :   9.130E-01
|
| Cw  :   2.926E+00     x0  :  -9.563E-01     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        10.000
|
| Design Parameters                                                 (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.91  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   120.00)
|
|                            λ          λp         λr        CASE
|
| Flange: NonSlender       5.91        N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      13.42        N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   120.00)
|
|                            λ          λp         λr        CASE
```

```
|
| Flange: Compact          5.91       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         13.42      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                               - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              65.00       79.04       0.822     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  132.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              65.00       77.91       0.834     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae     = 0.16647E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  155.81     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       57.82       0.000     Cl.E3          1      0.00
```

```
 Intermediate Results :
  Effective Slenderness     : Lcx/rx =  65.393               Cl.E2
  Elastic Buckling Stress   : Fex    =  9638.2     kip/ft2    Eq.E3-4
  Crit. Buckling Stress     : Fcrx   =  5266.7     kip/ft2    Eq.E3-2
  Nom. Flexural Buckling    : Pnx    =  96.557     kip        Eq.E3-1
-----------------------------------------------------------------------------
 FLEXURAL BUCKLING Y
              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
              0.000       6.513       0.000     Cl.E3          1      0.00

 Intermediate Results :
  Effective Slenderness     : Lcy/ry =  246.83               Cl.E2
  Elastic Buckling Stress   : Fey    =  676.52     kip/ft2    Eq.E3-4
  Crit. Buckling Stress     : Fcry   =  593.30     kip/ft2    Eq.E3-3
  Nom. Flexural Buckling    : Pny    =  10.877     kip        Eq.E3-1
-----------------------------------------------------------------------------
 FLEXURAL-TORSIONAL-BUCKLING
              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
              0.000       54.05       0.000     Cl.E4          1      0.00

 Intermediate Results :
  Constant                  : H      = 0.79760               Eq.E4-8
  Euler Buckling Stress     : Fez    =  15379.     kip/ft2    Eq.E4-7
  Elastic F-T-B Stress      : Fe     =  7929.7     kip/ft2    Eq.E4-3
  Crit. F-T-B Stress        : Fcr    =  4923.6     kip/ft2    Eq.E3-2
  Nom. Flex-tor Buckling    : Pn     =  90.267     kip        Eq.E4-1
-----------------------------------------------------------------------------
     STAAD SPACE                                            -- PAGE NO.
```

```
6
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR SHEAR
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG X
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       21.73       0.000     Cl.G1          1      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
 |
 |  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
 |
 |  Nom. Shear Along X        : Vnx    =  36.288     kip       Eq.G6-1
 |
 |-----------------------------------------------------------------------------
 |
 | SHEAR ALONG Y
 |
 |             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
 |
 |             0.000       29.19       0.000     Cl.G1          1      0.00
 |
 |
 |
 | Intermediate Results :
 |
 |  Coefficient Cv Along Y    : Cv     =  1.0000               -
 |
 |  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
 |
 |  Nom. Shear Along Y        : Vny    =  48.750     kip       Eq.G2-1
 |
 |-----------------------------------------------------------------------------
 |
     STAAD SPACE                                               -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
 |-----------------------------------------------------------------------------
 |
 |                       CHECKS FOR BENDING
 |
 |-----------------------------------------------------------------------------
```

```
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       10.95       0.000     Cl.F2.1        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  18.292     kip-ft     Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1.819       0.000     Cl.F6.1        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  3.0369     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       6.269       0.000     Cl.F2.2        1      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  10.470     kip-ft     Eq.F2-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  1.7173     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0807                Eq.F2-8b
|
|  Effective Rad. of Gyr.    : Rts    = 0.51367E-01 ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  10.104     ft         Eq.F2-6
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                   ****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                  - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                          RATIO      CRITERIA           L/C      LOC
|
|                          0.834      Eq.H1-1a            3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity            : Pc     =  77.906     kip        Cl.H1.1
|
|  Moment Capacity           : Mcx    =  6.2693     kip-ft     Cl.H1.1
|
|  Moment Capacity           : Mcy    =  1.8185     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
   45. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME= 10: 3:24 ****
```

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

V.AISC 360-16 C Tension LRFD

Verify the tensile yield and tensile rupture strength of channel section with LRFD method for AISC 360-16 code.

Details

The member is a C5x9 section. Steel grade is ASTM A992. Member is loaded with dead load of 15 kips and live load of 50 kips. The member is welded (i.e., no bolt holes) and the shear lag factor, U, is calculated as 0.908.

Validation

Ultimate load: $P_u = 1.2 \times DL + 1.6 \times LL = 98\ kips$

Tensile Yielding

Nominal tensile yield strength:

$$P_{nY} = F_y A_g = 50 \times 2.64 = 132 \text{ kip} \quad \text{[Eqn. D2-1]}$$

$$\phi_{tY} P_{nY} = 0.9 \times 132 = 118.8 \text{ kip}$$

Ratio for tensile yield = 98 / 118.8 = 0.825

Tensile Rupture

Net area, $A_n = A_g = 2.64\ in^2$

Effective net area based on D3: $A_e = A_n \times U = 2.397\ in^2$

Nominal tensile rupture strength:

$$P_{nR} = F_u A_e = 65 \times 2.397 = 155.8 \text{ kip} \qquad \text{[Eqn. D2-2]}$$

$$\phi_{tR} P_{nR} = 0.75 \times 155.8 = 116.9 \text{ kip}$$

Ratio for tensile yield = 98 / 116.9 = 0.839

Results

Table 742: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yield strength, $\phi_{tY}P_{nY}$ (kip)	118.8	118.8	none	
Ratio	0.825	0.825	none	
Tensile rupture strength, $\phi_{tR}P_{nR}$ (kip)	116.9	116.9	negligible	
Ratio	0.839	0.839	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 C Tension LRFD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C5X9
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 15
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
SLF 0.908 ALL
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
PERFORM ANALYSIS
LOAD LIST 3
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
```

```
       Mx              MZ          Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
 |------------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  C5X9               (AISC
SECTIONS)|
|  Status:        FAIL       Ratio:         2.057      Loadcase:       3
|
|  Location:      0.00       Ref:      Slenderness (T)
|
|  Pz:       98.00     T     Vy:        0.000          Vx:      0.000
|
|  Tz:       0.000           My:        0.000          Mx:      0.000
|
|------------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    617.065
|
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00
|
|------------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        2.057(FAIL)
```

```
|
|         Loc  :    0.00            Condition :    Cl.D1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   2.640E+00     Axx :   1.210E+00     Ayy :   1.625E+00
|
| Ixx :   8.890E+00     Iyy :   6.240E-01     J   :   1.090E-01
|
| Sxx+:   3.556E+00     Sxx-:   3.556E+00     Zxx :   4.390E+00
|
| Syy+:   4.555E-01     Syy-:   1.200E+00     Zyy :   9.130E-01
|
| Cw  :   2.926E+00     x0  :  -9.563E-01     y0  :   0.000E+00
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        7200.000             Fu:        9359.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        25.000
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.91  CSP:    1.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   300.00)
|
|                            λ         λp        λr       CASE
|
| Flange: NonSlender       5.91       N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      13.42       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   300.00)
|
|                            λ         λp        λr       CASE
|
| Flange: Compact          5.91       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         13.42      90.55    137.27     Table.4.1b.Case15
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                          - Member :     1 Contd.
|--------------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL TENSION
|
|--------------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             98.00       118.8       0.825     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn      =  132.00     kip        Eq.D2-1
|
|--------------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             98.00       116.9       0.839     Cl.D2          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae      = 0.16647E-01 ft2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn      =  155.81     kip        Eq.D2-2
|
|--------------------------------------------------------------------------------
|
|                           CHECKS FOR AXIAL COMPRESSION
|
|--------------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       22.32       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  163.48                Cl.E2
|
|  Elastic Buckling Stress   : Fex     =  1542.1     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx    =  1352.4     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnx     =  24.795     kip        Eq.E3-1
```

```
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       1.566       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  617.07               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  108.24     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  94.929     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  1.7404     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       21.81       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Constant                  : H      = 0.79760               Eq.E4-8
|
|  Euler Buckling Stress     : Fez    =  14789.     kip/ft2    Eq.E4-7
|
|  Elastic F-T-B Stress      : Fe     =  1507.5     kip/ft2    Eq.E4-3
|
|  Crit. F-T-B Stress        : Fcr    =  1322.1     kip/ft2    Eq.E3-3
|
|  Nom. Flex-tor Buckling    : Pn     =  24.238     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
6
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
```

```
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       32.66       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  36.288     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       43.88       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  48.750     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                           CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       16.46       0.000     Cl.F2.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  18.292     kip-ft    Eq.F2-1
```

```
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       2.733       0.000     Cl.F6.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  3.0369     kip-ft     Eq.F6-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       3.699       0.000     Cl.F2.2        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  4.1104     kip-ft     Eq.F2-3
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  1.7173     ft         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0807                Eq.F2-8b
|
|  Effective Rad. of Gyr.    : Rts    = 0.51367E-01 ft         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  10.104     ft         Eq.F2-6
|
|  Crit. Elas. L-T-B Stress  : FcrX   =  1997.4     kip/ft2    Eq.F2-4
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                    - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                            RATIO      CRITERIA           L/C      LOC
```

```
|
|                           0.839      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
|  Axial Capacity             : Pc     =  116.86     kip        Cl.H1.1
|
|  Moment Capacity            : Mcx    =  3.6994     kip-ft     Cl.H1.1
|
|  Moment Capacity            : Mcy    =  2.7333     kip-ft     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    46. PERFORM ANALYSIS
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
```

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

V. AISC 360-16 I Tension LRFD

To verify the tensile yield strength of an I section using the LRFD method from the AISC 360-16 code.

Reference

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

Verify the tensile yield and tensile rupture strength of an I section (W8X21) using the LRFD method. The material is A992 ($F_y = 50\ ksi$, $F_u = 65\ ksi$)

The area of the bolt holes is assumed to be zero. A known shear lag factor of 0.908 is directly specified.

The length of the member is 25 ft. It is loaded axial in tension with a dead load of 30 kips and a live load of 90 kips.

Validation

The factored axial load, $P_u = 1.2 \times 30 + 1.6 \times 90 = 180\ kips$

<u>Tensile Yield Strength</u>

The tensile yield strength: $\phi_{tY} P_{nY} = \phi_{tY} \times F_y \times A_g = 0.9 \times 50 \times 6.16 = 277.2\ kips$

Ratio for yield strength: 0.649

<u>Tensile Rupture Strength</u>

For this example, A_n is assumed equal to A_g .

Effective area, $A_e = A_n \times U = 6.16 \times 0.908 = 5.593\ in^2$

The tensile rupture strength = $\phi_{tR}P_{nR} = \phi_{tR}{\times}F_u{\times}A_e = 0.75{\times}65{\times}5.593 = 272.672\ kips$

Ratio for rupture strength: 0.660

Results

Table 743: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Tensile yield strength (kips)	277.2	277.2	none	
Ratio	0.649	0.649	none	
Tensile rupture strength (kips)	272.7	272.7	none	
Ratio	0.660	0.660	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 I Tension LRFD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 30
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
```

```
2 FX 90
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2016
FU 9360 ALL
FYLD 7200 ALL
KX 1 ALL
KY 1 ALL
KZ 1 ALL
METHOD LRFD
PROFILE W8X ALL
*SGR 28 ALL
SLF 0.908 ALL
STP 1 ALL
TRACK 2 ALL
UNIT KIP INCH
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections
principal major axis.
       Y               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
```

```
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  W8X21              (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.660       Loadcase:        3
|
|  Location:      0.00       Ref:      Cl.D2
|
|  Pz:       180.0     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    238.212
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.660(PASS)
|
|          Loc  :    0.00            Condition :    Cl.D2
|
```

```
|--------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   6.160E+00     Axx :   4.216E+00     Ayy :   2.070E+00
|
| Ixx :   7.530E+01     Iyy :   9.770E+00     J   :   2.820E-01
|
| Sxx+:   1.819E+01     Sxx-:   1.819E+01     Zxx :   2.040E+01
|
| Syy+:   3.708E+00     Syy-:   3.708E+00     Zyy :   5.690E+00
|
| Cw  :   1.515E+02     x0  :   0.000E+00     y0  :   0.000E+00
|
|--------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000             Fu:          65.000
|
|--------------------------------------------------------------------------
|
| Actual Member Length:       300.000
|
| Design Parameters                                         (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    0.91  CSP:   12.00
|
|--------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:   300.00)
|
|                           λ          λp         λr       CASE
|
| Flange: NonSlender       6.59       N/A      13.49     Table.4.1a.Case1
|
| Web   : NonSlender      27.52       N/A      35.88     Table.4.1a.Case5
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:   300.00)
|
|                           λ          λp         λr       CASE
|
| Flange: Compact          6.59       9.15     24.08     Table.4.1b.Case10
|
| Web   : Compact         27.52      90.55    137.27     Table.4.1b.Case15
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
5
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|--------------------------------------------------------------------------
|
```

```
                         CHECKS FOR AXIAL TENSION
--------------------------------------------------------------------------------
 TENSILE YIELDING
              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
              180.0       277.2       0.649     Cl.D2          3      0.00

 Intermediate Results :
  Nom. Ten. Yld Cap         : Pn     =  308.00     kip        Eq.D2-1
--------------------------------------------------------------------------------
 TENSILE RUPTURE
              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
              180.0       272.7       0.660     Cl.D2          3      0.00

 Intermediate Results :
  Effective area            : Ae     =  5.5933     in2        Eq.D3-1
  Nom. Ten. Rpt Cap         : Pn     =  363.56     kip        Eq.D2-2
--------------------------------------------------------------------------------
                         CHECKS FOR AXIAL COMPRESSION
--------------------------------------------------------------------------------
 FLEXURAL BUCKLING X
              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
              0.000       161.8       0.000     Cl.E3          3      0.00

 Intermediate Results :
  Effective Slenderness     : Lcx/rx =  85.805                Cl.E2
  Elastic Buckling Stress   : Fex    =  38.875     ksi        Eq.E3-4
  Crit. Buckling Stress     : Fcrx   =  29.186     ksi        Eq.E3-2
  Nom. Flexural Buckling    : Pnx    =  179.79     kip        Eq.E3-1
--------------------------------------------------------------------------------
```

```
| FLEXURAL BUCKLING Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       24.52       0.000     Cl.E3          3      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  238.21               Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  5.0439     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  4.4235     ksi        Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  27.249     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL-TORSIONAL-BUCKLING
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       169.7       0.000     Cl.E4          3      0.00
|
|
|
| Intermediate Results :
|
|  Elastic F-T-B Stress      : Fe     =  42.637     ksi        Eq.E4-2
|
|  Crit. F-T-B Stress        : Fcr    =  30.606     ksi        Eq.E3-2
|
|  Nom. Flex-tor Buckling    : Pn     =  188.53     kip        Eq.E4-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       113.8       0.000     Cl.G1          3      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000               Cl.G6
|
|  Nom. Shear Along X        : Vnx    =  126.48     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       62.10       0.000     Cl.G1          3      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000               -
|
|  Coefficient Kv Along Y    : Kv     =  5.3400               Eq.G2-5
|
|  Nom. Shear Along Y        : Vny    =  62.100     kip       Eq.G2-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                       CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       918.0       0.000     Cl.F2.1        3      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along X : Mnx    =  1020.0     kip-in    Eq.F2-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

```
|              0.000       256.1       0.000     Cl.F6.1        3      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  284.50     kip-in     Eq.F6-1
|
|--------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       302.2       0.000     Cl.F2.2        3      0.00
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  335.72     kip-in     Eq.F2-3
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Limiting Unbraced Length  : LpX    =  53.381     in         Eq.F2-5
|
|  coefficient C             : Cx     =  1.0000                Eq.F2-8a
|
|  Effective Rad. of Gyr.    : Rts    =  1.4543     in         Eq.F2-7
|
|  Limiting Unbraced Length  : LrX    =  176.96     in         Eq.F2-6
|
|  Crit. Elas. L-T-B Stress  : FcrX   =  18.458     ksi        Eq.F2-4
|
|--------------------------------------------------------------------------
|
     STAAD SPACE                                          -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|--------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL BEND INTERACTION
|
|--------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H1
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.660      Eq.H1-1a           3        0.00
|
|
|
| Intermediate Results :
|
```

```
|  Modification Fact. for Cb : Cbf    =  2.6064               Eq.H1-2
|
|  Axial Capacity           : Pc     =  272.67     kip        Cl.H1.1
|
|  Moment Capacity          : Mcx    =  787.52     kip-in     Cl.H1.1
|
|  Moment Capacity          : Mcy    =  256.05     kip-in     Cl.H1.1
|
|-----------------------------------------------------------------------------
|
    53. FINISH
```

V.AISC 360-16 L Tension ASD

Verify the tensile yield and tensile rupture strength of an angle section with ASD method for AISC 360-16 code.

Details

The member is an L4x4x1/2 section. Steel grade is ASTM A36. The member is axially loaded with dead load of 20 kips and a live load of 60 kips. The member is connected using bolts in 3/4"⌀ holes. The shear lag factor, U, is calculated as 0.869.

Validation

Service load: $P = DL + LL = 80\ kips$

<u>Tensile Yielding</u>

Nominal tensile yield strength:

$$P_{nY} = F_y A_g = 36 \times 3.75 = 135 \text{ kip} \qquad \text{[Eqn. D2-1]}$$

$$P_{nY}/\Omega_{tY} = 132 / 1.67 = 80.83 \text{ kip}$$

Ratio for tensile yield = 80 / 80.83 = 0.990

<u>Tensile Rupture</u>

Bolt hole area = $\pi(0.75/2)^2 = 0.44\ in^2$

Net area, $A_n = A_g - A_{bolthole} = 3.75 - 0.44 = 3.31\ in^2$

Effective net area based on D3: $A_e = A_n \times U = 2.875\ in^2$

Nominal tensile rupture strength:

$$P_{nR} = F_u A_e = 58 \times 2.875 = 166.7 \text{ kip} \qquad \text{[Eqn. D2-2]}$$

$$P_{nR}/\Omega_{tR} = 166.7 / 2.00 = 83.37 \text{ kip}$$

Ratio for tensile yield = 80 / 83.4 = 0.960

Results

Table 744: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yield strength, P_{nY}/Ω_{tY} (kip)	80.83	80.84	negligible	
Ratio	0.990	0.99	none	
Tensile rupture strength, P_{nR}/Ω_{tR} (kip)	83.37	83.45	negligible	
Ratio	0.960	0.959	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 L Tension ASD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-May-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 19.4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GX 80 9.7
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
```

```
CODE AISC UNIFIED 2016
FU 8352 ALL
FYLD 5184 ALL
METHOD ASD
SLF 0.869 ALL
STP 1 ALL
NSF 0.883 ALL
TRACK 2 ALL
UNIT KIP INCH
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       U               Z          Axis typically parallel to the sections
principal major axis.
       V               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
```

```
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                     ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |---------------------------------------------------------------------------
 |
 |  Member No:        1       Profile:  ST  L40408              (AISC
SECTIONS)|
 |  Status:        PASS       Ratio:         0.990       Loadcase:        1
 |
 |  Location:      0.00       Ref:      Cl.D2
 |
 |  Pz:       80.00     T     Vy:        0.000           Vx:      0.000
 |
 |  Tz:       0.000           My:        0.000           Mx:      0.000
 |
 |---------------------------------------------------------------------------
 |
 | TENSION SLENDERNESS
 |
 | Actual Slenderness Ratio    :    300.000
 |
 | Allowable Slenderness Ratio :    300.000            LOC  :     0.00
 |
 |---------------------------------------------------------------------------
 |
 | STRENGTH CHECKS
 |
 | Critical L/C  :      1             Ratio     :        0.990(PASS)
 |
 |          Loc  :    0.00            Condition :    Cl.D2
 |
 |---------------------------------------------------------------------------
 |
 | SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
 |
 | Ag  :   3.750E+00     Axx :   2.000E+00     Ayy :   2.000E+00
 |
 | Ixx :   5.561E+00     Iyy :   5.561E+00     J   :   3.125E-01
 |
 | Sxx+:   1.349E+00     Sxx-:   1.955E+00     Zxx :   2.429E+00
 |
 | Syy+:   3.134E+00     Syy-:   3.134E+00     Zyy :   5.641E+00
 |
```

```
| Cw  :   3.662E-01     x0  :   9.333E-01     y0  :   9.333E-01
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          36.000             Fu:          58.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:       232.800
|
| Design Parameters                                               (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    0.88  SLF:    0.87  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   232.80)
|
|                           λ         λp        λr       CASE
|
| Flange: NonSlender       8.00       N/A      12.77     Table.4.1a.Case3
|
| Web   : NonSlender       8.00       N/A      12.77     Table.4.1a.Case3
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   232.80)
|
|                           λ         λp        λr       CASE
|
| Flange: Compact          8.00      15.33     25.83     Table.4.1b.Case12
|
| Web   : Compact          8.00      15.33     25.83     Table.4.1b.Case12
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                      ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                          - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C     LOC
|
|             80.00       80.84       0.990     Cl.D2          1      0.00
|
|
|
```

```
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn     =  135.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              80.00       83.45       0.959     Cl.D2          1      0.00
|
|
| Intermediate Results :
|
|  Effective area            : Ae     =  2.8775     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn     =  166.89     kip        Eq.D2-2
|
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       15.42       0.000     Cl.E3          1      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  191.16                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  7.8323     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  6.8689     ksi        Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnx    =  25.758     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       15.42       0.000     Cl.E3          1      0.00
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  191.16                Cl.E2
|
```

```
|  Elastic Buckling Stress   : Fey    =  7.8323     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  6.8689     ksi         Eq.E3-3
|
|  Nom. Flexural Buckling    : Pny    =  25.758     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING U
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       24.59       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcu/ru =  151.41                 Cl.E2
|
|  Elastic Buckling Stress   : Feu    =  12.484     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcru   =  10.949     ksi         Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnu    =  41.058     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING V
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       6.263       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcv/rv =  300.00                 Cl.E2
|
|  Elastic Buckling Stress   : Fev    =  3.1802     ksi         Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrv   =  2.7890     ksi         Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnv    =  10.459     kip         Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                    ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                     - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR SHEAR
|
```

```
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|                DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|                0.000        25.87       0.000     Cl.G1          1      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G3
|
|  Nom. Shear Along X        : Vnx    =  43.200     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|                DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|                0.000        25.87       0.000     Cl.G1          1      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along Y    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along Y    : Kv     =  1.2000                Cl.G3
|
|  Nom. Shear Along Y        : Vny    =  43.200     kip        Eq.G3-1
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
10
                         STAAD.PRO CODE CHECKING - AISC 360-16 ASD (V1.2)
                       ****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                            CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|                DEMAND       CAPACITY    RATIO     REFERENCE     L/C    LOC
|
```

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

V.AISC 360-16 L Tension LRFD

Verify the tensile yield and tensile rupture strength of an angle section with LRFD method for AISC 360-16 code.

Details

The member is an L4x4x1/2 section. Steel grade is ASTM A36. The member is axially loaded with dead load of 20 kips and a live load of 60 kips. The member is connected using bolts in 3/4"⌀ holes. The shear lag factor, U, is calculated as 0.869.

Validation

Ultimate load: $P_u = 1.2 \times DL + 1.6 \times LL = 120\ kips$

Tensile Yielding

Nominal tensile yield strength:

$$P_{nY} = F_y A_g = 36 \times 3.75 = 135 \text{ kip} \qquad \text{[Eqn. D2-1]}$$

$$\phi_{tY} P_{nY} = 0.9 \times 135 = 121.5 \text{ kip}$$

Ratio for tensile yield = 120 / 121.5 = 0.988

Tensile Rupture

Bolt hole area = $\pi(0.75/2)^2 = 0.44 \text{ in}^2$

Net area, $A_n = A_g - A_{bolthole} = 3.75 - 0.44 = 3.31\ in^2$

Effective net area based on D3: $A_e = A_n \times U = 2.875\ in^2$

Nominal tensile rupture strength:

$$P_{nR} = F_u A_e = 58 \times 2.875 = 166.7 \text{ kip} \qquad \text{[Eqn. D2-2]}$$

$$\phi_{tR} P_{nR} = 0.75 \times 166.7 = 125.1 \text{ kip}$$

Ratio for tensile yield = 80 / 83.4 = 0.960

Results

Table 745: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yield strength, $\phi_{tY}P_{nY}$ (kip)	121.5	121.5	none	
Ratio	0.988	0.988	none	
Tensile rupture strength, $\phi_{tR}P_{nR}$ (kip)	125.1	125.2	negligible	
Ratio	0.960	0.959	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 L Tension LRFD.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-May-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 19.4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GX 20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GX 60
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2016
FU 8352 ALL
FYLD 5184 ALL
METHOD LRFD
SLF 0.869 ALL
STP 1 ALL
NSF 0.883 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       U               Z          Axis typically parallel to the sections
principal major axis.
       V               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                                 -- PAGE NO.
```

```
7
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ******************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------
|
|  Member No:        1       Profile:  ST  L40408             (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.988       Loadcase:        3
|
|  Location:      0.00       Ref:      Eq.H2-1
|
|  Pz:       120.0     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       0.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :    300.000
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      3             Ratio     :        0.988(PASS)
|
|          Loc  :    0.00            Condition :    Eq.H2-1
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   3.750E+00     Axx :   2.000E+00     Ayy :   2.000E+00
|
| Ixx :   5.561E+00     Iyy :   5.561E+00     J   :   3.125E-01
|
| Sxx+:   1.349E+00     Sxx-:   1.955E+00     Zxx :   2.429E+00
|
| Syy+:   3.134E+00     Syy-:   3.134E+00     Zyy :   5.641E+00
|
| Cw  :   3.662E-01     x0  :   9.333E-01     y0  :   9.333E-01
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:        5184.000             Fu:        8351.999
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        19.400
|
| Design Parameters                                              (Rolled)
```

```
||
|| Kx:    1.00  Ky:    1.00  NSF:    0.88  SLF:    0.87  CSP:    1.00
||
||--------------------------------------------------------------------------
||
|| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   232.80)
||
||                           λ         λp        λr       CASE
||
|| Flange: NonSlender       8.00       N/A      12.77     Table.4.1a.Case3
||
|| Web   : NonSlender       8.00       N/A      12.77     Table.4.1a.Case3
||
||
||
|| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   232.80)
||
||                           λ         λp        λr       CASE
||
|| Flange: Compact          8.00      15.33     25.83     Table.4.1b.Case12
||
|| Web   : Compact          8.00      15.33     25.83     Table.4.1b.Case12
||
||--------------------------------------------------------------------------
||
     STAAD SPACE                                                -- PAGE NO.
8
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
||--------------------------------------------------------------------------
||
||                       CHECKS FOR AXIAL TENSION
||
||--------------------------------------------------------------------------
||
|| TENSILE YIELDING
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             120.0       121.5       0.988     Cl.D2          3      0.00
||
||
||
|| Intermediate Results :
||
||  Nom. Ten. Yld Cap         : Pn     =  135.00     kip        Eq.D2-1
||
||--------------------------------------------------------------------------
||
|| TENSILE RUPTURE
||
||             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
||
||             120.0       125.2       0.959     Cl.D2          3      0.00
||
||
```

```
Intermediate Results :

 Effective area           : Ae     = 0.19982E-01 ft2       Eq.D3-1

 Nom. Ten. Rpt Cap        : Pn     =  166.89     kip       Eq.D2-2

-----------------------------------------------------------------------------

                         CHECKS FOR AXIAL COMPRESSION

-----------------------------------------------------------------------------

FLEXURAL BUCKLING X
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC

             0.000       23.18       0.000     Cl.E3          1      0.00


Intermediate Results :

 Effective Slenderness     : Lcx/rx =  191.16                Cl.E2

 Elastic Buckling Stress   : Fex    =  1127.9     kip/ft2    Eq.E3-4

 Crit. Buckling Stress     : Fcrx   =  989.13     kip/ft2    Eq.E3-3

 Nom. Flexural Buckling    : Pnx    =  25.758     kip        Eq.E3-1

-----------------------------------------------------------------------------

FLEXURAL BUCKLING Y
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC

             0.000       23.18       0.000     Cl.E3          1      0.00


Intermediate Results :

 Effective Slenderness     : Lcy/ry =  191.16                Cl.E2

 Elastic Buckling Stress   : Fey    =  1127.9     kip/ft2    Eq.E3-4

 Crit. Buckling Stress     : Fcry   =  989.13     kip/ft2    Eq.E3-3

 Nom. Flexural Buckling    : Pny    =  25.758     kip        Eq.E3-1

-----------------------------------------------------------------------------

FLEXURAL BUCKLING U
             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC

             0.000       36.95       0.000     Cl.E3          1      0.00
```

```
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcu/ru =  151.41               Cl.E2
|
|  Elastic Buckling Stress   : Feu    =  1797.8     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcru   =  1576.6     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnu    =  41.058     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING V
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       9.413       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcv/rv =  300.00               Cl.E2
|
|  Elastic Buckling Stress   : Fev    =  457.95     kip/ft2    Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrv   =  401.62     kip/ft2    Eq.E3-3
|
|  Nom. Flexural Buckling    : Pnv    =  10.459     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       38.88       0.000     Cl.G1          1      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000               Eq.G2-9
```

```
|
| Coefficient Kv Along X    : Kv     =  1.2000              Cl.G3
|
| Nom. Shear Along X        : Vnx    =  43.200     kip       Eq.G6-1
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       38.88       0.000     Cl.G1          1      0.00
|
|
|
| Intermediate Results :
|
| Coefficient Cv Along Y    : Cv     =  1.0000              Eq.G2-9
|
| Coefficient Kv Along Y    : Kv     =  1.2000              Cl.G3
|
| Nom. Shear Along Y        : Vny    =  43.200     kip       Eq.G3-1
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                             -- PAGE NO.
10
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (X)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       7.997       0.000     Cl.F10.1       1      0.00
|
|
|
| Intermediate Results :
|
| Nom Flex Yielding Along X : Mnx    =  8.8852     kip-ft    Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (Y)
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       7.997       0.000     Cl.F10.1       1      0.00
|
|
```

```
| Intermediate Results :
|
|  Nom Flex Yielding Along Y : Mny    =  8.8852     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (U)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       12.69       0.000     Cl.F10.1       1      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along U : Mnu    =  14.104     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (V)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       5.465       0.000     Cl.F10.1       1      0.00
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along V : Mnv    =  6.0722     kip-ft     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       6.064       0.000     Cl.F10.2       1      0.00
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  6.7380     kip-ft     Eq.F10-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Crit. Elastic L-T-B Mom.  : McrX   =  26.144     kip-ft     Eq.F10-5b
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       6.064       0.000     Cl.F10.2       1      0.00
```

```
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About Y : Mny    =  6.7380     kip-ft     Eq.F10-2
|
|  Mod. factor Cb            : CbY    =  1.0000                Custom
|
|  Crit. Elas. L-T-B Moment  : McrY   =  26.144     kip-ft     Eq.F10-5b
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT U
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       8.883       0.000     Cl.F10.2       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About U : Mnu    =  9.8695     kip-ft     Eq.F10-2
|
|  modification factor Cb    : CbU    =  1.0000                Custom
|
|  Crit. Elast. L-T-B Moment : McrU   =  16.992     kip-ft     Eq.F10-4
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                           - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR AXIAL BEND INTERACTION
|
|-----------------------------------------------------------------------------
|
| COMBINED FORCES CLAUSE H2
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.988      Eq.H2-1            3        0.00
|
|
|
| Intermediate Results :
|
|  Applied Stress            : fra    =  4608.0     kip/ft2    Eq.H2-1
|
|  Allowable Stress          : Fca    =  4665.6     kip/ft2    Eq.H2-1
|
|  Allowable Stress          : Fcbz   =  4897.4     kip/ft2    Eq.H2-1
```

```
|
|  Allowable Stress          : Fcbw    =  4829.9     kip/ft2    Eq.H2-1
|
|-----------------------------------------------------------------------------
|
    47. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
```

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

V. AISC 360-16 - Torsion

Verify the torsional strength of an angle section using the LRFD method in the AISC 360-16 code. Use the simple method for calculating torsional shears (i.e., no full twist analysis). Perform torsion checks per AISC Design Guide 9.

Reference

1. *Steel Construction Manual*, 15th Edition, American Institute of Steel Construction, 2017
2. *Torsional Analysis of Structural Steel Members* Deign Guide 9, American Institute of Steel Construction

Related Links

- *D1.A.5.6 Design for Torsion* (on page 1096)

Details

The angle section is an L 3x3x1/2 angle of ASTM A992 material. The member is a 2 ft long cantilever with a total end torque of 3.0 in·kips.

Validation

Maximum torsional shear stress: $\tau_f = T\frac{t_f}{J} = 3.0\frac{0.5}{0.234} = 6.41 ksi$

By observation, direct shear stress is zero (torsion loading only). Therefore, total torsional stress is = τ_f .

Torsional shear resistance: $\phi_T \times 0.6 \times F_y = 0.9 \times 0.6 \times 50 = 27\ ksi$

Ratio: $6.41\ ksi\ /\ 27\ ksi = 0.237$

Results

Table 746: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Torsional shear stress ratio	0.237	0.242	2.1%	Negligible difference when considering simplified method used by hand calculations

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-16 - Torsion.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 2 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L30308
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CMOM GX -0.25 2
PERFORM ANALYSIS
PARAMETER 1
CODE AISC UNIFIED 2016
ALH 1 ALL
FU 7200 ALL
FYLD 7200 ALL
METHOD LRFD
SOE 0 ALL
TND 9 ALL
TORSION 1 ALL
TRACK 2 ALL
UNIT KIP INCH
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
```

```
 ========================
 The capacity results and intermediate results in the report follow the
notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC
360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       U               Z          Axis typically parallel to the sections
principal major axis.
       V               Y          Axis typically parallel to the sections
principal minor axis.
       Z               X          Longitudinal axis perpendicular to the
cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 ======================
    1. Section classification reported is for the cross section and loadcase
that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/
loadcase
       based on the code clause in AISC 360-16 will not be shown in the
report.
    3. Bending results are reported as being �about� the relevant axis (X/Y),
while
       the results for shear are reported as being for shear forces �along�
the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear
along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
 - Member :     1
 |-----------------------------------------------------------------------------
 |
```

```
|  Member No:        1       Profile:  ST  L30308            (AISC
SECTIONS)|
|  Status:        PASS       Ratio:         0.242       Loadcase:        1
|
|  Location:      0.00       Ref:      DG9:Eq: 4.13
|
|  Pz:       0.000     T     Vy:        0.000           Vx:      0.000
|
|  Tz:       3.000           My:        0.000           Mx:      0.000
|
|-----------------------------------------------------------------------------
|
| TENSION SLENDERNESS
|
| Actual Slenderness Ratio    :     41.379
|
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00
|
|-----------------------------------------------------------------------------
|
| STRENGTH CHECKS
|
| Critical L/C  :      1            Ratio     :        0.242(PASS)
|
|          Loc  :    0.00           Condition :    DG9:Eq: 4.13
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )
|
| Ag  :   2.760E+00     Axx :   1.500E+00     Ayy :   1.500E+00
|
| Ixx :   2.216E+00     Iyy :   2.216E+00     J   :   2.300E-01
|
| Sxx+:   7.071E-01     Sxx-:   1.149E+00     Zxx :   1.273E+00
|
| Syy+:   1.652E+00     Syy-:   1.652E+00     Zyy :   2.974E+00
|
| Cw  :   1.444E-01     x0  :   6.784E-01     y0  :   6.784E-01
|
|-----------------------------------------------------------------------------
|
| MATERIAL PROPERTIES
|
| Fyld:          50.000              Fu:          50.000
|
|-----------------------------------------------------------------------------
|
| Actual Member Length:        24.000
|
| Design Parameters                                                (Rolled)
|
| Kx:    1.00  Ky:    1.00  NSF:    1.00  SLF:    1.00  CSP:   12.00
|
|-----------------------------------------------------------------------------
|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    24.00)
|
```

```
|                            λ          λp         λr        CASE
|
| Flange: NonSlender       6.00       N/A      10.84     Table.4.1a.Case3
|
| Web   : NonSlender       6.00       N/A      10.84     Table.4.1a.Case3
|
|
|
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    24.00)
|
|                            λ          λp         λr        CASE
|
| Flange: Compact          6.00      13.00     21.92     Table.4.1b.Case12
|
| Web   : Compact          6.00      13.00     21.92     Table.4.1b.Case12
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
5
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                      - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL TENSION
|
|-----------------------------------------------------------------------------
|
| TENSILE YIELDING
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       124.2       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Nom. Ten. Yld Cap         : Pn      =  138.00     kip        Eq.D2-1
|
|-----------------------------------------------------------------------------
|
| TENSILE RUPTURE
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       103.5       0.000     Cl.D2          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective area            : Ae      =  2.7600     in2        Eq.D3-1
|
|  Nom. Ten. Rpt Cap         : Pn      =  138.00     kip        Eq.D2-2
|
```

```
|-----------------------------------------------------------------------------
|
|                          CHECKS FOR AXIAL COMPRESSION
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       117.9       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcx/rx =  26.782                Cl.E2
|
|  Elastic Buckling Stress   : Fex    =  399.04     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrx   =  47.445     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnx    =  130.95     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING Y
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       117.9       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcy/ry =  26.782                Cl.E2
|
|  Elastic Buckling Stress   : Fey    =  399.04     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcry   =  47.445     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pny    =  130.95     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING U
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       120.1       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcu/ru =  21.299                Cl.E2
|
```

```
|  Elastic Buckling Stress   : Feu    =  630.92     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcru   =  48.369     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnu    =  133.50     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL BUCKLING V
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       109.6       0.000     Cl.E3          1      0.00
|
|
|
| Intermediate Results :
|
|  Effective Slenderness     : Lcv/rv =  41.379                Cl.E2
|
|  Elastic Buckling Stress   : Fev    =  167.16     ksi        Eq.E3-4
|
|  Crit. Buckling Stress     : Fcrv   =  44.116     ksi        Eq.E3-2
|
|  Nom. Flexural Buckling    : Pnv    =  121.76     kip        Eq.E3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                               -- PAGE NO.
6
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                        - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                        CHECKS FOR SHEAR
|
|-----------------------------------------------------------------------------
|
| SHEAR ALONG X
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       40.50       0.000     Cl.G1          1      0.00
|
|
|
| Intermediate Results :
|
|  Coefficient Cv Along X    : Cv     =  1.0000                Eq.G2-9
|
|  Coefficient Kv Along X    : Kv     =  1.2000                Cl.G3
|
|  Nom. Shear Along X        : Vnx    =  45.000     kip        Eq.G6-1
|
|-----------------------------------------------------------------------------
|
```

```
|  SHEAR ALONG Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       40.50       0.000     Cl.G1          1      0.00
|
|
|
|  Intermediate Results :
|
|   Coefficient Cv Along Y    : Cv     =  1.0000              Eq.G2-9
|
|   Coefficient Kv Along Y    : Kv     =  1.2000              Cl.G3
|
|   Nom. Shear Along Y        : Vny    =  45.000     kip      Eq.G3-1
|
|-----------------------------------------------------------------------------
|
     STAAD SPACE                                             -- PAGE NO.
7
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                       - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR BENDING
|
|-----------------------------------------------------------------------------
|
|  FLEXURAL YIELDING (X)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       72.22       0.000     Cl.F10.1       1      0.00
|
|
|
|  Intermediate Results :
|
|   Nom Flex Yielding Along X : Mnx    =  80.244     kip-in     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
|  FLEXURAL YIELDING (Y)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       72.22       0.000     Cl.F10.1       1      0.00
|
|
|
|  Intermediate Results :
|
|   Nom Flex Yielding Along Y : Mny    =  80.244     kip-in     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
```

```
| FLEXURAL YIELDING (U)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       111.5       0.000     Cl.F10.1       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along U : Mnu    =  123.90     kip-in     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| FLEXURAL YIELDING (V)
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       47.73       0.000     Cl.F10.1       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding Along V : Mnv    =  53.034     kip-in     Eq.F10-1
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT X
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       57.78       0.000     Cl.F10.2       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom L-T-B Cap             : Mnx    =  64.196     kip-in     Eq.F10-2
|
|  Mom. Distr. factor        : CbX    =  1.0000                Custom
|
|  Crit. Elastic L-T-B Mom.  : McrX   =  3076.5     kip-in     Eq.F10-5b
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT Y
|
|              DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|              0.000       57.78       0.000     Cl.F10.2       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About Y : Mny    =  64.196     kip-in     Eq.F10-2
|
```

```
|  Mod. factor Cb            : CbY    =  1.0000                Custom
|
|  Crit. Elas. L-T-B Moment  : McrY   =  3076.5     kip-in     Eq.F10-5b
|
|-----------------------------------------------------------------------------
|
| LAT TOR BUCK ABOUT U
|
|             DEMAND      CAPACITY    RATIO     REFERENCE     L/C    LOC
|
|             0.000       111.5       0.000     Cl.F10.2       1      0.00
|
|
|
| Intermediate Results :
|
|  Nom Flex Yielding About U : Mnu    =  123.90     kip-in     Eq.F10-2
|
|  modification factor Cb    : CbU    =  1.0000                Custom
|
|  Crit. Elast. L-T-B Moment : McrU   =  1088.0     kip-in     Eq.F10-4
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted).
                                                         - Member :     1 Contd.
|-----------------------------------------------------------------------------
|
|                         CHECKS FOR TORSION
|
|-----------------------------------------------------------------------------
|
| TORSION - NORMAL STRESS RATIO
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.000      DG9:Eq: 4.12        1        0.00
|
|
|
| Intermediate Results :
|
|  Torque                    : Tq     =  3.0000     kip-in     DG9:Eq: 3.1
|
|  Mod. Factor               : Phi    =  1.0000                DG9:Eq: 4.22
|
|-----------------------------------------------------------------------------
|
| TORSION - SHEAR STRESS RATIO
|
|                           RATIO      CRITERIA          L/C      LOC
|
|                           0.242      DG9:Eq: 4.13        1        0.00
|
```

```
|---------------------------------------------------------------------------
|
| TORSION - COMB. BUCK. RATIO
|
|                           RATIO      CRITERIA           L/C       LOC
|
|                           0.000      DG9:Eq: 4.16(b)     1         0.00
|
|---------------------------------------------------------------------------
|
```

V. AISC 360-10 W Flex Memb F.1-1

Verify the major axis flexural strength of a wide flange section which is continuously braced per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F. 1-1A and F.1-1B, pp. F6-F8

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Details

From reference (2):

> Select an ASTM A992 W-shape beam with a simple span of 35 ft. Limit the member to a maximum nominal depth of 18 in. Limit the live load deflection to L/360. The nominal loads are a uniform dead load of 0.45 kip/ft and a uniform live load of 0.75 kip/ft. Assume the beam is continuously braced.

See example F.1-1A and F.1-1B in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	379	379	none	
M_n / Ω_b (ft·k) [ASD]	252	252	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W Flex Memb F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 9360 ALL
```

```
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FU 9360 ALL
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.213         3
                        0.00            0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     3.213(FAIL)                |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03             |
| Szz:   8.889E+01     Syy:   1.069E+01                                |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1  |
|                Slender        45.23     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10 |
|                Compact        45.23    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
7
```

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   7.17E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   5.09E+02   0.000      Eq. E7-1      3     0.00 |
| Min Buck    0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.59E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r       Fcr        Fe         Pn          |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02        |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    1.02E-01   2.82E+03   2.88E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     5.94E-02     1.00      1.20     6.58   2.56E+02           |
| Local-Y     4.44E-02     1.00      0.00    45.23   1.92E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   3.79E+02   0.703      Eq. F2-1      3    17.50 |
| Minor       0.00E+00   6.23E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       4.21E+02   0.00E+00                                       |
| Minor       6.92E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   8.29E+01   3.213      Eq. F2-3      3    17.50 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb     |
| Major       9.21E+01   0.00E+00    1.00     5.83    16.97    35.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         3.213      Eq. H1-1b     3    17.50              |
```

```
| Flexure Tens          3.213      Eq. H1-1b     3    17.50              |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         8.29E+01   2.66E+02   5.14E+01                  |
|                      6.23E+01   0.00E+00   0.00E+00                  |
| Flexure Tens         8.29E+01   2.66E+02   6.61E+02                  |
|                      6.23E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2010
    59. METHOD ASD
    60. FU 9360 ALL
    61. FYLD 7200 ALL
    62. TRACK 2 ALL
    63. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                  (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.331         4
                        0.00            0.00        -183.75       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     3.331(FAIL)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03             |
| Szz:   8.889E+01     Syy:   1.069E+01                                 |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1  |
|                Slender        45.23     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10 |
```

```
|                Compact        45.23    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   3.39E+02   0.000      Eq. E7-1      4     0.00 |
| Min Buck    0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.73E+02   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02        |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.02E-01   2.82E+03   2.88E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     5.94E-02     1.00     1.20     6.58   2.56E+02            |
| Local-Y     4.44E-02     1.00     0.00    45.23   1.92E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.84E+02   2.52E+02   0.729      Eq. F2-1      4    17.50 |
| Minor       0.00E+00   4.14E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.21E+02   0.00E+00                                       |
| Minor       6.92E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.84E+02   5.52E+01   3.331      Eq. F2-3      4    17.50 |
| Intermediate   Mn         Me        Cb       Lp       Lr       Lb    |
| Major       9.21E+01   0.00E+00    1.00     5.83    16.97    35.00    |
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                   |
|                      RATIO      CRITERIA    L/C     LOC                   |
| Flexure Comp          3.331      Eq. H1-1b     4    17.50                 |
| Flexure Tens          3.331      Eq. H1-1b     4    17.50                 |
| Intermediate        Mcx /      Mrx /      Pc /                            |
|                     Mcy        Mry        Pr                              |
| Flexure Comp        5.52E+01   1.84E+02   3.42E+01                        |
|                     4.14E+01   0.00E+00   0.00E+00                        |
| Flexure Tens        5.52E+01   1.84E+02   4.40E+02                        |
|                     4.14E+01   0.00E+00   0.00E+00                        |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 W LTB Test F.1-2

Verify the lateral-torsional buckling flexural strength of a wide flange section that is braced at the third points per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F. 1-2A and F.1-2B, pp. F9 - F11

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Details

From reference (2):

> Verify the available flexural strength of the W18×50, ASTM A992 beam selected in Example F.1-1A with the beam braced at the ends and third points.

See example F.1-2A and F.1-2B in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	305	306	negligible	
M_n / Ω_b (ft·k) [ASD]	203	203	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W LTB Test F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 9360 ALL
FYLD 7200 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
```

```
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FU 9360 ALL
FYLD 7200 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.872         3
                        0.00            0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.872(PASS)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
| Szz:   8.889E+01     Syy:   1.069E+01                                 |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1  |
|                Slender        45.23     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10 |
|                Compact        45.23    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
7
```

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                     |
|                                                                             |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Yield       0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00       |
| Rupture     0.00E+00   7.17E+02   0.000      Eq. D2-2      3     0.00       |
|-----------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                                 |
|                                                                             |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Maj Buck    0.00E+00   5.09E+02   0.000      Eq. E7-1      3     0.00       |
| Min Buck    0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00       |
| Flexural                                                                    |
| Tor Buck    0.00E+00   4.81E+02   0.000      Eq. E7-1      3     0.00       |
| Intermediate                                                                |
| Results     Eff Area     KL/r      Fcr        Fe         Pn                |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02             |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01             |
| Flexural       Ag        Fcr        Pn                                      |
| Tor Buck    9.97E-02   5.24E+03   5.34E+02                                  |
|-----------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                             |
|                                                                             |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Local-Z     0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00       |
| Local-Y     3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00       |
| Intermediate                                                                |
| Results        Aw         Cv        Kv      h/tw      Vn                    |
| Local-Z     5.94E-02     1.00      1.20     6.58   2.56E+02                 |
| Local-Y     4.44E-02     1.00      0.00    45.23   1.92E+02                 |
|-----------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                                  |
|                                                                             |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Major       2.66E+02   3.79E+02   0.703      Eq. F2-1      3    17.50       |
| Minor       0.00E+00   6.23E+01   0.000      Eq. F6-1      3     0.00       |
| Intermediate   Mn         My                                                |
| Major       4.21E+02   0.00E+00                                             |
| Minor       6.92E+01   0.00E+00                                             |
|-----------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                                |
|                                                                             |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Major       2.66E+02   3.06E+02   0.872      Eq. F2-2      3    17.50       |
| Intermediate   Mn         Me        Cb       Lp        Lr       Lb         |
| Major       3.39E+02   0.00E+00    1.01     5.83    16.97    11.67         |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                     |
|                        RATIO      CRITERIA    L/C     LOC                   |
| Flexure Comp           0.872      Eq. H1-1b     3    17.50                  |
```

```
| Flexure Tens          0.872      Eq. H1-1b     3    17.50              |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         3.06E+02   2.66E+02   5.14E+01                   |
|                      6.23E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         3.06E+02   2.66E+02   6.61E+02                   |
|                      6.23E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2010
    55. METHOD ASD
    56. FU 9360 ALL
    57. FYLD 7200 ALL
    58. CB 1.01 ALL
    59. LX 11.67 ALL
    60. UNB 11.67 ALL
    61. UNT 11.67 ALL
    62. TRACK 2 ALL
    63. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                  -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX              MY            MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.904         4
                        0.00            0.00       -183.75       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.904(PASS)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
| Szz:   8.889E+01     Syy:   1.069E+01                                 |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
```

```
|                STIFFENED                                                 |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1    |
|                Slender        45.23     N/A     35.88    T.B4.1(a)-5    |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10   |
|                Compact        45.23    90.55   137.27    T.B4.1(b)-15   |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00    |
| Rupture    0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00    |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   0.00E+00   3.39E+02   0.000      Eq. E7-1      4     0.00    |
| Min Buck   0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00    |
| Flexural                                                                |
| Tor Buck   0.00E+00   3.20E+02   0.000      Eq. E7-1      4     0.00    |
| Intermediate                                                            |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02           |
| Min Buck   1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01           |
| Flexural      Ag       Fcr        Pn                                    |
| Tor Buck   9.97E-02   5.24E+03   5.34E+02                               |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z    0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00    |
| Local-Y    2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00    |
| Intermediate                                                            |
| Results       Aw         Cv        Kv      h/tw      Vn                 |
| Local-Z    5.94E-02     1.00      1.20     6.58   2.56E+02              |
| Local-Y    4.44E-02     1.00      0.00    45.23   1.92E+02              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      1.84E+02   2.52E+02   0.729      Eq. F2-1      4    17.50    |
| Minor      0.00E+00   4.14E+01   0.000      Eq. F6-1      4     0.00    |
| Intermediate   Mn        My                                             |
| Major      4.21E+02   0.00E+00                                          |
| Minor      6.92E+01   0.00E+00                                          |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      1.84E+02   2.03E+02   0.904      Eq. F2-2      4    17.50    |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb       |
| Major      3.39E+02   0.00E+00    1.01     5.83    16.97    11.67       |
     STAAD SPACE                                              -- PAGE NO.
11
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.904      Eq. H1-1b     4    17.50              |
| Flexure Tens         0.904      Eq. H1-1b     4    17.50              |
| Intermediate        Mcx /      Mrx /      Pc /                        |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        2.03E+02   1.84E+02   3.42E+01                    |
|                     4.14E+01   0.00E+00   0.00E+00                    |
| Flexure Tens        2.03E+02   1.84E+02   4.40E+02                    |
|                     4.14E+01   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 W LTB Test F.1-3B

Reference

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F. 1-3B, pp. F14 - F15

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Problem

From reference (2):

> Verify the available flexural strength of the W18×50, ASTM A992 beam selected in Example F.1-1A with the beam braced at the ends and center point.

See example F.1-3B in reference (2) for calculations.

Comparison

Table 747: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·kips) [LRFD]	289	3,460 in-kips = 288.3	negligible	
M_n/Ω_b (ft·kips) [ASD]	191	2,300 in-kips = 191.7	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 - F.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-OCT-12
JOB NAME AISC 360-10 Validation
JOB CLIENT Bentley
JOB NO 7
END JOB INFORMATION
************************************************************
* AISC Example F.1-3
* W-SHAPE FLEXURAL MEMBERDESIGN IN STRONG-AXIS BENDING,
* BRACED AT MIDSPAN
* (1 analytical member with 2 unbraced segment)
************************************************************
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 420 0 0;
MEMBER INCIDENCES
1 1 2;
***************************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 59 RY 1.5 RT 1.2
END DEFINE MATERIAL
***************************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
***************************************************
CONSTANTS
MATERIAL STEEL ALL
***************************************************
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
***************************************************
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.0375
**
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.0625
**
```

```
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
**
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
***************************************************
PERFORM ANALYSIS
**
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
*
* ASTM A992
FYLD 50 ALL
FU 65 ALL
*CB 0 ALL
CB 1.3 ALL
*
LX 210 ALL
UNT 210 ALL
UNB 210 ALL
*
MAIN 1 ALL
*
TRACK 2 ALL
CHECK CODE ALL
***************************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
*
* ASTM A992
FYLD 50 ALL
FU 65 ALL
*CB 0 ALL
CB 1.3 ALL
*
LX 210 ALL
UNT 210 ALL
UNB 210 ALL
*
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.924         3
```

```
                     0.00            0.00       -3197.25      210.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    254.294  L/C  :     3                |
 | Allowable Slenderness Ratio :    300.000  LOC  :   210.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.924(PASS)                 |
 |          Loc  :  210.00   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -3.197E+03               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
 | Szz:   8.889E+01     Syy:   1.069E+01                                 |
 | Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          65.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:       420.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1  |
 |                Slender        45.23     N/A     35.88    T.B4.1(a)-5  |
 | Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10 |
 |                Compact        45.23    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00 |
 | Rupture     0.00E+00   7.17E+02   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck    0.00E+00   5.09E+02   0.000      Eq. E7-1      3     0.00 |
 | Min Buck    0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   3.92E+02   0.000      Eq. E7-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.42E+01    56.93   3.85E+01   8.83E+01   5.65E+02        |
 | Min Buck    1.47E+01   254.29   3.88E+00   4.43E+00   5.71E+01        |
 | Flexural       Ag        Fcr        Pn                                |
 | Tor Buck    1.47E+01   2.96E+01   4.36E+02                            |
 |-----------------------------------------------------------------------|
```

```
| CHECK FOR SHEAR                                                      |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z    0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y    3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00 |
| Intermediate                                                         |
| Results        Aw        Cv       Kv      h/tw      Vn              |
| Local-Z    8.55E+00     1.00     1.20     6.58   2.56E+02            |
| Local-Y    6.39E+00     1.00     0.00    45.23   1.92E+02            |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      3.20E+03   4.54E+03   0.703      Eq. F2-1      3   210.00 |
| Minor      0.00E+00   7.47E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                          |
| Major      5.05E+03   0.00E+00                                       |
| Minor      8.30E+02   0.00E+00                                       |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      3.20E+03   3.46E+03   0.924      Eq. F2-3      3   210.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb     |
| Major      3.84E+03   0.00E+00    1.30    70.01   203.60   210.00   |
     STAAD SPACE                                            -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp          0.924      Eq. H1-1b     3   210.00           |
| Flexure Tens          0.924      Eq. H1-1b     3   210.00           |
| Intermediate         Mcx /      Mrx /      Pc /                    |
|                      Mcy        Mry        Pr                      |
| Flexure Comp         3.46E+03   3.20E+03   5.14E+01                 |
|                      7.47E+02   0.00E+00   0.00E+00                 |
| Flexure Tens         3.46E+03   3.20E+03   6.61E+02                 |
|                      7.47E+02   0.00E+00   0.00E+00                 |
|----------------------------------------------------------------------|
    77. ***************************************************
    78. LOAD LIST 4
    79. PARAMETER 2
    80. CODE AISC UNIFIED 2010
    81. METHOD ASD
    82. *
    83. * ASTM A992
    84. FYLD 50 ALL
    85. FU 65 ALL
    86. *CB 0 ALL
    87. CB 1.3 ALL
    88. *
    89. LX 210 ALL
    90. UNT 210 ALL
    91. UNB 210 ALL
    92. *
```

```
   93. TRACK 2 ALL
   94. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W18X50                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.958         4
                        0.00            0.00       -2205.00      210.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    254.294  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :   210.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.958(PASS)                 |
 |          Loc  :  210.00   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.205E+03               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
 | Szz:   8.889E+01     Syy:   1.069E+01                                 |
 | Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          65.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:       420.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1(a)-1  |
 |                Slender        45.23     N/A     35.88    T.B4.1(a)-5  |
 | Flexure     :  Compact         6.58     9.15    24.08    T.B4.1(b)-10 |
 |                Compact        45.23    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
```

```
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   3.39E+02   0.000      Eq. E7-1      4     0.00  |
| Min Buck    0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00  |
| Flexural                                                                 |
| Tor Buck    0.00E+00   2.61E+02   0.000      Eq. E7-1      4     0.00  |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    1.42E+01    56.93   3.85E+01   8.83E+01   5.65E+02          |
| Min Buck    1.47E+01   254.29   3.88E+00   4.43E+00   5.71E+01          |
| Flexural       Ag       Fcr        Pn                                    |
| Tor Buck    1.47E+01   2.96E+01   4.36E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00  |
| Local-Y     2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00  |
| Intermediate                                                             |
| Results        Aw         Cv       Kv      h/tw      Vn                |
| Local-Z     8.55E+00     1.00     1.20     6.58   2.56E+02              |
| Local-Y     6.39E+00     1.00     0.00    45.23   1.92E+02              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       2.20E+03   3.02E+03   0.729      Eq. F2-1      4   210.00  |
| Minor       0.00E+00   4.97E+02   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn        My                                            |
| Major       5.05E+03   0.00E+00                                         |
| Minor       8.30E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       2.20E+03   2.30E+03   0.958      Eq. F2-3      4   210.00  |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb       |
| Major       3.84E+03   0.00E+00    1.30    70.01   203.60   210.00     |
    STAAD SPACE                                                -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                      RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp          0.958      Eq. H1-1b     4   210.00               |
| Flexure Tens          0.958      Eq. H1-1b     4   210.00               |
| Intermediate         Mcx /      Mrx /      Pc /                         |
|                      Mcy        Mry        Pr                           |
| Flexure Comp         2.30E+03   2.20E+03   3.42E+01                     |
|                      4.97E+02   0.00E+00   0.00E+00                     |
| Flexure Tens         2.30E+03   2.20E+03   4.40E+02                     |
|                      4.97E+02   0.00E+00   0.00E+00                     |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 C Flex Mem F.2-1

Verify the major axis flexural strength of a compact channel section which is continuously braced per the LRFD and ASD methods for AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F. 2-1A and F.2-1B, pp. F16-F18

Details

From reference (2):

> Select an ASTM A36 channel to serve as a roof edge beam with a simple span of 25 ft. Limit the live load deflection to L/360. The nominal loads are a uniform dead load of 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.23 \text{ k/ft}) + 1.6(0.69 \text{ k/ft}) = 1.38 \text{ k/ft}$$

$$M_u = \frac{1.38(25)^2}{8} = 108 \text{ ft-k}$$

Service moment:

$$w_a = 0.23 \text{ k/ft} + 0.69 \text{ k/ft} = 0.92 \text{ k/ft}$$

$$M_a = \frac{0.92(25)^2}{8} = 71.9 \text{ ft-k}$$

Try a C15×33.9, which has a plastic section modulus, $Z_x = 50.8\ in^3$.

All ASTM A36 grade C-shapes are compact. The member is continuously braced, so therefore the yielding limit state governs.

$$M_n = M_p = F_y \times Z_x = 36 \times 50.8 = 1{,}830 \text{ in-k} = 152 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(152) = 137\ ft\text{-}k$

ASD: $M_n / \Omega_b = {}^{152}/_{1.67} = 91.0\ ft\text{-}k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	137	137	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
M_n/Ω_b (ft·k) [ASD]	91	91.3	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 C Flex Mem F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
```

```
PARAMETER 2
CODE AISC UNIFIED 2010
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 3
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 5184 ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                  (AISC SECTIONS)
                           FAIL   Eq. H1-1b           2.548         3
                        0.00            0.00        -107.81       12.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    333.952  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    12.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     2.548(FAIL)                |
|          Loc  :   12.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.078E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1 |
|                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5 |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10 |
|                Compact        34.25   106.72   161.78    T.B4.1(b)-15 |
      STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
```

```
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00  |
 | Rupture    0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   2.79E+02   0.000      Eq. E3-1      3     0.00  |
 | Min Buck   0.00E+00   2.03E+01   0.000      Eq. E3-1      3     0.00  |
 | Flexural                                                              |
 | Tor Buck   0.00E+00   2.07E+02   0.000      Eq. E4-1      3     0.00  |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck   6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02         |
 | Min Buck   6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01         |
 | Flexural      Ag        Fcr        Pn                                 |
 | Tor Buck   6.94E-02   3.31E+03   2.30E+02                             |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00  |
 | Local-Y    1.72E+01   1.17E+02   0.148      Eq. G2-1      3     0.00  |
 | Intermediate                                                          |
 | Results       Aw        Cv       Kv      h/tw      Vn                 |
 | Local-Z    3.07E-02     1.00     1.20     5.23   9.55E+01             |
 | Local-Y    4.17E-02     1.00     5.00    34.25   1.30E+02             |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      1.08E+02   1.37E+02   0.786      Eq. F2-1      3    12.50  |
 | Minor      0.00E+00   1.37E+01   0.000      Eq. F6-1      3     0.00  |
 | Intermediate   Mn        My                                           |
 | Major      1.52E+02   0.00E+00                                        |
 | Minor      1.53E+01   0.00E+00                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      1.08E+02   4.23E+01   2.548      Eq. F2-3      3    12.50  |
 | Intermediate   Mn        Me        Cb      Lp       Lr       Lb       |
 | Major      4.70E+01   0.00E+00    1.00     3.74    14.49    25.00     |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
 |                       RATIO      CRITERIA    L/C     LOC              |
 | Flexure Comp          2.548      Eq. H1-1b     3    12.50             |
 | Flexure Tens          2.548      Eq. H1-1b     3    12.50             |
```

```
| Intermediate         Mcx /      Mrx /      Pc /                        |
|                      Mcy        Mry        Pr                          |
| Flexure Comp         4.23E+01   1.08E+02   2.03E+01                    |
|                      1.37E+01   0.00E+00   0.00E+00                    |
| Flexure Tens         4.23E+01   1.08E+02   3.24E+02                    |
|                      1.37E+01   0.00E+00   0.00E+00                    |
|------------------------------------------------------------------------|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2010
    50. FLX 2 ALL
    51. METHOD ASD
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b            2.553         4
                        0.00            0.00         -71.88       12.50
|------------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :    333.952  L/C  :     4                 |
| Allowable Slenderness Ratio :    300.000  LOC  :    12.50              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     2.553(FAIL)                  |
|          Loc  :   12.50   Condition :   Eq. H1-1b                      |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -7.188E+01                |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02              |
| Szz:   4.200E+01     Syy:   3.178E+00                                  |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00              |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:        5183.999   Fu:        8351.999                           |
|------------------------------------------------------------------------|
| Actual Member Length:        25.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                               |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1   |
|                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5   |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10  |
|                Compact        34.25   106.72   161.78    T.B4.1(b)-15  |
      STAAD SPACE                                              -- PAGE NO.
```

```
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00  |
 | Rupture    0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   1.85E+02   0.000      Eq. E3-1      4     0.00  |
 | Min Buck   0.00E+00   1.35E+01   0.000      Eq. E3-1      4     0.00  |
 | Flexural                                                              |
 | Tor Buck   0.00E+00   1.38E+02   0.000      Eq. E4-1      4     0.00  |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck   6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02         |
 | Min Buck   6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01         |
 | Flexural      Ag        Fcr        Pn                                 |
 | Tor Buck   6.94E-02   3.31E+03   2.30E+02                             |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00  |
 | Local-Y    1.15E+01   7.76E+01   0.148      Eq. G2-1      4     0.00  |
 | Intermediate                                                          |
 | Results       Aw         Cv       Kv      h/tw      Vn                |
 | Local-Z    3.07E-02     1.00     1.20     5.23   9.55E+01             |
 | Local-Y    4.17E-02     1.00     5.00    34.25   1.30E+02             |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      7.19E+01   9.13E+01   0.788      Eq. F2-1      4    12.50  |
 | Minor      0.00E+00   9.14E+00   0.000      Eq. F6-1      4     0.00  |
 | Intermediate   Mn         My                                          |
 | Major      1.52E+02   0.00E+00                                        |
 | Minor      1.53E+01   0.00E+00                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      7.19E+01   2.82E+01   2.553      Eq. F2-3      4    12.50  |
 | Intermediate   Mn         Me        Cb       Lp       Lr       Lb     |
 | Major      4.70E+01   0.00E+00    1.00     3.74    14.49    25.00     |
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
 |                        RATIO      CRITERIA    L/C     LOC             |
```

```
| Flexure Comp          2.553      Eq. H1-1b     4    12.50                   |
| Flexure Tens          2.553      Eq. H1-1b     4    12.50                   |
| Intermediate         Mcx /      Mrx /      Pc /                             |
|                      Mcy        Mry        Pr                               |
| Flexure Comp         2.82E+01   7.19E+01   1.35E+01                         |
|                      9.14E+00   0.00E+00   0.00E+00                         |
| Flexure Tens         2.82E+01   7.19E+01   2.16E+02                         |
|                      9.14E+00   0.00E+00   0.00E+00                         |
|---------------------------------------------------------------------------|
```

V. AISC 360-10 C LTB Test F.2-2

Verify the lateral flexural strength of a compact channel section braced at the fifth points per the LRFD and ASD methods of the AISC 360-10 code.

Reference

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F. 2-2A and F.2-2B, pp F-19 - F21

Problem

Check a C15x33.9 channel of ASTM A36 steel which serves as a roof edge beam. The simple span is 25 ft. The nominal loads on the member are a dead load 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The beam is braced at fifth points and at ends.

Validation

Ultimate moment:

$$w_u = 1.2(0.23\text{ k/ft}) + 1.6(0.69\text{ k/ft}) = 1.38\text{ k/ft}$$

$$M_u = \frac{1.38(25)^2}{8} = 108\text{ ft-k}$$

Service moment:

$$w_a = 0.23\text{ k/ft} + 0.69\text{ k/ft} = 0.92\text{ k/ft}$$

$$M_a = \frac{0.92(25)^2}{8} = 71.9\text{ ft-k}$$

The unbraced length, >$L_b = 5\ ft$.

Try a C15×33.9, which has a plastic section modulus, $Z_x = 50.8\ in^3$.

The lateral-torsional buckling modification factor, $C_b = 1.0$ for the center span of this bracing & loading condition.

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76(0.901)\sqrt{\frac{29,000}{36}}\left(\frac{1}{12}\right) = 3.75\text{ft} \quad \text{(F2-5)}$$

$$L_r = \pi r_{ts}\sqrt{\frac{E}{0.7F_y}} = \pi(1.13)\sqrt{\frac{29,000}{0.7 \times 36}}\left(\frac{1}{12}\right) = 10.0 \text{ ft} \tag{F2-6}$$

Note: This is the simplified, conservative calculation for L_r per the User Note used to verify that the section is in the plastic-elastic range. The reference uses the more exact calculation but the resulting flexural strength values are essentially equal.

$$L_p < L_b \leq L_r$$

All ASTM A36 grade C-shapes are compact.

The nominal flexural strength for the flexural yielding limit state is:

$$M_n = M_p = F_y \times Z_x = 36 \times 50.8 = 1{,}830 \text{ in-k} = 152 \text{ ft-k}$$

The nominal flexural strength for the lateral-torsional buckling limit state is:

$$M_n = C_b\left[M_p - (M_p - 0.7F_y S_x)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right]$$

$$= 1.0\left[1,830 - (1,830 - 0.7 \times 36 \times 42)\left(\frac{5 - 3.75}{10 - 3.75}\right)\right]\left(\frac{1}{12}\right) = 145 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(145) = 131 \text{ ft-k}$

ASD: $M_n / \Omega_b = {}^{131}/_{1.67} = 86.8 \text{ ft-k}$

Comparison

Table 748: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (kips) [LRFD]	131	130	negligible	
M_n/Ω_b (kips) [ASD]	86.8	86.7	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 C LTB Test F.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Mar-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FYLD 5184 ALL
FU 8352 ALL
CB 1 ALL
FLX 1 ALL
METHOD LRFD
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 5184 ALL
FU 8352 ALL
CB 1 ALL
FLX 1 ALL
METHOD ASD
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                    (AISC SECTIONS)
                           FAIL   SLENDERNESS          1.113         3
                       0.00            0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    333.952  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.113(FAIL)                |
|          Loc  :    0.00   Condition :   SLENDERNESS                   |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.725E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                 |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1  |
|                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5  |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10 |
|                Compact        34.25   106.72   161.78    T.B4.1(b)-15 |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   2.79E+02   0.000      Eq. E3-1      3     0.00 |
```

```
| Min Buck    0.00E+00   2.03E+01   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                               |
| Tor Buck    0.00E+00   2.07E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02         |
| Min Buck    6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01         |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    6.94E-02   3.31E+03   2.30E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     1.72E+01   1.17E+02   0.148      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw         Cv        Kv      h/tw      Vn              |
| Local-Z     3.07E-02     1.00      1.20     5.23   9.55E+01            |
| Local-Y     4.17E-02     1.00      5.00    34.25   1.30E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.08E+02   1.37E+02   0.786      Eq. F2-1      3    12.50 |
| Minor       0.00E+00   1.37E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                           |
| Major       1.52E+02   0.00E+00                                       |
| Minor       1.53E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.08E+02   1.30E+02   0.827      Eq. F2-2      3    12.50 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major       1.45E+02   0.00E+00    1.00     3.74    14.49     5.00    |
    STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.827      Eq. H1-1b     3    12.50             |
| Flexure Tens          0.827      Eq. H1-1b     3    12.50             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         1.30E+02   1.08E+02   2.03E+01                   |
|                      1.37E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         1.30E+02   1.08E+02   3.24E+02                   |
|                      1.37E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    51. LOAD LIST 4
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2010
    54. FYLD 5184 ALL
    55. FU 8352 ALL
    56. CB 1 ALL
```

```
   57. FLX 1 ALL
   58. METHOD ASD
   59. TRACK 2 ALL
   60. UNB 5 ALL
   61. UNT 5 ALL
   62. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                   (AISC SECTIONS)
                           FAIL   SLENDERNESS          1.113         4
                        0.00            0.00           0.00       0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    333.952  L/C  :     4                 |
 | Allowable Slenderness Ratio :    300.000  LOC  :     0.00              |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     1.113(FAIL)                  |
 |          Loc  :    0.00   Condition :   SLENDERNESS                    |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   1.150E+01     Fz:   0.000E+00                |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02               |
 | Szz:   4.200E+01     Syy:   3.178E+00                                  |
 | Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00               |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        5183.999   Fu:        8351.999                            |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        25.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1 |
 |                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5 |
 | Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10 |
 |                Compact        34.25   106.72   161.78    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Yield       0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.85E+02   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   1.35E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.38E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02        |
| Min Buck    6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    6.94E-02   3.31E+03   2.30E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.15E+01   7.76E+01   0.148      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     3.07E-02     1.00     1.20      5.23   9.55E+01           |
| Local-Y     4.17E-02     1.00     5.00     34.25   1.30E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       7.19E+01   9.13E+01   0.788      Eq. F2-1      4    12.50 |
| Minor       0.00E+00   9.14E+00   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.52E+02   0.00E+00                                       |
| Minor       1.53E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       7.19E+01   8.67E+01   0.829      Eq. F2-2      4    12.50 |
| Intermediate   Mn         Me        Cb       Lp       Lr       Lb   |
| Major       1.45E+02   0.00E+00    1.00     3.74    14.49     5.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.829      Eq. H1-1b     4    12.50           |
| Flexure Tens           0.829      Eq. H1-1b     4    12.50           |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          8.67E+01   7.19E+01   1.35E+01                  |
|                       9.14E+00   0.00E+00   0.00E+00                  |
| Flexure Tens          8.67E+01   7.19E+01   2.16E+02                  |
|                       9.14E+00   0.00E+00   0.00E+00                  |
```

```
|--------------------------------------------------------------------|
   63. FINISH
```

V. AISC 360-10 W Local Buckling F.3

Verify the leg local buckling flexural strength of a non-compact wide flange section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.3A and F.3B, pp. F22 - F25

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Details

From reference (2):

> Select an ASTM A992 W-shape beam with a simple span of 40 ft. The nominal loads are a uniform dead load of 0.05 kip/ft and two equal 18 kip concentrated live loads acting at the third points of the beam. The beam is continuously braced.

See example F.3A and F.3B in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	398	398	none	
M_n / Ω_b (ft·k) [ASD]	265	265	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W Local Buckling F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.05
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -18 13.333
1 CON GY -18 26.667
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           6.470         3
                       0.00            0.00       -395.99       20.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    289.731  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    20.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     6.470(FAIL)                 |
|          Loc  :   20.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -3.960E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03             |
| Szz:   9.311E+01     Syy:   9.509E+00                                |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        40.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.49    T.B4.1(a)-1  |
|                Slender        53.54     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.47     9.15    24.08    T.B4.1(b)-10 |
|                Compact        53.54    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                            -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   6.35E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   6.87E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   4.61E+02   0.000      Eq. E7-1      3     0.00 |
| Min Buck    0.00E+00   3.79E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.54E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
```

```
| Maj Buck    8.92E-02    58.20   5.23E+03   1.22E+04   5.12E+02          |
| Min Buck    9.79E-02   289.73   4.31E+02   4.91E+02   4.22E+01          |
| Flexural       Ag       Fcr        Pn                                   |
| Tor Buck    9.79E-02   1.75E+03   1.72E+02                              |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.89E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     3.00E+01   2.16E+02   0.139      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02            |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   4.01E+02   0.987      Eq. F2-1      3    20.00 |
| Minor       0.00E+00   5.59E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   6.12E+01   6.470      Eq. F2-3      3    20.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       6.80E+01   0.00E+00    1.00     5.85    16.53    40.00    |
    STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   3.98E+02   0.995      Eq. F3-1      3    20.00 |
| Minor       0.00E+00   5.52E+01   0.000      Eq. F6-2      3     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       4.42E+02   0.00E+00                                       |
| Minor       6.14E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          6.470      Eq. H1-1b     3    20.00             |
| Flexure Tens          6.470      Eq. H1-1b     3    20.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         6.12E+01   3.96E+02   3.79E+01                  |
|                      5.52E+01   0.00E+00   0.00E+00                  |
| Flexure Tens         6.12E+01   3.96E+02   6.35E+02                  |
|                      5.52E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
   50. LOAD LIST 4
   51. PARAMETER 2
```

```
    52. CODE AISC UNIFIED 2010
    53. FU 9360 ALL
    54. FYLD 7200 ALL
    55. FLX 2 ALL
    56. METHOD ASD
    57. TRACK 2 ALL
    58. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                   (AISC SECTIONS)
                            FAIL   Eq. H1-1b            6.139         4
                         0.00            0.00        -249.99       20.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    289.731  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    20.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     6.139(FAIL)                 |
|          Loc  :   20.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.500E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03              |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        40.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.49    T.B4.1(a)-1  |
|                Slender        53.54     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.47     9.15    24.08    T.B4.1(b)-10 |
|                Compact        53.54    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
```

```
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   4.22E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.58E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   3.07E+02   0.000      Eq. E7-1      4     0.00 |
| Min Buck    0.00E+00   2.52E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.03E+02   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.92E-02    58.20   5.23E+03   1.22E+04   5.12E+02        |
| Min Buck    9.79E-02   289.73   4.31E+02   4.91E+02   4.22E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    9.79E-02   1.75E+03   1.72E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.26E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.90E+01   1.44E+02   0.132      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02            |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   2.67E+02   0.936      Eq. F2-1      4    20.00 |
| Minor       0.00E+00   3.72E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   4.07E+01   6.139      Eq. F2-3      4    20.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       6.80E+01   0.00E+00    1.00     5.85    16.53    40.00   |
     STAAD SPACE                                           -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   2.65E+02   0.944      Eq. F3-1      4    20.00 |
| Minor       0.00E+00   3.67E+01   0.000      Eq. F6-2      4     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       4.42E+02   0.00E+00                                       |
| Minor       6.14E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                      RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp          6.139      Eq. H1-1b     4    20.00               |
| Flexure Tens          6.139      Eq. H1-1b     4    20.00               |
| Intermediate        Mcx /      Mrx /      Pc /                          |
|                     Mcy        Mry        Pr                            |
| Flexure Comp        4.07E+01   2.50E+02   2.52E+01                      |
|                     3.67E+01   0.00E+00   0.00E+00                      |
| Flexure Tens        4.07E+01   2.50E+02   4.22E+02                      |
|                     3.67E+01   0.00E+00   0.00E+00                      |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 W Memb Selection F.4

Select a wide flange section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.4, pp. F26-F27

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Details

From reference (2):

> Select an ASTM A992 W-shape flexural member by the moment of inertia, to limit the live load deflection to 1 in. The span length is 30 ft. The loads are a uniform dead load of 0.80 kip/ft and a uniform live load of 2 kip/ft. The beam is continuously braced.

See example F.4 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Selected section [LRFD]	W24x55	W24x55	none	
Selected section [ASD]	W24x55	W24x55	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W Memb Selection F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X192
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.8
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
DEFINE ENVELOPE
3 ENVELOPE 1 TYPE STRENGTH
4 ENVELOPE 2 TYPE SERVICEABILITY
END DEFINE ENVELOPE
LOAD LIST ENV 2
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
BEAM 1 ALL
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
TRACK 2 ALL
```

```
FLX 2 ALL
PROFILE W24 ALL
METHOD LRFD
SELECT OPTIMIZED
LOAD LIST ENV 2
PARAMETER 2
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
BEAM 1 ALL
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
TRACK 2 ALL
FLX 2 ALL
PROFILE W24 ALL
METHOD ASD
SELECT OPTIMIZED
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X192                  (AISC SECTIONS)
                           PASS   DEFLECTION           0.186         4
                                                                 15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  1.29E+03  Deflection :     0.280       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.186  (PASS)                       |
|-----------------------------------------------------------------------|
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
4
                     STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864         4
                                                                 15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
```

```
| Ratio                       :     0.864  (PASS)                        |
|-----------------------------------------------------------------------|
   ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 STEEL DESIGN
      STAAD SPACE                                                  -- PAGE NO.
5
                     STAAD.PRO MEMBER SELECTION - (AISC-360-10-LRFD)   v1.4a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864         4
                                                                 15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
   57. LOAD LIST ENV 2
   58. PARAMETER 2
   59. CODE AISC UNIFIED 2010
   60. FU 9360 ALL
   61. FYLD 7200 ALL
   62. BEAM 1 ALL
   63. DFF 240 ALL
   64. DJ1 1 ALL
   65. DJ2 2 ALL
   66. TRACK 2 ALL
   67. FLX 2 ALL
   68. PROFILE W24 ALL
   69. METHOD ASD
   70. SELECT OPTIMIZED
 STEEL DESIGN
      STAAD SPACE                                                  -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864         4
                                                                 15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
 STEEL DESIGN
```

```
     STAAD SPACE                                             -- PAGE NO.
7
                     STAAD.PRO MEMBER SELECTION - ( AISC-360-10-ASD)   v1.4a
                     **********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION          0.864        4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
   ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 STEEL DESIGN
     STAAD SPACE                                             -- PAGE NO.
8
                     STAAD.PRO MEMBER SELECTION - ( AISC-360-10-ASD)   v1.4a
                     **********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION          0.864        4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 I Minor Axis Bending F.5

Verify the minor axis bending capacity of an I-shaped flexural member under minor axis bending per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.5, pp. F28-F29

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Details

From reference (2):

> Select an ASTM A992 W-shape beam loaded in its minor axis with a simple span of 15 ft. The loads are a total uniform dead load of 0.667 kip/ft and a uniform live load of 2 kip/ft. Limit the live load deflection to L/240. The beam is braced at the ends only.

See example F.5 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	122	122	none	
M_n/Ω_b (ft·k) [ASD]	81.4	81.1	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 I Minor Axis Bending F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 15 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X58
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MY
```

```
1 FIXED BUT MY
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GZ -0.667
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GZ -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W12X58                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.923         3
                        0.00          112.51           0.00        7.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     71.747  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     7.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.923(PASS)                |
|          Loc  :    7.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   1.125E+02     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.280E+01     Ayy:   4.392E+00     Cw:  3.575E+03              |
| Szz:   7.787E+01     Syy:   2.140E+01                                 |
```

```
| Izz:   4.750E+02     Iyy:   1.070E+02     Ix:  2.100E+00                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        15.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     7.81     N/A     13.49    T.B4.1(a)-1    |
|                Non-Slender    27.00     N/A     35.88    T.B4.1(a)-5    |
| Flexure     :  Compact         7.81     9.15    24.08    T.B4.1(b)-10   |
|                Compact        27.00    90.55   137.27    T.B4.1(b)-15   |
     STAAD SPACE                                                -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   7.65E+02   0.000      Eq. D2-1      3     0.00   |
| Rupture     0.00E+00   8.29E+02   0.000      Eq. D2-2      3     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    0.00E+00   7.03E+02   0.000      Eq. E3-1      3     0.00   |
| Min Buck    0.00E+00   5.25E+02   0.000      Eq. E3-1      3     0.00   |
| Flexural                                                                |
| Tor Buck    0.00E+00   6.13E+02   0.000      Eq. E4-1      3     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    1.18E-01    34.05   6.61E+03   3.55E+04   7.81E+02          |
| Min Buck    1.18E-01    71.75   4.94E+03   8.01E+03   5.83E+02          |
| Flexural       Ag       Fcr        Pn                                   |
| Tor Buck    1.18E-01   5.77E+03   6.81E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z    -3.00E+01   3.46E+02   0.087      Eq. G2-1      3     0.00   |
| Local-Y     0.00E+00   1.32E+02   0.000      Eq. G2-1      3     0.00   |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                  |
| Local-Z     8.89E-02     1.00     1.20     7.81   3.84E+02               |
| Local-Y     3.05E-02     1.00     0.00    27.00   1.32E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   3.24E+02   0.000      Eq. F2-1      3     0.00   |
| Minor       1.13E+02   1.22E+02   0.923      Eq. F6-1      3     7.50   |
| Intermediate   Mn        My                                             |
| Major       3.60E+02   0.00E+00                                         |
```

```
| Minor       1.35E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.89E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       3.21E+02   0.00E+00    1.00     8.86    29.98    15.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.923      Eq. H1-1b     3     7.50            |
| Flexure Tens           0.923      Eq. H1-1b     3     7.50            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          2.89E+02   0.00E+00   5.25E+02                  |
|                       1.22E+02   1.13E+02   0.00E+00                  |
| Flexure Tens          2.89E+02   0.00E+00   7.65E+02                  |
|                       1.22E+02   1.13E+02   0.00E+00                  |
|-----------------------------------------------------------------------|
   47. LOAD LIST 4
   48. PARAMETER 2
   49. CODE AISC UNIFIED 2010
   50. FU 9360 ALL
   51. FYLD 7200 ALL
   52. METHOD ASD
   53. TRACK 2 ALL
   54. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W12X58                  (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.925        4
                        0.00           75.01           0.00       7.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     71.747  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     7.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.925(PASS)                 |
|          Loc  :    7.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   7.501E+01     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
```

```
|| SECTION PROPERTIES  (UNIT: INCH)                                      |
|| Azz:   1.280E+01     Ayy:   4.392E+00     Cw:  3.575E+03              |
|| Szz:   7.787E+01     Syy:   2.140E+01                                 |
|| Izz:   4.750E+02     Iyy:   1.070E+02     Ix:  2.100E+00              |
||-----------------------------------------------------------------------|
|| MATERIAL PROPERTIES                                                    |
|| Fyld:        7199.999   Fu:        9359.999                            |
||-----------------------------------------------------------------------|
|| Actual Member Length:        15.000                                    |
|| Design Parameters                                                      |
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
||-----------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
||                STIFFENED                                               |
|| Compression :  Non-Slender     7.81     N/A     13.49    T.B4.1(a)-1  |
||                Non-Slender    27.00     N/A     35.88    T.B4.1(a)-5  |
|| Flexure     :  Compact         7.81     9.15    24.08    T.B4.1(b)-10 |
||                Compact        27.00    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   5.09E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   5.52E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   4.68E+02   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   3.49E+02   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   4.08E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.18E-01    34.05   6.61E+03   3.55E+04   7.81E+02        |
| Min Buck    1.18E-01    71.75   4.94E+03   8.01E+03   5.83E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.18E-01   5.77E+03   6.81E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    -2.00E+01   2.30E+02   0.087      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   8.78E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     8.89E-02     1.00      1.20     7.81   3.84E+02           |
| Local-Y     3.05E-02     1.00      0.00    27.00   1.32E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.16E+02   0.000      Eq. F2-1      4     0.00 |
```

```
| Minor       7.50E+01   8.11E+01   0.925      Eq. F6-1      4     7.50 |
| Intermediate   Mn        My                                           |
| Major       3.60E+02   0.00E+00                                       |
| Minor       1.35E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.92E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major       3.21E+02   0.00E+00    1.00     8.86    29.98    15.00   |
     STAAD SPACE                                             -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.925      Eq. H1-1b     4     7.50              |
| Flexure Tens         0.925      Eq. H1-1b     4     7.50              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        1.92E+02   0.00E+00   3.49E+02                   |
|                     8.11E+01   7.50E+01   0.00E+00                   |
| Flexure Tens        1.92E+02   0.00E+00   5.09E+02                   |
|                     8.11E+01   7.50E+01   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 HSST Compact Flange F.6

Verify the available flexural strength of a square HSS flexural member with compact flanges per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.6, pp. F30-F31

Details

From reference (2):

> Select a square ASTM A500 Grade B HSS beam to span 7.5 ft. The loads are a uniform dead load of 0.145 kip/ft and a uniform live load of 0.435 kip/ft. Limit the live load deflection to L/240. The beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.145 \text{ k/ft}) + 1.6(0.435 \text{ k/ft}) = 0.87 \text{ k/ft}$$

$$M_u = \frac{0.87(7.5)^2}{8} = 6.12 \text{ ft}\cdot\text{k}$$

Service moment:

$$w_a = 0.145 \text{ k/ft} + 0.435 \text{ k/ft} = 0.580 \text{ k/ft}$$

$$M_a = \frac{0.58(7.5)^2}{8} = 4.08 \text{ ft}\cdot\text{k}$$

Try a HSS 3 1/2×3 1/2×1/8, which has a plastic section modulus, $Z_x = 1.93\ in^3$.

The yield strength, $F_y = 46\ ksi$

Flange compactness: $\lambda = b/t = 27.2$

Determine the compact flange ratio limit from Table B4.1b: $\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29,000}{46}} = 28.1 > \lambda$ (Compact)

$$M_n = M_p = F_y \times Z_x = 46 \times 1.93 = 88.8 \text{ in-k} = 7.40 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(7.40) = 6.66\ ft\cdot k$

ASD: $M_n / \Omega_b = {}^{7.40}\!/_{1.67} = 4.43\ ft\cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	6.66*	6.66	none	
M_n/Ω_b (ft·k) [ASD]	4.43*	4.43	none	

* The reference gives $\phi_b M_n = 6.67\ ft\cdot k$ *and* $M_n/\Omega_b = 4.44$, *respectively. The differences are due to either rounding differences.*

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST Compact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST3.5X3.5X0.125
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.145
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.435
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST3.5X3.5X0.125        (AISC SECTIONS)
```

```
                          PASS   Eq. H1-1b           0.919        3
                       0.00            0.00          -6.12        3.75
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     65.585  L/C  :     3                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     3.75             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.919(PASS)                 |
 |          Loc  :    3.75   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -6.117E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   7.313E-01     Ayy:   7.313E-01     Cw:  0.000E+00              |
 | Szz:   1.657E+00     Syy:   1.657E+00                                 |
 | Izz:   2.900E+00     Iyy:   2.900E+00     Ix:  4.580E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        6623.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:         7.500                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Non-Slender    27.17     N/A     35.15    T.B4.1(a)-6  |
 | Flexure     :  Compact        27.17    28.12    35.15    T.B4.1(b)-17 |
 |                Compact        27.17    60.76   143.12    T.B4.1(b)-19 |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       0.00E+00   6.38E+01   0.000      Eq. D2-1      3     0.00 |
 | Rupture     0.00E+00   6.70E+01   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    0.00E+00   4.77E+01   0.000      Eq. E3-1      3     0.00 |
 | Min Buck    0.00E+00   4.77E+01   0.000      Eq. E3-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.07E-02    65.58   4.96E+03   9.58E+03   5.30E+01        |
 | Min Buck    1.07E-02    65.58   4.96E+03   9.58E+03   5.30E+01        |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Local-Z     0.00E+00   1.82E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y    -3.26E+00   1.82E+01   0.180      Eq. G2-1      3     7.50 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     5.08E-03     1.00     5.00    27.17   2.02E+01            |
| Local-Y     5.08E-03     1.00     5.00    27.17   2.02E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   5.49E+00   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   6.10E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       6.12E+00   6.66E+00   0.919      Eq. F7-1      3     3.75 |
| Minor       0.00E+00   6.66E+00   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.40E+00   0.00E+00                                       |
| Minor       7.40E+00   0.00E+00                                       |
      STAAD SPACE                                           -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.919      Eq. H1-1b     3     3.75            |
| Flexure Tens           0.919      Eq. H1-1b     3     3.75            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          6.66E+00   6.12E+00   4.77E+01                  |
|                       6.66E+00   0.00E+00   0.00E+00                  |
| Flexure Tens          6.66E+00   6.12E+00   6.38E+01                  |
|                       6.66E+00   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2010
    51. FU 8352 ALL
    52. FYLD 6624 ALL
    53. FLX 2 ALL
    54. METHOD ASD
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                           -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
```

```
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST3.5X3.5X0.125        (AISC SECTIONS)
                          PASS   Eq. H1-1b            0.921         4
                       0.00            0.00          -4.08        3.75
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     65.585  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     3.75             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.921(PASS)                 |
|          Loc  :    3.75   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.078E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.313E-01     Ayy:   7.313E-01     Cw:  0.000E+00             |
| Szz:   1.657E+00     Syy:   1.657E+00                                 |
| Izz:   2.900E+00     Iyy:   2.900E+00     Ix:  4.580E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         7.500                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A     0.00      N/A        |
|                Non-Slender    27.17     N/A    35.15    T.B4.1(a)-6  |
| Flexure     :  Compact        27.17    28.12    35.15    T.B4.1(b)-17 |
|                Compact        27.17    60.76   143.12    T.B4.1(b)-19 |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.24E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.47E+01   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   3.18E+01   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   3.18E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe          Pn          |
```

```
| Maj Buck    1.07E-02    65.58   4.96E+03   9.58E+03   5.30E+01         |
| Min Buck    1.07E-02    65.58   4.96E+03   9.58E+03   5.30E+01         |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.21E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y    -2.17E+00   1.21E+01   0.180      Eq. G2-1      4     7.50 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     5.08E-03     1.00     5.00    27.17   2.02E+01            |
| Local-Y     5.08E-03     1.00     5.00    27.17   2.02E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   3.65E+00   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   6.10E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       4.08E+00   4.43E+00   0.921      Eq. F7-1      4     3.75 |
| Minor       0.00E+00   4.43E+00   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.40E+00   0.00E+00                                       |
| Minor       7.40E+00   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.921      Eq. H1-1b     4     3.75            |
| Flexure Tens           0.921      Eq. H1-1b     4     3.75            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.43E+00   4.08E+00   3.18E+01                  |
|                       4.43E+00   0.00E+00   0.00E+00                  |
| Flexure Tens          4.43E+00   4.08E+00   4.24E+01                  |
|                       4.43E+00   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 HSST NonCompact Flange F.7

Verify the available flexural strength of a square HSS flexural member with non-compact flanges per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.

2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.7A and F.7B, pp. F32-F35

Details

From reference (2):

Select a rectangular ASTM A500 Grade B HSS beam with a span of 21 ft. The loads include a uniform dead load of 0.15 kip/ft and a uniform live load of 0.4 kip/ft. Limit the live load deflection to L/240. The beam is braced at the end points only.

Validation

Ultimate moment:

$$w_u = 1.2(0.15\text{ k/ft}) + 1.6(0.4\text{ k/ft}) = 0.82\text{ k/ft}$$

$$M_u = \frac{0.82(21)^2}{8} = 45.2\text{ ft-k}$$

Service moment:

$$w_a = 0.15\text{ k/ft} + 0.4\text{ k/ft} = 0.55\text{ k/ft}$$

$$M_a = \frac{0.55(21)^2}{8} = 30.3\text{ ft-k}$$

Try a HSS 10×6×3/16, which has a plastic section modulus, $Z_x = 18.0\ in^3$ and a section modulus, $S_x = 14.9\ in^3$.

Flange compactness: $\lambda = b/t = 31.5$

Determine the compact and slender flange ratio limits from Table B4.1b:

$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{46}} = 28.1$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29{,}000}{46}} = 35.2$$

Thus, the section has a non-compact flange.

Web compactness: $\lambda = h/t = 54.5$

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

Thus, the section has a compact web.

$$M_p = F_y \times Z_x = 46 \times 18.0 = 828\text{ in-k} = 69.0\text{ ft}\cdot\text{k}$$

$$M_n = M_p - (M_p - F_y S)\left(3.57\frac{b}{t}\sqrt{\frac{F_y}{E}} - 4.0\right) \qquad \text{(F7-2)}$$

$$= 828 - (828 - 0.7 \times 46 \times 14.9)\left(3.57 \times 31.5\sqrt{\frac{46}{29{,}000}} - 4.0\right) = 760\text{ in-k} = 63.3\text{ ft}\cdot\text{k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(63.3) = 57.0\ ft \cdot k$

ASD: $M_n / \Omega_b = {}^{63.3}\!/_{1.67} = 37.9\ ft \cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	57.0	57.0	none	
M_n/Ω_b (ft·k) [ASD]	37.9	37.9	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST NonCompact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST10X6X0.188
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.15
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.4
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
```

```
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST10X6X0.188           (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.792         3
                        0.00            0.00         -45.20       10.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    100.002  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.792(PASS)                 |
|          Loc  :   10.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.520E+01               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.906E+00     Ayy:   3.298E+00     Cw:  0.000E+00              |
| Szz:   1.492E+01     Syy:   1.137E+01                                 |
| Izz:   7.460E+01     Iyy:   3.410E+01     Ix:  7.380E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        21.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /      λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Slender        54.47     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Non-Compact    31.48    28.12    35.15    T.B4.1(b)-17 |
```

```
|               Compact        54.47    60.76   143.12    T.B4.1(b)-19 |
     STAAD SPACE                                              -- PAGE NO.
4
                     STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.22E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   2.34E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.43E+02   0.000      Eq. E7-1      3     0.00 |
| Min Buck    0.00E+00   1.06E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    3.11E-02    67.61   4.28E+03   9.02E+03   1.59E+02        |
| Min Buck    3.11E-02   100.00   3.15E+03   4.12E+03   1.18E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   4.74E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     8.61E+00   8.19E+01   0.105      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv      h/tw      Vn              |
| Local-Z     1.32E-02     1.00      5.00    31.48   5.26E+01            |
| Local-Y     2.29E-02     1.00      5.00    54.47   9.10E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   4.12E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   4.58E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       4.52E+01   6.21E+01   0.728      Eq. F7-1      3    10.50 |
| Minor       0.00E+00   4.38E+01   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       6.90E+01   0.00E+00                                       |
| Minor       4.87E+01   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
5
                     STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       4.52E+01   5.70E+01   0.792      Eq. F7-2      3    10.50 |
```

```
| Minor       0.00E+00   3.05E+01   0.000      Eq. F7-3      3     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       6.34E+01   0.00E+00                                       |
| Minor       3.39E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          0.792      Eq. H1-1b     3    10.50             |
| Flexure Tens          0.792      Eq. H1-1b     3    10.50             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         5.70E+01   4.52E+01   1.06E+02                   |
|                      3.05E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         5.70E+01   4.52E+01   2.22E+02                   |
|                      3.05E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2010
    50. FU 8352 ALL
    51. FYLD 6624 ALL
    52. METHOD ASD
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST10X6X0.188           (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.799         4
                        0.00            0.00         -30.32       10.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    100.002  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.799(PASS)                 |
|          Loc  :   10.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -3.032E+01              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.906E+00     Ayy:   3.298E+00     Cw:  0.000E+00             |
| Szz:   1.492E+01     Syy:   1.137E+01                                |
```

```
| Izz:   7.460E+01     Iyy:   3.410E+01     Ix:  7.380E+01                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:        6623.999   Fu:        8351.999                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        21.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          |
|                STIFFENED                                                 |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A          |
|                Slender        54.47     N/A     35.15    T.B4.1(a)-6    |
| Flexure     :  Non-Compact    31.48    28.12    35.15    T.B4.1(b)-17   |
|                Compact        54.47    60.76   143.12    T.B4.1(b)-19   |
     STAAD SPACE                                               -- PAGE NO.
7
                     STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Yield       0.00E+00   1.48E+02   0.000      Eq. D2-1      4     0.00    |
| Rupture     0.00E+00   1.56E+02   0.000      Eq. D2-2      4     0.00    |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Maj Buck    0.00E+00   9.55E+01   0.000      Eq. E7-1      4     0.00    |
| Min Buck    0.00E+00   7.04E+01   0.000      Eq. E7-1      4     0.00    |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    3.11E-02    67.61   4.28E+03   9.02E+03   1.59E+02          |
| Min Buck    3.11E-02   100.00   3.15E+03   4.12E+03   1.18E+02          |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Local-Z     0.00E+00   3.15E+01   0.000      Eq. G2-1      4     0.00    |
| Local-Y     5.78E+00   5.45E+01   0.106      Eq. G2-1      4     0.00    |
| Intermediate                                                             |
| Results        Aw         Cv        Kv      h/tw      Vn                |
| Local-Z     1.32E-02     1.00      5.00    31.48   5.26E+01              |
| Local-Y     2.29E-02     1.00      5.00    54.47   9.10E+01              |
|--------------------------------------------------------------------------|
| CHECK FOR TORSION                                                        |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
|             0.00E+00   2.74E+01   0.000      Eq. H3-1      4     0.00    |
| Intermediate Fcr        Tn                                               |
|             3.97E+03   4.58E+01                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Major       3.03E+01   4.13E+01   0.734      Eq. F7-1      4    10.50    |
```

```
| Minor       0.00E+00   2.92E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       6.90E+01   0.00E+00                                       |
| Minor       4.87E+01   0.00E+00                                       |
     STAAD SPACE                                            -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.03E+01   3.79E+01   0.799      Eq. F7-2      4    10.50 |
| Minor       0.00E+00   2.03E+01   0.000      Eq. F7-3      4     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       6.34E+01   0.00E+00                                       |
| Minor       3.39E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.799      Eq. H1-1b     4    10.50            |
| Flexure Tens           0.799      Eq. H1-1b     4    10.50            |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         3.79E+01   3.03E+01   7.04E+01                   |
|                      2.03E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         3.79E+01   3.03E+01   1.48E+02                   |
|                      2.03E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 HSST Slender Flange F.8

Verify the available flexural strength of a square HSS flexural member with slender flanges per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.8A and F.8B, pp. F36-F40

Details

From reference (2):

> Verify the strength of an ASTM A500 Grade B HSS8×8×x with a span of 21 ft. The loads are a dead load of 0.125 kip/ft and a live load of 0.375 kip/ft. Limit the live load deflection to L/240. The beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.125\text{ k/ft}) + 1.6(0.375\text{ k/ft}) = 0.75\text{ k/ft}$$

$$M_u = \frac{0.75(21)^2}{8} = 41.3\text{ ft}\cdot\text{k}$$

Service moment:

$$w_a = 0.125\text{ k/ft} + 0.375\text{ k/ft} = 0.50\text{ k/ft}$$

$$M_a = \frac{0.5(21)^2}{8} = 27.6\text{ ft}\cdot\text{k}$$

Try a HSS 8×8×3/16, which has a plastic section modulus, $Z_x = 15.7\ in^3$ *and a section modulus,* $S_x = 13.6\ in^3$.

The yield strength, $F_y = 46\ ksi$

Flange compactness: $\lambda = b / t = 43.0$

Determine the slender flange ratio limit from Table B4.1b: $\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{46}} = 35.2 < \lambda$ (Slender)

Web compactness: $\lambda = h / t = 43.0$

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

Thus, the section has a compact web.

$$M_n = F_y \times S_e \qquad \text{(F7-3)}$$

where

S_e = the effective section modulus determined with the effective compression flange taken as b_e

b_e = $1.93t_f\sqrt{\frac{E}{F_y}}\left[1 - \frac{0.38}{b / t_f}\sqrt{\frac{E}{F_y}}\right] = 1.93 \times 0.174\sqrt{\frac{29{,}000}{46}}\left[1 - \frac{0.38}{0.43}\sqrt{\frac{29{,}000}{46}}\right] = 6.53\ in$

The width of the compression flange between the corners (assumed to be $1.5t$) is:

$$b = 8 - 2 \times 1.5 \times t = 8 - 3(0.174) = 7.48\text{ in} > 6.53\text{ in}$$

Thus, the ineffective width of the compression flange is: $b - b_e = 7.48 - 6.53 = 0.95\ in$

Subtract this contribution of this ineffective flange area from the gross moment of inertia to approximate the effective moment of inertia:

$$I_{\text{eff}} \approx 54.4 - 2\left[\frac{0.95(0.174)^3}{12} + 0.95(0.174)(3.91)^3\right] = 49.3\text{ in}^4$$

$$S_e = \frac{I_{\text{eff}}}{d / 2} = \frac{49.3}{8.0 / 2} = 12.3\text{ in}^3$$

$$M_n = \ = 46 \times 12.3 = 566\text{ in-k} = 47.2\text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(47.2) = 42.5\ ft\cdot k$

ASD: $M_n / \Omega_b = {}^{47.2}/_{1.67} = 28.3\ ft\cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	42.5	42.5	none	
M_n/Ω_b (ft·k) [ASD]	28.3	28.3	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST Slender Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST8X8X0.188
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.125
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.375
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
```

```
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST8X8X0.188            (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.972         3
                        0.00            0.00         -41.34       10.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     79.175  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.972(PASS)                 |
|          Loc  :   10.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.134E+01              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.602E+00     Ayy:   2.602E+00     Cw:  0.000E+00             |
| Szz:   1.360E+01     Syy:   1.360E+01                                 |
| Izz:   5.440E+01     Iyy:   5.440E+01     Ix:  8.450E+01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        21.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A       |
|                Slender        42.98     N/A     35.15    T.B4.1(a)-6 |
| Flexure     :  Slender        42.98    28.12    35.15    T.B4.1(b)-17|
|                Compact        42.98    60.76   143.12    T.B4.1(b)-19|
```

```
      STAAD SPACE                                             -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   2.22E+02   0.000      Eq. D2-1      3     0.00  |
 | Rupture    0.00E+00   2.34E+02   0.000      Eq. D2-2      3     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   1.35E+02   0.000      Eq. E7-1      3     0.00  |
 | Min Buck   0.00E+00   1.35E+02   0.000      Eq. E7-1      3     0.00  |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r       Fcr         Fe         Pn          |
 | Maj Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02         |
 | Min Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02         |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   6.46E+01   0.000      Eq. G2-1      3     0.00  |
 | Local-Y    7.88E+00   6.46E+01   0.122      Eq. G2-1      3     0.00  |
 | Intermediate                                                          |
 | Results       Aw         Cv       Kv      h/tw      Vn                |
 | Local-Z    1.81E-02     1.00     5.00    42.98   7.18E+01             |
 | Local-Y    1.81E-02     1.00     5.00    42.98   7.18E+01             |
 |-----------------------------------------------------------------------|
 | CHECK FOR TORSION                                                     |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 |            0.00E+00   4.41E+01   0.000      Eq. H3-1      3     0.00  |
 | Intermediate Fcr        Tn                                            |
 |            3.97E+03   4.90E+01                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      4.13E+01   5.42E+01   0.763      Eq. F7-1      3    10.50  |
 | Minor      0.00E+00   5.42E+01   0.000      Eq. F7-1      3     0.00  |
 | Intermediate   Mn        My                                           |
 | Major      6.02E+01   0.00E+00                                        |
 | Minor      6.02E+01   0.00E+00                                        |
      STAAD SPACE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      4.13E+01   4.25E+01   0.972      Eq. F7-3      3    10.50  |
 | Minor      0.00E+00   4.25E+01   0.000      Eq. F7-3      3     0.00  |
```

```
| Intermediate   Mn        Fcr                                            |
| Major       4.73E+01   0.00E+00                                          |
| Minor       4.73E+01   0.00E+00                                          |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                      RATIO      CRITERIA    L/C     LOC                  |
| Flexure Comp          0.972      Eq. H1-1b     3    10.50                |
| Flexure Tens          0.972      Eq. H1-1b     3    10.50                |
| Intermediate         Mcx /      Mrx /      Pc /                          |
|                      Mcy        Mry        Pr                            |
| Flexure Comp         4.25E+01   4.13E+01   1.35E+02                      |
|                      4.25E+01   0.00E+00   0.00E+00                      |
| Flexure Tens         4.25E+01   4.13E+01   2.22E+02                      |
|                      4.25E+01   0.00E+00   0.00E+00                      |
|-------------------------------------------------------------------------|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2010
    51. FU 8352 ALL
    52. FYLD 6624 ALL
    53. FLX 2 ALL
    54. METHOD ASD
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSST8X8X0.188            (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.974         4
                        0.00            0.00         -27.56       10.50
|-------------------------------------------------------------------------|
| SLENDERNESS                                                              |
| Actual Slenderness Ratio    :     79.175  L/C  :     4                   |
| Allowable Slenderness Ratio :    300.000  LOC  :    10.50                |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      4    Ratio     :     0.974(PASS)                    |
|          Loc  :   10.50   Condition :   Eq. H1-1b                        |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.756E+01                  |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   2.602E+00     Ayy:   2.602E+00     Cw:  0.000E+00                 |
| Szz:   1.360E+01     Syy:   1.360E+01                                    |
```

```
| Izz:   5.440E+01     Iyy:   5.440E+01     Ix:  8.450E+01                  |
|---------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                       |
| Fyld:        6623.999   Fu:        8351.999                               |
|---------------------------------------------------------------------------|
| Actual Member Length:        21.000                                       |
| Design Parameters                                                         |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00            |
|---------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE           |
|                STIFFENED                                                  |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A            |
|                Slender        42.98     N/A     35.15    T.B4.1(a)-6     |
| Flexure     :  Slender        42.98    28.12    35.15    T.B4.1(b)-17    |
|                Compact        42.98    60.76   143.12    T.B4.1(b)-19    |
    STAAD SPACE                                                -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                   |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
| Yield      0.00E+00   1.48E+02   0.000      Eq. D2-1      4     0.00      |
| Rupture    0.00E+00   1.56E+02   0.000      Eq. D2-2      4     0.00      |
|---------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                               |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
| Maj Buck   0.00E+00   8.96E+01   0.000      Eq. E7-1      4     0.00      |
| Min Buck   0.00E+00   8.96E+01   0.000      Eq. E7-1      4     0.00      |
| Intermediate                                                              |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02            |
| Min Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02            |
|---------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                           |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
| Local-Z    0.00E+00   4.30E+01   0.000      Eq. G2-1      4     0.00      |
| Local-Y    5.25E+00   4.30E+01   0.122      Eq. G2-1      4     0.00      |
| Intermediate                                                              |
| Results       Aw         Cv       Kv      h/tw      Vn                    |
| Local-Z    1.81E-02     1.00     5.00    42.98   7.18E+01                 |
| Local-Y    1.81E-02     1.00     5.00    42.98   7.18E+01                 |
|---------------------------------------------------------------------------|
| CHECK FOR TORSION                                                         |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
|            0.00E+00   2.93E+01   0.000      Eq. H3-1      4     0.00      |
| Intermediate Fcr        Tn                                                |
|            3.97E+03   4.90E+01                                            |
|---------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                                |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
| Major      2.76E+01   3.60E+01   0.765      Eq. F7-1      4    10.50      |
```

```
| Minor       0.00E+00   3.60E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       6.02E+01   0.00E+00                                       |
| Minor       6.02E+01   0.00E+00                                       |
    STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       2.76E+01   2.83E+01   0.974      Eq. F7-3      4    10.50 |
| Minor       0.00E+00   2.83E+01   0.000      Eq. F7-3      4     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       4.73E+01   0.00E+00                                       |
| Minor       4.73E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          0.974      Eq. H1-1b     4    10.50             |
| Flexure Tens          0.974      Eq. H1-1b     4    10.50             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.83E+01   2.76E+01   8.96E+01                   |
|                      2.83E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         2.83E+01   2.76E+01   1.48E+02                   |
|                      2.83E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 Pipe F.9

Verify the available flexural strength of a compact pipe member per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F.9A and F.9B, pp. F41-F43

Details

From reference (2):

> Select an ASTM A53 Grade B Pipe shape with an 8-in. nominal depth and a simple span of 16 ft. The loads are a total uniform dead load of 0.32 kip/ft and a uniform live load of 0.96 kip/ft. There is no deflection limit for this beam. The beam is braced only at the ends.

Validation

Ultimate moment:

$$w_u = 1.2(0.32 \text{ k/ft}) + 1.6(0.96 \text{ k/ft}) = 1.92 \text{ k/ft}$$

$$M_u = \frac{1.92(16)^2}{8} = 61.4 \text{ ft·k}$$

Service moment:

$$w_a = 0.32 \text{ k/ft} + 0.96 \text{ k/ft} = 1.28 \text{ k/ft}$$

$$M_a = \frac{1.28(16)^2}{8} = 41.0 \text{ ft·k}$$

Try a pipe 8 x-strong, which has a plastic section modulus, $Z_x = 31.0\ in^3$.

The yield strength, $F_y = 35\ ksi$

Compactness ratio: $\lambda = D/t = 18.5$

Determine the slender flange ratio limit from Table B4.1b: $\lambda_p = \frac{0.07E}{F_y} = \frac{0.07(29{,}000)}{35} = 58.0 > \lambda$ (Compact)

By observation, $\lambda < \frac{0.45E}{F_y}$, so Section F8 applies.

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

$$M_n = M_p = F_y \times Z_x = 35 \times 31.0 = 1{,}090 \text{ in-k} = 90.4 \text{ ft-k} \tag{F8-1}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(90.4) = 81.4\ ft·k$

ASD: $M_n / \Omega_b = {}^{90.4}/_{1.67} = 54.1\ ft·k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	81.4	81.4	none	
M_n/Ω_b (ft·k) [ASD]	54.1	54.1	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Pipe F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 16 0 0;
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPX80
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.32
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.96
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 8640 ALL
FYLD 5040 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FU 8640 ALL
FYLD 5040 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   PIPX80                    (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.755         3
                        0.00            0.00        -61.44       8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     66.233  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.755(PASS)                 |
|          Loc  :    8.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -6.144E+01              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.344E+00     Ayy:   6.344E+00     Cw:  0.000E+00              |
| Szz:   2.319E+01     Syy:   2.319E+01                                 |
| Izz:   1.000E+02     Iyy:   1.000E+02     Ix:  2.000E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5039.999   Fu:        8639.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        16.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    18.55     N/A     91.14    T.B4.1(a)-9  |
|                Non-Slender    18.55     N/A     91.14    T.B4.1(a)-9  |
| Flexure     :  Compact        18.55    58.00   256.86    T.B4.1(b)-20 |
|                Compact        18.55    58.00   256.86    T.B4.1(b)-20 |
      STAAD SPACE                                            -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   3.75E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   5.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   2.99E+02   0.000      Eq. E3-1      3     0.00 |
```

```
| Min Buck    0.00E+00   2.99E+02   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02        |
| Min Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.12E+02   0.000      Eq. G6-1      3     0.00 |
| Local-Y     1.54E+01   1.12E+02   0.137      Eq. G6-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv      h/tw      Vn             |
| Local-Z     8.26E-02     0.00      0.00     0.00   1.25E+02          |
| Local-Y     8.26E-02     0.00      0.00     0.00   1.25E+02          |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             0.00E+00   7.66E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.02E+03   8.51E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       6.14E+01   8.14E+01   0.755      Eq. F8-1      3     8.00 |
| Minor       0.00E+00   8.14E+01   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       9.04E+01   0.00E+00                                       |
| Minor       9.04E+01   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.755      Eq. H1-1b     3     8.00            |
| Flexure Tens           0.755      Eq. H1-1b     3     8.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          8.14E+01   6.14E+01   2.99E+02                  |
|                       8.14E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          8.14E+01   6.14E+01   3.75E+02                  |
|                       8.14E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    55. LOAD LIST 4
    56. PARAMETER 2
    57. CODE AISC UNIFIED 2010
    58. METHOD ASD
    59. FU 8640 ALL
    60. FYLD 5040 ALL
    61. TRACK 2 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
```

```
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   PIPX80                   (AISC SECTIONS)
                          PASS   Eq. H1-1b            0.757         4
                       0.00            0.00         -40.96       8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     66.233  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.757(PASS)                 |
|          Loc  :    8.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.096E+01              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.344E+00     Ayy:   6.344E+00     Cw:  0.000E+00              |
| Szz:   2.319E+01     Syy:   2.319E+01                                 |
| Izz:   1.000E+02     Iyy:   1.000E+02     Ix:  2.000E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5039.999   Fu:        8639.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        16.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    18.55     N/A     91.14    T.B4.1(a)-9  |
|                Non-Slender    18.55     N/A     91.14    T.B4.1(a)-9  |
| Flexure     :  Compact        18.55    58.00   256.86    T.B4.1(b)-20 |
|                Compact        18.55    58.00   256.86    T.B4.1(b)-20 |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.49E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   3.57E+02   0.000      Eq. D2-2      4     0.00 |
```

```
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   1.99E+02   0.000      Eq. E3-1      4     0.00  |
| Min Buck    0.00E+00   1.99E+02   0.000      Eq. E3-1      4     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02         |
| Min Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02         |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   7.48E+01   0.000      Eq. G6-1      4     0.00  |
| Local-Y     1.02E+01   7.48E+01   0.137      Eq. G6-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn              |
| Local-Z     8.26E-02     0.00      0.00     0.00   1.25E+02            |
| Local-Y     8.26E-02     0.00      0.00     0.00   1.25E+02            |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   5.10E+01   0.000      Eq. H3-1      4     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.02E+03   8.51E+01                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       4.10E+01   5.41E+01   0.757      Eq. F8-1      4     8.00  |
| Minor       0.00E+00   5.41E+01   0.000      Eq. F8-1      4     0.00  |
| Intermediate   Mn         My                                            |
| Major       9.04E+01   0.00E+00                                        |
| Minor       9.04E+01   0.00E+00                                        |
    STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp           0.757      Eq. H1-1b     4     8.00             |
| Flexure Tens           0.757      Eq. H1-1b     4     8.00             |
| Intermediate          Mcx /      Mrx /      Pc /                       |
|                       Mcy        Mry        Pr                         |
| Flexure Comp          5.41E+01   4.10E+01   1.99E+02                   |
|                       5.41E+01   0.00E+00   0.00E+00                   |
| Flexure Tens          5.41E+01   4.10E+01   2.49E+02                   |
|                       5.41E+01   0.00E+00   0.00E+00                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 WT Shape F.10

Verify the available flexural strength of a WT section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example F10, pp. F44 - F46

Details

From reference (2):

> Select an ASTM A992 WT beam with a 5-in. nominal depth and a simple span of 6 ft. The toe of the stem of the WT is in tension. The loads are a uniform dead load of 0.08 kip/ft and a uniform live load of 0.24 kip/ft. There is no deflection limit for this member. The beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.08\text{ k/ft}) + 1.6(0.24\text{ k/ft}) = 0.48\text{ k/ft}$$

$$M_u = \frac{0.48(6)^2}{8} = 2.16\text{ ft-k}$$

Service moment:

$$w_a = 0.08\text{ k/ft} + 0.24\text{ k/ft} = 0.32\text{ k/ft}$$

$$M_a = \frac{0.32(6)^2}{8} = 1.44\text{ ft-k}$$

Try a WT5×6, which has a plastic section modulus, $Z_x = 2.20\ in^3$ and a section modulus, $S_x = 1.22\ in^3$.

Flexural Yielding

$$M_n = M_p = F_y \times Z_x = 50 \times 2.20 = 110\text{ in}\cdot\text{k} < 1.6F_yS_x = 1.6 \times 50 \times 1.22 = 97.6\text{ in-k} \quad \text{(F9-2)}$$

Lateral-torsional buckling does not govern since the member is continuously braced.

Flange compactness: $\lambda = {}^{b_f}\!/_{2t_f} = 9.43$

Determine the compact and slender flange ratio limits from Table B4.1b:

$$\lambda_{pf} = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29{,}000}{46}} = 9.15$$

$$\lambda_{rf} = 1.0\sqrt{\frac{E}{F_y}} = 1.0\sqrt{\frac{29{,}000}{46}} = 24.1$$

Thus, the section has a non-compact flange.

$$M_n = \left[M_p - (M_p - 0.7F_yS_{xc})\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)\right] = \ = \left[110 - (110 - 0.7 \times 50 \times 3.20)\left(\frac{9.43 - 9.15}{24.1 - 9.15}\right)\right] = 110 \text{ in}\cdot\text{k} >$$

So use $M_n = 97.6\ in \cdot k = 8.13\ ft \cdot k$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(8.13) = 7.32\ ft \cdot k$

ASD: $M_n / \Omega_b = {}^{8.13}/_{1.67} = 4.87\ ft \cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	7.32	7.21	1.5%	
M_n/Ω_b (ft·k) [ASD]	4.87	4.80	1.5%	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 WT Shape F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W10X12
CONSTANTS
```

```
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.08
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.24
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W10X12                    (AISC SECTIONS)
                           PASS   Eq. H2-1            0.300         3
                        0.00            0.00         -2.16        3.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     91.750  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     3.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.300(PASS)                |
|          Loc  :    3.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.160E+00                   |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   8.316E-01     Ayy:   9.376E-01     Cw:  2.546E-02                |
| Szz:   1.202E+00     Syy:   5.505E-01                                   |
| Izz:   4.303E+00     Iyy:   1.090E+00     Ix:  2.735E-02                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:         6.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Slender        25.97     N/A     18.06    T.B4.1(a)-4   |
|                  N/A           N/A      N/A      N/A      N/A          |
| Flexure     :  Non-Compact     9.43     9.15    24.08    T.B4.1(b)-10  |
|                Slender        25.97    20.23    24.81    T.B4.1(b)-14  |
      STAAD SPACE                                            -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   7.96E+01   0.000      Eq. D2-1      3     0.00  |
| Rupture     0.00E+00   8.63E+01   0.000      Eq. D2-2      3     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   4.31E+01   0.000      Eq. E7-1      3     0.00  |
| Min Buck    0.00E+00   3.28E+01   0.000      Eq. E7-1      3     0.00  |
| Flexural                                                                |
| Tor Buck    0.00E+00   2.72E+01   0.000      Eq. E7-1      3     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    1.23E-02    46.18   3.89E+03   1.93E+04   4.79E+01         |
| Min Buck    1.23E-02    91.75   2.96E+03   4.90E+03   3.64E+01         |
| Flexural       Ag        Fcr        Pn                                  |
| Tor Buck    1.23E-02   2.46E+03   3.02E+01                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   2.25E+01   0.000      Eq. G2-1      3     0.00  |
| Local-Y    -1.44E+00   2.53E+01   0.057      Eq. G2-1      3     6.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     5.78E-03     1.00     1.20     9.43   2.49E+01              |
| Local-Y     6.51E-03     1.00     1.20    25.97   2.81E+01              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
```

```
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.16E+00   7.21E+00   0.300      Eq. F9-1      3     3.00 |
| Minor       0.00E+00   3.26E+00   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       8.01E+00   0.00E+00                                      |
| Minor       3.63E+00   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.16E+00   2.45E+01   0.088      Eq. F9-4      3     3.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major       2.72E+01   0.00E+00    1.00     0.00     0.00     6.00    |
     STAAD SPACE                                            -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.16E+00   7.21E+00   0.300      Eq. F9-6      3     3.00 |
| Minor       0.00E+00   3.23E+00   0.000      Eq. F6-2      3     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       8.01E+00   0.00E+00                                      |
| Minor       3.59E+00   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.300      Eq. H2-1      3     3.00              |
| Flexure Tens         0.300      Eq. H2-1      3     3.00              |
| Intermediate        frbw /     frbz /     fra /                      |
|                     Fcbw       Fcbz       Fca                        |
| Flexure Comp        1.17E+03   0.00E+00   0.00E+00                   |
|                     3.92E+03   1.01E+04   2.21E+03                   |
| Flexure Tens        3.11E+03   0.00E+00   0.00E+00                   |
|                     1.04E+04   1.01E+04   6.48E+03                   |
|-----------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2010
    59. FU 9360 ALL
    60. FYLD 7200 ALL
    61. FLX 2 ALL
    62. METHOD ASD
    63. TRACK 2 ALL
    64. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                            -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 T    W10X12                   (AISC SECTIONS)
```

```
                           PASS    Eq. H2-1            0.300        4
                        0.00            0.00          -1.44       3.00
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :     91.750  L/C  :     4                  |
| Allowable Slenderness Ratio :    300.000  LOC  :     3.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      4    Ratio     :     0.300(PASS)                   |
|          Loc  :    3.00   Condition :   Eq. H2-1                        |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.440E+00                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   8.316E-01     Ayy:   9.376E-01     Cw:  2.546E-02                |
| Szz:   1.202E+00     Syy:   5.505E-01                                   |
| Izz:   4.303E+00     Iyy:   1.090E+00     Ix:  2.735E-02                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:         6.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Slender        25.97     N/A     18.06    T.B4.1(a)-4    |
|                  N/A           N/A      N/A      N/A      N/A           |
| Flexure     :  Non-Compact     9.43     9.15    24.08    T.B4.1(b)-10   |
|                Slender        25.97    20.23    24.81    T.B4.1(b)-14   |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   5.30E+01   0.000      Eq. D2-1      4     0.00  |
| Rupture     0.00E+00   5.75E+01   0.000      Eq. D2-2      4     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   2.87E+01   0.000      Eq. E7-1      4     0.00  |
| Min Buck    0.00E+00   2.18E+01   0.000      Eq. E7-1      4     0.00  |
| Flexural                                                                |
| Tor Buck    0.00E+00   1.81E+01   0.000      Eq. E7-1      4     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr         Fe         Pn            |
| Maj Buck    1.23E-02    46.18   3.89E+03   1.93E+04   4.79E+01          |
| Min Buck    1.23E-02    91.75   2.96E+03   4.90E+03   3.64E+01          |
| Flexural       Ag        Fcr        Pn                                  |
| Tor Buck    1.23E-02   2.46E+03   3.02E+01                              |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.49E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y    -9.60E-01   1.68E+01   0.057      Eq. G2-1      4     6.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     5.78E-03     1.00      1.20     9.43   2.49E+01          |
| Local-Y     6.51E-03     1.00      1.20    25.97   2.81E+01          |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.44E+00   4.80E+00   0.300      Eq. F9-1      4     3.00 |
| Minor       0.00E+00   2.17E+00   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       8.01E+00   0.00E+00                                       |
| Minor       3.63E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.44E+00   1.63E+01   0.089      Eq. F9-4      4     3.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       2.72E+01   0.00E+00    1.00     0.00     0.00     6.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.44E+00   4.80E+00   0.300      Eq. F9-6      4     3.00 |
| Minor       0.00E+00   2.15E+00   0.000      Eq. F6-2      4     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       8.01E+00   0.00E+00                                       |
| Minor       3.59E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.300      Eq. H2-1      4     3.00           |
| Flexure Tens            0.300      Eq. H2-1      4     3.00           |
| Intermediate           frbw /     frbz /     fra /                    |
|                        Fcbw       Fcbz       Fca                      |
| Flexure Comp           7.83E+02   0.00E+00   0.00E+00                 |
|                        2.61E+03   6.74E+03   1.47E+03                 |
| Flexure Tens           2.07E+03   0.00E+00   0.00E+00                 |
|                        6.90E+03   6.74E+03   4.31E+03                 |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 Angle F.11A

Verify the available flexural strength of a single angle member per the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example Example F.11A, pp. F47-F49

Details

From reference (2):

> Select an ASTM A36 single angle with a simple span of 6 ft. The vertical leg of the single angle is up and the toe is in compression. The vertical loads are a uniform dead load of 0.05 kip/ft and a uniform live load of 0.15 kip/ft. There are no horizontal loads. There is no deflection limit for this angle. The angle is braced at the end points only. Assume bending about the geometric x-x axis and that there is no lateral-torsional restraint.

Note: The angle section is given a beta angle as the geometrical axis and principal axis are different. In STAAD.Pro the calculation is done with reference with the principal axis, but in AISC 360-10 Code geometrical axis is used calculating the capacities of the section.

Validation

Ultimate moment:

$$w_u = 1.2(0.05 \text{ k/ft}) + 1.6(0.15 \text{ k/ft}) = 0.30 \text{ k/ft}$$

$$M_u = \frac{0.30(6)^2}{8} = 1.35 \text{ ft} \cdot \text{k}$$

Service moment:

$$w_a = 0.05 \text{ k/ft} + 0.15 \text{ k/ft} = 0.20 \text{ k/ft}$$

$$M_a = \frac{0.2(6)^2}{8} = 0.90 \text{ ft} \cdot \text{k}$$

Try an L4×4×1/4, which has a section modulus, $S_x = 1.03\ in^3$.

The yield strength, $F_y = 36\ ksi$

The nominal flexural strength for the flexural yielding limit state:

$$M_n = 1.5M_y = 1.5F_yS_x = 1.5 \times 36 \times 1.03 = 55.6 \text{ in} \cdot \text{k} \qquad \text{(F10-1)}$$

The yield moment for a single angle bent about its geometric axis with no lateral-torsional restraint:

$$M_y = 0.8F_yS_x = 0.8 \times 36 \times 1.03 = 29.7 \text{ in} \cdot \text{k}$$

Determine M_e:

$$M_e = \frac{0.66Eb^4tC_b}{L_b^2}\left[\sqrt{1 + 0.78\left(\frac{L_bt}{b^2}\right)^2} - 1\right] \qquad \text{(F10-6a)}$$

where

C_b = 1.14 from Ref. 1, Table 3-1

$$M_e = \frac{0.66(29,\ 000)(4.0)^4(1/4)1.14}{(6 \times 12)^2}\left[\sqrt{1 + 0.78\left(\frac{6 \times 12(1/4)}{(4.0)^2}\right)^2} - 1\right] = 110 \text{ in}\cdot\text{k} > M_y$$

$$M_n = \left(1.92 - 1.17\sqrt{\frac{M_y}{M_e}}\right)M_y \le 1.5M_y \tag{F10-3}$$

$$M_n = \left(1.92 - 1.17\sqrt{\frac{29.7}{110}}\right)29.7 = 39.0 \text{ in}\cdot\text{k} \le 1.5(29.7) = 44.6 \text{ in}\cdot\text{k}$$

Leg local buckling

Leg slenderness ratio: $\lambda = \frac{b}{t} = \frac{4.0}{1/4} = 16.0$

Determine the compact and slender leg ratio limits from Table B4.1b:

$$\lambda_p = 0.54\sqrt{\frac{E}{F_y}} = 0.54\sqrt{\frac{29{,}000}{36}} = 15.3$$

$$\lambda_r = 0.91\sqrt{\frac{E}{F_y}} = 0.91\sqrt{\frac{29{,}000}{36}} = 25.8$$

$\lambda_p < \lambda < \lambda_r$; thus, the leg is non-compact in flexure.

$$M_n = F_y S_c\left[2.43 - 1.72\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{E}}\right] \tag{F10-3}$$

where

S_c = $0.80S_x = 0.80 \times 1.03 = 0..824 \ in^3$

$$M_n = 36 \times 0.824\left[2.43 - 1.72(16.0)\sqrt{\frac{36}{29,\ 000}}\right] = 43.3 \text{ in}\cdot\text{k}$$

The lateral-torsional buckling limit state governs.

$$M_n = 43.3 \text{ in}\cdot\text{k} = 3.25 \text{ ft}\cdot\text{k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(3.25) = 2.93$ *ft-k*

ASD: $M_n / \Omega_b = {}^{3.25}/_{1.67} = 1.95$ *ft-k*

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
LTB $\phi_b M_n$ (ft·k) [LRFD]	2.93	2.96	negligible	
LTB M_n/Ω_b (ft·k) [ASD]	1.95	1.97	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Angle F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40404
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GZ 0.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GZ 0.2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 8352 ALL
FYLD 5184 ALL
CB 1.14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
```

```
FU 8352 ALL
FYLD 5184 ALL
CB 1.14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   L40404                   (AISC SECTIONS)
                           PASS   Sec. F1              0.444         3
                        0.00           -1.86           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     91.954  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.444(PASS)                |
|          Loc  :    0.00   Condition :   Sec. F1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -1.860E+00              |
| Mx:   0.000E+00     My:  -1.860E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.000E+00     Ayy:   1.000E+00     Cw:  5.051E-02              |
| Szz:   7.837E-01     Syy:   1.719E+00                                 |
| Izz:   1.183E+00     Iyy:   4.895E+00     Ix:  4.021E-02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        16.00     N/A     12.77    T.B4.1(a)-3  |
|                  N/A          N/A       N/A     N/A      N/A          |
| Flexure     :  Non-Compact    16.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A          N/A       N/A     N/A      N/A          |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
```

```
||              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   6.25E+01   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   8.40E+01   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Min Buck    0.00E+00   3.33E+01   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Min Buck    1.34E-02   105.90   2.76E+03   3.68E+03   3.70E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     1.86E+00   1.94E+01   0.096      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.94E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     6.94E-03     1.00     1.20    16.00   2.16E+01            |
| Local-Y     6.94E-03     1.00     1.20    16.00   2.16E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.86E+00   6.96E+00   0.267      Eq. F10-1     3     0.00 |
| Minor       0.00E+00   3.17E+00   0.000      Eq. F10-1     3     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.74E+00   5.16E+00                                       |
| Minor       3.53E+00   2.35E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.86E+00   5.97E+00   0.311      Eq. F10-3     3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       6.64E+00   1.76E+01    1.14     0.00     0.00     6.00   |
    STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.86E+00   6.78E+00   0.274      Eq. F10-7     3     0.00 |
| Intermediate   Mn         Me                                          |
| Major       7.53E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|              STRESS     RATIO      CRITERIA     L/C    LOC           |
| Shear       2.80E+03    0.096      Eq. H3-8      3     0.00          |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.311      Eq. H2-1      3     0.00            |
```

```
| Flexure Tens          0.311      Eq. H2-1      3     0.00              |
| Intermediate         frbw /     frbz /     fra /                      |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         1.87E+03   0.00E+00   0.00E+00                   |
|                      6.00E+03   7.00E+03   2.48E+03                   |
| Flexure Tens         1.87E+03   0.00E+00   0.00E+00                   |
|                      6.00E+03   7.00E+03   4.67E+03                   |
  GEOMETRIC AXIS DESIGN
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      1.32E+00   4.24E+00   0.310      Eq. F10-1     3     0.00  |
| Minor      1.32E+00   4.24E+00   0.310      Eq. F10-1     3     0.00  |
| Intermediate   Mn         My                                          |
| Major      4.71E+00   3.14E+00                                        |
| Minor      4.71E+00   3.14E+00                                        |
     STAAD SPACE                                             -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      1.32E+00   2.96E+00   0.444      Eq. F10-3     3     0.00  |
| Minor      1.32E+00   3.39E+00   0.388      Eq. F10-3     3     0.00  |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major      3.29E+00   9.20E+00    1.14     0.00     0.00     6.00     |
| Minor      3.77E+00   5.41E+01    1.14     0.00     0.00     6.00     |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      1.32E+00   3.30E+00   0.398      Eq. F10-7     3     0.00  |
| Minor     -6.58E-01   3.30E+00   0.199      Eq. F10-7     3     3.00  |
| Intermediate   Mn         Me                                          |
| Major      3.67E+00   0.00E+00                                        |
| Minor      3.67E+00   0.00E+00                                        |
|-----------------------------------------------------------------------|
   49. LOAD LIST 4
   50. PARAMETER 2
   51. CODE AISC UNIFIED 2010
   52. METHOD ASD
   53. FU 8352 ALL
   54. FYLD 5184 ALL
   55. CB 1.14 ALL
   56. TRACK 2 ALL
   57. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                             -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
```

```
 ======================================================================
      1 ST   L40404                  (AISC SECTIONS)
                          PASS   Sec. F1             0.485        4
                       0.00           -1.35          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     91.954  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.485(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. F1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -1.350E+00              |
| Mx:   0.000E+00     My:  -1.350E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.000E+00     Ayy:   1.000E+00     Cw:  5.051E-02             |
| Szz:   7.837E-01     Syy:   1.719E+00                                |
| Izz:   1.183E+00     Iyy:   4.895E+00     Ix:  4.021E-02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        16.00     N/A     12.77    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    16.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   4.16E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   5.60E+01   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Min Buck    0.00E+00   2.21E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Min Buck    1.34E-02   105.90   2.76E+03   3.68E+03   3.70E+01       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
```

```
| Local-Z     1.35E+00   1.29E+01   0.104      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.29E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     6.94E-03     1.00     1.20    16.00   2.16E+01            |
| Local-Y     6.94E-03     1.00     1.20    16.00   2.16E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.35E+00   4.63E+00   0.291      Eq. F10-1     4     0.00 |
| Minor       0.00E+00   2.11E+00   0.000      Eq. F10-1     4     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.74E+00   5.16E+00                                       |
| Minor       3.53E+00   2.35E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.35E+00   3.97E+00   0.340      Eq. F10-3     4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       6.64E+00   1.76E+01    1.14     0.00     0.00     6.00    |
     STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.35E+00   4.51E+00   0.299      Eq. F10-7     4     0.00 |
| Intermediate   Mn         Me                                          |
| Major       7.53E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|              STRESS     RATIO      CRITERIA     L/C    LOC            |
| Shear       1.86E+03    0.104      Eq. H3-8      4     0.00           |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.340      Eq. H2-1      4     0.00             |
| Flexure Tens          0.340      Eq. H2-1      4     0.00             |
| Intermediate         frbw /     frbz /     fra /                      |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         1.36E+03   0.00E+00   0.00E+00                   |
|                      3.99E+03   4.66E+03   1.65E+03                   |
| Flexure Tens         1.36E+03   0.00E+00   0.00E+00                   |
|                      3.99E+03   4.66E+03   3.10E+03                   |
  GEOMETRIC AXIS DESIGN
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       9.55E-01   2.82E+00   0.338      Eq. F10-1     4     0.00 |
| Minor       9.55E-01   2.82E+00   0.338      Eq. F10-1     4     0.00 |
| Intermediate   Mn         My                                          |
```

```
| Major       4.71E+00   3.14E+00                                              |
| Minor       4.71E+00   3.14E+00                                              |
      STAAD SPACE                                                  -- PAGE NO.
13
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       9.55E-01   1.97E+00   0.485      Eq. F10-3     4     0.00 |
| Minor       9.55E-01   2.26E+00   0.423      Eq. F10-3     4     0.00 |
| Intermediate   Mn         Me        Cb       Lp       Lr       Lb    |
| Major       3.29E+00   9.20E+00    1.14     0.00     0.00     6.00   |
| Minor       3.77E+00   5.41E+01    1.14     0.00     0.00     6.00   |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       9.55E-01   2.20E+00   0.434      Eq. F10-7     4     0.00 |
| Minor      -4.77E-01   2.20E+00   0.217      Eq. F10-7     4     3.00 |
| Intermediate   Mn         Me                                          |
| Major       3.67E+00   0.00E+00                                       |
| Minor       3.67E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 W E.1C

Verify the compressive strength of an unbraced wide-flange section column about the Y-Y axis per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.1C, pp. E4 - E8

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From the reference example E.1C:

Calculate the available strength of a W14×132 column with unbraced lengths of 30 ft in both axes.

From the reference example E.1A:

> … an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips. The column is 30 ft long and is pinned top and bottom in both axes.

See example E.1C in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	892	893	negligible	
P_n / Ω_c (kips) [ASD]	598	594	negligible	
Slenderness ratio	95.7	95.792	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 LRFD
```

```
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
KX 1 ALL
KY 1 ALL
METHOD LRFD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 7200 ALL
FU 9360 ALL
KX 1 ALL
KY 1 ALL
METHOD ASD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X132                  (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.941         3
                      840.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     95.792  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.941(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04             |
| Szz:   2.082E+02     Syy:   7.456E+01                                 |
```

```
| Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01                  |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        30.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1(a)-1    |
|                Non-Slender    17.74     N/A     35.88    T.B4.1(a)-5    |
| Flexure     :  Compact         7.14     9.15    24.08    T.B4.1(b)-10   |
|                Compact        17.74    90.55   137.27    T.B4.1(b)-15   |
     STAAD SPACE                                                 -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   1.75E+03   0.000      Eq. D2-1      3     0.00    |
| Rupture    0.00E+00   1.89E+03   0.000      Eq. D2-2      3     0.00    |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   8.40E+02   1.37E+03   0.612      Eq. E3-1      3     0.00    |
| Min Buck   8.40E+02   8.93E+02   0.941      Eq. E3-1      3     0.00    |
| Flexural                                                                |
| Tor Buck   8.40E+02   1.39E+03   0.602      Eq. E4-1      3     0.00    |
| Intermediate                                                            |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   2.69E-01    57.33   5.66E+03   1.25E+04   1.53E+03           |
| Min Buck   2.69E-01    95.79   3.68E+03   4.49E+03   9.92E+02           |
| Flexural      Ag        Fcr        Pn                                   |
| Tor Buck   2.69E-01   5.75E+03   1.55E+03                               |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z    0.00E+00   8.18E+02   0.000      Eq. G2-1      3     0.00    |
| Local-Y    0.00E+00   2.84E+02   0.000      Eq. G2-1      3     0.00    |
| Intermediate                                                            |
| Results       Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z    2.10E-01     1.00     1.20     7.14   9.08E+02               |
| Local-Y    6.58E-02     1.00     0.00    17.74   2.84E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      0.00E+00   8.78E+02   0.000      Eq. F2-1      3     0.00    |
| Minor      0.00E+00   4.24E+02   0.000      Eq. F6-1      3     0.00    |
| Intermediate   Mn        My                                             |
| Major      9.75E+02   0.00E+00                                          |
```

```
| Minor       4.71E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   7.48E+02   0.000      Eq. F2-2      3     0.00  |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb       |
| Major       8.31E+02   0.00E+00    1.00    13.27    56.15    30.00     |
     STAAD SPACE                                             -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.941      Eq. H1-1a     3     0.00              |
| Flexure Tens           0.000      Eq. H1-1b     3     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          7.48E+02   0.00E+00   8.93E+02                    |
|                       4.24E+02   0.00E+00   8.40E+02                    |
| Flexure Tens          7.48E+02   0.00E+00   1.75E+03                    |
|                       4.24E+02   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2010
    55. FYLD 7200 ALL
    56. FU 9360 ALL
    57. KX 1 ALL
    58. KY 1 ALL
    59. METHOD ASD
    60. PROFILE W14 ALL
    61. TRACK 2 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                             -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY            MZ       LOCATION
 =======================================================================
       1 ST   W14X132                 (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.943         4
                      560.00 C          0.00           0.00       0.00
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :     95.792  L/C  :     4                  |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      4    Ratio     :     0.943(PASS)                   |
|          Loc  :    0.00   Condition :   Eq. H1-1a                       |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
```

```
| Fx:   5.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04                |
| Szz:   2.082E+02     Syy:   7.456E+01                                   |
| Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        30.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1(a)-1   |
|                Non-Slender    17.74     N/A     35.88    T.B4.1(a)-5   |
| Flexure     :  Compact         7.14     9.15    24.08    T.B4.1(b)-10  |
|                Compact        17.74    90.55   137.27    T.B4.1(b)-15  |
    STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield       0.00E+00   1.16E+03   0.000      Eq. D2-1      4     0.00   |
| Rupture     0.00E+00   1.26E+03   0.000      Eq. D2-2      4     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck    5.60E+02   9.14E+02   0.613      Eq. E3-1      4     0.00   |
| Min Buck    5.60E+02   5.94E+02   0.943      Eq. E3-1      4     0.00   |
| Flexural                                                                |
| Tor Buck    5.60E+02   9.28E+02   0.603      Eq. E4-1      4     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    2.69E-01    57.33   5.66E+03   1.25E+04   1.53E+03          |
| Min Buck    2.69E-01    95.79   3.68E+03   4.49E+03   9.92E+02          |
| Flexural       Ag       Fcr        Pn                                   |
| Tor Buck    2.69E-01   5.75E+03   1.55E+03                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z     0.00E+00   5.44E+02   0.000      Eq. G2-1      4     0.00   |
| Local-Y     0.00E+00   1.90E+02   0.000      Eq. G2-1      4     0.00   |
| Intermediate                                                            |
| Results        Aw        Cv        Kv      h/tw      Vn                 |
| Local-Z     2.10E-01     1.00     1.20     7.14   9.08E+02              |
| Local-Y     6.58E-02     1.00     0.00    17.74   2.84E+02              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
```

```
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   5.84E+02   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   2.82E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       9.75E+02   0.00E+00                                      |
| Minor       4.71E+02   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   4.98E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major       8.31E+02   0.00E+00    1.00     13.27    56.15    30.00  |
     STAAD SPACE                                             -- PAGE NO.
11
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.943      Eq. H1-1a     4     0.00             |
| Flexure Tens         0.000      Eq. H1-1b     4     0.00             |
| Intermediate        Mcx /      Mrx /       Pc /                      |
|                     Mcy        Mry         Pr                        |
| Flexure Comp        4.98E+02   0.00E+00   5.94E+02                   |
|                     2.82E+02   0.00E+00   5.60E+02                   |
| Flexure Tens        4.98E+02   0.00E+00   1.16E+03                   |
|                     2.82E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 W E.1D

Verify the compressive strength of a column torsionally braced at the midpoint in the weak axis per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.1D, p E-6, pp.E-9 - E-10

> **Note:** This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From the reference example E.1C

Calculate the available strength of a W14×90 with a strong axis unbraced length of 30.0 ft and weak axis and torsional unbraced lengths of 15.0 ft.

From the reference example E.1A:

... an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips. The column is 30 ft long and is pinned top and bottom in both axes.

See example E.1D in reference (2) for calculations.

Results

Result Type	**Reference**	**STAAD.Pro**	**Difference**	**Comments**
$\phi_c P_n$ (kips) [LRFD]	927	927	none	
P_n/Ω_c (kips) [ASD]	617	617	none	
Slenderness ratio	58.6	58.63	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
```

```
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
KX 0.5 ALL
KY 0.5 ALL
METHOD LRFD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 7200 ALL
FU 9360 ALL
KX 0.5 ALL
KY 0.5 ALL
METHOD ASD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X90                    (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.906         3
                      840.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     58.633  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.906(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
```

```
 | Fx:   8.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                 |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                 |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   2.059E+01     Ayy:   6.160E+00     Cw:  1.598E+04              |
 | Szz:   1.427E+02     Syy:   4.993E+01                                 |
 | Izz:   9.990E+02     Iyy:   3.620E+02     Ix:  4.060E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        30.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    0.50 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender    10.21     N/A     13.49    T.B4.1(a)-1 |
 |                Non-Slender    25.86     N/A     35.88    T.B4.1(a)-5 |
 | Flexure     :  Non-Compact    10.21     9.15    24.08    T.B4.1(b)-10 |
 |                Compact        25.86    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   1.19E+03   0.000      Eq. D2-1      3     0.00 |
 | Rupture    0.00E+00   1.29E+03   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   8.40E+02   9.27E+02   0.906      Eq. E3-1      3     0.00 |
 | Min Buck   8.40E+02   1.00E+03   0.838      Eq. E3-1      3     0.00 |
 | Flexural                                                              |
 | Tor Buck   8.40E+02   1.02E+03   0.821      Eq. E4-1      3     0.00 |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn            |
 | Maj Buck   1.84E-01    58.63   5.60E+03   1.20E+04   1.03E+03         |
 | Min Buck   1.84E-01    48.70   6.05E+03   1.74E+04   1.11E+03         |
 | Flexural      Ag        Fcr        Pn                                 |
 | Tor Buck   1.84E-01   6.18E+03   1.14E+03                             |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   5.56E+02   0.000      Eq. G2-1      3     0.00 |
 | Local-Y    0.00E+00   1.85E+02   0.000      Eq. G2-1      3     0.00 |
 | Intermediate                                                          |
 | Results       Aw        Cv        Kv      h/tw      Vn                |
 | Local-Z    1.43E-01     1.00      1.20    10.21   6.18E+02            |
 | Local-Y    4.28E-02     1.00      0.00    25.86   1.85E+02            |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
```

```
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   5.89E+02   0.000      Eq. F2-1      3     0.00  |
| Minor       0.00E+00   2.84E+02   0.000      Eq. F6-1      3     0.00  |
| Intermediate   Mn         My                                           |
| Major       6.54E+02   0.00E+00                                        |
| Minor       3.15E+02   0.00E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   4.66E+02   0.000      Eq. F2-2      3     0.00  |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb      |
| Major       5.18E+02   0.00E+00    1.00    13.05    42.60    30.00     |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   5.74E+02   0.000      Eq. F3-1      3     0.00  |
| Minor       0.00E+00   2.73E+02   0.000      Eq. F6-2      3     0.00  |
| Intermediate   Mn         Fcr                                          |
| Major       6.37E+02   0.00E+00                                        |
| Minor       3.03E+02   0.00E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.906      Eq. H1-1a     3     0.00              |
| Flexure Tens           0.000      Eq. H1-1b     3     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                       |
|                       Mcy        Mry        Pr                         |
| Flexure Comp          4.66E+02   0.00E+00   9.27E+02                   |
|                       2.73E+02   0.00E+00   8.40E+02                   |
| Flexure Tens          4.66E+02   0.00E+00   1.19E+03                   |
|                       2.73E+02   0.00E+00   0.00E+00                   |
|-------------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2010
    55. FYLD 7200 ALL
    56. FU 9360 ALL
    57. KX 0.5 ALL
    58. KY 0.5 ALL
    59. METHOD ASD
    60. PROFILE W14 ALL
    61. TRACK 2 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
```

```
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W14X90                  (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.908        4
                      560.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     58.633  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.908(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.059E+01     Ayy:   6.160E+00     Cw:  1.598E+04              |
| Szz:   1.427E+02     Syy:   4.993E+01                                 |
| Izz:   9.990E+02     Iyy:   3.620E+02     Ix:  4.060E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    0.50 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.21     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    25.86     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Non-Compact    10.21     9.15    24.08    T.B4.1(b)-10 |
|                Compact        25.86    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   7.93E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   8.61E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    5.60E+02   6.17E+02   0.908      Eq. E3-1      4     0.00 |
| Min Buck    5.60E+02   6.67E+02   0.839      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    5.60E+02   6.81E+02   0.822      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.84E-01    58.63   5.60E+03   1.20E+04   1.03E+03        |
```

```
| Min Buck    1.84E-01    48.70   6.05E+03   1.74E+04   1.11E+03         |
| Flexural       Ag        Fcr        Pn                                  |
| Tor Buck    1.84E-01   6.18E+03   1.14E+03                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   3.70E+02   0.000      Eq. G2-1      4     0.00  |
| Local-Y     0.00E+00   1.23E+02   0.000      Eq. G2-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv        Kv      h/tw      Vn                |
| Local-Z     1.43E-01     1.00      1.20    10.21   6.18E+02            |
| Local-Y     4.28E-02     1.00      0.00    25.86   1.85E+02            |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   3.92E+02   0.000      Eq. F2-1      4     0.00  |
| Minor       0.00E+00   1.89E+02   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn         My                                            |
| Major       6.54E+02   0.00E+00                                         |
| Minor       3.15E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   3.10E+02   0.000      Eq. F2-2      4     0.00  |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb     |
| Major       5.18E+02   0.00E+00    1.00    13.05    42.60    30.00     |
    STAAD SPACE                                            -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   3.82E+02   0.000      Eq. F3-1      4     0.00  |
| Minor       0.00E+00   1.81E+02   0.000      Eq. F6-2      4     0.00  |
| Intermediate   Mn         Fcr                                           |
| Major       6.37E+02   0.00E+00                                         |
| Minor       3.03E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                       RATIO      CRITERIA    L/C     LOC                |
| Flexure Comp          0.908      Eq. H1-1a     4     0.00               |
| Flexure Tens          0.000      Eq. H1-1b     4     0.00               |
| Intermediate         Mcx /      Mrx /      Pc /                        |
|                      Mcy        Mry        Pr                          |
| Flexure Comp         3.10E+02   0.00E+00   6.17E+02                    |
|                      1.81E+02   0.00E+00   5.60E+02                    |
| Flexure Tens         3.10E+02   0.00E+00   7.93E+02                    |
|                      1.81E+02   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 - E-2

Reference

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. . Example E.2, pp. E-11 - E-15

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Problem

From the reference:

Verify that a built-up, ASTM A572 Grade 50 column with PL 1 in. × 8 in. flanges and a PL 4 in. × 15 in. web is sufficient to carry a dead load of 70 kips and live load of 210 kips in axial compression. The column length is 15 ft and the ends are pinned in both axes.

Validation

Ultimate load:

$$P_u = 1.2(70\text{ kips}) + 1.6(210\text{ kips}) = 420\text{ kips}$$

Service load:

$$P_a = 70\text{ kips} + 210\text{ kips} = 280\text{ kips}$$

Section Properties

Built-up section properties (ignoring fillet welds):

$$A = 2(8.0)(1.0) + 15.0(0.25) = 19.8\text{ in}^2$$

$$I_x = \Sigma Ad^2 + \Sigma\frac{bh^3}{12} = 2\left[8(8.0)^2 + \frac{8.0(1.0)^3}{12}\right] + \frac{0.25(15.0)^3}{12} = 1,\ 100\text{ in}^4$$

$$I_y = 2\left(\frac{1.0 \times 8.0^3}{12}\right) + \frac{15.0 \times 0.25^3}{12} = 85.4\text{ in}^4$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{85.4}{19.8}} = 2.08\text{ in}$$

Elastic Flexural Buckling Stress

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{KL}{r_y} = \frac{1.0(15.0)(12)}{2.08} = 86.5$$

$$F_e = \frac{\pi^2 E}{(KL\ /r)^2} = \frac{\pi^2 29,\ 000}{(86.5)^2} = 38.3\text{ ksi} \qquad \text{(Eq. E3-4)}$$

<u>Slenderness</u>

Check for slender flanges and then determine the unstiffened element (flange) reduction factor.

$$k_c = \frac{4}{\sqrt{h / t_w}} = \frac{4}{\sqrt{15.0 / 0.25}} = 0.516$$ (Table B4.1b note a)

$0.35 < k_c < 0.76$

Flange slenderness:

$$\lambda = \frac{b}{t} = \frac{4.0}{1.0} = 4.0$$

The limiting flange slenderness ratio, λ_r:

$$\lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.516(29,\ 000)}{50}} = 11.1 > \lambda$$ (Table B4.1a, case 2)

Thus, the flanges are not slender and there is no reduction ($Q_s = 1.0$).

Check for slender web and then determine the stiffened element (web) reduction factor.

Web slenderness:

$$\lambda = \frac{h}{t} = \frac{15.0}{0.25} = 60.0$$

The limiting web slenderness ratio, λ_r:

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,\ 000}{50}} = 35.9 < \lambda$$ (Table B4.1a, case 5)

Thus, the web is slender.

$$Q_a = \frac{A_e}{A_g}$$ (Eq. E7-16)

where

A_e	=	the effective area based on the reduced effective width, b_e
b_e	=	$1.92t\sqrt{\frac{E}{f}}\left[1 - \frac{0.34}{(b / t)}\sqrt{\frac{E}{f}}\right] \leq b$ (Eq. E7-17)
f	=	F_{cr} calculated based on $Q = 1.0$
F_{cr}	=	$Q\left(0.658^{QF_y/F_e}\right)F_y = 1.0\left[0.658^{1.0(50)/(38.3)}\right]50 = 29.0\ ksi$ because $\frac{KL}{r} = 86.5 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{1.0(50)}} = 113$

$$b_e = 1.92(0.25)\sqrt{\frac{29,\ 000}{29.0}}\left[1 - \frac{0.34}{(15 / 0.25)}\sqrt{\frac{29,\ 000}{29.0}}\right] = 12.5 \text{ in} < 15 \text{ in}$$

$$A_e = b_e t_w + 2b_f t_f = 12.5(0.25) + 2(8.0)(1.0) = 19.1 \text{ in}^2$$

$$Q_a = \frac{19.1}{19.8} = 0.965$$

$$Q = Q_s Q_a = 1.0 \times 0.965 = 0.965$$

$$\frac{KL}{r} = 86.5 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{0.965(50)}} = 115$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.965\left[0.658^{0.965(50)/(38.3)}\right]50 = 28.5 \text{ ksi}$$

<u>Nominal Compressive Strength</u>

$$P_n = F_{cr}A_g = 28.5(19.8) = 564 \text{ kips} \qquad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(564) = 508 \; kips$

ASD: $P_n / \Omega_c = {}^{564}/_{1.67} = 338 \; kips$

Comparison

Table 749: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
$\phi_c P_n$ (kips) [LRFD]	508	513	1.0%
P_n/Ω_c (kips) [ASD]	338	341	0.9%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\US\AISC\AISC 360-10 - E-2.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
*TABLE 1
*UNIT INCHES KIP
*ISECTION
*I1
*17 0.25 17 8 1 8 1 0 0 0
TABLE 2
UNIT INCHES KIP
WIDE FLANGE
I
19.75 17 0.25 8 1 1095.65 85.3529 5.41146 4.25 10.6667 8 1
END
UNIT inch KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
```

```
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 UPTABLE 2 I
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -70
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -210
UNIT INCHES KIP
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                          (UPT)
                           PASS   Eq. H1-1a            0.819         3
```

```
                      420.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     86.586  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.819(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   4.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.067E+01     Ayy:   4.250E+00     Cw:  5.461E+03             |
| Szz:   1.289E+02     Syy:   2.134E+01                                |
| Izz:   1.096E+03     Iyy:   8.535E+01     Ix:  5.411E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:         50.000   Fu:          65.000                            |
|-----------------------------------------------------------------------|
| Actual Member Length:       180.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     4.00     N/A     13.66    T.B4.1(a)-1 |
|                Slender        60.00     N/A     36.33    T.B4.1(a)-5 |
| Flexure     :  Compact         4.00     9.27    24.39    T.B4.1(b)-10 |
|                Compact        60.00    91.69   139.00    T.B4.1(b)-15 |
     STAAD PLANE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   8.89E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   9.63E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    4.20E+02   8.03E+02   0.523      Eq. E7-1      3     0.00 |
| Min Buck    4.20E+02   5.13E+02   0.819      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    4.20E+02   6.84E+02   0.614      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.86E+01    24.17   4.52E+01   5.02E+02   8.93E+02        |
| Min Buck    1.91E+01    86.59   2.89E+01   3.91E+01   5.70E+02        |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    1.88E+01   3.85E+01   7.60E+02                            |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   2.88E+02   0.000      Eq. G2-1      3     0.00  |
| Local-Y     0.00E+00   1.15E+02   0.000      Eq. G2-1      3     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     1.07E+01     1.00     1.20     4.00   3.20E+02             |
| Local-Y     4.25E+00     1.00     5.00    60.00   1.27E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   6.39E+03   0.000      Eq. F2-1      3     0.00  |
| Minor       0.00E+00   1.45E+03   0.000      Eq. F6-1      3     0.00  |
| Intermediate   Mn         My                                            |
| Major       7.10E+03   0.00E+00                                         |
| Minor       1.61E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   5.44E+03   0.000      Eq. F2-2      3     0.00  |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb      |
| Major       6.04E+03   0.00E+00    1.00    89.22   311.02   180.00    |
    STAAD PLANE                                               -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.819      Eq. H1-1a     3     0.00              |
| Flexure Tens           0.000      Eq. H1-1b     3     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          5.44E+03   0.00E+00   5.13E+02                    |
|                       1.45E+03   0.00E+00   4.20E+02                    |
| Flexure Tens          5.44E+03   0.00E+00   8.89E+02                    |
|                       1.45E+03   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
    63. LOAD LIST 4
    64. PARAMETER 2
    65. CODE AISC UNIFIED 2010
    66. FU 65 ALL
    67. FYLD 50 ALL
    68. METHOD ASD
    69. TRACK 2 ALL
    70. CHECK CODE ALL
 STEEL DESIGN
    STAAD PLANE                                               -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
```

```
======================================================================
      1 ST   I                        (UPT)
                          PASS   Eq. H1-1a           0.820        4
                     280.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     86.586  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.820(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.800E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.067E+01     Ayy:   4.250E+00     Cw:  5.461E+03             |
| Szz:   1.289E+02     Syy:   2.134E+01                                 |
| Izz:   1.096E+03     Iyy:   8.535E+01     Ix:  5.411E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       180.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     4.00     N/A     13.66    T.B4.1(a)-1 |
|                Slender        60.00     N/A     36.33    T.B4.1(a)-5 |
| Flexure     :  Compact         4.00     9.27    24.39    T.B4.1(b)-10|
|                Compact        60.00    91.69   139.00    T.B4.1(b)-15|
     STAAD PLANE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   5.91E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   6.42E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    2.80E+02   5.35E+02   0.524      Eq. E7-1      4     0.00 |
| Min Buck    2.80E+02   3.41E+02   0.820      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    2.80E+02   4.55E+02   0.615      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.86E+01    24.17   4.52E+01   5.02E+02   8.93E+02        |
| Min Buck    1.91E+01    86.59   2.89E+01   3.91E+01   5.70E+02        |
```

```
| Flexural       Ag       Fcr        Pn                                     |
| Tor Buck    1.88E+01   3.85E+01   7.60E+02                                |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z     0.00E+00   1.92E+02   0.000      Eq. G2-1      4     0.00   |
| Local-Y     0.00E+00   7.63E+01   0.000      Eq. G2-1      4     0.00   |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn               |
| Local-Z     1.07E+01     1.00      1.20      4.00   3.20E+02           |
| Local-Y     4.25E+00     1.00      5.00     60.00   1.27E+02           |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   4.25E+03   0.000      Eq. F2-1      4     0.00   |
| Minor       0.00E+00   9.65E+02   0.000      Eq. F6-1      4     0.00   |
| Intermediate   Mn         My                                            |
| Major       7.10E+03   0.00E+00                                         |
| Minor       1.61E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   3.62E+03   0.000      Eq. F2-2      4     0.00   |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb      |
| Major       6.04E+03   0.00E+00    1.00    89.22   311.02   180.00     |
     STAAD PLANE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp            0.820      Eq. H1-1a     4     0.00             |
| Flexure Tens            0.000      Eq. H1-1b     4     0.00             |
| Intermediate           Mcx /      Mrx /      Pc /                      |
|                        Mcy        Mry        Pr                        |
| Flexure Comp           3.62E+03   0.00E+00   3.41E+02                   |
|                        9.65E+02   0.00E+00   2.80E+02                   |
| Flexure Tens           3.62E+03   0.00E+00   5.91E+02                   |
|                        9.65E+02   0.00E+00   0.00E+00                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Built up I E.3

Verify the compressive strength of a built-up I section with slender flanges per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010. *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.

2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.3, pp. E-16 - E-20
3. Seaburg, Paul A. and Charles J. Carter. 1997. Design Guide 9: *Torsional Analysis of Structural Steel members*. AISC: Chicago, IL.

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference (2):

Determine if a built-up, ASTM A572 Grade 50 column with PL3/8 in. × 102 in. flanges and a PL1/4 in. × 74 in. web has sufficient available strength to carry a dead load of 40 kips and a live load of 120 kips in axial compression. The column's unbraced length is 15.0 ft in both axes and the ends are pinned.

Validation

Ultimate load:

$$P_u = 1.2(40 \text{ kips}) + 1.6(120 \text{ kips}) = 240 \text{ kips}$$

Service load:

$$P_a = 40 \text{ kips} + 120 \text{ kips} = 160 \text{ kips}$$

Section Properties

Built-up section properties (ignoring fillet welds):

$$A_g = 2(10.5)(0.375) + 7.25(0.25) = 9.69 \text{ in}^2$$

$$I_x = \Sigma Ad^2 + \Sigma\frac{bh^3}{12} = 2\left[10.5 \times 0.375(3.81)^2 + \frac{10.5(0.25)^3}{12}\right] + \frac{0.25(7.25)^3}{12} = 122 \text{ in}^4$$

$$I_y = 2\left(\frac{0.375 \times 10.5^3}{12}\right) + \frac{7.25 \times 0.25^3}{12} = 72.4 \text{ in}^4$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{72.4}{9.69}} = 2.73 \text{ in}$$

From the User Note in Section E4 of Reference 1:

$$C_w = \frac{I_y h_o^2}{4} = \frac{72.4(7.63)^2}{4} = 1,\ 050 \text{ in}^6$$

From Eq. 3.4 in Reference 3:

$$J = \Sigma\frac{bt^3}{3} = \frac{2(10.5)(0.375)^3 + 7.25(0.25)^3}{3} = 0.407 \text{ in}^4$$

Slenderness

Check for slender web and then determine the stiffened element (web) reduction factor.

Web slenderness:

$$\lambda = \frac{h}{t} = \frac{7.25}{0.25} = 29.0$$

The limiting web slenderness ratio, λ_r:

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,\ 000}{50}} = 35.9 > \lambda$$ (Table B4.1a, case 5)

Thus, the web is not slender and there is no reduction (i.e., Q_a *1.0)*

Check for slender flanges and then determine the unstiffened element (flange) reduction factor.

$$k_c = \frac{4}{\sqrt{h\ /t_w}} = \frac{4}{\sqrt{7.25\,/\,0.25}} = 0.743$$ (Table B4.1b note a)

$0.35 < k_c < 0.76$, OK.

Flange slenderness:

$$\lambda = \frac{b}{t} = \frac{5.25}{0.375} = 14.0$$

The limiting flange slenderness ratio, λ_r:

$$\lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.743(29,\ 000)}{50}} = 13.3 < \lambda$$ (Table B4.1a, case 2)

Thus, the flanges are slender.

$$0.64\sqrt{\frac{k_c E}{F_y}} = 13.3 < \frac{b}{t} = 14.0 < 1.17\sqrt{\frac{k_c E}{F_y}} = 1.17\sqrt{\frac{0.743(29,\ 000)}{50}} = 24.3$$

$$Q_s = 1.415 - 0.65\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{k_c E}} = 1.415 - 0.65(14.0)\sqrt{\frac{50}{0.743(29,\ 000)}} = 0.977$$ (Eq. E7-8)

$$Q = Q_s Q_a = 0.977(1.0) = 0.977$$

<u>Elastic Flexural Buckling Stress</u>

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{K_y L}{r_y} = \frac{1.0(15.0)(12)}{2.73} = 65.9$$

$$F_e = \frac{\pi^2 E}{(KL\ \ /\,r)^2} = \frac{\pi^2 29,\ 000}{(65.9)^2} = 65.9 \text{ ksi}$$ (Eq. E3-4)

<u>Nominal Compressive Strength</u>

$$\frac{KL}{r} = 65.9 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{0.977(50)}} = 115$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.977\left[0.658^{0.977(50)/(65.9)}\right]50 = 35.8 \text{ ksi}$$ (Eq E7-2)

$$P_n = F_{cr} A_g = 35.8(6.69) = 347 \text{ kips}$$ (Eq E7-1)

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(347) = 312$ *kips*

ASD: $P_n / \Omega_c = {}^{347}/_{1.67} = 208\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	312	314	negligible	
P_n / Ω_c (kips) [ASD]	208	209	negligible	
Slenderness ratio	65.9	65.86	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Built up I E.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 2
UNIT INCHES KIP
WIDE FLANGE
I
9.6875 8 0.25 10.5 0.375 122.496 72.361 0.406901 2 5.25 10.5 0.375
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
```

```
MATERIAL STEEL_36_KSI ALL
MEMBER PROPERTY AMERICAN
1 UPTABLE 2 I
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -120
UNIT INCHES KIP
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 65 ALL
FYLD 50 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                        (UPT)
                           PASS   Eq. H1-1a           0.764         3
                      240.00 C          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     65.861  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.764(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
```

```
|| DESIGN FORCES                                                            | |
|| Fx:   2.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                  | |
|| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  | |
||--------------------------------------------------------------------------| |
|| SECTION PROPERTIES  (UNIT: INCH)                                         | |
|| Azz:   5.250E+00     Ayy:   2.000E+00     Cw:  1.052E+03                 | |
|| Szz:   3.062E+01     Syy:   1.378E+01                                    | |
|| Izz:   1.225E+02     Iyy:   7.236E+01     Ix:  4.069E-01                 | |
||--------------------------------------------------------------------------| |
|| MATERIAL PROPERTIES                                                      | |
|| Fyld:          50.000   Fu:          65.000                              | |
||--------------------------------------------------------------------------| |
|| Actual Member Length:       180.000                                      | |
|| Design Parameters                                                        | |
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           | |
||--------------------------------------------------------------------------| |
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          | |
||                STIFFENED                                                 | |
|| Compression :  Slender        14.00     N/A     13.49    T.B4.1(a)-1     | |
||                Non-Slender    29.00     N/A     35.88    T.B4.1(a)-5     | |
|| Flexure     :  Non-Compact    14.00     9.15    24.08    T.B4.1(b)-10    | |
||                Compact        29.00    90.55   137.27    T.B4.1(b)-15    | |
     STAAD PLANE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
||--------------------------------------------------------------------------| |
|| CHECK FOR AXIAL TENSION                                                  | |
||                                                                          | |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      | |
|| Yield       0.00E+00   4.36E+02   0.000      Eq. D2-1      3     0.00    | |
|| Rupture     0.00E+00   4.72E+02   0.000      Eq. D2-2      3     0.00    | |
||--------------------------------------------------------------------------| |
|| CHECK FOR AXIAL COMPRESSION                                              | |
||                                                                          | |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      | |
|| Maj Buck    2.40E+02   3.57E+02   0.672      Eq. E7-1      3     0.00    | |
|| Min Buck    2.40E+02   3.14E+02   0.764      Eq. E7-1      3     0.00    | |
|| Flexural                                                                 | |
|| Tor Buck    2.40E+02   3.21E+02   0.747      Eq. E7-1      3     0.00    | |
|| Intermediate                                                             | |
|| Results     Eff Area     KL/r      Fcr        Fe         Pn              | |
|| Maj Buck    9.69E+00    50.62   4.09E+01   1.12E+02   3.97E+02          | |
|| Min Buck    9.69E+00    65.86   3.60E+01   6.60E+01   3.49E+02          | |
|| Flexural       Ag        Fcr        Pn                                   | |
|| Tor Buck    9.69E+00   3.68E+01   3.57E+02                              | |
||--------------------------------------------------------------------------| |
|| CHECK FOR SHEAR                                                          | |
||                                                                          | |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      | |
|| Local-Z     0.00E+00   1.42E+02   0.000      Eq. G2-1      3     0.00    | |
|| Local-Y     0.00E+00   6.00E+01   0.000      Eq. G2-1      3     0.00    | |
|| Intermediate                                                             | |
|| Results        Aw        Cv       Kv      h/tw      Vn                   | |
|| Local-Z     5.25E+00     1.00     1.20    14.00   1.58E+02              | |
|| Local-Y     2.00E+00     1.00     0.00    29.00   6.00E+01              | |
||--------------------------------------------------------------------------| |
```

```
| CHECK FOR BENDING-YIELDING                                                 |
|                                                                            |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Major       0.00E+00   1.50E+03   0.000      Eq. F2-1      3     0.00     |
| Minor       0.00E+00   9.35E+02   0.000      Eq. F6-1      3     0.00     |
| Intermediate   Mn        My                                                |
| Major       1.67E+03   0.00E+00                                           |
| Minor       1.04E+03   0.00E+00                                           |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                               |
|                                                                            |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Major       0.00E+00   1.36E+03   0.000      Eq. F2-2      3     0.00     |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb        |
| Major       1.51E+03   0.00E+00    1.00   115.84   354.09   180.00       |
     STAAD PLANE                                          -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                    |
|                                                                            |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Major       0.00E+00   1.33E+03   0.000      Eq. F3-1      3     0.00     |
| Minor       0.00E+00   7.73E+02   0.000      Eq. F6-2      3     0.00     |
| Intermediate   Mn        Fcr                                               |
| Major       1.47E+03   0.00E+00                                           |
| Minor       8.58E+02   0.00E+00                                           |
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                    |
|                       RATIO      CRITERIA    L/C     LOC                   |
| Flexure Comp          0.764      Eq. H1-1a     3     0.00                  |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00                  |
| Intermediate         Mcx /      Mrx /      Pc /                           |
|                      Mcy        Mry        Pr                             |
| Flexure Comp         1.33E+03   0.00E+00   3.14E+02                       |
|                      7.73E+02   0.00E+00   2.40E+02                       |
| Flexure Tens         1.33E+03   0.00E+00   4.36E+02                       |
|                      7.73E+02   0.00E+00   0.00E+00                       |
|------------------------------------------------------------------------|
    65. LOAD LIST 4
    66. PARAMETER 2
    67. CODE AISC UNIFIED 2010
    68. FU 65 ALL
    69. FYLD 50 ALL
    70. METHOD ASD
    71. TRACK 2 ALL
    72. CHECK CODE ALL
 STEEL DESIGN
     STAAD PLANE                                          -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   I                        (UPT)
                          PASS   Eq. H1-1a           0.765         4
                     160.00 C          0.00          0.00        0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     65.861  L/C  :     4                |
 | Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.765(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H1-1a                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   1.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   5.250E+00     Ayy:   2.000E+00     Cw:  1.052E+03              |
 | Szz:   3.062E+01     Syy:   1.378E+01                                 |
 | Izz:   1.225E+02     Iyy:   7.236E+01     Ix:  4.069E-01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          65.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:       180.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Slender        14.00     N/A     13.49    T.B4.1(a)-1  |
 |                Non-Slender    29.00     N/A     35.88    T.B4.1(a)-5  |
 | Flexure     :  Non-Compact    14.00     9.15    24.08    T.B4.1(b)-10 |
 |                Compact        29.00    90.55   137.27    T.B4.1(b)-15 |
      STAAD PLANE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       0.00E+00   2.90E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   3.15E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck    1.60E+02   2.38E+02   0.674      Eq. E7-1      4     0.00 |
 | Min Buck    1.60E+02   2.09E+02   0.765      Eq. E7-1      4     0.00 |
 | Flexural                                                              |
 | Tor Buck    1.60E+02   2.14E+02   0.749      Eq. E7-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    9.69E+00    50.62   4.09E+01   1.12E+02   3.97E+02        |
 | Min Buck    9.69E+00    65.86   3.60E+01   6.60E+01   3.49E+02        |
 | Flexural       Ag       Fcr        Pn                                 |
```

```
| Tor Buck    9.69E+00   3.68E+01   3.57E+02                               |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   9.43E+01   0.000      Eq. G2-1      4     0.00  |
| Local-Y     0.00E+00   4.00E+01   0.000      Eq. G2-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     5.25E+00     1.00     1.20    14.00   1.58E+02              |
| Local-Y     2.00E+00     1.00     0.00    29.00   6.00E+01              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   9.97E+02   0.000      Eq. F2-1      4     0.00  |
| Minor       0.00E+00   6.22E+02   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn        My                                             |
| Major       1.67E+03   0.00E+00                                         |
| Minor       1.04E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   9.02E+02   0.000      Eq. F2-2      4     0.00  |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       1.51E+03   0.00E+00    1.00   115.84   354.09   180.00     |
    STAAD PLANE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   8.82E+02   0.000      Eq. F3-1      4     0.00  |
| Minor       0.00E+00   5.14E+02   0.000      Eq. F6-2      4     0.00  |
| Intermediate   Mn        Fcr                                            |
| Major       1.47E+03   0.00E+00                                         |
| Minor       8.58E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp           0.765      Eq. H1-1a     4     0.00              |
| Flexure Tens           0.000      Eq. H1-1b     4     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          8.82E+02   0.00E+00   2.09E+02                    |
|                       5.14E+02   0.00E+00   1.60E+02                    |
| Flexure Tens          8.82E+02   0.00E+00   2.90E+02                    |
|                       5.14E+02   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Double L E.5

Reference

American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E5, pp. E-26 - E-30.

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links
- *D1.A.5.2 Axial Compression* (on page 1093)

Problem

From the reference:

> Verify the strength of a 2L 4×3-1/2×3/8 LLBB (3/8-in. separation) strut, ASTM A36, with a length of 8 ft and pinned ends carrying an axial dead load of 20 kips and live load of 60 kips.

Refer to the reference for calculations.

Comparison

Table 750: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	128	127	negligible	
P_n/Ω_c (kips) [ASD]	85.0	84.6	negligible	

Note: STAAD.Pro does *not* perform design per Section E6 of the specification. It is assumed that sufficient intermediate connectors of an appropriate type are present. It is left to the engineer to verify the intermediate connector design.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\360-2010\AISC 360-10 - E.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
```

```
UNIT FEET KIP
JOINT COORDINATES
1 0 8 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L40356 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -60
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
LX 56 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 36 ALL
FU 58 ALL
LX 56 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L40356                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.943         3
                      120.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     76.914  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.943(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.625E+00     Ayy:   1.500E+00     Cw:  0.000E+00              |
| Szz:   0.000E+00     Syy:   0.000E+00                                 |
| Izz:   8.350E+00     Iyy:   1.541E+01     Ix:  2.512E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.67     N/A     12.77    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         9.33    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      0.00E+00   1.74E+02   0.000      Eq. D2-1      3     0.00  |
| Rupture    0.00E+00   2.33E+02   0.000      Eq. D2-2      3     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck   1.20E+02   1.27E+02   0.943      Eq. E3-1      3     0.00  |
```

```
| Min Buck    1.20E+02   1.47E+02   0.818      Eq. E3-1      3     0.00 |
| Flexural                                                               |
| Tor Buck    1.20E+02   1.36E+02   0.882      Eq. E4-1      3     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    5.36E+00    76.91   2.64E+01   4.84E+01   1.41E+02        |
| Min Buck    5.36E+00    56.61   3.04E+01   8.93E+01   1.63E+02        |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    0.00E+00   2.82E+01   1.51E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   5.10E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.92E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw         Cv        Kv      h/tw      Vn              |
| Local-Z     2.62E+00     1.00      1.20     9.33   5.67E+01           |
| Local-Y     1.50E+00     1.00      1.20    10.67   3.24E+01           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.55E+02   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   2.06E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                           |
| Major       1.72E+02   0.00E+00                                        |
| Minor       2.29E+02   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.09E+03   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       2.32E+03   0.00E+00    1.00     0.00     0.00    96.00    |
     STAAD SPACE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.943      Eq. H2-1      3     0.00           |
| Flexure Tens           0.000      Eq. H2-1      3     0.00           |
| Intermediate          frbw /     frbz /     fra /                     |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   2.24E+01                  |
|                       5.18E+01   5.18E+01   2.37E+01                  |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                  |
|                       2.23E+01   5.18E+01   3.24E+01                  |
|-----------------------------------------------------------------------|
    53. LOAD LIST 4
    54. PARAMETER 2
    55. CODE AISC UNIFIED 2010
    56. METHOD ASD
    57. FYLD 36 ALL
    58. FU 58 ALL
```

```
   59. LX 56 ALL
   60. TRACK 2 ALL
   61. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 LD   L40356                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.945         4
                       80.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     76.914  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.945(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.625E+00     Ayy:   1.500E+00     Cw:  0.000E+00              |
| Szz:   0.000E+00     Syy:   0.000E+00                                 |
| Izz:   8.350E+00     Iyy:   1.541E+01     Ix:  2.512E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.67     N/A     12.77    T.B4.1(a)-3 |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         9.33    15.33    25.83    T.B4.1(b)-12|
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.16E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.55E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    8.00E+01   8.46E+01   0.945      Eq. E3-1      4     0.00   |
| Min Buck    8.00E+01   9.76E+01   0.820      Eq. E3-1      4     0.00   |
| Flexural                                                                |
| Tor Buck    8.00E+01   9.05E+01   0.884      Eq. E4-1      4     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r       Fcr        Fe          Pn           |
| Maj Buck    5.36E+00    76.91   2.64E+01   4.84E+01   1.41E+02          |
| Min Buck    5.36E+00    56.61   3.04E+01   8.93E+01   1.63E+02          |
| Flexural       Ag        Fcr         Pn                                 |
| Tor Buck    0.00E+00   2.82E+01   1.51E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   3.40E+01   0.000      Eq. G2-1      4     0.00   |
| Local-Y     0.00E+00   1.94E+01   0.000      Eq. G2-1      4     0.00   |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn               |
| Local-Z     2.62E+00     1.00      1.20      9.33   5.67E+01            |
| Local-Y     1.50E+00     1.00      1.20     10.67   3.24E+01            |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.03E+02   0.000      Eq. F9-1      4     0.00   |
| Minor       0.00E+00   1.37E+02   0.000      Eq. F6-1      4     0.00   |
| Intermediate   Mn         My                                            |
| Major       1.72E+02   0.00E+00                                         |
| Minor       2.29E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.39E+03   0.000      Eq. F9-4      4     0.00   |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb      |
| Major       2.32E+03   0.00E+00    1.00     0.00     0.00    96.00      |
     STAAD SPACE                                            -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp            0.945      Eq. H2-1      4     0.00             |
| Flexure Tens            0.000      Eq. H2-1      4     0.00             |
| Intermediate           frbw /     frbz /     fra /                      |
|                        Fcbw       Fcbz       Fca                        |
| Flexure Comp           0.00E+00   0.00E+00   1.49E+01                   |
|                        3.45E+01   3.45E+01   1.58E+01                   |
| Flexure Tens           0.00E+00   0.00E+00   0.00E+00                   |
|                        1.48E+01   3.45E+01   2.16E+01                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Double L E.6

Verify the compressive strength of an unbraced wide-flange section column about the Y-Y axis per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.6, pp. E-31 - E-36

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From the reference example E.1C:

Determine if a 2L5×3×1/4 LLBB (3/4-in. separation) strut, ASTM A36, with a length of 8 ft and pinned ends has sufficient available strength to support a dead load of 10 kips and live load of 30 kips in axial compression. Also, calculate the required number of pretensioned bolted or welded intermediate connectors.

Validation

Ultimate load:

$$P_u = 1.2(10\text{ kips}) + 1.6(30\text{ kips}) = 60\text{ kips}$$

Service load:

$$P_a = 10\text{ kips} + 30\text{ kips} = 40\text{ kips}$$

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

Slenderness

$$\lambda = \frac{b}{t} = \frac{5.0}{0.25} = 20.0$$

The limiting slenderness ratio, λ_r:

$$\lambda_r = 0.45\sqrt{\frac{E}{F_y}} = 0.45\sqrt{\frac{29,000}{36}} = 12.8 < \lambda \qquad \text{(Table B4.1a, case 3)}$$

Thus, the outstanding legs are slender.

Check for slender web and then determine the stiffened element (web) reduction factor.

$$0.45\sqrt{\frac{E}{F_y}} = 12.8 < \frac{b}{t} = 20.0 < 0.91\sqrt{\frac{E}{F_y}} = 0.91\sqrt{\frac{29,000}{36}} = 25.8$$

$$Q_s = 1.34 - 0.76\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{E}} = 1.34 - 0.76(20.0)\sqrt{\frac{36}{29,000}} = 0.804 \qquad \text{(Eq. E7-11)}$$

$$Q = Q_s Q_a = 0.804(1.0) = 0.804$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y$$

where

F_e	=	$\left(\frac{F_{ey} + F_{ez}}{2H}\right)\left[1 - \sqrt{1 - \frac{4F_{ey}F_{ez}H}{(F_{ey} + F_{ez})^2}}\right]$ (Eq E4-5)
K_i	=	0.5 for angles back-to-back
r_i	=	$r_z = 0.652$ in; the radius of gyration for a single angle
a	=	32 in spacing between connectors (using the bolts at 1/3 points from Reference 2)
$\left(\frac{K_y L}{r_y}\right)_m$	=	$\sqrt{\left(\frac{KL}{r}\right)_0^2 + \left(\frac{K_i a}{r_i}\right)^2} = \sqrt{\left(\frac{1.0 \times 96}{1.33}\right)^2 + \left(\frac{0.5 \times 32}{0.652}\right)^2} = 76.3$ for $\frac{a}{r_i} = \frac{32\ in}{0.652\ in} = 49.1 > 40$
F_{ey}	=	$\frac{\pi^2 E}{\left(\frac{K_y L}{r_y}\right)_m^2} = \frac{\pi^2(29,\ 000)}{(76.3)^2} = 49.2\ ksi$
F_{ez}	=	$\frac{GJ}{A_g \bar{r}_0^2} = \frac{11,\ 200(0.04328) \times 2\ angles}{3.88(2.59)^2} = 37.7\ ksi$

Thus, $\left(\frac{49.2 + 37.7}{2 \times 0.657}\right)\left[1 - \sqrt{1 - \frac{4(49.2)(37.7)(0.657)}{(49.2 + 37.7)^2}}\right] = 26.8\ ksi$

Nominal Compressive Strength

$$\frac{QF_y}{2.25} = \frac{0.804(36)}{2.25} = 12.9 \text{ ksi} < 26.8 \text{ ksi}$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.804\left[0.658^{0.804(36)/(26.8)}\right]36 = 18.4 \text{ ksi} \qquad \text{(Eq E7-2)}$$

$$P_n = F_{cr}A_g = 18.4(3.88) = 71.4 \text{ kips} \qquad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(71.4) = 64.3\ kips$

ASD: $P_n / \Omega_c = {}^{71.4}/_{1.67} = 42.8\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	64.3	63.8	negligible	
P_n/Ω_c (kips) [ASD]	42.8	42.4	negligible	

Note: STAAD.Pro does *not* perform design per Section E6 of the specification. It is assumed that sufficient intermediate connectors of an appropriate type are present. It is left to the engineer to verify the intermediate connector design.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Double L E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 8 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L50304 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -10
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
```

```
JOINT LOAD
1 FY -30
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 58 ALL
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FU 58 ALL
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L50304                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.941         3
                       60.00 C         0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     75.420  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.941(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   6.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.500E+00     Ayy:   1.250E+00     Cw:  0.000E+00             |
| Szz:   3.055E+00     Syy:   2.075E+00                                 |
| Izz:   1.022E+01     Iyy:   7.003E+00     Ix:  8.083E-02             |
|-----------------------------------------------------------------------|
```

```
| MATERIAL PROPERTIES                                                    |
| Fyld:          36.000   Fu:          58.000                            |
|------------------------------------------------------------------------|
| Actual Member Length:        96.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00         |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Slender        20.00     N/A     12.77    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   1.26E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   1.69E+02   0.000      Eq. D2-2      3     0.00 |
|------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                            |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    6.00E+01   8.72E+01   0.688      Eq. E7-1      3     0.00 |
| Min Buck    6.00E+01   8.15E+01   0.737      Eq. E7-1      3     0.00 |
| Flexural                                                               |
| Tor Buck    6.00E+01   6.38E+01   0.941      Eq. E7-1      3     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    3.88E+00    59.15   2.50E+01   8.18E+01   9.69E+01        |
| Min Buck    3.88E+00    71.46   2.33E+01   5.61E+01   9.05E+01        |
| Flexural       Ag       Fcr         Pn                                 |
| Tor Buck    3.88E+00   1.83E+01   7.08E+01                            |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   2.92E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.43E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     1.50E+00     1.00     1.20    12.00   3.24E+01             |
| Local-Y     1.25E+00     1.00     1.20    20.00   2.70E+01             |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.58E+02   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   1.08E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                           |
| Major       1.76E+02   0.00E+00                                        |
| Minor       1.20E+02   0.00E+00                                        |
|------------------------------------------------------------------------|
```

```
|| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
||                                                                       |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|| Major      0.00E+00   1.04E+03   0.000      Eq. F9-4      3     0.00 |
|| Intermediate   Mn        Me         Cb       Lp       Lr       Lb     |
|| Major      1.16E+03   0.00E+00    1.00     0.00     0.00    96.00    |
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.941      Eq. H2-1      3     0.00            |
| Flexure Tens           0.000      Eq. H2-1      3     0.00            |
| Intermediate          frbw /     frbz /     fra /                    |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   1.55E+01                  |
|                       5.18E+01   5.18E+01   1.64E+01                  |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                  |
|                       2.57E+01   5.18E+01   3.24E+01                  |
|-----------------------------------------------------------------------|
    70. LOAD LIST 4
    71. PARAMETER 2
    72. CODE AISC UNIFIED 2010
    73. METHOD ASD
    74. FU 58 ALL
    75. FYLD 36 ALL
    76. CSPACING 32 ALL
    77. TRACK 2 ALL
    78. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 LD   L50304                   (AISC SECTIONS)
                          PASS   Eq. H2-1            0.943        4
                       40.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     75.420  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.943(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   4.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
```

```
| Azz:   1.500E+00     Ayy:   1.250E+00     Cw:  0.000E+00                |
| Szz:   3.055E+00     Syy:   2.075E+00                                     |
| Izz:   1.022E+01     Iyy:   7.003E+00     Ix:  8.083E-02                 |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.00     N/A     12.77    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   8.36E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.13E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    4.00E+01   5.80E+01   0.689      Eq. E7-1      4     0.00 |
| Min Buck    4.00E+01   5.42E+01   0.738      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    4.00E+01   4.24E+01   0.943      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    3.88E+00    59.15   2.50E+01   8.18E+01   9.69E+01        |
| Min Buck    3.88E+00    71.46   2.33E+01   5.61E+01   9.05E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    3.88E+00   1.83E+01   7.08E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.94E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.62E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.50E+00     1.00     1.20    12.00   3.24E+01            |
| Local-Y     1.25E+00     1.00     1.20    20.00   2.70E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.05E+02   0.000      Eq. F9-1      4     0.00 |
| Minor       0.00E+00   7.16E+01   0.000      Eq. F6-1      4     0.00 |
```

```
| Intermediate   Mn        My                                         |
| Major       1.76E+02   0.00E+00                                     |
| Minor       1.20E+02   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                     |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   6.93E+02   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn        Me         Cb       Lp        Lr       Lb    |
| Major       1.16E+03   0.00E+00    1.00     0.00     0.00    96.00    |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp           0.943      Eq. H2-1      4     0.00          |
| Flexure Tens           0.000      Eq. H2-1      4     0.00          |
| Intermediate          frbw /     frbz /     fra /                   |
|                       Fcbw       Fcbz       Fca                     |
| Flexure Comp          0.00E+00   0.00E+00   1.03E+01                |
|                       3.45E+01   3.45E+01   1.09E+01                |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                |
|                       1.71E+01   3.45E+01   2.16E+01                |
|---------------------------------------------------------------------|
```

V. AISC 360-10 WT E.7

Verify the compressive strength of a WT section per the LRFD and ADS methods of the AISC 360-10 codes.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.7, pp.E-37 - E-41

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference (2):

> Select an ASTM A992 nonslender WT-shape compression member with a length of 20 ft to support a dead load of 20 kips and live load of 60 kips in axial compression. The ends are pinned.

See example E.7 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	128	127	negligible	
P_n / Ω_c (kips) [ASD]	85.0	84.3	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 WT E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W14X68
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -20
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
```

```
2 FY -60
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 50 ALL
FU 65 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 50 ALL
FU 65 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W14X68                   (AISC SECTIONS)
                            PASS   Eq. H2-1             0.947         3
                      120.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    133.504  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.947(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.200E+00     Ayy:   2.905E+00     Cw:  3.173E+00             |
| Szz:   5.635E+00     Syy:   1.210E+01                                |
| Izz:   3.232E+01     Iyy:   6.050E+01     Ix:  1.505E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
```

```
| Fyld:          50.000   Fu:          65.000                          |
|----------------------------------------------------------------------|
| Actual Member Length:       240.000                                  |
| Design Parameters                                                    |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                             |
| Compression :  Non-Slender    16.87     N/A     18.06    T.B4.1(a)-4 |
|                  N/A           N/A      N/A      N/A      N/A        |
| Flexure     :  Compact         6.94     9.15    24.08    T.B4.1(b)-10|
|                Compact        16.87    20.23    24.81    T.B4.1(b)-14|
     STAAD SPACE                                               -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                              |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      0.00E+00   4.50E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture    0.00E+00   4.88E+02   0.000      Eq. D2-2      3     0.00 |
|----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                          |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   1.20E+02   1.27E+02   0.947      Eq. E3-1      3     0.00 |
| Min Buck   1.20E+02   2.24E+02   0.535      Eq. E3-1      3     0.00 |
| Flexural                                                             |
| Tor Buck   1.20E+02   2.21E+02   0.542      Eq. E4-1      3     0.00 |
| Intermediate                                                         |
| Results    Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck   1.00E+01   133.50   1.41E+01   1.61E+01   1.41E+02        |
| Min Buck   1.00E+01    97.57   2.49E+01   3.01E+01   2.49E+02        |
| Flexural      Ag        Fcr        Pn                                |
| Tor Buck   0.00E+00   2.46E+01   2.46E+02                            |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    0.00E+00   1.94E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y    0.00E+00   7.84E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                         |
| Results       Aw         Cv       Kv      h/tw      Vn              |
| Local-Z    7.20E+00     1.00     1.20     6.94   2.16E+02            |
| Local-Y    2.90E+00     1.00     1.20    16.87   8.71E+01            |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      0.00E+00   4.06E+02   0.000      Eq. F9-1      3     0.00 |
| Minor      0.00E+00   8.30E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                         |
| Major      4.51E+02   0.00E+00                                       |
| Minor      9.23E+02   0.00E+00                                       |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
```

```
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   3.06E+03   0.000      Eq. F9-4      3     0.00   |
| Intermediate   Mn        Me         Cb       Lp       Lr       Lb      |
| Major       3.40E+03   0.00E+00    1.00     0.00     0.00   240.00     |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp           0.947      Eq. H2-1      3     0.00             |
| Flexure Tens           0.000      Eq. H2-1      3     0.00             |
| Intermediate          frbw /     frbz /     fra /                      |
|                       Fcbw       Fcbz       Fca                        |
| Flexure Comp          0.00E+00   0.00E+00   1.20E+01                   |
|                       7.20E+01   6.86E+01   1.27E+01                   |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                   |
|                       1.59E+01   6.86E+01   4.50E+01                   |
|------------------------------------------------------------------------|
    58. LOAD LIST 4
    59. PARAMETER 2
    60. CODE AISC UNIFIED 2010
    61. FYLD 50 ALL
    62. FU 65 ALL
    63. METHOD ASD
    64. TRACK 2 ALL
    65. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1 T    W14X68                   (AISC SECTIONS)
                            PASS   Sec. E1               0.949        4
                       80.00 C         0.00           0.00       0.00
|------------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :    133.504  L/C  :     4                 |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     0.949(PASS)                  |
|          Loc  :    0.00   Condition :   Sec. E1                        |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   8.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00                |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   7.200E+00     Ayy:   2.905E+00     Cw:  3.173E+00              |
| Szz:   5.635E+00     Syy:   1.210E+01                                  |
```

```
| Izz:   3.232E+01     Iyy:   6.050E+01     Ix:  1.505E+00                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:          50.000   Fu:          65.000                              |
|--------------------------------------------------------------------------|
| Actual Member Length:       240.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          |
|                STIFFENED                                                 |
| Compression :  Non-Slender    16.87     N/A     18.06    T.B4.1(a)-4    |
|                  N/A           N/A      N/A      N/A      N/A            |
| Flexure     :  Compact         6.94     9.15    24.08    T.B4.1(b)-10   |
|                Compact        16.87    20.23    24.81    T.B4.1(b)-14   |
    STAAD SPACE                                                -- PAGE NO.
10
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   2.99E+02   0.000      Eq. D2-1      4     0.00    |
| Rupture    0.00E+00   3.25E+02   0.000      Eq. D2-2      4     0.00    |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   8.00E+01   8.43E+01   0.949      Eq. E3-1      4     0.00    |
| Min Buck   8.00E+01   1.49E+02   0.536      Eq. E3-1      4     0.00    |
| Flexural                                                                 |
| Tor Buck   8.00E+01   1.47E+02   0.544      Eq. E4-1      4     0.00    |
| Intermediate                                                             |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   1.00E+01   133.50   1.41E+01   1.61E+01   1.41E+02           |
| Min Buck   1.00E+01    97.57   2.49E+01   3.01E+01   2.49E+02           |
| Flexural      Ag        Fcr        Pn                                    |
| Tor Buck   0.00E+00   2.46E+01   2.46E+02                                |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z    0.00E+00   1.29E+02   0.000      Eq. G2-1      4     0.00    |
| Local-Y    0.00E+00   5.22E+01   0.000      Eq. G2-1      4     0.00    |
| Intermediate                                                             |
| Results       Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z    7.20E+00     1.00     1.20     6.94   2.16E+02                |
| Local-Y    2.90E+00     1.00     1.20    16.87   8.71E+01                |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      0.00E+00   2.70E+02   0.000      Eq. F9-1      4     0.00    |
| Minor      0.00E+00   5.52E+02   0.000      Eq. F6-1      4     0.00    |
| Intermediate   Mn         My                                             |
| Major      4.51E+02   0.00E+00                                           |
```

```
| Minor       9.23E+02   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   2.03E+03   0.000      Eq. F9-4      4     0.00  |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb     |
| Major       3.40E+03   0.00E+00    1.00     0.00     0.00   240.00     |
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp            0.949      Eq. H2-1      4     0.00             |
| Flexure Tens            0.000      Eq. H2-1      4     0.00             |
| Intermediate           frbw /     frbz /     fra /                      |
|                        Fcbw       Fcbz       Fca                       |
| Flexure Comp           0.00E+00   0.00E+00   8.00E+00                   |
|                        4.79E+01   4.57E+01   8.43E+00                   |
| Flexure Tens           0.00E+00   0.00E+00   0.00E+00                   |
|                        1.06E+01   4.57E+01   2.99E+01                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Rect HSS E.9

Verify the compressive strength of an unbraced column with a rectangular HSS section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.9, pp. E-47 - E-49

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference (2):

> Select an ASTM A500 Grade B rectangular HSS compression member, with a length of 20 ft, to support a dead load of 85 kips and live load of 255 kips in axial compression. The base is fixed and the top is pinned.

Validation

Ultimate load:

$$P_u = 1.2(85\text{ kips}) + 1.6(255\text{ kips}) = 510\text{ kips}$$

Service load:

$$P_a = 85\text{ kips} + 255\text{ kips} = 340\text{ kips}$$

Slenderness

Note: The corner radius is taken as 1.5× the design wall thickness per Section B4.1b of Reference 1.

$$\lambda = \frac{h}{t} = \frac{12.0 - 3(0.349)}{0.349} = 31.4$$

The limiting slenderness ratio, λ_r:

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29,\,000}{46}} = 35.2 > \lambda \qquad \text{(Table B4.1a, case 3)}$$

Thus, the section does not contain slender elements.

Per Table C-A-7.1 in Reference 1, $K = 0.8$ for fixed-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{K_y L}{r_y} = \frac{0.8(20)(12)}{4.01} = 47.9 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\,000}{46}} = 118$$

$$F_e = \frac{\pi^2 E}{\left(\frac{K_y L}{r_y}\right)_m^2} = \frac{\pi^2(29,\,000)}{(47.9)^2} = 125\text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$F_{cr} = \left(0.658^{F_y/F_e}\right)F_y = (0.658^{46/125})46 = 39.4\text{ ksi} \qquad \text{(Eq. E3-2)}$$

Nominal Compressive Strength

$$P_n = F_{cr}A_g = 39.4(14.6) = 575\text{ kips} \qquad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(575) = 518\ kips$

ASD: $P_n / \Omega_c = {}^{575}\!/_{1.67} = 344\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	518	518	none	
P_n/Ω_c (kips) [ASD]	344	345	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Rect HSS E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-19
```

```
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 240 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST12X10X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -85
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -255
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
KX 0.8 ALL
KY 0.8 ALL
METHOD LRFD
TRACK 2 ALL
KZ 0.8 ALL
FU 58 ALL
FYLD 46 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
KX 0.8 ALL
KY 0.8 ALL
KZ 0.8 ALL
FU 58 ALL
FYLD 46 ALL
METHOD ASD
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST12X10X0.375          (AISC SECTIONS)
                            PASS   Eq. H1-1a            0.985         3
                      510.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     47.959  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.985(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.100E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.249E+00     Ayy:   7.645E+00     Cw:  0.000E+00             |
| Szz:   5.167E+01     Syy:   4.680E+01                                 |
| Izz:   3.100E+02     Iyy:   2.340E+02     Ix:  4.210E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       240.000                                   |
| Design Parameters                                                     |
| Kz:    0.80 Ky:    0.80 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    31.38     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact        25.65    28.12    35.15    T.B4.1(b)-17 |
|                Compact        31.38    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                             -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   6.04E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   6.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL COMPRESSION                                            |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    5.10E+02   5.38E+02   0.948      Eq. E3-1      3     0.00 |
| Min Buck    5.10E+02   5.18E+02   0.985      Eq. E3-1      3     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.46E+01    41.67   4.09E+01   1.65E+02   5.98E+02        |
| Min Buck    1.46E+01    47.96   3.94E+01   1.24E+02   5.75E+02        |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.55E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.90E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     6.25E+00     1.00     5.00    25.65   1.72E+02            |
| Local-Y     7.65E+00     1.00     5.00    31.38   2.11E+02            |
|------------------------------------------------------------------------|
| CHECK FOR TORSION                                                      |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             0.00E+00   1.94E+03   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                             |
|             2.76E+01   2.16E+03                                        |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.53E+03   0.000      Eq. F7-1      3     0.00 |
| Minor       0.00E+00   2.24E+03   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                           |
| Major       2.81E+03   0.00E+00                                        |
| Minor       2.48E+03   0.00E+00                                        |
    STAAD SPACE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Minor       0.00E+00   2.10E+03   0.000      Eq. F7-2      3     0.00 |
| Intermediate   Mn         Fcr                                          |
| Minor       2.33E+03   0.00E+00                                        |
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.985      Eq. H1-1a     3     0.00              |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00              |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.53E+03   0.00E+00   5.18E+02                   |
|                      2.10E+03   0.00E+00   5.10E+02                   |
| Flexure Tens         2.53E+03   0.00E+00   6.04E+02                   |
|                      2.10E+03   0.00E+00   0.00E+00                   |
```

```
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. KX 0.8 ALL
    54. KY 0.8 ALL
    55. KZ 0.8 ALL
    56. FU 58 ALL
    57. FYLD 46 ALL
    58. METHOD ASD
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST12X10X0.375          (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.987         4
                      340.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     47.959  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.987(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   3.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.249E+00     Ayy:   7.645E+00     Cw:  0.000E+00              |
| Szz:   5.167E+01     Syy:   4.680E+01                                 |
| Izz:   3.100E+02     Iyy:   2.340E+02     Ix:  4.210E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       240.000                                   |
| Design Parameters                                                     |
| Kz:    0.80 Ky:    0.80 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
```

```
|                Non-Slender    31.38     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact        25.65    28.12    35.15    T.B4.1(b)-17 |
|                Compact        31.38    60.76   143.12    T.B4.1(b)-19 |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   4.02E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.23E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    3.40E+02   3.58E+02   0.950      Eq. E3-1      4     0.00 |
| Min Buck    3.40E+02   3.45E+02   0.987      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.46E+01    41.67   4.09E+01   1.65E+02   5.98E+02        |
| Min Buck    1.46E+01    47.96   3.94E+01   1.24E+02   5.75E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.03E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.26E+02   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     6.25E+00     1.00     5.00    25.65   1.72E+02            |
| Local-Y     7.65E+00     1.00     5.00    31.38   2.11E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.29E+03   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.76E+01   2.16E+03                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.68E+03   0.000      Eq. F7-1      4     0.00 |
| Minor       0.00E+00   1.49E+03   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       2.81E+03   0.00E+00                                      |
| Minor       2.48E+03   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
```

```
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C     LOC |
| Minor       0.00E+00   1.40E+03   0.000      Eq. F7-2      4     0.00 |
| Intermediate   Mn        Fcr                                            |
| Minor       2.33E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.987      Eq. H1-1a    4     0.00              |
| Flexure Tens           0.000      Eq. H1-1b    4     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                       |
|                       Mcy        Mry        Pr                         |
| Flexure Comp          1.68E+03   0.00E+00   3.45E+02                   |
|                       1.40E+03   0.00E+00   3.40E+02                   |
| Flexure Tens          1.68E+03   0.00E+00   4.02E+02                   |
|                       1.40E+03   0.00E+00   0.00E+00                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Pipe E.11

Verify the compressive strength of a pipe section braced at the midpoint in the Y-Y direction per the LRFD and ADS methods of the AISC 360-10 codes.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.11, pp. E-54 - E-56

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference (2):

> Select an ASTM A53 Grade B Pipe compression member with a length of 30 ft to support a dead load of 35 kips and live load of 105 kips in axial compression. The column is pin-connected at the ends in both axes and braced at the midpoint in the y-y direction.

Validation

Ultimate load:

$$P_u = 1.2(35 \text{ kips}) + 1.6(105 \text{ kips}) = 210 \text{ kips}$$

Service load:

$$P_a = 35 \text{ kips} + 105 \text{ kips} = 140 \text{ kips}$$

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns.

$$\left(\frac{KL}{r}\right)_x = \frac{1.0 \times 30 \times 12}{3.68} = 97.8$$

$$\left(\frac{KL}{r}\right)_y = \frac{1.0 \times 15 \times 12}{3.68} = 48.9$$

So buckling about the x-x axis governs. For 35 ksi, no pipe sections are slender per Table 4-6 in Reference 1.

$$\left(\frac{KL}{r}\right)_{max} = 97.8 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{35}} = 136$$

The critical stress

$$F_e = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)^2} = \frac{\pi^2 29,\ 000}{(97.8)^2} = 29.9 \text{ ksi} \quad \text{(Eq. E3-4)}$$

Nominal Compressive Strength

$$F_{cr} = \left(0.658^{F_y/F_e}\right) F_y = 0.804\left[0.658^{35/29.9}\right]35 = 21.4 \text{ ksi} \quad \text{(Eq E7-2)}$$

$$P_n = F_{cr} A_g = 21.4(11.5) = 246 \text{ kips} \quad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(246) = 221 \ kips$

ASD: $P_n / \Omega_c = {}^{246}/_{1.67} = 147 \ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	221	219	negligible	
P_n / Ω_c (kips) [ASD]	147	145	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Pipe E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
```

```
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS100
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -35
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -105
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 60 ALL
FYLD 35 ALL
LY 180 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 60 ALL
FYLD 35 ALL
LY 180 ALL
METHOD ASD
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
       1 ST   PIPS100                   (AISC SECTIONS)
                            PASS   Eq. H1-1a            0.961         3
                      210.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.349  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.961(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.100E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   5.926E+00     Ayy:   5.926E+00     Cw:  0.000E+00              |
| Szz:   2.809E+01     Syy:   2.809E+01                                 |
| Izz:   1.510E+02     Iyy:   1.510E+02     Ix:  3.020E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          35.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       360.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    31.62     N/A     91.14    T.B4.1(a)-9 |
|                Non-Slender    31.62     N/A     91.14    T.B4.1(a)-9 |
| Flexure     :  Compact        31.62    58.00   256.86    T.B4.1(b)-20 |
|                Compact        31.62    58.00   256.86    T.B4.1(b)-20 |
      STAAD SPACE                                             -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   3.62E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   5.18E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    2.10E+02   2.19E+02   0.961      Eq. E3-1      3     0.00 |
| Min Buck    2.10E+02   3.19E+02   0.658      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.15E+01    99.35   2.11E+01   2.90E+01   2.43E+02        |
| Min Buck    1.15E+01    49.67   3.08E+01   1.16E+02   3.55E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.09E+02   0.000      Eq. G6-1      3     0.00 |
| Local-Y     0.00E+00   1.09E+02   0.000      Eq. G6-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.15E+01     0.00     0.00     0.00   1.21E+02            |
| Local-Y     1.15E+01     0.00     0.00     0.00   1.21E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.09E+03   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.10E+01   1.22E+03                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.16E+03   0.000      Eq. F8-1      3     0.00 |
| Minor       0.00E+00   1.16E+03   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.29E+03   0.00E+00                                      |
| Minor       1.29E+03   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.961      Eq. H1-1a     3     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         1.16E+03   0.00E+00   2.19E+02                  |
|                      1.16E+03   0.00E+00   2.10E+02                  |
| Flexure Tens         1.16E+03   0.00E+00   3.62E+02                  |
|                      1.16E+03   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    66. LOAD LIST 4
    67. PARAMETER 2
    68. CODE AISC UNIFIED 2010
    69. FU 60 ALL
    70. FYLD 35 ALL
    71. LY 180 ALL
```

```
    72. METHOD ASD
    73. TRACK 2 ALL
    74. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   PIPS100                  (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.963         4
                      140.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.349  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.963(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   5.926E+00     Ayy:   5.926E+00     Cw:  0.000E+00             |
| Szz:   2.809E+01     Syy:   2.809E+01                                 |
| Izz:   1.510E+02     Iyy:   1.510E+02     Ix:  3.020E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          35.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       360.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    31.62     N/A     91.14    T.B4.1(a)-9  |
|                Non-Slender    31.62     N/A     91.14    T.B4.1(a)-9  |
| Flexure     :  Compact        31.62    58.00   256.86    T.B4.1(b)-20 |
|                Compact        31.62    58.00   256.86    T.B4.1(b)-20 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL TENSION                                              |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.41E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   3.45E+02   0.000      Eq. D2-2      4     0.00 |
|----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                          |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    1.40E+02   1.45E+02   0.963      Eq. E3-1      4     0.00 |
| Min Buck    1.40E+02   2.12E+02   0.659      Eq. E3-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    1.15E+01    99.35   2.11E+01   2.90E+01   2.43E+02       |
| Min Buck    1.15E+01    49.67   3.08E+01   1.16E+02   3.55E+02       |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   7.23E+01   0.000      Eq. G6-1      4     0.00 |
| Local-Y     0.00E+00   7.23E+01   0.000      Eq. G6-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw        Cv        Kv       h/tw      Vn            |
| Local-Z     1.15E+01     0.00      0.00      0.00   1.21E+02          |
| Local-Y     1.15E+01     0.00      0.00      0.00   1.21E+02          |
|----------------------------------------------------------------------|
| CHECK FOR TORSION                                                    |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   7.28E+02   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                           |
|             2.10E+01   1.22E+03                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   7.73E+02   0.000      Eq. F8-1      4     0.00 |
| Minor       0.00E+00   7.73E+02   0.000      Eq. F8-1      4     0.00 |
| Intermediate   Mn        My                                          |
| Major       1.29E+03   0.00E+00                                      |
| Minor       1.29E+03   0.00E+00                                      |
    STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.963      Eq. H1-1a     4     0.00            |
| Flexure Tens          0.000      Eq. H1-1b     4     0.00            |
| Intermediate         Mcx /      Mrx /      Pc /                     |
|                      Mcy        Mry        Pr                       |
| Flexure Comp         7.73E+02   0.00E+00   1.45E+02                  |
|                      7.73E+02   0.00E+00   1.40E+02                  |
| Flexure Tens         7.73E+02   0.00E+00   2.41E+02                  |
```

```
|                       7.73E+02   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 Built Up I E 12

Verify the compressive strength of a built-up I section with different flange widths per the LRFD and ADS methods of the AISC 360-10 codes.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example E.12, pp. E-57 - E-62
3. Seaburg, Paul A. and Charles J. Carter. 1997. Design Guide 9: *Torsional Analysis of Structural Steel members*. AISC: Chicago, IL.

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

From reference (2):

> Compute the available strength of a built-up compression member with a length of 14 ft. The ends are pinned. The outside flange is PL3/4-in.×5-in., the inside flange is PL3/4-in.×8-in., and the web is PL3/8-in.×10 1/2-in. Material is ASTM A572 Grade 50.

Validation

Material Properties

$$F_y = 50 \text{ ksi}$$

$$F_u = 65 \text{ ksi}$$

Section Properties

Built-up section properties (ignoring fillet welds):

$$A_g = 8.0(0.75) + 10.5(0.375) + 5.0(0.75) = 13.7 \text{ in}^2$$

$$\bar{y} = \frac{\Sigma A_i y_i}{\Sigma A_i} = \frac{6.0(11.6) + 3.94(6.0) + 3.75(0.375)}{6.0 + 3.94 + 3.75} = 6.91 \text{ in}$$

where $\bar{y}$ is measured from the bottom of the outside flange.

$$I_x = \left[8.0(0.75)(4.72)^2 + \frac{8.0(0.75)^3}{12}\right] + \left[0.375(10.5)(0.910)^2 + \frac{0.375(10.5)^3}{12}\right] + \left[5.0(0.75)(6.54)^2 + \frac{5.0(0.75)^3}{12}\right] =$$

$$I_y = \frac{0.75 \times 8.0^3}{12} + \frac{10.5 \times 0.375^3}{12} + \frac{0.75 \times 5.0^3}{12} = 39.9 \text{ in}^4$$

$$r_x = \sqrt{\frac{I_x}{A}} = \sqrt{\frac{334}{13.7}} = 4.94 \text{ in}$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{39.9}{13.7}} = 1.71 \text{ in}$$

From Eq. 3.4 in Reference 3:

$$J = \Sigma\frac{bt^3}{3} = \frac{8.0(0.75)^3 + 10.5(0.375)^3 + 5.0(0.75)^3}{3} = 2.01 \text{ in}^4$$

Distance between flange centroids:

$$h_0 = 12.0 - \frac{0.75}{2} - \frac{0.75}{2} = 11.3 \text{ in}$$

$$C_w = \frac{t_f h_o^2}{12}\left(\frac{b_1^3 b_2^3}{b_1^3 + b_2^3}\right) = \frac{0.75(11.3)^2}{12}\left[\frac{(8.0)^3(5.0)^3}{(8.0)^3 + (5.0)^3}\right] = 802 \text{ in}^6$$

Due to symmetry about the y-axis, the shear center lies on the y axis ($x_0 = 0$). The distance from the centroid to the shear center can be calculated by finding the shear center from outside flange plus one-half the outside flange and then subtracting the centroid distance from that same reference point.

$$y_0 = \left(e + \frac{t_f}{2}\right) - \bar{y} = h_0\left[\frac{b_1^3}{b_1^3 + b_2^3}\right] + \frac{t_f}{2} - \bar{y} = 11.3\left[\frac{(8.0)^3}{(8.0)^3 + (5.0)^3}\right] + \frac{0.75}{2} - 6.91 = 2.55 \text{ in}$$

$$\bar{r}_0^2 = x_0^2 + y_0^2 + \frac{I_x + I_y}{A_g} = 0 + (2.55)^2 + \frac{344 + 39.9}{13.7} = 33.8 \text{ in}^2$$

$$H = 1 - \frac{x_0^2 + y_0^2}{\bar{r}_0^2} = 1 - \frac{0 + (2.55)^2}{33.8} = 0.808$$

<u>Elastic Flexural Buckling Stress</u>

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{K_y L}{r_y} = \frac{1.0(14.0)(12)}{1.71} = 98.2$$

$$F_{ey} = \frac{\pi^2 E}{(KL\ /r_y)^2} = \frac{\pi^2 29,\ 000}{(98.2)^2} = 29.7 \text{ ksi} \quad \text{(Eq. E4-8)}$$

$$F_{ez} = \left[\frac{\pi^2 E C_w}{(K_z L\ )^2} + GJ\right]\left(\frac{1}{A_g \bar{r}_0^2}\right) = \left[\frac{\pi^2(29,\ 000)(802)}{(1.0 \times 14.0 \times 12^2} + 11,\ 200(2.01)\right]\left(\frac{1}{13.7 \times 33.8}\right) = 66.2 \text{ ksi} \quad \text{(Eq. E4-9)}$$

$$F_e = \left(\frac{F_{ey} + F_{ez}}{2H}\right)\left[1 - \sqrt{1 - \frac{4F_{ey}F_{ez}H}{(F_{ey} + F_{ez})^2}}\right] = \left(\frac{29.7 + 66.2}{2 \times 0.808}\right)\left[1 - \sqrt{1 - \frac{4(29.7)(66.2)(0.808)}{(29.7 + 66.2)^2}}\right] = 26.4 \text{ ksi} \quad \text{(Eq. E4-5)}$$

<u>Nominal Compressive Strength</u>

$$\frac{F_y}{2.25} = \frac{50}{2.25} = 22.2 \text{ ksi} < 26.4 \text{ ksi}$$

$$F_{cr} = \left(0.658^{F_y/F_e}\right)F_y = \left[0.658^{(50)/(26.4)}\right]50 = 22.6 \text{ ksi} \quad \text{(Eq E7-2)}$$

$$P_n = F_{cr}A_g = 22.6(13.7) = 310 \text{ kips} \quad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(310) = 279 \text{ kips}$

ASD: $P_n / \Omega_c = {}^{310}\!/_{1.67} = 186 \text{ kips}$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	279	278	negligible	
P_n/Ω_c (kips) [ASD]	186	185	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\US\AISC\AISC 360-10 Built Up I E 12.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 14 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
I
13.6875 12 0.375 8 0.75 333.426 39.8586 2.0127 4.5 6.5 5 0.75
END
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 I
CONSTANTS
MATERIAL STEEL_36_KSI ALL
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -40
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
STP 2 ALL
FU 65 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
STP 2 ALL
FU 65 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT MEMBER PROPERTIES
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                      (UPT)
                           PASS   Eq. H1-1a           0.316        3
                       88.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
```

```
| Actual Slenderness Ratio    :     98.449  L/C  :     3                   |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      3    Ratio     :     0.316(PASS)                   |
|          Loc  :    0.00   Condition :   Eq. H1-1a                        |
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   8.800E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   6.500E+00     Ayy:   4.500E+00     Cw:  7.947E+02                 |
| Szz:   4.815E+01     Syy:   9.965E+00                                   |
| Izz:   3.334E+02     Iyy:   3.986E+01     Ix:  2.013E+00                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:          50.000   Fu:          65.000                              |
|--------------------------------------------------------------------------|
| Actual Member Length:       168.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                 |
| Compression :  Non-Slender     5.33     N/A     13.40    T.B4.1(a)-2   |
|                Non-Slender    28.00     N/A     35.88    T.B4.1(a)-5   |
| Flexure     :  Compact         5.33     9.15    23.84    T.B4.1(b)-11  |
|                Compact        23.17   127.82   137.27    T.B4.1(b)-16  |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   6.16E+02   0.000      Eq. D2-1      3     0.00   |
| Rupture     0.00E+00   6.67E+02   0.000      Eq. D2-2      3     0.00   |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    8.80E+01   5.66E+02   0.156      Eq. E3-1      3     0.00   |
| Min Buck    8.80E+01   3.03E+02   0.290      Eq. E3-1      3     0.00   |
| Flexural                                                                 |
| Tor Buck    8.80E+01   2.78E+02   0.316      Eq. E4-1      3     0.00   |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    1.37E+01    34.04   4.59E+01   2.47E+02   6.29E+02          |
| Min Buck    1.37E+01    98.45   2.46E+01   2.95E+01   3.37E+02          |
| Flexural       Ag        Fcr        Pn                                   |
| Tor Buck    1.37E+01   2.26E+01   3.09E+02                              |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
```

```
| Local-Z     0.00E+00   1.76E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.22E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     6.50E+00     1.00     1.20    10.67   1.95E+02            |
| Local-Y     4.50E+00     1.00     5.00    28.00   1.35E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.78E+03   0.000      Sec. F4.2     3     0.00 |
| Minor       0.00E+00   7.17E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.09E+03   0.00E+00                                       |
| Minor       7.97E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.30E+03   0.000      Eq. F4-2      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       2.56E+03   0.00E+00    1.00    60.83   343.65   168.00   |
     STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.78E+03   0.000      Eq. F4-1      3     0.00 |
| Intermediate   Mn         Rpc                                         |
|             3.09E+03   1.28E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.316      Eq. H1-1a     3     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         2.30E+03   0.00E+00   2.78E+02                  |
|                      7.17E+02   0.00E+00   8.80E+01                  |
| Flexure Tens         2.30E+03   0.00E+00   6.16E+02                  |
|                      7.17E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    67. LOAD LIST 4
    68. PARAMETER 2
    69. CODE AISC UNIFIED 2010
    70. METHOD ASD
    71. STP 2 ALL
    72. FU 65 ALL
    73. FYLD 50 ALL
    74. TRACK 2 ALL
    75. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
```

```
                       STAADD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   I                         (UPT)
                           PASS   Eq. H1-1a            0.324        4
                       60.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     98.449  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.324(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   6.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.500E+00     Ayy:   4.500E+00     Cw:  7.947E+02              |
| Szz:   4.815E+01     Syy:   9.965E+00                                 |
| Izz:   3.334E+02     Iyy:   3.986E+01     Ix:  2.013E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       168.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.33     N/A     13.40    T.B4.1(a)-2  |
|                Non-Slender    28.00     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.33     9.15    23.84    T.B4.1(b)-11 |
|                Compact        23.17   127.82   137.27    T.B4.1(b)-16 |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.10E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.45E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    6.00E+01   3.77E+02   0.159      Eq. E3-1      4     0.00 |
| Min Buck    6.00E+01   2.02E+02   0.297      Eq. E3-1      4     0.00 |
| Flexural                                                              |
```

```
| Tor Buck    6.00E+01   1.85E+02   0.324      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.37E+01    34.04   4.59E+01   2.47E+02   6.29E+02        |
| Min Buck    1.37E+01    98.45   2.46E+01   2.95E+01   3.37E+02        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    1.37E+01   2.26E+01   3.09E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.17E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   8.08E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv      h/tw      Vn             |
| Local-Z     6.50E+00     1.00      1.20    10.67   1.95E+02          |
| Local-Y     4.50E+00     1.00      5.00    28.00   1.35E+02          |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.85E+03   0.000      Sec. F4.2     4     0.00 |
| Minor       0.00E+00   4.77E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.09E+03   0.00E+00                                       |
| Minor       7.97E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.53E+03   0.000      Eq. F4-2      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       2.56E+03   0.00E+00    1.00    60.83   343.65   168.00    |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             0.00E+00   1.85E+03   0.000      Eq. F4-1      4     0.00 |
| Intermediate   Mn         Rpc                                         |
|             3.09E+03   1.28E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.324      Eq. H1-1a     4     0.00           |
| Flexure Tens            0.000      Eq. H1-1b     4     0.00           |
| Intermediate           Mcx /      Mrx /      Pc /                    |
|                        Mcy        Mry        Pr                      |
| Flexure Comp           1.53E+03   0.00E+00   1.85E+02                 |
|                        4.77E+02   0.00E+00   6.00E+01                 |
| Flexure Tens           1.53E+03   0.00E+00   4.10E+02                 |
|                        4.77E+02   0.00E+00   0.00E+00                 |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 - H.1B

Reference

Design Examples, Version 14.0, American Institute of Steel Construction, 2011, page H-4.

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.5 Design for Combined Forces* (on page 1096)

Objective

From the reference:

> Using AISC Manual tables to determine the available compressive and flexural strengths, determine if an ASTM A992 W14×99 has sufficient available strength to support the axial forces and moments listed as follows, obtained from a second-order analysis that includes P-δ effects. The unbraced length is 14 ft and the member has pinned ends. KLx = KLy = Lb = 14.0 ft.

Refer to the reference for calculations.

Note: A static analysis is used in STAAD.Pro as the reference supplies only the resulting forces from an assumed P-δ analysis.

Comparison

Table 751: Comparison of results

Result Type		Reference	STAAD.Pro	Difference
LRFD	Interaction Ratio	0.928	0.929	none
	ϕM_{nx} (kip-ft)	642	642	none
	ϕM_{ny} (kip-ft)	311	311	none
	$\phi_c P_n$ (kips)	1,130	1,130	none
ASD	Interaction Ratio	0.931	0.932	negligible
	M_{nx}/Ω (kip-ft)	428	427	negligible
	M_{ny}/Ω (kip-ft)	207	207	none
	Compression (kips)	750	750	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 - H.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Nov-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 14 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X99
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -400 MX 80 MZ 250
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -267 MX 53.3 MZ 167
PERFORM ANALYSIS
LOAD LIST 1
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 2
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
```

```
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X99                   (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.929         1
                      400.00 C         80.00        250.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     45.200  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.929(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   4.000E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   8.000E+01     Mz:   2.500E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.278E+01     Ayy:   6.887E+00     Cw:  1.810E+04              |
| Szz:   1.563E+02     Syy:   5.507E+01                                 |
| Izz:   1.110E+03     Iyy:   4.020E+02     Ix:  5.370E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        14.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.36     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    23.59     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.36     9.15    24.08    T.B4.1(b)-10 |
|                Compact        23.59    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                         -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
```

```
| Yield       0.00E+00   1.31E+03   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.42E+03   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    4.00E+02   1.24E+03   0.322      Eq. E3-1      1     0.00 |
| Min Buck    4.00E+02   1.13E+03   0.355      Eq. E3-1      1     0.00 |
| Flexural                                                              |
| Tor Buck    4.00E+02   1.15E+03   0.348      Eq. E4-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    2.02E-01    27.20   6.82E+03   5.57E+04   1.38E+03        |
| Min Buck    2.02E-01    45.20   6.20E+03   2.02E+04   1.25E+03        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    2.02E-01   6.32E+03   1.28E+03                           |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   6.15E+02   0.000      Eq. G2-1      1     0.00 |
| Local-Y     0.00E+00   2.07E+02   0.000      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.58E-01     1.00     1.20     9.36   6.83E+02            |
| Local-Y     4.78E-02     1.00     0.00    23.59   2.07E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.50E+02   6.49E+02   0.385      Eq. F2-1      1     0.00 |
| Minor       8.00E+01   3.14E+02   0.255      Eq. F6-1      1     0.00 |
| Intermediate   Mn        My                                           |
| Major       7.21E+02   0.00E+00                                      |
| Minor       3.48E+02   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.50E+02   6.42E+02   0.389      Eq. F2-2      1     0.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb     |
| Major       7.14E+02   0.00E+00    1.00    13.13    45.47    14.00   |
    STAAD SPACE                                            -- PAGE NO.
5
                     STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.50E+02   6.45E+02   0.387      Eq. F3-1      1     0.00 |
| Minor       8.00E+01   3.11E+02   0.257      Eq. F6-2      1     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       7.17E+02   0.00E+00                                      |
| Minor       3.46E+02   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
```

```
|                        RATIO      CRITERIA    L/C     LOC                |
| Flexure Comp           0.929      Eq. H1-1a     1     0.00               |
| Flexure Tens           0.646      Eq. H1-1b     1     0.00               |
| Intermediate          Mcx /      Mrx /      Pc /                         |
|                       Mcy        Mry        Pr                           |
| Flexure Comp          6.42E+02  -2.50E+02   1.13E+03                     |
|                       3.11E+02   8.00E+01   4.00E+02                     |
| Flexure Tens          6.42E+02  -2.50E+02   1.31E+03                     |
|                       3.11E+02   8.00E+01   0.00E+00                     |
|--------------------------------------------------------------------------|
    44. LOAD LIST 2
    45. PARAMETER 2
    46. CODE AISC UNIFIED 2010
    47. METHOD ASD
    48. FYLD 7200 ALL
    49. FU 9360 ALL
    50. LZ 14 ALL
    51. LY 14 ALL
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                 -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   W14X99                  (AISC SECTIONS)
                          PASS   Eq. H1-1a           0.932         2
                      267.00 C         53.30        167.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     45.200  L/C  :     2                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      2    Ratio     :     0.932(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.670E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   5.330E+01     Mz:   1.670E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.278E+01     Ayy:   6.887E+00     Cw:  1.810E+04              |
| Szz:   1.563E+02     Syy:   5.507E+01                                 |
| Izz:   1.110E+03     Iyy:   4.020E+02     Ix:  5.370E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        14.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
```

```
|                STIFFENED                                                   |
| Compression :  Non-Slender     9.36     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    23.59     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.36     9.15    24.08    T.B4.1(b)-10 |
|                Compact        23.59    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   8.71E+02   0.000      Eq. D2-1      2     0.00 |
| Rupture     0.00E+00   9.46E+02   0.000      Eq. D2-2      2     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    2.67E+02   8.25E+02   0.323      Eq. E3-1      2     0.00 |
| Min Buck    2.67E+02   7.50E+02   0.356      Eq. E3-1      2     0.00 |
| Flexural                                                              |
| Tor Buck    2.67E+02   7.65E+02   0.349      Eq. E4-1      2     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    2.02E-01    27.20   6.82E+03   5.57E+04   1.38E+03       |
| Min Buck    2.02E-01    45.20   6.20E+03   2.02E+04   1.25E+03       |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    2.02E-01   6.32E+03   1.28E+03                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   4.09E+02   0.000      Eq. G2-1      2     0.00 |
| Local-Y     0.00E+00   1.38E+02   0.000      Eq. G2-1      2     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.58E-01     1.00     1.20     9.36   6.83E+02           |
| Local-Y     4.78E-02     1.00     0.00    23.59   2.07E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -1.67E+02   4.32E+02   0.387      Eq. F2-1      2     0.00 |
| Minor       5.33E+01   2.09E+02   0.256      Eq. F6-1      2     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.21E+02   0.00E+00                                       |
| Minor       3.48E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -1.67E+02   4.27E+02   0.391      Eq. F2-2      2     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb   |
| Major       7.14E+02   0.00E+00    1.00    13.13    45.47    14.00   |
     STAAD SPACE                                              -- PAGE NO.
8
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -1.67E+02   4.29E+02   0.389      Eq. F3-1      2     0.00 |
| Minor       5.33E+01   2.07E+02   0.257      Eq. F6-2      2     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       7.17E+02   0.00E+00                                       |
| Minor       3.46E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.932      Eq. H1-1a     2     0.00             |
| Flexure Tens          0.648      Eq. H1-1b     2     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         4.27E+02  -1.67E+02   7.50E+02                   |
|                      2.07E+02   5.33E+01   2.67E+02                   |
| Flexure Tens         4.27E+02  -1.67E+02   8.71E+02                   |
|                      2.07E+02   5.33E+01   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 W Tens BM H3

Verify the interaction checks due to tension and bending moments in both axes of a W section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example H.3, pp. H-8 - H-11

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.5 Design for Combined Forces* (on page 1096)

Details

From reference (2):

> Select an ASTM A992 W-shape with a 14-in. nominal depth to carry forces of 29.0 kips from dead load and 87.0 kips from live load in axial tension, as well as the following moments due to uniformly distributed loads:
>
> MxD = 32.0 kip-ft
> MxL = 96.0 kip-ft

MyD = 11.3 kip-ft
MyL = 33.8 kip-ft

The unbraced length is 30.0 ft and the ends are pinned. Assume the connections are made with no holes.

See example H.3 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Interaction ratio per H1-1b [LRFD]	0.873	0.874	negligible	
Interaction ratio per H1-1b [ASD]	0.874	0.876	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W Tens BM H3.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X82
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MY MZ
1 PINNED
LOAD 1 LOADTYPE None  TITLE FORCE
JOINT LOAD
2 FX 174
1 FX -174
```

```
LOAD 2 LOADTYPE None  TITLE MOMENT MAJOR
MEMBER LOAD
1 UNI GY -1.70668
LOAD 3 LOADTYPE None  TITLE MOMENT MINOR
MEMBER LOAD
1 UNI GZ 0.6009
UNIT INCHES KIP
LOAD 4 LOADTYPE None  TITLE FORCE2
JOINT LOAD
1 FX 116
2 FX 116
LOAD 5 LOADTYPE None  TITLE MOMENT MAJOR 2
MEMBER LOAD
1 UNI GY -0.0948
LOAD 6 LOADTYPE None  TITLE MOMENT MINOR 2
MEMBER LOAD
1 UNI GZ 0.0334
UNIT FEET KIP
LOAD COMB 7 LRFD
1 1.0 2 1.0 3 1.0
LOAD COMB 8 ASD
4 1.0 5 1.0 6 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 7
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 50 ALL
FU 65 ALL
CB 1.41 ALL
TRACK 2 ALL
UNIT FEET KIP
CHECK CODE ALL
UNIT INCHES KIP
LOAD LIST 8
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 50 ALL
FU 65 ALL
CB 1.41 ALL
TRACK 2 ALL
UNIT FEET KIP
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X82                    (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.874         7
```

```
                    174.00 T        -67.60        -192.00       15.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    144.970  L/C  :     7                |
| Allowable Slenderness Ratio :    300.000  LOC  :    15.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      7    Ratio     :     0.874(PASS)                 |
|          Loc  :   15.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.740E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:  -6.760E+01     Mz:  -1.920E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.727E+01     Ayy:   7.293E+00     Cw:  6.688E+03              |
| Szz:   1.232E+02     Syy:   2.931E+01                                 |
| Izz:   8.810E+02     Iyy:   1.480E+02     Ix:  5.070E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.91     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    22.35     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.91     9.15    24.08    T.B4.1(b)-10 |
|                Compact        22.35    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                             -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       1.74E+02   1.08E+03   0.161      Eq. D2-1      7     0.00 |
| Rupture     1.74E+02   1.17E+03   0.149      Eq. D2-2      7     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   8.34E+02   0.000      Eq. E3-1      7     0.00 |
| Min Buck    0.00E+00   2.58E+02   0.000      Eq. E3-1      7     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   7.99E+02   0.000      Eq. E4-1      7     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.67E-01    59.42   5.56E+03   1.17E+04   9.27E+02        |
| Min Buck    1.67E-01   144.97   1.72E+03   1.96E+03   2.87E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.67E-01   5.32E+03   8.87E+02                            |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     9.01E+00   4.66E+02   0.019      Eq. G2-1      7     0.00 |
| Local-Y     2.56E+01   2.19E+02   0.117      Eq. G2-1      7     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn              |
| Local-Z     1.20E-01     1.00     1.20     5.91   5.18E+02           |
| Local-Y     5.06E-02     1.00     0.00    22.35   2.19E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.92E+02   5.21E+02   0.368      Eq. F2-1      7    15.00 |
| Minor      -6.76E+01   1.68E+02   0.402      Eq. F6-1      7    15.00 |
| Intermediate   Mn        My                                           |
| Major       5.79E+02   0.00E+00                                       |
| Minor       1.87E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.92E+02   4.91E+02   0.391      Eq. F2-2      7    15.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb     |
| Major       5.45E+02   0.00E+00    1.41     8.77    33.01    30.00   |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp            0.794      Eq. H1-1b     7    15.00           |
| Flexure Tens            0.874      Eq. H1-1b     7    15.00           |
| Intermediate           Mcx /      Mrx /      Pc /                    |
|                        Mcy        Mry        Pr                      |
| Flexure Comp           4.91E+02   1.92E+02   2.58E+02                |
|                        1.68E+02  -6.76E+01   0.00E+00                |
| Flexure Tens           4.91E+02   1.92E+02   1.08E+03                |
|                        1.68E+02  -6.76E+01   1.74E+02                |
|-----------------------------------------------------------------------|
    66. UNIT INCHES KIP
    67. LOAD LIST 8
    68. PARAMETER 2
    69. CODE AISC UNIFIED 2010
    70. METHOD ASD
    71. FYLD 50 ALL
    72. FU 65 ALL
    73. CB 1.41 ALL
    74. TRACK 2 ALL
    75. UNIT FEET KIP
    76. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X82                   (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.876         8
                      116.00 T        -45.09        -127.98       15.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    144.970  L/C  :     8                |
| Allowable Slenderness Ratio :    300.000  LOC  :    15.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      8    Ratio     :     0.876(PASS)                 |
|          Loc  :   15.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.160E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:  -4.509E+01     Mz:  -1.280E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.727E+01     Ayy:   7.293E+00     Cw:  6.688E+03             |
| Szz:   1.232E+02     Syy:   2.931E+01                                 |
| Izz:   8.810E+02     Iyy:   1.480E+02     Ix:  5.070E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.91     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    22.35     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.91     9.15    24.08    T.B4.1(b)-10 |
|                Compact        22.35    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       1.16E+02   7.19E+02   0.161      Eq. D2-1      8     0.00 |
| Rupture     1.16E+02   7.80E+02   0.149      Eq. D2-2      8     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   5.55E+02   0.000      Eq. E3-1      8     0.00 |
| Min Buck    0.00E+00   1.72E+02   0.000      Eq. E3-1      8     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   5.31E+02   0.000      Eq. E4-1      8     0.00 |
| Intermediate                                                          |
```

```
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    1.67E-01    59.42   5.56E+03   1.17E+04   9.27E+02      |
| Min Buck    1.67E-01   144.97   1.72E+03   1.96E+03   2.87E+02      |
| Flexural       Ag       Fcr        Pn                               |
| Tor Buck    1.67E-01   5.32E+03   8.87E+02                          |
|---------------------------------------------------------------------|
| CHECK FOR SHEAR                                                     |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     6.01E+00   3.10E+02   0.019      Eq. G2-1      8     0.00 |
| Local-Y     1.71E+01   1.46E+02   0.117      Eq. G2-1      8     0.00 |
| Intermediate                                                        |
| Results        Aw         Cv       Kv      h/tw      Vn            |
| Local-Z     1.20E-01     1.00     1.20     5.91   5.18E+02          |
| Local-Y     5.06E-02     1.00     0.00    22.35   2.19E+02          |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.28E+02   3.47E+02   0.369      Eq. F2-1      8    15.00 |
| Minor      -4.51E+01   1.12E+02   0.403      Eq. F6-1      8    15.00 |
| Intermediate   Mn         My                                        |
| Major       5.79E+02   0.00E+00                                     |
| Minor       1.87E+02   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                        |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.28E+02   3.26E+02   0.392      Eq. F2-2      8    15.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       5.45E+02   0.00E+00    1.41     8.77    33.01    30.00   |
     STAAD SPACE                                          -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                       RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp          0.795      Eq. H1-1b     8    15.00           |
| Flexure Tens          0.876      Eq. H1-1b     8    15.00           |
| Intermediate         Mcx /      Mrx /      Pc /                    |
|                      Mcy        Mry        Pr                      |
| Flexure Comp         3.26E+02   1.28E+02   1.72E+02                |
|                      1.12E+02  -4.51E+01   0.00E+00                |
| Flexure Tens         3.26E+02   1.28E+02   7.19E+02                |
|                      1.12E+02  -4.51E+01   1.16E+02                |
|---------------------------------------------------------------------|
```

V. AISC 360-10 HSST Torsional Strength H.5A

Verify the torsional strength of an rectangular HSS section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example H. 5A, p. H-16

Related Links

- *D1.A.5.6 Design for Torsion* (on page 1096)

Details

From reference (2):

Determine the available torsional strength of an ASTM A500 Grade B HSS6×4×1/4.

Validation

$h/t > b/t$, so h/t governs.

$$\frac{h}{t} = 22.8 < 2.45\sqrt{\frac{E}{F_y}} = 2.45\sqrt{\frac{29,\ 000}{46}} = 61.5$$

$$F_{cr} = 0.6F_y = 0.6(46) = 27.6\ \text{ksi} \qquad \text{(Eq. H3-3)}$$

Nominal Torsional Strength

$$T_n = F_{cr}C = 27.6(10.1) = 279\ \text{in}\cdot\text{k} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_T T_n = 0.90(279) = 251\ in\cdot k$

ASD: $T_n / \Omega_T = {}^{279}\!/_{1.67} = 167\ in\cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_G T_n$ (in·k) [LRFD]	251	251	none	
G_n/Ω_T (in·k) [ASD]	167	167	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST Torsional Strength H.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
```

```
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
UNIT INCHES KIP
MEMBER LOAD
1 UMOM GX -1.5
UNIT FEET KIP
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 46 ALL
FU 58 ALL
TORSION 1 ALL
TND 3 ALL
TRACK 2 ALL
CHECK CODE ALL
UNIT INCHES KIP
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 46 ALL
FU 58 ALL
TORSION 1 ALL
TND 3 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   HSST6X4X0.25             (AISC SECTIONS)
                          PASS   Eq. H3-1             0.287         1
                       0.00           0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.875  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.287(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   7.200E+01     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.538E+00     Ayy:   2.470E+00     Cw:  0.000E+00             |
| Szz:   6.967E+00     Syy:   5.550E+00                                 |
| Izz:   2.090E+01     Iyy:   1.110E+01     Ix:  2.360E+01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    22.75     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact        14.17    28.12    35.15    T.B4.1(b)-17 |
|                Compact        22.75    60.76   143.12    T.B4.1(b)-19 |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      0.00E+00   1.78E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture    0.00E+00   1.87E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   0.00E+00   1.72E+02   0.000      Eq. E3-1      1     0.00 |
| Min Buck   0.00E+00   1.68E+02   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck   4.30E+00    21.77   4.46E+01   6.04E+02   1.92E+02         |
| Min Buck   4.30E+00    29.88   4.33E+01   3.21E+02   1.86E+02         |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
```

```
|                                                                           |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z     0.00E+00   3.82E+01   0.000      Eq. G2-1      1     0.00    |
| Local-Y     0.00E+00   6.14E+01   0.000      Eq. G2-1      1     0.00    |
| Intermediate                                                              |
| Results        Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z     1.54E+00     1.00     5.00    14.17   4.25E+01               |
| Local-Y     2.47E+00     1.00     5.00    22.75   6.82E+01               |
|---------------------------------------------------------------------------|
| CHECK FOR TORSION                                                         |
|                                                                           |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
|             7.20E+01   2.51E+02   0.287      Eq. H3-1      1     0.00    |
| Intermediate Fcr        Tn                                                |
|             2.76E+01   2.79E+02                                          |
|---------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                                |
|                                                                           |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major       0.00E+00   3.53E+02   0.000      Eq. F7-1      1     0.00    |
| Minor       0.00E+00   2.67E+02   0.000      Eq. F7-1      1     0.00    |
| Intermediate   Mn         My                                              |
| Major       3.92E+02   0.00E+00                                           |
| Minor       2.97E+02   0.00E+00                                           |
    STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                   |
|                      RATIO      CRITERIA    L/C     LOC                  |
| Flexure Comp         0.000      Eq. H1-1b     1     0.00                 |
| Flexure Tens         0.000      Eq. H1-1b     1     0.00                 |
| Intermediate        Mcx /      Mrx /      Pc /                           |
|                     Mcy        Mry        Pr                              |
| Flexure Comp        3.53E+02   0.00E+00   1.68E+02                       |
|                     2.67E+02   0.00E+00   0.00E+00                       |
| Flexure Tens        3.53E+02   0.00E+00   1.78E+02                       |
|                     2.67E+02   0.00E+00   0.00E+00                       |
|---------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                     |
|                      RATIO      CRITERIA    L/C     LOC                  |
| Flx Tor Comp Shr     0.082      Eq. H3-6      1     0.00                 |
| Flx Tor Tens Shr     0.082      Eq. H3-6      1     0.00                 |
| Intermediate        Mcx /      Mcy /      Mrx /      Mry /               |
|                     Vcx /      Vcy /      Vrx /      Vry /               |
|                     Tc         Tr         Pc         Pr                  |
| Flx Tor Comp Shr    3.53E+02   2.67E+02   0.00E+00   0.00E+00            |
|                     3.82E+01   6.14E+01   0.00E+00   0.00E+00            |
|                     2.51E+02   7.20E+01   1.68E+02   0.00E+00            |
| Flx Tor Tens Shr    3.53E+02   2.67E+02   0.00E+00   0.00E+00            |
|                     3.82E+01   6.14E+01   0.00E+00   0.00E+00            |
|                     2.51E+02   7.20E+01   1.78E+02   0.00E+00            |
|---------------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                               |
```

```
|-------------------------------------------------------------------------------
----|
| Member:     1 Section:       HSST6X4X0.25
AISC                |
| End Condition:  Fix-
Free                                                                  |
| Shear Capacity :  2.4840E+01  Shear Criteria : DG9 Eq.
4.13                 |
| Normal Capacity:  4.1400E+01  Normal Criteria: DG9 Eq.
4.12                 |
|-------------------------------------------------------------------------------
----|
      STAAD SPACE                                                  -- PAGE NO.
6
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Mid Flange:   0.00000E+00 0.0000     1 | Flange Edge:  0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
| FL/WB Junc:   6.43777E+00 0.2592     1
|                                        |
|                                  0.000
|                                        |
| Mid-Web:      6.43777E+00 0.2592     1
|                                        |
|                                  0.000
|                                        |
|-------------------------------------------------------------------------------
----|
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                          |
| Flange Edge:  6.43777E+00     Web Edge:     6.43777E
+00                        |
|-------------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                             |
| Flange Edge:  0.00000E+00  FL/WB Junc:   0.00000E+00  Mid-Web:      0.00000E
+00 |
|-------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                             |
| Flange Edge:  0.00000E
+00                                                       |
|-------------------------------------------------------------------------------
----|
    43. UNIT INCHES KIP
    44. PARAMETER 2
    45. CODE AISC UNIFIED 2010
    46. METHOD ASD
    47. FYLD 46 ALL
    48. FU 58 ALL
    49. TORSION 1 ALL
    50. TND 3 ALL
```

```
    51. TRACK 2 ALL
    52. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.25             (AISC SECTIONS)
                           PASS   Eq. H3-1             0.431         1
                        0.00            0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.875  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.431(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   7.200E+01     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.538E+00     Ayy:   2.470E+00     Cw:  0.000E+00              |
| Szz:   6.967E+00     Syy:   5.550E+00                                 |
| Izz:   2.090E+01     Iyy:   1.110E+01     Ix:  2.360E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:           58.000                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    22.75     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact        14.17    28.12    35.15    T.B4.1(b)-17 |
|                Compact        22.75    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
```

```
||                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.18E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.25E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.15E+02   0.000      Eq. E3-1      1     0.00 |
| Min Buck    0.00E+00   1.12E+02   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.30E+00    21.77   4.46E+01   6.04E+02   1.92E+02        |
| Min Buck    4.30E+00    29.88   4.33E+01   3.21E+02   1.86E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.54E+01   0.000      Eq. G2-1      1     0.00 |
| Local-Y     0.00E+00   4.08E+01   0.000      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.54E+00     1.00     5.00    14.17   4.25E+01            |
| Local-Y     2.47E+00     1.00     5.00    22.75   6.82E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             7.20E+01   1.67E+02   0.431      Eq. H3-1      1     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.76E+01   2.79E+02                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.35E+02   0.000      Eq. F7-1      1     0.00 |
| Minor       0.00E+00   1.78E+02   0.000      Eq. F7-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.92E+02   0.00E+00                                       |
| Minor       2.97E+02   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.000      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         2.35E+02   0.00E+00   1.12E+02                  |
|                      1.78E+02   0.00E+00   0.00E+00                  |
| Flexure Tens         2.35E+02   0.00E+00   1.18E+02                  |
|                      1.78E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
```

```
|                       RATIO      CRITERIA    L/C      LOC                |
| Flx Tor Comp Shr      0.186      Eq. H3-6      1     0.00                |
| Flx Tor Tens Shr      0.186      Eq. H3-6      1     0.00                |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /              |
|                      Vcx /      Vcy /      Vrx /      Vry /              |
|                      Tc         Tr         Pc         Pr                 |
| Flx Tor Comp Shr     2.35E+02   1.78E+02   0.00E+00   0.00E+00           |
|                      2.54E+01   4.08E+01   0.00E+00   0.00E+00           |
|                      1.67E+02   7.20E+01   1.12E+02   0.00E+00           |
| Flx Tor Tens Shr     2.35E+02   1.78E+02   0.00E+00   0.00E+00           |
|                      2.54E+01   4.08E+01   0.00E+00   0.00E+00           |
|                      1.67E+02   7.20E+01   1.18E+02   0.00E+00           |
|-----------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                               |
|-----------------------------------------------------------------------------
----|
| Member:     1 Section:       HSST6X4X0.25
AISC                       |
| End Condition:  Fix-
Free                                                          |
| Shear Capacity :  1.6527E+01  Shear Criteria : DG9 Eq.
4.18                       |
| Normal Capacity:  2.7545E+01  Normal Criteria: DG9 Eq.
4.17                       |
|-----------------------------------------------------------------------------
----|
      STAAD SPACE                                                -- PAGE NO.
10
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Mid Flange:   0.00000E+00 0.0000     1 | Flange Edge:  0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
| FL/WB Junc:   6.43777E+00 0.3895     1
|                                        |
|                                  0.000
|                                        |
| Mid-Web:      6.43777E+00 0.3895     1
|                                        |
|                                  0.000
|                                        |
|-----------------------------------------------------------------------------
----|
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                |
| Flange Edge:  6.43777E+00     Web Edge:     6.43777E
+00                         |
|-----------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                                 |
| Flange Edge:  0.00000E+00  FL/WB Junc:   0.00000E+00  Mid-Web:      0.00000E
```

```
+00 |
|---------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                              |
| Flange Edge:  0.00000E
+00                                                        |
|---------------------------------------------------------------------------
----|
```

V. AISC 360-10 HSSP Torsional Strength H.5B

Verify the torsional strength of an round HSS section per the LRFD and ASD methods of the AISC 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example H. 5B, p. H-17 - H-18

Related Links

- *D1.A.5.6 Design for Torsion* (on page 1096)

Details

From reference (2):

Determine the available torsional strength of an ASTM A500 Grade B HSS5.000×0.250 that is 14 ft long.

Validation

Critical Stress

$$F_{cr} = \max \left| \begin{array}{l} \dfrac{1.23E}{\sqrt{\dfrac{L}{D}}\left(\dfrac{D}{t}\right)^{5/4}} = \dfrac{1.23(29,\ 000)}{\sqrt{\dfrac{14 \times 12}{5.0}}(21.5)^{5/4}} = 133 \text{ ksi} \\ \dfrac{0.60E}{\left(D/t\right)^{3/2}} = \dfrac{0.60(29,\ 000)}{(21.5)^{3/2}} = 175 \text{ ksi} \end{array} \right.$$

The maximum permitted value is $F_{cr} \leq 0.6F_y = 0.6(42) = 25.2\ ksi$ (governs).

Nominal Torsional Strength

$$T_n = F_{cr}C = 25.2(7.95) = 200 \text{ in}\cdot\text{k} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_T T_n = 0.90(200) = 180\ in \cdot k$

ASD: $T_n / \Omega_T = {}^{200}/_{1.67} = 120\ in \cdot k$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_G T_n$ (in·k) [LRFD]	180	180	none	
G_n/Ω_T (in·k) [ASD]	120	120	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSSP Torsional Strength H.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 14 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP5X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
UNIT INCHES KIP
MEMBER LOAD
1 UMOM GX -0.6
UNIT FEET KIP
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 58 ALL
FYLD 42 ALL
TND 3 ALL
TORSION 1 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
```

```
CODE AISC UNIFIED 2010
METHOD ASD
FU 58 ALL
FYLD 42 ALL
TND 3 ALL
TORSION 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSSP5X0.25               (AISC SECTIONS)
                           PASS   Eq. H3-1             0.559        1
                        0.00            0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.547  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.559(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   1.008E+02     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.840E+00     Ayy:   1.840E+00     Cw:  0.000E+00             |
| Szz:   3.976E+00     Syy:   3.976E+00                                 |
| Izz:   9.940E+00     Iyy:   9.940E+00     Ix:  1.988E+01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          42.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       168.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
|                Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
|                Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
```

```
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield      0.00E+00   1.32E+02   0.000      Eq. D2-1      1     0.00 |
 | Rupture    0.00E+00   1.52E+02   0.000      Eq. D2-2      1     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck   0.00E+00   7.18E+01   0.000      Eq. E3-1      1     0.00 |
 | Min Buck   0.00E+00   7.18E+01   0.000      Eq. E3-1      1     0.00 |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck   3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
 | Min Buck   3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Local-Z    0.00E+00   3.96E+01   0.000      Eq. G6-1      1     0.00 |
 | Local-Y    0.00E+00   3.96E+01   0.000      Eq. G6-1      1     0.00 |
 | Intermediate                                                          |
 | Results       Aw        Cv        Kv      h/tw      Vn               |
 | Local-Z    3.49E+00     0.00      0.00     0.00   4.40E+01           |
 | Local-Y    3.49E+00     0.00      0.00     0.00   4.40E+01           |
 |-----------------------------------------------------------------------|
 | CHECK FOR TORSION                                                     |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 |            1.01E+02   1.80E+02   0.559      Eq. H3-1      1     0.00 |
 | Intermediate Fcr        Tn                                            |
 |            2.52E+01   2.00E+02                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Major      0.00E+00   2.00E+02   0.000      Eq. F8-1      1     0.00 |
 | Minor      0.00E+00   2.00E+02   0.000      Eq. F8-1      1     0.00 |
 | Intermediate   Mn        My                                           |
 | Major      2.23E+02   0.00E+00                                        |
 | Minor      2.23E+02   0.00E+00                                        |
     STAAD SPACE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
 |                        RATIO      CRITERIA    L/C     LOC             |
 | Flexure Comp           0.000      Eq. H1-1b     1     0.00            |
 | Flexure Tens           0.000      Eq. H1-1b     1     0.00            |
 | Intermediate         Mcx /      Mrx /      Pc /                       |
 |                      Mcy        Mry        Pr                         |
 | Flexure Comp         2.00E+02   0.00E+00   7.18E+01                   |
 |                      2.00E+02   0.00E+00   0.00E+00                   |
 | Flexure Tens         2.00E+02   0.00E+00   1.32E+02                   |
 |                      2.00E+02   0.00E+00   0.00E+00                   |
```

```
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                  |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flx Tor Comp Shr      0.313      Eq. H3-6      1     0.00            |
| Flx Tor Tens Shr      0.313      Eq. H3-6      1     0.00            |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /          |
|                      Vcx /      Vcy /      Vrx /      Vry /          |
|                      Tc         Tr         Pc         Pr             |
| Flx Tor Comp Shr     2.00E+02   2.00E+02   0.00E+00   0.00E+00       |
|                      3.96E+01   3.96E+01   0.00E+00   0.00E+00       |
|                      1.80E+02   1.01E+02   7.18E+01   0.00E+00       |
| Flx Tor Tens Shr     2.00E+02   2.00E+02   0.00E+00   0.00E+00       |
|                      3.96E+01   3.96E+01   0.00E+00   0.00E+00       |
|                      1.80E+02   1.01E+02   1.32E+02   0.00E+00       |
|------------------------------------------------------------------------|
|------------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                               |
|------------------------------------------------------------------------------
----|
| Member:     1 Section:       HSSP5X0.25
AISC                      |
| End Condition:  Fix-
Free                                                          |
| Shear Capacity :  2.2680E+01  Shear Criteria : DG9 Eq.
4.13                     |
| Normal Capacity:  3.7800E+01  Normal Criteria: DG9 Eq.
4.12                     |
|------------------------------------------------------------------------------
----|
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Uniform:      1.14946E+01 0.5068     1 | Uniform:      0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
|------------------------------------------------------------------------------
----|
      STAAD SPACE                                              -- PAGE NO.
6
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                |
| Uniform:      1.14946E
+01                                                        |
|------------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                             |
| Uniform:      0.00000E
+00                                                        |
|------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                             |
| Uniform:      0.00000E
```

```
+00                                                        |
|-------------------------------------------------------------------------------
----|
    43. PARAMETER 2
    44. CODE AISC UNIFIED 2010
    45. METHOD ASD
    46. FU 58 ALL
    47. FYLD 42 ALL
    48. TND 3 ALL
    49. TORSION 1 ALL
    50. TRACK 2 ALL
    51. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSSP5X0.25              (AISC SECTIONS)
                           PASS   Eq. H3-1             0.840         1
                        0.00            0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.547  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.840(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   1.008E+02     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.840E+00     Ayy:   1.840E+00     Cw:  0.000E+00              |
| Szz:   3.976E+00     Syy:   3.976E+00                                 |
| Izz:   9.940E+00     Iyy:   9.940E+00     Ix:  1.988E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          42.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       168.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
```

```
|               Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
|               Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   8.78E+01   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.01E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   4.78E+01   0.000      Eq. E3-1      1     0.00 |
| Min Buck    0.00E+00   4.78E+01   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
| Min Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.63E+01   0.000      Eq. G6-1      1     0.00 |
| Local-Y     0.00E+00   2.63E+01   0.000      Eq. G6-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     3.49E+00     0.00     0.00     0.00   4.40E+01           |
| Local-Y     3.49E+00     0.00     0.00     0.00   4.40E+01           |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             1.01E+02   1.20E+02   0.840      Eq. H3-1      1     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.52E+01   2.00E+02                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.33E+02   0.000      Eq. F8-1      1     0.00 |
| Minor       0.00E+00   1.33E+02   0.000      Eq. F8-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       2.23E+02   0.00E+00                                      |
| Minor       2.23E+02   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
```

```
| Flexure Comp          0.000      Eq. H1-1b     1     0.00               |
| Flexure Tens          0.000      Eq. H1-1b     1     0.00               |
| Intermediate         Mcx /      Mrx /      Pc /                        |
|                      Mcy        Mry        Pr                          |
| Flexure Comp         1.33E+02   0.00E+00   4.78E+01                    |
|                      1.33E+02   0.00E+00   0.00E+00                    |
| Flexure Tens         1.33E+02   0.00E+00   8.78E+01                    |
|                      1.33E+02   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flx Tor Comp Shr      0.706      Eq. H3-6      1     0.00             |
| Flx Tor Tens Shr      0.706      Eq. H3-6      1     0.00             |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /          |
|                      Vcx /      Vcy /      Vrx /      Vry /          |
|                      Tc         Tr         Pc         Pr             |
| Flx Tor Comp Shr     1.33E+02   1.33E+02   0.00E+00   0.00E+00       |
|                      2.63E+01   2.63E+01   0.00E+00   0.00E+00       |
|                      1.20E+02   1.01E+02   4.78E+01   0.00E+00       |
| Flx Tor Tens Shr     1.33E+02   1.33E+02   0.00E+00   0.00E+00       |
|                      2.63E+01   2.63E+01   0.00E+00   0.00E+00       |
|                      1.20E+02   1.01E+02   8.78E+01   0.00E+00       |
|-----------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                          |
|-----------------------------------------------------------------------------
----|
| Member:     1 Section:       HSSP5X0.25
AISC                       |
| End Condition:  Fix-
Free                                                   |
| Shear Capacity :  1.5090E+01  Shear Criteria : DG9 Eq.
4.18                      |
| Normal Capacity:  2.5150E+01  Normal Criteria: DG9 Eq.
4.17                      |
|-----------------------------------------------------------------------------
----|
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Uniform:      1.14946E+01 0.7617     1 | Uniform:      0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
|-----------------------------------------------------------------------------
----|
      STAAD SPACE                                                -- PAGE NO.
10
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                |
| Uniform:      1.14946E
+01                                                  |
|-----------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
```

```
4.6                                |
| Uniform:      0.00000E
+00                                                        |
|-------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                                |
| Uniform:      0.00000E
+00                                                        |
|-------------------------------------------------------------------------------
----|
```

V. AISC 360-10 W Shape Strong Axis Shear G.1

Verify the available shear strength along the strong axis of a wide flange section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.1A and G.1B, pp. G-3 - G-4

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

> Determine the available shear strength and adequacy of a W24×62 ASTM A992 beam ... with end shears of 48 kips from dead load and 145 kips from live load.

See example G.1A and G.1B in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	306	306	none	
V_n / Ω_v (kips) [ASD]	204	204	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 W Shape Strong Axis Shear G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 16 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X62
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -100 8
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -285 8
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
```

```
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 7200 ALL
FU 9360 ALL
METHOD ASD
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W24X62                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           8.014         3
                       0.00           0.00       -2304.00        8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    139.453  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     8.014(FAIL)                 |
|          Loc  :    8.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   2.880E+02     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.304E+03               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.307E+00     Ayy:   1.019E+01     Cw:  4.606E+03              |
| Szz:   1.308E+02     Syy:   9.801E+00                                 |
| Izz:   1.550E+03     Iyy:   3.450E+01     Ix:  1.710E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        16.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.97     N/A     13.49    T.B4.1(a)-1 |
|                Slender        50.05     N/A     35.88    T.B4.1(a)-5 |
| Flexure     :  Compact         5.97     9.15    24.08    T.B4.1(b)-10 |
|                Compact        50.05    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                        -- PAGE NO.
```

```
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   8.19E+02   0.000      Eq. D2-1      3     0.00  |
 | Rupture    0.00E+00   8.87E+02   0.000      Eq. D2-2      3     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   7.08E+02   0.000      Eq. E7-1      3     0.00  |
 | Min Buck   0.00E+00   2.11E+02   0.000      Eq. E7-1      3     0.00  |
 | Flexural                                                              |
 | Tor Buck   0.00E+00   4.45E+02   0.000      Eq. E7-1      3     0.00  |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.12E-01    20.81   6.23E+03   9.52E+04   7.87E+02        |
 | Min Buck    1.26E-01   139.45   1.86E+03   2.12E+03   2.35E+02        |
 | Flexural       Ag        Fcr        Pn                                |
 | Tor Buck    1.25E-01   3.91E+03   4.94E+02                            |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   2.24E+02   0.000      Eq. G2-1      3     0.00  |
 | Local-Y    2.88E+02   3.06E+02   0.942      Eq. G2-1      3     0.00  |
 | Intermediate                                                          |
 | Results        Aw        Cv        Kv      h/tw      Vn               |
 | Local-Z     5.77E-02     1.00      1.20     5.97   2.49E+02           |
 | Local-Y     7.08E-02     1.00      0.00    50.05   3.06E+02           |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      2.30E+03   5.74E+02   4.016      Eq. F2-1      3     8.00  |
 | Minor      0.00E+00   5.88E+01   0.000      Eq. F6-1      3     0.00  |
 | Intermediate   Mn        My                                           |
 | Major      6.38E+02   0.00E+00                                        |
 | Minor      6.53E+01   0.00E+00                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      2.30E+03   2.87E+02   8.014      Eq. F2-3      3     8.00  |
 | Intermediate   Mn        Me        Cb      Lp       Lr       Lb       |
 | Major      3.19E+02   0.00E+00    1.00     4.86    14.40    16.00     |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
 |                        RATIO      CRITERIA    L/C     LOC             |
```

```
| Flexure Comp          8.014      Eq. H1-1b     3     8.00              |
| Flexure Tens          8.014      Eq. H1-1b     3     8.00              |
| Intermediate         Mcx /      Mrx /      Pc /                        |
|                      Mcy        Mry        Pr                          |
| Flexure Comp         2.87E+02   2.30E+03   2.11E+02                    |
|                      5.88E+01   0.00E+00   0.00E+00                    |
| Flexure Tens         2.87E+02   2.30E+03   8.19E+02                    |
|                      5.88E+01   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2010
    59. FYLD 7200 ALL
    60. FU 9360 ALL
    61. METHOD ASD
    62. STP 1 ALL
    63. TRACK 2 ALL
    64. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W24X62                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           8.051         4
                        0.00            0.00       -1540.00        8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    139.453  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     8.051(FAIL)                 |
|          Loc  :    8.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.925E+02     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.540E+03               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.307E+00     Ayy:   1.019E+01     Cw:  4.606E+03              |
| Szz:   1.308E+02     Syy:   9.801E+00                                 |
| Izz:   1.550E+03     Iyy:   3.450E+01     Ix:  1.710E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        16.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.97     N/A     13.49    T.B4.1(a)-1  |
```

```
|                Slender        50.05     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.97     9.15    24.08    T.B4.1(b)-10 |
|                Compact        50.05    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   5.45E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   5.92E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   4.71E+02   0.000      Eq. E7-1      4     0.00 |
| Min Buck    0.00E+00   1.41E+02   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.96E+02   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr         Fe         Pn          |
| Maj Buck    1.12E-01    20.81   6.23E+03   9.52E+04   7.87E+02        |
| Min Buck    1.26E-01   139.45   1.86E+03   2.12E+03   2.35E+02        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    1.25E-01   3.91E+03   4.94E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.49E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.92E+02   2.04E+02   0.944      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     5.77E-02     1.00      1.20     5.97   2.49E+02            |
| Local-Y     7.08E-02     1.00      0.00    50.05   3.06E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.54E+03   3.82E+02   4.034      Eq. F2-1      4     8.00 |
| Minor       0.00E+00   3.91E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       6.38E+02   0.00E+00                                       |
| Minor       6.53E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.54E+03   1.91E+02   8.051      Eq. F2-3      4     8.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb     |
| Major       3.19E+02   0.00E+00    1.00     4.86    14.40    16.00    |
     STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------|
| CHECK  FOR FLEXURE TENS/COMP INTERACTION                           |
|                      RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp         8.051      Eq. H1-1b     4     8.00           |
| Flexure Tens         8.051      Eq. H1-1b     4     8.00           |
| Intermediate        Mcx /      Mrx /      Pc /                    |
|                     Mcy        Mry        Pr                      |
| Flexure Comp        1.91E+02   1.54E+03   1.41E+02                |
|                     3.91E+01   0.00E+00   0.00E+00                |
| Flexure Tens        1.91E+02   1.54E+03   5.45E+02                |
|                     3.91E+01   0.00E+00   0.00E+00                |
|--------------------------------------------------------------------|
```

V. AISC 360-10 C Strong Axis Shear G.2

Verify the available shear strength along the strong axis of a channel section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.2A and G.2B, pp. G-5 - G-6

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

> Verify the available shear strength and adequacy of a C15×33.9 ASTM A36 channel with end shears of 17.5 kips from dead load and 52.5 kips from live load.

See example G.2A and G.2B in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	117	117	none	
V_n / Ω_v (kips) [ASD]	77.8	77.6	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 C Strong Axis Shear G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -35
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -105
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 36 ALL
```

```
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            1.547         3
                        0.00            0.00       -2520.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     53.432  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.547(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.050E+02     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.520E+03              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1 |
|                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5 |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10 |
|                Compact        34.25   106.72   161.78    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00 |
```

```
| Rupture     0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   3.23E+02   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   2.79E+02   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.94E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.00E+01     8.55   3.59E+01   3.91E+03   3.59E+02        |
| Min Buck    1.00E+01    53.43   3.10E+01   1.00E+02   3.10E+02        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    1.00E+01   3.27E+01   3.27E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     1.05E+02   1.17E+02   0.900      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     4.42E+00     1.00     1.20     5.23   9.55E+01            |
| Local-Y     6.00E+00     1.00     5.00    34.25   1.30E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.52E+03   1.65E+03   1.531      Eq. F2-1      3    24.00 |
| Minor       0.00E+00   1.65E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.83E+03   0.00E+00                                      |
| Minor       1.83E+02   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.52E+03   1.63E+03   1.547      Eq. F2-2      3    24.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       1.81E+03   0.00E+00    1.00    44.87   173.87    48.00   |
    STAAD SPACE                                               -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         1.547      Eq. H1-1b     3    24.00              |
| Flexure Tens         1.547      Eq. H1-1b     3    24.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        1.63E+03   2.52E+03   2.79E+02                   |
|                     1.65E+02   0.00E+00   0.00E+00                   |
| Flexure Tens        1.63E+03   2.52E+03   3.24E+02                   |
|                     1.65E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

```
   48. LOAD LIST 4
   49. PARAMETER 2
   50. CODE AISC UNIFIED 2010
   51. METHOD ASD
   52. FYLD 36 ALL
   53. FU 58 ALL
   54. TRACK 2 ALL
   55. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                 -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                   (AISC SECTIONS)
                            FAIL   Eq. H1-1b            1.550         4
                         0.00            0.00       -1680.00       24.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     53.432  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     1.550(FAIL)                 |
 |          Loc  :   24.00   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   7.000E+01     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.680E+03               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02              |
 | Szz:   4.200E+01     Syy:   3.178E+00                                 |
 | Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          36.000   Fu:          58.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        48.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1(a)-1  |
 |                Non-Slender    34.25     N/A     42.29    T.B4.1(a)-5  |
 | Flexure     :  Compact         5.23    10.79    28.38    T.B4.1(b)-10 |
 |                Compact        34.25   106.72   161.78    T.B4.1(b)-15 |
      STAAD SPACE                                                 -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
```

```
 |                                                                     |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Yield      0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture    0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00 |
 |---------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                         |
 |                                                                     |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Maj Buck   0.00E+00   2.15E+02   0.000      Eq. E3-1      4     0.00 |
 | Min Buck   0.00E+00   1.85E+02   0.000      Eq. E3-1      4     0.00 |
 | Flexural                                                            |
 | Tor Buck   0.00E+00   1.96E+02   0.000      Eq. E4-1      4     0.00 |
 | Intermediate                                                        |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn          |
 | Maj Buck   1.00E+01     8.55   3.59E+01   3.91E+03   3.59E+02       |
 | Min Buck   1.00E+01    53.43   3.10E+01   1.00E+02   3.10E+02       |
 | Flexural      Ag        Fcr        Pn                               |
 | Tor Buck   1.00E+01   3.27E+01   3.27E+02                           |
 |---------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                     |
 |                                                                     |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Local-Z    0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00 |
 | Local-Y    7.00E+01   7.76E+01   0.902      Eq. G2-1      4     0.00 |
 | Intermediate                                                        |
 | Results       Aw        Cv       Kv      h/tw      Vn               |
 | Local-Z    4.42E+00     1.00     1.20     5.23   9.55E+01           |
 | Local-Y    6.00E+00     1.00     5.00    34.25   1.30E+02           |
 |---------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                          |
 |                                                                     |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Major      1.68E+03   1.10E+03   1.534      Eq. F2-1      4    24.00 |
 | Minor      0.00E+00   1.10E+02   0.000      Eq. F6-1      4     0.00 |
 | Intermediate   Mn        My                                         |
 | Major      1.83E+03   0.00E+00                                      |
 | Minor      1.83E+02   0.00E+00                                      |
 |---------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                        |
 |                                                                     |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Major      1.68E+03   1.08E+03   1.550      Eq. F2-2      4    24.00 |
 | Intermediate   Mn        Me         Cb      Lp       Lr       Lb    |
 | Major      1.81E+03   0.00E+00    1.00    44.87   173.87    48.00   |
      STAAD SPACE                                           -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |---------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
 |                      RATIO      CRITERIA    L/C     LOC             |
 | Flexure Comp          1.550      Eq. H1-1b     4    24.00            |
 | Flexure Tens          1.550      Eq. H1-1b     4    24.00            |
 | Intermediate         Mcx /      Mrx /      Pc /                     |
 |                      Mcy        Mry        Pr                       |
 | Flexure Comp         1.08E+03   1.68E+03   1.85E+02                 |
 |                      1.10E+02   0.00E+00   0.00E+00                 |
```

```
| Flexure Tens         1.08E+03   1.68E+03   2.16E+02                  |
|                      1.10E+02   0.00E+00   0.00E+00                  |
|----------------------------------------------------------------------|
```

V. AISC 360-10 L Shear Capacity G.3

Verify the available shear strength along the strong axis of a single angle section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.3, pp. G-7 - G-8

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

Determine the available shear strength and adequacy of a L5×3×1/4 (LLV) ASTM A36 with end shears of 3.50 kips from dead load and 10.5 kips from live load.

Validation

Ultimate load:

$$V_u = 1.2(3.50 \text{ kips}) + 1.6(10.5 \text{ kips}) = 21.0 \text{ kips}$$

Service load:

$$V_a = 3.50 \text{ kips} + 10.5 \text{ kips} = 14.0 \text{ kips}$$

Area of the web for shear:

$$A_w = bt = 5.0(0.25) = 1.25 \text{ in}^2$$

Section G4 of Reference 1 gives $k_v = 1.2$.

$$\frac{h}{t_w} = \frac{b}{t} = \frac{5.0}{0.25} = 20 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{1.2(29,\ 000)}{36}} = 34.2$$

Therefore, $C_v = 1.0$.

<u>Nominal Shear Strength</u>

$$V_n = 0.6F_y A_w C_v = 0.6(36)(1.25)(1.0) = 27.0 \text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(27.0) = 24.3 \text{ kips}$

ASD: $V_n / \Omega_v = {}^{27.0}\!/_{1.67} = 16.2\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	24.3	24.3	none	
V_n / Ω_v (kips) [ASD]	16.2	16.2	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 L Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L50304
CONSTANTS
BETA 45 ALL
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON Y -10
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON Y -20
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
```

```
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   L50304                   (AISC SECTIONS)
                           FAIL   Eq. H2-1            14.492         3
                        0.00          230.75         297.25       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     73.620  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :       3    Ratio     :    14.492(FAIL)               |
|          Loc  :   24.00   Condition :   Eq. H2-1                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   2.200E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   2.307E+02     Mz:   2.973E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.250E+00     Ayy:   7.500E-01     Cw:  6.060E-02             |
| Szz:   5.259E-01     Syy:   1.692E+00                                |
| Izz:   8.247E-01     Iyy:   5.723E+00     Ix:  4.042E-02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender       20.00     N/A     12.77    T.B4.1(a)-3  |
```

```
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield      0.00E+00   6.29E+01   0.000      Eq. D2-1      3     0.00 |
| Rupture    0.00E+00   8.44E+01   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck   0.00E+00   4.89E+01   0.000      Eq. E7-1      3     0.00 |
| Min Buck   0.00E+00   4.02E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck   0.00E+00   3.49E+01   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck   1.94E+00    27.95   2.80E+01   3.66E+02   5.44E+01        |
| Min Buck   1.94E+00    73.62   2.30E+01   5.28E+01   4.47E+01        |
| Flexural      Ag        Fcr        Pn                                 |
| Tor Buck   1.94E+00   2.00E+01   3.88E+01                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z    0.00E+00   1.46E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y    2.20E+01   2.43E+01   0.905      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results       Aw        Cv        Kv      h/tw      Vn               |
| Local-Z    7.50E-01     1.00      1.20    12.00   1.62E+01            |
| Local-Y    1.25E+00     1.00      1.20    20.00   2.70E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major     -2.31E+02   8.23E+01   2.805      Eq. F10-1     3     0.00 |
| Minor      2.97E+02   2.56E+01  11.631      Eq. F10-1     3    24.00 |
| Intermediate   Mn        My                                          |
| Major      9.14E+01   6.09E+01                                        |
| Minor      2.84E+01   1.89E+01                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major     -2.31E+02   8.06E+01   2.861      Eq. F10-3     3     0.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb     |
| Major      8.96E+01   4.13E+02    1.00     0.00     0.00    48.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
```

```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Minor      -2.31E+02   4.17E+01   5.539      Eq. F10-7     3     0.00  |
| Intermediate   Mn        Me                                           |
| Minor       4.63E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|             STRESS     RATIO      CRITERIA     L/C    LOC            |
| Shear       1.94E+01    0.905      Eq. H3-8      3     0.00           |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp        14.492      Eq. H2-1      3    24.00              |
| Flexure Tens        14.492      Eq. H2-1      3    24.00              |
| Intermediate        frbw /     frbz /     fra /                       |
|                     Fcbw       Fcbz       Fca                         |
| Flexure Comp        9.59E+01   5.65E+02   0.00E+00                    |
|                     3.35E+01   4.86E+01   1.80E+01                    |
| Flexure Tens        1.36E+02   5.65E+02   0.00E+00                    |
|                     4.76E+01   4.86E+01   3.24E+01                    |
|-----------------------------------------------------------------------|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2010
    52. METHOD ASD
    53. FYLD 36 ALL
    54. FU 58 ALL
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                               -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   L50304                   (AISC SECTIONS)
                           FAIL   Eq. H2-1            14.851        4
                        0.00          157.33         202.67       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     73.620  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :    14.851(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.500E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   1.573E+02     Mz:   2.027E+02               |
|-----------------------------------------------------------------------|
```

```
|| SECTION PROPERTIES  (UNIT: INCH)                                       ||
|| Azz:   1.250E+00     Ayy:   7.500E-01     Cw:  6.060E-02               ||
|| Szz:   5.259E-01     Syy:   1.692E+00                                  ||
|| Izz:   8.247E-01     Iyy:   5.723E+00     Ix:  4.042E-02               ||
||------------------------------------------------------------------------||
|| MATERIAL PROPERTIES                                                    ||
|| Fyld:          36.000   Fu:          58.000                            ||
||------------------------------------------------------------------------||
|| Actual Member Length:        48.000                                    ||
|| Design Parameters                                                      ||
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         ||
||------------------------------------------------------------------------||
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        ||
||                STIFFENED                                               ||
|| Compression :  Slender        20.00     N/A     12.77    T.B4.1(a)-3   ||
||                  N/A           N/A      N/A      N/A      N/A          ||
|| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1(b)-12  ||
||                  N/A           N/A      N/A      N/A      N/A          ||
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
||------------------------------------------------------------------------||
|| CHECK FOR AXIAL TENSION                                                ||
||                                                                        ||
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    ||
|| Yield      0.00E+00   4.18E+01   0.000      Eq. D2-1      4     0.00   ||
|| Rupture    0.00E+00   5.63E+01   0.000      Eq. D2-2      4     0.00   ||
||------------------------------------------------------------------------||
|| CHECK FOR AXIAL COMPRESSION                                            ||
||                                                                        ||
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    ||
|| Maj Buck   0.00E+00   3.25E+01   0.000      Eq. E7-1      4     0.00   ||
|| Min Buck   0.00E+00   2.67E+01   0.000      Eq. E7-1      4     0.00   ||
|| Flexural                                                               ||
|| Tor Buck   0.00E+00   2.32E+01   0.000      Eq. E7-1      4     0.00   ||
|| Intermediate                                                           ||
|| Results     Eff Area     KL/r      Fcr        Fe         Pn            ||
|| Maj Buck   1.94E+00    27.95   2.80E+01   3.66E+02   5.44E+01          ||
|| Min Buck   1.94E+00    73.62   2.30E+01   5.28E+01   4.47E+01          ||
|| Flexural      Ag       Fcr        Pn                                   ||
|| Tor Buck   1.94E+00   2.00E+01   3.88E+01                              ||
||------------------------------------------------------------------------||
|| CHECK FOR SHEAR                                                        ||
||                                                                        ||
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    ||
|| Local-Z    0.00E+00   9.70E+00   0.000      Eq. G2-1      4     0.00   ||
|| Local-Y    1.50E+01   1.62E+01   0.928      Eq. G2-1      4     0.00   ||
|| Intermediate                                                           ||
|| Results       Aw        Cv        Kv      h/tw      Vn                 ||
|| Local-Z    7.50E-01    1.00      1.20    12.00   1.62E+01              ||
|| Local-Y    1.25E+00    1.00      1.20    20.00   2.70E+01              ||
||------------------------------------------------------------------------||
|| CHECK FOR BENDING-YIELDING                                             ||
||                                                                        ||
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    ||
|| Major     -1.57E+02   5.47E+01   2.875      Eq. F10-1     4     0.00   ||
```

```
| Minor        2.03E+02   1.70E+01  11.919      Eq. F10-1     4    24.00 |
| Intermediate   Mn         My                                           |
| Major        9.14E+01   6.09E+01                                       |
| Minor        2.84E+01   1.89E+01                                       |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                        |
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       -1.57E+02   5.37E+01   2.932      Eq. F10-3     4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major        8.96E+01   4.13E+02    1.00     0.00     0.00    48.00   |
    STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                   |
|                                                                        |
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Minor       -1.57E+02   2.77E+01   5.676      Eq. F10-7     4     0.00 |
| Intermediate   Mn         Me                                           |
| Minor        4.63E+01   0.00E+00                                       |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                          |
|                                                                        |
|               STRESS     RATIO      CRITERIA     L/C    LOC           |
| Shear        1.29E+01    0.928      Eq. H3-8      4     0.00           |
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                          RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           14.851      Eq. H2-1      4    24.00            |
| Flexure Tens           14.851      Eq. H2-1      4    24.00            |
| Intermediate           frbw /     frbz /     fra /                     |
|                        Fcbw       Fcbz       Fca                       |
| Flexure Comp           6.54E+01   3.85E+02   0.00E+00                  |
|                        2.23E+01   3.23E+01   1.20E+01                  |
| Flexure Tens           9.30E+01   3.85E+02   0.00E+00                  |
|                        3.17E+01   3.23E+01   2.16E+01                  |
|------------------------------------------------------------------------|
```

V. AISC 360-10 HSST Shear Capacity G.4

Verify the available shear strength along the strong axis of a rectangular HSS section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.4, pp. G-9 - G-10

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

> Determine the available shear strength and adequacy of an HSS6×4×3/8 ASTM A500 Grade B member with end shears of 11.0 kips from dead load and 33.0 kips from live load. The beam is oriented with the shear parallel to the 6 in. dimension.

Validation

Ultimate load:

$$V_u = 1.2(11.0\text{ kips}) + 1.6(33.0\text{ kips}) = 66.0\text{ kips}$$

Service load:

$$V_a = 11.0\text{ kips} + 33.0\text{ kips} = 44.0\text{ kips}$$

Area of the web for shear:

Note: The corner radius is taken as 1.5× the design wall thickness per Section B4.1b of Reference 1.

$$h = H - 3t = 6.0 - 3\times 0.349 = 4.95\text{ in}$$

Section G5 of Reference 1 gives $k_v = 5$.

$$A_w = 2ht = 2(4.95)(0.349) = 3.46\text{ in}^2$$

$$\frac{h}{t_w} = \frac{4.95}{0.349} = 14.2 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{5(29,\ 000)}{46}} = 61.8$$

Therefore, $C_v = 1.0$.

Nominal Shear Strength

$$V_n = 0.6F_y A_w C_v = 0.6(46)(3.46)(1.0) = 95.5\text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(95.5) = 86.0\ kips$

ASD: $V_n / \Omega_v = {}^{95.5}\!/_{1.67} = 57.2\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	86.0	85.9	negligible	
V_n/Ω_v (kips) [ASD]	57.2	57.1	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -22
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -66
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FYLD 6624 ALL
FU 8352 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FYLD 6624 ALL
FU 8352 ALL
METHOD ASD
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.215         3
                        0.00            0.00        -132.00        2.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     30.913  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     2.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     3.215(FAIL)                 |
|          Loc  :    2.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   6.600E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.320E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00              |
| Szz:   9.433E+00     Syy:   7.450E+00                                 |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         4.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    14.19     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1(b)-17 |
|                Compact        14.19    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   2.56E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   2.69E+02   0.000      Eq. D2-2      3     0.00 |
```

```
|-------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                       |
|                                                                   |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   2.47E+02   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   2.40E+02   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                      |
| Results     Eff Area     KL/r      Fcr        Fe         Pn       |
| Maj Buck    4.29E-02    22.43   6.40E+03   8.19E+04   2.75E+02     |
| Min Buck    4.29E-02    30.91   6.21E+03   4.31E+04   2.67E+02     |
|-------------------------------------------------------------------|
| CHECK FOR SHEAR                                                   |
|                                                                   |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.12E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     6.60E+01   8.59E+01   0.769      Eq. G2-1      3     0.00 |
| Intermediate                                                      |
| Results        Aw         Cv       Kv      h/tw      Vn          |
| Local-Z     1.43E-02     1.00      5.00     8.46   5.69E+01        |
| Local-Y     2.40E-02     1.00      5.00    14.19   9.54E+01        |
|-------------------------------------------------------------------|
| CHECK FOR TORSION                                                 |
|                                                                   |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.94E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                        |
|             3.97E+03   3.27E+01                                  |
|-------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                        |
|                                                                   |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.32E+02   4.11E+01   3.215      Eq. F7-1      3     2.00 |
| Minor       0.00E+00   3.08E+01   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                      |
| Major       4.56E+01   0.00E+00                                   |
| Minor       3.43E+01   0.00E+00                                   |
     STAAD SPACE                                          -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                           |
|                      RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp         3.215      Eq. H1-1b     3     2.00          |
| Flexure Tens         3.215      Eq. H1-1b     3     2.00          |
| Intermediate        Mcx /      Mrx /      Pc /                    |
|                     Mcy        Mry        Pr                      |
| Flexure Comp        4.11E+01   1.32E+02   2.40E+02                |
|                     3.08E+01   0.00E+00   0.00E+00                |
| Flexure Tens        4.11E+01   1.32E+02   2.56E+02                |
|                     3.08E+01   0.00E+00   0.00E+00                |
|-------------------------------------------------------------------|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2010
    50. FYLD 6624 ALL
    51. FU 8352 ALL
```

```
    52. METHOD ASD
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.222         4
                        0.00            0.00         -88.00        2.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     30.913  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     2.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     3.222(FAIL)                 |
 |          Loc  :    2.00   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   4.400E+01     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -8.800E+01               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00              |
 | Szz:   9.433E+00     Syy:   7.450E+00                                 |
 | Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        6623.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:         4.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Non-Slender    14.19     N/A     35.15    T.B4.1(a)-6  |
 | Flexure     :  Compact         8.46    28.12    35.15    T.B4.1(b)-17 |
 |                Compact        14.19    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL TENSION                                              |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.70E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.79E+02   0.000      Eq. D2-2      4     0.00 |
|----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                          |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.65E+02   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   1.60E+02   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.29E-02    22.43   6.40E+03   8.19E+04   2.75E+02       |
| Min Buck    4.29E-02    30.91   6.21E+03   4.31E+04   2.67E+02       |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   3.41E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     4.40E+01   5.71E+01   0.770      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.43E-02     1.00     5.00     8.46   5.69E+01            |
| Local-Y     2.40E-02     1.00     5.00    14.19   9.54E+01            |
|----------------------------------------------------------------------|
| CHECK FOR TORSION                                                    |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.96E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                           |
|             3.97E+03   3.27E+01                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       8.80E+01   2.73E+01   3.222      Eq. F7-1      4     2.00 |
| Minor       0.00E+00   2.05E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       4.56E+01   0.00E+00                                      |
| Minor       3.43E+01   0.00E+00                                      |
     STAAD SPACE                                            -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           3.222      Eq. H1-1b     4     2.00           |
| Flexure Tens           3.222      Eq. H1-1b     4     2.00           |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          2.73E+01   8.80E+01   1.60E+02                 |
|                       2.05E+01   0.00E+00   0.00E+00                 |
| Flexure Tens          2.73E+01   8.80E+01   1.70E+02                 |
```

```
|                    2.05E+01   0.00E+00   0.00E+00                    |
|----------------------------------------------------------------------|
```

V. AISC 360-10 HSSP Shear Capacity G.5

Verify the available shear strength along the strong axis of a round HSS section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.5, pp. G-11 - G-12

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

> Verify the available shear strength and adequacy of a round HSS16.000×0.375 ASTM A500 Grade B member spanning 32 ft with end shears of 30.0 kips from dead load and 90.0 kips from live load.

Validation

Ultimate load:

$$V_u = 1.2(30.0\text{ kips}) + 1.6(90.0\text{ kips}) = 180\text{ kips}$$

Service load:

$$V_a = 30.0\text{ kips} + 90.0\text{ kips} = 120\text{ kips}$$

Shear Buckling Stress

$$F_{cr} = \max\begin{cases} \dfrac{1.60E}{\sqrt{\dfrac{L_v}{D}}\left(\dfrac{D}{t}\right)^{5/4}} = \dfrac{1.60(29,\ 000)}{\sqrt{\dfrac{192}{16.0}}\left(\dfrac{16.0}{0.349}\right)^{5/4}} = 112\text{ ksi} \\ \dfrac{0.78E}{\left(D/t\right)^{3/2}} = \dfrac{0.78(29,\ 000)}{\left(16.0/0.349\right)^{3/2}} = 72.9\text{ ksi} \end{cases}$$

The maximum permitted value is $F_{cr} \le 0.6F_y = 0.6(42) = 25.2\ ksi$ (governs).

Nominal Shear Strength

$$V_n = F_{cr}A_g / 2 = (25.2)(17.2) / 2 = 217\text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(217) = 195\ kips$

ASD: $V_n / \Omega_v = {}^{217}/_{1.67} = 130\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	195	195	none	
V_n/Ω_v (kips) [ASD]	130	130	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSSP Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 32 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP16X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
UNIT INCHES KIP
1 UNI GY -0.15625
UNIT FEET KIP
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
UNIT INCHES KIP
1 UNI GY -0.46875
UNIT FEET KIP
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
```

```
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 42 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 42 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   HSSP16X0.375             (AISC SECTIONS)
                           FAIL   Eq. H1-1b            5.347         3
                        0.00            0.00      -17280.00      192.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     69.439  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :   192.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     5.347(FAIL)                 |
|          Loc  :  192.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.728E+04              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.728E+00     Ayy:   8.728E+00     Cw:  0.000E+00             |
| Szz:   6.575E+01     Syy:   6.575E+01                                 |
| Izz:   5.260E+02     Iyy:   5.260E+02     Ix:  1.052E+03             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          42.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       384.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
```

```
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                              |
| Compression :  Non-Slender    45.85     N/A     75.95    T.B4.1(a)-9  |
|                Non-Slender    45.85     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        45.85    48.33   214.05    T.B4.1(b)-20 |
|                Compact        45.85    48.33   214.05    T.B4.1(b)-20 |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   6.50E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   7.48E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   4.84E+02   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   4.84E+02   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02        |
| Min Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.95E+02   0.000      Eq. G6-1      3     0.00 |
| Local-Y     1.80E+02   1.95E+02   0.923      Eq. G6-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.72E+01     0.00     0.00     0.00   2.17E+02            |
| Local-Y     1.72E+01     0.00     0.00     0.00   2.17E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             0.00E+00   2.97E+03   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.52E+01   3.30E+03                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.73E+04   3.23E+03   5.347      Eq. F8-1      3   192.00 |
| Minor       0.00E+00   3.23E+03   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.59E+03   0.00E+00                                      |
| Minor       3.59E+03   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
```

```
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         5.347      Eq. H1-1b     3   192.00              |
| Flexure Tens         5.347      Eq. H1-1b     3   192.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        3.23E+03   1.73E+04   4.84E+02                    |
|                     3.23E+03   0.00E+00   0.00E+00                    |
| Flexure Tens        3.23E+03   1.73E+04   6.50E+02                    |
|                     3.23E+03   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2010
    55. METHOD ASD
    56. FYLD 42 ALL
    57. FU 58 ALL
    58. TRACK 2 ALL
    59. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSSP16X0.375             (AISC SECTIONS)
                           FAIL   Eq. H1-1b            5.357         4
                        0.00           0.00      -11520.00      192.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     69.439  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :   192.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     5.357(FAIL)                |
|          Loc  :  192.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.152E+04               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.728E+00     Ayy:   8.728E+00     Cw:  0.000E+00             |
| Szz:   6.575E+01     Syy:   6.575E+01                                |
| Izz:   5.260E+02     Iyy:   5.260E+02     Ix:  1.052E+03             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          42.000   Fu:          58.000                           |
```

```
 |-----------------------------------------------------------------------|
 | Actual Member Length:       384.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender    45.85     N/A     75.95    T.B4.1(a)-9  |
 |                Non-Slender    45.85     N/A     75.95    T.B4.1(a)-9  |
 | Flexure     :  Compact        45.85    48.33   214.05    T.B4.1(b)-20 |
 |                Compact        45.85    48.33   214.05    T.B4.1(b)-20 |
     STAAD SPACE                                               -- PAGE NO.
7
                    STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   4.33E+02   0.000      Eq. D2-1      4     0.00  |
 | Rupture    0.00E+00   4.99E+02   0.000      Eq. D2-2      4     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   3.22E+02   0.000      Eq. E3-1      4     0.00  |
 | Min Buck   0.00E+00   3.22E+02   0.000      Eq. E3-1      4     0.00  |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn            |
 | Maj Buck   1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02         |
 | Min Buck   1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02         |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   1.30E+02   0.000      Eq. G6-1      4     0.00  |
 | Local-Y    1.20E+02   1.30E+02   0.925      Eq. G6-1      4     0.00  |
 | Intermediate                                                          |
 | Results       Aw         Cv       Kv      h/tw      Vn                |
 | Local-Z    1.72E+01     0.00     0.00     0.00   2.17E+02             |
 | Local-Y    1.72E+01     0.00     0.00     0.00   2.17E+02             |
 |-----------------------------------------------------------------------|
 | CHECK FOR TORSION                                                     |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 |            0.00E+00   1.98E+03   0.000      Eq. H3-1      4     0.00  |
 | Intermediate Fcr        Tn                                            |
 |            2.52E+01   3.30E+03                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major      1.15E+04   2.15E+03   5.357      Eq. F8-1      4   192.00  |
 | Minor      0.00E+00   2.15E+03   0.000      Eq. F8-1      4     0.00  |
 | Intermediate   Mn        My                                           |
 | Major      3.59E+03   0.00E+00                                        |
 | Minor      3.59E+03   0.00E+00                                        |
```

```
      STAAD SPACE                                                 -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          5.357      Eq. H1-1b     4   192.00             |
| Flexure Tens          5.357      Eq. H1-1b     4   192.00             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.15E+03   1.15E+04   3.22E+02                   |
|                      2.15E+03   0.00E+00   0.00E+00                   |
| Flexure Tens         2.15E+03   1.15E+04   4.33E+02                   |
|                      2.15E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 - G-6

Reference

Design Examples, Version 14.0, American Institute of Steel Construction, 2011, page G-13.

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Problem

From the reference:

> Verify the available shear strength and adequacy of a W21×48 ASTM A992 beam with end shears of 20.0 kips from dead load and 60.0 kips from live load in the weak direction.

Refer to the reference for calculations.

Comparison

Table 752: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
$\phi_v V_n$ (kips) [LRFD]	189	189	none
V_n / Ω_v (kips) [ASD]	126	126	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 - G-6.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.28151e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GZ -20 0.665
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GZ -60 0.665
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FYLD 7200
FU 9360
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
```

```
FYLD 7200
FU 9360
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            1.439         3
                        0.00          -79.80           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     57.946  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.439(FAIL)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   1.200E+02              |
| Mx:   0.000E+00     My:  -7.980E+01     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03              |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         8.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.66    T.B4.1(a)-1  |
|                Slender        53.54     N/A     36.33    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.47     9.27    24.39    T.B4.1(b)-10 |
|                Compact        53.54    91.69   139.00    T.B4.1(b)-15 |
      STAAD SPACE                                            -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
```

```
| Yield       0.00E+00   6.35E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   6.87E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   5.54E+02   0.000      Eq. E7-1      3     0.00 |
| Min Buck    0.00E+00   4.66E+02   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   4.97E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    8.62E-02    11.64   6.28E+03   3.12E+05   6.15E+02       |
| Min Buck    8.94E-02    57.95   5.28E+03   1.26E+04   5.17E+02       |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    8.82E-02   5.64E+03   5.52E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z    -1.20E+02   1.89E+02   0.635      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.16E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02           |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.01E+02   0.000      Eq. F2-1      3     0.00 |
| Minor      -7.98E+01   5.59E+01   1.428      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.71E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major       4.12E+02   0.00E+00    1.00     5.93    16.76     8.00   |
    STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.99E+02   0.000      Eq. F3-1      3     0.00 |
| Minor      -7.98E+01   5.55E+01   1.439      Eq. F6-2      3     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       4.44E+02   0.00E+00                                       |
| Minor       6.16E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
```

```
|                        RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp           1.439      Eq. H1-1b     3     0.00             |
| Flexure Tens           1.439      Eq. H1-1b     3     0.00             |
| Intermediate          Mcx /      Mrx /      Pc /                       |
|                       Mcy        Mry        Pr                         |
| Flexure Comp          3.71E+02   0.00E+00   4.66E+02                   |
|                       5.55E+01  -7.98E+01   0.00E+00                   |
| Flexure Tens          3.71E+02   0.00E+00   6.35E+02                   |
|                       5.55E+01  -7.98E+01   0.00E+00                   |
|------------------------------------------------------------------------|
   46. LOAD LIST 4
   47. PARAMETER 2
   48. CODE AISC UNIFIED 2010
   49. METHOD ASD
   50. FYLD 7200
   51. FU 9360
   52. TRACK 2 ALL
   53. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                  (AISC SECTIONS)
                           FAIL   Eq. H1-1b            1.442         4
                        0.00          -53.20           0.00        0.00
|------------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :     57.946  L/C  :     4                 |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     1.442(FAIL)                  |
|          Loc  :    0.00   Condition :   Eq. H1-1b                      |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   8.000E+01                |
| Mx:   0.000E+00     My:  -5.320E+01     Mz:   0.000E+00                |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03              |
| Szz:   9.311E+01     Syy:   9.509E+00                                  |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01              |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:        7199.999   Fu:        9359.999                            |
|------------------------------------------------------------------------|
| Actual Member Length:         8.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Non-Slender     9.47     N/A     13.66    T.B4.1(a)-1   |
```

```
|               Slender        53.54     N/A     36.33    T.B4.1(a)-5  |
| Flexure     :  Non-Compact     9.47     9.27    24.39    T.B4.1(b)-10 |
|               Compact        53.54    91.69   139.00    T.B4.1(b)-15 |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.22E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.58E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   3.68E+02   0.000      Eq. E7-1      4     0.00 |
| Min Buck    0.00E+00   3.10E+02   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   3.31E+02   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    8.62E-02    11.64   6.28E+03   3.12E+05   6.15E+02       |
| Min Buck    8.94E-02    57.95   5.28E+03   1.26E+04   5.17E+02       |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    8.82E-02   5.64E+03   5.52E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    -8.00E+01   1.26E+02   0.636      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.44E+02   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02           |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.67E+02   0.000      Eq. F2-1      4     0.00 |
| Minor      -5.32E+01   3.72E+01   1.431      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.47E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb    |
| Major       4.12E+02   0.00E+00    1.00     5.93    16.76     8.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
```

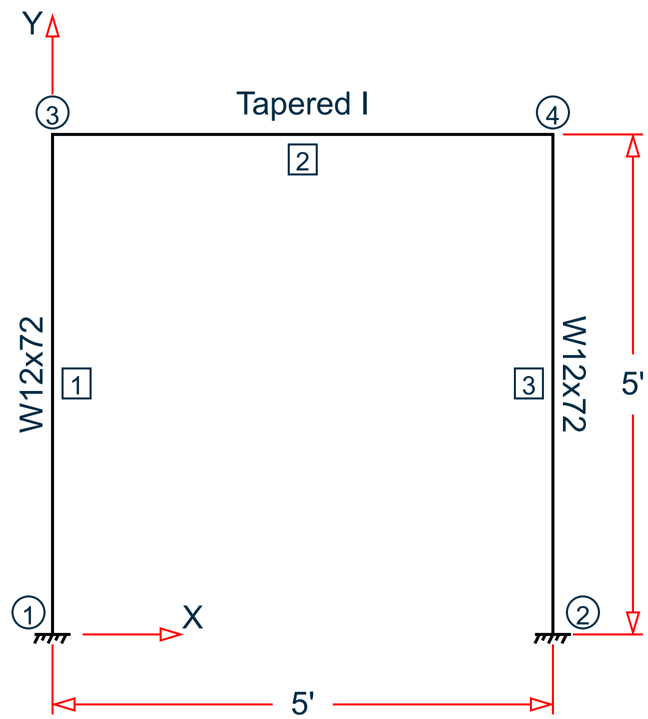
```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.66E+02   0.000      Eq. F3-1      4     0.00 |
| Minor      -5.32E+01   3.69E+01   1.442      Eq. F6-2      4     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       4.44E+02   0.00E+00                                      |
| Minor       6.16E+01   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         1.442      Eq. H1-1b     4     0.00              |
| Flexure Tens         1.442      Eq. H1-1b     4     0.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        2.47E+02   0.00E+00   3.10E+02                    |
|                     3.69E+01  -5.32E+01   0.00E+00                    |
| Flexure Tens        2.47E+02   0.00E+00   4.22E+02                    |
|                     3.69E+01  -5.32E+01   0.00E+00                    |
|-----------------------------------------------------------------------|
```

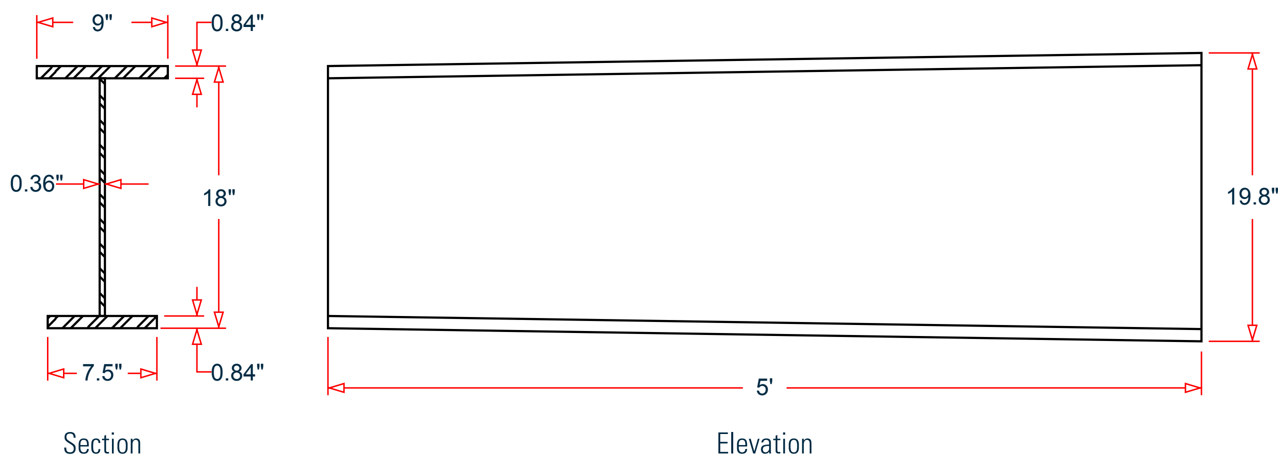
V. AISC 360-10 Tapered I Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered I section member per both the LRFD and ASD methods of the AISC 360-10 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





The following load cases are evaluated:

1. A uniformly distributed load of -3.5 k/ft along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. Lateral loads at the top of the right column of -50 kips in the global X direction and 45 kips in the global Z direction (out of plane).
4. A concreted load of -100 kips in the global Y direction and a concentrated torque of 9 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the left end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 32.26 in·kips, F_y = 8.755 kips
2. M_z = 677.6 in·kips, M_y = 2.942 in·kips, M_x = 5.106 in·kips, F_y = 22.49 kips, F_z = 0.098 kips, F_x = 24.70 kips
3. M_z = 275.9 in·kips, M_x = 4.497 in·kips, F_y = 50.23 kips, F_x = 5.473 kips
4. M_z = 654.5 in·kips, M_y = 5.298 in·kips, M_x = 9.192 in·kips, F_y = 22.50 kips, F_z = 0.177 kips, F_x = 24.35 kips

Section Properties

The section at the left end ($D = 18\ in.$) has the following properties:

Web height, $d_w = 18 - 2{\times}0.84 = 16.32\ in.$
Area, $A_g = (0.84)(9 + 7.5) + 16.32\ (0.36) = 19.74\ in^2$
Shear area in local y, $A_y = 16.32(0.36) = 5.875\ in^2$
Shear area in local x, $A_x = 0.84(9 + 7.5) = 13.86\ in^2$

Dist to centroid y, $c = \dfrac{7.5(0.84)\left(\frac{0.84}{2}\right) + 0.36(16.32)\left(0.84 + \frac{16.32}{2}\right) + 9(0.84)\left(18 - \frac{0.84}{2}\right)}{19.74} = 9.548\ in.$

Moment of inertia about major axis,

$$I_{zz} = \begin{aligned} &\frac{7.5(0.84)^3}{12} + 7.5(0.84)\left(9.548 - \frac{0.84}{2}\right)^2 \\ &+ \frac{0.36(16.32)^3}{12} + 0.36(16.32)\left(9.548 - \frac{16.32}{2} - 0.84\right)^2 \\ &+ \frac{9(0.84)^3}{12} + 9(0.84)\left(18 - \frac{0.84}{2} - 9.548\right)^2 \end{aligned} = 1,\ 147\ \text{in}^4$$

Moment of inertia about minor axis,

$$I_{yy} = \frac{0.84(9)^3}{12} + \frac{16.32(0.36)^3}{12} + \frac{0.84(7.5)^3}{12} = 80.62\ \text{in}^4$$

Elastic section modulus about major axis top flange fiber, $S_{xx,top} = \dfrac{I_{zz}}{D - c} = \dfrac{1,\ 146}{18 - 9.548} = 135.5\ in^3$

Elastic section modulus about major axis bottom flange fiber, $S_{xx,bottom} = \dfrac{I_{zz}}{c} = \dfrac{1,\ 146}{9.548} = 120.0\ in^3$

Elastic section modulus about minor axis, $S_{yy} = \dfrac{I_{yy}}{b_{f1} / 2} = \dfrac{80.62}{9 / 2} = 17.92\ in^3$

Radius of gyration about major axis, $r_x = \sqrt{\dfrac{I_{zz}}{A_g}} = \sqrt{\dfrac{1,\ 147}{19.74}} = 7.619\ in$

Radius of gyration about minor axis, $r_y = \sqrt{\dfrac{I_{yy}}{A_g}} = \sqrt{\dfrac{80.62}{19.74}} = 2.021\ in$

Location of the plastic neutral axis, $y_{PNA} = 0.84 + \frac{19.74 - 2(7.5)(0.84)}{2 \times 0.36} = 10.75\ in.$

Plastic section modulus about major axis,

$$Z_{zt} = \frac{7.5(10.75)^2}{2} - \frac{(7.5 - 0.36)(10.75 - 0.84)^2}{2} + \frac{9(18 - 10.75)^2}{2} - \frac{(9 - 0.36)(18 - 0.84 - 10.75)^2}{2} = 141.8\ \text{in}^3$$

Plastic section modulus about minor axis, $Z_y = \frac{0.84(9)^2}{4} + \frac{16.32(0.36)^2}{4} + \frac{0.84(7.5)^2}{4} = 29.35\ in^3$

Torsional constant, $J = \frac{9(0.84)^3 + 16.32(0.36)^3 + 7.5(0.84)^3}{3} = 3.514\ in.^4$

Section Classification

Flange in compression

$$\lambda = \frac{b}{t} = \frac{1/2 \times 7.5}{0.84} = 4.464$$

$$k_c = \frac{4}{\sqrt{d_w / t_w}} = \frac{4}{\sqrt{\frac{16.32}{0.36}}} = 0.594$$

Per Table B4.1a, $\lambda < \lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.594(29,000)}{50}} = 11.88$, therefore flanges are non-slender for compression.

Web in compression

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table B4.1a, Case 5, $\lambda > \lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,000}{50}} = 35.88$, therefore the web is slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{b_f}{t_f} = \frac{9}{2(0.84)} = 5.36$$

Per Table B4.1b, Case 11, $\lambda < \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29,000}{50}} = 9.15$, therefore flange is compact for bending.

Web in bending

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table B4.1b, Case 15, $\lambda < \lambda_p = 3.76\sqrt{\frac{E}{F_y}} = 3.76\sqrt{\frac{29,000}{50}} = 90.55$, therefore web is compact for bending.

Compression Capacity

About X Axis:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{7.619} = 7.875 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(7.875)^2} = 4{,}615 \text{ ksi} \quad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ex}} = \frac{50}{4{,}615} = 0.011 < 2.25$$

$$F_{crx} = 0.658^{F_y / F_{ex}} F_y = 0.658^{(0.011)} 50 = 49.77 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Since the web is slender, use the effective area. The effective width imperfection adjustment factors (Table E7.1):

$c_1 = 0.18$

$c_2 = 1.31$

$$F_{el} = \left(c_2 \frac{\lambda_r}{\lambda}\right)^2 F_y = \left(1.31 \frac{35.88}{45.33}\right)^2 = 53.76 \text{ ksi} \quad \text{(Eq. E7-5)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w \left(1 - c_1 \sqrt{\frac{F_{el}}{F_{crx}}}\right) \sqrt{\frac{F_{el}}{F_{crx}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{49.77}}\right)\sqrt{\frac{53.76}{49.77}} = 13.78 \text{ in.} \quad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 13.78)0.36 = 18.84 \text{ in.}^2$$

$$P_{nx} = A_e \times F_{crx} = 18.84 \times 49.77 = 937.4 \text{ kips} \quad \text{(Eq. E7-1)}$$

About Y Axis:

$$\frac{k_y L_y}{r_y} = \frac{1.0(60)}{2.021} = 29.69 < 4.71\sqrt{\frac{E}{F_y}} = 113.4$$

$$F_{ey} = \frac{\pi^2 E}{\left(\frac{k_y L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(29.69)^2} = 324.7 \text{ ksi} \quad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ey}} = \frac{50}{324.7} = 0.154 < 2.25$$

$$F_{cry} = 0.658^{(F_y / F_{ey})} F_y = 0.658^{(0.154)} 50 = 46.88 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w \left(1 - c_1 \sqrt{\frac{F_{el}}{F_{cry}}}\right) \sqrt{\frac{F_{el}}{F_{cry}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{46.88}}\right)\sqrt{\frac{53.76}{46.88}} = 14.11 \text{ in.} \quad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 14.11)0.36 = 18.93 \text{ in.}^2$$

$$P_{ny} = A_e \times F_{cry} = 18.93 \times 46.88 = 887.4 \text{ kips} \quad \text{(Eq. E7-1)}$$

The Y axis governs; $P_n = P_{ny}$. The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 887.4 = 798.3 \; kips$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 887.4 \;/\; 1.67 = 531.4 \; kips$

Calculate Bending Capacity

Bending capacity in plastic yielding about Y axis:

$$M_y = F_y \times Z_y = 50 \times 29.35 = 1{,}468 \text{ in·kips} > 1.6F_y \times S_y = 1.6\ (50)\ (17.92) = 1{,}434 \text{ in·kips}$$

Use $M_y = 1{,}434 \; in \cdot kips$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}434 = 1{,}290 \; in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}434 \;/\; 1.67 = 858.4 \; in \cdot kips$

Bending capacity in plastic yielding about X axis:

The section is capacity is determined as per F4 in the specification. The design moments for this are from Load Case 4 and occur at mid-span (d = 18.9 in.). The section properties at this location are:

$I_{zz} = 1{,}278 \; in^4$
$I_{yy} = 80.63 \; in^4$
$c = 10.01 \; in$
$h_c = 2 \times (10.01 - 0.84) = 18.35 \; in.$
$S_x = 143.8 \; in^3$
$Z_x = 150.7 \; in^3$

$$M_{yc} = F_y \times S_{yc} = 50 \times 143.8 = 7{,}192 \text{ ksi}$$

$$M_p = F_y \times Z_x = 50 \times 150.7 = 7{,}535 \text{ ksi} \le 1.6\ F_y \times S_x = 11{,}504 \text{ ksi}$$

The moment of inertia of the compression flange about the Y axis $I_{yc} = \frac{0.84(7.5)^3}{12} = 29.53 \; in.^4$, and thus $I_{yc} \,/\, I_y = {}^{29.53}\!/_{80.63} = 0.366 > 0.23.\ \frac{h_c}{t_w} = \frac{18.35}{0.36} = 50.99 < \lambda_{pw} = 90.55$ (limiting slenderness for a compact web per Table B4.1b case 15), so the web plastification factor, R_{pc} is determined as:

$$R_{pc} = \frac{M_p}{M_{yc}} = \frac{7{,}535}{7{,}192} = 1.048 \quad \text{(F4-9a)}$$

Bending capacity in plastic yielding about Y axis:

$$M_n = R_{pc} \times M_{yc} = 1.048 \times 7{,}192 = 7{,}535 \text{ in·kips}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 7{,}535 = 6{,}782 \; in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 7{,}535 \;/\; 1.67 = 4{,}512 \; in \cdot kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 3:

$$\frac{P_r}{P_c} = \frac{5.473}{798.3} = 0.007 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{5.479}{531.4} = 0.010 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.007}{2} + \left(\frac{1{,}225}{6{,}783} + \frac{0}{1{,}290}\right) = 0.184 \text{ (LRFD)}$$

$$\frac{0.010}{2} + \left(\frac{1{,}225}{4{,}513} + \frac{0}{858}\right) = 0.277 \text{ (ASD)}$$

Results

Table 753:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	798.3	799	negligible	
	Major axis bending capacity (in·kips)	6,782	6,780	negligible	
	Minor axis bending capacity (in·kips)	1,290	1,290	none	
	Interaction ratio	0.184	0.184	none	
ASD	Compression capacity (kips)	531.4	532	negligible	
	Major axis bending capacity (in·kips)	4,512	4,510	negligible	
	Minor axis bending capacity (in·kips)	858.4	858	negligible	
	Interaction ratio	0.277	0.277	none	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Tapered I Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 0 0; 4 5 5 0;
MEMBER INCIDENCES
1 1 2; 2 2 4; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY
2 TAPERED 1.5 0.03 1.65 0.75 0.07 0.625 0.07
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -3.5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 4 LOADTYPE None  TITLE LOAD CASE 4
JOINT LOAD
4 FX -50 FZ 45
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CON GY -100
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2010
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
```

```
CODE AISC UNIFIED 2010
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2      TAP ERED                 (AISC SECTIONS)
                           PASS   Sec. G1              0.315         3
                        5.47 C          0.00         275.86        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.927  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.315(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.473E+00(C ) Fy:   5.002E+01     Fz:   0.000E+00               |
| Mx:  -4.497E+00     My:   1.731E-05     Mz:   2.759E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.386E+01     Ayy:   5.875E+00     Cw:  5.508E+03             |
| Szz:   1.200E+02     Syy:   1.792E+01                                 |
| Izz:   1.146E+03     Iyy:   8.062E+01     Ix:  3.514E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.36     N/A     11.88    T.B4.1(a)-2  |
|                Slender        45.33     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.36     9.15    21.15    T.B4.1(b)-11 |
|                Compact        44.73    94.06   137.27    T.B4.1(b)-16 |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield       0.00E+00   8.88E+02   0.000      Eq. D2-1      1     0.00    |
| Rupture     0.00E+00   8.88E+02   0.000      Eq. D2-2      1     0.00    |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck    2.47E+01   8.41E+02   0.029      Eq. E7-1      2     0.00    |
| Min Buck    2.47E+01   7.99E+02   0.031      Eq. E7-1      2     0.00    |
| Flexural                                                                 |
| Tor Buck    2.47E+01   7.93E+02   0.031      Eq. E7-1      2     0.00    |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    1.88E+01     7.88   4.74E+01   4.62E+03   9.35E+02          |
| Min Buck    1.89E+01    29.69   4.50E+01   3.25E+02   8.88E+02          |
| Flexural       Ag        Fcr        Pn                                   |
| Tor Buck    1.89E+01   4.46E+01   8.81E+02                               |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z     1.77E-01   3.74E+02   0.000      Eq. G2-1      4     0.00    |
| Local-Y     5.00E+01   1.59E+02   0.315      Eq. G2-1      3     0.00    |
| Intermediate                                                             |
| Results        Aw         Cv       Kv      h/tw      Vn                 |
| Local-Z     1.39E+01     1.00     1.20    10.71   4.16E+02               |
| Local-Y     5.88E+00     1.00     5.00    45.33   1.76E+02               |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major       1.22E+03   6.78E+03   0.181      Sec. F4.2     3    30.00    |
| Minor       5.30E+00   1.29E+03   0.004      Eq. F6-1      4     0.00    |
| Intermediate   Mn         My                                             |
| Major       7.54E+03   0.00E+00                                          |
| Minor       1.43E+03   0.00E+00                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      -6.55E+02   6.31E+03   0.104      Eq. F4-2      4     0.00    |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb       |
| Major       7.01E+03   0.00E+00    1.00    54.78   242.74    60.00      |
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                            |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
|             1.22E+03   6.78E+03   0.181      Eq. F4-1      3    30.00    |
| Intermediate   Mn         Rpc                                            |
|             7.54E+03   1.05E+00                                          |
|--------------------------------------------------------------------------|
```

```
| CHECK FOR BENDING-TENSION FLANGE YIELDING                              |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
|            1.22E+03   6.78E+03   0.181      Eq. F4-15     3    30.00   |
| Intermediate   Mn        Rpt                                           |
|            7.54E+03   1.18E+00                                         |
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.184      Eq. H1-1b     3    30.00              |
| Flexure Tens         0.181      Eq. H1-1b     3    30.00              |
| Intermediate        Mcx /      Mrx /      Pc /                        |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        6.78E+03   1.22E+03   7.95E+02                    |
|                     1.29E+03   0.00E+00   5.47E+00                    |
| Flexure Tens        6.78E+03   1.22E+03   0.00E+00                    |
|                     1.29E+03   0.00E+00   0.00E+00                    |
|------------------------------------------------------------------------|
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2010
    55. FYLD 50 ALL
    56. FU 60 ALL
    57. METHOD ASD
    58. STP 2 MEMB 2
    59. TRACK 2 MEMB 2
    60. CHECK CODE MEMB 2
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
       2      TAP ERED                 (AISC SECTIONS)
                          PASS   Sec. G1              0.474        3
                       5.47 C          0.00         275.86       0.00
|------------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :     29.927  L/C  :     3                 |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      3    Ratio     :     0.474(PASS)                  |
|          Loc  :    0.00   Condition :   Sec. G1                        |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   5.473E+00(C ) Fy:   5.002E+01     Fz:   0.000E+00               |
| Mx:  -4.497E+00     My:   1.731E-05     Mz:   2.759E+02               |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   1.386E+01     Ayy:   5.875E+00     Cw:  5.508E+03              |
| Szz:   1.200E+02     Syy:   1.792E+01                                  |
| Izz:   1.146E+03     Iyy:   8.062E+01     Ix:  3.514E+00              |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:          50.000   Fu:          60.000                            |
```

```
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.36     N/A     11.88    T.B4.1(a)-2  |
|                Slender        45.33     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         5.36     9.15    21.15    T.B4.1(b)-11 |
|                Compact        44.73    94.06   137.27    T.B4.1(b)-16 |
     STAAD SPACE                                               -- PAGE NO.
11
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   5.91E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   5.92E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.47E+01   5.60E+02   0.044      Eq. E7-1      2     0.00 |
| Min Buck    2.47E+01   5.32E+02   0.046      Eq. E7-1      2     0.00 |
| Flexural                                                              |
| Tor Buck    2.47E+01   5.27E+02   0.047      Eq. E7-1      2     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.88E+01     7.88   4.74E+01   4.62E+03   9.35E+02        |
| Min Buck    1.89E+01    29.69   4.50E+01   3.25E+02   8.88E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.89E+01   4.46E+01   8.81E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     1.77E-01   2.49E+02   0.001      Eq. G2-1      4     0.00 |
| Local-Y     5.00E+01   1.06E+02   0.474      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.39E+01     1.00     1.20    10.71   4.16E+02            |
| Local-Y     5.88E+00     1.00     5.00    45.33   1.76E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       1.22E+03   4.51E+03   0.271      Sec. F4.2     3    30.00 |
| Minor       5.30E+00   8.58E+02   0.006      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.54E+03   0.00E+00                                       |
| Minor       1.43E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
```

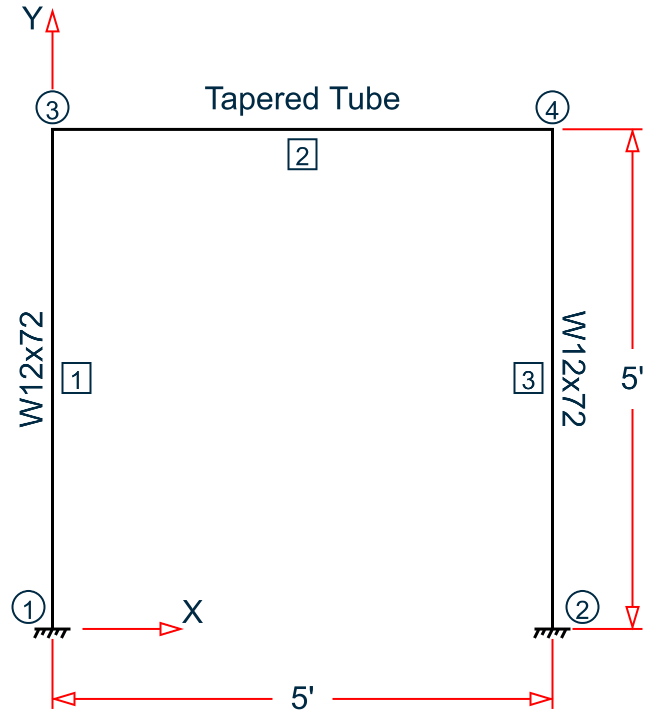
```
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -6.55E+02   4.20E+03   0.156      Eq. F4-2      4     0.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       7.01E+03   0.00E+00    1.00    54.78   242.74    60.00    |
      STAAD SPACE                                              -- PAGE NO.
12
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             1.22E+03   4.51E+03   0.271      Eq. F4-1      3    30.00 |
| Intermediate   Mn        Rpc                                          |
|             7.54E+03   1.05E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-TENSION FLANGE YIELDING                             |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             1.22E+03   4.51E+03   0.271      Eq. F4-15     3    30.00 |
| Intermediate   Mn        Rpt                                          |
|             7.54E+03   1.18E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.277      Eq. H1-1b     3    30.00             |
| Flexure Tens          0.271      Eq. H1-1b     3    30.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         4.51E+03   1.22E+03   5.29E+02                  |
|                      8.58E+02   0.00E+00   5.47E+00                  |
| Flexure Tens         4.51E+03   1.22E+03   0.00E+00                  |
|                      8.58E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

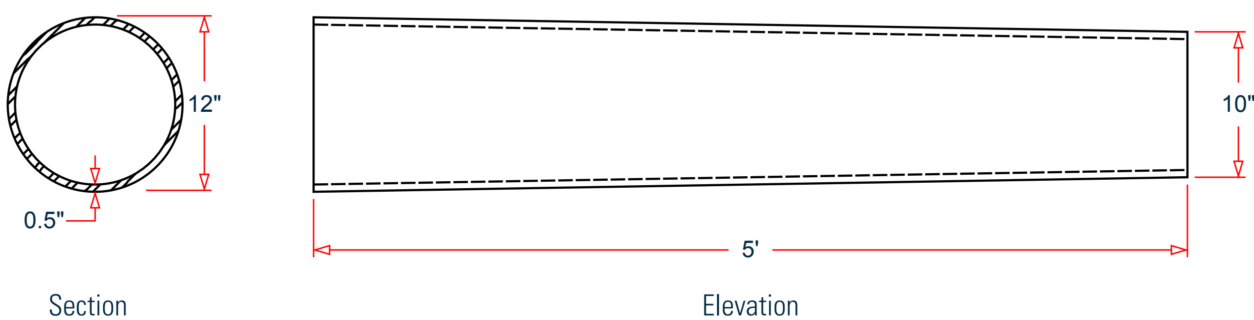
V. AISC 360-10 Tapered Tube Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered tube section member per both the LRFD and ASD methods of the AISC 360-10 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concreted torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the right end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

$M_z = 505.66\ in \cdot kips$
$F_y = 67.2\ kips$
$F_x = 10.23\ kips$

Section Properties

The section at the right end ($OD = 10\ in.$) has the following properties:

Inner diameter, ID = 9 in.
Area, $A_g = (\pi/4)(OD^2 - ID^2) = 14.92\ in^2$
Form factor = $0.49 + 0.8\ (t\ /\ OD) = 0.53$
Shear area, $A_y = 14.92(0.53) = 7.909\ in^2$
Moment of inertia,

$$I = \frac{\pi}{64}\left(\mathrm{OD}^4\text{-}\mathrm{ID}^4\right) = 168.8\ \mathrm{in}^4$$

Elastic section modulus, $S = \frac{\pi}{32}\left(\frac{OD^4\text{-}ID^4}{OD}\right) = 33.76\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{168.8}{14.92}} = 3.363\ in$

Plastic section modulus, $Z = \frac{OD^3\text{-}ID^3}{6} = 45.17\ in^3$

HSS Torsional constant, $C = \frac{\pi}{2}(OD\text{-}t)^2 t = 70.88\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{\mathrm{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1a, Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 63.8$, therefore flanges are non-slender for compression. The web is similarly non-slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{\mathrm{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1b, Case20, $\lambda < \lambda_r = 0.07\frac{E}{F_y} = 40.6$, therefore flange is compact for bending.

Compression Capacity

$$\frac{kL}{r} = \frac{1.0(60)}{3.363} = 17.84 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

$$F_e = \frac{\pi^2 E}{\left(\frac{kL}{r}\right)^2} = \frac{\pi^2 \times 29{,}000}{(17.84)^2} = 899.4\ \mathrm{ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{899.4} = 0.056 < 2.25$$

$$F_{\mathrm{crx}} = 0.658^{F_y/F_e} F_y = 0.658^{(0.056)} 50 = 48.85\ \mathrm{ksi} \qquad \text{(Eq. E3-2)}$$

$$P_n = A_g \times F_{cr} = 14.92 \times 48.85 = 729.0 \text{ kips} \quad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 729.0 = 656.1 \text{ kips}$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 729.0 / 1.67 = 436.5 \text{ kips}$

Shear Capacity

The distance from the maximum shear force (ends) to zero shear force (mid-span), $L_v = L / 2 = 30 \text{ in}$.

The critical shear stress, F_{cr}, *is the minimum of*:

$$F_{cr1} = \frac{1.60E}{\sqrt{\frac{L_v}{OD}}\left(\frac{OD}{t}\right)^{5/4}} = 633.4 \text{ ksi}$$

$$F_{cr2} = \frac{0.78E}{\left(\frac{OD}{t}\right)^{3/2}} = 252.9 \text{ ksi}$$

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30 \text{ ksi}$

Nominal shear capacity:

$$V_n = F_{cr} A_g / 2 = 223.8 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_c V_n = 0.9 \times 223.8 = 201.5 \text{ kips}$
The allowable shear capacity (ASD) = $V_n / \Omega_c = 223.8 / 1.67 = 134.0 \text{ kips}$

Calculate Bending Capacity

Bending capacity in plastic yielding:

$$M_p = F_y \times Z = 50 \times 45.17 = 2{,}259 \text{ in·kips} < 1.6F_y \times S = 1.6 (50) (33.76) = 2{,}701 \text{ in·kips}$$

The bending capacity for flexural yielding:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 2{,}259 = 2{,}033 \text{ in·kips}$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 2{,}259 / 1.67 = 1{,}352 \text{ in·kips}$

Torsional Capacity

The critical stress, F_{cr}, *is the larger of*:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{OD}}\left(\frac{OD}{t}\right)^{5/4}} = 344.3 \text{ ksi}$$

$$F_{cr2} = \frac{1.23E}{\left(\frac{OD}{t}\right)^{3/2}} = 194.5 \text{ ksi}$$

But shall not exceed:

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30\ ksi$

Nominal torsional capacity:

$$T_n = F_{cr}C = 2{,}126 \text{ in·kips}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_c T_n = 0.9 \times 2{,}126 = 1{,}914\ kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_c = 2{,}126\ /\ 1.67 = 1{,}273\ kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{10.23}{656.1} = 0.016 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{10.23}{436.5} = 0.023 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.016}{2} + \left(\frac{505.7}{2{,}033} + \frac{0}{2{,}033}\right) = 0.257 \text{ (LRFD)}$$

$$\frac{0.023}{2} + \left(\frac{505.7}{1{,}352} + \frac{0}{1{,}352}\right) = 0.386 \text{ (ASD)}$$

Results

Table 754:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	656.1	656	negligible	
	Shear capacity (kips)	201.5	201	negligible	
	Bending capacity (in·kips)	2,033	2,030	negligible	
	Torsional capacity (in·kips)	1,914	1,910	negligible	
	Interaction ratio	0.257	0.257	none	
ASD	Compression capacity (kips)	436.5	437	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Shear capacity (kips)	134.0	134	negligible	
	Bending capacity (in·kips)	1,352	1,350	negligible	
	Torsional capacity (in·kips)	1,273	1,270	negligible	
	Interaction ratio	0.386	0.386	none	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Tapered Tube Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 PRIS ROUND STA 12 END 10 THI 0.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
```

```
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=    10: 2:26                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: AISC 360-10 Tapered Tube Section.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 13-APR-21
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT INCHES KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0
     9. MEMBER INCIDENCES
```

```
    10. 1 1 2; 2 2 3; 3 3 4
    11. DEFINE MATERIAL START
    12. ISOTROPIC STEEL
    13. E 29000
    14. POISSON 0.3
    15. DENSITY 0.000283
    16. ALPHA 6.5E-06
    17. DAMP 0.03
    18. G 11200
    19. TYPE STEEL
    20. STRENGTH RY 1.5 RT 1.2
    21. END DEFINE MATERIAL
    22. MEMBER PROPERTY AMERICAN
    23. 1 TABLE ST W12X72
    24. 3 TABLE ST W12X72
    25. MEMBER PROPERTY
    26. 2 PRIS ROUND STA 12 END 10 THI 0.5
    27. CONSTANTS
    28. MATERIAL STEEL ALL
    29. SUPPORTS
    30. 1 FIXED
    31. 4 FIXED
    32. LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
    33. MEMBER LOAD
    34. 2 UNI GY -2.25
    35. LOAD 2 LOADTYPE NONE  TITLE LOAD CASE 2
    36. JOINT LOAD
    37. 2 FX 50 FZ 25
    38. LOAD 3 LOADTYPE NONE  TITLE LOAD CASE 3
      STAAD SPACE                                                -- PAGE NO.
2
    39. MEMBER LOAD
    40. 2 CMOM GX 0.75
    41. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =      12
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    42. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                                -- PAGE NO.
3
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00162  -0.00665   0.00000   0.00000   0.00000  -0.00075
           2    0.07748   0.00168   0.25417   0.00541   0.00296  -0.00079
           3    0.00000   0.00000   0.00013   0.00000   0.00000   0.00000
      3    1    0.00033  -0.00659   0.00000   0.00000   0.00000   0.00069
           2    0.07448  -0.00168   0.07663   0.00255   0.00296  -0.00085
```

```
          3    0.00000   0.00000   0.00011   0.00000   0.00000   0.00000
      4   1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      STAAD SPACE                                              -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     10.23     67.81      0.00      0.00      0.00    -89.36
           2    -26.14    -17.09    -24.95  -1258.27     -1.62   1013.33
           3      0.00      0.00      0.00     -0.39     -0.00      0.00
      4    1    -10.23     67.19      0.00      0.00      0.00    108.13
           2    -23.86     17.09     -0.05   -241.73     -1.62    961.14
           3      0.00      0.00     -0.00     -0.36     -0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     67.81    -10.23     0.00      0.00      0.00     -89.36
                 2    -67.81     10.23     0.00      0.00      0.00    -524.44
           2     1    -17.09     26.14   -24.95     -1.62   1258.27    1013.33
                 2     17.09    -26.14    24.95      1.62    238.50     554.84
           3     1      0.00      0.00     0.00     -0.00      0.39       0.00
                 2      0.00      0.00    -0.00      0.00     -0.39       0.00
      2    1     2     10.23     67.81     0.00      0.00      0.00     524.44
                 3    -10.23     67.19     0.00      0.00      0.00    -505.66
           2     2     23.86    -17.09     0.05    238.50     -1.62    -554.84
                 3    -23.86     17.09    -0.05   -238.50     -1.62    -470.69
           3     2      0.00      0.00     0.00     -0.39     -0.00       0.00
                 3      0.00      0.00    -0.00     -0.36     -0.00       0.00
      3    1     3     67.19     10.23     0.00      0.00      0.00     505.66
                 4    -67.19    -10.23     0.00      0.00      0.00     108.13
           2     3     17.09     23.86     0.05     -1.62    238.50     470.69
                 4    -17.09    -23.86    -0.05      1.62   -241.73     961.14
           3     3      0.00      0.00     0.00     -0.00      0.36       0.00
                 4      0.00      0.00    -0.00      0.00     -0.36       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    43. PARAMETER 1
    44. CODE AISC UNIFIED 2010
    45. SLF 0.8 MEMB 2
    46. FYLD 50 ALL
    47. FU 60 ALL
    48. METHOD LRFD
    49. STP 2 MEMB 2
    50. TRACK 2 MEMB 2
    51. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
```

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       2      PRISMAT                  (AISC SECTIONS)
                           PASS   Sec. G1              0.334         1
                       10.23 C          0.00         505.66       60.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.839  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :    60.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.334(PASS)                |
|          Loc  :   60.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.023E+01(C ) Fy:  -6.719E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.057E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.909E+00     Ayy:   7.909E+00     Cw:  0.000E+00             |
| Szz:   3.376E+01     Syy:   3.376E+01                                 |
| Izz:   1.688E+02     Iyy:   1.688E+02     Ix:  3.376E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
|                Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
| Flexure     :  Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
|                Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   8.13E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   6.50E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    2.39E+01   6.56E+02   0.036      Eq. E3-1      2    60.00 |
| Min Buck    2.39E+01   6.56E+02   0.036      Eq. E3-1      2    60.00 |
| Intermediate                                                          |
```

```
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02        |
| Min Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    -5.39E-02   2.01E+02   0.000      Eq. G6-1      2    60.00 |
| Local-Y    -6.72E+01   2.01E+02   0.334      Eq. G6-1      1    60.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.49E+01     0.00     0.00     0.00   2.24E+02            |
| Local-Y     1.49E+01     0.00     0.00     0.00   2.24E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             2.38E+02   1.91E+03   0.125      Eq. H3-1      2    60.00 |
| Intermediate Fcr        Tn                                            |
|             3.00E+01   2.13E+03                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -5.06E+02   2.03E+03   0.249      Eq. F8-1      1    60.00 |
| Minor       1.62E+00   2.03E+03   0.001      Eq. F8-1      2    60.00 |
| Intermediate   Mn        My                                           |
| Major       2.26E+03   0.00E+00                                      |
| Minor       2.26E+03   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.257      Eq. H1-1b     1    60.00            |
| Flexure Tens          0.249      Eq. H1-1b     1    60.00            |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         2.03E+03  -5.06E+02   6.56E+02                  |
|                      2.03E+03   0.00E+00   1.02E+01                  |
| Flexure Tens         2.03E+03  -5.06E+02   0.00E+00                  |
|                      2.03E+03   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2010
    54. SLF 0.8 MEMB 2
    55. FYLD 50 ALL
    56. FU 60 ALL
    57. METHOD ASD
    58. STP 2 MEMB 2
    59. TRACK 2 MEMB 2
    60. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
```

```
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      2      PRISMAT                  (AISC SECTIONS)
                           PASS   Sec. G1              0.501         1
                       10.23 C          0.00         505.66       60.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     17.839  L/C  :     1                |
 | Allowable Slenderness Ratio :    200.000  LOC  :    60.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      1    Ratio     :     0.501(PASS)                 |
 |          Loc  :   60.00   Condition :   Sec. G1                       |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   1.023E+01(C ) Fy:  -6.719E+01     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   5.057E+02               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   7.909E+00     Ayy:   7.909E+00     Cw:  0.000E+00              |
 | Szz:   3.376E+01     Syy:   3.376E+01                                 |
 | Izz:   1.688E+02     Iyy:   1.688E+02     Ix:  3.376E+02              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          60.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        60.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
 |                Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
 | Flexure     :  Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
 |                Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   5.41E+02   0.000      Eq. D2-1      1     0.00  |
 | Rupture    0.00E+00   4.34E+02   0.000      Eq. D2-2      1     0.00  |
 |-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.39E+01   4.37E+02   0.055      Eq. E3-1      2    60.00  |
| Min Buck    2.39E+01   4.37E+02   0.055      Eq. E3-1      2    60.00  |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr         Fe         Pn            |
| Maj Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02         |
| Min Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02         |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    -5.39E-02   1.34E+02   0.000      Eq. G6-1      2    60.00  |
| Local-Y    -6.72E+01   1.34E+02   0.501      Eq. G6-1      1    60.00  |
| Intermediate                                                             |
| Results        Aw         Cv        Kv      h/tw      Vn                |
| Local-Z     1.49E+01     0.00      0.00     0.00   2.24E+02             |
| Local-Y     1.49E+01     0.00      0.00     0.00   2.24E+02             |
|--------------------------------------------------------------------------|
| CHECK FOR TORSION                                                        |
|                                                                          |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             2.38E+02   1.27E+03   0.187      Eq. H3-1      2    60.00  |
| Intermediate Fcr        Tn                                               |
|             3.00E+01   2.13E+03                                         |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      -5.06E+02   1.35E+03   0.374      Eq. F8-1      1    60.00  |
| Minor       1.62E+00   1.35E+03   0.001      Eq. F8-1      2    60.00  |
| Intermediate   Mn         My                                             |
| Major       2.26E+03   0.00E+00                                         |
| Minor       2.26E+03   0.00E+00                                         |
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                      RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp         0.386      Eq. H1-1b     1    60.00                |
| Flexure Tens         0.374      Eq. H1-1b     1    60.00                |
| Intermediate        Mcx /      Mrx /      Pc /                           |
|                     Mcy        Mry        Pr                             |
| Flexure Comp        1.35E+03  -5.06E+02   4.37E+02                       |
|                     1.35E+03   0.00E+00   1.02E+01                       |
| Flexure Tens        1.35E+03  -5.06E+02   0.00E+00                       |
|                     1.35E+03   0.00E+00   0.00E+00                       |
|--------------------------------------------------------------------------|
    61. FINISH
              *********** END OF THE STAAD.Pro RUN ***********
                **** DATE= MAR 24,2022   TIME= 10: 2:27 ****
     STAAD SPACE                                                -- PAGE NO.
12
         **********************************************************
```

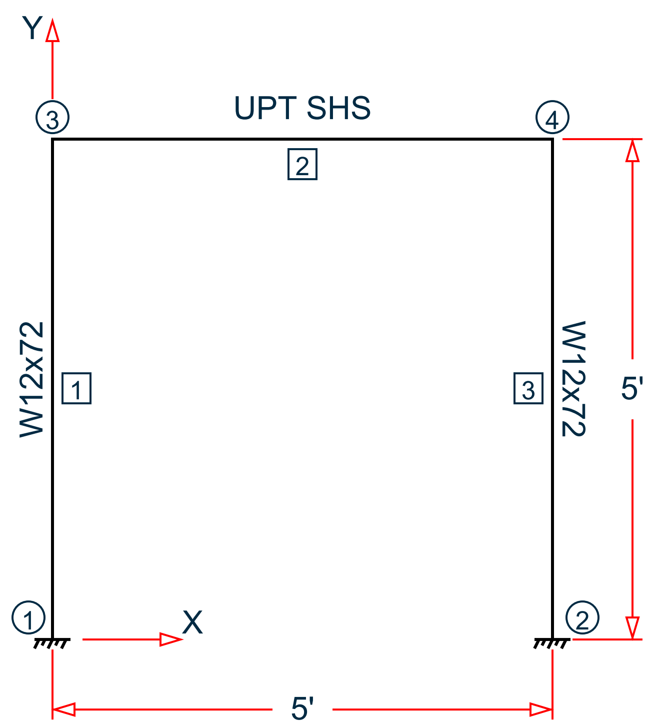
```
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

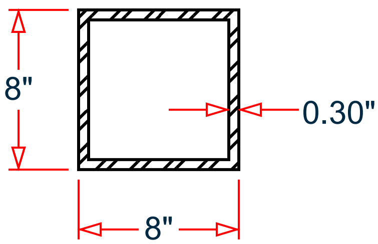
V. AISC 360-10 UPT Square Hollow Section

Verify the axial compression capacity, shear capacity, flexure capacity, torsional capacity, and interaction ratio of a user-provided hollow square section member per both the LRFD and ASD methods of the AISC 360-10 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT SHS member for the beam. The section outer depth, D, is 8" and the thickness, t, is 0.30".





Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner depth, $d = 8 - 2\times0.30 = 7.40$ in.

Area, $A_g = D^2 - d^2 = 9.24\ in^2$

Shear area, $A_w = 2\times D\times t_w = 4.80\ in^2$

Moment of inertia, $I = \frac{1}{12}\left(D^4 - d^4\right) = 91.45\ in^4$

Elastic section modulus, $S = \frac{I}{D / 2} = \frac{91.45}{8 / 2} = 22.86\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{91.45}{9.24}} = 3.146\ in$

Plastic section modulus, $Z = \frac{1}{6}\left(D^3 - d^3\right) = 26.69\ in^3$

HSS Torsional constant, $C = 2\left(B - t_w\right)\left(D - t_w\right)t_w - 4.5\left(4 - \pi\right)t_w{}^3 = 35.47\ in^3$

Section Classification

The design thickness, $t = 0.93\times t_w = 0.279\ in$

The design depth, $d = D - 3t = 7.163\ in$

Flange in compression

$$\lambda = \frac{d}{t} = \frac{7.163}{0.279} = 25.67$$

Per Table 4.1a Case 6, $\lambda < \lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29,000}{50}} = 33.72$, therefore flanges are non-slender for compression. Similar for the web of the square section.

Flexure classification of flange:

Per Table 4.1b, Case 17, $\lambda < \lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29,000}{50}} = 26.97$, therefore flange is compact for bending.

Flexure classification of web:

Per Table 4.1b, Case 19, $\lambda < \lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29,000}{50}} = 58.28$, therefore web is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.146} = 19.07 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,000}{(19.07)^2} = 787 \text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{787} = 0.064 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y/F_e} F_y = 0.658^{(0.064)} 50 = 48.68 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 9.24 \times 48.68 = 449.9 \text{ kips} \qquad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 449.9 = 404.9$ *kips*

The allowable compression capacity (ASD) = $P_n / \Omega_c = 449.9 \ / \ 1.67 = 269.4$ *kips*

Calculate Shear Capacity The nominal shear strength:

$$V_n = 0.6 \times F_y \times A_w \times C_{v2} \qquad \text{(Eq. G4-1)}$$

where

C_{v2} = 1.0 per Section G2.2 with $h/t = 25.67$ and $K_v = 5$.

$$V_n = 0.6\ (50)\ (4.80)\ (1.0) = 144 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 144 = 129.6$ *kips*

The allowable shear capacity (ASD) = $V_n / \Omega_v = 144 \ / \ 1.67 = 86.23 \ kips$

Calculate Bending Capacity

Bending capacity in plastic yielding (compact section):

$$M_p = F_y \times Z_x = 50 \times 26.69 = 1{,}335 \text{ in·kips} \qquad \text{(Eq F7-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}335 = 1{,}201 \ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}335 \ / \ 1.67 = 799.2 \ in \cdot kips$

Torsional Capacity

$$\frac{h}{t} = 25.67 < 2.45\sqrt{\frac{E}{F_y}} = 59.0$$

$$F_{cr} = 0.6F_y = 30 \text{ ksi}$$

Nominal torsional capacity:

$$T_n = F_{cr} \times C = 1{,}064 \text{ ksi}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1{,}064 = 957.7 \ kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 1{,}064 \ / \ 1.67 = 637.2 \ kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.80}{404.9} = 0.029 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.80}{269.4} = 0.044 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0$.

$$\frac{0.029}{2} + \left(\frac{599.7}{1{,}201} + \frac{0}{1{,}201}\right) = 0.514 \text{ (LRFD)}$$

$$\frac{0.044}{2} + \left(\frac{599.7}{799.2} + \frac{0}{799.2}\right) = 0.765 \text{ (ASD)}$$

Results

Table 755:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	404.9	405	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Bending capacity (in·kips)	1,201	1,200	negligible	
	Shear capacity (kips)	129.6	130	negligible	
	Torsional capacity (in·kips)	957.7	958	negligible	
	Interaction ratio	0.514	0.514	negligible	
ASD	Compression capacity (kips)	269.4	269	negligible	
	Bending capacity (in·kips)	799.2	799	negligible	
	Shear capacity (kips)	86.23	86.2	negligible	
	Torsional capacity (in·kips)	637.2	637	negligible	
	Interaction ratio	0.765	0.772	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 UPT Square Hollow Section.STD` is typically installed with the program.

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded SHS member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
```

```
TUBE
SHS
9.24 8 8 0.3 91.4452 91.4452 136.96 4.8 4.8
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 SHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             **************************************************
             *                                                *
             *           STAAD.Pro CONNECT Edition            *
             *           Version  22.10.00.***                *
             *           Proprietary Program of               *
             *           Bentley Systems, Inc.                *
             *           Date=    MAR 24, 2022                *
             *           Time=    10: 2:32                    *
             *                                                *
             *  Licensed to: Bentley Systems Inc              *
             **************************************************
     1. STAAD SPACE
INPUT FILE: AISC 360-10 UPT Square Hollow Section.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 13-APR-21
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT INCHES KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0
     9. MEMBER INCIDENCES
    10. 1 1 2; 2 2 3; 3 3 4
    11. START USER TABLE
    12. TABLE 1
    13. UNIT INCHES KIP
    14. TUBE
    15. SHS
    16. 9.24 8 8 0.3 91.4452 91.4452 136.96 4.8 4.8
    17. END
    18. DEFINE MATERIAL START
    19. ISOTROPIC STEEL
    20. E 29000
    21. POISSON 0.3
    22. DENSITY 0.000283
    23. ALPHA 6.5E-06
    24. DAMP 0.03
    25. G 11200
    26. TYPE STEEL
    27. STRENGTH RY 1.5 RT 1.2
    28. END DEFINE MATERIAL
    29. MEMBER PROPERTY AMERICAN
    30. 1 TABLE ST W12X72
    31. 3 TABLE ST W12X72
    32. MEMBER PROPERTY
    33. 2 UPTABLE 1 SHS
    34. CONSTANTS
    35. MATERIAL STEEL ALL
    36. SUPPORTS
    37. 1 FIXED
    38. 4 FIXED
      STAAD SPACE                                              -- PAGE NO.
2
    39. LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
    40. MEMBER LOAD
```

```
    41. 2 UNI GY -2.25
    42. LOAD 2 LOADTYPE NONE  TITLE LOAD CASE 2
    43. JOINT LOAD
    44. 2 FX 50 FZ 25
    45. LOAD 3 LOADTYPE NONE  TITLE LOAD CASE 3
    46. MEMBER LOAD
    47. 2 CMOM GX 0.75
    48. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =      12
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    49. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                              -- PAGE NO.
3
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00132  -0.00662   0.00000   0.00000   0.00000  -0.00085
           2    0.09809   0.00107   0.28808   0.00654   0.00408  -0.00152
           3    0.00000   0.00000   0.00012   0.00000   0.00000   0.00000
      3    1   -0.00132  -0.00662   0.00000   0.00000   0.00000   0.00085
           2    0.09263  -0.00107   0.04272   0.00141   0.00408  -0.00142
           3    0.00000   0.00000   0.00012   0.00000   0.00000   0.00000
      4    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      STAAD SPACE                                              -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     11.80     67.50      0.00      0.00      0.00   -108.35
           2    -25.61    -10.88    -24.93  -1364.43     -2.23   1206.26
           3      0.00      0.00      0.00     -0.38      0.00      0.00
      4    1    -11.80     67.50      0.00      0.00      0.00    108.35
           2    -24.39     10.88     -0.07   -135.57     -2.23   1141.12
           3      0.00      0.00      0.00     -0.38      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     67.50    -11.80     0.00      0.00      0.00    -108.35
                 2    -67.50     11.80     0.00      0.00      0.00    -599.73
           2     1    -10.88     25.61   -24.93     -2.23   1364.43    1206.26
                 2     10.88    -25.61    24.93      2.23    131.11     330.63
           3     1      0.00      0.00    -0.00     -0.00      0.38       0.00
```

```
                 2      0.00      0.00      0.00      0.00     -0.38       0.00
      2    1     2     11.80     67.50      0.00      0.00      0.00     599.73
                 3    -11.80     67.50      0.00      0.00      0.00    -599.73
           2     2     24.39    -10.88      0.07    131.11     -2.23    -330.63
                 3    -24.39     10.88     -0.07   -131.11     -2.23    -321.98
           3     2      0.00      0.00      0.00     -0.38     -0.00       0.00
                 3      0.00      0.00     -0.00     -0.38      0.00       0.00
      3    1     3     67.50     11.80      0.00      0.00      0.00     599.73
                 4    -67.50    -11.80      0.00      0.00      0.00     108.35
           2     3     10.88     24.39      0.07     -2.23    131.11     321.98
                 4    -10.88    -24.39     -0.07      2.23   -135.57    1141.12
           3     3      0.00      0.00     -0.00     -0.00      0.38       0.00
                 4      0.00      0.00      0.00      0.00     -0.38       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    50. PARAMETER 1
    51. CODE AISC UNIFIED 2010
    52. SLF 0.8 MEMB 2
    53. FYLD 50 ALL
    54. FU 60 ALL
    55. METHOD LRFD
    56. STP 2 MEMB 2
    57. TRACK 2 MEMB 2
    58. CHECK CODE MEMB 2
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   SHS                     (UPT)
                           PASS   Sec. G1             0.521        1
                      11.80 C          0.00        599.73        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     19.072  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.521(PASS)                |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.180E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.997E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.800E+00     Ayy:   4.800E+00     Cw:  0.000E+00             |
| Szz:   2.286E+01     Syy:   2.286E+01                                |
| Izz:   9.145E+01     Iyy:   9.145E+01     Ix:  1.370E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
```

```
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                                |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A         |
|                Non-Slender    25.67     N/A     33.72    T.B4.1(a)-6   |
| Flexure     :  Compact        25.67    26.97    33.72    T.B4.1(b)-17  |
|                Compact        25.67    58.28   137.27    T.B4.1(b)-19  |
     STAAD SPACE                                             -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   4.16E+02   0.000      Eq. D2-1      1     0.00  |
| Rupture     0.00E+00   3.33E+02   0.000      Eq. D2-2      1     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    2.44E+01   4.05E+02   0.060      Eq. E3-1      2     0.00  |
| Min Buck    2.44E+01   4.05E+02   0.060      Eq. E3-1      2     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr         Fe         Pn            |
| Maj Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02          |
| Min Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02          |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z    -7.44E-02   1.30E+02   0.001      Eq. G2-1      2     0.00  |
| Local-Y     6.75E+01   1.30E+02   0.521      Eq. G2-1      1     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn                |
| Local-Z     4.80E+00     1.00      5.00    25.67   1.44E+02             |
| Local-Y     4.80E+00     1.00      5.00    25.67   1.44E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
|             1.31E+02   9.58E+02   0.137      Eq. H3-1      2     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.00E+01   1.06E+03                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major      -6.00E+02   1.20E+03   0.499      Eq. F7-1      1     0.00  |
| Minor      -2.23E+00   1.20E+03   0.002      Eq. F7-1      2     0.00  |
| Intermediate   Mn         My                                            |
| Major       1.33E+03   0.00E+00                                        |
| Minor       1.33E+03   0.00E+00                                        |
     STAAD SPACE                                             -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
```

```
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.514      Eq. H1-1b     1     0.00              |
| Flexure Tens         0.499      Eq. H1-1b     1     0.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        1.20E+03  -6.00E+02   4.05E+02                    |
|                     1.20E+03   0.00E+00   1.18E+01                    |
| Flexure Tens        1.20E+03  -6.00E+02   3.33E+02                    |
|                     1.20E+03   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
    59. PARAMETER 2
    60. CODE AISC UNIFIED 2010
    61. SLF 0.8 MEMB 2
    62. FYLD 50 ALL
    63. FU 60 ALL
    64. METHOD ASD
    65. STP 2 MEMB 2
    66. TRACK 2 MEMB 2
    67. CHECK CODE MEMB 2
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   SHS                      (UPT)
                          PASS   Sec. G1             0.783         1
                      11.80 C          0.00        599.73        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     19.072  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.783(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.180E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.997E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.800E+00     Ayy:   4.800E+00     Cw:  0.000E+00             |
| Szz:   2.286E+01     Syy:   2.286E+01                                 |
| Izz:   9.145E+01     Iyy:   9.145E+01     Ix:  1.370E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
```

```
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A          |
|                Non-Slender    25.67     N/A     33.72    T.B4.1(a)-6    |
| Flexure     :  Compact        25.67    26.97    33.72    T.B4.1(b)-17   |
|                Compact        25.67    58.28   137.27    T.B4.1(b)-19   |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield       0.00E+00   2.77E+02   0.000      Eq. D2-1      1     0.00   |
| Rupture     0.00E+00   2.22E+02   0.000      Eq. D2-2      1     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck    2.44E+01   2.69E+02   0.091      Eq. E3-1      2     0.00   |
| Min Buck    2.44E+01   2.69E+02   0.091      Eq. E3-1      2     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r       Fcr        Fe          Pn           |
| Maj Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02          |
| Min Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02          |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z    -7.44E-02   8.62E+01   0.001      Eq. G2-1      2     0.00   |
| Local-Y     6.75E+01   8.62E+01   0.783      Eq. G2-1      1     0.00   |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn               |
| Local-Z     4.80E+00     1.00      5.00    25.67   1.44E+02            |
| Local-Y     4.80E+00     1.00      5.00    25.67   1.44E+02            |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
|             1.31E+02   6.37E+02   0.206      Eq. H3-1      2     0.00   |
| Intermediate Fcr        Tn                                              |
|             3.00E+01   1.06E+03                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      -6.00E+02   7.99E+02   0.750      Eq. F7-1      1     0.00   |
| Minor      -2.23E+00   7.99E+02   0.003      Eq. F7-1      2     0.00   |
| Intermediate   Mn         My                                            |
| Major       1.33E+03   0.00E+00                                         |
| Minor       1.33E+03   0.00E+00                                         |
     STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
```

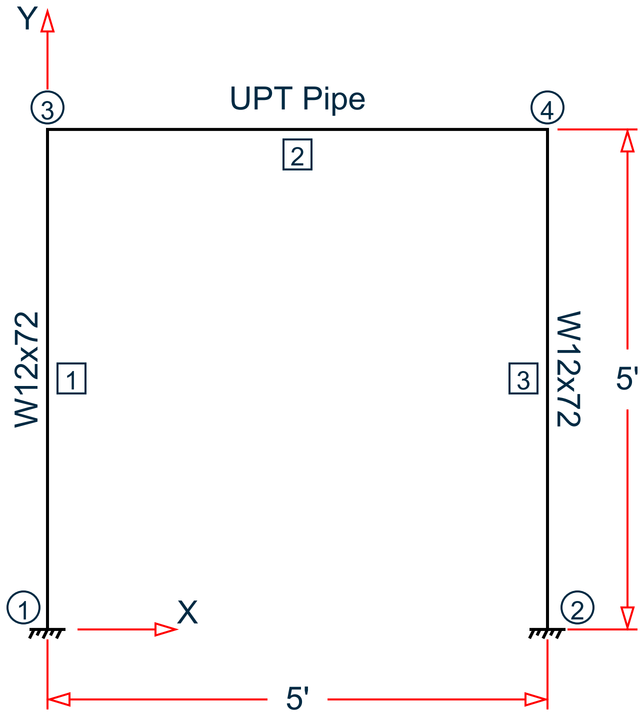
```
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          0.772      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.750      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         7.99E+02   3.31E+02   2.69E+02                   |
|                      7.99E+02  -2.23E+00   2.44E+01                   |
| Flexure Tens         7.99E+02   3.31E+02   2.22E+02                   |
|                      7.99E+02  -2.23E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flx Tor Comp Shr      0.618      Eq. H3-6      2     0.00             |
| Flx Tor Tens Shr      0.527      Eq. H3-6      2     0.00             |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /          |
|                      Vcx /      Vcy /      Vrx /      Vry /          |
|                      Tc         Tr         Pc         Pr             |
| Flx Tor Comp Shr     7.99E+02   7.99E+02   3.31E+02  -2.23E+00       |
|                      8.62E+01   8.62E+01  -7.44E-02  -1.09E+01       |
|                      6.37E+02   1.31E+02   2.69E+02   2.44E+01       |
| Flx Tor Tens Shr     7.99E+02   7.99E+02   3.31E+02  -2.23E+00       |
|                      8.62E+01   8.62E+01  -7.44E-02  -1.09E+01       |
|                      6.37E+02   1.31E+02   2.22E+02   0.00E+00       |
|-----------------------------------------------------------------------|
    68. FINISH
      STAAD SPACE                                                -- PAGE NO.
12
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME= 10: 2:33 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

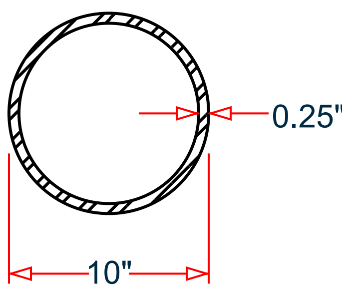
V. AISC 360-10 UPT Pipe Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered I section member per both the LRFD and ASD methods of the AISC 360-10 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT pipe member for the beam. The pipe outer diameter, D, is 10" and the thickness, t, is 0.25".





Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner diameter, $d = 10 - 2\times 0.25 = 9.5\ in.$
Area, $A_g = \frac{\pi}{4}\left(D^2 - d^2\right) = 7.66\ in^2$
Shear area, $A_y = A_x = A_g\ /\ 2 = 3.83\ in^2$
Moment of inertia, $I_x = I_y = \frac{\pi}{64}\left(D^4 - d^4\right) = 91.05\ in^4$
Elastic section modulus, $S_{xx} = \frac{I_{xx}}{D\ /\ 2} = \frac{91.05}{10\ /\ 2} = 18.21\ in^3$
Radius of gyration, $r_x = \sqrt{\frac{I_{xx}}{A_g}} = \sqrt{\frac{91.05}{7.66}} = 3.448\ in$
Plastic section modulus, $Z_x = \frac{1}{6}\left(D^3 - d^3\right) = 23.77\ in^3$
HSS Torsional constant, $C = \frac{\pi}{2}(D - t)^2 t = 37.33\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{D}{t} = \frac{10}{0.25} = 40$$

Per Table 4.1a Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29,000}{50} = 63.8$, therefore flanges are non-slender for compression.

Web in compression

Per Table 4.1a, Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29,000}{50} = 63.8$, therefore the web is non-slender for compression.

Flexure classification:

Per Table 4.1b, Case 20, $\lambda < \lambda_p = 0.07\frac{E}{F_y} = 0.11\frac{29,000}{50} = 40.6$, therefore flange is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.448} = 17.40 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(17.40)^2} = 945\text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{945} = 0.053 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y / F_e} F_y = 0.658^{(0.053)} 50 = 48.91 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 7.66 \times 48.91 = 374.5 \text{ kips} \qquad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 374.5 = 337.0 \text{ kips}$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 374.5 \; / \; 1.67 = 224.3 \text{ kips}$

<u>Calculate Shear Capacity</u>

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{D}}\left(\frac{D}{t}\right)^{\left(5/4\right)}} = 144.8 \text{ ksi}$$

$$F_{cr2} = \frac{0.78E}{\left(\frac{D}{t}\right)^{\left(3/2\right)}} = 89.4 \text{ ksi}$$

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

So, $F_{cr} = 30 \text{ ksi}$.

The nominal shear strength:

$$V_n = F_{cr} \times A_x = 30 \times 3.83 = 114.9 \text{ kips}$$

The axial capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 114.9 = 103.4 \text{ kips}$
The allowable shear capacity (ASD) = $V_n / \Omega_v = 114.9 \; / \; 1.67 = 68.78 \text{ kips}$

<u>Calculate Bending Capacity</u>

Bending capacity in plastic yielding (compact section):

$$M_y = F_y \times Z_x = 50 \times 23.77 = 1{,}189 \text{ in}\cdot\text{kips} \qquad \text{(Eq F8-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}189 = 1{,}070 \text{ in}\cdot\text{kips}$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}189 \; / \; 1.67 = 711.7 \text{ in}\cdot\text{kips}$

<u>Interaction Ratio for Bending and Compression</u>

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.75}{337.0} = 0.034 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.75}{224.3} = 0.052 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \le 1.0$.

$$\frac{0.034}{2} + \left(\frac{599.6}{1{,}070} + \frac{0}{1{,}070}\right) = 0.578 \text{ (LRFD)}$$

$$\frac{0.052}{2} + \left(\frac{599.6}{711.7} + \frac{0}{711.7}\right) = 0.869 \text{ (ASD)}$$

Calculate Torsion Capacity

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}. From shear calculations, $F_{cr} = 30\ ksi$.

The nominal shear strength:

$$T_n = F_{cr} \times C = 30 \times 37.33 = 1{,}120 \text{ in·kips}$$

The axial capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1{,}120 = 1{,}008$ *in·kips*
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 114.9\ /\ 1.67 = 670.6$ *in·kips*

Results

Table 756:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	337	337	none	
	Bending capacity (in·kips)	1,070	1,070	none	
	Shear capacity (kips)	103.4	103	negligible	
	Interaction ratio	0.578	0.578	none	
	Torsional capacity (in·kips)	1,008	1,010	negligible	
ASD	Compression capacity (kips)	224.3	224.3	none	
	Bending capacity (in·kips)	711.7	712	negligible	
	Shear capacity (kips)	68.78	68.8	negligible	
	Interaction ratio	0.869	0.869	none	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Torsional capacity (in·kips)	670.6	671	negligible	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded pipe member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 UPT Pipe Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
PIPE
PIPE
10 9.5 0 0
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 PIPE
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   PIPE                     (UPT)
                           PASS   Sec. G1              0.653         1
                       11.75 C         0.00         599.60        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.400  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.653(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.175E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.996E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   3.829E+00     Ayy:   3.829E+00     Cw:  0.000E+00              |
| Szz:   1.821E+01     Syy:   1.821E+01                                 |
| Izz:   9.105E+01     Iyy:   9.105E+01     Ix:  1.821E+02              |
```

```
 |-----------------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                         |
 | Fyld:          50.000   Fu:          60.000                                 |
 |-----------------------------------------------------------------------------|
 | Actual Member Length:        60.000                                         |
 | Design Parameters                                                           |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00              |
 |-----------------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE             |
 |                STIFFENED                                                    |
 | Compression :  Non-Slender    40.00     N/A     63.80    T.B4.1(a)-9        |
 |                Non-Slender    40.00     N/A     63.80    T.B4.1(a)-9        |
 | Flexure     :  Compact        40.00    40.60   179.80    T.B4.1(b)-20       |
 |                Compact        40.00    40.60   179.80    T.B4.1(b)-20       |
     STAAD SPACE                                                -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                                     |
 |                                                                             |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
 | Yield       0.00E+00   3.45E+02   0.000      Eq. D2-1      1     0.00       |
 | Rupture     0.00E+00   2.76E+02   0.000      Eq. D2-2      1     0.00       |
 |-----------------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                                 |
 |                                                                             |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
 | Maj Buck    2.43E+01   3.37E+02   0.072      Eq. E3-1      2     0.00       |
 | Min Buck    2.43E+01   3.37E+02   0.072      Eq. E3-1      2     0.00       |
 | Intermediate                                                                |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn                 |
 | Maj Buck    7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02              |
 | Min Buck    7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02              |
 |-----------------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                             |
 |                                                                             |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
 | Local-Z    -6.96E-02   1.03E+02   0.001      Eq. G6-1      2     0.00       |
 | Local-Y     6.75E+01   1.03E+02   0.653      Eq. G6-1      1     0.00       |
 | Intermediate                                                                |
 | Results        Aw        Cv       Kv      h/tw      Vn                      |
 | Local-Z     7.66E+00     0.00     0.00     0.00   1.15E+02                  |
 | Local-Y     7.66E+00     0.00     0.00     0.00   1.15E+02                  |
 |-----------------------------------------------------------------------------|
 | CHECK FOR TORSION                                                           |
 |                                                                             |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
 |             1.56E+02   1.01E+03   0.155      Eq. H3-1      2     0.00       |
 | Intermediate Fcr        Tn                                                  |
 |             3.00E+01   1.12E+03                                             |
 |-----------------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                                  |
 |                                                                             |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
 | Major      -6.00E+02   1.07E+03   0.561      Eq. F8-1      1     0.00       |
 | Minor      -2.09E+00   1.07E+03   0.002      Eq. F8-1      2     0.00       |
```

```
| Intermediate   Mn        My                                           |
| Major       1.19E+03   0.00E+00                                       |
| Minor       1.19E+03   0.00E+00                                       |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.578      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.561      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         1.07E+03  -6.00E+02   3.37E+02                   |
|                      1.07E+03   0.00E+00   1.17E+01                   |
| Flexure Tens         1.07E+03  -6.00E+02   2.76E+02                   |
|                      1.07E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2010
    59. SLF 0.8 MEMB 2
    60. FYLD 50 ALL
    61. FU 60 ALL
    62. METHOD ASD
    63. STP 2 MEMB 2
    64. TRACK 2 MEMB 2
    65. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   PIPE                    (UPT)
                           PASS   Sec. G1              0.981        1
                       11.75 C         0.00         599.60        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.400  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.981(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.175E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.996E+02                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   3.829E+00     Ayy:   3.829E+00     Cw:  0.000E+00                |
| Szz:   1.821E+01     Syy:   1.821E+01                                   |
| Izz:   9.105E+01     Iyy:   9.105E+01     Ix:  1.821E+02                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:          50.000   Fu:          60.000                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        60.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                                |
| Compression :  Non-Slender    40.00     N/A     63.80    T.B4.1(a)-9   |
|                Non-Slender    40.00     N/A     63.80    T.B4.1(a)-9   |
| Flexure     :  Compact        40.00    40.60   179.80    T.B4.1(b)-20  |
|                Compact        40.00    40.60   179.80    T.B4.1(b)-20  |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   2.29E+02   0.000      Eq. D2-1      1     0.00    |
| Rupture    0.00E+00   1.84E+02   0.000      Eq. D2-2      1     0.00    |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   2.43E+01   2.24E+02   0.108      Eq. E3-1      2     0.00    |
| Min Buck   2.43E+01   2.24E+02   0.108      Eq. E3-1      2     0.00    |
| Intermediate                                                            |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02           |
| Min Buck   7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02           |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z   -6.96E-02   6.88E+01   0.001      Eq. G6-1      2     0.00    |
| Local-Y    6.75E+01   6.88E+01   0.981      Eq. G6-1      1     0.00    |
| Intermediate                                                            |
| Results       Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z    7.66E+00     0.00     0.00     0.00   1.15E+02               |
| Local-Y    7.66E+00     0.00     0.00     0.00   1.15E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
|            1.56E+02   6.71E+02   0.233      Eq. H3-1      2     0.00    |
| Intermediate Fcr        Tn                                              |
|            3.00E+01   1.12E+03                                          |
```

```
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -6.00E+02   7.12E+02   0.843      Eq. F8-1      1     0.00 |
| Minor      -2.09E+00   7.12E+02   0.003      Eq. F8-1      2     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.19E+03   0.00E+00                                      |
| Minor       1.19E+03   0.00E+00                                      |
     STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.869      Eq. H1-1b     1     0.00              |
| Flexure Tens         0.843      Eq. H1-1b     1     0.00              |
| Intermediate        Mcx /      Mrx /      Pc /                        |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        7.12E+02   3.58E+02   2.24E+02                    |
|                     7.12E+02  -2.09E+00   2.43E+01                    |
| Flexure Tens        7.12E+02   3.58E+02   1.84E+02                    |
|                     7.12E+02  -2.09E+00   0.00E+00                    |
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flx Tor Comp Shr     0.775      Eq. H3-6      2     0.00              |
| Flx Tor Tens Shr     0.667      Eq. H3-6      2     0.00              |
| Intermediate        Mcx /      Mcy /      Mrx /      Mry /            |
|                     Vcx /      Vcy /      Vrx /      Vry /            |
|                     Tc         Tr         Pc         Pr               |
| Flx Tor Comp Shr    7.12E+02   7.12E+02   3.58E+02  -2.09E+00         |
|                     6.88E+01   6.88E+01  -6.96E-02  -1.18E+01         |
|                     6.71E+02   1.56E+02   2.24E+02   2.43E+01         |
| Flx Tor Tens Shr    7.12E+02   7.12E+02   3.58E+02  -2.09E+00         |
|                     6.88E+01   6.88E+01  -6.96E-02  -1.18E+01         |
|                     6.71E+02   1.56E+02   1.84E+02   0.00E+00         |
|----------------------------------------------------------------------|
```

V. AISC 360-10 C Weak Axis Shear G.7

Verify the available shear strength along the weak axis of a singly symmetric section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example G.7, pp. G-15 - G-16

Related Links

- *D1.A.5.4 Design for Shear* (on page 1094)

Details

From reference (2):

> Verify the available shear strength and adequacy of a C9×20 ASTM A36 channel with end shears of 5.00 kips from dead load and 15.0 kips from live load in the weak direction.

Validation

Ultimate load:

$$V_u = 1.2(5.0\text{ kips}) + 1.6(15.0\text{ kips}) = 30.0\text{ kips}$$

Service load:

$$V_a = 5.0\text{ kips} + 15.0\text{ kips} = 20.0\text{ kips}$$

Area of the flanges for shear:

$$A_w = 2b_f t_f = 2(2.65)(0.413) = 2.19\text{ in}^2$$

Section G2.1 of Reference 1 gives $k_v = 1.2$.

$$\frac{b_f}{t_f} = \frac{2.65}{0.413} = 6.42 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{1.2(29,\ 000)}{36}} = 34.2$$

Therefore, $C_v = 1.0$.

<u>Nominal Shear Strength</u>

$$V_n = 0.6F_y A_w C_v = 0.6(36)(2.19)(1.0) = 47.3\text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(47.3) = 42.6\ kips$

ASD: $V_n / \Omega_v = {}^{47.3}\!/_{1.67} = 28.3\ kips$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	42.6	42.6	none	
V_n/Ω_v (kips) [ASD]	28.3	28.3	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 C Weak Axis Shear G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Jun-20
END JOB INFORMATION
```

```
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C9X20
CONSTANTS
BETA 90 ALL
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -30
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FYLD 36 ALL
FU 58 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C9X20                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           11.619         3
                        0.00         -720.00           0.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     74.912  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :    11.619(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -3.000E+01              |
| Mx:   0.000E+00     My:  -7.200E+02     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.189E+00     Ayy:   4.032E+00     Cw:  3.933E+01              |
| Szz:   1.353E+01     Syy:   1.195E+00                                 |
| Izz:   6.090E+01     Iyy:   2.410E+00     Ix:  4.270E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.42     N/A     15.89    T.B4.1(a)-1 |
|                Non-Slender    18.25     N/A     42.29    T.B4.1(a)-5 |
| Flexure     :  Compact         6.42    10.79    28.38    T.B4.1(b)-10 |
|                Compact        18.25   106.72   161.78    T.B4.1(b)-15 |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.90E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   2.55E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.88E+02   0.000      Eq. E3-1      3     0.00 |
```

```
| Min Buck    0.00E+00   1.42E+02   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                            |
| Tor Buck    0.00E+00   1.70E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                        |
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    5.87E+00    14.90   3.56E+01   1.29E+03   2.09E+02       |
| Min Buck    5.87E+00    74.91   2.68E+01   5.10E+01   1.57E+02       |
| Flexural       Ag        Fcr        Pn                              |
| Tor Buck    5.87E+00   3.22E+01   1.89E+02                          |
|---------------------------------------------------------------------|
| CHECK FOR SHEAR                                                     |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     3.00E+01   4.26E+01   0.705      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   7.84E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                        |
| Results        Aw         Cv        Kv      h/tw      Vn            |
| Local-Z     2.19E+00     1.00     1.20     6.42   4.73E+01          |
| Local-Y     4.03E+00     1.00     5.00    18.25   8.71E+01          |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.48E+02   0.000      Eq. F2-1      3     0.00 |
| Minor      -7.20E+02   6.20E+01  11.619      Eq. F6-1      3    24.00 |
| Intermediate   Mn         My                                        |
| Major       6.08E+02   0.00E+00                                     |
| Minor       6.89E+01   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                        |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.21E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb  |
| Major       5.78E+02   0.00E+00    1.00    32.01   174.62    48.00  |
    STAAD SPACE                                           -- PAGE NO.
5
                    STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                     RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp        11.619      Eq. H1-1b     3    24.00            |
| Flexure Tens        11.619      Eq. H1-1b     3    24.00            |
| Intermediate        Mcx /      Mrx /      Pc /                      |
|                     Mcy        Mry        Pr                        |
| Flexure Comp        5.21E+02   0.00E+00   1.42E+02                   |
|                     6.20E+01  -7.20E+02   0.00E+00                   |
| Flexure Tens        5.21E+02   0.00E+00   1.90E+02                   |
|                     6.20E+01  -7.20E+02   0.00E+00                   |
|---------------------------------------------------------------------|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2010
    52. METHOD ASD
    53. FYLD 36 ALL
    54. FU 58 ALL
```

```
   55. TRACK 2 ALL
   56. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
6
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C9X20                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b           11.642         4
                        0.00        -480.00           0.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     74.912  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :    11.642(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -2.000E+01              |
| Mx:   0.000E+00     My:  -4.800E+02     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.189E+00     Ayy:   4.032E+00     Cw:  3.933E+01             |
| Szz:   1.353E+01     Syy:   1.195E+00                                |
| Izz:   6.090E+01     Iyy:   2.410E+00     Ix:  4.270E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.42     N/A     15.89    T.B4.1(a)-1 |
|                Non-Slender    18.25     N/A     42.29    T.B4.1(a)-5 |
| Flexure     :  Compact         6.42    10.79    28.38    T.B4.1(b)-10 |
|                Compact        18.25   106.72   161.78    T.B4.1(b)-15 |
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.27E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.70E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
```

```
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.25E+02   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   9.42E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                               |
| Tor Buck    0.00E+00   1.13E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    5.87E+00    14.90   3.56E+01   1.29E+03   2.09E+02        |
| Min Buck    5.87E+00    74.91   2.68E+01   5.10E+01   1.57E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    5.87E+00   3.22E+01   1.89E+02                            |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     2.00E+01   2.83E+01   0.706      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   5.22E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                           |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     2.19E+00     1.00      1.20     6.42   4.73E+01            |
| Local-Y     4.03E+00     1.00      5.00    18.25   8.71E+01            |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.64E+02   0.000      Eq. F2-1      4     0.00 |
| Minor      -4.80E+02   4.12E+01  11.642      Eq. F6-1      4    24.00 |
| Intermediate   Mn         My                                          |
| Major       6.08E+02   0.00E+00                                       |
| Minor       6.89E+01   0.00E+00                                       |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.46E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       5.78E+02   0.00E+00    1.00    32.01   174.62    48.00   |
    STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp          11.642      Eq. H1-1b     4    24.00            |
| Flexure Tens          11.642      Eq. H1-1b     4    24.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.46E+02   0.00E+00   9.42E+01                  |
|                       4.12E+01  -4.80E+02   0.00E+00                  |
| Flexure Tens          3.46E+02   0.00E+00   1.27E+02                  |
|                       4.12E+01  -4.80E+02   0.00E+00                  |
|------------------------------------------------------------------------|
```

V. AISC 360-10 W D.1

Verify the tensile strength a wide flange section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example D.1, pp. D-2 - D-4

> **Note:** This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

From reference (2):

> Select an 8-in. W-shape, ASTM A992, to carry a dead load of 30 kips and a live load of 90 kips in tension. The member is 25 ft long. Verify the member strength by both LRFD and ASD with the bolted end connection shown [*in the reference*]. Verify that the member satisfies the recommended slenderness limit. Assume that connection limit states do not govern.

See example D.1 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	277	277	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	211	211	none	
Tensile yielding P_n/Ω_t (kips) [ASD]	184	184	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	141	141	none	
Slenderness ratio	238	238.212	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\US\AISC\AISC 360-10 W D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 30
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 90
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
```

```
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   W8X21                    (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854         3
                      180.00 T          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    238.212  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.854(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.800E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02              |
| Szz:   1.819E+01     Syy:   3.708E+00                                 |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    27.52     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1(b)-10 |
|                Compact        27.52    90.55   137.27    T.B4.1(b)-15 |
      STAAD SPACE                                           -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
||-----------------------------------------------------------------------|
|| CHECK FOR AXIAL TENSION                                               |
||                                                                       |
||              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Yield       1.80E+02   2.77E+02   0.649      Eq. D2-1      3     0.00 |
|| Rupture     1.80E+02   2.11E+02   0.854      Eq. D2-2      3     0.00 |
||-----------------------------------------------------------------------|
|| CHECK FOR AXIAL COMPRESSION                                           |
||                                                                       |
||              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Maj Buck    0.00E+00   1.62E+02   0.000      Eq. E3-1      3     0.00 |
|| Min Buck    0.00E+00   2.45E+01   0.000      Eq. E3-1      3     0.00 |
|| Flexural                                                              |
|| Tor Buck    0.00E+00   1.70E+02   0.000      Eq. E4-1      3     0.00 |
|| Intermediate                                                          |
|| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
|| Maj Buck    4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02        |
|| Min Buck    4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01        |
|| Flexural       Ag        Fcr        Pn                                |
|| Tor Buck    4.28E-02   4.41E+03   1.89E+02                            |
||-----------------------------------------------------------------------|
|| CHECK FOR SHEAR                                                       |
||                                                                       |
||              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Local-Z     0.00E+00   1.14E+02   0.000      Eq. G2-1      3     0.00 |
|| Local-Y     0.00E+00   6.21E+01   0.000      Eq. G2-1      3     0.00 |
|| Intermediate                                                          |
|| Results        Aw         Cv       Kv      h/tw      Vn              |
|| Local-Z     2.93E-02     1.00     1.20     6.59   1.26E+02           |
|| Local-Y     1.44E-02     1.00     0.00    27.52   6.21E+01           |
||-----------------------------------------------------------------------|
|| CHECK FOR BENDING-YIELDING                                            |
||                                                                       |
||              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Major       0.00E+00   7.65E+01   0.000      Eq. F2-1      3     0.00 |
|| Minor       0.00E+00   2.13E+01   0.000      Eq. F6-1      3     0.00 |
|| Intermediate   Mn         My                                          |
|| Major       8.50E+01   0.00E+00                                       |
|| Minor       2.37E+01   0.00E+00                                       |
||-----------------------------------------------------------------------|
|| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
||                                                                       |
||              FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Major       0.00E+00   2.52E+01   0.000      Eq. F2-3      3     0.00 |
|| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
|| Major       2.80E+01   0.00E+00    1.00     4.45    14.75    25.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
||-----------------------------------------------------------------------|
|| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
||                       RATIO      CRITERIA    L/C     LOC              |
|| Flexure Comp          0.000      Eq. H1-1b     3     0.00             |
|| Flexure Tens          0.854      Eq. H1-1a     3     0.00             |
|| Intermediate         Mcx /      Mrx /      Pc /                       |
||                      Mcy        Mry        Pr                         |
```

```
| Flexure Comp         2.52E+01   0.00E+00   2.45E+01                      |
|                      2.13E+01   0.00E+00   0.00E+00                      |
| Flexure Tens         2.52E+01   0.00E+00   2.11E+02                      |
|                      2.13E+01   0.00E+00   1.80E+02                      |
|--------------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. FU 9360 ALL
    54. FYLD 7200 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.773 ALL
    58. SLF 0.908 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W8X21                    (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854         4
                      120.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :    238.212  L/C  :     4                 |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00              |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     0.854(PASS)                  |
|          Loc  :    0.00   Condition :   Eq. H1-1a                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   1.200E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                 |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02               |
| Szz:   1.819E+01     Syy:   3.708E+00                                  |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01               |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:        7199.999   Fu:        9359.999                            |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00         |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1(a)-1  |
|                Non-Slender    27.52     N/A     35.88    T.B4.1(a)-5  |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1(b)-10 |
```

```
|               Compact        27.52    90.55   137.27    T.B4.1(b)-15 |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      1.20E+02   1.84E+02   0.651      Eq. D2-1      4     0.00 |
| Rupture    1.20E+02   1.41E+02   0.854      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   0.00E+00   1.08E+02   0.000      Eq. E3-1      4     0.00 |
| Min Buck   0.00E+00   1.63E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck   0.00E+00   1.13E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck   4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02        |
| Min Buck   4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01        |
| Flexural      Ag        Fcr        Pn                                 |
| Tor Buck   4.28E-02   4.41E+03   1.89E+02                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    0.00E+00   7.57E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y    0.00E+00   4.14E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results       Aw        Cv       Kv       h/tw      Vn               |
| Local-Z    2.93E-02     1.00     1.20      6.59   1.26E+02            |
| Local-Y    1.44E-02     1.00     0.00     27.52   6.21E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      0.00E+00   5.09E+01   0.000      Eq. F2-1      4     0.00 |
| Minor      0.00E+00   1.42E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major      8.50E+01   0.00E+00                                        |
| Minor      2.37E+01   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      0.00E+00   1.68E+01   0.000      Eq. F2-3      4     0.00 |
| Intermediate   Mn        Me        Cb       Lp        Lr       Lb    |
| Major      2.80E+01   0.00E+00    1.00     4.45     14.75    25.00   |
     STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                          RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp             0.000      Eq. H1-1b     4     0.00            |
| Flexure Tens             0.854      Eq. H1-1a     4     0.00            |
| Intermediate            Mcx /      Mrx /      Pc /                      |
|                         Mcy        Mry        Pr                        |
| Flexure Comp            1.68E+01   0.00E+00   1.63E+01                  |
|                         1.42E+01   0.00E+00   0.00E+00                  |
| Flexure Tens            1.68E+01   0.00E+00   1.41E+02                  |
|                         1.42E+01   0.00E+00   1.20E+02                  |
|-------------------------------------------------------------------------|
```

V. AISC 360-10 - D-2

Reference

Design Examples, Version 14.0, American Institute of Steel Construction, 2011, page D-5.

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Problem

From the reference:

> Verify, by both ASD and LRFD, the tensile strength of an L4×4×1/2, ASTM A36 [...]. The member carries a dead load of 20 kips and a live load of 60 kips in tension. Calculate at what length this tension member would cease to satisfy the recommended slenderness limit. Assume that connection limit states do not govern.

Refer to the reference for calculations.

Comparison

Table 757: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	122	122	none
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	125	125	none
Tensile yielding P_n / Ω_t (kips) [ASD]	80.8	80.8	none

Result Type	Reference	STAAD.Pro	Difference
Tensile rupture P_n/Ω_t (kips) [ASD]	83.5	83.4	negligible
L at which λ = *300* (ft)	19.4	19.4	none

To calculate the length at which the member would exceed the slenderness limit of 300: (6 in)/(slenderness ratio) × 300

$$(6\text{ in})/(7.732)\times 300 = 232.8\text{ in} = 19.4\text{ ft}$$

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 - D-2.STD` is typically installed with the program.

Tip: You can copy and paste this content directly into a `.std` file to run in STAAD.Pro.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead TITLE LOAD CASE 1
JOINT LOAD
2 FX 20
LOAD 2 LOADTYPE Live TITLE LOAD CASE 2
JOINT LOAD
2 FX 60
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
```

```
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
NSF 0.88267 ALL
SLF 0.869 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
NSF 0.88267 ALL
SLF 0.869 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      STAAD SPACE                                              -- PAGE NO.
3
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   L40408                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.988         3
                      120.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :      7.732  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.988(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.000E+00     Ayy:   2.000E+00     Cw:  3.662E-01              |
| Szz:   1.371E+00     Syy:   3.121E+00                                 |
| Izz:   2.258E+00     Iyy:   8.865E+00     Ix:  3.125E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.87 CSP:   12.00        |
|-----------------------------------------------------------------------|
```

```
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                             |
| Compression :  Non-Slender     8.00     N/A     12.93    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         8.00    15.52    26.15    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       1.20E+02   1.22E+02   0.988      Eq. D2-1      3     0.00 |
| Rupture     1.20E+02   1.25E+02   0.959      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Min Buck    0.00E+00   9.85E+01   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Min Buck    3.75E+00    63.94   2.92E+01   7.18E+01   1.09E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     2.00E+00     1.00     1.20     8.00   4.32E+01            |
| Local-Y     2.00E+00     1.00     1.20     8.00   4.32E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.52E+02   0.000      Eq. F10-1     3     0.00 |
| Minor       0.00E+00   6.66E+01   0.000      Eq. F10-1     3     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.69E+02   1.12E+02                                       |
| Minor       7.40E+01   4.94E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.52E+02   0.000      Eq. F10-3     3     0.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major       1.69E+02   9.12E+03    1.00     0.00     0.00     6.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                          |
```

```
|                                                                          |
|              STRESS     RATIO      CRITERIA     L/C    LOC              |
| Shear       1.94E+01    0.000      Eq. H3-8      3     0.00             |
|--------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                       RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          0.000      Eq. H2-1      3     0.00               |
| Flexure Tens          0.988      Eq. H2-1      3     0.00               |
| Intermediate         frbw /     frbz /     fra /                       |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
|                      4.86E+01   4.86E+01   2.63E+01                   |
| Flexure Tens         0.00E+00   0.00E+00   3.20E+01                   |
|                      4.86E+01   4.86E+01   3.24E+01                   |
  GEOMETRIC AXIS DESIGN
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   9.60E+01   0.000      Eq. F10-1     3     0.00  |
| Minor       0.00E+00   9.60E+01   0.000      Eq. F10-1     3     0.00  |
| Intermediate   Mn         My                                             |
| Major       1.07E+02   7.11E+01                                          |
| Minor       1.07E+02   7.11E+01                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   7.68E+01   0.000      Eq. F10-3     3     0.00  |
| Minor       0.00E+00   7.68E+01   0.000      Eq. F10-3     3     0.00  |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb     |
| Major       8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
| Minor       8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
|--------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
6
    46. LOAD LIST 4
    47. PARAMETER 2
    48. CODE AISC UNIFIED 2010
    49. METHOD ASD
    50. NSF 0.88267 ALL
    51. SLF 0.869 ALL
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   L40408                    (AISC SECTIONS)
                           PASS   Eq. H2-1             0.990        4
                       80.00 T          0.00           0.00       0.00
|--------------------------------------------------------------------------|
| SLENDERNESS                                                              |
```

```
| Actual Slenderness Ratio    :      7.732  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.990(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.000E+01(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.000E+00     Ayy:   2.000E+00     Cw:  3.662E-01              |
| Szz:   1.371E+00     Syy:   3.121E+00                                 |
| Izz:   2.258E+00     Iyy:   8.865E+00     Ix:  3.125E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.87 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     8.00     N/A     12.93    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         8.00    15.52    26.15    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       8.00E+01   8.08E+01   0.990      Eq. D2-1      4     0.00 |
| Rupture     8.00E+01   8.34E+01   0.959      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Min Buck    0.00E+00   6.55E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Min Buck    3.75E+00    63.94   2.92E+01   7.18E+01   1.09E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     2.00E+00     1.00      1.20     8.00   4.32E+01           |
| Local-Y     2.00E+00     1.00      1.20     8.00   4.32E+01           |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.01E+02   0.000      Eq. F10-1     4     0.00 |
| Minor       0.00E+00   4.43E+01   0.000      Eq. F10-1     4     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.69E+02   1.12E+02                                       |
| Minor       7.40E+01   4.94E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.01E+02   0.000      Eq. F10-3     4     0.00 |
| Intermediate   Mn        Me         Cb       Lp       Lr       Lb     |
| Major       1.69E+02   9.12E+03    1.00     0.00     0.00     6.00    |
    STAAD SPACE                                          -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|             STRESS     RATIO      CRITERIA     L/C    LOC            |
| Shear       1.29E+01    0.000      Eq. H3-8      4     0.00           |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H2-1      4     0.00              |
| Flexure Tens          0.990      Eq. H2-1      4     0.00              |
| Intermediate         frbw /     frbz /     fra /                      |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
|                      3.23E+01   3.23E+01   1.75E+01                   |
| Flexure Tens         0.00E+00   0.00E+00   2.13E+01                   |
|                      3.23E+01   3.23E+01   2.16E+01                   |
  GEOMETRIC AXIS DESIGN
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   6.38E+01   0.000      Eq. F10-1     4     0.00 |
| Minor       0.00E+00   6.38E+01   0.000      Eq. F10-1     4     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.07E+02   7.11E+01                                       |
| Minor       1.07E+02   7.11E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.11E+01   0.000      Eq. F10-3     4     0.00 |
| Minor       0.00E+00   5.11E+01   0.000      Eq. F10-3     4     0.00 |
| Intermediate   Mn        Me         Cb       Lp       Lr       Lb     |
| Major       8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
| Minor       8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 WT D.3

Verify the tensile strength a WT section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example D.3, pp. D-8 - D-10

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

From reference (2):

A WT6×20, ASTM A992 member has a length of 30 ft and carries a dead load of 40 kips and a live load of 120 kips in tension. The end connection is fillet welded on each side for 16 in. Verify the member tensile strength by both LRFD and ASD. Assume that the gusset plate and the weld are satisfactory.

See example D.3 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	263	263	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	266	266	none	
Tensile yielding P_n/Ω_t (kips) [ASD]	175	175	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	177	177	none	
Slenderness ratio	229	230.607	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 WT D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W12X40
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 1 ALL
SLF 0.932 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
```

```
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
TMAIN 300 ALL
NSF 1 ALL
SLF 0.932 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W12X40                   (AISC SECTIONS)
                           PASS   Eq. Sec. D2          0.912         3
                      240.00 T          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    230.607  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.912(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. Sec. D2                   |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.125E+00     Ayy:   1.755E+00     Cw:  6.190E-01             |
| Szz:   2.920E+00     Syy:   5.506E+00                                |
| Izz:   1.426E+01     Iyy:   2.205E+01     Ix:  4.530E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.93 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.17     N/A     18.06    T.B4.1(a)-4  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         7.78     9.15    24.08    T.B4.1(b)-10 |
|                Compact        20.17    20.23    24.81    T.B4.1(b)-14 |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       2.40E+02   2.63E+02   0.912      Eq. D2-1      3     0.00 |
 | Rupture     2.40E+02   2.66E+02   0.903      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck    0.00E+00   2.49E+01   0.000      Eq. E7-1      3     0.00 |
 | Min Buck    0.00E+00   3.84E+01   0.000      Eq. E7-1      3     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   3.82E+01   0.000      Eq. E7-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    4.06E-02   230.61   6.80E+02   7.75E+02   2.76E+01        |
 | Min Buck    4.06E-02   185.43   1.05E+03   1.20E+03   4.27E+01        |
 | Flexural       Ag        Fcr        Pn                                |
 | Tor Buck    4.06E-02   1.04E+03   4.24E+01                            |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Local-Z     0.00E+00   1.11E+02   0.000      Eq. G2-1      3     0.00 |
 | Local-Y     0.00E+00   4.74E+01   0.000      Eq. G2-1      3     0.00 |
 | Intermediate                                                          |
 | Results        Aw         Cv       Kv      h/tw      Vn               |
 | Local-Z     2.86E-02     1.00     1.20     7.78   1.24E+02            |
 | Local-Y     1.22E-02     1.00     1.20    20.17   5.27E+01            |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Major       0.00E+00   1.75E+01   0.000      Eq. F9-1      3     0.00 |
 | Minor       0.00E+00   3.15E+01   0.000      Eq. F6-1      3     0.00 |
 | Intermediate   Mn         My                                          |
 | Major       1.95E+01   0.00E+00                                       |
 | Minor       3.50E+01   0.00E+00                                       |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Major       0.00E+00   4.84E+01   0.000      Eq. F9-4      3     0.00 |
 | Intermediate   Mn         Me        Cb       Lp       Lr       Lb     |
 | Major       5.37E+01   0.00E+00    1.00     0.00     0.00    30.00    |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
 |                       RATIO      CRITERIA    L/C     LOC              |
 | Flexure Comp          0.000      Eq. H2-1      3     0.00             |
 | Flexure Tens          0.912      Eq. H2-1      3     0.00             |
 | Intermediate         frbw /     frbz /     fra /                      |
 |                      Fcbw       Fcbz       Fca                        |
```

```
|| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
||                      1.04E+04   9.89E+03   6.12E+02                   |
|| Flexure Tens         0.00E+00   0.00E+00   5.91E+03                   |
||                      2.27E+03   9.89E+03   6.48E+03                   |
||-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. FU 9360 ALL
    54. FYLD 7200 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 1 ALL
    58. SLF 0.932 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W12X40                  (AISC SECTIONS)
                           PASS   Eq. H2-1             0.914         4
                      160.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    230.607  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.914(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.125E+00     Ayy:   1.755E+00     Cw:  6.190E-01             |
| Szz:   2.920E+00     Syy:   5.506E+00                                |
| Izz:   1.426E+01     Iyy:   2.205E+01     Ix:  4.530E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.93 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.17     N/A     18.06    T.B4.1(a)-4  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         7.78     9.15    24.08    T.B4.1(b)-10 |
```

```
|                Compact        20.17    20.23    24.81    T.B4.1(b)-14 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       1.60E+02   1.75E+02   0.914      Eq. D2-1      4     0.00 |
| Rupture     1.60E+02   1.77E+02   0.903      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.65E+01   0.000      Eq. E7-1      4     0.00 |
| Min Buck    0.00E+00   2.56E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.54E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r       Fcr        Fe         Pn          |
| Maj Buck    4.06E-02   230.61   6.80E+02   7.75E+02   2.76E+01        |
| Min Buck    4.06E-02   185.43   1.05E+03   1.20E+03   4.27E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    4.06E-02   1.04E+03   4.24E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   7.41E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   3.15E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv       h/tw      Vn            |
| Local-Z     2.86E-02     1.00      1.20      7.78   1.24E+02          |
| Local-Y     1.22E-02     1.00      1.20     20.17   5.27E+01          |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.17E+01   0.000      Eq. F9-1      4     0.00 |
| Minor       0.00E+00   2.10E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.95E+01   0.00E+00                                      |
| Minor       3.50E+01   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.22E+01   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       5.37E+01   0.00E+00    1.00     0.00     0.00    30.00   |
      STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.000      Eq. H2-1      4     0.00             |
| Flexure Tens         0.914      Eq. H2-1      4     0.00             |
| Intermediate        frbw /     frbz /     fra /                      |
|                     Fcbw       Fcbz       Fca                        |
| Flexure Comp        0.00E+00   0.00E+00   0.00E+00                   |
|                     6.90E+03   6.58E+03   4.07E+02                   |
| Flexure Tens        0.00E+00   0.00E+00   3.94E+03                   |
|                     1.51E+03   6.58E+03   4.31E+03                   |
|----------------------------------------------------------------------|
```

V. AISC 360-10 HSST D.4

Verify the tensile strength of a rectangular HSS section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example D.4, pp. D-11 - D-13

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

From reference (2):

> Verify the tensile strength of an HSS6×4×3/8, ASTM A500 Grade B, with a length of 30 ft. The member is carrying a dead load of 35 kips and a live load of 105 kips in tension. The end connection is a fillet welded 1/2-in.-thick single concentric gusset plate with a weld length of 16 in. Assume that the gusset plate and weld are satisfactory.

Validation

Ultimate load:

$$P_u = 1.2(35\text{ kips}) + 1.6(105\text{ kips}) = 210\text{ kips}$$

Service load:

$$P_a = 35\text{ kips} + 105 = 140\text{ kips}$$

<u>Tensile Yielding</u>

LRFD: $\phi_t P_n = \phi_t A_g F_y = 0.9(6.18)(46) = 256\ kips$

ASD: $P_n \big/ \Omega_c = \frac{A_g F_y}{\Omega_c} = \frac{(6.18)(46)}{1.67} = 170\ kips$

<u>Tensile Rupture</u>

Determine the shear lag factor for the effective area:

$$A_e = A_n U \quad \text{(Eq. D3-1)}$$

where

A_n = the net area; in this case allows for a 1/16-in gap in fit-up between the HSS and the gusset plate,

$= A_g - 2(t_p + 1/16)t = 6.18 - 2(0.5 + 1/16)0.349 = 5.79\ in^2$

$\bar{x}$ = $\frac{B^2 + 2BH}{4(B+H)} = \frac{(4.0)^2 + 2(4.0)(6.0)}{4(4.0 + 6.0)} = 1.60\ in$

U = $1 - \frac{\bar{x}}{l} = 1 - \frac{1.60}{16.0} = 0.90$

$$A_e = 5.79(0.90) = 5.21\ \text{in}^2$$

Nominal Tensile Rupture Strength

$$P_n = F_u A_e = 58(5.21) = 302\ \text{kips} \quad \text{(Eq D2-2)}$$

The available tensile strength:

LRFD: $\phi_t P_n = 0.75(302) = 227\ kips$

ASD: $P_n / \Omega_t = {}^{302}\!/_{2.00} = 151\ kips$

Slenderness

$$\frac{L}{r} = \frac{30 \times 12}{1.55} = 232$$

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	256	256	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	227	226	negligible	
Tensile yielding P_n/Ω_t (kips) [ASD]	170	170	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	151	151	none	
Slenderness ratio	232	231.848	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSST D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 35
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 105
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 0.9 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6624 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 0.9 ALL
TRACK 2 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.927         3
                      210.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    231.848  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.927(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.100E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00             |
| Szz:   9.433E+00     Syy:   7.450E+00                                |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    0.90 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    14.19     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1(b)-17 |
|                Compact        14.19    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       2.10E+02   2.56E+02   0.821      Eq. D2-1      3     0.00 |
| Rupture     2.10E+02   2.26E+02   0.927      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL COMPRESSION                                         |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   4.93E+01   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   2.60E+01   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                        |
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    4.29E-02   168.23   1.28E+03   1.46E+03   5.48E+01      |
| Min Buck    4.29E-02   231.85   6.72E+02   7.67E+02   2.89E+01      |
|---------------------------------------------------------------------|
| CHECK FOR SHEAR                                                     |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.12E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                        |
| Results        Aw         Cv        Kv      h/tw      Vn            |
| Local-Z     1.43E-02     1.00      5.00     8.46   5.69E+01         |
| Local-Y     2.40E-02     1.00      5.00    14.19   9.54E+01         |
|---------------------------------------------------------------------|
| CHECK FOR TORSION                                                   |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.94E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                          |
|             3.97E+03   3.27E+01                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.11E+01   0.000      Eq. F7-1      3     0.00 |
| Minor       0.00E+00   3.08E+01   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                        |
| Major       4.56E+01   0.00E+00                                     |
| Minor       3.43E+01   0.00E+00                                     |
    STAAD SPACE                                               -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                      RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp         0.000      Eq. H1-1b     3     0.00           |
| Flexure Tens         0.927      Eq. H1-1a     3     0.00           |
| Intermediate        Mcx /      Mrx /      Pc /                     |
|                     Mcy        Mry        Pr                       |
| Flexure Comp        4.11E+01   0.00E+00   2.60E+01                 |
|                     3.08E+01   0.00E+00   0.00E+00                 |
| Flexure Tens        4.11E+01   0.00E+00   2.26E+02                 |
|                     3.08E+01   0.00E+00   2.10E+02                 |
|---------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. FU 8352 ALL
    54. FYLD 6624 ALL
    55. METHOD ASD
```

```
    56. TMAIN 300 ALL
    57. NSF 0.936 ALL
    58. SLF 0.9 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.927         4
                      140.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    231.848  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.927(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00              |
| Szz:   9.433E+00     Syy:   7.450E+00                                 |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    0.90 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    14.19     N/A     35.15    T.B4.1(a)-6  |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1(b)-17 |
|                Compact        14.19    60.76   143.12    T.B4.1(b)-19 |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       1.40E+02   1.70E+02   0.822      Eq. D2-1      4     0.00 |
| Rupture     1.40E+02   1.51E+02   0.927      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   3.28E+01   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   1.73E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    4.29E-02   168.23   1.28E+03   1.46E+03   5.48E+01        |
| Min Buck    4.29E-02   231.85   6.72E+02   7.67E+02   2.89E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   3.41E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   5.71E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.43E-02     1.00     5.00     8.46   5.69E+01            |
| Local-Y     2.40E-02     1.00     5.00    14.19   9.54E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   1.96E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   3.27E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   2.73E+01   0.000      Eq. F7-1      4     0.00 |
| Minor       0.00E+00   2.05E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       4.56E+01   0.00E+00                                       |
| Minor       3.43E+01   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H1-1b     4     0.00             |
| Flexure Tens          0.927      Eq. H1-1a     4     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.73E+01   0.00E+00   1.73E+01                   |
|                      2.05E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         2.73E+01   0.00E+00   1.51E+02                   |
```

```
|                      2.05E+01   0.00E+00   1.40E+02                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 HSSP D.5

Verify the tensile strength a round HSS section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example D.5, pp. D-14 - D-16

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

From reference (2):

> Verify the tensile strength of an HSS6×0.500, ASTM A500 Grade B, with a length of 30 ft. The member carries a dead load of 40 kips and a live load of 120 kips in tension. Assume the end connection is a fillet welded 1/2-in.- thick single concentric gusset plate with a weld length of 16 in. Assume that the gusset plate and weld are satisfactory.

See example D.5 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	306	306	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	329	329	negligible	
Tensile yielding P_n / Ω_t (kips) [ASD]	203	203	none	
Tensile rupture P_n / Ω_t (kips) [ASD]	220	220	none	
Slenderness ratio	184	183.316	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 HSSP D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP6X0.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 6048 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
```

```
FU 8352 ALL
FYLD 6048 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSSP6X0.5               (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.785         3
                      240.00 T          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    183.316  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.785(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.466E+00     Ayy:   4.466E+00     Cw:  0.000E+00              |
| Szz:   1.040E+01     Syy:   1.040E+01                                 |
| Izz:   3.120E+01     Iyy:   3.120E+01     Ix:  6.240E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6047.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    12.90     N/A     75.95    T.B4.1(a)-9  |
|                Non-Slender    12.90     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        12.90    48.33   214.05    T.B4.1(b)-20 |
|                Compact        12.90    48.33   214.05    T.B4.1(b)-20 |
      STAAD SPACE                                             -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       2.40E+02   3.06E+02   0.785      Eq. D2-1      3     0.00 |
| Rupture     2.40E+02   3.29E+02   0.729      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   5.44E+01   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   5.44E+01   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr         Fe         Pn         |
| Maj Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01        |
| Min Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   9.17E+01   0.000      Eq. G6-1      3     0.00 |
| Local-Y     0.00E+00   9.17E+01   0.000      Eq. G6-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     5.62E-02     0.00     0.00     0.00   1.02E+02            |
| Local-Y     5.62E-02     0.00     0.00     0.00   1.02E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   3.93E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.63E+03   4.37E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.50E+01   0.000      Eq. F8-1      3     0.00 |
| Minor       0.00E+00   4.50E+01   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       5.01E+01   0.00E+00                                       |
| Minor       5.01E+01   0.00E+00                                       |
    STAAD SPACE                                             -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.000      Eq. H1-1b     3     0.00            |
| Flexure Tens           0.785      Eq. H1-1a     3     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.50E+01   0.00E+00   5.44E+01                  |
|                       4.50E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          4.50E+01   0.00E+00   3.06E+02                  |
|                       4.50E+01   0.00E+00   2.40E+02                  |
```

```
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. FU 8352 ALL
    54. FYLD 6048 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.936 ALL
    58. SLF 1 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1 ST   HSSP6X0.5                (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.786         4
                      160.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    183.316  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.786(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.466E+00     Ayy:   4.466E+00     Cw:  0.000E+00              |
| Szz:   1.040E+01     Syy:   1.040E+01                                 |
| Izz:   3.120E+01     Iyy:   3.120E+01     Ix:  6.240E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6047.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    12.90     N/A     75.95    T.B4.1(a)-9  |
```

```
|                Non-Slender    12.90     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        12.90    48.33   214.05    T.B4.1(b)-20 |
|                Compact        12.90    48.33   214.05    T.B4.1(b)-20 |
     STAAD SPACE                                               -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       1.60E+02   2.03E+02   0.786      Eq. D2-1      4     0.00 |
| Rupture     1.60E+02   2.20E+02   0.729      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   3.62E+01   0.000      Eq. E3-1      4     0.00 |
| Min Buck    0.00E+00   3.62E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01       |
| Min Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   6.10E+01   0.000      Eq. G6-1      4     0.00 |
| Local-Y     0.00E+00   6.10E+01   0.000      Eq. G6-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     5.62E-02     0.00     0.00     0.00   1.02E+02           |
| Local-Y     5.62E-02     0.00     0.00     0.00   1.02E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.62E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.63E+03   4.37E+01                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.00E+01   0.000      Eq. F8-1      4     0.00 |
| Minor       0.00E+00   3.00E+01   0.000      Eq. F8-1      4     0.00 |
| Intermediate   Mn        My                                          |
| Major       5.01E+01   0.00E+00                                      |
| Minor       5.01E+01   0.00E+00                                      |
     STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
```

```
| Flexure Comp          0.000      Eq. H1-1b     4     0.00               |
| Flexure Tens          0.786      Eq. H1-1a     4     0.00               |
| Intermediate         Mcx /      Mrx /      Pc /                         |
|                      Mcy        Mry        Pr                           |
| Flexure Comp         3.00E+01   0.00E+00   3.62E+01                     |
|                      3.00E+01   0.00E+00   0.00E+00                     |
| Flexure Tens         3.00E+01   0.00E+00   2.03E+02                     |
|                      3.00E+01   0.00E+00   1.60E+02                     |
|-----------------------------------------------------------------------|
```

V. AISC 360-10 2L D.6

Verify the tensile strength a double angle section per the LRFD and ASD methods of the AISD 360-10 code.

References

1. American Institute of Steel Construction. 2010 *Steel Construction Manual. Fourteenth Edition*. AISC: Chicago, IL.
2. American Institute of Steel Construction. 2011. *Design Examples, Version 14.0*. AISC:Chicago, IL. Example D.6, pp. D-17 - D-19

Note: This reference is freely available for download from the AISC website at https://www.aisc.org/globalassets/aisc/university-programs/teaching-aids/first-semester-design-examples.pdf

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

From reference (2):

> A 2L4×4×1/2 (3/8-in. separation), ASTM A36, has one line of (8) 3/4-in.-diameter bolts in standard holes and is 25 ft in length. The double angle is carrying a dead load of 40 kips and a live load of 120 kips in tension. Verify the member tensile strength. Assume that the gusset plate and bolts are satisfactory.

See example D.6 in reference (2) for calculations.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	243	243	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	272	272	none	
Tensile yielding P_n/Ω_t (kips) [ASD]	162	162	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile rupture P_n/Ω_t (kips) [ASD]	182	182	none	
Slenderness ratio	248	246.344	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 2L D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE LD L40408 SP 0.03125
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 5184 ALL
```

```
METHOD LRFD
TMAIN 300 ALL
NSF 0.884 ALL
SLF 0.944 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2010
FU 8352 ALL
FYLD 5184 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.884 ALL
SLF 0.944 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L40408                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.988         3
                      240.00 T          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    246.344  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.988(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.000E+00     Ayy:   2.000E+00     Cw:  0.000E+00             |
| Szz:   3.949E+00     Syy:   6.022E+00                                |
| Izz:   1.112E+01     Iyy:   2.522E+01     Ix:  6.250E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.94 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
```

```
| Compression :  Non-Slender     8.00     N/A     12.77    T.B4.1(a)-3  |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         8.00    15.33    25.83    T.B4.1(b)-12 |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       2.40E+02   2.43E+02   0.988      Eq. D2-1      3     0.00 |
| Rupture     2.40E+02   2.72E+02   0.882      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   2.79E+01   0.000      Eq. E3-1      3     0.00 |
| Min Buck    0.00E+00   6.33E+01   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   6.27E+01   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    5.21E-02   246.34   5.96E+02   6.79E+02   3.10E+01        |
| Min Buck    5.21E-02   163.61   1.35E+03   1.54E+03   7.03E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    0.00E+00   1.34E+03   6.97E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   7.78E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     2.78E-02     1.00     1.20     8.00   8.64E+01            |
| Local-Y     1.39E-02     1.00     1.20     8.00   4.32E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.71E+01   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   2.60E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.90E+01   0.00E+00                                       |
| Minor       2.89E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   6.81E+01   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       7.56E+01   0.00E+00    1.00     0.00     0.00    25.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
```

```
                      *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H2-1      3     0.00             |
| Flexure Tens          0.988      Eq. H2-1      3     0.00             |
| Intermediate         frbw /     frbz /     fra /                      |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
|                      7.46E+03   7.46E+03   5.36E+02                   |
| Flexure Tens         0.00E+00   0.00E+00   4.61E+03                   |
|                      3.14E+03   7.46E+03   4.67E+03                   |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2010
    53. FU 8352 ALL
    54. FYLD 5184 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.884 ALL
    58. SLF 0.944 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                      *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L40408                  (AISC SECTIONS)
                           PASS   Eq. H2-1             0.990        4
                     160.00 T          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    246.344  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.990(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.000E+00     Ayy:   2.000E+00     Cw:  0.000E+00              |
| Szz:   3.949E+00     Syy:   6.022E+00                                 |
| Izz:   1.112E+01     Iyy:   2.522E+01     Ix:  6.250E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
```

```
|| Actual Member Length:        25.000                                    |
|| Design Parameters                                                      |
|| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.94 CSP:   12.00         |
||------------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
||                STIFFENED                                               |
|| Compression :  Non-Slender     8.00     N/A     12.77    T.B4.1(a)-3   |
||                  N/A           N/A      N/A      N/A      N/A          |
|| Flexure     :  Compact         8.00    15.33    25.83    T.B4.1(b)-12  |
||                  N/A           N/A      N/A      N/A      N/A          |
      STAAD SPACE                                                -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
  ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |------------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                                |
 |                                                                        |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       1.60E+02   1.62E+02   0.990      Eq. D2-1      4     0.00  |
 | Rupture     1.60E+02   1.82E+02   0.882      Eq. D2-2      4     0.00  |
 |------------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                            |
 |                                                                        |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    0.00E+00   1.86E+01   0.000      Eq. E3-1      4     0.00  |
 | Min Buck    0.00E+00   4.21E+01   0.000      Eq. E3-1      4     0.00  |
 | Flexural                                                               |
 | Tor Buck    0.00E+00   4.17E+01   0.000      Eq. E4-1      4     0.00  |
 | Intermediate                                                           |
 | Results     Eff Area     KL/r      Fcr         Fe         Pn           |
 | Maj Buck    5.21E-02   246.34   5.96E+02   6.79E+02   3.10E+01         |
 | Min Buck    5.21E-02   163.61   1.35E+03   1.54E+03   7.03E+01         |
 | Flexural       Ag        Fcr        Pn                                 |
 | Tor Buck    0.00E+00   1.34E+03   6.97E+01                             |
 |------------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                        |
 |                                                                        |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z     0.00E+00   5.17E+01   0.000      Eq. G2-1      4     0.00  |
 | Local-Y     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00  |
 | Intermediate                                                           |
 | Results        Aw         Cv       Kv      h/tw      Vn                |
 | Local-Z     2.78E-02     1.00     1.20     8.00   8.64E+01             |
 | Local-Y     1.39E-02     1.00     1.20     8.00   4.32E+01             |
 |------------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                             |
 |                                                                        |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Major       0.00E+00   1.14E+01   0.000      Eq. F9-1      4     0.00  |
 | Minor       0.00E+00   1.73E+01   0.000      Eq. F6-1      4     0.00  |
 | Intermediate   Mn         My                                           |
 | Major       1.90E+01   0.00E+00                                        |
 | Minor       2.89E+01   0.00E+00                                        |
 |------------------------------------------------------------------------|
 | CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
 |                                                                        |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Major       0.00E+00   4.53E+01   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb      |
| Major       7.56E+01   0.00E+00    1.00     0.00     0.00    25.00    |
    STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H2-1      4     0.00             |
| Flexure Tens          0.990      Eq. H2-1      4     0.00             |
| Intermediate         frbw /     frbz /     fra /                     |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
|                      4.97E+03   4.97E+03   3.57E+02                   |
| Flexure Tens         0.00E+00   0.00E+00   3.07E+03                   |
|                      2.09E+03   4.97E+03   3.10E+03                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Bending

To verify the flexural member strength in strong-axis bending of an I section for both LRFD and ASD methods per the AISC 360-05 code.

References

1. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, page F-6. Example F.1-1a
2. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005

Related Links

- *D1.A.5.3 Flexural Design Strength* (on page 1094)

Details

Verify the Flexural Member Strength in Strong- Axis Bending by both LRFD and ASD method with ASTM A992 W18X50 beam with a simple span of 35 feet. The nominal loads are a uniform dead load of 0.45 kip/ft and a uniform live load of 0.75 kip/ft. Assuming the beam is continuously braced.

Validation

ASTM A992: $F_y = 50\ ksi$, $F_u = 65\ ksi$

Calculate the required flexural strength:

LRFD

$$w_u = 1.2(0.450\text{ kip/ft}) + 1.6(0.750\text{ kip/ft}) = 1.74\text{ kip/ft}$$

$$M_u = \frac{1.74\text{kip/ft}\left(35.0\text{ft}^2\right)}{8} = 266\text{kip}\cdot\text{ft}$$

ASD

$$w_a = 0.450\text{ kip/ft} + 0.750\text{ kip/ft} = 1.20\text{ kip/ft}$$

$$M_a = \frac{1.20\text{kip/ft}\left(35.0\text{ft}^2\right)}{8} = 184\text{kip}\cdot\text{ft}$$

Per the User Note in section F-2 of ref 2 (p.16.1-47), the W18X50 section is compact. Since the beam is continuously braced and compact, only the yielding limit state applies.

Tip: Per Table 3-2 of ref. 2 (p.3-17), the W18X50 is *not* the most efficient shape for this individual beam. This example follows that given in ref 1. STAAD.Pro will select a lighter beam if the member selection command is used.

LRFD

$$\phi_b M_n = \phi_b M_{px} = 0.9\times (101\text{ in}^3 \times 50\text{ ksi})(1\text{ ft}/12\text{ in}) = 379\text{ ft}\cdot\text{kip} > 266\text{ ft}\cdot\text{kip}$$

ASD

$$M_n/\Omega_b = M_{px}/\Omega_b = (101\text{ in}^3 \times 50\text{ ksi})(1\text{ ft}/12\text{ in}) / 1.67 = 252\text{ ft}\cdot\text{kip} > 180\text{ ft}\cdot\text{kip}$$

Results

Table 758: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (kips) [LRFD]	379	379	none	
M_n/Ω_b (kips) [ASD]	252	252	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Bending.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 27-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI Y -0.45
LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI Y -0.75
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.331         3
                       0.00            0.00        -183.75       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     3.331(FAIL)                |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
```

```
 | Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03                  |
 | Szz:   8.889E+01     Syy:   1.069E+01                                     |
 | Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00                  |
 |---------------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                       |
 | Fyld:        7199.999   Fu:        9359.999                               |
 |---------------------------------------------------------------------------|
 | Actual Member Length:        35.000                                       |
 | Design Parameters                                                         |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00            |
 |---------------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE           |
 |                STIFFENED                                                  |
 | Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3         |
 |                Slender        45.23     N/A     35.88    T.B4.1-10        |
 | Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1         |
 |                Compact        45.23    90.55   137.27    T.B4.1-9         |
     STAAD PLANE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |---------------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                                   |
 |                                                                           |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
 | Yield      0.00E+00   4.40E+02   0.000      Eq. D2-1      3     0.00      |
 | Rupture    0.00E+00   4.78E+02   0.000      Eq. D2-2      3     0.00      |
 |---------------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                               |
 |                                                                           |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
 | Maj Buck   0.00E+00   3.39E+02   0.000      Sec. E1       3     0.00      |
 | Min Buck   0.00E+00   3.42E+01   0.000      Eq. E7-1      3     0.00      |
 | Flexural                                                                  |
 | Tor Buck   0.00E+00   1.73E+02   0.000      Eq. E7-1      3     0.00      |
 | Intermediate                                                              |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn                |
 | Maj Buck   9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02             |
 | Min Buck   1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01             |
 | Flexural      Ag        Fcr        Pn                                     |
 | Tor Buck   1.02E-01   2.82E+03   2.88E+02                                 |
 |---------------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                           |
 |                                                                           |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
 | Local-Z    0.00E+00   1.54E+02   0.000      Eq. G2-1      3     0.00      |
 | Local-Y    2.10E+01   1.28E+02   0.164      Eq. G2-1      3     0.00      |
 | Intermediate                                                              |
 | Results       Aw        Cv       Kv      h/tw      Vn                     |
 | Local-Z    5.94E-02     1.00     1.20     6.58   2.56E+02                 |
 | Local-Y    4.44E-02     1.00     0.00    45.23   1.92E+02                 |
 |---------------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                                |
 |                                                                           |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC       |
 | Major      1.84E+02   2.52E+02   0.729      Eq. F2-1      3    17.50      |
 | Minor      0.00E+00   4.14E+01   0.000      Eq. F6-1      3     0.00      |
```

```
| Intermediate   Mn         My                                          |
| Major       4.21E+02   0.00E+00                                       |
| Minor       6.92E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.84E+02   5.52E+01   3.331      Eq. F2-3      3    17.50 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       9.21E+01   0.00E+00    1.00     5.83    16.97    35.00    |
    STAAD PLANE                                             -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          3.331      Eq. H1-1b     3    17.50            |
| Flexure Tens          3.331      Eq. H1-1b     3    17.50            |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         5.52E+01   1.84E+02   3.42E+01                  |
|                      4.14E+01   0.00E+00   0.00E+00                  |
| Flexure Tens         5.52E+01   1.84E+02   4.40E+02                  |
|                      4.14E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    45. LOAD LIST 4
    46. PARAMETER 2
    47. CODE AISC UNIFIED 2005
    48. METHOD LRFD
    49. FYLD 7200 ALL
    50. FU 9360 ALL
    51. TRACK 2 ALL
    52. CHECK CODE ALL
 STEEL DESIGN
     STAAD PLANE                                             -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                 (AISC SECTIONS)
                           FAIL   Eq. H1-1b           3.213        4
                       0.00            0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     4               |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50            |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     3.213(FAIL)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03                 |
| Szz:   8.889E+01     Syy:   1.069E+01                                    |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:        7199.999   Fu:        9359.999                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        35.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          |
|                STIFFENED                                                 |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3        |
|                Slender        45.23     N/A     35.88    T.B4.1-10       |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1        |
|                Compact        45.23    90.55   137.27    T.B4.1-9        |
    STAAD PLANE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Yield      0.00E+00   6.61E+02   0.000      Eq. D2-1      4     0.00     |
| Rupture    0.00E+00   7.17E+02   0.000      Eq. D2-2      4     0.00     |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Maj Buck   0.00E+00   5.09E+02   0.000      Sec. E1       4     0.00     |
| Min Buck   0.00E+00   5.14E+01   0.000      Eq. E7-1      4     0.00     |
| Flexural                                                                 |
| Tor Buck   0.00E+00   2.59E+02   0.000      Eq. E7-1      4     0.00     |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02            |
| Min Buck   1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01            |
| Flexural      Ag        Fcr        Pn                                    |
| Tor Buck   1.02E-01   2.82E+03   2.88E+02                                |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Local-Z    0.00E+00   2.31E+02   0.000      Eq. G2-1      4     0.00     |
| Local-Y    3.05E+01   1.92E+02   0.159      Eq. G2-1      4     0.00     |
| Intermediate                                                             |
| Results       Aw        Cv        Kv      h/tw      Vn                   |
| Local-Z    5.94E-02     1.00      1.20     6.58   2.56E+02                |
| Local-Y    4.44E-02     1.00      0.00    45.23   1.92E+02                |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
```

```
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       2.66E+02   3.79E+02   0.703      Eq. F2-1      4    17.50 |
| Minor       0.00E+00   6.23E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                            |
| Major       4.21E+02   0.00E+00                                        |
| Minor       6.92E+01   0.00E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       2.66E+02   8.29E+01   3.213      Eq. F2-3      4    17.50 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb     |
| Major       9.21E+01   0.00E+00    1.00     5.83     16.97    35.00    |
    STAAD PLANE                                                -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp            3.213      Eq. H1-1b     4    17.50             |
| Flexure Tens            3.213      Eq. H1-1b     4    17.50             |
| Intermediate          Mcx /      Mrx /       Pc /                       |
|                       Mcy        Mry         Pr                         |
| Flexure Comp          8.29E+01   2.66E+02   5.14E+01                    |
|                       6.23E+01   0.00E+00   0.00E+00                    |
| Flexure Tens          8.29E+01   2.66E+02   6.61E+02                    |
|                       6.23E+01   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-05 Compression

Verify the axial compression capacity for a the y-y axis of an I section for both the LRFD and ASD methods per the AISC 360-05 code.

References

1. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, page E-4. Example E.1a
2. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005

Related Links

- *D1.A.5.2 Axial Compression* (on page 1093)

Details

The column is 30 ft long, pinned top and bottom in both axis.

The section is an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips. The column is 30 feet long

Validation

Calculate the required strength.

LRFD:

$$P_u = 1.2(140 \text{ kips}) + 1.6(420 \text{ kips}) = 840 \text{ kips}$$

ASD:

$$P_a = 140 \text{ kips} + 420 \text{ kips} = 560 \text{ kips}$$

For a pinned-pinned condition, K = 1.0 (Table C-C2.2 of ref. 2). Because the unbraced length is equal about both x-x and y-y directions, y-y axis buckling governs.

Use Table 4-1 in ref 2 and select a W14x132 ($KLy = 30\ ft$). From this table:

$$\phi_c P_n = 892 \text{ kips} > 840 \text{ kips}$$

$$P_n/\Omega_c = 594 \text{ kips} > 560 \text{ kips}$$

Results

Table 759: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	892	893	negligible	
P_n/Ω_c (kips) [ASD]	594	594	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Compression.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 29-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MZ
LOAD 1 DEAD LOAD
JOINT LOAD
2 FY -140
LOAD 2 LIVE LOAD
JOINT LOAD
2 FY -420
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X132                  (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.943         3
                      560.00 C          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     95.792  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.943(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
```

```
|| Fx:   5.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00             |
|| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00             |
||---------------------------------------------------------------------|
|| SECTION PROPERTIES  (UNIT: INCH)                                    |
|| Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04            |
|| Szz:   2.082E+02     Syy:   7.456E+01                               |
|| Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01            |
||---------------------------------------------------------------------|
|| MATERIAL PROPERTIES                                                 |
|| Fyld:          50.000   Fu:          60.000                         |
||---------------------------------------------------------------------|
|| Actual Member Length:       360.000                                 |
|| Design Parameters                                                   |
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00      |
||---------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE     |
||                STIFFENED                                            |
|| Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1-3   |
||                Non-Slender    17.74     N/A     35.88    T.B4.1-10  |
|| Flexure     :  Compact         7.14     9.15    24.08    T.B4.1-1   |
||                Compact        17.74    90.55   137.27    T.B4.1-9   |
     STAAD PLANE                                          -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
||---------------------------------------------------------------------|
|| CHECK FOR AXIAL TENSION                                             |
||                                                                     |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Yield      0.00E+00   1.16E+03   0.000      Eq. D2-1      3     0.00 |
|| Rupture    0.00E+00   1.16E+03   0.000      Eq. D2-2      3     0.00 |
||---------------------------------------------------------------------|
|| CHECK FOR AXIAL COMPRESSION                                         |
||                                                                     |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Maj Buck   5.60E+02   9.14E+02   0.613      Sec. E1       3     0.00 |
|| Min Buck   5.60E+02   5.94E+02   0.943      Eq. E3-1      3     0.00 |
|| Flexural                                                            |
|| Tor Buck   5.60E+02   9.28E+02   0.603      Eq. E4-1      3     0.00 |
|| Intermediate                                                        |
|| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
|| Maj Buck   3.88E+01    57.33   3.93E+01   8.71E+01   1.53E+03       |
|| Min Buck   3.88E+01    95.79   2.56E+01   3.12E+01   9.92E+02       |
|| Flexural       Ag       Fcr        Pn                               |
|| Tor Buck   3.88E+01   3.99E+01   1.55E+03                           |
||---------------------------------------------------------------------|
|| CHECK FOR SHEAR                                                     |
||                                                                     |
||             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|| Local-Z    0.00E+00   5.44E+02   0.000      Eq. G2-1      3     0.00 |
|| Local-Y    0.00E+00   1.90E+02   0.000      Eq. G2-1      3     0.00 |
|| Intermediate                                                        |
|| Results        Aw        Cv        Kv      h/tw      Vn             |
|| Local-Z    3.03E+01     1.00      1.20     7.14   9.08E+02          |
|| Local-Y    9.48E+00     1.00      0.00    17.74   2.84E+02          |
||---------------------------------------------------------------------|
|| CHECK FOR BENDING-YIELDING                                          |
```

```
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      0.00E+00   7.01E+03   0.000      Eq. F2-1      3     0.00    |
| Minor      0.00E+00   3.38E+03   0.000      Eq. F6-1      3     0.00    |
| Intermediate   Mn        My                                              |
| Major      1.17E+04   0.00E+00                                          |
| Minor      5.65E+03   0.00E+00                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major      0.00E+00   5.97E+03   0.000      Eq. F2-2      3     0.00    |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb        |
| Major      9.98E+03   0.00E+00    1.00   159.29   673.75   360.00      |
     STAAD PLANE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                       RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp          0.943      Eq. H1-1a     3     0.00                |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00                |
| Intermediate         Mcx /      Mrx /      Pc /                         |
|                      Mcy        Mry        Pr                           |
| Flexure Comp         5.97E+03   0.00E+00   5.94E+02                     |
|                      3.38E+03   0.00E+00   5.60E+02                     |
| Flexure Tens         5.97E+03   0.00E+00   1.16E+03                     |
|                      3.38E+03   0.00E+00   0.00E+00                     |
|--------------------------------------------------------------------------|
    53. LOAD LIST 4
    54. PARAMETER 2
    55. CODE AISC UNIFIED 2005
    56. METHOD LRFD
    57. FYLD 50 ALL
    58. TRACK 2 ALL
    59. CHECK CODE ALL
 STEEL DESIGN
     STAAD PLANE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X132                  (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.941         4
                      840.00 C          0.00           0.00        0.00
|--------------------------------------------------------------------------|
| SLENDERNESS                                                              |
| Actual Slenderness Ratio    :     95.792  L/C  :     4                   |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      4    Ratio     :     0.941(PASS)                    |
|          Loc  :    0.00   Condition :   Eq. H1-1a                        |
```

```
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   8.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04                 |
| Szz:   2.082E+02     Syy:   7.456E+01                                    |
| Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:          50.000   Fu:          60.000                              |
|--------------------------------------------------------------------------|
| Actual Member Length:       360.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                 |
| Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1-3       |
|                Non-Slender    17.74     N/A     35.88    T.B4.1-10      |
| Flexure     :  Compact         7.14     9.15    24.08    T.B4.1-1       |
|                Compact        17.74    90.55   137.27    T.B4.1-9       |
     STAAD PLANE                                               -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   1.75E+03   0.000      Eq. D2-1      4     0.00    |
| Rupture    0.00E+00   1.75E+03   0.000      Eq. D2-2      4     0.00    |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   8.40E+02   1.37E+03   0.612      Sec. E1       4     0.00    |
| Min Buck   8.40E+02   8.93E+02   0.941      Eq. E3-1      4     0.00    |
| Flexural                                                                 |
| Tor Buck   8.40E+02   1.39E+03   0.602      Eq. E4-1      4     0.00    |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck   3.88E+01    57.33   3.93E+01   8.71E+01   1.53E+03           |
| Min Buck   3.88E+01    95.79   2.56E+01   3.12E+01   9.92E+02           |
| Flexural      Ag        Fcr        Pn                                    |
| Tor Buck   3.88E+01   3.99E+01   1.55E+03                                |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z    0.00E+00   8.18E+02   0.000      Eq. G2-1      4     0.00    |
| Local-Y    0.00E+00   2.84E+02   0.000      Eq. G2-1      4     0.00    |
| Intermediate                                                             |
| Results       Aw        Cv       Kv      h/tw      Vn                   |
| Local-Z    3.03E+01     1.00     1.20     7.14   9.08E+02                |
| Local-Y    9.48E+00     1.00     0.00    17.74   2.84E+02                |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.05E+04   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   5.08E+03   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.17E+04   0.00E+00                                       |
| Minor       5.65E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   8.98E+03   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       9.98E+03   0.00E+00    1.00   159.29   673.75   360.00   |
    STAAD PLANE                                            -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.941      Eq. H1-1a    4     0.00             |
| Flexure Tens           0.000      Eq. H1-1b    4     0.00             |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          8.98E+03   0.00E+00   8.93E+02                  |
|                       5.08E+03   0.00E+00   8.40E+02                  |
| Flexure Tens          8.98E+03   0.00E+00   1.75E+03                  |
|                       5.08E+03   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Tension

Verify the tensile yield strength and tensile rupture strength of an I section using both the LRFD and ASD methods per the AISC 360-05 code.

Reference

1. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, page D-2. Example D.1
2. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005

Related Links

- *D1.A.5.1 Axial Tension* (on page 1092)

Details

Verify the member Tensile Yield strength and Tensile Rupture Strength by both LRFD and ASD with the bolted end connection of a I Section (W8X21) using the LRFD and ASD method

Steel Gr. ASTM A992.

The connection consists of 2 rows of 4- 3/4"Ø bolts in each flange (16 bolts total) in standard holes. The holes are spaced evenly at 3" on center, with a 1-1/4" edge to center at the end of the flange in line with the tensile force.

Validation

ASTM A992: $F_y = 50\ ksi$, $F_u = 65\ ksi$

Assume a W8X21 section:

$A_g = 6.16\ in^2$
$b_f = 5.27\ in$
$t_f = 0.400\ in$
$d = 8.28\ in$
$r_y = 1.26\ in$
$y = 0.831\ in$ (for a WT4X10.5)

Calculate the required tensile strength.

LRFD

$$P_u = 1.2(30\text{ kips}) + 1.6(90\text{ kips}) = 180\text{ kips}$$

ASD

$$P_a = 30\text{ kips} + 90\text{ kips} = 120\text{ kips}$$

Check the tensile yield strength limit state using tabulated values per Table 5-1 in ref. 2:

$$\phi_t P_n = 277\text{ kips} > 180\text{ kips}$$

$$P_n/\Omega_t = 184\text{ kips} > 120\text{ kips}$$

Check the available tensile rupture strength at the end connection. The tabulated values are for $A_e/A_g \geq 0.75$. Verify this assumption is applicable.

Calculate U as the larger of the values from Table D3.1 case 2 or case 7 (ref. 2).

Case 2 - Check as two WT shapes:

$$U = 1 - x/l = 1 - 0.831\text{ in} / 9.0\text{ in} = 0.908$$

where

l	=	length of the bolted connection = $3x\ 3.0\ in = 9.0\ in$
x	=	distance from the flange face to the plastic neutral axis. This is taken from Table 1-8 of ref. 2.

Case 7

$$b_f = 5.27\text{ in} < 2/3d = 2/3(8.28\text{ in}) = 5.52\text{ in};\ U = 0.85$$

Use U = 0.908

Calculate the net area at the bolted connection:

$$A_n = A_g - 4(Ø_h + 1/16\text{ in})t_f = 6.16\text{ in}^2 - 4\times(13/16\text{ in} + 1/16\text{ in})(0.400\text{ in}) = 4.76\text{ in}^2$$

Calculate the effective area:

$$A_e = A_nU = 4.76\text{ in}^2 \times 0.908 = 4.32\text{ in}^2$$

$$A_e/A_g = 4.32\text{ in}^2/ 6.16\text{ in}^2 = 0.701 < 0.75$$

Therefore, the tabulated values for rupture do not apply. Calculate the rupture strength:

$$P_n = F_u A_e = 65 \text{ ksi } (4.32 \text{ in}^2) = 281 \text{ kips}$$

LRFD

$$\phi_t P_n = 0.75(281 \text{ kips}) = 211 \text{ kips} > 180 \text{ kips}$$

ASD

$$P_n/\Omega_t = 281 \text{ kips}/2.00 = 141 \text{ kips} > 120 \text{ kips}$$

Results

Table 760: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
$\phi_t P_n$ (kips) [LRFD]	Yield	277	277	none	
	Rupture	211	211	none	
P_n/Ω_t (kips) [ASD]	Yield	184	184	none	
	Rupture	141	141	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Tension.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 27-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MZ
LOAD 1 DEAD LOAD
JOINT LOAD
1 FY -30
LOAD 2 LIVE LOAD
JOINT LOAD
1 FY -90
LOAD COMB 3 SERVICE LOAD
1 1.0 2 1.0
LOAD COMB 4 ULTIMATE LOAD
1 1.2 2 1.6
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD LRFD
TRACK 2 ALL
NSF 0.773 ALL
SLF 0.908 ALL
FU 9360 ALL
FYLD 7200 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W8X21                    (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854        3
                      120.00 T          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    238.212  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
```

```
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.854(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02              |
| Szz:   1.819E+01     Syy:   3.708E+00                                 |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1-3     |
|                Non-Slender    27.52     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1-1     |
|                Compact        27.52    90.55   137.27    T.B4.1-9     |
     STAAD PLANE                                            -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      1.20E+02   1.84E+02   0.651      Eq. D2-1      3     0.00 |
| Rupture    1.20E+02   1.41E+02   0.854      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   0.00E+00   1.08E+02   0.000      Sec. E1       3     0.00 |
| Min Buck   0.00E+00   1.63E+01   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck   0.00E+00   1.13E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck   4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02        |
| Min Buck   4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01        |
| Flexural      Ag       Fcr        Pn                                  |
| Tor Buck   4.28E-02   4.41E+03   1.89E+02                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    0.00E+00   7.57E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y    0.00E+00   4.14E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
```

```
| Results         Aw        Cv       Kv      h/tw      Vn              |
| Local-Z     2.93E-02     1.00     1.20     6.59   1.26E+02           |
| Local-Y     1.44E-02     1.00     0.00    27.52   6.21E+01           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.09E+01   0.000      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   1.42E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                         |
| Major       8.50E+01   0.00E+00                                      |
| Minor       2.37E+01   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.68E+01   0.000      Eq. F2-3      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       2.80E+01   0.00E+00    1.00     4.45    14.75    25.00    |
    STAAD PLANE                                               -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.000      Eq. H1-1b     3     0.00           |
| Flexure Tens           0.854      Eq. H1-1a     3     0.00           |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          1.68E+01   0.00E+00   1.63E+01                 |
|                       1.42E+01   0.00E+00   0.00E+00                 |
| Flexure Tens          1.68E+01   0.00E+00   1.41E+02                 |
|                       1.42E+01   0.00E+00   1.20E+02                 |
|----------------------------------------------------------------------|
    55. LOAD LIST 4
    56. PARAMETER 2
    57. CODE AISC UNIFIED 2005
    58. METHOD LRFD
    59. TRACK 2 ALL
    60. NSF 0.773 ALL
    61. SLF 0.908 ALL
    62. FU 9360 ALL
    63. FYLD 7200 ALL
    64. CHECK CODE ALL
 STEEL DESIGN
    STAAD PLANE                                               -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W8X21                    (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854        4
                      180.00 T          0.00           0.00        0.00
```

```
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    238.212  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.854(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.800E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02              |
| Szz:   1.819E+01     Syy:   3.708E+00                                 |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1-3     |
|                Non-Slender    27.52     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1-1     |
|                Compact        27.52    90.55   137.27    T.B4.1-9     |
     STAAD PLANE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       1.80E+02   2.77E+02   0.649      Eq. D2-1      4     0.00 |
| Rupture     1.80E+02   2.11E+02   0.854      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   1.62E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   2.45E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.70E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02        |
| Min Buck    4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    4.28E-02   4.41E+03   1.89E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
```

```
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.14E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   6.21E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.93E-02     1.00     1.20     6.59   1.26E+02           |
| Local-Y     1.44E-02     1.00     0.00    27.52   6.21E+01           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   7.65E+01   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   2.13E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       8.50E+01   0.00E+00                                      |
| Minor       2.37E+01   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.52E+01   0.000      Eq. F2-3      4     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       2.80E+01   0.00E+00    1.00     4.45    14.75    25.00   |
     STAAD PLANE                                         -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp         0.000      Eq. H1-1b     4     0.00            |
| Flexure Tens         0.854      Eq. H1-1a     4     0.00            |
| Intermediate        Mcx /      Mrx /      Pc /                      |
|                     Mcy        Mry        Pr                        |
| Flexure Comp        2.52E+01   0.00E+00   2.45E+01                  |
|                     2.13E+01   0.00E+00   0.00E+00                  |
| Flexure Tens        2.52E+01   0.00E+00   2.11E+02                  |
|                     2.13E+01   0.00E+00   1.80E+02                  |
|----------------------------------------------------------------------|
```

V. AISC 360-05 2L D.6

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.6, pp. D-12,13

Details

From reference (2):

A 2L4×4×2 (3/8-in. separation), ASTM A36, has one line of (8) ¾-in. diameter bolts in standard holes and is 25 ft in length. The double angle is carrying a dead load of 40 kips and a live load of 120 kips in tension. Verify the strength by both LRFD and ASD.

Validation

Refer to reference.

Results

Table 761: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	243	243	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	272	272	none	
Tensile yielding P_n / Ω_t (kips) [ASD]	162	162	none	
Tensile rupture P_n / Ω_t (kips) [ASD]	182	182	none	
Slenderness ratio	248	246.344	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 2L D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE LD L40408 SP 0.03125
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 5184 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.884 ALL
SLF 0.944 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 5184 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.884 ALL
SLF 0.944 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L40408                   (AISC SECTIONS)
                          PASS   Eq. H2-1            0.988         3
                     240.00 T          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    246.344  L/C  :     3                |
```

```
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      3    Ratio     :     0.988(PASS)                    |
|          Loc  :    0.00   Condition :   Eq. H2-1                         |
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   4.000E+00     Ayy:   2.000E+00     Cw:  0.000E+00                 |
| Szz:   3.949E+00     Syy:   6.022E+00                                    |
| Izz:   1.112E+01     Iyy:   2.522E+01     Ix:  6.250E-01                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:        5183.999   Fu:        8351.999                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        25.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.94 CSP:   12.00           |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                 |
| Compression :  Non-Slender     8.00     N/A     12.77    T.B4.1-5       |
|                  N/A           N/A      N/A      N/A      N/A           |
| Flexure     :  Compact         8.00    15.33    25.83    T.B4.1-6       |
|                  N/A           N/A      N/A      N/A      N/A           |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       2.40E+02   2.43E+02   0.988      Eq. D2-1      3     0.00   |
| Rupture     2.40E+02   2.72E+02   0.882      Eq. D2-2      3     0.00   |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    0.00E+00   2.79E+01   0.000      Sec. E1       3     0.00   |
| Min Buck    0.00E+00   6.33E+01   0.000      Eq. E3-1      3     0.00   |
| Flexural                                                                 |
| Tor Buck    0.00E+00   6.27E+01   0.000      Eq. E4-1      3     0.00   |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    5.21E-02   246.34   5.96E+02   6.79E+02   3.10E+01           |
| Min Buck    5.21E-02   163.64   1.35E+03   1.54E+03   7.03E+01           |
| Flexural       Ag       Fcr        Pn                                    |
| Tor Buck    0.00E+00   1.34E+03   6.97E+01                              |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z     0.00E+00   7.78E+01   0.000      Eq. G2-1      3     0.00   |
```

```
| Local-Y     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     2.78E-02     1.00     1.20     8.00   8.64E+01            |
| Local-Y     1.39E-02     1.00     1.20     8.00   4.32E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.71E+01   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   2.60E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.90E+01   0.00E+00                                       |
| Minor       2.89E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   6.81E+01   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       7.56E+01   0.00E+00    1.00     0.00     0.00    25.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.000      Eq. H2-1      3     0.00            |
| Flexure Tens           0.988      Eq. H2-1      3     0.00            |
| Intermediate          frbw /     frbz /     fra /                     |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   0.00E+00                  |
|                       7.46E+03   7.46E+03   5.36E+02                  |
| Flexure Tens          0.00E+00   0.00E+00   4.61E+03                  |
|                       3.14E+03   7.46E+03   4.67E+03                  |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. FU 8352 ALL
    54. FYLD 5184 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.884 ALL
    58. SLF 0.944 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
       1 LD   L40408                   (AISC SECTIONS)
                           PASS   Eq. H2-1            0.990        4
                      160.00 T          0.00          0.00        0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    246.344  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.990(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H2-1                      |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   4.000E+00     Ayy:   2.000E+00     Cw:  0.000E+00              |
 | Szz:   3.949E+00     Syy:   6.022E+00                                 |
 | Izz:   1.112E+01     Iyy:   2.522E+01     Ix:  6.250E-01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        5183.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        25.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.94 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     8.00     N/A     12.77    T.B4.1-5     |
 |                  N/A           N/A      N/A      N/A      N/A         |
 | Flexure     :  Compact         8.00    15.33    25.83    T.B4.1-6     |
 |                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       1.60E+02   1.62E+02   0.990      Eq. D2-1      4     0.00 |
 | Rupture     1.60E+02   1.82E+02   0.882      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    0.00E+00   1.86E+01   0.000      Sec. E1       4     0.00 |
 | Min Buck    0.00E+00   4.21E+01   0.000      Eq. E3-1      4     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   4.17E+01   0.000      Eq. E4-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    5.21E-02   246.34   5.96E+02   6.79E+02   3.10E+01        |
 | Min Buck    5.21E-02   163.64   1.35E+03   1.54E+03   7.03E+01        |
 | Flexural       Ag       Fcr        Pn                                 |
```

```
| Tor Buck    0.00E+00   1.34E+03   6.97E+01                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   5.17E+01   0.000      Eq. G2-1      4     0.00  |
| Local-Y     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn                |
| Local-Z     2.78E-02     1.00     1.20     8.00   8.64E+01              |
| Local-Y     1.39E-02     1.00     1.20     8.00   4.32E+01              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.14E+01   0.000      Eq. F9-1      4     0.00  |
| Minor       0.00E+00   1.73E+01   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn        My                                            |
| Major       1.90E+01   0.00E+00                                         |
| Minor       2.89E+01   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   4.53E+01   0.000      Eq. F9-4      4     0.00  |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       7.56E+01   0.00E+00    1.00     0.00     0.00    25.00    |
     STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.000      Eq. H2-1      4     0.00              |
| Flexure Tens           0.990      Eq. H2-1      4     0.00              |
| Intermediate          frbw /     frbz /     fra /                       |
|                       Fcbw       Fcbz       Fca                         |
| Flexure Comp          0.00E+00   0.00E+00   0.00E+00                    |
|                       4.97E+03   4.97E+03   3.57E+02                    |
| Flexure Tens          0.00E+00   0.00E+00   3.07E+03                    |
|                       2.09E+03   4.97E+03   3.10E+03                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-05 Angle Section D-2

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.2, pp. D-4,5

Details

From reference (2):

> Verify, by both ASD and LRFD, the strength of an L4×4×½, ASTM A36, with one line of (4) ¾ in. diameter bolts in standard holes. The member carries a dead load of 20 kips and a live load of 60 kips in tension. Calculate at what length this tension member would cease to satisfy the recommended slenderness limit.

Validation

Refer to reference.

Results

Table 762: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	122	122	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	125	125	none	
Tensile yielding P_n / Ω_t (kips) [ASD]	80.8	80.8	none	
Tensile rupture P_n / Ω_t (kips) [ASD]	83.5	83.5	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Angle Section D-2.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
```

```
DAMP 0.03
TYPE STEEL
STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L40408
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 20
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 60
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
SLF 0.869 ALL
NSF 0.883 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
SLF 0.869 ALL
NSF 0.883 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   L40408                   (AISC SECTIONS)
                           PASS   Eq. H2-1            0.988        3
                      120.00 T         0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :      7.732  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.988(PASS)                |
```

```
||          Loc  :    0.00   Condition :   Eq. H2-1                        |
||-------------------------------------------------------------------------|
|| DESIGN FORCES                                                           |
|| Fx:   1.200E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00                 |
|| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                 |
||-------------------------------------------------------------------------|
|| SECTION PROPERTIES  (UNIT: INCH)                                        |
|| Azz:   2.000E+00     Ayy:   2.000E+00     Cw:  3.662E-01                |
|| Szz:   1.371E+00     Syy:   3.121E+00                                   |
|| Izz:   2.258E+00     Iyy:   8.865E+00     Ix:  3.125E-01                |
||-------------------------------------------------------------------------|
|| MATERIAL PROPERTIES                                                     |
|| Fyld:          36.000   Fu:          58.000                             |
||-------------------------------------------------------------------------|
|| Actual Member Length:         6.000                                     |
|| Design Parameters                                                       |
|| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.87 CSP:   12.00          |
||-------------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
||                STIFFENED                                                |
|| Compression :  Non-Slender     8.00     N/A     12.93    T.B4.1-5       |
||                  N/A           N/A      N/A      N/A      N/A           |
|| Flexure     :  Compact         8.00    15.52    26.15    T.B4.1-6       |
||                  N/A           N/A      N/A      N/A      N/A           |
      STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       1.20E+02   1.22E+02   0.988      Eq. D2-1      3     0.00   |
| Rupture     1.20E+02   1.25E+02   0.959      Eq. D2-2      3     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Min Buck    0.00E+00   9.85E+01   0.000      Eq. E3-1      3     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Min Buck    3.75E+00    63.94   2.92E+01   7.18E+01   1.09E+02          |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00   |
| Local-Y     0.00E+00   3.89E+01   0.000      Eq. G2-1      3     0.00   |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn                 |
| Local-Z     2.00E+00     1.00      1.20     8.00   4.32E+01             |
| Local-Y     2.00E+00     1.00      1.20     8.00   4.32E+01             |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   1.52E+02   0.000      Eq. F10-1     3     0.00   |
```

```
| Minor        0.00E+00   6.66E+01   0.000      Eq. F10-1     3     0.00 |
| Intermediate   Mn         My                                           |
| Major        1.69E+02   1.12E+02                                       |
| Minor        7.40E+01   4.94E+01                                       |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major        0.00E+00   1.52E+02   0.000      Eq. F10-3     3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb      |
| Major        1.69E+02   9.12E+03    1.00     0.00     0.00     6.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                           |
|                                                                         |
|               STRESS     RATIO      CRITERIA     L/C    LOC            |
| Shear        1.94E+01    0.000      Eq. H3-8      3     0.00           |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp            0.000      Eq. H2-1      3     0.00             |
| Flexure Tens            0.988      Eq. H2-1      3     0.00             |
| Intermediate           frbw /     frbz /     fra /                     |
|                        Fcbw       Fcbz       Fca                       |
| Flexure Comp           0.00E+00   0.00E+00   0.00E+00                  |
|                        4.86E+01   4.86E+01   2.63E+01                  |
| Flexure Tens           0.00E+00   0.00E+00   3.20E+01                  |
|                        4.86E+01   4.86E+01   3.24E+01                  |
  GEOMETRIC AXIS DESIGN
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major        0.00E+00   9.60E+01   0.000      Eq. F10-1     3     0.00 |
| Minor        0.00E+00   9.60E+01   0.000      Eq. F10-1     3     0.00 |
| Intermediate   Mn         My                                           |
| Major        1.07E+02   7.11E+01                                       |
| Minor        1.07E+02   7.11E+01                                       |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major        0.00E+00   7.68E+01   0.000      Eq. F10-3     3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb      |
| Major        8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
|-------------------------------------------------------------------------|
     STAAD SPACE                                              -- PAGE NO.
6
    46. LOAD LIST 4
    47. PARAMETER 2
    48. CODE AISC UNIFIED 2005
    49. METHOD ASD
    50. SLF 0.869 ALL
    51. NSF 0.883 ALL
```

```
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   L40408                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.990        4
                       80.00 T          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :      7.732  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.990(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.000E+01(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.000E+00     Ayy:   2.000E+00     Cw:  3.662E-01              |
| Szz:   1.371E+00     Syy:   3.121E+00                                 |
| Izz:   2.258E+00     Iyy:   8.865E+00     Ix:  3.125E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.88 SLF:    0.87 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     8.00     N/A     12.93    T.B4.1-5     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         8.00    15.52    26.15    T.B4.1-6     |
|                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      8.00E+01   8.08E+01   0.990      Eq. D2-1      4     0.00  |
| Rupture    8.00E+01   8.34E+01   0.959      Eq. D2-2      4     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
```

```
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Min Buck    0.00E+00   6.55E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Min Buck    3.75E+00    63.94   2.92E+01   7.18E+01   1.09E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   2.59E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.00E+00     1.00     1.20     8.00   4.32E+01            |
| Local-Y     2.00E+00     1.00     1.20     8.00   4.32E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.01E+02   0.000      Eq. F10-1     4     0.00 |
| Minor       0.00E+00   4.43E+01   0.000      Eq. F10-1     4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.69E+02   1.12E+02                                       |
| Minor       7.40E+01   4.94E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.01E+02   0.000      Eq. F10-3     4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       1.69E+02   9.12E+03    1.00     0.00     0.00     6.00   |
    STAAD SPACE                                           -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|               STRESS     RATIO      CRITERIA     L/C    LOC          |
| Shear       1.29E+01    0.000      Eq. H3-8      4     0.00          |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.000      Eq. H2-1      4     0.00             |
| Flexure Tens         0.990      Eq. H2-1      4     0.00             |
| Intermediate        frbw /     frbz /     fra /                      |
|                     Fcbw       Fcbz       Fca                        |
| Flexure Comp        0.00E+00   0.00E+00   0.00E+00                   |
|                     3.23E+01   3.23E+01   1.75E+01                   |
| Flexure Tens        0.00E+00   0.00E+00   2.13E+01                   |
|                     3.23E+01   3.23E+01   2.16E+01                   |
 GEOMETRIC AXIS DESIGN
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
```

```
| Major       0.00E+00   6.38E+01   0.000      Eq. F10-1     4     0.00 |
| Minor       0.00E+00   6.38E+01   0.000      Eq. F10-1     4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.07E+02   7.11E+01                                       |
| Minor       1.07E+02   7.11E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   5.11E+01   0.000      Eq. F10-3     4     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb      |
| Major       8.53E+01   1.40E+05    1.00     0.00     0.00     6.00    |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Built Up I E 12

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.12, pp.E-44 - E-47

Details

From reference (2):

> Compute the available strength of a built-up compression member with a length of 14 ft. The ends are pinned. The outside flange is PL3/4×5, the inside flange is PL3/4×8, and the web is PL3/8×10½. Material is ASTM A572 Grade 50.

Validation

Material Properties

$$F_y = 50 \text{ ksi}$$

$$F_u = 65 \text{ ksi}$$

Section Properties

Built-up section properties (ignoring fillet welds):

$$A_g = 8.0(0.75) + 10.5(0.375) + 5.0(0.75) = 13.7 \text{ in}^2$$

$$\bar{y} = \frac{\Sigma A_i y_i}{\Sigma A_i} = \frac{6.0(11.6) + 3.94(6.0) + 3.75(0.375)}{6.0 + 3.94 + 3.75} = 6.91 \text{ in}$$

where $\bar{y}$ is measured from the bottom of the outside flange.

$$I_x = \left[8.0(0.75)(4.72)^2 + \frac{8.0(0.75)^3}{12}\right] + \left[0.375(10.5)(0.910)^2 + \frac{0.375(10.5)^3}{12}\right] + \left[5.0(0.75)(6.54)^2 + \frac{5.0(0.75)^3}{12}\right] =$$

$$I_y = \frac{0.75 \times 8.0^3}{12} + \frac{10.5 \times 0.375^3}{12} + \frac{0.75 \times 5.0^3}{12} = 39.9 \text{ in}^4$$

$$r_x = \sqrt{\frac{I_x}{A}} = \sqrt{\frac{334}{13.7}} = 4.94 \text{ in}$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{39.9}{13.7}} = 1.71 \text{ in}$$

From Eq. 3.4 in Reference 3:

$$J = \Sigma\frac{bt^3}{3} = \frac{8.0(0.75)^3 + 10.5(0.375)^3 + 5.0(0.75)^3}{3} = 2.01 \text{ in}^4$$

Distance between flange centroids:

$$h_0 = 12.0 - \frac{0.75}{2} - \frac{0.75}{2} = 11.3 \text{ in}$$

$$C_w = \frac{t_f h_o^2}{12}\left(\frac{b_1^3 b_2^3}{b_1^3 + b_2^3}\right) = \frac{0.75(11.3)^2}{12}\left[\frac{(8.0)^3(5.0)^3}{(8.0)^3 + (5.0)^3}\right] = 802 \text{ in}^6$$

Due to symmetry about the y-axis, the shear center lies on the y axis ($x_0 = 0$). The distance from the centroid to the shear center can be calculated by finding the shear center from outside flange plus one-half the outside flange and then subtracting the centroid distance from that same reference point.

$$y_0 = \left(e + \frac{t_f}{2}\right) - \bar{y} = h_0\left[\frac{b_1^3}{b_1^3 + b_2^3}\right] + \frac{t_f}{2} - \bar{y} = 11.3\left[\frac{(8.0)^3}{(8.0)^3 + (5.0)^3}\right] + \frac{0.75}{2} - 6.91 = 2.55 \text{ in}$$

$$\bar{r}_0^2 = x_0^2 + y_0^2 + \frac{I_x + I_y}{A_g} = 0 + (2.55)^2 + \frac{344 + 39.9}{13.7} = 33.8 \text{ in}^2$$

$$H = 1 - \frac{x_0^2 + y_0^2}{\bar{r}_0^2} = 1 - \frac{0 + (2.55)^2}{33.8} = 0.808$$

<u>Elastic Flexural Buckling Stress</u>

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{K_y L}{r_y} = \frac{1.0(14.0)(12)}{1.71} = 98.2$$

$$F_{ey} = \frac{\pi^2 E}{(KL/r_y)^2} = \frac{\pi^2 29,000}{(98.2)^2} = 29.7 \text{ ksi} \qquad \text{(Eq. E4-8)}$$

$$F_{ez} = \left[\frac{\pi^2 E C_w}{(K_z L)^2} + GJ\right]\left(\frac{1}{A_g \bar{r}_0^2}\right) = \left[\frac{\pi^2(29,000)(802)}{(1.0 \times 14.0 \times 12)^2} + 11,200(2.01)\right]\left(\frac{1}{13.7 \times 33.8}\right) = 66.2 \text{ ksi} \qquad \text{(Eq. E4-9)}$$

$$F_e = \left(\frac{F_{ey} + F_{ez}}{2H}\right)\left[1 - \sqrt{1 - \frac{4F_{ey}F_{ez}H}{(F_{ey} + F_{ez})^2}}\right] = \left(\frac{29.7 + 66.2}{2 \times 0.808}\right)\left[1 - \sqrt{1 - \frac{4(29.7)(66.2)(0.808)}{(29.7 + 66.2)^2}}\right] = 26.4 \text{ ksi} \qquad \text{(Eq. E4-5)}$$

<u>Nominal Compressive Strength</u>

$$\frac{F_y}{2.25} = \frac{50}{2.25} = 22.2 \text{ ksi} < 26.4 \text{ ksi}$$

$$F_{cr} = \left(0.658^{F_y/F_e}\right)F_y = \left[0.658^{(50)/(26.4)}\right]50 = 22.6 \text{ ksi} \qquad \text{(Eq E7-2)}$$

$$P_n = F_{cr}A_g = 22.6(13.7) = 310 \text{ kips} \quad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(310) = 279 \; kips$

ASD: $P_n / \Omega_c = {}^{310}/_{1.67} = 186 \; kips$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	279	278	negligible	
P_n/Ω_c (kips) [ASD]	186	185	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Built Up I E 12.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 14 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
I
13.6875 12 0.375 8 0.75 333.426 39.8586 2.0127 4.5 6.5 5 0.75
END
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
```

```
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 I
CONSTANTS
MATERIAL STEEL_36_KSI ALL
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -40
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
STP 2 ALL
FU 65 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
STP 2 ALL
FU 65 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   I                         (UPT)
                           PASS   Eq. H1-1a           0.316         3
                       88.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     98.449  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
```

```
 | Critical L/C  :      3    Ratio     :     0.316(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H1-1a                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   8.800E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   6.500E+00     Ayy:   4.500E+00     Cw:  7.947E+02              |
 | Szz:   4.815E+01     Syy:   9.965E+00                                 |
 | Izz:   3.334E+02     Iyy:   3.986E+01     Ix:  2.013E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          65.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:       168.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     5.33     N/A     13.40    T.B4.1-4     |
 |                Non-Slender    28.00     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         5.33     9.15    23.84    T.B4.1-2     |
 |                Compact        23.17   127.82   137.27    T.B4.1-11    |
     STAAD SPACE                                              -- PAGE NO.
4
                     STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       0.00E+00   6.16E+02   0.000      Eq. D2-1      3     0.00 |
 | Rupture     0.00E+00   6.67E+02   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    8.80E+01   5.66E+02   0.156      Sec. E1       3     0.00 |
 | Min Buck    8.80E+01   3.03E+02   0.290      Eq. E3-1      3     0.00 |
 | Flexural                                                              |
 | Tor Buck    8.80E+01   2.78E+02   0.316      Eq. E4-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.37E+01    34.04   4.59E+01   2.47E+02   6.29E+02        |
 | Min Buck    1.37E+01    98.45   2.46E+01   2.95E+01   3.37E+02        |
 | Flexural       Ag       Fcr        Pn                                 |
 | Tor Buck    1.37E+01   2.26E+01   3.09E+02                            |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z     0.00E+00   1.76E+02   0.000      Eq. G2-1      3     0.00 |
 | Local-Y     0.00E+00   1.22E+02   0.000      Eq. G2-1      3     0.00 |
 | Intermediate                                                          |
 | Results        Aw        Cv       Kv      h/tw      Vn                |
```

```
| Local-Z     6.50E+00     1.00     1.20    10.67   1.95E+02            |
| Local-Y     4.50E+00     1.00     5.00    28.00   1.35E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.78E+03   0.000      Sec. F4.2     3     0.00 |
| Minor       0.00E+00   7.17E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.09E+03   0.00E+00                                       |
| Minor       7.97E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.30E+03   0.000      Eq. F4-2      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       2.56E+03   0.00E+00    1.00    60.83   343.65   168.00   |
     STAAD SPACE                                              -- PAGE NO.
5
                          STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                          ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.78E+03   0.000      Eq. F4-1      3     0.00 |
| Intermediate   Mn         Rpc                                         |
|             3.09E+03   1.28E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.316      Eq. H1-1a     3     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         2.30E+03   0.00E+00   2.78E+02                   |
|                      7.17E+02   0.00E+00   8.80E+01                   |
| Flexure Tens         2.30E+03   0.00E+00   6.16E+02                   |
|                      7.17E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    67. LOAD LIST 4
    68. PARAMETER 2
    69. CODE AISC UNIFIED 2005
    70. METHOD ASD
    71. STP 2 ALL
    72. FU 65 ALL
    73. FYLD 50 ALL
    74. TRACK 2 ALL
    75. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                          STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                          ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
```

```
                          FX            MY              MZ       LOCATION
=======================================================================
      1 ST   I                           (UPT)
                           PASS   Eq. H1-1a           0.324         4
                       60.00 C          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     98.449  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.324(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   6.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.500E+00     Ayy:   4.500E+00     Cw:  7.947E+02              |
| Szz:   4.815E+01     Syy:   9.965E+00                                 |
| Izz:   3.334E+02     Iyy:   3.986E+01     Ix:  2.013E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       168.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.33     N/A     13.40    T.B4.1-4     |
|                Non-Slender    28.00     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         5.33     9.15    23.84    T.B4.1-2     |
|                Compact        23.17   127.82   137.27    T.B4.1-11    |
    STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   4.10E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.45E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    6.00E+01   3.77E+02   0.159      Sec. E1       4     0.00 |
| Min Buck    6.00E+01   2.02E+02   0.297      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    6.00E+01   1.85E+02   0.324      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.37E+01    34.04   4.59E+01   2.47E+02   6.29E+02        |
```

```
| Min Buck    1.37E+01    98.45   2.46E+01   2.95E+01   3.37E+02        |
| Flexural       Ag       Fcr        Pn                                  |
| Tor Buck    1.37E+01   2.26E+01   3.09E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.17E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   8.08E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     6.50E+00     1.00     1.20    10.67   1.95E+02            |
| Local-Y     4.50E+00     1.00     5.00    28.00   1.35E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.85E+03   0.000      Sec. F4.2     4     0.00 |
| Minor       0.00E+00   4.77E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major       3.09E+03   0.00E+00                                       |
| Minor       7.97E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.53E+03   0.000      Eq. F4-2      4     0.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb     |
| Major       2.56E+03   0.00E+00    1.00    60.83   343.65   168.00   |
     STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.85E+03   0.000      Eq. F4-1      4     0.00 |
| Intermediate   Mn        Rpc                                          |
|             3.09E+03   1.28E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.324      Eq. H1-1a     4     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     4     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         1.53E+03   0.00E+00   1.85E+02                  |
|                      4.77E+02   0.00E+00   6.00E+01                  |
| Flexure Tens         1.53E+03   0.00E+00   4.10E+02                  |
|                      4.77E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Built up I E.2

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.2, pp.E-10 - E-13

Details

From reference (2):

> Verify that a built-up, ASTM A572 grade 50, column with PL1in.×8in. flanges and a PL1/4in.×15in. web is sufficient to carry a dead load of 70 kips and live load of 210 kips in axial compression. The column length is 15 ft and the ends are pinned in both axes.

Validation

Ultimate load:

$$P_u = 1.2(70\text{ kips}) + 1.6(210\text{ kips}) = 420\text{ kips}$$

Service load:

$$P_a = 70\text{ kips} + 210\text{ kips} = 280\text{ kips}$$

<u>Section Properties</u>

Built-up section properties (ignoring fillet welds):

$$A = 2(8.0)(1.0) + 15.0(0.25) = 19.8\text{ in}^2$$

$$I_x = \Sigma Ad^2 + \Sigma\frac{bh^3}{12} = 2\left[8(8.0)^2 + \frac{8.0(1.0)^3}{12}\right] + \frac{0.25(15.0)^3}{12} = 1,\ 100\text{ in}^4$$

$$I_y = 2\left(\frac{1.0 \times 8.0^3}{12}\right) + \frac{15.0 \times 0.25^3}{12} = 85.4\text{ in}^4$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{85.4}{19.8}} = 2.08\text{ in}$$

<u>Elastic Flexural Buckling Stress</u>

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{KL}{r_y} = \frac{1.0(15.0)(12)}{2.08} = 86.5$$

$$F_e = \frac{\pi^2 E}{(KL\ /r)^2} = \frac{\pi^2 29,\ 000}{(86.5)^2} = 38.3\text{ ksi} \qquad \text{(Eq. E3-4)}$$

<u>Slenderness</u>

Check for slender flanges and then determine the unstiffened element (flange) reduction factor.

$$k_c = \frac{4}{\sqrt{h\ /t_w}} = \frac{4}{\sqrt{15.0/0.25}} = 0.516 \qquad \text{(Table B4.1b note a)}$$

$0.35 < k_c < 0.76$

Flange slenderness:

$$\lambda = \frac{b}{t} = \frac{4.0}{1.0} = 4.0$$

The limiting flange slenderness ratio, λ_r:

$$\lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.516(29,\ 000)}{50}} = 11.1 > \lambda \qquad \text{(Table B4.1a, case 2)}$$

Thus, the flanges are not slender and there is no reduction ($Q_s = 1.0$).

Check for slender web and then determine the stiffened element (web) reduction factor.

Web slenderness:

$$\lambda = \frac{h}{t} = \frac{15.0}{0.25} = 60.0$$

The limiting web slenderness ratio, λ_r:

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,\ 000}{50}} = 35.9 < \lambda \qquad \text{(Table B4.1a, case 5)}$$

Thus, the web is slender.

$$Q_a = \frac{A_e}{A_g} \qquad \text{(Eq. E7-16)}$$

where

A_e	=	the effective area based on the reduced effective width, b_e
b_e	=	$1.92t\sqrt{\frac{E}{f}}\left[1 - \frac{0.34}{(b\ /\ t)}\sqrt{\frac{E}{f}}\right] \leq b$ (Eq. E7-17)
f	=	F_{cr} calculated based on $Q = 1.0$
F_{cr}	=	$Q\left(0.658^{QF_y/F_e}\right)F_y = 1.0\left[0.658^{1.0(50)/(38.3)}\right]50 = 29.0\ ksi$ because $\frac{KL}{r} = 86.5 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{1.0(50)}} = 113$

$$b_e = 1.92(0.25)\sqrt{\frac{29,\ 000}{29.0}}\left[1 - \frac{0.34}{(15/0.25)}\sqrt{\frac{29,\ 000}{29.0}}\right] = 12.5 \text{ in} < 15 \text{ in}$$

$$A_e = b_e t_w + 2b_f t_f = 12.5(0.25) + 2(8.0)(1.0) = 19.1 \text{ in}^2$$

$$Q_a = \frac{19.1}{19.8} = 0.965$$

$$Q = Q_s Q_a = 1.0 \times 0.965 = 0.965$$

$$\frac{KL}{r} = 86.5 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{0.965(50)}} = 115$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.965\left[0.658^{0.965(50)/(38.3)}\right]50 = 28.5 \text{ ksi}$$

<u>Nominal Compressive Strength</u>

$$P_n = F_{cr}A_g = 28.5(19.8) = 564 \text{ kips} \qquad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(564) = 508\ kips$

ASD: $P_n / \Omega_c = {}^{564}\!/_{1.67} = 338\ kips$

Table 763: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
$\phi_c P_n$ (kips) [LRFD]	508	513	1.0%
P_n / Ω_c (kips) [ASD]	338	341	0.9%

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Built up I E.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 2
UNIT INCHES KIP
WIDE FLANGE
I
19.75 17 0.25 8 1 1095.65 85.3529 5.41146 4.25 10.6667 8 1
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 UPTABLE 2 I
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
```

```
2 FY -70
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -210
UNIT INCHES KIP
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 65 ALL
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
STP 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 65 ALL
FYLD 50 ALL
METHOD ASD
STP 2 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                      (UPT)
                           PASS   Eq. H1-1a            0.819         3
                      420.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     86.586  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.819(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   4.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.067E+01     Ayy:   4.250E+00     Cw:  5.461E+03             |
| Szz:   1.289E+02     Syy:   2.134E+01                                 |
```

```
| Izz:   1.096E+03     Iyy:   8.535E+01     Ix:  5.411E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       180.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     4.00     N/A     11.22    T.B4.1-4     |
|                Slender        60.00     N/A     36.33    T.B4.1-10    |
| Flexure     :  Compact         4.00     9.27    19.90    T.B4.1-2     |
|                Compact        60.00    91.69   139.00    T.B4.1-9     |
    STAAD PLANE                                             -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   8.89E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   9.63E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    4.20E+02   8.03E+02   0.523      Sec. E1       3     0.00 |
| Min Buck    4.20E+02   5.13E+02   0.819      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    4.20E+02   6.84E+02   0.614      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.86E+01    24.17   4.52E+01   5.02E+02   8.93E+02        |
| Min Buck    1.91E+01    86.59   2.89E+01   3.91E+01   5.70E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.88E+01   3.85E+01   7.60E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   2.88E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.15E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.07E+01     1.00     1.20     8.00   3.20E+02            |
| Local-Y     4.25E+00     1.00     5.00    60.00   1.27E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   6.39E+03   0.000      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   1.45E+03   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.10E+03   0.00E+00                                       |
```

```
| Minor       1.61E+03   0.00E+00                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                              |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       0.00E+00   5.44E+03   0.000      Eq. F2-2      3     0.00   |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb         |
| Major       6.04E+03   0.00E+00    1.00    89.22   311.02   180.00      |
     STAAD PLANE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.819      Eq. H1-1a     3     0.00              |
| Flexure Tens           0.000      Eq. H1-1b     3     0.00              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          5.44E+03   0.00E+00   5.13E+02                    |
|                       1.45E+03   0.00E+00   4.20E+02                    |
| Flexure Tens          5.44E+03   0.00E+00   8.89E+02                    |
|                       1.45E+03   0.00E+00   0.00E+00                    |
|--------------------------------------------------------------------------|
    58. LOAD LIST 4
    59. PARAMETER 2
    60. CODE AISC UNIFIED 2005
    61. FU 65 ALL
    62. FYLD 50 ALL
    63. METHOD ASD
    64. STP 2 ALL
    65. TRACK 2 ALL
    66. CHECK CODE ALL
 STEEL DESIGN
     STAAD PLANE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                         (UPT)
                           PASS   Eq. H1-1a            0.820        4
                      280.00 C          0.00           0.00       0.00
|--------------------------------------------------------------------------|
| SLENDERNESS                                                              |
| Actual Slenderness Ratio    :     86.586  L/C  :     4                   |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      4    Ratio     :     0.820(PASS)                    |
|          Loc  :    0.00   Condition :   Eq. H1-1a                        |
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   2.800E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  |
```

```
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                             |
| Azz:   1.067E+01     Ayy:   4.250E+00     Cw:  5.461E+03                     |
| Szz:   1.289E+02     Syy:   2.134E+01                                        |
| Izz:   1.096E+03     Iyy:   8.535E+01     Ix:  5.411E+00                     |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                          |
| Fyld:          50.000   Fu:          65.000                                  |
|-----------------------------------------------------------------------------|
| Actual Member Length:       180.000                                          |
| Design Parameters                                                            |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00               |
|-----------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE              |
|                STIFFENED                                                     |
| Compression :  Non-Slender     4.00     N/A     11.22    T.B4.1-4           |
|                Slender        60.00     N/A     36.33    T.B4.1-10          |
| Flexure     :  Compact         4.00     9.27    19.90    T.B4.1-2           |
|                Compact        60.00    91.69   139.00    T.B4.1-9           |
    STAAD PLANE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                      |
|                                                                              |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Yield       0.00E+00   5.91E+02   0.000      Eq. D2-1      4     0.00       |
| Rupture     0.00E+00   6.42E+02   0.000      Eq. D2-2      4     0.00       |
|-----------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                                  |
|                                                                              |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Maj Buck    2.80E+02   5.35E+02   0.524      Sec. E1       4     0.00       |
| Min Buck    2.80E+02   3.41E+02   0.820      Eq. E7-1      4     0.00       |
| Flexural                                                                     |
| Tor Buck    2.80E+02   4.55E+02   0.615      Eq. E7-1      4     0.00       |
| Intermediate                                                                 |
| Results     Eff Area     KL/r      Fcr        Fe         Pn                 |
| Maj Buck    1.86E+01    24.17   4.52E+01   5.02E+02   8.93E+02              |
| Min Buck    1.91E+01    86.59   2.89E+01   3.91E+01   5.70E+02              |
| Flexural       Ag       Fcr        Pn                                        |
| Tor Buck    1.88E+01   3.85E+01   7.60E+02                                  |
|-----------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                              |
|                                                                              |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Local-Z     0.00E+00   1.92E+02   0.000      Eq. G2-1      4     0.00       |
| Local-Y     0.00E+00   7.63E+01   0.000      Eq. G2-1      4     0.00       |
| Intermediate                                                                 |
| Results        Aw        Cv        Kv      h/tw      Vn                      |
| Local-Z     1.07E+01     1.00      1.20     8.00   3.20E+02                  |
| Local-Y     4.25E+00     1.00      5.00    60.00   1.27E+02                  |
|-----------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                                   |
|                                                                              |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
```

```
| Major       0.00E+00   4.25E+03   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   9.65E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.10E+03   0.00E+00                                       |
| Minor       1.61E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   3.62E+03   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb      |
| Major       6.04E+03   0.00E+00    1.00    89.22   311.02   180.00    |
    STAAD PLANE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.820      Eq. H1-1a     4     0.00            |
| Flexure Tens           0.000      Eq. H1-1b     4     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.62E+03   0.00E+00   3.41E+02                  |
|                       9.65E+02   0.00E+00   2.80E+02                  |
| Flexure Tens          3.62E+03   0.00E+00   5.91E+02                  |
|                       9.65E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Built up I E.3

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.3, pp.E-14 - E.17
3. Seaburg, Paul A. and Charles J. Carter. 1997. Design Guide 9: *Torsional Analysis of Structural Steel members*. AISC: Chicago, IL.

Details

From reference (2):

> Determine if a built-up, ASTM A572 grade 50 column with PL3/8 in.×102 in. flanges and a PL1/4 in.×7 1/4 in. web has sufficient available strength to carry a dead load of 40 kips and a live load of 120 kips in axial compression. The column unbraced length is 15 ft in both axes and the ends are pinned.

Validation

Ultimate load:

$$P_u = 1.2(40\text{ kips}) + 1.6(120\text{ kips}) = 240\text{ kips}$$

Service load:

$$P_a = 40\text{ kips} + 120\text{ kips} = 160\text{ kips}$$

Section Properties

Built-up section properties (ignoring fillet welds):

$$A_g = 2(10.5)(0.375) + 7.25(0.25) = 9.69\text{ in}^2$$

$$I_x = \Sigma A d^2 + \Sigma \frac{bh^3}{12} = 2\left[10.5 \times 0.375(3.81)^2 + \frac{10.5(0.25)^3}{12}\right] + \frac{0.25(7.25)^3}{12} = 122\text{ in}^4$$

$$I_y = 2\left(\frac{0.375 \times 10.5^3}{12}\right) + \frac{7.25 \times 0.25^3}{12} = 72.4\text{ in}^4$$

$$r_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{72.4}{9.69}} = 2.73\text{ in}$$

From the User Note in Section E4 of Reference 1:

$$C_w = \frac{I_y h_o^2}{4} = \frac{72.4(7.63)^2}{4} = 1,050\text{ in}^6$$

From Eq. 3.4 in Reference 3:

$$J = \Sigma \frac{bt^3}{3} = \frac{2(10.5)(0.375)^3 + 7.25(0.25)^3}{3} = 0.407\text{ in}^4$$

Slenderness

Check for slender web and then determine the stiffened element (web) reduction factor.

Web slenderness:

$$\lambda = \frac{h}{t} = \frac{7.25}{0.25} = 29.0$$

The limiting web slenderness ratio, λ_r:

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,000}{50}} = 35.9 > \lambda \qquad \text{(Table B4.1a, case 5)}$$

Thus, the web is not slender and there is no reduction (i.e., Q_a *1.0)*

Check for slender flanges and then determine the unstiffened element (flange) reduction factor.

$$k_c = \frac{4}{\sqrt{h/t_w}} = \frac{4}{\sqrt{7.25/0.25}} = 0.743 \qquad \text{(Table B4.1b note a)}$$

$0.35 < k_c < 0.76$, OK.

Flange slenderness:

$$\lambda = \frac{b}{t} = \frac{5.25}{0.375} = 14.0$$

The limiting flange slenderness ratio, λ_r:

$$\lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.743(29,000)}{50}} = 13.3 < \lambda \qquad \text{(Table B4.1a, case 2)}$$

Thus, the flanges are slender.

$$0.64\sqrt{\frac{k_c E}{F_y}} = 13.3 < \frac{b}{t} = 14.0 < 1.17\sqrt{\frac{k_c E}{F_y}} = 1.17\sqrt{\frac{0.743(29,\ 000)}{50}} = 24.3$$

$$Q_s = 1.415 - 0.65\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{k_c E}} = 1.415 - 0.65(14.0)\sqrt{\frac{50}{0.743(29,\ 000)}} = 0.977 \quad \text{(Eq. E7-8)}$$

$$Q = Q_s Q_a = 0.977(1.0) = 0.977$$

Elastic Flexural Buckling Stress

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

$$\frac{K_y L}{r_y} = \frac{1.0(15.0)(12)}{2.73} = 65.9$$

$$F_e = \frac{\pi^2 E}{(KL\ / r)^2} = \frac{\pi^2 29,\ 000}{(65.9)^2} = 65.9 \text{ ksi} \quad \text{(Eq. E3-4)}$$

Nominal Compressive Strength

$$\frac{KL}{r} = 65.9 < 4.71\sqrt{\frac{E}{QF_y}} = 4.71\sqrt{\frac{29,\ 000}{0.977(50)}} = 115$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.977\left[0.658^{0.977(50)/(65.9)}\right]50 = 35.8 \text{ ksi} \quad \text{(Eq E7-2)}$$

$$P_n = F_{cr}A_g = 35.8(6.69) = 347 \text{ kips} \quad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_c P_n = 0.90(347) = 312\ kips$

ASD: $P_n\ /\ \Omega_c = {}^{347}\!/_{1.67} = 208\ kips$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	312	314	negligible	
P_n/Ω_c (kips) [ASD]	208	209	negligible	
Slenderness ratio	65.9	65.86	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Built up I E.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 31-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
```

```
JOINT COORDINATES
1 0 0 0; 2 0 15 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 2
UNIT INCHES KIP
WIDE FLANGE
I
9.6875 8 0.25 10.5 0.375 122.496 72.361 0.406901 2 5.25 10.5 0.375
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 65 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL_36_KSI ALL
MEMBER PROPERTY AMERICAN
1 UPTABLE 2 I
UNIT FEET KIP
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MY MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -120
UNIT INCHES KIP
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 65 ALL
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
```

```
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 65 ALL
FYLD 50 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                         (UPT)
                           PASS   Eq. H1-1a            0.764         3
                      240.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     65.861  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.764(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   5.250E+00     Ayy:   2.000E+00     Cw:  1.052E+03              |
| Szz:   3.062E+01     Syy:   1.378E+01                                 |
| Izz:   1.225E+02     Iyy:   7.236E+01     Ix:  4.069E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       180.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        14.00     N/A     13.49    T.B4.1-3     |
|                Non-Slender    29.00     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact    14.00     9.15    24.08    T.B4.1-1     |
|                Compact        29.00    90.55   137.27    T.B4.1-9     |
      STAAD PLANE                                            -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
```

```
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   4.36E+02   0.000      Eq. D2-1      3     0.00  |
| Rupture     0.00E+00   4.72E+02   0.000      Eq. D2-2      3     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.40E+02   3.57E+02   0.672      Sec. E1       3     0.00  |
| Min Buck    2.40E+02   3.14E+02   0.764      Eq. E7-1      3     0.00  |
| Flexural                                                                |
| Tor Buck    2.40E+02   3.21E+02   0.747      Eq. E7-1      3     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr         Fe         Pn           |
| Maj Buck    9.69E+00    50.62   4.09E+01   1.12E+02   3.97E+02         |
| Min Buck    9.69E+00    65.86   3.60E+01   6.60E+01   3.49E+02         |
| Flexural       Ag        Fcr        Pn                                  |
| Tor Buck    9.69E+00   3.68E+01   3.57E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.42E+02   0.000      Eq. G2-1      3     0.00  |
| Local-Y     0.00E+00   6.00E+01   0.000      Eq. G2-1      3     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     5.25E+00     1.00     1.20    14.00   1.58E+02             |
| Local-Y     2.00E+00     1.00     0.00    29.00   6.00E+01             |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.50E+03   0.000      Eq. F2-1      3     0.00  |
| Minor       0.00E+00   9.35E+02   0.000      Eq. F6-1      3     0.00  |
| Intermediate   Mn        My                                             |
| Major       1.67E+03   0.00E+00                                         |
| Minor       1.04E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.36E+03   0.000      Eq. F2-2      3     0.00  |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       1.51E+03   0.00E+00    1.00   115.84   354.09   180.00     |
     STAAD PLANE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.33E+03   0.000      Eq. F3-1      3     0.00  |
| Minor       0.00E+00   7.73E+02   0.000      Eq. F6-2      3     0.00  |
| Intermediate   Mn        Fcr                                            |
```

```
| Major       1.47E+03   0.00E+00                                        |
| Minor       8.58E+02   0.00E+00                                        |
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                      RATIO      CRITERIA    L/C     LOC                |
| Flexure Comp         0.764      Eq. H1-1a     3     0.00               |
| Flexure Tens         0.000      Eq. H1-1b     3     0.00               |
| Intermediate        Mcx /      Mrx /      Pc /                         |
|                     Mcy        Mry        Pr                           |
| Flexure Comp        1.33E+03   0.00E+00   3.14E+02                     |
|                     7.73E+02   0.00E+00   2.40E+02                     |
| Flexure Tens        1.33E+03   0.00E+00   4.36E+02                     |
|                     7.73E+02   0.00E+00   0.00E+00                     |
|------------------------------------------------------------------------|
    65. LOAD LIST 4
    66. PARAMETER 2
    67. CODE AISC UNIFIED 2005
    68. FU 65 ALL
    69. FYLD 50 ALL
    70. METHOD ASD
    71. TRACK 2 ALL
    72. CHECK CODE ALL
 STEEL DESIGN
      STAAD PLANE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   I                        (UPT)
                           PASS   Eq. H1-1a           0.765         4
                      160.00 C          0.00           0.00       0.00
|------------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :     65.861  L/C  :     4                 |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     0.765(PASS)                  |
|          Loc  :    0.00   Condition :   Eq. H1-1a                      |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   1.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00                |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   5.250E+00     Ayy:   2.000E+00     Cw:  1.052E+03              |
| Szz:   3.062E+01     Syy:   1.378E+01                                  |
| Izz:   1.225E+02     Iyy:   7.236E+01     Ix:  4.069E-01              |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:          50.000   Fu:          65.000                            |
|------------------------------------------------------------------------|
| Actual Member Length:       180.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
```

```
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        14.00     N/A     13.49    T.B4.1-3     |
|                Non-Slender    29.00     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact    14.00     9.15    24.08    T.B4.1-1     |
|                Compact        29.00    90.55   137.27    T.B4.1-9     |
     STAAD PLANE                                               -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   2.90E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   3.15E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    1.60E+02   2.38E+02   0.674      Sec. E1       4     0.00 |
| Min Buck    1.60E+02   2.09E+02   0.765      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    1.60E+02   2.14E+02   0.749      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    9.69E+00    50.62   4.09E+01   1.12E+02   3.97E+02        |
| Min Buck    9.69E+00    65.86   3.60E+01   6.60E+01   3.49E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    9.69E+00   3.68E+01   3.57E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   9.43E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   4.00E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     5.25E+00     1.00     1.20    14.00   1.58E+02            |
| Local-Y     2.00E+00     1.00     0.00    29.00   6.00E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   9.97E+02   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   6.22E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.67E+03   0.00E+00                                       |
| Minor       1.04E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   9.02E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       1.51E+03   0.00E+00    1.00   115.84   354.09   180.00   |
```

```
     STAAD PLANE                                                 -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      0.00E+00   8.82E+02   0.000      Eq. F3-1      4     0.00 |
| Minor      0.00E+00   5.14E+02   0.000      Eq. F6-2      4     0.00 |
| Intermediate   Mn        Fcr                                         |
| Major      1.47E+03   0.00E+00                                       |
| Minor      8.58E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.765      Eq. H1-1a     4     0.00              |
| Flexure Tens         0.000      Eq. H1-1b     4     0.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        8.82E+02   0.00E+00   2.09E+02                   |
|                     5.14E+02   0.00E+00   1.60E+02                   |
| Flexure Tens        8.82E+02   0.00E+00   2.90E+02                   |
|                     5.14E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 C Flex Mem F.2-1

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.2-1a, p.F-15

Details

From reference (2):

> Select an ASTM A36 channel to serve as a roof edge beam with a simple span of 25 ft. Limit the live load deflection to L/360. The nominal loads are a uniform dead load of 0.23 kip/ft and a uniform live load of 0.69 kip/ft. The beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.23 \text{ k/ft}) + 1.6(0.69 \text{ k/ft}) = 1.38 \text{ k/ft}$$

$$M_u = \frac{1.38(25)^2}{8} = 108 \text{ ft-k}$$

Service moment:

$$w_a = 0.23 \text{ k/ft} + 0.69 \text{ k/ft} = 0.92 \text{ k/ft}$$

$$M_a = \frac{0.92(25)^2}{8} = 71.9 \text{ ft-k}$$

Try a C15×33.9, which has a plastic section modulus, $Z_x = 50.8\ in^3$.

All ASTM A36 grade C-shapes are compact. The member is continuously braced, so therefore the yielding limit state governs.

$$M_n = M_p = F_y \times Z_x = 36 \times 50.8 = 1{,}830 \text{ in-k} = 152 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(152) = 137\ ft\text{-}k$

ASD: $M_n \big/ \Omega_b = {}^{152}\!\big/_{1.67} = 91.0\ ft\text{-}k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	137	137	none	
M_n/Ω_b (ft·k) [ASD]	91	91.3	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 C Flex Mem F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 07-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
```

```
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FLX 2 ALL
FYLD 5184 ALL
FU 8352 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FLX 2 ALL
FYLD 5184 ALL
FU 8352 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b            2.548         3
                        0.00            0.00        -107.81       12.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    333.952  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    12.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     2.548(FAIL)                |
|          Loc  :   12.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.078E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
```

```
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02                 |
| Szz:   4.200E+01     Syy:   3.178E+00                                    |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:        5183.999   Fu:        8351.999                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        25.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          |
|                STIFFENED                                                 |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3        |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10       |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1        |
|                Compact        34.25   106.72   161.78    T.B4.1-9        |
    STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Yield      0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00     |
| Rupture    0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00     |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Maj Buck   0.00E+00   2.79E+02   0.000      Sec. E1       3     0.00     |
| Min Buck   0.00E+00   2.03E+01   0.000      Eq. E3-1      3     0.00     |
| Flexural                                                                 |
| Tor Buck   0.00E+00   2.07E+02   0.000      Eq. E4-1      3     0.00     |
| Intermediate                                                             |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02            |
| Min Buck   6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01            |
| Flexural      Ag        Fcr        Pn                                    |
| Tor Buck   6.94E-02   3.31E+03   2.30E+02                                |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Local-Z    0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00     |
| Local-Y    1.72E+01   1.17E+02   0.148      Eq. G2-1      3     0.00     |
| Intermediate                                                             |
| Results       Aw        Cv       Kv      h/tw      Vn                    |
| Local-Z    3.07E-02     1.00     1.20     5.23   9.55E+01                |
| Local-Y    4.17E-02     1.00     5.00    34.25   1.30E+02                |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Major      1.08E+02   1.37E+02   0.786      Eq. F2-1      3    12.50     |
| Minor      0.00E+00   1.37E+01   0.000      Eq. F6-1      3     0.00     |
```

```
| Intermediate   Mn        My                                            |
| Major       1.52E+02   0.00E+00                                        |
| Minor       1.53E+01   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.08E+02   4.23E+01   2.548      Eq. F2-3      3    12.50 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       4.70E+01   0.00E+00    1.00     3.74    14.49    25.00    |
     STAAD SPACE                                             -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          2.548      Eq. H1-1b     3    12.50              |
| Flexure Tens          2.548      Eq. H1-1b     3    12.50              |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         4.23E+01   1.08E+02   2.03E+01                   |
|                      1.37E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         4.23E+01   1.08E+02   3.24E+02                   |
|                      1.37E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
   49. LOAD LIST 4
   50. PARAMETER 2
   51. CODE AISC UNIFIED 2005
   52. FLX 2 ALL
   53. FYLD 5184 ALL
   54. FU 8352 ALL
   55. METHOD ASD
   56. TRACK 2 ALL
   57. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                             -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                  (AISC SECTIONS)
                           FAIL   Eq. H1-1b           2.553         4
                        0.00            0.00         -71.88       12.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                            |
| Actual Slenderness Ratio    :    333.952  L/C  :     4                 |
| Allowable Slenderness Ratio :    300.000  LOC  :    12.50              |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      4    Ratio     :     2.553(FAIL)                  |
|          Loc  :   12.50   Condition :   Eq. H1-1b                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                          |
```

```
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -7.188E+01                 |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02               |
| Szz:   4.200E+01     Syy:   3.178E+00                                  |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00               |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3     |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1     |
|                Compact        34.25   106.72   161.78    T.B4.1-9     |
     STAAD SPACE                                               -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.85E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   1.35E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.38E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r       Fcr        Fe         Pn          |
| Maj Buck    6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02        |
| Min Buck    6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    6.94E-02   3.31E+03   2.30E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.15E+01   7.76E+01   0.148      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     3.07E-02     1.00      1.20     5.23   9.55E+01            |
| Local-Y     4.17E-02     1.00      5.00    34.25   1.30E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
```

```
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       7.19E+01   9.13E+01   0.788      Eq. F2-1      4    12.50 |
| Minor       0.00E+00   9.14E+00   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.52E+02   0.00E+00                                      |
| Minor       1.53E+01   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       7.19E+01   2.82E+01   2.553      Eq. F2-3      4    12.50 |
| Intermediate   Mn         Me         Cb       Lp        Lr       Lb    |
| Major       4.70E+01   0.00E+00    1.00     3.74     14.49    25.00   |
    STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          2.553      Eq. H1-1b     4    12.50             |
| Flexure Tens          2.553      Eq. H1-1b     4    12.50             |
| Intermediate         Mcx /      Mrx /       Pc /                      |
|                      Mcy        Mry         Pr                        |
| Flexure Comp         2.82E+01   7.19E+01   1.35E+01                   |
|                      9.14E+00   0.00E+00   0.00E+00                   |
| Flexure Tens         2.82E+01   7.19E+01   2.16E+02                   |
|                      9.14E+00   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 C LTB Test F.2-2

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.2-2, pp.F-16 - F-19

Details

From reference (2):

> Check the C15×33.9 beam selected in V. AISC 360-05 C Flex Mem F.2-1 (on page 5441), assuming it is braced at the ends and the fifth points rather than continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.23\text{ k/ft}) + 1.6(0.69\text{ k/ft}) = 1.38\text{ k/ft}$$

$$M_u = \frac{1.38(25)^2}{8} = 108 \text{ ft-k}$$

Service moment:

$$w_a = 0.23 \text{ k/ft} + 0.69 \text{ k/ft} = 0.92 \text{ k/ft}$$

$$M_a = \frac{0.92(25)^2}{8} = 71.9 \text{ ft-k}$$

The unbraced length, >$L_b = 5\ ft$.

Try a C15×33.9, which has a plastic section modulus, $Z_x = 50.8\ in^3$.

The lateral-torsional buckling modification factor, $C_b = 1.0$ for the center span of this bracing & loading condition.

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76(0.901)\sqrt{\frac{29,000}{36}}\left(\frac{1}{12}\right) = 3.75\text{ft} \quad \text{(F2-5)}$$

$$L_r = \pi r_{ts} \sqrt{\frac{E}{0.7F_y}} = \pi(1.13)\sqrt{\frac{29,000}{0.7 \times 36}}\left(\frac{1}{12}\right) = 10.0 \text{ ft} \quad \text{(F2-6)}$$

Note: This is the simplified, conservative calculation for L_r per the User Note used to verify that the section is in the plastic-elastic range. The reference uses the more exact calculation but the resulting flexural strength values are essentially equal.

$$L_p < L_b \leq L_r$$

All ASTM A36 grade C-shapes are compact.

The nominal flexural strength for the flexural yielding limit state is:

$$M_n = M_p = F_y \times Z_x = 36 \times 50.8 = 1{,}830 \text{ in-k} = 152 \text{ ft-k}$$

The nominal flexural strength for the lateral-torsional buckling limit state is:

$$M_n = C_b\left[M_p - (M_p - 0.7F_y S_x)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right]$$

$$= 1.0\left[1,830 - (1,830 - 0.7 \times 36 \times 42)\left(\frac{5 - 3.75}{10 - 3.75}\right)\right]\left(\frac{1}{12}\right) = 145 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(145) = 131\ ft\text{-}k$

ASD: $M_n / \Omega_b = {}^{131}/_{1.67} = 86.8\ ft\text{-}k$

Table 764: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (kips) [LRFD]	131	130	negligible	
M_n/Ω_b (kips) [ASD]	86.8	86.7	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 C LTB Test F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 30-Mar-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.23
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.69
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FYLD 5184 ALL
FU 8352 ALL
CB 1 ALL
FLX 1 ALL
METHOD LRFD
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
```

```
CODE AISC UNIFIED 2005
FYLD 5184 ALL
FU 8352 ALL
CB 1 ALL
FLX 1 ALL
METHOD ASD
TRACK 2 ALL
UNB 5 ALL
UNT 5 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                  (AISC SECTIONS)
                           FAIL   SLENDERNESS          1.113         3
                        0.00            0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    333.952  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.113(FAIL)                 |
|          Loc  :    0.00   Condition :   SLENDERNESS                   |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.725E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                 |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5183.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3     |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1     |
|                Compact        34.25   106.72   161.78    T.B4.1-9     |
      STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
```

```
                       *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   2.79E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   2.03E+01   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.07E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02        |
| Min Buck    6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    6.94E-02   3.31E+03   2.30E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     1.72E+01   1.17E+02   0.148      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     3.07E-02     1.00     1.20     5.23   9.55E+01            |
| Local-Y     4.17E-02     1.00     5.00    34.25   1.30E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.08E+02   1.37E+02   0.786      Eq. F2-1      3    12.50 |
| Minor       0.00E+00   1.37E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.52E+02   0.00E+00                                       |
| Minor       1.53E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.08E+02   1.30E+02   0.827      Eq. F2-2      3    12.50 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       1.45E+02   0.00E+00    1.00     3.74    14.49     5.00   |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                     RATIO      CRITERIA    L/C     LOC                |
| Flexure Comp         0.827      Eq. H1-1b     3    12.50              |
| Flexure Tens         0.827      Eq. H1-1b     3    12.50              |
```

```
|| Intermediate         Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          1.30E+02   1.08E+02   2.03E+01                    |
|                       1.37E+01   0.00E+00   0.00E+00                    |
| Flexure Tens          1.30E+02   1.08E+02   3.24E+02                    |
|                       1.37E+01   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
    51. LOAD LIST 4
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2005
    54. FYLD 5184 ALL
    55. FU 8352 ALL
    56. CB 1 ALL
    57. FLX 1 ALL
    58. METHOD ASD
    59. TRACK 2 ALL
    60. UNB 5 ALL
    61. UNT 5 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                  (AISC SECTIONS)
                           FAIL   SLENDERNESS          1.113         4
                        0.00            0.00           0.00       0.00
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :    333.952  L/C  :     4                  |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      4    Ratio     :     1.113(FAIL)                   |
|          Loc  :    0.00   Condition :   SLENDERNESS                     |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   1.150E+01     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02               |
| Szz:   4.200E+01     Syy:   3.178E+00                                   |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00               |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        5183.999   Fu:        8351.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        25.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                                |
```

```
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3      |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10     |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1      |
|                Compact        34.25   106.72   161.78    T.B4.1-9      |
     STAAD SPACE                                               -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00  |
| Rupture    0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck   0.00E+00   1.85E+02   0.000      Sec. E1       4     0.00  |
| Min Buck   0.00E+00   1.35E+01   0.000      Eq. E3-1      4     0.00  |
| Flexural                                                              |
| Tor Buck   0.00E+00   1.38E+02   0.000      Eq. E4-1      4     0.00  |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck   6.94E-02    53.45   4.46E+03   1.44E+04   3.10E+02         |
| Min Buck   6.94E-02   333.95   3.24E+02   3.70E+02   2.25E+01         |
| Flexural      Ag        Fcr        Pn                                 |
| Tor Buck   6.94E-02   3.31E+03   2.30E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00  |
| Local-Y    1.15E+01   7.76E+01   0.148      Eq. G2-1      4     0.00  |
| Intermediate                                                          |
| Results       Aw         Cv        Kv      h/tw      Vn              |
| Local-Z    3.07E-02     1.00      1.20     5.23   9.55E+01            |
| Local-Y    4.17E-02     1.00      5.00    34.25   1.30E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      7.19E+01   9.13E+01   0.788      Eq. F2-1      4    12.50  |
| Minor      0.00E+00   9.14E+00   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn         My                                          |
| Major      1.52E+02   0.00E+00                                        |
| Minor      1.53E+01   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      7.19E+01   8.67E+01   0.829      Eq. F2-2      4    12.50  |
| Intermediate   Mn         Me        Cb       Lp       Lr       Lb     |
| Major      1.45E+02   0.00E+00    1.00     3.74    14.49     5.00     |
     STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
```

```
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.829      Eq. H1-1b     4    12.50              |
| Flexure Tens         0.829      Eq. H1-1b     4    12.50              |
| Intermediate        Mcx /      Mrx /      Pc /                        |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        8.67E+01   7.19E+01   1.35E+01                    |
|                     9.14E+00   0.00E+00   0.00E+00                    |
| Flexure Tens        8.67E+01   7.19E+01   2.16E+02                    |
|                     9.14E+00   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 C Strong Axis Shear G.2

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.2, pp.G-5,6

Details

From reference (2):

> Verify the shear strength of a C15×33.9 channel with end shears of 17.5 kips from dead load and 52.5 kips from live load.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	117	117	none	
V_n/Ω_v (kips) [ASD]	77.8	77.6	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 C Strong Axis Shear G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X33
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -35
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -105
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
```

```
 =======================================================================
 *     1 ST   C15X33                   (AISC SECTIONS)
                          FAIL   Eq. H1-1b           1.547         3
                       0.00            0.00       -2520.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     53.432  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.547(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.050E+02     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.520E+03               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                 |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3     |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1     |
|                Compact        34.25   106.72   161.78    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   3.24E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   4.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   3.23E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   2.79E+02   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.94E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.00E+01     8.55   3.59E+01   3.91E+03   3.59E+02        |
| Min Buck    1.00E+01    53.43   3.10E+01   1.00E+02   3.10E+02        |
```

```
| Flexural       Ag       Fcr        Pn                                    |
| Tor Buck    1.00E+01   3.27E+01   3.27E+02                               |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00  |
| Local-Y     1.05E+02   1.17E+02   0.900      Eq. G2-1      3     0.00  |
| Intermediate                                                           |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     4.42E+00     1.00     1.20     5.23   9.55E+01             |
| Local-Y     6.00E+00     1.00     5.00    34.25   1.30E+02             |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       2.52E+03   1.65E+03   1.531      Eq. F2-1      3    24.00  |
| Minor       0.00E+00   1.65E+02   0.000      Eq. F6-1      3     0.00  |
| Intermediate   Mn        My                                            |
| Major       1.83E+03   0.00E+00                                        |
| Minor       1.83E+02   0.00E+00                                        |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       2.52E+03   1.63E+03   1.547      Eq. F2-2      3    24.00  |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb       |
| Major       1.81E+03   0.00E+00    1.00    44.87   173.87    48.00     |
    STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       *******************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         1.547      Eq. H1-1b     3    24.00              |
| Flexure Tens         1.547      Eq. H1-1b     3    24.00              |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        1.63E+03   2.52E+03   2.79E+02                   |
|                     1.65E+02   0.00E+00   0.00E+00                   |
| Flexure Tens        1.63E+03   2.52E+03   3.24E+02                   |
|                     1.65E+02   0.00E+00   0.00E+00                   |
|------------------------------------------------------------------------|
   48. LOAD LIST 4
   49. PARAMETER 2
   50. CODE AISC UNIFIED 2005
   51. METHOD ASD
   52. FYLD 36 ALL
   53. FU 58 ALL
   54. TRACK 2 ALL
   55. CHECK CODE ALL
 STEEL DESIGN
    STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       *******************************************
```

```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C15X33                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           1.550         4
                        0.00            0.00       -1680.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     53.432  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     1.550(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   7.000E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.680E+03               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.420E+00     Ayy:   6.000E+00     Cw:  3.555E+02             |
| Szz:   4.200E+01     Syy:   3.178E+00                                 |
| Izz:   3.150E+02     Iyy:   8.070E+00     Ix:  1.010E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.23     N/A     15.89    T.B4.1-3     |
|                Non-Slender    34.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         5.23    10.79    28.38    T.B4.1-1     |
|                Compact        34.25   106.72   161.78    T.B4.1-9     |
     STAAD SPACE                                             -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   2.16E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   2.90E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   2.15E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   1.85E+02   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.96E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
```

```
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.00E+01     8.55   3.59E+01   3.91E+03   3.59E+02       |
| Min Buck    1.00E+01    53.43   3.10E+01   1.00E+02   3.10E+02       |
| Flexural       Ag       Fcr        Pn                                |
| Tor Buck    1.00E+01   3.27E+01   3.27E+02                           |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.72E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     7.00E+01   7.76E+01   0.902      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     4.42E+00     1.00      1.20     5.23   9.55E+01           |
| Local-Y     6.00E+00     1.00      5.00    34.25   1.30E+02           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.68E+03   1.10E+03   1.534      Eq. F2-1      4    24.00 |
| Minor       0.00E+00   1.10E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                          |
| Major       1.83E+03   0.00E+00                                     |
| Minor       1.83E+02   0.00E+00                                     |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.68E+03   1.08E+03   1.550      Eq. F2-2      4    24.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb      |
| Major       1.81E+03   0.00E+00    1.00    44.87   173.87    48.00    |
    STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp         1.550      Eq. H1-1b     4    24.00            |
| Flexure Tens         1.550      Eq. H1-1b     4    24.00            |
| Intermediate        Mcx /      Mrx /      Pc /                     |
|                     Mcy        Mry        Pr                       |
| Flexure Comp        1.08E+03   1.68E+03   1.85E+02                  |
|                     1.10E+02   0.00E+00   0.00E+00                  |
| Flexure Tens        1.08E+03   1.68E+03   2.16E+02                  |
|                     1.10E+02   0.00E+00   0.00E+00                  |
|----------------------------------------------------------------------|
```

V. AISC 360-05 C Weak Axis Shear G.7

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005

2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.7, p.G-12

Details

From reference (2):

> Verify the strength of a C9×20 ASTM A36 channel with end shears of 5 kips from dead load and 15 kips from live load in the weak direction.

Validation

Ultimate load:

$$V_u = 1.2(5.0\text{ kips}) + 1.6(15.0\text{ kips}) = 30.0\text{ kips}$$

Service load:

$$V_a = 5.0\text{ kips} + 15.0\text{ kips} = 20.0\text{ kips}$$

Area of the flanges for shear:

$$A_w = 2b_f t_f = 2(2.65)(0.413) = 2.19\text{ in}^2$$

Section G2.1 of Reference 1 gives $k_v = 1.2$.

$$\frac{b_f}{t_f} = \frac{2.65}{0.413} = 6.42 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{1.2(29,\ 000)}{36}} = 34.2$$

Therefore, $C_v = 1.0$.

<u>Nominal Shear Strength</u>

$$V_n = 0.6F_y A_w C_v = 0.6(36)(2.19)(1.0) = 47.3\text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(47.3) = 42.6\ kips$

ASD: $V_n / \Omega_v = {}^{47.3}/_{1.67} = 28.3\ kips$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	42.6	42.6	none	
V_n / Ω_v (kips) [ASD]	28.3	28.3	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 C Weak Axis Shear G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
```

```
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C9X20
CONSTANTS
BETA 90 ALL
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -10
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -30
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FYLD 36 ALL
FU 58 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
```

```
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C9X20                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b           11.619         3
                        0.00        -720.00           0.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     74.912  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :    11.619(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -3.000E+01              |
| Mx:   0.000E+00     My:  -7.200E+02     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.189E+00     Ayy:   4.032E+00     Cw:  3.933E+01              |
| Szz:   1.353E+01     Syy:   1.195E+00                                 |
| Izz:   6.090E+01     Iyy:   2.410E+00     Ix:  4.270E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.42     N/A     15.89    T.B4.1-3     |
|                Non-Slender    18.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         6.42    10.79    28.38    T.B4.1-1     |
|                Compact        18.25   106.72   161.78    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   1.90E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   2.55E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   1.88E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   1.42E+02   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.70E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe          Pn          |
```

```
| Maj Buck    5.87E+00    14.90   3.56E+01   1.29E+03   2.09E+02         |
| Min Buck    5.87E+00    74.91   2.68E+01   5.10E+01   1.57E+02         |
| Flexural       Ag       Fcr        Pn                                  |
| Tor Buck    5.87E+00   3.22E+01   1.89E+02                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     3.00E+01   4.26E+01   0.705      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   7.84E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     2.19E+00     1.00     1.20     6.42   4.73E+01            |
| Local-Y     4.03E+00     1.00     5.00    18.25   8.71E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   5.48E+02   0.000      Eq. F2-1      3     0.00 |
| Minor      -7.20E+02   6.20E+01  11.619      Eq. F6-1      3    24.00 |
| Intermediate   Mn        My                                          |
| Major       6.08E+02   0.00E+00                                      |
| Minor       6.89E+01   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   5.21E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb     |
| Major       5.78E+02   0.00E+00    1.00    32.01   174.62    48.00   |
    STAAD SPACE                                              -- PAGE NO.
5
                     STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp        11.619      Eq. H1-1b     3    24.00             |
| Flexure Tens        11.619      Eq. H1-1b     3    24.00             |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        5.21E+02   0.00E+00   1.42E+02                   |
|                     6.20E+01  -7.20E+02   0.00E+00                   |
| Flexure Tens        5.21E+02   0.00E+00   1.90E+02                   |
|                     6.20E+01  -7.20E+02   0.00E+00                   |
|-----------------------------------------------------------------------|
   49. LOAD LIST 4
   50. PARAMETER 2
   51. CODE AISC UNIFIED 2005
   52. METHOD ASD
   53. FYLD 36 ALL
   54. FU 58 ALL
   55. TRACK 2 ALL
   56. CHECK CODE ALL
 STEEL DESIGN
    STAAD SPACE                                              -- PAGE NO.
6
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   C9X20                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b          11.642         4
                        0.00         -480.00          0.00       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     74.912  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :    11.642(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:  -2.000E+01              |
| Mx:   0.000E+00     My:  -4.800E+02     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.189E+00     Ayy:   4.032E+00     Cw:  3.933E+01              |
| Szz:   1.353E+01     Syy:   1.195E+00                                 |
| Izz:   6.090E+01     Iyy:   2.410E+00     Ix:  4.270E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.42     N/A     15.89    T.B4.1-3     |
|                Non-Slender    18.25     N/A     42.29    T.B4.1-10    |
| Flexure     :  Compact         6.42    10.79    28.38    T.B4.1-1     |
|                Compact        18.25   106.72   161.78    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.27E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.70E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   1.25E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   9.42E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                              |
```

```
| Tor Buck    0.00E+00   1.13E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    5.87E+00    14.90   3.56E+01   1.29E+03   2.09E+02        |
| Min Buck    5.87E+00    74.91   2.68E+01   5.10E+01   1.57E+02        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    5.87E+00   3.22E+01   1.89E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     2.00E+01   2.83E+01   0.706      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   5.22E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.19E+00     1.00     1.20     6.42   4.73E+01            |
| Local-Y     4.03E+00     1.00     5.00    18.25   8.71E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.64E+02   0.000      Eq. F2-1      4     0.00 |
| Minor      -4.80E+02   4.12E+01  11.642      Eq. F6-1      4    24.00 |
| Intermediate   Mn         My                                          |
| Major       6.08E+02   0.00E+00                                       |
| Minor       6.89E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.46E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       5.78E+02   0.00E+00    1.00    32.01   174.62    48.00   |
    STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          11.642      Eq. H1-1b     4    24.00            |
| Flexure Tens          11.642      Eq. H1-1b     4    24.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.46E+02   0.00E+00   9.42E+01                   |
|                       4.12E+01  -4.80E+02   0.00E+00                   |
| Flexure Tens          3.46E+02   0.00E+00   1.27E+02                   |
|                       4.12E+01  -4.80E+02   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Double L E.5

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.5, pp.E-22 - E-25

Details

From reference (2):

> Verify the strength of a 2L4×3 1/2×3/8 LLBB (3/4-in. separation) strut with a length of 8 ft and pinned ends carrying an axial dead load of 20 kips and live load of 60 kips. Also, calculate the required number of fully tightened or welded intermediate connectors required.

Validation

Refer to reference.

Table 765: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	127	127	none	
P_n / Ω_c (kips) [ASD]	85.0	84.6	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Double L E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 8 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L40356 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
```

```
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -20
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -60
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FU 58 ALL
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FU 58 ALL
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
```

```
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L40356                   (AISC SECTIONS)
                           PASS   Eq. H2-1            0.943         3
                      120.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     76.914  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.943(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.625E+00     Ayy:   1.500E+00     Cw:  0.000E+00              |
| Szz:   2.984E+00     Syy:   3.978E+00                                 |
| Izz:   8.350E+00     Iyy:   1.541E+01     Ix:  2.512E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.67     N/A     12.77    T.B4.1-5     |
|                  N/A          N/A      N/A      N/A      N/A          |
| Flexure     :  Compact         9.33    15.33    25.83    T.B4.1-6     |
|                  N/A          N/A      N/A      N/A      N/A          |
     STAAD SPACE                                              -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.74E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   2.33E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    1.20E+02   1.27E+02   0.943      Sec. E1       3     0.00 |
| Min Buck    1.20E+02   1.46E+02   0.822      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    1.20E+02   1.35E+02   0.887      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
```

```
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    5.36E+00    76.91   2.64E+01   4.84E+01   1.41E+02       |
| Min Buck    5.36E+00    57.51   3.02E+01   8.65E+01   1.62E+02       |
| Flexural       Ag        Fcr        Pn                               |
| Tor Buck    0.00E+00   2.81E+01   1.50E+02                          |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.10E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.92E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn             |
| Local-Z     2.62E+00     1.00     1.20     9.33   5.67E+01           |
| Local-Y     1.50E+00     1.00     1.20    10.67   3.24E+01           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.55E+02   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   2.06E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                         |
| Major       1.72E+02   0.00E+00                                      |
| Minor       2.29E+02   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.09E+03   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       2.32E+03   0.00E+00    1.00     0.00     0.00    96.00   |
    STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                         RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp           0.943      Eq. H2-1      3     0.00          |
| Flexure Tens           0.000      Eq. H2-1      3     0.00          |
| Intermediate          frbw /     frbz /     fra /                   |
|                       Fcbw       Fcbz       Fca                     |
| Flexure Comp          0.00E+00   0.00E+00   2.24E+01                |
|                       5.18E+01   5.18E+01   2.37E+01                |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                |
|                       2.23E+01   5.18E+01   3.24E+01                |
|----------------------------------------------------------------------|
    62. LOAD LIST 4
    63. PARAMETER 2
    64. CODE AISC UNIFIED 2005
    65. METHOD ASD
    66. FU 58 ALL
    67. FYLD 36 ALL
    68. CSPACING 32 ALL
    69. TRACK 2 ALL
    70. CHECK CODE ALL
 STEEL DESIGN
```

```
     STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 LD   L40356                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.945        4
                       80.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     76.914  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.945(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.625E+00     Ayy:   1.500E+00     Cw:  0.000E+00              |
| Szz:   2.984E+00     Syy:   3.978E+00                                 |
| Izz:   8.350E+00     Iyy:   1.541E+01     Ix:  2.512E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.67     N/A     12.77    T.B4.1-5     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         9.33    15.33    25.83    T.B4.1-6     |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.16E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.55E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    8.00E+01   8.46E+01   0.945      Sec. E1       4     0.00 |
```

```
| Min Buck    8.00E+01   9.71E+01   0.824      Eq. E3-1      4     0.00 |
| Flexural                                                             |
| Tor Buck    8.00E+01   9.01E+01   0.888      Eq. E4-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    5.36E+00    76.91   2.64E+01   4.84E+01   1.41E+02       |
| Min Buck    5.36E+00    57.51   3.02E+01   8.65E+01   1.62E+02       |
| Flexural       Ag        Fcr        Pn                               |
| Tor Buck    0.00E+00   2.81E+01   1.50E+02                           |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   3.40E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.94E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv        Kv      h/tw      Vn            |
| Local-Z     2.62E+00     1.00      1.20     9.33   5.67E+01          |
| Local-Y     1.50E+00     1.00      1.20    10.67   3.24E+01          |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.03E+02   0.000      Eq. F9-1      4     0.00 |
| Minor       0.00E+00   1.37E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       1.72E+02   0.00E+00                                      |
| Minor       2.29E+02   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.39E+03   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       2.32E+03   0.00E+00    1.00     0.00     0.00    96.00   |
    STAAD SPACE                                        -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          0.945      Eq. H2-1      4     0.00            |
| Flexure Tens          0.000      Eq. H2-1      4     0.00            |
| Intermediate         frbw /     frbz /     fra /                     |
|                      Fcbw       Fcbz       Fca                       |
| Flexure Comp         0.00E+00   0.00E+00   1.49E+01                  |
|                      3.45E+01   3.45E+01   1.58E+01                  |
| Flexure Tens         0.00E+00   0.00E+00   0.00E+00                  |
|                      1.48E+01   3.45E+01   2.16E+01                  |
|----------------------------------------------------------------------|
```

V. AISC 360-05 Double L E.6

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.6, pp.E-26 - E-30

Details

From reference (2):

> Determine if a 2L5×3×1/4 LLBB (3/4-in. separation) strut with a length of 8 ft and pinned ends has sufficient available strength to support a dead load of 10 kips and live load of 30 kips in axial compression. Also, calculate the required number of fully tightened or welded intermediate connectors.

Validation

Ultimate load:

$$P_u = 1.2(10\text{ kips}) + 1.6(30\text{ kips}) = 60\text{ kips}$$

Service load:

$$P_a = 10\text{ kips} + 30\text{ kips} = 40\text{ kips}$$

Per Table C-A-7.1 in Reference 1, $K = 1.0$ for pinned-pinned columns. Since the unbraced length is the same both directions, by inspection the y-y axis governs.

Slenderness

$$\lambda = \frac{b}{t} = \frac{5.0}{0.25} = 20.0$$

The limiting slenderness ratio, λ_r:

$$\lambda_r = 0.45\sqrt{\frac{E}{F_y}} = 0.45\sqrt{\frac{29,000}{36}} = 12.8 < \lambda \qquad \text{(Table B4.1a, case 3)}$$

Thus, the outstanding legs are slender.

Check for slender web and then determine the stiffened element (web) reduction factor.

$$0.45\sqrt{\frac{E}{F_y}} = 12.8 < \frac{b}{t} = 20.0 < 0.91\sqrt{\frac{E}{F_y}} = 0.91\sqrt{\frac{29,000}{36}} = 25.8$$

$$Q_s = 1.34 - 0.76\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{E}} = 1.34 - 0.76(20.0)\sqrt{\frac{36}{29,000}} = 0.804 \qquad \text{(Eq. E7-11)}$$

$$Q = Q_s Q_a = 0.804(1.0) = 0.804$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y$$

where

F_e	=	$\left(\frac{F_{ey}+F_{ez}}{2H}\right)\left[1-\sqrt{1-\frac{4F_{ey}F_{ez}H}{(F_{ey}+F_{ez})^2}}\right]$ (Eq E4-5)
K_i	=	0.5 for angles back-to-back
r_i	=	$r_z = 0.652\ in$; the radius of gyration for a single angle
a	=	32 in spacing between connectors (using the bolts at 1/3 points from Reference 2)
$\left(\frac{K_yL}{r_y}\right)_m$	=	$\sqrt{\left(\frac{KL}{r}\right)_0^2+\left(\frac{K_ia}{r_i}\right)^2} = \sqrt{\left(\frac{1.0\times 96}{1.33}\right)^2+\left(\frac{0.5\times 32}{0.652}\right)^2} = 76.3$ for $\frac{a}{r_i} = \frac{32\ in}{0.652\ in} = 49.1 > 40$
F_{ey}	=	$\frac{\pi^2E}{\left(\frac{K_yL}{r_y}\right)_m^2} = \frac{\pi^2(29,\ 000)}{(76.3)^2} = 49.2\ ksi$
F_{ez}	=	$\frac{GJ}{A_g\bar{r}_0^2} = \frac{11,\ 200(0.04328)\times 2\ angles}{3.88(2.59)^2} = 37.7\ ksi$

Thus, $\left(\frac{49.2+37.7}{2\times 0.657}\right)\left[1-\sqrt{1-\frac{4(49.2)(37.7)(0.657)}{(49.2+37.7)^2}}\right] = 26.8\ ksi$

Nominal Compressive Strength

$$\frac{QF_y}{2.25} = \frac{0.804(36)}{2.25} = 12.9\text{ ksi} < 26.8\text{ ksi}$$

$$F_{cr} = Q\left(0.658^{QF_y/F_e}\right)F_y = 0.804\left[0.658^{0.804(36)/(26.8)}\right]36 = 18.4\text{ ksi} \quad \text{(Eq E7-2)}$$

$$P_n = F_{cr}A_g = 18.4(3.88) = 71.4\text{ kips} \quad \text{(Eq E7-1)}$$

The available compressive strength:

LRFD: $\phi_cP_n = 0.90(71.4) = 64.3\ kips$

ASD: $P_n\ /\ \Omega_c = {}^{71.4}\!/_{1.67} = 42.8\ kips$

Result Type	Reference	STAAD.Pro	Difference	Comments
ϕ_cP_n (kips) [LRFD]	64.3	63.8	negligible	
P_n/Ω_c (kips) [ASD]	42.8	42.4	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Double L E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 5-Dec-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 8 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE LD L50304 SP 0.0625
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 PINNED
1 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -10
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -30
LOAD COMB 3 ULTIMATE LOADS
1 1.2 2 1.6
LOAD COMB 4 SERVICE LOADS
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FU 58 ALL
```

```
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FU 58 ALL
FYLD 36 ALL
CSPACING 32 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 LD   L50304                   (AISC SECTIONS)
                           PASS   Eq. H2-1            0.945         3
                       60.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     78.939  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.945(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   6.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.500E+00     Ayy:   1.250E+00     Cw:  0.000E+00              |
| Szz:   3.055E+00     Syy:   2.075E+00                                 |
| Izz:   1.022E+01     Iyy:   7.003E+00     Ix:  8.083E-02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        96.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.00     N/A     12.77    T.B4.1-5     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1-6     |
|                  N/A           N/A      N/A      N/A      N/A         |
```

```
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      0.00E+00   1.26E+02   0.000      Eq. D2-1      3     0.00  |
| Rupture    0.00E+00   1.69E+02   0.000      Eq. D2-2      3     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck   6.00E+01   8.72E+01   0.688      Sec. E1       3     0.00  |
| Min Buck   6.00E+01   8.09E+01   0.742      Eq. E7-1      3     0.00  |
| Flexural                                                              |
| Tor Buck   6.00E+01   6.35E+01   0.945      Eq. E7-1      3     0.00  |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck   3.88E+00    59.15   2.50E+01   8.18E+01   9.69E+01        |
| Min Buck   3.88E+00    72.59   2.32E+01   5.43E+01   8.99E+01        |
| Flexural      Ag        Fcr        Pn                                 |
| Tor Buck   3.88E+00   1.82E+01   7.05E+01                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    0.00E+00   2.92E+01   0.000      Eq. G2-1      3     0.00  |
| Local-Y    0.00E+00   2.43E+01   0.000      Eq. G2-1      3     0.00  |
| Intermediate                                                          |
| Results       Aw        Cv       Kv      h/tw      Vn                |
| Local-Z    1.50E+00     1.00     1.20    12.00   3.24E+01             |
| Local-Y    1.25E+00     1.00     1.20    20.00   2.70E+01             |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      0.00E+00   1.58E+02   0.000      Eq. F9-1      3     0.00  |
| Minor      0.00E+00   1.08E+02   0.000      Eq. F6-1      3     0.00  |
| Intermediate   Mn        My                                           |
| Major      1.76E+02   0.00E+00                                        |
| Minor      1.20E+02   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      0.00E+00   1.04E+03   0.000      Eq. F9-4      3     0.00  |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major      1.16E+03   0.00E+00    1.00     0.00     0.00    96.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
```

```
|                       RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp          0.945      Eq. H2-1      3     0.00                |
| Flexure Tens          0.000      Eq. H2-1      3     0.00                |
| Intermediate         frbw /     frbz /     fra /                         |
|                      Fcbw       Fcbz       Fca                           |
| Flexure Comp         0.00E+00   0.00E+00   1.55E+01                      |
|                      5.18E+01   5.18E+01   1.64E+01                      |
| Flexure Tens         0.00E+00   0.00E+00   0.00E+00                      |
|                      2.57E+01   5.18E+01   3.24E+01                      |
|--------------------------------------------------------------------------|
    70. LOAD LIST 4
    71. PARAMETER 2
    72. CODE AISC UNIFIED 2005
    73. METHOD ASD
    74. FU 58 ALL
    75. FYLD 36 ALL
    76. CSPACING 32 ALL
    77. TRACK 2 ALL
    78. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1 LD   L50304                   (AISC SECTIONS)
                           PASS   Eq. H2-1            0.947         4
                       40.00 C          0.00          0.00       0.00
|--------------------------------------------------------------------------|
| SLENDERNESS                                                              |
| Actual Slenderness Ratio    :     78.939  L/C  :     4                   |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00                |
|--------------------------------------------------------------------------|
| STRENGTH CHECKS                                                          |
| Critical L/C  :      4    Ratio     :     0.947(PASS)                    |
|          Loc  :    0.00   Condition :   Eq. H2-1                         |
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   4.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   1.500E+00     Ayy:   1.250E+00     Cw:  0.000E+00                 |
| Szz:   3.055E+00     Syy:   2.075E+00                                    |
| Izz:   1.022E+01     Iyy:   7.003E+00     Ix:  8.083E-02                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:          36.000   Fu:          58.000                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        96.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   32.00          |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE          |
|                STIFFENED                                                 |
```

```
| Compression :  Slender        20.00     N/A     12.77    T.B4.1-5     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1-6     |
|                  N/A           N/A      N/A      N/A      N/A         |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield      0.00E+00   8.36E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture    0.00E+00   1.13E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck   4.00E+01   5.80E+01   0.689      Sec. E1       4     0.00 |
| Min Buck   4.00E+01   5.38E+01   0.743      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck   4.00E+01   4.22E+01   0.947      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck   3.88E+00    59.15   2.50E+01   8.18E+01   9.69E+01         |
| Min Buck   3.88E+00    72.59   2.32E+01   5.43E+01   8.99E+01         |
| Flexural      Ag        Fcr        Pn                                 |
| Tor Buck   3.88E+00   1.82E+01   7.05E+01                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z    0.00E+00   1.94E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y    0.00E+00   1.62E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results       Aw        Cv       Kv      h/tw      Vn                |
| Local-Z    1.50E+00     1.00     1.20    12.00   3.24E+01             |
| Local-Y    1.25E+00     1.00     1.20    20.00   2.70E+01             |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      0.00E+00   1.05E+02   0.000      Eq. F9-1      4     0.00 |
| Minor      0.00E+00   7.16E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn        My                                           |
| Major      1.76E+02   0.00E+00                                        |
| Minor      1.20E+02   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      0.00E+00   6.93E+02   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn        Me        Cb       Lp       Lr       Lb    |
| Major      1.16E+03   0.00E+00    1.00     0.00     0.00    96.00    |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
```

```
                      ********************************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.947      Eq. H2-1      4     0.00             |
| Flexure Tens         0.000      Eq. H2-1      4     0.00             |
| Intermediate        frbw /     frbz /     fra /                      |
|                     Fcbw       Fcbz       Fca                        |
| Flexure Comp        0.00E+00   0.00E+00   1.03E+01                   |
|                     3.45E+01   3.45E+01   1.09E+01                   |
| Flexure Tens        0.00E+00   0.00E+00   0.00E+00                   |
|                     1.71E+01   3.45E+01   2.16E+01                   |
|----------------------------------------------------------------------|
```

V. AISC 360-05 F.1-3 LTB I Section

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F1-3, pp.F-11 - F-14

Details

From reference (2):

> Verify the strength of the W18×50 beam selected in Example F.1-1a if the beam is braced at the ends and center point rather than continuously braced.

From Example F1-1a (also found in Reference (1)):

> Select an ASTM A992 W-shape beam with a simple span of 35 feet. Limit the member to a maximum nominal depth of 18 in. Limit the live load deflection to L/360. The nominal loads are a uniform dead load of 0.45 kip/ft and a uniform live load of 0.75 kip/ft.

Validation

Refer to reference.

Table 766: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·kips) [LRFD]	288	288	none	
M_n / Ω_b (ft·kips) [ASD]	192	192	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-OCT-12
JOB NAME AISC 360-10 Validation
JOB CLIENT Bentley
JOB NO 7
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 420 0 0;
MEMBER INCIDENCES
1 1 2;
***************************************************
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 59 RY 1.5 RT 1.2
END DEFINE MATERIAL
***************************************************
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
***************************************************
CONSTANTS
MATERIAL STEEL ALL
***************************************************
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
***************************************************
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.0375
**
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.0625
**
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
**
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
***************************************************
PERFORM ANALYSIS
UNIT FEET KIP
LOAD LIST 3
```

```
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 65 ALL
CB 1.3 ALL
LX 210 ALL
UNT 17.5 ALL
UNB 17.5 ALL
MAIN 1 ALL
TRACK 2 ALL
CHECK CODE ALL
***************************************************
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
CB 1.3 ALL
LX 210 ALL
UNT 17.5 ALL
UNB 210 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.924         3
                        0.00            0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.924(PASS)                |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03             |
| Szz:   8.889E+01     Syy:   1.069E+01                                |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
```

```
| Fyld:        7199.999   Fu:        8639.999                              |
|-------------------------------------------------------------------------|
| Actual Member Length:        35.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3       |
|                Slender        45.23     N/A     35.88    T.B4.1-10      |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1       |
|                Compact        45.23    90.55   137.27    T.B4.1-9       |
    STAAD SPACE                                              -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield      0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00   |
| Rupture    0.00E+00   6.61E+02   0.000      Eq. D2-2      3     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck   0.00E+00   5.09E+02   0.000      Sec. E1       3     0.00   |
| Min Buck   0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00   |
| Flexural                                                                |
| Tor Buck   0.00E+00   1.93E+02   0.000      Eq. E7-1      3     0.00   |
| Intermediate                                                            |
| Results    Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck   9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02          |
| Min Buck   1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01          |
| Flexural      Ag        Fcr        Pn                                   |
| Tor Buck   1.02E-01   2.10E+03   2.14E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z    0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00   |
| Local-Y    3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00   |
| Intermediate                                                            |
| Results       Aw        Cv       Kv      h/tw      Vn                  |
| Local-Z    5.94E-02     1.00     1.20     6.58   2.56E+02               |
| Local-Y    4.44E-02     1.00     0.00    45.23   1.92E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major      2.66E+02   3.79E+02   0.703      Eq. F2-1      3    17.50   |
| Minor      0.00E+00   6.23E+01   0.000      Eq. F6-1      3     0.00   |
| Intermediate   Mn         My                                            |
| Major      4.21E+02   0.00E+00                                         |
| Minor      6.92E+01   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
```

```
||                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       2.66E+02   2.88E+02   0.924      Eq. F2-3      3    17.50 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb       |
| Major       3.20E+02   0.00E+00    1.30     5.83    16.97    17.50    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           0.924      Eq. H1-1b     3    17.50              |
| Flexure Tens           0.924      Eq. H1-1b     3    17.50              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          2.88E+02   2.66E+02   5.14E+01                    |
|                       6.23E+01   0.00E+00   0.00E+00                    |
| Flexure Tens          2.88E+02   2.66E+02   6.61E+02                    |
|                       6.23E+01   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
    65. ***************************************************
    66. LOAD LIST 4
    67. PARAMETER 2
    68. CODE AISC UNIFIED 2005
    69. METHOD ASD
    70. FYLD 7200 ALL
    71. FU 9360 ALL
    72. CB 1.3 ALL
    73. LX 210 ALL
    74. UNT 17.5 ALL
    75. UNB 210 ALL
    76. TRACK 2 ALL
    77. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                          PASS   Eq. H1-1b           0.958         4
                        0.00            0.00       -183.75       17.50
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :    254.294  L/C  :     4                  |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      4    Ratio     :     0.958(PASS)                   |
|          Loc  :   17.50   Condition :   Eq. H1-1b                       |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                 |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02                  |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03                |
| Szz:   8.889E+01     Syy:   1.069E+01                                   |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        35.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                                |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3      |
|                Slender        45.23     N/A     35.88    T.B4.1-10     |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1      |
|                Compact        45.23    90.55   137.27    T.B4.1-9      |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield       0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00   |
| Rupture     0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck    0.00E+00   3.39E+02   0.000      Sec. E1       4     0.00   |
| Min Buck    0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00   |
| Flexural                                                                |
| Tor Buck    0.00E+00   1.28E+02   0.000      Eq. E7-1      4     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02          |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01          |
| Flexural       Ag        Fcr        Pn                                  |
| Tor Buck    1.02E-01   2.10E+03   2.14E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z     0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00   |
| Local-Y     2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00   |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                  |
| Local-Z     5.94E-02     1.00     1.20     6.58   2.56E+02               |
| Local-Y     4.44E-02     1.00     0.00    45.23   1.92E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
```

```
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.84E+02   2.52E+02   0.729      Eq. F2-1      4    17.50 |
| Minor       0.00E+00   4.14E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.21E+02   0.00E+00                                        |
| Minor       6.92E+01   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.84E+02   1.92E+02   0.958      Eq. F2-3      4    17.50 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       3.20E+02   0.00E+00    1.30     5.83    16.97    17.50    |
    STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.958      Eq. H1-1b     4    17.50           |
| Flexure Tens           0.958      Eq. H1-1b     4    17.50           |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          1.92E+02   1.84E+02   3.42E+01                 |
|                       4.14E+01   0.00E+00   0.00E+00                 |
| Flexure Tens          1.92E+02   1.84E+02   4.40E+02                 |
|                       4.14E+01   0.00E+00   0.00E+00                 |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 HSSP D.5

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.5, pp.D-10,11

Details

From reference (2):

> An HSS6.000×0.500, ASTM A500 grade B, has a length of 30 ft. The member carries a dead load of 40 kips and a live load of 120 kips in tension. Assume the end connection is a fillet welded 1/2 in. thick single concentric gusset plate that has a length of 16 in. Verify the strength by both LRFD and ASD.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	306	306	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	329	329	none	
Tensile yielding P_n / Ω_t (kips) [ASD]	203	203	none	
Tensile rupture P_n / Ω_t (kips) [ASD]	220	220	none	
Slenderness ratio	184	183.316	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSSP D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP6X0.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
```

```
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6048 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6048 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSSP6X0.5                (AISC SECTIONS)
                            PASS   Eq. H1-1a            0.785         3
                      240.00 T         0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    183.316  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.785(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.466E+00     Ayy:   4.466E+00     Cw:  0.000E+00             |
| Szz:   1.040E+01     Syy:   1.040E+01                                |
| Izz:   3.120E+01     Iyy:   3.120E+01     Ix:  6.240E+01             |
|-----------------------------------------------------------------------|
```

```
| MATERIAL PROPERTIES                                                    |
| Fyld:        6047.999   Fu:        8351.999                            |
|------------------------------------------------------------------------|
| Actual Member Length:        30.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    1.00 CSP:   12.00         |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Non-Slender    12.90     N/A     75.95    T.B4.1-15a    |
|                Non-Slender    12.90     N/A     75.95    T.B4.1-15a    |
| Flexure     :  Compact        12.90    48.33   214.05    T.B4.1-15b    |
|                Compact        12.90    48.33   214.05    T.B4.1-15b    |
    STAAD SPACE                                                -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       2.40E+02   3.06E+02   0.785      Eq. D2-1      3     0.00  |
| Rupture     2.40E+02   3.29E+02   0.729      Eq. D2-2      3     0.00  |
|------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                            |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   5.44E+01   0.000      Sec. E1       3     0.00  |
| Min Buck    0.00E+00   5.44E+01   0.000      Eq. E3-1      3     0.00  |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01         |
| Min Buck    5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01         |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   9.17E+01   0.000      Eq. G6-1      3     0.00  |
| Local-Y     0.00E+00   9.17E+01   0.000      Eq. G6-1      3     0.00  |
| Intermediate                                                           |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     5.62E-02     0.00     0.00     0.00   1.02E+02             |
| Local-Y     5.62E-02     0.00     0.00     0.00   1.02E+02             |
|------------------------------------------------------------------------|
| CHECK FOR TORSION                                                      |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   3.93E+01   0.000      Eq. H3-1      3     0.00  |
| Intermediate Fcr        Tn                                             |
|             3.63E+03   4.37E+01                                       |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   4.50E+01   0.000      Eq. F8-1      3     0.00  |
| Minor       0.00E+00   4.50E+01   0.000      Eq. F8-1      3     0.00  |
| Intermediate   Mn         My                                           |
```

```
| Major       5.01E+01   0.00E+00                                        |
| Minor       5.01E+01   0.00E+00                                        |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.000      Eq. H1-1b     3     0.00            |
| Flexure Tens           0.785      Eq. H1-1a     3     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.50E+01   0.00E+00   5.44E+01                  |
|                       4.50E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          4.50E+01   0.00E+00   3.06E+02                  |
|                       4.50E+01   0.00E+00   2.40E+02                  |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. FU 8352 ALL
    54. FYLD 6048 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.936 ALL
    58. SLF 1 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSSP6X0.5               (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.786         4
                      160.00 T          0.00          0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    183.316  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.786(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
```

```
|| Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00            |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00            |
 |--------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                   |
 | Azz:   4.466E+00     Ayy:   4.466E+00     Cw:  0.000E+00           |
 | Szz:   1.040E+01     Syy:   1.040E+01                              |
 | Izz:   3.120E+01     Iyy:   3.120E+01     Ix:  6.240E+01           |
 |--------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                |
 | Fyld:        6047.999   Fu:        8351.999                        |
 |--------------------------------------------------------------------|
 | Actual Member Length:        30.000                                |
 | Design Parameters                                                  |
 | Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    1.00 CSP:   12.00     |
 |--------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE    |
 |                STIFFENED                                           |
 | Compression :  Non-Slender    12.90     N/A     75.95    T.B4.1-15a |
 |                Non-Slender    12.90     N/A     75.95    T.B4.1-15a |
 | Flexure     :  Compact        12.90    48.33   214.05    T.B4.1-15b |
 |                Compact        12.90    48.33   214.05    T.B4.1-15b |
     STAAD SPACE                                           -- PAGE NO.
10
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |--------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                            |
 |                                                                    |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Yield      1.60E+02   2.03E+02   0.786      Eq. D2-1      4     0.00 |
 | Rupture    1.60E+02   2.20E+02   0.729      Eq. D2-2      4     0.00 |
 |--------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                        |
 |                                                                    |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Maj Buck   0.00E+00   3.62E+01   0.000      Sec. E1       4     0.00 |
 | Min Buck   0.00E+00   3.62E+01   0.000      Eq. E3-1      4     0.00 |
 | Intermediate                                                       |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn         |
 | Maj Buck   5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01      |
 | Min Buck   5.62E-02   183.32   1.08E+03   1.23E+03   6.04E+01      |
 |--------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                    |
 |                                                                    |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Local-Z    0.00E+00   6.10E+01   0.000      Eq. G6-1      4     0.00 |
 | Local-Y    0.00E+00   6.10E+01   0.000      Eq. G6-1      4     0.00 |
 | Intermediate                                                       |
 | Results       Aw         Cv        Kv      h/tw      Vn             |
 | Local-Z    5.62E-02     0.00      0.00      0.00   1.02E+02         |
 | Local-Y    5.62E-02     0.00      0.00      0.00   1.02E+02         |
 |--------------------------------------------------------------------|
 | CHECK FOR TORSION                                                  |
 |                                                                    |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 |            0.00E+00   2.62E+01   0.000      Eq. H3-1      4     0.00 |
 | Intermediate Fcr       Tn                                          |
```

```
|             3.63E+03   4.37E+01                                         |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.00E+01   0.000      Eq. F8-1      4     0.00 |
| Minor       0.00E+00   3.00E+01   0.000      Eq. F8-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       5.01E+01   0.00E+00                                       |
| Minor       5.01E+01   0.00E+00                                       |
     STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.000      Eq. H1-1b     4     0.00            |
| Flexure Tens           0.786      Eq. H1-1a     4     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.00E+01   0.00E+00   3.62E+01                  |
|                       3.00E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          3.00E+01   0.00E+00   2.03E+02                  |
|                       3.00E+01   0.00E+00   1.60E+02                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 HSSP Shear Capacity G.5

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.5, pp.G-9,10

Details

From reference (2):

> Verify the shear strength of a round HSS16.000×0.375 ASTM A500 grade B member spanning 32 feet with end shears of 30 kips from dead load and 90 kips from live load.

Validation

Ultimate load:

$$V_u = 1.2(30.0 \text{ kips}) + 1.6(90.0 \text{ kips}) = 180 \text{ kips}$$

Service load:

$$V_a = 30.0 \text{ kips} + 90.0 \text{ kips} = 120 \text{ kips}$$

Shear Buckling Stress

$$F_{cr} = \max \left| \begin{array}{l} \dfrac{1.60E}{\sqrt{\dfrac{L_v}{D}}\left(\dfrac{D}{t}\right)^{5/4}} = \dfrac{1.60(29,\ 000)}{\sqrt{\dfrac{192}{16.0}}\left(\dfrac{16.0}{0.349}\right)^{5/4}} = 112 \text{ ksi} \\ \dfrac{0.78E}{\left(D/t\right)^{3/2}} = \dfrac{0.78(29,\ 000)}{\left(16.0/0.349\right)^{3/2}} = 72.9 \text{ ksi} \end{array} \right.$$

The maximum permitted value is $F_{cr} \le 0.6F_y = 0.6(42) = 25.2\ ksi$ (governs).

Nominal Shear Strength

$$V_n = F_{cr}A_g / 2 = (25.2)(17.2) / 2 = 217 \text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(217) = 195\ kips$

ASD: $V_n / \Omega_v = {}^{217}/_{1.67} = 130\ kips$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	195	195	none	
V_n / Ω_v (kips) [ASD]	130	130	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSSP Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 32 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP16X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
```

```
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
UNIT INCHES KIP
1 UNI GY -0.15625
UNIT FEET KIP
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
UNIT INCHES KIP
1 UNI GY -0.46875
UNIT FEET KIP
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 42 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 42 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSSP16X0.375             (AISC SECTIONS)
                           FAIL   Eq. H1-1b            5.347         3
                        0.00            0.00      -17280.00      192.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     69.439  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :   192.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     5.347(FAIL)                 |
|          Loc  :  192.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.728E+04                   |
|---------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                          |
| Azz:   8.728E+00     Ayy:   8.728E+00     Cw:  0.000E+00                  |
| Szz:   6.575E+01     Syy:   6.575E+01                                     |
| Izz:   5.260E+02     Iyy:   5.260E+02     Ix:  1.052E+03                  |
|---------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                       |
| Fyld:          42.000   Fu:          58.000                               |
|---------------------------------------------------------------------------|
| Actual Member Length:       384.000                                       |
| Design Parameters                                                         |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00           |
|---------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE           |
|                STIFFENED                                                  |
| Compression :  Non-Slender    45.85     N/A     75.95    T.B4.1-15a      |
|                Non-Slender    45.85     N/A     75.95    T.B4.1-15a      |
| Flexure     :  Compact        45.85    48.33   214.05    T.B4.1-15b      |
|                Compact        45.85    48.33   214.05    T.B4.1-15b      |
     STAAD SPACE                                              -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                   |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Yield       0.00E+00   6.50E+02   0.000      Eq. D2-1      3     0.00    |
| Rupture     0.00E+00   7.48E+02   0.000      Eq. D2-2      3     0.00    |
|---------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                               |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Maj Buck    0.00E+00   4.84E+02   0.000      Sec. E1       3     0.00    |
| Min Buck    0.00E+00   4.84E+02   0.000      Eq. E3-1      3     0.00    |
| Intermediate                                                              |
| Results     Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02          |
| Min Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02          |
|---------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                           |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
| Local-Z     0.00E+00   1.95E+02   0.000      Eq. G6-1      3     0.00    |
| Local-Y     1.80E+02   1.95E+02   0.923      Eq. G6-1      3     0.00    |
| Intermediate                                                              |
| Results        Aw         Cv        Kv      h/tw      Vn                 |
| Local-Z     1.72E+01     0.00      0.00     0.00   2.17E+02              |
| Local-Y     1.72E+01     0.00      0.00     0.00   2.17E+02              |
|---------------------------------------------------------------------------|
| CHECK FOR TORSION                                                         |
|                                                                           |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
|             0.00E+00   2.97E+03   0.000      Eq. H3-1      3     0.00    |
| Intermediate Fcr        Tn                                                |
|             2.52E+01   3.30E+03                                          |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.73E+04   3.23E+03   5.347      Eq. F8-1      3   192.00 |
| Minor       0.00E+00   3.23E+03   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       3.59E+03   0.00E+00                                       |
| Minor       3.59E+03   0.00E+00                                       |
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           5.347      Eq. H1-1b     3   192.00            |
| Flexure Tens           5.347      Eq. H1-1b     3   192.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.23E+03   1.73E+04   4.84E+02                  |
|                       3.23E+03   0.00E+00   0.00E+00                  |
| Flexure Tens          3.23E+03   1.73E+04   6.50E+02                  |
|                       3.23E+03   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2005
    55. METHOD ASD
    56. FYLD 42 ALL
    57. FU 58 ALL
    58. TRACK 2 ALL
    59. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSSP16X0.375             (AISC SECTIONS)
                           FAIL   Eq. H1-1b            5.357         4
                        0.00            0.00      -11520.00      192.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     69.439  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :   192.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
```

```
|  Critical L/C  :      4    Ratio     :     5.357(FAIL)                       |
|          Loc  :  192.00   Condition :   Eq. H1-1b                            |
|--------------------------------------------------------------------------|
|  DESIGN FORCES                                                             |
|  Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                    |
|  Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.152E+04                    |
|--------------------------------------------------------------------------|
|  SECTION PROPERTIES  (UNIT: INCH)                                          |
|  Azz:   8.728E+00     Ayy:   8.728E+00     Cw:  0.000E+00                   |
|  Szz:   6.575E+01     Syy:   6.575E+01                                      |
|  Izz:   5.260E+02     Iyy:   5.260E+02     Ix:  1.052E+03                   |
|--------------------------------------------------------------------------|
|  MATERIAL PROPERTIES                                                       |
|  Fyld:          42.000   Fu:          58.000                                |
|--------------------------------------------------------------------------|
|  Actual Member Length:       384.000                                       |
|  Design Parameters                                                         |
|  Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00            |
|--------------------------------------------------------------------------|
|  SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE           |
|                 STIFFENED                                                  |
|  Compression :  Non-Slender    45.85     N/A     75.95    T.B4.1-15a       |
|                 Non-Slender    45.85     N/A     75.95    T.B4.1-15a       |
|  Flexure     :  Compact        45.85    48.33   214.05    T.B4.1-15b       |
|                 Compact        45.85    48.33   214.05    T.B4.1-15b       |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
|  CHECK FOR AXIAL TENSION                                                   |
|                                                                            |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
|  Yield       0.00E+00   4.33E+02   0.000      Eq. D2-1      4     0.00     |
|  Rupture     0.00E+00   4.99E+02   0.000      Eq. D2-2      4     0.00     |
|--------------------------------------------------------------------------|
|  CHECK FOR AXIAL COMPRESSION                                               |
|                                                                            |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
|  Maj Buck    0.00E+00   3.22E+02   0.000      Sec. E1       4     0.00     |
|  Min Buck    0.00E+00   3.22E+02   0.000      Eq. E3-1      4     0.00     |
|  Intermediate                                                              |
|  Results     Eff Area     KL/r      Fcr        Fe         Pn               |
|  Maj Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02            |
|  Min Buck    1.72E+01    69.44   3.12E+01   5.94E+01   5.37E+02            |
|--------------------------------------------------------------------------|
|  CHECK FOR SHEAR                                                           |
|                                                                            |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
|  Local-Z     0.00E+00   1.30E+02   0.000      Eq. G6-1      4     0.00     |
|  Local-Y     1.20E+02   1.30E+02   0.925      Eq. G6-1      4     0.00     |
|  Intermediate                                                              |
|  Results        Aw         Cv       Kv      h/tw      Vn                   |
|  Local-Z     1.72E+01     0.00     0.00     0.00   2.17E+02                |
|  Local-Y     1.72E+01     0.00     0.00     0.00   2.17E+02                |
|--------------------------------------------------------------------------|
|  CHECK FOR TORSION                                                         |
```

```
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|            0.00E+00   1.98E+03   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                           |
|            2.52E+01   3.30E+03                                       |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      1.15E+04   2.15E+03   5.357      Eq. F8-1      4   192.00 |
| Minor      0.00E+00   2.15E+03   0.000      Eq. F8-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major      3.59E+03   0.00E+00                                       |
| Minor      3.59E+03   0.00E+00                                       |
    STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp         5.357      Eq. H1-1b     4   192.00             |
| Flexure Tens         5.357      Eq. H1-1b     4   192.00             |
| Intermediate        Mcx /      Mrx /      Pc /                       |
|                     Mcy        Mry        Pr                         |
| Flexure Comp        2.15E+03   1.15E+04   3.22E+02                   |
|                     2.15E+03   0.00E+00   0.00E+00                   |
| Flexure Tens        2.15E+03   1.15E+04   4.33E+02                   |
|                     2.15E+03   0.00E+00   0.00E+00                   |
|----------------------------------------------------------------------|
```

V. AISC 360-05 HSSP Torsional Strength H.5B

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example H.5b, p.H-13

Details

From reference (2):

Determine the available torsional strength of an ASTM A500 Gr. B HSS5.000×0.250 that is 14 ft long.

Validation

Critical Stress

$$F_{cr} = \max \begin{cases} \dfrac{1.23E}{\sqrt{\dfrac{L}{D}}\left(\dfrac{D}{t}\right)^{5/4}} = \dfrac{1.23(29,\ 000)}{\sqrt{\dfrac{14 \times 12}{5.0}}(21.5)^{5/4}} = 133 \text{ ksi} \\ \dfrac{0.60E}{\left(D/t\right)^{3/2}} = \dfrac{0.60(29,\ 000)}{(21.5)^{3/2}} = 175 \text{ ksi} \end{cases}$$

The maximum permitted value is $F_{cr} \leq 0.6F_y = 0.6(42) = 25.2\ ksi$ (governs).

Nominal Torsional Strength

$$T_n = F_{cr}C = 25.2(7.95) = 200 \text{ in}\cdot\text{k} \quad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_T T_n = 0.90(200) = 180\ in \cdot k$

ASD: $T_n / \Omega_T = {}^{200}/_{1.67} = 120\ in \cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_G T_n$ (in·k) [LRFD]	180	180	none	
G_n/Ω_T (in·k) [ASD]	120	120	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSSP Torsional Strength H.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 14 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSSP5X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
```

```
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
UNIT INCHES KIP
MEMBER LOAD
1 UMOM GX -0.6
UNIT FEET KIP
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2010
METHOD LRFD
FU 58 ALL
FYLD 42 ALL
TND 3 ALL
TORSION 1 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISC UNIFIED 2010
METHOD ASD
FU 58 ALL
FYLD 42 ALL
TND 3 ALL
TORSION 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSSP5X0.25               (AISC SECTIONS)
                           PASS   Eq. H3-1             0.559         1
                        0.00            0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.547  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.559(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   1.008E+02     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.840E+00     Ayy:   1.840E+00     Cw:  0.000E+00             |
| Szz:   3.976E+00     Syy:   3.976E+00                                |
| Izz:   9.940E+00     Iyy:   9.940E+00     Ix:  1.988E+01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
```

```
| Fyld:          42.000   Fu:          58.000                             |
|-----------------------------------------------------------------------|
| Actual Member Length:       168.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
|                Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9  |
| Flexure     :  Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
|                Compact        21.46    48.33   214.05    T.B4.1(b)-20 |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   1.32E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.52E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   7.18E+01   0.000      Eq. E3-1      1     0.00 |
| Min Buck    0.00E+00   7.18E+01   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
| Min Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   3.96E+01   0.000      Eq. G6-1      1     0.00 |
| Local-Y     0.00E+00   3.96E+01   0.000      Eq. G6-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     3.49E+00     0.00     0.00     0.00   4.40E+01            |
| Local-Y     3.49E+00     0.00     0.00     0.00   4.40E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             1.01E+02   1.80E+02   0.559      Eq. H3-1      1     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.52E+01   2.00E+02                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.00E+02   0.000      Eq. F8-1      1     0.00 |
| Minor       0.00E+00   2.00E+02   0.000      Eq. F8-1      1     0.00 |
| Intermediate   Mn        My                                           |
| Major       2.23E+02   0.00E+00                                       |
```

```
| Minor       2.23E+02   0.00E+00                                      |
      STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.000      Eq. H1-1b     1     0.00           |
| Flexure Tens           0.000      Eq. H1-1b     1     0.00           |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          2.00E+02   0.00E+00   7.18E+01                 |
|                       2.00E+02   0.00E+00   0.00E+00                 |
| Flexure Tens          2.00E+02   0.00E+00   1.32E+02                 |
|                       2.00E+02   0.00E+00   0.00E+00                 |
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flx Tor Comp Shr       0.313      Eq. H3-6      1     0.00           |
| Flx Tor Tens Shr       0.313      Eq. H3-6      1     0.00           |
| Intermediate          Mcx /      Mcy /      Mrx /      Mry /         |
|                       Vcx /      Vcy /      Vrx /      Vry /         |
|                       Tc         Tr         Pc         Pr            |
| Flx Tor Comp Shr      2.00E+02   2.00E+02   0.00E+00   0.00E+00      |
|                       3.96E+01   3.96E+01   0.00E+00   0.00E+00      |
|                       1.80E+02   1.01E+02   7.18E+01   0.00E+00      |
| Flx Tor Tens Shr      2.00E+02   2.00E+02   0.00E+00   0.00E+00      |
|                       3.96E+01   3.96E+01   0.00E+00   0.00E+00      |
|                       1.80E+02   1.01E+02   1.32E+02   0.00E+00      |
|----------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                              |
|-----------------------------------------------------------------------------
----|
| Member:     1 Section:       HSSP5X0.25
AISC                        |
| End Condition:  Fix-
Free                                                         |
| Shear Capacity :  2.2680E+01  Shear Criteria : DG9 Eq.
4.13                       |
| Normal Capacity:  3.7800E+01  Normal Criteria: DG9 Eq.
4.12                       |
|-----------------------------------------------------------------------------
----|
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Uniform:      1.14946E+01 0.5068     1 | Uniform:      0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
|-----------------------------------------------------------------------------
----|
```

```
      STAAD SPACE                                              -- PAGE NO.
6
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                |
| Uniform:      1.14946E
+01                                                        |
|-------------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                             |
| Uniform:      0.00000E
+00                                                        |
|-------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                             |
| Uniform:      0.00000E
+00                                                        |
|-------------------------------------------------------------------------------
----|
    43. PARAMETER 2
    44. CODE AISC UNIFIED 2010
    45. METHOD ASD
    46. FU 58 ALL
    47. FYLD 42 ALL
    48. TND 3 ALL
    49. TORSION 1 ALL
    50. TRACK 2 ALL
    51. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSSP5X0.25              (AISC SECTIONS)
                          PASS   Eq. H3-1            0.840         1
                        0.00            0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     99.547  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.840(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
```

```
||  Mx:   1.008E+02     My:   0.000E+00     Mz:   0.000E+00                   |
||----------------------------------------------------------------------------|
||  SECTION PROPERTIES  (UNIT: INCH)                                          |
||  Azz:   1.840E+00     Ayy:   1.840E+00     Cw:  0.000E+00                  |
||  Szz:   3.976E+00     Syy:   3.976E+00                                     |
||  Izz:   9.940E+00     Iyy:   9.940E+00     Ix:  1.988E+01                  |
||----------------------------------------------------------------------------|
||  MATERIAL PROPERTIES                                                       |
||  Fyld:          42.000   Fu:          58.000                               |
||----------------------------------------------------------------------------|
||  Actual Member Length:       168.000                                       |
||  Design Parameters                                                         |
||  Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00            |
||----------------------------------------------------------------------------|
||  SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE           |
||                 STIFFENED                                                  |
||  Compression :  Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9      |
||                 Non-Slender    21.46     N/A     75.95    T.B4.1(a)-9      |
||  Flexure     :  Compact        21.46    48.33   214.05    T.B4.1(b)-20     |
||                 Compact        21.46    48.33   214.05    T.B4.1(b)-20     |
      STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
||----------------------------------------------------------------------------|
||  CHECK FOR AXIAL TENSION                                                   |
||                                                                            |
||               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
||  Yield       0.00E+00   8.78E+01   0.000      Eq. D2-1      1     0.00     |
||  Rupture     0.00E+00   1.01E+02   0.000      Eq. D2-2      1     0.00     |
||----------------------------------------------------------------------------|
||  CHECK FOR AXIAL COMPRESSION                                               |
||                                                                            |
||               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
||  Maj Buck    0.00E+00   4.78E+01   0.000      Eq. E3-1      1     0.00     |
||  Min Buck    0.00E+00   4.78E+01   0.000      Eq. E3-1      1     0.00     |
||  Intermediate                                                              |
||  Results     Eff Area     KL/r      Fcr         Fe         Pn              |
||  Maj Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01            |
||  Min Buck    3.49E+00    99.55   2.29E+01   2.89E+01   7.98E+01            |
||----------------------------------------------------------------------------|
||  CHECK FOR SHEAR                                                           |
||                                                                            |
||               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
||  Local-Z     0.00E+00   2.63E+01   0.000      Eq. G6-1      1     0.00     |
||  Local-Y     0.00E+00   2.63E+01   0.000      Eq. G6-1      1     0.00     |
||  Intermediate                                                              |
||  Results        Aw        Cv       Kv      h/tw      Vn                    |
||  Local-Z     3.49E+00     0.00     0.00     0.00   4.40E+01                |
||  Local-Y     3.49E+00     0.00     0.00     0.00   4.40E+01                |
||----------------------------------------------------------------------------|
||  CHECK FOR TORSION                                                         |
||                                                                            |
||               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC      |
||              1.01E+02   1.20E+02   0.840      Eq. H3-1      1     0.00     |
||  Intermediate Fcr        Tn                                                |
||              2.52E+01   2.00E+02                                           |
```

```
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major      0.00E+00   1.33E+02   0.000      Eq. F8-1      1     0.00   |
| Minor      0.00E+00   1.33E+02   0.000      Eq. F8-1      1     0.00   |
| Intermediate   Mn        My                                             |
| Major      2.23E+02   0.00E+00                                          |
| Minor      2.23E+02   0.00E+00                                          |
    STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                      RATIO      CRITERIA    L/C     LOC                 |
| Flexure Comp         0.000      Eq. H1-1b     1     0.00                |
| Flexure Tens         0.000      Eq. H1-1b     1     0.00                |
| Intermediate        Mcx /      Mrx /      Pc /                          |
|                     Mcy        Mry        Pr                            |
| Flexure Comp        1.33E+02   0.00E+00   4.78E+01                      |
|                     1.33E+02   0.00E+00   0.00E+00                      |
| Flexure Tens        1.33E+02   0.00E+00   8.78E+01                      |
|                     1.33E+02   0.00E+00   0.00E+00                      |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                   |
|                      RATIO      CRITERIA    L/C     LOC                 |
| Flx Tor Comp Shr     0.706      Eq. H3-6      1     0.00                |
| Flx Tor Tens Shr     0.706      Eq. H3-6      1     0.00                |
| Intermediate        Mcx /      Mcy /      Mrx /      Mry /              |
|                     Vcx /      Vcy /      Vrx /      Vry /              |
|                     Tc         Tr         Pc         Pr                 |
| Flx Tor Comp Shr    1.33E+02   1.33E+02   0.00E+00   0.00E+00           |
|                     2.63E+01   2.63E+01   0.00E+00   0.00E+00           |
|                     1.20E+02   1.01E+02   4.78E+01   0.00E+00           |
| Flx Tor Tens Shr    1.33E+02   1.33E+02   0.00E+00   0.00E+00           |
|                     2.63E+01   2.63E+01   0.00E+00   0.00E+00           |
|                     1.20E+02   1.01E+02   8.78E+01   0.00E+00           |
|-------------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                                   |
|-----------------------------------------------------------------------------
----|
| Member:     1 Section:       HSSP5X0.25
AISC                    |
| End Condition:  Fix-
Free                                                  |
| Shear Capacity :  1.5090E+01  Shear Criteria : DG9 Eq.
4.18                    |
| Normal Capacity:  2.5150E+01  Normal Criteria: DG9 Eq.
4.17                    |
|-----------------------------------------------------------------------------
----|
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
```

```
|                                   Sect |
Sect |
| Uniform:      1.14946E+01 0.7617     1 | Uniform:      0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
|------------------------------------------------------------------------------
----|
      STAAD SPACE                                                -- PAGE NO.
10
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                |
| Uniform:      1.14946E
+01                                                       |
|------------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                             |
| Uniform:      0.00000E
+00                                                       |
|------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                             |
| Uniform:      0.00000E
+00                                                       |
|------------------------------------------------------------------------------
----|
```

V. AISC 360-05 HSST Compact Flange F.6

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.6, p.F-27

Details

From reference (2):

> Select a square ASTM A500 Gr. B HSS beam to span 7.5 feet. The nominal loads are a uniform dead load of 0.145 kip/ft and a uniform live load of 0.435 kip/ft. Limit the live load deflection to L/240 . Assume the beam is continuously braced.

Validation

Ultimate moment:

$$w_u = 1.2(0.145 \text{ k/ft}) + 1.6(0.435 \text{ k/ft}) = 0.87 \text{ k/ft}$$

$$M_u = \frac{0.87(7.5)^2}{8} = 6.12 \text{ ft} \cdot \text{k}$$

Service moment:

$$w_a = 0.145 \text{ k/ft} + 0.435 \text{ k/ft} = 0.580 \text{ k/ft}$$

$$M_a = \frac{0.58(7.5)^2}{8} = 4.08 \text{ ft}\cdot\text{k}$$

Calculate the minimum required moment of inertia:

$$\Delta_{max} = \frac{L}{240} = \frac{7.50(12)}{240} = 0.375 \text{ in.}$$

$$I_{req} = \frac{5wl^4}{384E\Delta_{max}} = \frac{5(0.435)(7.5)^4(12)^3}{384(29{,}000)(0.375)} = 2.85 \text{ in.}^4$$ (Ref. 1, Table 3-23, Diagram 1)

Try a HSS 3×3×1/4, which has a plastic section modulus, $I_x = 3.02\ in^4$, $Z_x = 2.48\ in^3$.

The yield strength, $F_y = 46\ ksi$

Flange compactness: $\lambda = b/t = 9.88$

Determine the compact flange ratio limit from Table B4.1b: $\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{46}} = 28.1 > \lambda$

(Compact)

$$M_n = M_p = F_y \times Z_x = 46 \times 2.48 = 114.1 \text{ in-k} = 9.51 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(9.51) = 8.56\ ft\cdot k$

ASD: $M_n / \Omega_b = {}^{9.51}\!/_{1.67} = 5.69\ ft\cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	8.56	8.56	none	the reference gives a nominally different answer due to rounding
M_n/Ω_b (ft·k) [ASD]	5.69	5.69	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST Compact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST3X3X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.145
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.435
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST3X3X0.25             (AISC SECTIONS)
```

```
                         PASS   Eq. H1-1b            0.715         3
                      0.00            0.00          -6.12        3.75
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     80.897  L/C  :     3                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     3.75             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.715(PASS)                 |
 |          Loc  :    3.75   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -6.117E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   1.072E+00     Ayy:   1.072E+00     Cw:  0.000E+00              |
 | Szz:   2.013E+00     Syy:   2.013E+00                                 |
 | Izz:   3.020E+00     Iyy:   3.020E+00     Ix:  5.080E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        6623.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:         7.500                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Non-Slender     9.88     N/A     35.15    T.B4.1-12    |
 | Flexure     :  Compact         9.88    28.12    35.15    T.B4.1-12    |
 |                Compact         9.88    60.76   143.12    T.B4.1-13    |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   1.01E+02   0.000      Eq. D2-1      3     0.00  |
 | Rupture    0.00E+00   1.06E+02   0.000      Eq. D2-2      3     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   6.50E+01   0.000      Sec. E1       3     0.00  |
 | Min Buck   0.00E+00   6.50E+01   0.000      Eq. E3-1      3     0.00  |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r      Fcr         Fe         Pn           |
 | Maj Buck   1.69E-02    80.90   4.27E+03   6.30E+03   7.23E+01         |
 | Min Buck   1.69E-02    80.90   4.27E+03   6.30E+03   7.23E+01         |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Local-Z     0.00E+00   2.66E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y    -3.26E+00   2.66E+01   0.122      Eq. G2-1      3     7.50 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     7.45E-03     1.00     5.00     9.88   2.96E+01           |
| Local-Y     7.45E-03     1.00     5.00     9.88   2.96E+01           |
|----------------------------------------------------------------------|
| CHECK FOR TORSION                                                    |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   7.29E+00   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                           |
|             3.97E+03   8.10E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       6.12E+00   8.56E+00   0.715      Eq. F7-1      3     3.75 |
| Minor       0.00E+00   8.56E+00   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn        My                                          |
| Major       9.51E+00   0.00E+00                                      |
| Minor       9.51E+00   0.00E+00                                      |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.715      Eq. H1-1b     3     3.75           |
| Flexure Tens           0.715      Eq. H1-1b     3     3.75           |
| Intermediate          Mcx /      Mrx /      Pc /                    |
|                       Mcy        Mry        Pr                      |
| Flexure Comp          8.56E+00   6.12E+00   6.50E+01                 |
|                       8.56E+00   0.00E+00   0.00E+00                 |
| Flexure Tens          8.56E+00   6.12E+00   1.01E+02                 |
|                       8.56E+00   0.00E+00   0.00E+00                 |
|----------------------------------------------------------------------|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2005
    51. FU 8352 ALL
    52. FYLD 6624 ALL
    53. FLX 2 ALL
    54. METHOD ASD
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                             -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
```

```
                      *********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST3X3X0.25            (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.716         4
                        0.00            0.00         -4.08        3.75
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     80.897  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     3.75             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.716(PASS)                 |
|          Loc  :    3.75   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.078E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.072E+00     Ayy:   1.072E+00     Cw:  0.000E+00             |
| Szz:   2.013E+00     Syy:   2.013E+00                                 |
| Izz:   3.020E+00     Iyy:   3.020E+00     Ix:  5.080E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         7.500                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender     9.88     N/A     35.15    T.B4.1-12    |
| Flexure     :  Compact         9.88    28.12    35.15    T.B4.1-12    |
|                Compact         9.88    60.76   143.12    T.B4.1-13    |
     STAAD SPACE                                              -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      *********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      0.00E+00   6.72E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture    0.00E+00   7.08E+01   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   0.00E+00   4.33E+01   0.000      Sec. E1       4     0.00 |
| Min Buck   0.00E+00   4.33E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
```

```
| Maj Buck    1.69E-02    80.90   4.27E+03   6.30E+03   7.23E+01          |
| Min Buck    1.69E-02    80.90   4.27E+03   6.30E+03   7.23E+01          |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.77E+01   0.000      Eq. G2-1      4     0.00  |
| Local-Y    -2.17E+00   1.77E+01   0.123      Eq. G2-1      4     7.50  |
| Intermediate                                                            |
| Results        Aw         Cv        Kv      h/tw      Vn               |
| Local-Z     7.45E-03     1.00      5.00     9.88   2.96E+01            |
| Local-Y     7.45E-03     1.00      5.00     9.88   2.96E+01            |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   4.85E+00   0.000      Eq. H3-1      4     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.97E+03   8.10E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       4.08E+00   5.69E+00   0.716      Eq. F7-1      4     3.75  |
| Minor       0.00E+00   5.69E+00   0.000      Eq. F7-1      4     0.00  |
| Intermediate   Mn         My                                           |
| Major       9.51E+00   0.00E+00                                        |
| Minor       9.51E+00   0.00E+00                                        |
     STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.716      Eq. H1-1b     4     3.75             |
| Flexure Tens           0.716      Eq. H1-1b     4     3.75             |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          5.69E+00   4.08E+00   4.33E+01                    |
|                       5.69E+00   0.00E+00   0.00E+00                    |
| Flexure Tens          5.69E+00   4.08E+00   6.72E+01                    |
|                       5.69E+00   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
```

V. AISC 360-05 HSST D.4

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.4, pp.D-8,9

Details

From reference (2):

> Verify, by LRFD and ASD, the strength of an HSS6×4×3/8, ASTM A500 grade B, with a length of 30 ft. The member is carrying a dead load of 35 kips and a live load of 105 kips in tension. Assume the end connection is fillet welded to a 1/2 in. thick single concentric gusset plate and has a length of 16 in.

Validation

Ultimate load:

$$P_u = 1.2(35\text{ kips}) + 1.6(105\text{ kips}) = 210\text{ kips}$$

Service load:

$$P_a = 35\text{ kips} + 105 = 140\text{ kips}$$

<u>Tensile Yielding</u>

LRFD: $\phi_t P_n = \phi_t A_g F_y = 0.9(6.18)(46) = 256\ kips$

ASD: $P_n \Big/ \Omega_c = \frac{A_g F_y}{\Omega_c} = \frac{(6.18)(46)}{1.67} = 170\ kips$

<u>Tensile Rupture</u>

Determine the shear lag factor for the effective area:

$$A_e = A_n U \qquad \text{(Eq. D3-1)}$$

where

A_n = the net area; in this case allows for a 1/16-in gap in fit-up between the HSS and the gusset plate,

$= A_g - 2(t_p + 1 / 16)t = 6.18 - 2(0.5 + 1 / 16)0.349 = 5.79\ in^2$

$\bar{x}$ = $\frac{B^2 + 2BH}{4(B + H)} = \frac{(4.0)^2 + 2(4.0)(6.0)}{4(4.0 + 6.0)} = 1.60\ in$

U = $1 - \frac{\bar{x}}{l} = 1 - \frac{1.60}{16.0} = 0.90$

$$A_e = 5.79(0.90) = 5.21\text{ in}^2$$

<u>Nominal Tensile Rupture Strength</u>

$$P_n = F_u A_e = 58(5.21) = 302\text{ kips} \qquad \text{(Eq D2-2)}$$

The available tensile strength:

LRFD: $\phi_t P_n = 0.75(302) = 227\ kips$

ASD: $P_n / \Omega_t = {}^{302}\!\big/_{2.00} = 151\ kips$

<u>Slenderness</u>

$$\frac{L}{r} = \frac{30 \times 12}{1.55} = 232$$

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	256	256	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	227	226	negligible	
Tensile yielding P_n/Ω_t (kips) [ASD]	170	170	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	151	151	none	
Slenderness ratio	232	231.848	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 35
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 105
LOAD COMB 3 LRFD
1 1.2 2 1.6
```

```
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 0.9 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.936 ALL
SLF 0.9 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.927         3
                      210.00 T         0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    231.848  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.927(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.100E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00             |
| Szz:   9.433E+00     Syy:   7.450E+00                                |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01             |
|-----------------------------------------------------------------------|
```

```
 | MATERIAL PROPERTIES                                                     |
 | Fyld:        6623.999   Fu:        8351.999                             |
 |-------------------------------------------------------------------------|
 | Actual Member Length:        30.000                                     |
 | Design Parameters                                                       |
 | Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    0.90 CSP:   12.00          |
 |-------------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
 |                STIFFENED                                                |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A          |
 |                Non-Slender    14.19     N/A     35.15    T.B4.1-12      |
 | Flexure     :  Compact         8.46    28.12    35.15    T.B4.1-12      |
 |                Compact        14.19    60.76   143.12    T.B4.1-13      |
     STAAD SPACE                                                -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-------------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                                 |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Yield      2.10E+02   2.56E+02   0.821      Eq. D2-1      3     0.00    |
 | Rupture    2.10E+02   2.26E+02   0.927      Eq. D2-2      3     0.00    |
 |-------------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                             |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Maj Buck   0.00E+00   4.93E+01   0.000      Sec. E1       3     0.00    |
 | Min Buck   0.00E+00   2.60E+01   0.000      Eq. E3-1      3     0.00    |
 | Intermediate                                                            |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn             |
 | Maj Buck   4.29E-02   168.23   1.28E+03   1.46E+03   5.48E+01           |
 | Min Buck   4.29E-02   231.85   6.72E+02   7.67E+02   2.89E+01           |
 |-------------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                         |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Local-Z    0.00E+00   5.12E+01   0.000      Eq. G2-1      3     0.00    |
 | Local-Y    0.00E+00   8.59E+01   0.000      Eq. G2-1      3     0.00    |
 | Intermediate                                                            |
 | Results        Aw        Cv       Kv      h/tw      Vn                  |
 | Local-Z    1.43E-02     1.00     5.00     8.46   5.69E+01               |
 | Local-Y    2.40E-02     1.00     5.00    14.19   9.54E+01               |
 |-------------------------------------------------------------------------|
 | CHECK FOR TORSION                                                       |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 |            0.00E+00   2.94E+01   0.000      Eq. H3-1      3     0.00    |
 | Intermediate Fcr        Tn                                              |
 |            3.97E+03   3.27E+01                                          |
 |-------------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                              |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Major      0.00E+00   4.11E+01   0.000      Eq. F7-1      3     0.00    |
 | Minor      0.00E+00   3.08E+01   0.000      Eq. F7-1      3     0.00    |
 | Intermediate   Mn        My                                             |
```

```
| Major       4.56E+01   0.00E+00                                        |
| Minor       3.43E+01   0.00E+00                                        |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H1-1b     3     0.00             |
| Flexure Tens          0.927      Eq. H1-1a     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         4.11E+01   0.00E+00   2.60E+01                   |
|                      3.08E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         4.11E+01   0.00E+00   2.26E+02                   |
|                      3.08E+01   0.00E+00   2.10E+02                   |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. FU 8352 ALL
    54. FYLD 6624 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.936 ALL
    58. SLF 0.9 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.927         4
                      140.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    231.848  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.927(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
```

```
| Fx:   1.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00                |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00                |
|----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                     |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00              |
| Szz:   9.433E+00     Syy:   7.450E+00                                 |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01              |
|----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                  |
| Fyld:        6623.999   Fu:        8351.999                          |
|----------------------------------------------------------------------|
| Actual Member Length:        30.000                                  |
| Design Parameters                                                    |
| Kz:    1.00 Ky:    1.00 NSF:    0.94 SLF:    0.90 CSP:   12.00       |
|----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                             |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A       |
|                Non-Slender    14.19     N/A     35.15    T.B4.1-12   |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1-12   |
|                Compact        14.19    60.76   143.12    T.B4.1-13   |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                              |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       1.40E+02   1.70E+02   0.822      Eq. D2-1      4     0.00 |
| Rupture     1.40E+02   1.51E+02   0.927      Eq. D2-2      4     0.00 |
|----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                          |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   3.28E+01   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   1.73E+01   0.000      Eq. E3-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.29E-02   168.23   1.28E+03   1.46E+03   5.48E+01       |
| Min Buck    4.29E-02   231.85   6.72E+02   7.67E+02   2.89E+01       |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   3.41E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   5.71E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn             |
| Local-Z     1.43E-02     1.00      5.00     8.46   5.69E+01          |
| Local-Y     2.40E-02     1.00      5.00    14.19   9.54E+01          |
|----------------------------------------------------------------------|
| CHECK FOR TORSION                                                    |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.96E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                           |
```

```
|             3.97E+03   3.27E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.73E+01   0.000      Eq. F7-1      4     0.00 |
| Minor       0.00E+00   2.05E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn        My                                          |
| Major       4.56E+01   0.00E+00                                       |
| Minor       3.43E+01   0.00E+00                                       |
     STAAD SPACE                                             -- PAGE NO.
11
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.000      Eq. H1-1b     4     0.00           |
| Flexure Tens            0.927      Eq. H1-1a     4     0.00           |
| Intermediate           Mcx /      Mrx /      Pc /                     |
|                        Mcy        Mry        Pr                       |
| Flexure Comp           2.73E+01   0.00E+00   1.73E+01                 |
|                        2.05E+01   0.00E+00   0.00E+00                 |
| Flexure Tens           2.73E+01   0.00E+00   1.51E+02                 |
|                        2.05E+01   0.00E+00   1.40E+02                 |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 HSST NonCompact Flange F.7

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.7, pp.F-28 - F-31

Details

From reference (2):

> Select a rectangular ASTM A500 Gr. B HSS beam with a span of 21 ft. The nominal loads include a uniform dead load of 0.15 kip/ft and a uniform live load of 0.40 kip/ft. Limit the live load deflection to L/240. Assume the beam is braced at the end points only.

Validation

Ultimate moment:

$$w_u = 1.2(0.15 \text{ k/ft}) + 1.6(0.4 \text{ k/ft}) = 0.82 \text{ k/ft}$$

$$M_u = \frac{0.82(21)^2}{8} = 45.2 \text{ ft-k}$$

Service moment:

$$w_a = 0.15 \text{ k/ft} + 0.4 \text{ k/ft} = 0.55 \text{ k/ft}$$

$$M_a = \frac{0.55(21)^2}{8} = 30.3 \text{ ft-k}$$

Try a HSS 10×6×3/16, which has a plastic section modulus, $Z_x = 18.0\ in^3$ and a section modulus, $S_x = 14.9\ in^3$.

Flange compactness: $\lambda = b/t = 31.5$

Determine the compact and slender flange ratio limits from Table B4.1b:

$$\lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{46}} = 28.1$$

$$\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29{,}000}{46}} = 35.2$$

Thus, the section has a non-compact flange.

Web compactness: $\lambda = h/t = 54.5$

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

Thus, the section has a compact web.

$$M_p = F_y \times Z_x = 46 \times 18.0 = 828 \text{ in-k} = 69.0 \text{ ft·k}$$

$$M_n = M_p - (M_p - F_y S)\left(3.57\frac{b}{t}\sqrt{\frac{F_y}{E}} - 4.0\right) \qquad \text{(F7-2)}$$

$$= 828 - (828 - 0.7 \times 46 \times 14.9)\left(3.57 \times 31.5\sqrt{\frac{46}{29{,}000}} - 4.0\right) = 760 \text{ in-k} = 63.3 \text{ ft·k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(63.3) = 57.0\ ft \cdot k$

ASD: $M_n / \Omega_b = {}^{63.3}\!/_{1.67} = 37.9\ ft \cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	57.0	57.0	none	
M_n/Ω_b (ft·k) [ASD]	37.9	37.9	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST NonCompact Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST10X6X0.188
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.15
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.4
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   HSST10X6X0.188           (AISC SECTIONS)
                         PASS   Eq. H1-1b            0.792         3
                      0.00            0.00          -45.20       10.50
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    100.002  L/C  :     3                |
 | Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.792(PASS)                 |
 |          Loc  :   10.50   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.520E+01               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   1.906E+00     Ayy:   3.298E+00     Cw:  0.000E+00              |
 | Szz:   1.492E+01     Syy:   1.137E+01                                 |
 | Izz:   7.460E+01     Iyy:   3.410E+01     Ix:  7.380E+01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        6623.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        21.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Slender        54.47     N/A     35.15    T.B4.1-12    |
 | Flexure     :  Non-Compact    31.48    28.12    35.15    T.B4.1-12    |
 |                Compact        54.47    60.76   143.12    T.B4.1-13    |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       0.00E+00   2.22E+02   0.000      Eq. D2-1      3     0.00 |
 | Rupture     0.00E+00   2.34E+02   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    0.00E+00   1.43E+02   0.000      Sec. E1       3     0.00 |
 | Min Buck    0.00E+00   1.06E+02   0.000      Eq. E7-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    3.11E-02    67.61   4.28E+03   9.02E+03   1.59E+02        |
 | Min Buck    3.11E-02   100.00   3.15E+03   4.12E+03   1.18E+02        |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
```

```
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   4.74E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y     8.61E+00   8.19E+01   0.105      Eq. G2-1      3     0.00 |
| Intermediate                                                        |
| Results        Aw        Cv       Kv      h/tw      Vn              |
| Local-Z     1.32E-02     1.00     5.00    31.48   5.26E+01           |
| Local-Y     2.29E-02     1.00     5.00    54.47   9.10E+01           |
|---------------------------------------------------------------------|
| CHECK FOR TORSION                                                   |
|                                                                     |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|           0.00E+00   4.12E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                          |
|           3.97E+03   4.58E+01                                        |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       4.52E+01   6.21E+01   0.728      Eq. F7-1      3    10.50 |
| Minor       0.00E+00   4.38E+01   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn        My                                         |
| Major       6.90E+01   0.00E+00                                      |
| Minor       4.87E+01   0.00E+00                                      |
      STAAD SPACE                                         -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                             |
|                                                                     |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       4.52E+01   5.70E+01   0.792      Eq. F7-2      3    10.50 |
| Minor       0.00E+00   3.05E+01   0.000      Eq. F7-3      3     0.00 |
| Intermediate   Mn        Fcr                                        |
| Major       6.34E+01   0.00E+00                                      |
| Minor       3.39E+01   0.00E+00                                      |
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                        RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp           0.792      Eq. H1-1b     3    10.50          |
| Flexure Tens           0.792      Eq. H1-1b     3    10.50          |
| Intermediate         Mcx /      Mrx /      Pc /                    |
|                      Mcy        Mry        Pr                      |
| Flexure Comp         5.70E+01   4.52E+01   1.06E+02                 |
|                      3.05E+01   0.00E+00   0.00E+00                 |
| Flexure Tens         5.70E+01   4.52E+01   2.22E+02                 |
|                      3.05E+01   0.00E+00   0.00E+00                 |
|---------------------------------------------------------------------|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2005
    50. FU 8352 ALL
    51. FYLD 6624 ALL
    52. METHOD ASD
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 STEEL DESIGN
```

```
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST10X6X0.188           (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.799         4
                        0.00            0.00         -30.32       10.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    100.002  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.799(PASS)                 |
|          Loc  :   10.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -3.032E+01               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.906E+00     Ayy:   3.298E+00     Cw:  0.000E+00              |
| Szz:   1.492E+01     Syy:   1.137E+01                                 |
| Izz:   7.460E+01     Iyy:   3.410E+01     Ix:  7.380E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        21.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Slender        54.47     N/A     35.15    T.B4.1-12    |
| Flexure     :  Non-Compact    31.48    28.12    35.15    T.B4.1-12    |
|                Compact        54.47    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.48E+02   0.000      Eq. D2-1      4     0.00 |
```

```
| Rupture     0.00E+00   1.56E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   9.55E+01   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   7.04E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    3.11E-02    67.61   4.28E+03   9.02E+03   1.59E+02       |
| Min Buck    3.11E-02   100.00   3.15E+03   4.12E+03   1.18E+02       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   3.15E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     5.78E+00   5.45E+01   0.106      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.32E-02     1.00     5.00    31.48   5.26E+01            |
| Local-Y     2.29E-02     1.00     5.00    54.47   9.10E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.74E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   4.58E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.03E+01   4.13E+01   0.734      Eq. F7-1      4    10.50 |
| Minor       0.00E+00   2.92E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       6.90E+01   0.00E+00                                       |
| Minor       4.87E+01   0.00E+00                                       |
     STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.03E+01   3.79E+01   0.799      Eq. F7-2      4    10.50 |
| Minor       0.00E+00   2.03E+01   0.000      Eq. F7-3      4     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       6.34E+01   0.00E+00                                       |
| Minor       3.39E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.799      Eq. H1-1b     4    10.50             |
| Flexure Tens          0.799      Eq. H1-1b     4    10.50             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
```

```
| Flexure Comp         3.79E+01   3.03E+01   7.04E+01                   |
|                      2.03E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         3.79E+01   3.03E+01   1.48E+02                   |
|                      2.03E+01   0.00E+00   0.00E+00                   |
|---------------------------------------------------------------------|
```

V. AISC 360-05 HSST Shear Capacity G.4

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.4, p.G-4

Details

From reference (2):

> Verify the shear strength of a HSS6×4×3/8 ASTM A500 grade B member with end shears of 11 kips from dead load and 33 kips from live load. The beam is oriented with the shear parallel to the 6 in. dimension.

Validation

Ultimate load:

$$V_u = 1.2(11.0 \text{ kips}) + 1.6(33.0 \text{ kips}) = 66.0 \text{ kips}$$

Service load:

$$V_a = 11.0 \text{ kips} + 33.0 \text{ kips} = 44.0 \text{ kips}$$

Area of the web for shear:

Note: The corner radius is taken as 1.5× the design wall thickness per Section B4.1b of Reference 1.

$$\text{h} = \text{H} - 3\text{t} = 6.0 - 3\times 0.349 = 4.95 \text{ in}$$

Section G5 of Reference 1 gives $k_v = 5$.

$$\text{A}_\text{w} = 2\text{ht} = 2(4.95)(0.349) = 3.46 \text{ in}^2$$

$$\frac{h}{t_w} = \frac{4.95}{0.349} = 14.2 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{5(29,\ 000)}{46}} = 61.8$$

Therefore, $C_v = 1.0$.

<u>Nominal Shear Strength</u>

$$V_n = 0.6F_y A_w C_v = 0.6(46)(3.46)(1.0) = 95.5 \text{ kips} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(95.5) = 86.0 \text{ kips}$

ASD: $V_n / \Omega_v = {}^{95.5}\!/_{1.67} = 57.2 \text{ kips}$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	86.0	85.9	negligible	
V_n / Ω_v (kips) [ASD]	57.2	57.1	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -22
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -66
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FYLD 6624 ALL
FU 8352 ALL
METHOD LRFD
TRACK 2 ALL
```

```
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FYLD 6624 ALL
FU 8352 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSST6X4X0.375            (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.215         3
                        0.00            0.00        -132.00        2.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     30.913  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     2.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     3.215(FAIL)                 |
|          Loc  :    2.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   6.600E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.320E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00              |
| Szz:   9.433E+00     Syy:   7.450E+00                                 |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        6623.999   Fu:        8351.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         4.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    14.19     N/A     35.15    T.B4.1-12    |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1-12    |
|                Compact        14.19    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                             -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield      0.00E+00   2.56E+02   0.000      Eq. D2-1      3     0.00  |
| Rupture    0.00E+00   2.69E+02   0.000      Eq. D2-2      3     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck   0.00E+00   2.47E+02   0.000      Sec. E1       3     0.00  |
| Min Buck   0.00E+00   2.40E+02   0.000      Eq. E3-1      3     0.00  |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck   4.29E-02    22.43   6.40E+03   8.19E+04   2.75E+02        |
| Min Buck   4.29E-02    30.91   6.21E+03   4.31E+04   2.67E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    0.00E+00   5.12E+01   0.000      Eq. G2-1      3     0.00  |
| Local-Y    6.60E+01   8.59E+01   0.769      Eq. G2-1      3     0.00  |
| Intermediate                                                          |
| Results       Aw         Cv        Kv      h/tw      Vn              |
| Local-Z    1.43E-02     1.00      5.00     8.46   5.69E+01           |
| Local-Y    2.40E-02     1.00      5.00    14.19   9.54E+01           |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|            0.00E+00   2.94E+01   0.000      Eq. H3-1      3     0.00  |
| Intermediate Fcr        Tn                                            |
|            3.97E+03   3.27E+01                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      1.32E+02   4.11E+01   3.215      Eq. F7-1      3     2.00  |
| Minor      0.00E+00   3.08E+01   0.000      Eq. F7-1      3     0.00  |
| Intermediate   Mn        My                                           |
| Major      4.56E+01   0.00E+00                                        |
| Minor      3.43E+01   0.00E+00                                        |
     STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           3.215      Eq. H1-1b     3     2.00            |
| Flexure Tens           3.215      Eq. H1-1b     3     2.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.11E+01   1.32E+02   2.40E+02                  |
|                       3.08E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          4.11E+01   1.32E+02   2.56E+02                  |
```

```
|                      3.08E+01   0.00E+00   0.00E+00                      |
|-------------------------------------------------------------------------|
    47. LOAD LIST 4
    48. PARAMETER 2
    49. CODE AISC UNIFIED 2005
    50. FYLD 6624 ALL
    51. FU 8352 ALL
    52. METHOD ASD
    53. TRACK 2 ALL
    54. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                          -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
 =======================================================================
 *     1 ST   HSST6X4X0.375            (AISC SECTIONS)
                          FAIL   Eq. H1-1b            3.222         4
                        0.00            0.00         -88.00        2.00
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :     30.913  L/C  :     4                  |
| Allowable Slenderness Ratio :    300.000  LOC  :     2.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      4    Ratio     :     3.222(FAIL)                   |
|          Loc  :    2.00   Condition :   Eq. H1-1b                       |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   4.400E+01     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -8.800E+01                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   2.061E+00     Ayy:   3.457E+00     Cw:  0.000E+00                |
| Szz:   9.433E+00     Syy:   7.450E+00                                   |
| Izz:   2.830E+01     Iyy:   1.490E+01     Ix:  3.280E+01                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        6623.999   Fu:        8351.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:         4.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A          |
|                Non-Slender    14.19     N/A     35.15    T.B4.1-12      |
| Flexure     :  Compact         8.46    28.12    35.15    T.B4.1-12      |
```

```
|                Compact        14.19    60.76   143.12    T.B4.1-13     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      0.00E+00   1.70E+02   0.000      Eq. D2-1      4     0.00  |
| Rupture    0.00E+00   1.79E+02   0.000      Eq. D2-2      4     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck   0.00E+00   1.65E+02   0.000      Sec. E1       4     0.00  |
| Min Buck   0.00E+00   1.60E+02   0.000      Eq. E3-1      4     0.00  |
| Intermediate                                                          |
| Results    Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck   4.29E-02    22.43   6.40E+03   8.19E+04   2.75E+02         |
| Min Buck   4.29E-02    30.91   6.21E+03   4.31E+04   2.67E+02         |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    0.00E+00   3.41E+01   0.000      Eq. G2-1      4     0.00  |
| Local-Y    4.40E+01   5.71E+01   0.770      Eq. G2-1      4     0.00  |
| Intermediate                                                          |
| Results       Aw         Cv       Kv      h/tw      Vn                |
| Local-Z    1.43E-02     1.00     5.00     8.46   5.69E+01             |
| Local-Y    2.40E-02     1.00     5.00    14.19   9.54E+01             |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|            0.00E+00   1.96E+01   0.000      Eq. H3-1      4     0.00  |
| Intermediate Fcr        Tn                                            |
|            3.97E+03   3.27E+01                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      8.80E+01   2.73E+01   3.222      Eq. F7-1      4     2.00  |
| Minor      0.00E+00   2.05E+01   0.000      Eq. F7-1      4     0.00  |
| Intermediate   Mn         My                                          |
| Major      4.56E+01   0.00E+00                                        |
| Minor      3.43E+01   0.00E+00                                        |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           3.222      Eq. H1-1b     4     2.00            |
| Flexure Tens           3.222      Eq. H1-1b     4     2.00            |
```

```
| Intermediate          Mcx /      Mrx /      Pc /                   |
|                       Mcy        Mry        Pr                     |
| Flexure Comp          2.73E+01   8.80E+01   1.60E+02               |
|                       2.05E+01   0.00E+00   0.00E+00               |
| Flexure Tens          2.73E+01   8.80E+01   1.70E+02               |
|                       2.05E+01   0.00E+00   0.00E+00               |
|--------------------------------------------------------------------|
```

V. AISC 360-05 HSST Slender Flange F.8

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.8, pp.F-32 - F-34

Details

From reference (2):

> Verify the strength of an ASTM A500 Gr. B HSS8×8×x with a span of 21 ft. The nominal loads are a dead load of 0.125 kip/ft and a live load of 0.375 kip/ft. Limit the live load deflection to L 240 .

Validation

Ultimate moment:

$$w_u = 1.2(0.125 \text{ k/ft}) + 1.6(0.375 \text{ k/ft}) = 0.75 \text{ k/ft}$$

$$M_u = \frac{0.75(21)^2}{8} = 41.3 \text{ ft} \cdot \text{k}$$

Service moment:

$$w_a = 0.125 \text{ k/ft} + 0.375 \text{ k/ft} = 0.50 \text{ k/ft}$$

$$M_a = \frac{0.5(21)^2}{8} = 27.6 \text{ ft} \cdot \text{k}$$

Try a HSS 8×8×3/16, which has a plastic section modulus, $Z_x = 15.7\ in^3$ *and a section modulus,* $S_x = 13.6\ in^3$.

The yield strength, $F_y = 46\ ksi$

Flange compactness: $\lambda = b/t = 43.0$

Determine the slender flange ratio limit from Table B4.1b: $\lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{46}} = 35.2 < \lambda$ (Slender)

Web compactness: $\lambda = h/t = 43.0$

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

Thus, the section has a compact web.

$$M_n = F_y \times S_e \tag{F7-3}$$

where

S_e = the effective section modulus determined with the effective compression flange taken as b_e

b_e = $1.93 t_f \sqrt{\frac{E}{F_y}}\left[1 - \frac{0.38}{b / t_f}\sqrt{\frac{E}{F_y}}\right] = 1.93 \times 0.174\sqrt{\frac{29{,}000}{46}}\left[1 - \frac{0.38}{0.43}\sqrt{\frac{29{,}000}{46}}\right] = 6.53\ in$

The width of the compression flange between the corners (assumed to be $1.5t$) is:

$$b = 8 - 2 \times 1.5 \times t = 8 - 3(0.174) = 7.48 \text{ in} > 6.53 \text{ in}$$

Thus, the ineffective width of the compression flange is: $b - b_e = 7.48 - 6.53 = 0.95\ in$

Subtract this contribution of this ineffective flange area from the gross moment of inertia to approximate the effective moment of inertia:

$$I_{\text{eff}} \approx 54.4 - 2\left[\frac{0.95(0.174)^3}{12} + 0.95(0.174)(3.91)^3\right] = 49.3 \text{ in}^4$$

$$S_e = \frac{I_{\text{eff}}}{d/2} = \frac{49.3}{8.0/2} = 12.3 \text{ in}^3$$

$$M_n = \ = 46 \times 12.3 = 566 \text{ in-k} = 47.2 \text{ ft-k}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(47.2) = 42.5\ ft \cdot k$

ASD: $M_n / \Omega_b = {}^{47.2}/_{1.67} = 28.3\ ft \cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	42.5	42.5	none	
M_n/Ω_b (ft·k) [ASD]	28.3	28.3	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST Slender Flange F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 21 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
```

```
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST8X8X0.188
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.125
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.375
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 8352 ALL
FYLD 6624 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST8X8X0.188            (AISC SECTIONS)
                          PASS   Eq. H1-1b            0.972         3
                       0.00            0.00         -41.34       10.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     79.175  L/C  :     3                |
```

```
|| Allowable Slenderness Ratio :    300.000  LOC  :    10.50               |
||-------------------------------------------------------------------------|
|| STRENGTH CHECKS                                                         |
|| Critical L/C  :      3    Ratio     :     0.972(PASS)                   |
||          Loc  :   10.50   Condition :   Eq. H1-1b                       |
||-------------------------------------------------------------------------|
|| DESIGN FORCES                                                           |
|| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                 |
|| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.134E+01                 |
||-------------------------------------------------------------------------|
|| SECTION PROPERTIES  (UNIT: INCH)                                        |
|| Azz:   2.602E+00     Ayy:   2.602E+00     Cw:  0.000E+00                |
|| Szz:   1.360E+01     Syy:   1.360E+01                                   |
|| Izz:   5.440E+01     Iyy:   5.440E+01     Ix:  8.450E+01                |
||-------------------------------------------------------------------------|
|| MATERIAL PROPERTIES                                                     |
|| Fyld:        6623.999   Fu:        8351.999                             |
||-------------------------------------------------------------------------|
|| Actual Member Length:        21.000                                     |
|| Design Parameters                                                       |
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
||-------------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
||                STIFFENED                                                |
|| Compression :  Non-Slender     0.00     N/A      0.00      N/A          |
||                Slender        42.98     N/A     35.15    T.B4.1-12      |
|| Flexure     :  Slender        42.98    28.12    35.15    T.B4.1-12      |
||                Compact        42.98    60.76   143.12    T.B4.1-13      |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-------------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                                 |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Yield      0.00E+00   2.22E+02   0.000      Eq. D2-1      3     0.00    |
 | Rupture    0.00E+00   2.34E+02   0.000      Eq. D2-2      3     0.00    |
 |-------------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                             |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Maj Buck   0.00E+00   1.35E+02   0.000      Sec. E1       3     0.00    |
 | Min Buck   0.00E+00   1.35E+02   0.000      Eq. E7-1      3     0.00    |
 | Intermediate                                                            |
 | Results    Eff Area     KL/r      Fcr        Fe          Pn             |
 | Maj Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02           |
 | Min Buck   3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02           |
 |-------------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                         |
 |                                                                         |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
 | Local-Z    0.00E+00   6.46E+01   0.000      Eq. G2-1      3     0.00    |
 | Local-Y    7.88E+00   6.46E+01   0.122      Eq. G2-1      3     0.00    |
 | Intermediate                                                            |
 | Results       Aw        Cv       Kv      h/tw      Vn                   |
|| Local-Z    1.81E-02     1.00     5.00    42.98   7.18E+01               |
```

```
| Local-Y     1.81E-02     1.00     5.00    42.98   7.18E+01             |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   4.41E+01   0.000      Eq. H3-1      3     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.97E+03   4.90E+01                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       4.13E+01   5.42E+01   0.763      Eq. F7-1      3    10.50  |
| Minor       0.00E+00   5.42E+01   0.000      Eq. F7-1      3     0.00  |
| Intermediate   Mn         My                                            |
| Major       6.02E+01   0.00E+00                                        |
| Minor       6.02E+01   0.00E+00                                        |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       4.13E+01   4.25E+01   0.972      Eq. F7-3      3    10.50  |
| Minor       0.00E+00   4.25E+01   0.000      Eq. F7-3      3     0.00  |
| Intermediate   Mn         Fcr                                           |
| Major       4.73E+01   0.00E+00                                        |
| Minor       4.73E+01   0.00E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.972      Eq. H1-1b     3    10.50             |
| Flexure Tens           0.972      Eq. H1-1b     3    10.50             |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.25E+01   4.13E+01   1.35E+02                    |
|                       4.25E+01   0.00E+00   0.00E+00                    |
| Flexure Tens          4.25E+01   4.13E+01   2.22E+02                    |
|                       4.25E+01   0.00E+00   0.00E+00                    |
|-------------------------------------------------------------------------|
    48. LOAD LIST 4
    49. PARAMETER 2
    50. CODE AISC UNIFIED 2005
    51. FU 8352 ALL
    52. FYLD 6624 ALL
    53. FLX 2 ALL
    54. METHOD ASD
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
```

```
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSST8X8X0.188           (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.974        4
                        0.00           0.00         -27.56       10.50
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     79.175  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :    10.50             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.974(PASS)                 |
 |          Loc  :   10.50   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.756E+01               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   2.602E+00     Ayy:   2.602E+00     Cw:  0.000E+00              |
 | Szz:   1.360E+01     Syy:   1.360E+01                                 |
 | Izz:   5.440E+01     Iyy:   5.440E+01     Ix:  8.450E+01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        6623.999   Fu:        8351.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        21.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Slender        42.98     N/A     35.15    T.B4.1-12    |
 | Flexure     :  Slender        42.98    28.12    35.15    T.B4.1-12    |
 |                Compact        42.98    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       0.00E+00   1.48E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   1.56E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Maj Buck    0.00E+00   8.96E+01   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   8.96E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02       |
| Min Buck    3.27E-02    79.18   4.01E+03   6.57E+03   1.50E+02       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   4.30E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     5.25E+00   4.30E+01   0.122      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv        Kv      h/tw      Vn             |
| Local-Z     1.81E-02     1.00      5.00    42.98   7.18E+01          |
| Local-Y     1.81E-02     1.00      5.00    42.98   7.18E+01          |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   2.93E+01   0.000      Eq. H3-1      4     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.97E+03   4.90E+01                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.76E+01   3.60E+01   0.765      Eq. F7-1      4    10.50 |
| Minor       0.00E+00   3.60E+01   0.000      Eq. F7-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       6.02E+01   0.00E+00                                      |
| Minor       6.02E+01   0.00E+00                                      |
     STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.76E+01   2.83E+01   0.974      Eq. F7-3      4    10.50 |
| Minor       0.00E+00   2.83E+01   0.000      Eq. F7-3      4     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       4.73E+01   0.00E+00                                      |
| Minor       4.73E+01   0.00E+00                                      |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.974      Eq. H1-1b     4    10.50            |
| Flexure Tens           0.974      Eq. H1-1b     4    10.50            |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          2.83E+01   2.76E+01   8.96E+01                  |
|                       2.83E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          2.83E+01   2.76E+01   1.48E+02                  |
```

```
|                      2.83E+01   0.00E+00   0.00E+00                      |
|-------------------------------------------------------------------------|
```

V. AISC 360-05 HSST Torsional Strength H.5A

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example H.5a, pp.H-12,13

Details

From reference (2):

> Determine the available torsional strength of an ASTM A500 Gr. B HSS6×4×1/4.

Validation

$h/t > b/t$, so h/t governs.

$$\frac{h}{t} = 22.8 < 2.45\sqrt{\frac{E}{F_y}} = 2.45\sqrt{\frac{29,000}{46}} = 61.5$$

$$F_{cr} = 0.6F_y = 0.6(46) = 27.6 \text{ ksi} \qquad \text{(Eq. H3-3)}$$

Nominal Torsional Strength

$$T_n = F_{cr}C = 27.6(10.1) = 279 \text{ in}\cdot\text{k} \qquad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_T T_n = 0.90(279) = 251\ in\cdot k$

ASD: $T_n / \Omega_T = {}^{279}\!/_{1.67} = 167\ in\cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_G T_n$ (in·k) [LRFD]	251	251	none	
G_n/Ω_T (in·k) [ASD]	167	167	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 HSST Torsional Strength H.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
```

```
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X4X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
UNIT INCHES KIP
MEMBER LOAD
1 UMOM GX -1.5
UNIT FEET KIP
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 46 ALL
FU 58 ALL
TORSION 1 ALL
TND 3 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 46 ALL
FU 58 ALL
TORSION 1 ALL
TND 3 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.25             (AISC SECTIONS)
```

```
                           PASS   Eq. H3-1            0.287        1
                        0.00           0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.875  L/C  :     1                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.287(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H3-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   7.200E+01     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.538E+00     Ayy:   2.470E+00     Cw:  0.000E+00              |
| Szz:   6.967E+00     Syy:   5.550E+00                                 |
| Izz:   2.090E+01     Iyy:   1.110E+01     Ix:  2.360E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    22.75     N/A     35.15    T.B4.1-12    |
| Flexure     :  Compact        14.17    28.12    35.15    T.B4.1-12    |
|                Compact        22.75    60.76   143.12    T.B4.1-13    |
     STAAD SPACE                                             -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   1.78E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.87E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.72E+02   0.000      Sec. E1       1     0.00 |
| Min Buck    0.00E+00   1.68E+02   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    4.30E+00    21.77   4.46E+01   6.04E+02   1.92E+02        |
| Min Buck    4.30E+00    29.88   4.33E+01   3.21E+02   1.86E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
```

```
| Local-Z     0.00E+00   3.82E+01   0.000      Eq. G2-1      1     0.00 |
| Local-Y     0.00E+00   6.14E+01   0.000      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     1.54E+00     1.00     5.00    14.17   4.25E+01            |
| Local-Y     2.47E+00     1.00     5.00    22.75   6.82E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             7.20E+01   2.51E+02   0.287      Eq. H3-1      1     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.76E+01   2.79E+02                                      |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   3.53E+02   0.000      Eq. F7-1      1     0.00 |
| Minor       0.00E+00   2.67E+02   0.000      Eq. F7-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.92E+02   0.00E+00                                       |
| Minor       2.97E+02   0.00E+00                                       |
     STAAD SPACE                                            -- PAGE NO.
5
                    STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp         0.000      Eq. H1-1b     1     0.00              |
| Flexure Tens         0.000      Eq. H1-1b     1     0.00              |
| Intermediate        Mcx /      Mrx /      Pc /                        |
|                     Mcy        Mry        Pr                          |
| Flexure Comp        3.53E+02   0.00E+00   1.68E+02                    |
|                     2.67E+02   0.00E+00   0.00E+00                    |
| Flexure Tens        3.53E+02   0.00E+00   1.78E+02                    |
|                     2.67E+02   0.00E+00   0.00E+00                    |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flx Tor Comp Shr     0.082      Eq. H3-6      1     0.00              |
| Flx Tor Tens Shr     0.082      Eq. H3-6      1     0.00              |
| Intermediate        Mcx /      Mcy /      Mrx /      Mry /            |
|                     Vcx /      Vcy /      Vrx /      Vry /            |
|                     Tc         Tr         Pc         Pr               |
| Flx Tor Comp Shr    3.53E+02   2.67E+02   0.00E+00   0.00E+00         |
|                     3.82E+01   6.14E+01   0.00E+00   0.00E+00         |
|                     2.51E+02   7.20E+01   1.68E+02   0.00E+00         |
| Flx Tor Tens Shr    3.53E+02   2.67E+02   0.00E+00   0.00E+00         |
|                     3.82E+01   6.14E+01   0.00E+00   0.00E+00         |
|                     2.51E+02   7.20E+01   1.78E+02   0.00E+00         |
|-----------------------------------------------------------------------|
|---------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                             |
|---------------------------------------------------------------------------
```

```
----|
| Member:     1 Section:       HSST6X4X0.25
AISC                         |
| End Condition:  Fix-
Free                                                             |
| Shear Capacity :  2.4840E+01  Shear Criteria : DG9 Eq.
4.13                        |
| Normal Capacity:  4.1400E+01  Normal Criteria: DG9 Eq.
4.12                        |
|-------------------------------------------------------------------------------
----|
      STAAD SPACE                                               -- PAGE NO.
6
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Mid Flange:   0.00000E+00 0.0000     1 | Flange Edge:  0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
| FL/WB Junc:   6.43777E+00 0.2592     1
|                                        |
|                                  0.000
|                                        |
| Mid-Web:      6.43777E+00 0.2592     1
|                                        |
|                                  0.000
|                                        |
|-------------------------------------------------------------------------------
----|
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                 |
| Flange Edge:  6.43777E+00     Web Edge:     6.43777E
+00                           |
|-------------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                               |
| Flange Edge:  0.00000E+00  FL/WB Junc:   0.00000E+00  Mid-Web:      0.00000E
+00 |
|-------------------------------------------------------------------------------
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                               |
| Flange Edge:  0.00000E
+00                                                         |
|-------------------------------------------------------------------------------
----|
    43. PARAMETER 2
    44. CODE AISC UNIFIED 2005
    45. METHOD ASD
    46. FYLD 46 ALL
    47. FU 58 ALL
    48. TORSION 1 ALL
    49. TND 3 ALL
    50. TRACK 2 ALL
    51. CHECK CODE ALL
```

```
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST6X4X0.25            (AISC SECTIONS)
                           PASS   Eq. H3-1             0.431         1
                        0.00            0.00           0.00       0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     29.875  L/C  :     1                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      1    Ratio     :     0.431(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H3-1                      |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   7.200E+01     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   1.538E+00     Ayy:   2.470E+00     Cw:  0.000E+00              |
 | Szz:   6.967E+00     Syy:   5.550E+00                                 |
 | Izz:   2.090E+01     Iyy:   1.110E+01     Ix:  2.360E+01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          46.000   Fu:          58.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        48.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Non-Slender    22.75     N/A     35.15    T.B4.1-12    |
 | Flexure     :  Compact        14.17    28.12    35.15    T.B4.1-12    |
 |                Compact        22.75    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Yield       0.00E+00   1.18E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   1.25E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   1.15E+02   0.000      Sec. E1       1     0.00 |
| Min Buck    0.00E+00   1.12E+02   0.000      Eq. E3-1      1     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr         Fe         Pn         |
| Maj Buck    4.30E+00    21.77   4.46E+01   6.04E+02   1.92E+02        |
| Min Buck    4.30E+00    29.88   4.33E+01   3.21E+02   1.86E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.54E+01   0.000      Eq. G2-1      1     0.00 |
| Local-Y     0.00E+00   4.08E+01   0.000      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.54E+00     1.00     5.00    14.17   4.25E+01            |
| Local-Y     2.47E+00     1.00     5.00    22.75   6.82E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             7.20E+01   1.67E+02   0.431      Eq. H3-1      1     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.76E+01   2.79E+02                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.35E+02   0.000      Eq. F7-1      1     0.00 |
| Minor       0.00E+00   1.78E+02   0.000      Eq. F7-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       3.92E+02   0.00E+00                                       |
| Minor       2.97E+02   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.35E+02   0.00E+00   1.12E+02                   |
|                      1.78E+02   0.00E+00   0.00E+00                   |
| Flexure Tens         2.35E+02   0.00E+00   1.18E+02                   |
|                      1.78E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flx Tor Comp Shr      0.186      Eq. H3-6      1     0.00             |
```

```
| Flx Tor Tens Shr      0.186      Eq. H3-6      1     0.00              |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /            |
|                      Vcx /      Vcy /      Vrx /      Vry /            |
|                      Tc         Tr         Pc         Pr               |
| Flx Tor Comp Shr     2.35E+02   1.78E+02   0.00E+00   0.00E+00         |
|                      2.54E+01   4.08E+01   0.00E+00   0.00E+00         |
|                      1.67E+02   7.20E+01   1.12E+02   0.00E+00         |
| Flx Tor Tens Shr     2.35E+02   1.78E+02   0.00E+00   0.00E+00         |
|                      2.54E+01   4.08E+01   0.00E+00   0.00E+00         |
|                      1.67E+02   7.20E+01   1.18E+02   0.00E+00         |
|-----------------------------------------------------------------------|
|-----------------------------------------------------------------------------
----|
| AISC DESIGN
GUIDE-9                                                             |
|-----------------------------------------------------------------------------
----|
| Member:     1 Section:       HSST6X4X0.25
AISC                        |
| End Condition:  Fix-
Free                                                        |
| Shear Capacity :  1.6527E+01  Shear Criteria : DG9 Eq.
4.18                       |
| Normal Capacity:  2.7545E+01  Normal Criteria: DG9 Eq.
4.17                       |
|-----------------------------------------------------------------------------
----|
      STAAD SPACE                                                -- PAGE NO.
10
| Stress Point   Shear     Ratio     LC/ | Stress Point   Normal    Ratio
LC/ |
|                                   Sect |
Sect |
| Mid Flange:   0.00000E+00 0.0000     1 | Flange Edge:  0.00000E+00
0.0000     1 |
|                                  0.000 |
0.000 |
| FL/WB Junc:   6.43777E+00 0.3895     1
|                                        |
|                                  0.000
|                                        |
| Mid-Web:      6.43777E+00 0.3895     1
|                                        |
|                                  0.000
|                                        |
|-----------------------------------------------------------------------------
----|
| Pure Torsion Shear Stress     Criteria: DG9
T4.1                                   |
| Flange Edge:  6.43777E+00     Web Edge:     6.43777E
+00                            |
|-----------------------------------------------------------------------------
----|
| Pure Flexure Shear Stress     Criteria: DG9 Eq.
4.6                               |
| Flange Edge:  0.00000E+00  FL/WB Junc:   0.00000E+00  Mid-Web:      0.00000E
+00 |
|-----------------------------------------------------------------------------
```

```
----|
| Direct Axial Bending Stress   Criteria: DG9 Eq.
4.5                             |
| Flange Edge:  0.00000E
+00                                                   |
|------------------------------------------------------------------------------
----|
```

V. AISC 360-05 I Minor Axis Bending F.5

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.5, pp.F-25,26

Details

From reference (2):

> Select an ASTM A992 W-shape beam loaded in its minor axis with a simple span of 15 ft. The nominal loads are a total uniform dead load of 0.667 kip/ft and a uniform live load of 2 kip/ft. Limit the live load deflection to L/240. Assume the beam is braced at the ends only.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	122	122	none	
M_n / Ω_b (ft·k) [ASD]	81.4	81.1	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 I Minor Axis Bending F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 15 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
```

```
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X58
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MY
1 FIXED BUT MY
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GZ -0.667
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GZ -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W12X58                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.923        3
                        0.00          112.51           0.00       7.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
```

```
| Actual Slenderness Ratio    :     71.747  L/C  :     3                     |
| Allowable Slenderness Ratio :    300.000  LOC  :     7.50                  |
|----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                            |
| Critical L/C  :      3    Ratio     :     0.923(PASS)                     |
|          Loc  :    7.50   Condition :   Eq. H1-1b                          |
|----------------------------------------------------------------------------|
| DESIGN FORCES                                                              |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                    |
| Mx:   0.000E+00     My:   1.125E+02     Mz:   0.000E+00                    |
|----------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                           |
| Azz:   1.280E+01     Ayy:   4.392E+00     Cw:  3.575E+03                   |
| Szz:   7.787E+01     Syy:   2.140E+01                                      |
| Izz:   4.750E+02     Iyy:   1.070E+02     Ix:  2.100E+00                   |
|----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                        |
| Fyld:        7199.999   Fu:        9359.999                                |
|----------------------------------------------------------------------------|
| Actual Member Length:        15.000                                        |
| Design Parameters                                                          |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00             |
|----------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE            |
|                STIFFENED                                                   |
| Compression :  Non-Slender     7.81     N/A     13.49    T.B4.1-3         |
|                Non-Slender    27.00     N/A     35.88    T.B4.1-10        |
| Flexure     :  Compact         7.81     9.15    24.08    T.B4.1-1         |
|                Compact        27.00    90.55   137.27    T.B4.1-9         |
     STAAD SPACE                                              -- PAGE NO.
4
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                    |
|                                                                            |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Yield       0.00E+00   7.65E+02   0.000      Eq. D2-1      3     0.00      |
| Rupture     0.00E+00   8.29E+02   0.000      Eq. D2-2      3     0.00      |
|----------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                                |
|                                                                            |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
| Maj Buck    0.00E+00   7.03E+02   0.000      Sec. E1       3     0.00      |
| Min Buck    0.00E+00   5.25E+02   0.000      Eq. E3-1      3     0.00      |
| Flexural                                                                   |
| Tor Buck    0.00E+00   6.13E+02   0.000      Eq. E4-1      3     0.00      |
| Intermediate                                                               |
| Results     Eff Area     KL/r      Fcr        Fe         Pn                |
| Maj Buck    1.18E-01    34.05   6.61E+03   3.55E+04   7.81E+02            |
| Min Buck    1.18E-01    71.75   4.94E+03   8.01E+03   5.83E+02            |
| Flexural       Ag       Fcr        Pn                                      |
| Tor Buck    1.18E-01   5.77E+03   6.81E+02                                 |
|----------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                            |
|                                                                            |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC        |
```

```
| Local-Z    -3.00E+01   3.46E+02   0.087      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.32E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     8.89E-02     1.00     1.20     7.81   3.84E+02             |
| Local-Y     3.05E-02     1.00     0.00    27.00   1.32E+02             |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   3.24E+02   0.000      Eq. F2-1      3     0.00 |
| Minor       1.13E+02   1.22E+02   0.923      Eq. F6-1      3     7.50 |
| Intermediate   Mn         My                                          |
| Major       3.60E+02   0.00E+00                                        |
| Minor       1.35E+02   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.89E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb      |
| Major       3.21E+02   0.00E+00    1.00     8.86    29.98    15.00    |
    STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                      RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp          0.923      Eq. H1-1b     3     7.50              |
| Flexure Tens          0.923      Eq. H1-1b     3     7.50              |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.89E+02   0.00E+00   5.25E+02                   |
|                      1.22E+02   1.13E+02   0.00E+00                   |
| Flexure Tens         2.89E+02   0.00E+00   7.65E+02                   |
|                      1.22E+02   1.13E+02   0.00E+00                   |
|-----------------------------------------------------------------------|
   47. LOAD LIST 4
   48. PARAMETER 2
   49. CODE AISC UNIFIED 2005
   50. FU 9360 ALL
   51. FYLD 7200 ALL
   52. METHOD ASD
   53. TRACK 2 ALL
   54. CHECK CODE ALL
 STEEL DESIGN
    STAAD SPACE                                              -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W12X58                   (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.925        4
```

```
                        0.00          75.01           0.00        7.50
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     71.747  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     7.50             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.925(PASS)                 |
 |          Loc  :    7.50   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   7.501E+01     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   1.280E+01     Ayy:   4.392E+00     Cw:  3.575E+03              |
 | Szz:   7.787E+01     Syy:   2.140E+01                                 |
 | Izz:   4.750E+02     Iyy:   1.070E+02     Ix:  2.100E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        15.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     7.81     N/A     13.49    T.B4.1-3     |
 |                Non-Slender    27.00     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         7.81     9.15    24.08    T.B4.1-1     |
 |                Compact        27.00    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       0.00E+00   5.09E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   5.52E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck    0.00E+00   4.68E+02   0.000      Sec. E1       4     0.00 |
 | Min Buck    0.00E+00   3.49E+02   0.000      Eq. E3-1      4     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   4.08E+02   0.000      Eq. E4-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.18E-01    34.05   6.61E+03   3.55E+04   7.81E+02        |
 | Min Buck    1.18E-01    71.75   4.94E+03   8.01E+03   5.83E+02        |
 | Flexural       Ag       Fcr        Pn                                 |
 | Tor Buck    1.18E-01   5.77E+03   6.81E+02                            |
 |-----------------------------------------------------------------------|
```

```
| CHECK FOR SHEAR                                                        |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    -2.00E+01   2.30E+02   0.087      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   8.78E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                           |
| Results        Aw        Cv        Kv      h/tw      Vn              |
| Local-Z     8.89E-02     1.00      1.20     7.81   3.84E+02           |
| Local-Y     3.05E-02     1.00      0.00    27.00   1.32E+02           |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   2.16E+02   0.000      Eq. F2-1      4     0.00 |
| Minor       7.50E+01   8.11E+01   0.925      Eq. F6-1      4     7.50 |
| Intermediate   Mn        My                                            |
| Major       3.60E+02   0.00E+00                                        |
| Minor       1.35E+02   0.00E+00                                        |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       0.00E+00   1.92E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major       3.21E+02   0.00E+00    1.00     8.86    29.98    15.00    |
    STAAD SPACE                                             -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.925      Eq. H1-1b     4     7.50           |
| Flexure Tens            0.925      Eq. H1-1b     4     7.50           |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          1.92E+02   0.00E+00   3.49E+02                  |
|                       8.11E+01   7.50E+01   0.00E+00                  |
| Flexure Tens          1.92E+02   0.00E+00   5.09E+02                  |
|                       8.11E+01   7.50E+01   0.00E+00                  |
|------------------------------------------------------------------------|
```

V. AISC 360-05 L Shear Capacity G.3

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.3, p.G-7

Details

From reference (2):

> Verify the shear strength of a 5×3×1/4 (LLV) ASTM A36 angle with end shears of 3.5 kips from dead load and 10.5 kips from live load.

Validation

Ultimate load:

$$V_u = 1.2(3.50 \text{ kips}) + 1.6(10.5 \text{ kips}) = 21.0 \text{ kips}$$

Service load:

$$V_a = 3.50 \text{ kips} + 10.5 \text{ kips} = 14.0 \text{ kips}$$

Area of the web for shear:

$$A_w = bt = 5.0(0.25) = 1.25 \text{ in}^2$$

Section G4 of Reference 1 gives $k_v = 1.2$.

$$\frac{h}{t_w} = \frac{b}{t} = \frac{5.0}{0.25} = 20 < 1.10\sqrt{\frac{k_v E}{F_y}} = 1.10\sqrt{\frac{1.2(29,\ 000)}{36}} = 34.2$$

Therefore, $C_v = 1.0$.

<u>Nominal Shear Strength</u>

$$V_n = 0.6F_y A_w C_v = 0.6(36)(1.25)(1.0) = 27.0 \text{ kips} \quad \text{(Eq. G2-1)}$$

The available shear strength:

LRFD: $\phi_v V_n = 0.90(27.0) = 24.3 \text{ kips}$

ASD: $V_n / \Omega_v = {}^{27.0}\!/_{1.67} = 16.2 \text{ kips}$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	24.3	24.3	none	
V_n/Ω_v (kips) [ASD]	16.2	16.2	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 L Shear Capacity G.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 4 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
```

```
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST L50304
CONSTANTS
BETA 45 ALL
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON Y -10
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON Y -20
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 36 ALL
FU 58 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   L50304                      (AISC SECTIONS)
                           FAIL   Eq. H2-1            14.492         3
                        0.00           230.75         297.25       24.00
```

```
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :     73.620  L/C  :     3                  |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      3    Ratio     :    14.492(FAIL)                   |
|          Loc  :   24.00   Condition :   Eq. H2-1                       |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   2.200E+01     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   2.307E+02     Mz:   2.973E+02                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   1.250E+00     Ayy:   7.500E-01     Cw:  6.060E-02                |
| Szz:   5.259E-01     Syy:   1.692E+00                                   |
| Izz:   8.247E-01     Iyy:   5.723E+00     Ix:  4.042E-02                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:          36.000   Fu:          58.000                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        48.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Slender        20.00     N/A     12.77    T.B4.1-5      |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1-6      |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   6.29E+01   0.000      Eq. D2-1      3     0.00  |
| Rupture     0.00E+00   8.44E+01   0.000      Eq. D2-2      3     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Min Buck    0.00E+00   3.18E+01   0.000      Eq. E7-1      3     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr         Fe         Pn           |
| Min Buck    1.94E+00   104.61   1.82E+01   2.62E+01   3.53E+01          |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.46E+01   0.000      Eq. G2-1      3     0.00  |
| Local-Y     2.20E+01   2.43E+01   0.905      Eq. G2-1      3     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
```

```
| Local-Z     7.50E-01     1.00     1.20    12.00   1.62E+01           |
| Local-Y     1.25E+00     1.00     1.20    20.00   2.70E+01           |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.31E+02   8.23E+01   2.805      Eq. F10-1     3     0.00 |
| Minor       2.97E+02   2.56E+01  11.631      Eq. F10-1     3    24.00 |
| Intermediate   Mn         My                                          |
| Major       9.14E+01   6.09E+01                                       |
| Minor       2.84E+01   1.89E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.31E+02   8.06E+01   2.861      Eq. F10-3     3     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb      |
| Major       8.96E+01   4.13E+02    1.00     0.00     0.00    48.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Minor      -2.31E+02   4.17E+01   5.539      Eq. F10-7     3     0.00 |
| Intermediate   Mn         Me                                          |
| Minor       4.63E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|              STRESS     RATIO      CRITERIA     L/C    LOC           |
| Shear       1.94E+01    0.905      Eq. H3-8      3     0.00          |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp         14.492      Eq. H2-1      3    24.00            |
| Flexure Tens         14.492      Eq. H2-1      3    24.00            |
| Intermediate         frbw /     frbz /     fra /                     |
|                      Fcbw       Fcbz       Fca                       |
| Flexure Comp         9.59E+01   5.65E+02   0.00E+00                  |
|                      3.35E+01   4.86E+01   1.64E+01                  |
| Flexure Tens         1.36E+02   5.65E+02   0.00E+00                  |
|                      4.76E+01   4.86E+01   3.24E+01                  |
|-----------------------------------------------------------------------|
    49. LOAD LIST 4
    50. PARAMETER 2
    51. CODE AISC UNIFIED 2005
    52. METHOD ASD
    53. FYLD 36 ALL
    54. FU 58 ALL
    55. TRACK 2 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   L50304                  (AISC SECTIONS)
                           FAIL   Eq. H2-1           14.851         4
                       0.00          157.33         202.67       24.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     73.620  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    24.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :    14.851(FAIL)                 |
|          Loc  :   24.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   1.500E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   1.573E+02     Mz:   2.027E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.250E+00     Ayy:   7.500E-01     Cw:  6.060E-02             |
| Szz:   5.259E-01     Syy:   1.692E+00                                |
| Izz:   8.247E-01     Iyy:   5.723E+00     Ix:  4.042E-02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          36.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        48.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.00     N/A     12.77    T.B4.1-5     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact    20.00    15.33    25.83    T.B4.1-6     |
|                  N/A           N/A      N/A      N/A      N/A         |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.18E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   5.63E+01   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Min Buck    0.00E+00   2.12E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe          Pn          |
```

```
| Min Buck    1.94E+00   104.61   1.82E+01   2.62E+01   3.53E+01        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   9.70E+00   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.50E+01   1.62E+01   0.928      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     7.50E-01     1.00     1.20    12.00   1.62E+01            |
| Local-Y     1.25E+00     1.00     1.20    20.00   2.70E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -1.57E+02   5.47E+01   2.875      Eq. F10-1     4     0.00 |
| Minor       2.03E+02   1.70E+01  11.919      Eq. F10-1     4    24.00 |
| Intermediate   Mn        My                                          |
| Major       9.14E+01   6.09E+01                                       |
| Minor       2.84E+01   1.89E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -1.57E+02   5.37E+01   2.932      Eq. F10-3     4     0.00 |
| Intermediate   Mn        Me        Cb      Lp       Lr       Lb      |
| Major       8.96E+01   4.13E+02    1.00     0.00     0.00    48.00    |
    STAAD SPACE                                             -- PAGE NO.
8
                    STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LEG LOCAL BUCKLING                                  |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Minor      -1.57E+02   2.77E+01   5.676      Eq. F10-7     4     0.00 |
| Intermediate   Mn        Me                                          |
| Minor       4.63E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR AND NORMAL STRESS INTERACTION                         |
|                                                                       |
|             STRESS     RATIO      CRITERIA     L/C    LOC            |
| Shear       1.29E+01    0.928      Eq. H3-8      4     0.00           |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp        14.851      Eq. H2-1      4    24.00             |
| Flexure Tens        14.851      Eq. H2-1      4    24.00             |
| Intermediate        frbw /     frbz /     fra /                       |
|                     Fcbw       Fcbz       Fca                         |
| Flexure Comp        6.54E+01   3.85E+02   0.00E+00                    |
|                     2.23E+01   3.23E+01   1.09E+01                    |
| Flexure Tens        9.30E+01   3.85E+02   0.00E+00                    |
|                     3.17E+01   3.23E+01   2.16E+01                    |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Pipe F.9

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.9, pp.F-35,36

Details

From reference (2):

Select an ASTM A53 grade B Pipe shape with a simple span of 16 ft. The nominal loads are a total uniform dead load of 0.32 kip/ft and a uniform live load of 0.96 kip/ft. Assume there is no deflection limit for this beam. The beam is braced only at the ends.

Validation

Ultimate moment:

$$w_u = 1.2(0.32 \text{ k/ft}) + 1.6(0.96 \text{ k/ft}) = 1.92 \text{ k/ft}$$

$$M_u = \frac{1.92(16)^2}{8} = 61.4 \text{ ft}\cdot\text{k}$$

Service moment:

$$w_a = 0.32 \text{ k/ft} + 0.96 \text{ k/ft} = 1.28 \text{ k/ft}$$

$$M_a = \frac{1.28(16)^2}{8} = 41.0 \text{ ft}\cdot\text{k}$$

Try a pipe 8 x-strong, which has a plastic section modulus, $Z_x = 31.0\ in^3$.

The yield strength, $F_y = 35\ ksi$

Compactness ratio: $\lambda = D/t = 18.5$

Determine the slender flange ratio limit from Table B4.1b: $\lambda_p = \frac{0.07E}{F_y} = \frac{0.07(29{,}000)}{35} = 58.0 > \lambda$ (Compact)

By observation, $\lambda < \frac{0.45E}{F_y}$, so Section F8 applies.

$$\lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{46}} = 60.8 > \lambda$$

$$M_n = M_p = F_y \times Z_x = 35 \times 31.0 = 1{,}090 \text{ in-k} = 90.4 \text{ ft-k} \quad \text{(F8-1)}$$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(90.4) = 81.4\ ft\cdot k$

ASD: $M_n / \Omega_b = {}^{90.4}/_{1.67} = 54.1\ ft\cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	81.4	81.4	none	
M_n / Ω_b (ft·k) [ASD]	54.1	54.1	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Pipe F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 16 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPX80
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.32
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.96
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
```

```
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FU 8640 ALL
FYLD 5040 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FU 8640 ALL
FYLD 5040 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY              MZ       LOCATION
 =======================================================================
       1 ST   PIPX80                   (AISC SECTIONS)
                           PASS   Eq. H1-1b            0.755         3
                        0.00            0.00         -61.44       8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     66.233  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.755(PASS)                 |
|          Loc  :    8.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -6.144E+01               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.344E+00     Ayy:   6.344E+00     Cw:  0.000E+00              |
| Szz:   2.319E+01     Syy:   2.319E+01                                 |
| Izz:   1.000E+02     Iyy:   1.000E+02     Ix:  2.000E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        5039.999   Fu:        8639.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        16.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
```

```
| Compression :  Non-Slender    18.55     N/A     91.14    T.B4.1-15a   |
|                Non-Slender    18.55     N/A     91.14    T.B4.1-15a   |
| Flexure     :  Compact        18.55    58.00   256.86    T.B4.1-15b   |
|                Compact        18.55    58.00   256.86    T.B4.1-15b   |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   3.75E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   5.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   2.99E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   2.99E+02   0.000      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02        |
| Min Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.12E+02   0.000      Eq. G6-1      3     0.00 |
| Local-Y     1.54E+01   1.12E+02   0.137      Eq. G6-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     8.26E-02     0.00     0.00     0.00   1.25E+02            |
| Local-Y     8.26E-02     0.00     0.00     0.00   1.25E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   7.66E+01   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.02E+03   8.51E+01                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       6.14E+01   8.14E+01   0.755      Eq. F8-1      3     8.00 |
| Minor       0.00E+00   8.14E+01   0.000      Eq. F8-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       9.04E+01   0.00E+00                                       |
| Minor       9.04E+01   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
```

```
|                         RATIO      CRITERIA    L/C     LOC                  |
| Flexure Comp            0.755      Eq. H1-1b     3     8.00                 |
| Flexure Tens            0.755      Eq. H1-1b     3     8.00                 |
| Intermediate           Mcx /      Mrx /      Pc /                           |
|                        Mcy        Mry        Pr                             |
| Flexure Comp           8.14E+01   6.14E+01   2.99E+02                       |
|                        8.14E+01   0.00E+00   0.00E+00                       |
| Flexure Tens           8.14E+01   6.14E+01   3.75E+02                       |
|                        8.14E+01   0.00E+00   0.00E+00                       |
|-----------------------------------------------------------------------------|
    55. LOAD LIST 4
    56. PARAMETER 2
    57. CODE AISC UNIFIED 2005
    58. METHOD ASD
    59. FU 8640 ALL
    60. FYLD 5040 ALL
    61. TRACK 2 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   PIPX80                   (AISC SECTIONS)
                          PASS   Eq. H1-1b           0.757         4
                       0.00            0.00        -40.96       8.00
|-----------------------------------------------------------------------------|
| SLENDERNESS                                                                 |
| Actual Slenderness Ratio    :     66.233  L/C  :     4                      |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00                   |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      4    Ratio     :     0.757(PASS)                      |
|          Loc  :    8.00   Condition :   Eq. H1-1b                           |
|-----------------------------------------------------------------------------|
| DESIGN FORCES                                                               |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00                     |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -4.096E+01                     |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                            |
| Azz:   6.344E+00     Ayy:   6.344E+00     Cw:  0.000E+00                    |
| Szz:   2.319E+01     Syy:   2.319E+01                                       |
| Izz:   1.000E+02     Iyy:   1.000E+02     Ix:  2.000E+02                    |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5039.999   Fu:        8639.999                                 |
|-----------------------------------------------------------------------------|
| Actual Member Length:        16.000                                         |
```

```
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender    18.55     N/A     91.14    T.B4.1-15a    |
|                Non-Slender    18.55     N/A     91.14    T.B4.1-15a    |
| Flexure     :  Compact        18.55    58.00   256.86    T.B4.1-15b    |
|                Compact        18.55    58.00   256.86    T.B4.1-15b    |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   2.49E+02   0.000      Eq. D2-1      4     0.00  |
| Rupture     0.00E+00   3.57E+02   0.000      Eq. D2-2      4     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   1.99E+02   0.000      Sec. E1       4     0.00  |
| Min Buck    0.00E+00   1.99E+02   0.000      Eq. E3-1      4     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02         |
| Min Buck    8.26E-02    66.23   4.03E+03   9.40E+03   3.33E+02         |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   7.48E+01   0.000      Eq. G6-1      4     0.00  |
| Local-Y     1.02E+01   7.48E+01   0.137      Eq. G6-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn                |
| Local-Z     8.26E-02     0.00     0.00     0.00   1.25E+02             |
| Local-Y     8.26E-02     0.00     0.00     0.00   1.25E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             0.00E+00   5.10E+01   0.000      Eq. H3-1      4     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.02E+03   8.51E+01                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       4.10E+01   5.41E+01   0.757      Eq. F8-1      4     8.00  |
| Minor       0.00E+00   5.41E+01   0.000      Eq. F8-1      4     0.00  |
| Intermediate   Mn        My                                             |
| Major       9.04E+01   0.00E+00                                        |
| Minor       9.04E+01   0.00E+00                                        |
     STAAD SPACE                                              -- PAGE NO.
8
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       *********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.757      Eq. H1-1b     4     8.00             |
| Flexure Tens          0.757      Eq. H1-1b     4     8.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         5.41E+01   4.10E+01   1.99E+02                   |
|                      5.41E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         5.41E+01   4.10E+01   2.49E+02                   |
|                      5.41E+01   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 Rect HSS E.9

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.9, pp.E-37,38

Details

From reference (2):

> Select a rectangular HSS compression member, with a length of 20 ft, to support a dead load of 85 kips and live load of 255 kips in axial compression. The base is fixed and the top is pinned.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	518	518	none	
P_n/Ω_c (kips) [ASD]	344	345	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Rect HSS E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
```

```
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 240 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST12X10X0.375
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -85
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FY -255
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
KX 0.8 ALL
KY 0.8 ALL
METHOD LRFD
TRACK 2 ALL
KZ 0.8 ALL
FU 58 ALL
FYLD 46 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
KX 0.8 ALL
KY 0.8 ALL
KZ 0.8 ALL
FU 58 ALL
FYLD 46 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   HSST12X10X0.375          (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.985         3
                      510.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     47.959  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.985(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.100E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.249E+00     Ayy:   7.645E+00     Cw:  0.000E+00              |
| Szz:   5.167E+01     Syy:   4.680E+01                                 |
| Izz:   3.100E+02     Iyy:   2.340E+02     Ix:  4.210E+02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       240.000                                   |
| Design Parameters                                                     |
| Kz:    0.80 Ky:    0.80 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    31.38     N/A     35.15    T.B4.1-12    |
| Flexure     :  Compact        25.65    28.12    35.15    T.B4.1-12    |
|                Compact        31.38    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   6.04E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   6.35E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    5.10E+02   5.38E+02   0.948      Sec. E1       3     0.00 |
```

```
| Min Buck    5.10E+02   5.18E+02   0.985      Eq. E3-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.46E+01    41.67   4.09E+01   1.65E+02   5.98E+02        |
| Min Buck    1.46E+01    47.96   3.94E+01   1.24E+02   5.75E+02        |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.55E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.90E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     6.25E+00     1.00     5.00    25.65   1.72E+02           |
| Local-Y     7.65E+00     1.00     5.00    31.38   2.11E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             0.00E+00   1.94E+03   0.000      Eq. H3-1      3     0.00 |
| Intermediate Fcr        Tn                                            |
|             2.76E+01   2.16E+03                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.53E+03   0.000      Eq. F7-1      3     0.00 |
| Minor       0.00E+00   2.24E+03   0.000      Eq. F7-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       2.81E+03   0.00E+00                                       |
| Minor       2.48E+03   0.00E+00                                       |
     STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Minor       0.00E+00   2.10E+03   0.000      Eq. F7-2      3     0.00 |
| Intermediate   Mn         Fcr                                         |
| Minor       2.33E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.985      Eq. H1-1a     3     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.53E+03   0.00E+00   5.18E+02                   |
|                      2.10E+03   0.00E+00   5.10E+02                   |
| Flexure Tens         2.53E+03   0.00E+00   6.04E+02                   |
|                      2.10E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
```

```
    53. KX 0.8 ALL
    54. KY 0.8 ALL
    55. KZ 0.8 ALL
    56. FU 58 ALL
    57. FYLD 46 ALL
    58. METHOD ASD
    59. TRACK 2 ALL
    60. CHECK CODE ALL
 STEEL DESIGN
   WARNING: For member#      1 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   HSST12X10X0.375          (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.987         4
                      340.00 C          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     47.959  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.987(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   3.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   6.249E+00     Ayy:   7.645E+00     Cw:  0.000E+00             |
| Szz:   5.167E+01     Syy:   4.680E+01                                 |
| Izz:   3.100E+02     Iyy:   2.340E+02     Ix:  4.210E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          46.000   Fu:          58.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       240.000                                   |
| Design Parameters                                                     |
| Kz:    0.80 Ky:    0.80 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
|                Non-Slender    31.38     N/A     35.15    T.B4.1-12    |
| Flexure     :  Compact        25.65    28.12    35.15    T.B4.1-12    |
|                Compact        31.38    60.76   143.12    T.B4.1-13    |
      STAAD SPACE                                              -- PAGE NO.
```

```
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield      0.00E+00   4.02E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture    0.00E+00   4.23E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck   3.40E+02   3.58E+02   0.950      Sec. E1       4     0.00 |
 | Min Buck   3.40E+02   3.45E+02   0.987      Eq. E3-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr         Fe         Pn          |
 | Maj Buck   1.46E+01    41.67   4.09E+01   1.65E+02   5.98E+02         |
 | Min Buck   1.46E+01    47.96   3.94E+01   1.24E+02   5.75E+02         |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Local-Z    0.00E+00   1.03E+02   0.000      Eq. G2-1      4     0.00 |
 | Local-Y    0.00E+00   1.26E+02   0.000      Eq. G2-1      4     0.00 |
 | Intermediate                                                          |
 | Results       Aw         Cv       Kv      h/tw      Vn                |
 | Local-Z    6.25E+00     1.00     5.00    25.65   1.72E+02             |
 | Local-Y    7.65E+00     1.00     5.00    31.38   2.11E+02             |
 |-----------------------------------------------------------------------|
 | CHECK FOR TORSION                                                     |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 |            0.00E+00   1.29E+03   0.000      Eq. H3-1      4     0.00 |
 | Intermediate Fcr        Tn                                            |
 |            2.76E+01   2.16E+03                                        |
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-YIELDING                                            |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Major      0.00E+00   1.68E+03   0.000      Eq. F7-1      4     0.00 |
 | Minor      0.00E+00   1.49E+03   0.000      Eq. F7-1      4     0.00 |
 | Intermediate   Mn        My                                           |
 | Major      2.81E+03   0.00E+00                                        |
 | Minor      2.48E+03   0.00E+00                                        |
     STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Minor      0.00E+00   1.40E+03   0.000      Eq. F7-2      4     0.00 |
 | Intermediate   Mn        Fcr                                          |
 | Minor      2.33E+03   0.00E+00                                        |
```

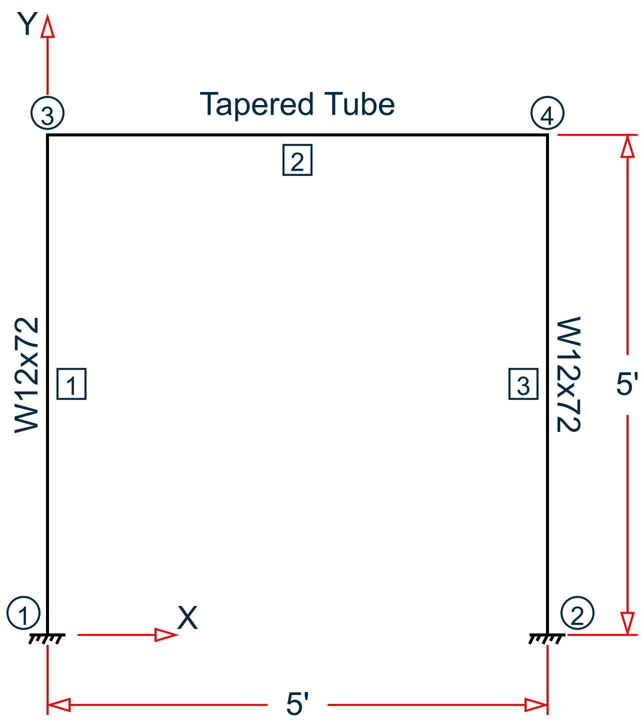
```
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.987      Eq. H1-1a     4     0.00             |
| Flexure Tens         0.000      Eq. H1-1b     4     0.00             |
| Intermediate        Mcx /      Mrx /      Pc /                      |
|                     Mcy        Mry        Pr                        |
| Flexure Comp        1.68E+03   0.00E+00   3.45E+02                   |
|                     1.40E+03   0.00E+00   3.40E+02                   |
| Flexure Tens        1.68E+03   0.00E+00   4.02E+02                   |
|                     1.40E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
```

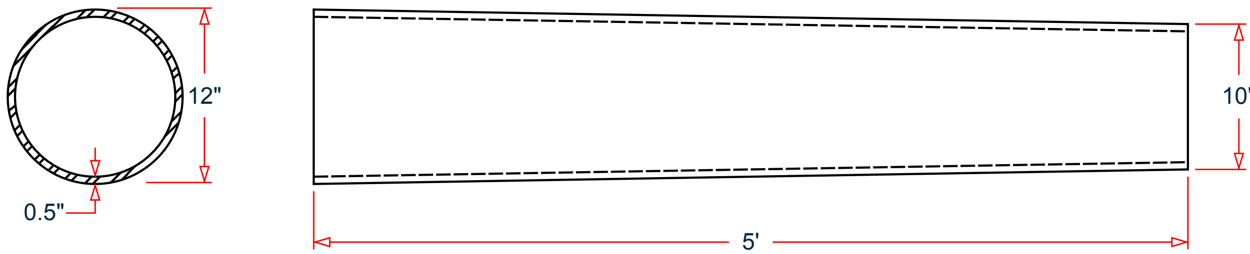
V. AISC 360-05 Tapered Tube Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered tube section member per both the LRFD and ASD methods of the AISC 360-05 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





Section Elevation

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concreted torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the right end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

$M_z = 505.66\ in \cdot kips$
$F_y = 67.2\ kips$
$F_x = 10.23\ kips$

Section Properties

The section at the right end ($OD = 10\ in.$) has the following properties:

Inner diameter, ID = 9 in.
Area, $A_g = (\pi / 4)(OD^2 - ID^2) = 14.92\ in^2$
Form factor = $0.49 + 0.8\ (t\ /\ OD) = 0.53$
Shear area, $A_y = 14.92(0.53) = 7.909\ in^2$
Moment of inertia,

$$I = \frac{\pi}{64}\left(\mathrm{OD}^4\text{-}\mathrm{ID}^4\right) = 168.8\ \mathrm{in}^4$$

Elastic section modulus, $S = \frac{\pi}{32}\left(\frac{OD^{\,4}\text{-}ID^{\,4}}{OD}\right) = 33.76\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{168.8}{14.92}} = 3.363\ in$

Plastic section modulus, $Z = \frac{OD^{\,3}\text{-}ID^{\,3}}{6} = 45.17\ in^3$

HSS Torsional constant, $C = \frac{\pi}{2}(OD\text{-}t)^2 t = 70.88\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{\text{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1-15a, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 63.8$, therefore flanges are non-slender for compression. The web is similarly non-slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{\text{OD}}{t} = \frac{10}{0.5} = 20$$

Per Table B4.1-15b, $\lambda < \lambda_r = 0.07\frac{E}{F_y} = 40.6$, therefore flange is compact for bending.

Compression Capacity

$$\frac{kL}{r} = \frac{1.0(60)}{3.363} = 17.84 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

$$F_e = \frac{\pi^2 E}{\left(\frac{kL}{r}\right)^2} = \frac{\pi^2 \times 29{,}000}{(17.84)^2} = 899.4 \text{ ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{899.4} = 0.056 < 2.25$$

$$F_{\text{crx}} = 0.658^{F_y/F_e} F_y = 0.658^{(0.056)} 50 = 48.85 \text{ ksi} \qquad \text{(Eq. E3-2)}$$

$$\text{P}_\text{n} = \text{A}_\text{g} \times \text{F}_\text{cr} = 14.92 \times 48.85 = 729.0 \text{ kips} \qquad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 729.0 = 656.1\ kips$
The allowable compression capacity (ASD) = $P_n/\Omega_c = 729.0\ /\ 1.67 = 436.5\ kips$

Shear Capacity

The distance from the maximum shear force (ends) to zero shear force (mid-span), $L_v = L\ /\ 2 = 30\ in.$

The critical shear stress, F_{cr}, *is the minimum of*:

$$F_{\text{cr1}} = \frac{1.60E}{\sqrt{\frac{L_v}{\text{OD}}}\left(\frac{\text{OD}}{t}\right)^{5/4}} = 633.4 \text{ ksi}$$

$$F_{\text{cr2}} = \frac{0.78E}{\left(\frac{\text{OD}}{t}\right)^{3/2}} = 252.9 \text{ ksi}$$

$$F_{\text{cr3}} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30\ ksi$

Nominal shear capacity:

$$V_n = F_{cr} {}^{A_g}\!/_2 = 223.8 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_c V_n = 0.9 \times 223.8 = 201.5 \ kips$
The allowable shear capacity (ASD) = $V_n/\Omega_c = 223.8 \ / \ 1.67 = 134.0 \ kips$

<u>Calculate Bending Capacity</u>

Bending capacity in plastic yielding:

$$\mathrm{M_p = F_y \times Z = 50 \times 45.17 = 2{,}259 \ in{\cdot}kips < 1.6F_y \times S = 1.6\ (50)\ (33.76) = 2{,}701\ in{\cdot}kips}$$

The bending capacity for flexural yielding:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 2{,}259 = 2{,}033 \ in{\cdot}kips$
The allowable bending capacity (ASD) = $M_y/\Omega_b = 2{,}259 \ / \ 1.67 = 1{,}352 \ in{\cdot}kips$

<u>Torsional Capacity</u>

The critical stress, F_{cr}, *is the larger of*:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{\mathrm{OD}}}\left(\frac{\mathrm{OD}}{t}\right)^{5/4}} = 344.3 \text{ ksi}$$

$$F_{cr2} = \frac{1.23E}{\left(\frac{\mathrm{OD}}{t}\right)^{3/2}} = 194.5 \text{ ksi}$$

But shall not exceed:

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

$F_{cr} = 30 \ ksi$

Nominal torsional capacity:

$$T_n = F_{cr}C = 2{,}126 \text{ in}{\cdot}\text{kips}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_c T_n = 0.9 \times 2{,}126 = 1{,}914 \ kips$
The allowable torsional capacity (ASD) = $T_n/\Omega_c = 2{,}126 \ / \ 1.67 = 1{,}273 \ kips$

<u>Interaction Ratio for Bending and Compression</u>

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{10.23}{656.1} = 0.016 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{10.23}{436.5} = 0.023 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.016}{2} + \left(\frac{505.7}{2{,}033} + \frac{0}{2{,}033}\right) = 0.257 \text{ (LRFD)}$$

$$\frac{0.023}{2} + \left(\frac{505.7}{1{,}352} + \frac{0}{1{,}352}\right) = 0.386 \text{ (ASD)}$$

Results

Table 767:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	656.1	656	negligible	
	Shear capacity (kips)	201.5	201	negligible	
	Bending capacity (in·kips)	2,033	2,030	negligible	
	Torsional capacity (in·kips)	1,914	1,910	negligible	
	Interaction ratio	0.257	0.257	none	
ASD	Compression capacity (kips)	436.5	437	negligible	
	Shear capacity (kips)	134.0	134	negligible	
	Bending capacity (in·kips)	1,352	1,350	negligible	
	Torsional capacity (in·kips)	1,273	1,270	negligible	
	Interaction ratio	0.386	0.386	none	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-10 Tapered Tube Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 PRIS ROUND STA 12 END 10 THI 0.5
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2010
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2010
```

```
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=    10: 2:26                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: AISC 360-10 Tapered Tube Section.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 13-APR-21
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT INCHES KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0
     9. MEMBER INCIDENCES
    10. 1 1 2; 2 2 3; 3 3 4
    11. DEFINE MATERIAL START
    12. ISOTROPIC STEEL
    13. E 29000
    14. POISSON 0.3
    15. DENSITY 0.000283
    16. ALPHA 6.5E-06
    17. DAMP 0.03
    18. G 11200
    19. TYPE STEEL
    20. STRENGTH RY 1.5 RT 1.2
    21. END DEFINE MATERIAL
    22. MEMBER PROPERTY AMERICAN
    23. 1 TABLE ST W12X72
    24. 3 TABLE ST W12X72
    25. MEMBER PROPERTY
    26. 2 PRIS ROUND STA 12 END 10 THI 0.5
    27. CONSTANTS
    28. MATERIAL STEEL ALL
    29. SUPPORTS
    30. 1 FIXED
    31. 4 FIXED
    32. LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
```

```
    33. MEMBER LOAD
    34. 2 UNI GY -2.25
    35. LOAD 2 LOADTYPE NONE  TITLE LOAD CASE 2
    36. JOINT LOAD
    37. 2 FX 50 FZ 25
    38. LOAD 3 LOADTYPE NONE  TITLE LOAD CASE 3
      STAAD SPACE                                              -- PAGE NO.
2
    39. MEMBER LOAD
    40. 2 CMOM GX 0.75
    41. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =      12
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    42. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                              -- PAGE NO.
3
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT   LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00162  -0.00665   0.00000   0.00000   0.00000  -0.00075
           2    0.07748   0.00168   0.25417   0.00541   0.00296  -0.00079
           3    0.00000   0.00000   0.00013   0.00000   0.00000   0.00000
      3    1    0.00033  -0.00659   0.00000   0.00000   0.00000   0.00069
           2    0.07448  -0.00168   0.07663   0.00255   0.00296  -0.00085
           3    0.00000   0.00000   0.00011   0.00000   0.00000   0.00000
      4    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      STAAD SPACE                                              -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT   LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     10.23     67.81      0.00      0.00      0.00    -89.36
           2    -26.14    -17.09    -24.95  -1258.27     -1.62   1013.33
           3      0.00      0.00      0.00     -0.39     -0.00      0.00
      4    1    -10.23     67.19      0.00      0.00      0.00    108.13
           2    -23.86     17.09     -0.05   -241.73     -1.62    961.14
           3      0.00      0.00     -0.00     -0.36     -0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER   LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     67.81    -10.23     0.00      0.00      0.00     -89.36
                 2    -67.81     10.23     0.00      0.00      0.00    -524.44
```

```
          2     1    -17.09     26.14   -24.95     -1.62   1258.27    1013.33
                2     17.09    -26.14    24.95      1.62    238.50     554.84
          3     1      0.00      0.00     0.00     -0.00      0.39       0.00
                2      0.00      0.00    -0.00      0.00     -0.39       0.00
     2    1     2     10.23     67.81     0.00      0.00      0.00     524.44
                3    -10.23     67.19     0.00      0.00      0.00    -505.66
          2     2     23.86    -17.09     0.05    238.50     -1.62    -554.84
                3    -23.86     17.09    -0.05   -238.50     -1.62    -470.69
          3     2      0.00      0.00     0.00     -0.39     -0.00       0.00
                3      0.00      0.00    -0.00     -0.36     -0.00       0.00
     3    1     3     67.19     10.23     0.00      0.00      0.00     505.66
                4    -67.19    -10.23     0.00      0.00      0.00     108.13
          2     3     17.09     23.86     0.05     -1.62    238.50     470.69
                4    -17.09    -23.86    -0.05      1.62   -241.73     961.14
          3     3      0.00      0.00     0.00     -0.00      0.36       0.00
                4      0.00      0.00    -0.00      0.00     -0.36       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    43. PARAMETER 1
    44. CODE AISC UNIFIED 2010
    45. SLF 0.8 MEMB 2
    46. FYLD 50 ALL
    47. FU 60 ALL
    48. METHOD LRFD
    49. STP 2 MEMB 2
    50. TRACK 2 MEMB 2
    51. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                            -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      2      PRISMAT                  (AISC SECTIONS)
                           PASS   Sec. G1             0.334         1
                       10.23 C          0.00        505.66       60.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.839  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :    60.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.334(PASS)                 |
|          Loc  :   60.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.023E+01(C ) Fy:  -6.719E+01     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.057E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
```

```
| Azz:   7.909E+00     Ayy:   7.909E+00     Cw:  0.000E+00               |
| Szz:   3.376E+01     Syy:   3.376E+01                                  |
| Izz:   1.688E+02     Iyy:   1.688E+02     Ix:  3.376E+02               |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:          50.000   Fu:          60.000                            |
|------------------------------------------------------------------------|
| Actual Member Length:        60.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00         |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
|                Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9  |
| Flexure     :  Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
|                Compact        20.00    40.60   179.80    T.B4.1(b)-20 |
    STAAD SPACE                                               -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   8.13E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   6.50E+02   0.000      Eq. D2-2      1     0.00 |
|------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                            |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    2.39E+01   6.56E+02   0.036      Eq. E3-1      2    60.00 |
| Min Buck    2.39E+01   6.56E+02   0.036      Eq. E3-1      2    60.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02        |
| Min Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02        |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z    -5.39E-02   2.01E+02   0.000      Eq. G6-1      2    60.00 |
| Local-Y    -6.72E+01   2.01E+02   0.334      Eq. G6-1      1    60.00 |
| Intermediate                                                           |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     1.49E+01     0.00     0.00     0.00   2.24E+02            |
| Local-Y     1.49E+01     0.00     0.00     0.00   2.24E+02            |
|------------------------------------------------------------------------|
| CHECK FOR TORSION                                                      |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             2.38E+02   1.91E+03   0.125      Eq. H3-1      2    60.00 |
| Intermediate Fcr        Tn                                             |
|             3.00E+01   2.13E+03                                       |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
```

```
|               FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -5.06E+02   2.03E+03   0.249      Eq. F8-1      1    60.00 |
| Minor       1.62E+00   2.03E+03   0.001      Eq. F8-1      2    60.00 |
| Intermediate   Mn        My                                           |
| Major       2.26E+03   0.00E+00                                       |
| Minor       2.26E+03   0.00E+00                                       |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-10-LRFD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                       RATIO      CRITERIA    L/C     LOC                |
| Flexure Comp          0.257      Eq. H1-1b     1    60.00               |
| Flexure Tens          0.249      Eq. H1-1b     1    60.00               |
| Intermediate         Mcx /      Mrx /      Pc /                        |
|                      Mcy        Mry        Pr                          |
| Flexure Comp         2.03E+03  -5.06E+02   6.56E+02                     |
|                      2.03E+03   0.00E+00   1.02E+01                     |
| Flexure Tens         2.03E+03  -5.06E+02   0.00E+00                     |
|                      2.03E+03   0.00E+00   0.00E+00                     |
|-------------------------------------------------------------------------|
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2010
    54. SLF 0.8 MEMB 2
    55. FYLD 50 ALL
    56. FU 60 ALL
    57. METHOD ASD
    58. STP 2 MEMB 2
    59. TRACK 2 MEMB 2
    60. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2      PRISMAT                  (AISC SECTIONS)
                           PASS   Sec. G1              0.501         1
                       10.23 C          0.00         505.66       60.00
|-------------------------------------------------------------------------|
| SLENDERNESS                                                             |
| Actual Slenderness Ratio    :     17.839  L/C  :     1                  |
| Allowable Slenderness Ratio :    200.000  LOC  :    60.00               |
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      1    Ratio     :     0.501(PASS)                   |
|          Loc  :   60.00   Condition :   Sec. G1                         |
```

```
|--------------------------------------------------------------------------|
| DESIGN FORCES                                                            |
| Fx:   1.023E+01(C ) Fy:  -6.719E+01     Fz:   0.000E+00                  |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.057E+02                  |
|--------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                         |
| Azz:   7.909E+00     Ayy:   7.909E+00     Cw:  0.000E+00                 |
| Szz:   3.376E+01     Syy:   3.376E+01                                    |
| Izz:   1.688E+02     Iyy:   1.688E+02     Ix:  3.376E+02                 |
|--------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                      |
| Fyld:          50.000   Fu:          60.000                              |
|--------------------------------------------------------------------------|
| Actual Member Length:        60.000                                      |
| Design Parameters                                                        |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00          |
|--------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                 |
| Compression :  Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9    |
|                Non-Slender    20.00     N/A     63.80    T.B4.1(a)-9    |
| Flexure     :  Compact        20.00    40.60   179.80    T.B4.1(b)-20   |
|                Compact        20.00    40.60   179.80    T.B4.1(b)-20   |
    STAAD SPACE                                              -- PAGE NO.
10
                    STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                    ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                  |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   5.41E+02   0.000      Eq. D2-1      1     0.00   |
| Rupture     0.00E+00   4.34E+02   0.000      Eq. D2-2      1     0.00   |
|--------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                              |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    2.39E+01   4.37E+02   0.055      Eq. E3-1      2    60.00   |
| Min Buck    2.39E+01   4.37E+02   0.055      Eq. E3-1      2    60.00   |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02           |
| Min Buck    1.49E+01    17.84   4.89E+01   8.99E+02   7.29E+02           |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z    -5.39E-02   1.34E+02   0.000      Eq. G6-1      2    60.00   |
| Local-Y    -6.72E+01   1.34E+02   0.501      Eq. G6-1      1    60.00   |
| Intermediate                                                             |
| Results        Aw        Cv       Kv      h/tw      Vn                  |
| Local-Z     1.49E+01     0.00     0.00     0.00   2.24E+02               |
| Local-Y     1.49E+01     0.00     0.00     0.00   2.24E+02               |
|--------------------------------------------------------------------------|
| CHECK FOR TORSION                                                        |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
```

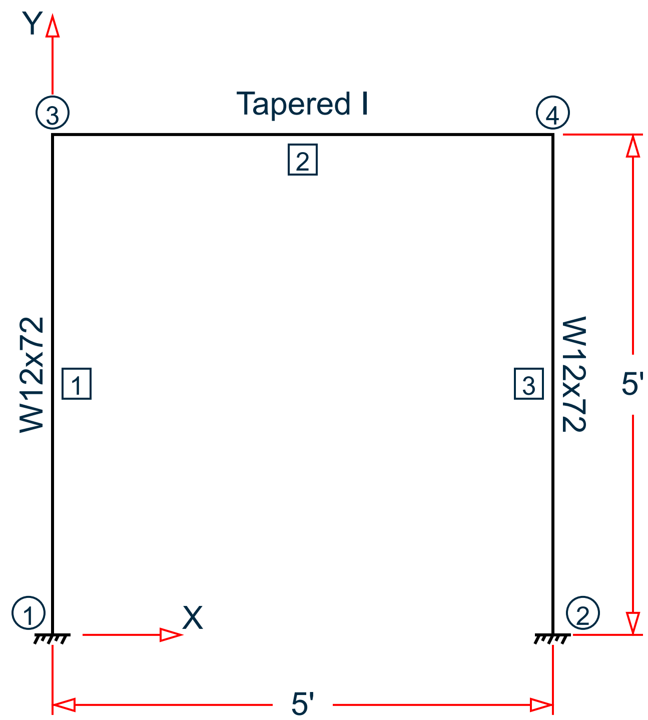
```
|             2.38E+02   1.27E+03   0.187      Eq. H3-1      2    60.00 |
| Intermediate Fcr        Tn                                            |
|             3.00E+01   2.13E+03                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -5.06E+02   1.35E+03   0.374      Eq. F8-1      1    60.00 |
| Minor       1.62E+00   1.35E+03   0.001      Eq. F8-1      2    60.00 |
| Intermediate   Mn        My                                           |
| Major       2.26E+03   0.00E+00                                       |
| Minor       2.26E+03   0.00E+00                                       |
     STAAD SPACE                                            -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-10-ASD)   v1.4a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.386      Eq. H1-1b     1    60.00             |
| Flexure Tens          0.374      Eq. H1-1b     1    60.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         1.35E+03  -5.06E+02   4.37E+02                   |
|                      1.35E+03   0.00E+00   1.02E+01                   |
| Flexure Tens         1.35E+03  -5.06E+02   0.00E+00                   |
|                      1.35E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
   61. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME= 10: 2:27 ****
     STAAD SPACE                                            -- PAGE NO.
12
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

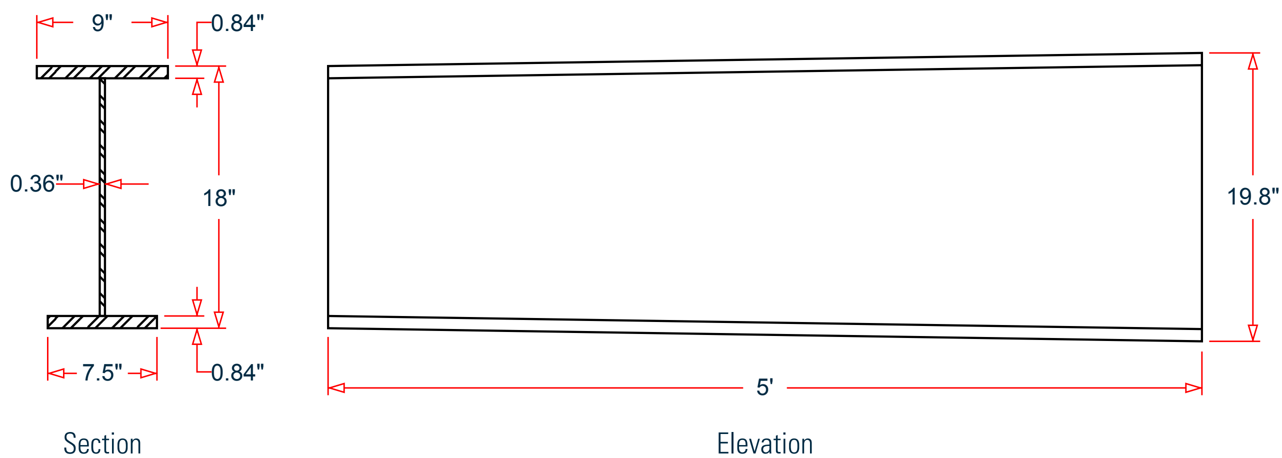
V. AISC 360-05 Tapered I Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered I section member per both the LRFD and ASD methods of the AISC 360-05 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a steel tapered member for the beam.





The following load cases are evaluated:

1. A uniformly distributed load of -3.5 k/ft along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. Lateral loads at the top of the right column of -50 kips in the global X direction and 45 kips in the global Z direction (out of plane).
4. A concreted load of -100 kips in the global Y direction and a concentrated torque of 9 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

By inspection, the left end of the beam (shallowest overall depth) governs under the symmetric loads present. The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 32.26 in·kips, F_y = 8.755 kips
2. M_z = 677.6 in·kips, M_y = 2.942 in·kips, M_x = 5.106 in·kips, F_y = 22.49 kips, F_z = 0.098 kips, F_x = 24.70 kips
3. M_z = 275.9 in·kips, M_x = 4.497 in·kips, F_y = 50.23 kips, F_x = 5.473 kips
4. M_z = 654.5 in·kips, M_y = 5.298 in·kips, M_x = 9.192 in·kips, F_y = 22.50 kips, F_z = 0.177 kips, F_x = 24.35 kips

Section Properties

The section at the left end ($D = 18\ in.$) has the following properties:

Web height, $d_w = 18 - 2 \times 0.84 = 16.32\ in.$
Area, $A_g = (0.84)(9 + 7.5) + 16.32\ (0.36) = 19.74\ in^2$
Shear area in local y, $A_y = 16.32(0.36) = 5.875\ in^2$
Shear area in local x, $A_x = 0.84(9 + 7.5) = 13.86\ in^2$

Dist to centroid y, $c = \dfrac{7.5(0.84)\left(\frac{0.84}{2}\right) + 0.36(16.32)\left(0.84 + \frac{16.32}{2}\right) + 9(0.84)\left(18 - \frac{0.84}{2}\right)}{19.74} = 9.548\ in.$

Moment of inertia about major axis,

$$I_{zz} = \begin{aligned} &\frac{7.5(0.84)^3}{12} + 7.5(0.84)\left(9.548 - \frac{0.84}{2}\right)^2 \\ &+ \frac{0.36(16.32)^3}{12} + 0.36(16.32)\left(9.548 - \frac{16.32}{2} - 0.84\right)^2 \\ &+ \frac{9(0.84)^3}{12} + 9(0.84)\left(18 - \frac{0.84}{2} - 9.548\right)^2 \end{aligned} = 1,147\ \text{in}^4$$

Moment of inertia about minor axis,

$$I_{yy} = \frac{0.84(9)^3}{12} + \frac{16.32(0.36)^3}{12} + \frac{0.84(7.5)^3}{12} = 80.62\ \text{in}^4$$

Elastic section modulus about major axis top flange fiber, $S_{xx,top} = \dfrac{I_{zz}}{D - c} = \dfrac{1,146}{18 - 9.548} = 135.5\ in^3$

Elastic section modulus about major axis bottom flange fiber, $S_{xx,bottom} = \dfrac{I_{zz}}{c} = \dfrac{1,146}{9.548} = 120.0\ in^3$

Elastic section modulus about minor axis, $S_{yy} = \dfrac{I_{yy}}{b_{f1}/2} = \dfrac{80.62}{9/2} = 17.92\ in^3$

Radius of gyration about major axis, $r_x = \sqrt{\dfrac{I_{zz}}{A_g}} = \sqrt{\dfrac{1,147}{19.74}} = 7.619\ in$

Radius of gyration about minor axis, $r_y = \sqrt{\dfrac{I_{yy}}{A_g}} = \sqrt{\dfrac{80.62}{19.74}} = 2.021\ in$

Location of the plastic neutral axis, $y_{PNA} = 0.84 + \frac{19.74 - 2(7.5)(0.84)}{2 \times 0.36} = 10.75\ in.$

Plastic section modulus about major axis,

$$Z_{zt} = \frac{7.5(10.75)^2}{2} - \frac{(7.5 - 0.36)(10.75 - 0.84)^2}{2} + \frac{9(18 - 10.75)^2}{2} - \frac{(9 - 0.36)(18 - 0.84 - 10.75)^2}{2} = 141.8\ \text{in}^3$$

Plastic section modulus about minor axis, $Z_y = \frac{0.84(9)^2}{4} + \frac{16.32(0.36)^2}{4} + \frac{0.84(7.5)^2}{4} = 29.35\ in^3$

Torsional constant, $J = \frac{9(0.84)^3 + 16.32(0.36)^3 + 7.5(0.84)^3}{3} = 3.514\ in.^4$

Section Classification

Flange in compression

$$\lambda = \frac{b}{t} = \frac{1/2 \times 7.5}{0.84} = 4.464$$

$$k_c = \frac{4}{\sqrt{d_w / t_w}} = \frac{4}{\sqrt{\frac{16.32}{0.36}}} = 0.594$$

Per Table B4.1a, $\lambda < \lambda_r = 0.64\sqrt{\frac{k_c E}{F_y}} = 0.64\sqrt{\frac{0.594(29,000)}{50}} = 11.88$, therefore flanges are non-slender for compression.

Web in compression

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table B4.1a, Case 5, $\lambda > \lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29,000}{50}} = 35.88$, therefore the web is slender for compression.

Flange in bending (use half the longer flange width as the outstanding width):

$$\lambda = \frac{b_f}{t_f} = \frac{9}{2(0.84)} = 5.36$$

Per Table B4.1b, Case 11, $\lambda < \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29,000}{50}} = 9.15$, therefore flange is compact for bending.

Web in bending

$$\lambda = \frac{d_w}{t_w} = \frac{16.32}{0.36} = 45.33$$

Per Table B4.1b, Case 15, $\lambda < \lambda_p = 3.76\sqrt{\frac{E}{F_y}} = 3.76\sqrt{\frac{29,000}{50}} = 90.55$, therefore web is compact for bending.

Compression Capacity

About X Axis:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{7.619} = 7.875 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,000}{50}} = 113.4$$

$$F_{ex} = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,000}{(7.875)^2} = 4{,}615 \text{ ksi} \quad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ex}} = \frac{50}{4{,}615} = 0.011 < 2.25$$

$$F_{crx} = 0.658^{F_y/F_{ex}} F_y = 0.658^{(0.011)} 50 = 49.77 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Since the web is slender, use the effective area. The effective width imperfection adjustment factors (Table E7.1):

$c_1 = 0.18$

$c_2 = 1.31$

$$F_{el} = \left(c_2 \frac{\lambda_r}{\lambda}\right)^2 F_y = \left(1.31 \frac{35.88}{45.33}\right)^2 = 53.76 \text{ ksi} \quad \text{(Eq. E7-5)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w\left(1 - c_1\sqrt{\frac{F_{el}}{F_{crx}}}\right)\sqrt{\frac{F_{el}}{F_{crx}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{49.77}}\right)\sqrt{\frac{53.76}{49.77}} = 13.78 \text{ in.} \quad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 13.78)0.36 = 18.84 \text{ in.}^2$$

$$P_{nx} = A_e \times F_{crx} = 18.84 \times 49.77 = 937.4 \text{ kips} \quad \text{(Eq. E7-1)}$$

About Y Axis:

$$\frac{k_y L_y}{r_y} = \frac{1.0(60)}{2.021} = 29.69 < 4.71\sqrt{\frac{E}{F_y}} = 113.4$$

$$F_{ey} = \frac{\pi^2 E}{\left(\frac{k_y L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29,000}{(29.69)^2} = 324.7 \text{ ksi} \quad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_{ey}} = \frac{50}{324.7} = 0.154 < 2.25$$

$$F_{cry} = 0.658^{(F_y/F_{ey})} F_y = 0.658^{(0.154)} 50 = 46.88 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Therefore, the effective depth of the web in compression for the effective area:

$$d_e = d_w\left(1 - c_1\sqrt{\frac{F_{el}}{F_{cry}}}\right)\sqrt{\frac{F_{el}}{F_{cry}}} = 16.32\left(1 - 0.18\sqrt{\frac{53.76}{46.88}}\right)\sqrt{\frac{53.76}{46.88}} = 14.11 \text{ in.} \quad \text{(Eq. E7-3)}$$

The reduced area of the section then becomes:

$$A_e = 19.75 - (16.32 - 14.11)0.36 = 18.93 \text{ in.}^2$$

$P_{ny} = A_e \times F_{cry} = 18.93 \times 46.88 = 887.4$ kips (Eq. E7-1)

The Y axis governs; $P_n = P_{ny}$. The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 887.4 = 798.3\ kips$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 887.4\ /\ 1.67 = 531.4\ kips$

<u>Calculate Bending Capacity</u>

Bending capacity in plastic yielding about Y axis:

$$M_y = F_y \times Z_y = 50 \times 29.35 = 1{,}468 \text{ in·kips} > 1.6F_y \times S_y = 1.6\ (50)\ (17.92) = 1{,}434 \text{ in·kips}$$

Use $M_y = 1{,}434\ in \cdot kips$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}434 = 1{,}290\ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}434\ /\ 1.67 = 858.4\ in \cdot kips$

Bending capacity in plastic yielding about X axis:

The section is capacity is determined as per F4 in the specification. The design moments for this are from Load Case 4 and occur at mid-span (d = 18.9 in.). The section properties at this location are:

$I_{zz} = 1{,}278\ in^4$
$I_{yy} = 80.63\ in^4$
$c = 10.01\ in$
$h_c = 2 \times (10.01 - 0.84) = 18.35\ in.$
$S_x = 143.8\ in^3$
$Z_x = 150.7\ in^3$

$$M_{yc} = F_y \times S_{yc} = 50 \times 143.8 = 7{,}192 \text{ ksi}$$

$$M_p = F_y \times Z_x = 50 \times 150.7 = 7{,}535 \text{ ksi} \leq 1.6\ F_y \times S_x = 11{,}504 \text{ ksi}$$

The moment of inertia of the compression flange about the Y axis $I_{yc} = \frac{0.84(7.5)^3}{12} = 29.53\ in.^4$, and thus $I_{yc} / I_y = {}^{29.53}\!/_{80.63} = 0.366 > 0.23$. $\frac{h_c}{t_w} = \frac{18.35}{0.36} = 50.99 < \lambda_{pw} = 90.55$ (limiting slenderness for a compact web per Table B4.1b case 15), so the web plastification factor, R_{pc} is determined as:

$$R_{pc} = \frac{M_p}{M_{yc}} = \frac{7{,}535}{7{,}192} = 1.048 \qquad \text{(F4-9a)}$$

Bending capacity in plastic yielding about Y axis:

$$M_n = R_{pc} \times M_{yc} = 1.048 \times 7{,}192 = 7{,}535 \text{ in·kips}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 7{,}535 = 6{,}782\ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 7{,}535\ /\ 1.67 = 4{,}512\ in \cdot kips$

<u>Interaction Ratio for Bending and Compression</u>

Check the interaction ratio for Load Case 3:

$$\frac{P_r}{P_c} = \frac{5.473}{798.3} = 0.007 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{5.479}{531.4} = 0.010 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0.$

$$\frac{0.007}{2} + \left(\frac{1{,}225}{6{,}783} + \frac{0}{1{,}290}\right) = 0.184 \text{ (LRFD)}$$

$$\frac{0.010}{2} + \left(\frac{1{,}225}{4{,}513} + \frac{0}{858}\right) = 0.277 \text{ (ASD)}$$

Results

Table 768:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	798.3	799	negligible	
	Major axis bending capacity (in·kips)	6,782	6,780	negligible	
	Minor axis bending capacity (in·kips)	1,290	1,290	none	
	Interaction ratio	0.184	0.184	none	
ASD	Compression capacity (kips)	531.4	532	negligible	
	Major axis bending capacity (in·kips)	4,512	4,510	negligible	
	Minor axis bending capacity (in·kips)	858.4	858	negligible	
	Interaction ratio	0.277	0.277	none	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded tapered member is specified using `STP 2`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 Tapered I Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0; 3 5 0 0; 4 5 5 0;
MEMBER INCIDENCES
1 1 2; 2 2 4; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY
2 TAPERED 1.5 0.03 1.65 0.75 0.07 0.625 0.07
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 3 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -3.5
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 4 LOADTYPE None  TITLE LOAD CASE 4
JOINT LOAD
4 FX -50 FZ 45
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CON GY -100
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
PARAMETER 1
CODE AISC UNIFIED 2005
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
```

```
CODE AISC UNIFIED 2005
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2      TAP ERED                 (AISC SECTIONS)
                           PASS   Sec. G1              0.315         3
                        5.47 C          0.00         275.86        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.927  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.315(PASS)                |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.473E+00(C ) Fy:   5.002E+01     Fz:   0.000E+00               |
| Mx:  -4.497E+00     My:   1.731E-05     Mz:   2.759E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.386E+01     Ayy:   5.875E+00     Cw:  5.508E+03             |
| Szz:   1.200E+02     Syy:   1.792E+01                                 |
| Izz:   1.146E+03     Iyy:   8.062E+01     Ix:  3.514E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.36     N/A     11.88    T.B4.1-4    |
|                Slender        45.33     N/A     35.88    T.B4.1-10   |
| Flexure     :  Compact         5.36     9.15    21.15    T.B4.1-2    |
|                Compact        44.73    94.06   137.27    T.B4.1-11   |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
```

```
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   8.88E+02   0.000      Eq. D2-1      1     0.00  |
| Rupture     0.00E+00   8.88E+02   0.000      Eq. D2-2      1     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.47E+01   8.41E+02   0.029      Sec. E1       2     0.00  |
| Min Buck    2.47E+01   7.99E+02   0.031      Eq. E7-1      2     0.00  |
| Flexural                                                                |
| Tor Buck    2.47E+01   7.93E+02   0.031      Eq. E7-1      2     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    1.88E+01     7.88   4.74E+01   4.62E+03   9.35E+02         |
| Min Buck    1.89E+01    29.69   4.50E+01   3.25E+02   8.88E+02         |
| Flexural       Ag       Fcr         Pn                                  |
| Tor Buck    1.89E+01   4.46E+01   8.81E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     1.77E-01   3.74E+02   0.000      Eq. G2-1      4     0.00  |
| Local-Y     5.00E+01   1.59E+02   0.315      Eq. G2-1      3     0.00  |
| Intermediate                                                            |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     1.39E+01     1.00     1.20    10.71   4.16E+02             |
| Local-Y     5.88E+00     1.00     5.00    45.33   1.76E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       1.22E+03   6.78E+03   0.181      Sec. F4.2     3    30.00  |
| Minor       5.30E+00   1.29E+03   0.004      Eq. F6-1      4     0.00  |
| Intermediate   Mn         My                                            |
| Major       7.54E+03   0.00E+00                                         |
| Minor       1.43E+03   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major      -6.55E+02   6.31E+03   0.104      Eq. F4-2      4     0.00  |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb     |
| Major       7.01E+03   0.00E+00    1.00    54.78   242.74    60.00    |
     STAAD SPACE                                              -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                           |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             1.22E+03   6.78E+03   0.181      Eq. F4-1      3    30.00  |
| Intermediate   Mn         Rpc                                           |
|             7.54E+03   1.05E+00                                        |
|-------------------------------------------------------------------------|
```

```
| CHECK FOR BENDING-TENSION FLANGE YIELDING                             |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|            1.22E+03   6.78E+03   0.181      Eq. F4-14     3    30.00 |
| Intermediate   Mn        Rpt                                        |
|            7.54E+03   1.18E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                      RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp         0.184      Eq. H1-1b     3    30.00             |
| Flexure Tens         0.181      Eq. H1-1b     3    30.00             |
| Intermediate        Mcx /      Mrx /      Pc /                      |
|                     Mcy        Mry        Pr                        |
| Flexure Comp        6.78E+03   1.22E+03   7.95E+02                   |
|                     1.29E+03   0.00E+00   5.47E+00                   |
| Flexure Tens        6.78E+03   1.22E+03   0.00E+00                   |
|                     1.29E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2005
    55. FYLD 50 ALL
    56. FU 60 ALL
    57. METHOD ASD
    58. STP 2 MEMB 2
    59. TRACK 2 MEMB 2
    60. CHECK CODE MEMB 2
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
10
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2      TAP ERED                 (AISC SECTIONS)
                          PASS   Sec. G1              0.474        3
                       5.47 C          0.00        275.86       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     29.927  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.474(PASS)                |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.473E+00(C ) Fy:   5.002E+01     Fz:   0.000E+00              |
| Mx:  -4.497E+00     My:   1.731E-05     Mz:   2.759E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.386E+01     Ayy:   5.875E+00     Cw:  5.508E+03             |
| Szz:   1.200E+02     Syy:   1.792E+01                                |
| Izz:   1.146E+03     Iyy:   8.062E+01     Ix:  3.514E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
```

```
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.36     N/A     11.88    T.B4.1-4     |
|                Slender        45.33     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         5.36     9.15    21.15    T.B4.1-2     |
|                Compact        44.73    94.06   137.27    T.B4.1-11    |
     STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   5.91E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   5.92E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.47E+01   5.60E+02   0.044      Sec. E1       2     0.00 |
| Min Buck    2.47E+01   5.32E+02   0.046      Eq. E7-1      2     0.00 |
| Flexural                                                              |
| Tor Buck    2.47E+01   5.27E+02   0.047      Eq. E7-1      2     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.88E+01     7.88   4.74E+01   4.62E+03   9.35E+02        |
| Min Buck    1.89E+01    29.69   4.50E+01   3.25E+02   8.88E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.89E+01   4.46E+01   8.81E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     1.77E-01   2.49E+02   0.001      Eq. G2-1      4     0.00 |
| Local-Y     5.00E+01   1.06E+02   0.474      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.39E+01     1.00     1.20    10.71   4.16E+02            |
| Local-Y     5.88E+00     1.00     5.00    45.33   1.76E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       1.22E+03   4.51E+03   0.271      Sec. F4.2     3    30.00 |
| Minor       5.30E+00   8.58E+02   0.006      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.54E+03   0.00E+00                                       |
| Minor       1.43E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
```

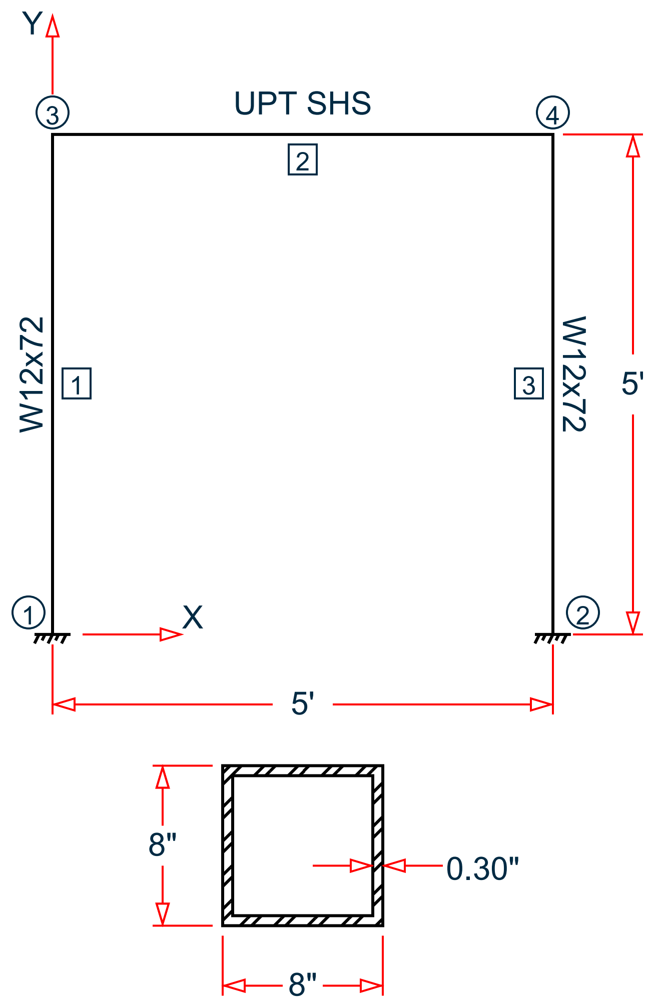
```
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -6.55E+02   4.20E+03   0.156      Eq. F4-2      4     0.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       7.01E+03   0.00E+00    1.00    54.78   242.74    60.00    |
     STAAD SPACE                                            -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-COMPRESSION FLANGE YIELDING                         |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             1.22E+03   4.51E+03   0.271      Eq. F4-1      3    30.00 |
| Intermediate   Mn        Rpc                                          |
|             7.54E+03   1.05E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-TENSION FLANGE YIELDING                             |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
|             1.22E+03   4.51E+03   0.271      Eq. F4-14     3    30.00 |
| Intermediate   Mn        Rpt                                          |
|             7.54E+03   1.18E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.277      Eq. H1-1b     3    30.00           |
| Flexure Tens           0.271      Eq. H1-1b     3    30.00           |
| Intermediate          Mcx /      Mrx /      Pc /                     |
|                       Mcy        Mry        Pr                       |
| Flexure Comp          4.51E+03   1.22E+03   5.29E+02                 |
|                       8.58E+02   0.00E+00   5.47E+00                 |
| Flexure Tens          4.51E+03   1.22E+03   0.00E+00                 |
|                       8.58E+02   0.00E+00   0.00E+00                 |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 UPT Square Hollow Section

Verify the axial compression capacity, shear capacity, flexure capacity, torsional capacity, and interaction ratio of a user-provided hollow square section member per both the LRFD and ASD methods of the AISC 360-05 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT SHS member for the beam. The section outer depth, D, is 8" and the thickness, t, is 0.30".



Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

<u>Design Forces</u>

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner depth, $d = 8 - 2\times 0.30 = 7.40\ in.$

Area, $A_g = D^2 - d^2 = 9.24\ in^2$

Shear area, $A_w = 2\times D\times t_w = 4.80\ in^2$

Moment of inertia, $I = \frac{1}{12}\left(D^4 - d^4\right) = 91.45\ in^4$

Elastic section modulus, $S = \frac{I}{D/2} = \frac{91.45}{8/2} = 22.86\ in^3$

Radius of gyration, $r = \sqrt{\frac{I}{A_g}} = \sqrt{\frac{91.45}{9.24}} = 3.146\ in$

Plastic section modulus, $Z = \frac{1}{6}\left(D^3 - d^3\right) = 26.69\ in^3$

HSS Torsional constant, $C = 2\left(B - t_w\right)\left(D - t_w\right)t_w - 4.5\left(4 - \pi\right)t_w{}^3 = 35.47\ in^3$

Section Classification

The design thickness, $t = 0.93\times t_w = 0.279\ in$

The design depth, $d = D - 3t = 7.163\ in$

Flange in compression

$$\lambda = \frac{d}{t} = \frac{7.163}{0.279} = 25.67$$

Per Table 4.1a Case 6, $\lambda < \lambda_r = 1.40\sqrt{\frac{E}{F_y}} = 1.40\sqrt{\frac{29{,}000}{50}} = 33.72$, therefore flanges are non-slender for compression. Similar for the web of the square section.

Flexure classification of flange:

Per Table B4.1-12, $\lambda < \lambda_p = 1.12\sqrt{\frac{E}{F_y}} = 1.12\sqrt{\frac{29{,}000}{50}} = 26.97$, therefore flange is compact for bending.

Flexure classification of web:

Per Table B4.1-13, $\lambda < \lambda_p = 2.42\sqrt{\frac{E}{F_y}} = 2.42\sqrt{\frac{29{,}000}{50}} = 58.28$, therefore web is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.146} = 19.07 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(19.07)^2} = 787\ \text{ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{787} = 0.064 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y / F_e} F_y = 0.658^{(0.064)} 50 = 48.68 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 9.24 \times 48.68 = 449.9 \text{ kips} \quad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 449.9 = 404.9 \ kips$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 449.9 \ / \ 1.67 = 269.4 \ kips$

Calculate Shear Capacity The nominal shear strength:

$$V_n = 0.6 \times F_y \times A_w \times C_{v2} \quad \text{(Eq. G4-1)}$$

where

C_{v2} = 1.0 per Section G2.2 with $h/t = 25.67$ and $K_v = 5$.

$$V_n = 0.6\ (50)\ (4.80)\ (1.0) = 144 \text{ kips}$$

The shear capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 144 = 129.6 \ kips$
The allowable shear capacity (ASD) = $V_n / \Omega_v = 144 \ / \ 1.67 = 86.23 \ kips$

Calculate Bending Capacity

Bending capacity in plastic yielding (compact section):

$$M_p = F_y \times Z_x = 50 \times 26.69 = 1{,}335 \text{ in} \cdot \text{kips} \quad \text{(Eq F7-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}335 = 1{,}201 \ in \cdot kips$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}335 \ / \ 1.67 = 799.2 \ in \cdot kips$

Torsional Capacity

$$\frac{h}{t} = 25.67 < 2.45\sqrt{\frac{E}{F_y}} = 59.0$$

$$F_{cr} = 0.6 F_y = 30 \text{ ksi}$$

Nominal torsional capacity:

$$T_n = F_{cr} \times C = 1{,}064 \text{ ksi}$$

The torsional capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1{,}064 = 957.7 \ kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 1{,}064 \ / \ 1.67 = 637.2 \ kips$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.80}{404.9} = 0.029 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.80}{269.4} = 0.044 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \le 1.0.$

$$\frac{0.029}{2} + \left(\frac{599.7}{1{,}201} + \frac{0}{1{,}201}\right) = 0.514 \text{ (LRFD)}$$

$$\frac{0.044}{2} + \left(\frac{599.7}{799.2} + \frac{0}{799.2}\right) = 0.765 \text{ (ASD)}$$

Results

Table 769:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	404.9	405	negligible	
	Bending capacity (in·kips)	1,201	1,200	negligible	
	Shear capacity (kips)	129.6	130	negligible	
	Torsional capacity (in·kips)	957.7	958	negligible	
	Interaction ratio	0.514	0.514	negligible	
ASD	Compression capacity (kips)	269.4	269	negligible	
	Bending capacity (in·kips)	799.2	799	negligible	
	Shear capacity (kips)	86.23	86.2	negligible	
	Torsional capacity (in·kips)	637.2	637	negligible	
	Interaction ratio	0.765	0.772	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 UPT Square Hollow Section.STD` is typically installed with the program.

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded SHS member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
SHS
9.24 8 8 0.3 91.4452 91.4452 136.96 4.8 4.8
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X72
3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 SHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
```

```
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2005
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2005
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=    10: 0: 9                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: AISC 360-05 UPT Square Hollow Section.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 13-APR-21
     4. END JOB INFORMATION
     5. INPUT WIDTH 79
     6. UNIT INCHES KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0
     9. MEMBER INCIDENCES
    10. 1 1 2; 2 2 3; 3 3 4
    11. START USER TABLE
    12. TABLE 1
    13. UNIT INCHES KIP
    14. TUBE
    15. SHS
    16. 9.24 8 8 0.3 91.4452 91.4452 136.96 4.8 4.8
    17. END
```

```
    18. DEFINE MATERIAL START
    19. ISOTROPIC STEEL
    20. E 29000
    21. POISSON 0.3
    22. DENSITY 0.000283
    23. ALPHA 6.5E-06
    24. DAMP 0.03
    25. G 11200
    26. TYPE STEEL
    27. STRENGTH RY 1.5 RT 1.2
    28. END DEFINE MATERIAL
    29. MEMBER PROPERTY AMERICAN
    30. 1 TABLE ST W12X72
    31. 3 TABLE ST W12X72
    32. MEMBER PROPERTY
    33. 2 UPTABLE 1 SHS
    34. CONSTANTS
    35. MATERIAL STEEL ALL
    36. SUPPORTS
    37. 1 FIXED
    38. 4 FIXED
      STAAD SPACE                                                     -- PAGE NO.
2
    39. LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
    40. MEMBER LOAD
    41. 2 UNI GY -2.25
    42. LOAD 2 LOADTYPE NONE  TITLE LOAD CASE 2
    43. JOINT LOAD
    44. 2 FX 50 FZ 25
    45. LOAD 3 LOADTYPE NONE  TITLE LOAD CASE 3
    46. MEMBER LOAD
    47. 2 CMOM GX 0.75
    48. PERFORM ANALYSIS
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
     NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =      12
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    49. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      STAAD SPACE                                                     -- PAGE NO.
3
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      2    1    0.00132  -0.00662   0.00000   0.00000   0.00000  -0.00085
           2    0.09809   0.00107   0.28808   0.00654   0.00408  -0.00152
           3    0.00000   0.00000   0.00012   0.00000   0.00000   0.00000
      3    1   -0.00132  -0.00662   0.00000   0.00000   0.00000   0.00085
           2    0.09263  -0.00107   0.04272   0.00141   0.00408  -0.00142
           3    0.00000   0.00000   0.00012   0.00000   0.00000   0.00000
```

```
      4    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
           3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
      STAAD SPACE                                              -- PAGE NO.
4
   SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     11.80     67.50      0.00      0.00      0.00   -108.35
           2    -25.61    -10.88    -24.93  -1364.43     -2.23   1206.26
           3      0.00      0.00      0.00     -0.38      0.00      0.00
      4    1    -11.80     67.50      0.00      0.00      0.00    108.35
           2    -24.39     10.88     -0.07   -135.57     -2.23   1141.12
           3      0.00      0.00      0.00     -0.38      0.00      0.00
      STAAD SPACE                                              -- PAGE NO.
5
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1     67.50    -11.80     0.00      0.00      0.00    -108.35
                 2    -67.50     11.80     0.00      0.00      0.00    -599.73
           2     1    -10.88     25.61   -24.93     -2.23   1364.43    1206.26
                 2     10.88    -25.61    24.93      2.23    131.11     330.63
           3     1      0.00      0.00    -0.00     -0.00      0.38       0.00
                 2      0.00      0.00     0.00      0.00     -0.38       0.00
      2    1     2     11.80     67.50     0.00      0.00      0.00     599.73
                 3    -11.80     67.50     0.00      0.00      0.00    -599.73
           2     2     24.39    -10.88     0.07    131.11     -2.23    -330.63
                 3    -24.39     10.88    -0.07   -131.11     -2.23    -321.98
           3     2      0.00      0.00     0.00     -0.38     -0.00       0.00
                 3      0.00      0.00    -0.00     -0.38      0.00       0.00
      3    1     3     67.50     11.80     0.00      0.00      0.00     599.73
                 4    -67.50    -11.80     0.00      0.00      0.00     108.35
           2     3     10.88     24.39     0.07     -2.23    131.11     321.98
                 4    -10.88    -24.39    -0.07      2.23   -135.57    1141.12
           3     3      0.00      0.00    -0.00     -0.00      0.38       0.00
                 4      0.00      0.00     0.00      0.00     -0.38       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    50. PARAMETER 1
    51. CODE AISC UNIFIED 2005
    52. SLF 0.8 MEMB 2
    53. FYLD 50 ALL
    54. FU 60 ALL
    55. METHOD LRFD
    56. STP 2 MEMB 2
    57. TRACK 2 MEMB 2
    58. CHECK CODE MEMB 2
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   SHS                         (UPT)
```

```
                           PASS   Sec. G1             0.521        1
                       11.80 C         0.00        599.73       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     19.072  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.521(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.180E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.997E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.800E+00     Ayy:   4.800E+00     Cw:  0.000E+00             |
| Szz:   2.286E+01     Syy:   2.286E+01                                |
| Izz:   9.145E+01     Iyy:   9.145E+01     Ix:  1.370E+02             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          60.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        60.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     0.00     N/A      0.00      N/A       |
|                Non-Slender    25.67     N/A     33.72    T.B4.1-12   |
| Flexure     :  Compact        25.67    26.97    33.72    T.B4.1-12   |
|                Compact        25.67    58.28   137.27    T.B4.1-13   |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.16E+02   0.000      Eq. D2-1      1     0.00 |
| Rupture     0.00E+00   3.33E+02   0.000      Eq. D2-2      1     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    2.44E+01   4.05E+02   0.060      Sec. E1       2     0.00 |
| Min Buck    2.44E+01   4.05E+02   0.060      Eq. E3-1      2     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02       |
| Min Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02       |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
```

```
| Local-Z    -7.44E-02   1.30E+02   0.001      Eq. G2-1      2     0.00 |
| Local-Y     6.75E+01   1.30E+02   0.521      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     4.80E+00     1.00      5.00    25.67   1.44E+02           |
| Local-Y     4.80E+00     1.00      5.00    25.67   1.44E+02           |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             1.31E+02   9.58E+02   0.137      Eq. H3-1      2     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.00E+01   1.06E+03                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -6.00E+02   1.20E+03   0.499      Eq. F7-1      1     0.00 |
| Minor      -2.23E+00   1.20E+03   0.002      Eq. F7-1      2     0.00 |
| Intermediate   Mn        My                                           |
| Major       1.33E+03   0.00E+00                                       |
| Minor       1.33E+03   0.00E+00                                       |
     STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.514      Eq. H1-1b     1     0.00            |
| Flexure Tens           0.499      Eq. H1-1b     1     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          1.20E+03  -6.00E+02   4.05E+02                  |
|                       1.20E+03   0.00E+00   1.18E+01                  |
| Flexure Tens          1.20E+03  -6.00E+02   3.33E+02                  |
|                       1.20E+03   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
   59. PARAMETER 2
   60. CODE AISC UNIFIED 2005
   61. SLF 0.8 MEMB 2
   62. FYLD 50 ALL
   63. FU 60 ALL
   64. METHOD ASD
   65. STP 2 MEMB 2
   66. TRACK 2 MEMB 2
   67. CHECK CODE MEMB 2
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      2 ST   SHS                         (UPT)
```

```
                            PASS    Sec. G1               0.783         1
                        11.80 C          0.00         599.73        0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     19.072  L/C  :     1                |
 | Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      1    Ratio     :     0.783(PASS)                 |
 |          Loc  :    0.00   Condition :   Sec. G1                       |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   1.180E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   5.997E+02               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   4.800E+00     Ayy:   4.800E+00     Cw:  0.000E+00              |
 | Szz:   2.286E+01     Syy:   2.286E+01                                 |
 | Izz:   9.145E+01     Iyy:   9.145E+01     Ix:  1.370E+02              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:          50.000   Fu:          60.000                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        60.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     0.00     N/A      0.00      N/A        |
 |                Non-Slender    25.67     N/A     33.72    T.B4.1-12    |
 | Flexure     :  Compact        25.67    26.97    33.72    T.B4.1-12    |
 |                Compact        25.67    58.28   137.27    T.B4.1-13    |
      STAAD SPACE                                              -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
  ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Yield       0.00E+00   2.77E+02   0.000      Eq. D2-1      1     0.00 |
 | Rupture     0.00E+00   2.22E+02   0.000      Eq. D2-2      1     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Maj Buck    2.44E+01   2.69E+02   0.091      Sec. E1       2     0.00 |
 | Min Buck    2.44E+01   2.69E+02   0.091      Eq. E3-1      2     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe          Pn          |
 | Maj Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02        |
 | Min Buck    9.24E+00    19.07   4.87E+01   7.87E+02   4.50E+02        |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
```

```
| Local-Z    -7.44E-02   8.62E+01   0.001      Eq. G2-1      2     0.00 |
| Local-Y     6.75E+01   8.62E+01   0.783      Eq. G2-1      1     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     4.80E+00     1.00     5.00    25.67   1.44E+02            |
| Local-Y     4.80E+00     1.00     5.00    25.67   1.44E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR TORSION                                                     |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
|             1.31E+02   6.37E+02   0.206      Eq. H3-1      2     0.00 |
| Intermediate Fcr        Tn                                            |
|             3.00E+01   1.06E+03                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major      -6.00E+02   7.99E+02   0.750      Eq. F7-1      1     0.00 |
| Minor      -2.23E+00   7.99E+02   0.003      Eq. F7-1      2     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.33E+03   0.00E+00                                       |
| Minor       1.33E+03   0.00E+00                                       |
     STAAD SPACE                                             -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.772      Eq. H1-1b     1     0.00            |
| Flexure Tens           0.750      Eq. H1-1b     1     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          7.99E+02   3.31E+02   2.69E+02                  |
|                       7.99E+02  -2.23E+00   2.44E+01                  |
| Flexure Tens          7.99E+02   3.31E+02   2.22E+02                  |
|                       7.99E+02  -2.23E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flx Tor Comp Shr       0.618      Eq. H3-6      2     0.00            |
| Flx Tor Tens Shr       0.527      Eq. H3-6      2     0.00            |
| Intermediate          Mcx /      Mcy /      Mrx /      Mry /          |
|                       Vcx /      Vcy /      Vrx /      Vry /          |
|                       Tc         Tr         Pc         Pr             |
| Flx Tor Comp Shr      7.99E+02   7.99E+02   3.31E+02  -2.23E+00       |
|                       8.62E+01   8.62E+01  -7.44E-02  -1.09E+01       |
|                       6.37E+02   1.31E+02   2.69E+02   2.44E+01       |
| Flx Tor Tens Shr      7.99E+02   7.99E+02   3.31E+02  -2.23E+00       |
|                       8.62E+01   8.62E+01  -7.44E-02  -1.09E+01       |
|                       6.37E+02   1.31E+02   2.22E+02   0.00E+00       |
|-----------------------------------------------------------------------|
   68. FINISH
     STAAD SPACE                                             -- PAGE NO.
12
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME= 10: 0:10 ****
```

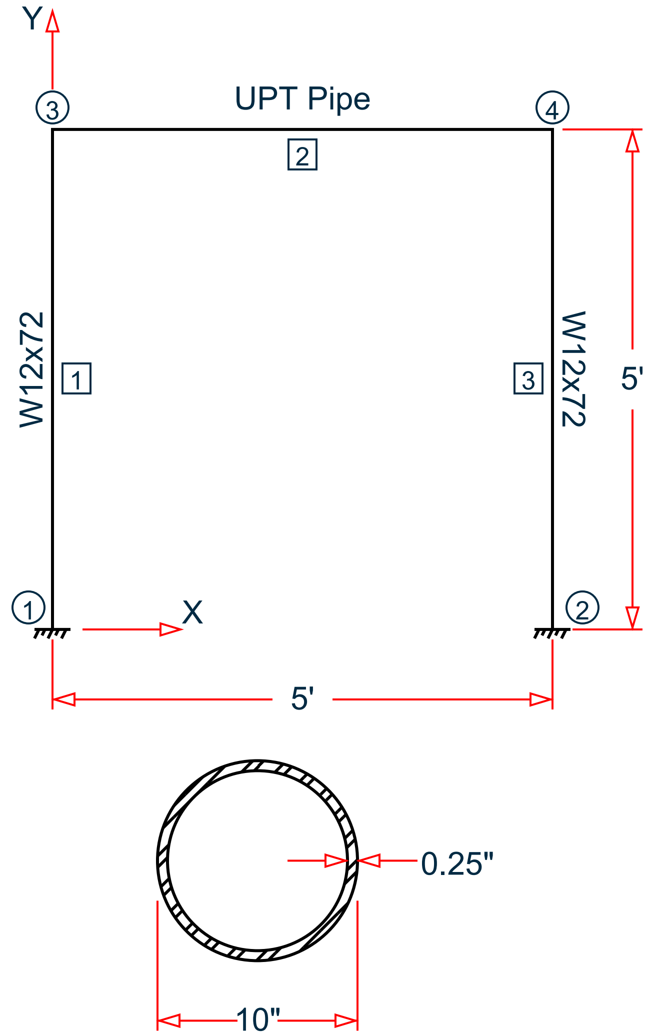
```
          **********************************************************
          *   For technical assistance on STAAD.Pro, please visit    *
          *   http://www.bentley.com/en/support/                     *
          *                                                          *
          *   Details about additional assistance from               *
          *   Bentley and Partners can be found at program menu      *
          *   Help->Technical Support                                *
          *                                                          *
          *           Copyright (c) Bentley Systems, Inc.            *
          *                http://www.bentley.com                    *
          **********************************************************
```

V. AISC 360-05 UPT Pipe Section

Verify the axial compression capacity, flexure capacity, and interaction ratio of a tapered I section member per both the LRFD and ASD methods of the AISC 360-05 code.

Details

A 5' × 5' portal frame consists of W12x72 columns and a UPT pipe member for the beam. The pipe outer diameter, D, is 10" and the thickness, t, is 0.25".



Section

The following load cases are evaluated:

1. A uniformly distributed load of -2.25 k/in along the beam in the global Y direction.
2. Lateral loads at the top of the left column of 50 kips in the global X direction and 25 kips in the global Z direction (out of plane).
3. A concentrated torque of 0.75 in·kips at mid-span of the beam.

Material Properties

E = 29,000 ksi
Fy = 50 ksi

Validation

Design Forces

The following design loads are used from the STAAD.Pro analysis are then used:

1. M_z = 599.6 in·kips, F_y = 67.5 kips

Section Properties

Inner diameter, $d = 10 - 2 \times 0.25 = 9.5\ in.$

Area, $A_g = \frac{\pi}{4}\left(D^2 - d^2\right) = 7.66\ in^2$

Shear area, $A_y = A_x = A_g\ /\ 2 = 3.83\ in^2$

Moment of inertia, $I_x = I_y = \frac{\pi}{64}\left(D^4 - d^4\right) = 91.05\ in^4$

Elastic section modulus, $S_{xx} = \frac{I_{xx}}{D\ /\ 2} = \frac{91.05}{10\ /\ 2} = 18.21\ in^3$

Radius of gyration, $r_x = \sqrt{\frac{I_{xx}}{A_g}} = \sqrt{\frac{91.05}{7.66}} = 3.448\ in$

Plastic section modulus, $Z_x = \frac{1}{6}\left(D^3 - d^3\right) = 23.77\ in^3$

HSS Torsional constant, $C = \frac{\pi}{2}(D - t)^2 t = 37.33\ in^3$

Section Classification

Flange in compression

$$\lambda = \frac{D}{t} = \frac{10}{0.25} = 40$$

Per Table 4.1a Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 63.8$, therefore flanges are non-slender for compression.

Web in compression

Per Table 4.1a, Case 9, $\lambda < \lambda_r = 0.11\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 63.8$, therefore the web is non-slender for compression.

Flexure classification:

Per Table 4.1b, Case 20, $\lambda < \lambda_p = 0.07\frac{E}{F_y} = 0.11\frac{29{,}000}{50} = 40.6$, therefore flange is compact for bending.

Compression Capacity

Effective slenderness ratio:

$$\frac{k_x L_x}{r_x} = \frac{1.0(60)}{3.448} = 17.40 < 4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29,\ 000}{50}} = 113.4$$

Elastic buckling stress:

$$F_e = \frac{\pi^2 E}{\left(\frac{k_x L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29,\ 000}{(17.40)^2} = 945\ \text{ksi} \qquad \text{(Eq. E3-4)}$$

$$\frac{F_y}{F_e} = \frac{50}{945} = 0.053 < 2.25$$

Critical buckling stress:

$$F_{cr} = 0.658^{F_y/F_e} F_y = 0.658^{(0.053)} 50 = 48.91 \text{ ksi} \quad \text{(Eq. E3-2)}$$

Nominal buckling strength:

$$P_n = A_g \times F_{cr} = 7.66 \times 48.91 = 374.5 \text{ kips} \quad \text{(Eq. E3-1)}$$

The axial capacity:

The ultimate compression capacity (LRFD) = $\phi_c P_n = 0.9 \times 374.5 = 337.0 \textit{ kips}$
The allowable compression capacity (ASD) = $P_n / \Omega_c = 374.5 \ / \ 1.67 = 224.3 \textit{ kips}$

Calculate Shear Capacity

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}:

$$F_{cr1} = \frac{1.23E}{\sqrt{\frac{L}{D}} \left(\frac{D}{t}\right)^{\left(5/4\right)}} = 144.8 \text{ ksi}$$

$$F_{cr2} = \frac{0.78E}{\left(\frac{D}{t}\right)^{\left(3/2\right)}} = 89.4 \text{ ksi}$$

$$F_{cr3} = 0.6F_y = 30 \text{ ksi}$$

So, $F_{cr} = 30 \textit{ ksi}$.

The nominal shear strength:

$$V_n = F_{cr} \times A_x = 30 \times 3.83 = 114.9 \text{ kips}$$

The axial capacity:

The ultimate shear capacity (LRFD) = $\phi_v V_n = 0.9 \times 114.9 = 103.4 \textit{ kips}$
The allowable shear capacity (ASD) = $V_n / \Omega_v = 114.9 \ / \ 1.67 = 68.78 \textit{ kips}$

Calculate Bending Capacity

Bending capacity in plastic yielding (compact section):

$$M_y = F_y \times Z_x = 50 \times 23.77 = 1{,}189 \text{ in} \cdot \text{kips} \quad \text{(Eq F8-1)}$$

The bending capacity for flexural yielding about the Y axis:

The ultimate bending capacity (LRFD) = $\phi_b M_y = 0.9 \times 1{,}189 = 1{,}070 \textit{ in} \cdot \textit{kips}$
The allowable bending capacity (ASD) = $M_y / \Omega_b = 1{,}189 \ / \ 1.67 = 711.7 \textit{ in} \cdot \textit{kips}$

Interaction Ratio for Bending and Compression

Check the interaction ratio for Load Case 1:

$$\frac{P_r}{P_c} = \frac{11.75}{337.0} = 0.034 < 0.2 \text{ (LRFD)}$$

$$\frac{P_r}{P_c} = \frac{11.75}{224.3} = 0.052 < 0.2 \text{ (ASD)}$$

So use Eq. H1-1b for both methods: $\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \le 1.0$.

$$\frac{0.034}{2} + \left(\frac{599.6}{1,070} + \frac{0}{1,070}\right) = 0.578 \text{ (LRFD)}$$

$$\frac{0.052}{2} + \left(\frac{599.6}{711.7} + \frac{0}{711.7}\right) = 0.869 \text{ (ASD)}$$

Calculate Torsion Capacity

Critical stress is the minimum of F_{cr1}, F_{cr2}, and F_{cr3}. From shear calculations, $F_{cr} = 30\ ksi$.

The nominal shear strength:

$$T_n = F_{cr} \times C = 30 \times 37.33 = 1,120 \text{ in·kips}$$

The axial capacity:

The ultimate torsional capacity (LRFD) = $\phi_T T_n = 0.9 \times 1,120 = 1,008\ in·kips$
The allowable torsional capacity (ASD) = $T_n / \Omega_T = 114.9\ /\ 1.67 = 670.6\ in·kips$

Results

Table 770:

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Compression capacity (kips)	337	337	none	
	Bending capacity (in·kips)	1,070	1,070	none	
	Shear capacity (kips)	103.4	103	negligible	
	Interaction ratio	0.578	0.578	none	
	Torsional capacity (in·kips)	1,008	1,010	negligible	
ASD	Compression capacity (kips)	224.3	224.3	none	
	Bending capacity (in·kips)	711.7	712	negligible	
	Shear capacity (kips)	68.78	68.8	negligible	
	Interaction ratio	0.869	0.869	none	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Torsional capacity (in·kips)	670.6	671	negligible	

The following D1.A.6 Design Parameters (on page 1100) are used:

- The welded pipe member is specified using `STP 2`
- Shear lag factor, U for tension rupture capacity is specified by `SLF 0.8`

The remaining parameters all use their default values.

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 UPT Pipe Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Apr-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 60 0; 3 60 60 0; 4 60 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
START USER TABLE
TABLE 1
UNIT INCHES KIP
PIPE
PIPE
10 9.5 0 0
END
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W12X72
MEMBER PROPERTY
2 UPTABLE 1 PIPE
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
2 UNI GY -2.25
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 FX 50 FZ 25
LOAD 3 LOADTYPE None  TITLE LOAD CASE 3
MEMBER LOAD
2 CMOM GX 0.75
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
PARAMETER 1
CODE AISC UNIFIED 2005
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD LRFD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
PARAMETER 2
CODE AISC UNIFIED 2005
SLF 0.8 MEMB 2
FYLD 50 ALL
FU 60 ALL
METHOD ASD
STP 2 MEMB 2
TRACK 2 MEMB 2
CHECK CODE MEMB 2
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   PIPE                     (UPT)
                           PASS   Sec. G1             0.653        1
                       11.75 C          0.00         599.60        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.400  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.653(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.175E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.996E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   3.829E+00     Ayy:   3.829E+00     Cw:  0.000E+00              |
| Szz:   1.821E+01     Syy:   1.821E+01                                 |
| Izz:   9.105E+01     Iyy:   9.105E+01     Ix:  1.821E+02              |
```

```
||-------------------------------------------------------------------------|
|| MATERIAL PROPERTIES                                                     |
|| Fyld:          50.000   Fu:          60.000                             |
||-------------------------------------------------------------------------|
|| Actual Member Length:        60.000                                     |
|| Design Parameters                                                       |
|| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00          |
||-------------------------------------------------------------------------|
|| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
||                STIFFENED                                                |
|| Compression :  Non-Slender   40.00     N/A     63.80    T.B4.1-15a      |
||                Non-Slender   40.00     N/A     63.80    T.B4.1-15a      |
|| Flexure     :  Compact       40.00    40.60   179.80    T.B4.1-15b      |
||                Compact       40.00    40.60   179.80    T.B4.1-15b      |
      STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield      0.00E+00   3.45E+02   0.000      Eq. D2-1      1     0.00    |
| Rupture    0.00E+00   2.76E+02   0.000      Eq. D2-2      1     0.00    |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck   2.43E+01   3.37E+02   0.072      Sec. E1       2     0.00    |
| Min Buck   2.43E+01   3.37E+02   0.072      Eq. E3-1      2     0.00    |
| Intermediate                                                            |
| Results    Eff Area     KL/r      Fcr        Fe         Pn              |
| Maj Buck   7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02           |
| Min Buck   7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02           |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z   -6.96E-02   1.03E+02   0.001      Eq. G6-1      2     0.00    |
| Local-Y    6.75E+01   1.03E+02   0.653      Eq. G6-1      1     0.00    |
| Intermediate                                                            |
| Results       Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z    7.66E+00     0.00     0.00     0.00   1.15E+02               |
| Local-Y    7.66E+00     0.00     0.00     0.00   1.15E+02               |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
|            1.56E+02   1.01E+03   0.155      Eq. H3-1      2     0.00    |
| Intermediate Fcr        Tn                                              |
|            3.00E+01   1.12E+03                                          |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major     -6.00E+02   1.07E+03   0.561      Eq. F8-1      1     0.00    |
| Minor     -2.09E+00   1.07E+03   0.002      Eq. F8-1      2     0.00    |
```

```
| Intermediate   Mn        My                                            |
| Major       1.19E+03   0.00E+00                                        |
| Minor       1.19E+03   0.00E+00                                        |
      STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.578      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.561      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         1.07E+03  -6.00E+02   3.37E+02                   |
|                      1.07E+03   0.00E+00   1.17E+01                   |
| Flexure Tens         1.07E+03  -6.00E+02   2.76E+02                   |
|                      1.07E+03   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2005
    59. SLF 0.8 MEMB 2
    60. FYLD 50 ALL
    61. FU 60 ALL
    62. METHOD ASD
    63. STP 2 MEMB 2
    64. TRACK 2 MEMB 2
    65. CHECK CODE MEMB 2
 STEEL DESIGN
   WARNING: For member#      2 the profile has been selected from HSS Rect/
Round (non A1085)/Pipe
            tables of AISC database. Weld type is considered to be Elect-
resist-weld.
            Thickness is already reduced from the table. Further reductions
will not be done.
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST   PIPE                    (UPT)
                           PASS   Sec. G1             0.981        1
                       11.75 C          0.00        599.60        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     17.400  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.981(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. G1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.175E+01(C ) Fy:   6.750E+01     Fz:   0.000E+00                |
```

```
| Mx:   0.000E+00     My:   0.000E+00     Mz:   5.996E+02                  |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   3.829E+00     Ayy:   3.829E+00     Cw:  0.000E+00                 |
| Szz:   1.821E+01     Syy:   1.821E+01                                    |
| Izz:   9.105E+01     Iyy:   9.105E+01     Ix:  1.821E+02                 |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:          50.000   Fu:          60.000                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        60.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.80 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender    40.00     N/A     63.80    T.B4.1-15a    |
|                Non-Slender    40.00     N/A     63.80    T.B4.1-15a    |
| Flexure     :  Compact        40.00    40.60   179.80    T.B4.1-15b    |
|                Compact        40.00    40.60   179.80    T.B4.1-15b    |
     STAAD SPACE                                              -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   2.29E+02   0.000      Eq. D2-1      1     0.00  |
| Rupture     0.00E+00   1.84E+02   0.000      Eq. D2-2      1     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    2.43E+01   2.24E+02   0.108      Sec. E1       2     0.00  |
| Min Buck    2.43E+01   2.24E+02   0.108      Eq. E3-1      2     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02         |
| Min Buck    7.66E+00    17.40   4.89E+01   9.45E+02   3.74E+02         |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    -6.96E-02   6.88E+01   0.001      Eq. G6-1      2     0.00  |
| Local-Y     6.75E+01   6.88E+01   0.981      Eq. G6-1      1     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     7.66E+00     0.00     0.00     0.00   1.15E+02             |
| Local-Y     7.66E+00     0.00     0.00     0.00   1.15E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR TORSION                                                       |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
|             1.56E+02   6.71E+02   0.233      Eq. H3-1      2     0.00  |
| Intermediate Fcr        Tn                                              |
|             3.00E+01   1.12E+03                                        |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|             FORCE      CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -6.00E+02   7.12E+02   0.843      Eq. F8-1      1     0.00 |
| Minor      -2.09E+00   7.12E+02   0.003      Eq. F8-1      2     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.19E+03   0.00E+00                                       |
| Minor       1.19E+03   0.00E+00                                       |
     STAAD SPACE                                             -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.869      Eq. H1-1b     1     0.00             |
| Flexure Tens          0.843      Eq. H1-1b     1     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                       |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         7.12E+02   3.58E+02   2.24E+02                   |
|                      7.12E+02  -2.09E+00   2.43E+01                   |
| Flexure Tens         7.12E+02   3.58E+02   1.84E+02                   |
|                      7.12E+02  -2.09E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP TORSION SHEAR INTERACTION                 |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flx Tor Comp Shr      0.775      Eq. H3-6      2     0.00             |
| Flx Tor Tens Shr      0.667      Eq. H3-6      2     0.00             |
| Intermediate         Mcx /      Mcy /      Mrx /      Mry /           |
|                      Vcx /      Vcy /      Vrx /      Vry /           |
|                      Tc         Tr         Pc         Pr              |
| Flx Tor Comp Shr     7.12E+02   7.12E+02   3.58E+02  -2.09E+00        |
|                      6.88E+01   6.88E+01  -6.96E-02  -1.18E+01        |
|                      6.71E+02   1.56E+02   2.24E+02   2.43E+01        |
| Flx Tor Tens Shr     7.12E+02   7.12E+02   3.58E+02  -2.09E+00        |
|                      6.88E+01   6.88E+01  -6.96E-02  -1.18E+01        |
|                      6.71E+02   1.56E+02   1.84E+02   0.00E+00        |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 W Comp and Bending H.1B

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example H.1b, p.H-3

Details

From reference (2):

> Verify if an ASTM A992 W14×99 has sufficient available strength to support the axial forces and moments listed below, obtained from a

second order analysis that includes second-order effects. The unbraced length is 14 ft and the member has pinned ends. KLx = KLy = Lb = 14.0 ft

LRFD	**ASD**
$P_u = 400$ *kips*	$P_a = 267$ *kips*
$M_{ux} = 250$ *kip-ft*	$M_{ax} = 167$ *kip-ft*
$M_{uy} = 80.0$ *kip-ft*	$M_{ay} = 53.3$ *kip-ft*

Validation

Refer to reference.

Table 771: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Interaction ratio [LRFD]	0.928	0.929	negligible	
Interaction ratio [ASD]	0.932	0.932	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Comp and Bending H.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 19-Nov-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 14 0; 2 0 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 5184 FU 8352 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST W14X99
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
1 FY -400 MX 80 MZ 250
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
1 FY -267 MX 53.3 MZ 167
PERFORM ANALYSIS
LOAD LIST 1
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 2
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
LZ 14 ALL
LY 14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X99                    (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.929        1
                     400.00 C         80.00        250.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     45.200  L/C  :     1                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      1    Ratio     :     0.929(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   4.000E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   8.000E+01     Mz:   2.500E+02              |
```

```
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   2.278E+01     Ayy:   6.887E+00     Cw:  1.810E+04                |
| Szz:   1.563E+02     Syy:   5.507E+01                                   |
| Izz:   1.110E+03     Iyy:   4.020E+02     Ix:  5.370E+00                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        14.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00          |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE         |
|                STIFFENED                                                |
| Compression :  Non-Slender     9.36     N/A     13.49    T.B4.1-3       |
|                Non-Slender    23.59     N/A     35.88    T.B4.1-10      |
| Flexure     :  Non-Compact     9.36     9.15    24.08    T.B4.1-1       |
|                Compact        23.59    90.55   137.27    T.B4.1-9       |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.31E+03   0.000      Eq. D2-1      1     0.00  |
| Rupture     0.00E+00   1.42E+03   0.000      Eq. D2-2      1     0.00  |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    4.00E+02   1.24E+03   0.322      Sec. E1       1     0.00  |
| Min Buck    4.00E+02   1.13E+03   0.355      Eq. E3-1      1     0.00  |
| Flexural                                                                |
| Tor Buck    4.00E+02   1.15E+03   0.348      Eq. E4-1      1     0.00  |
| Intermediate                                                            |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    2.02E-01    27.20   6.82E+03   5.57E+04   1.38E+03         |
| Min Buck    2.02E-01    45.20   6.20E+03   2.02E+04   1.25E+03         |
| Flexural       Ag       Fcr        Pn                                  |
| Tor Buck    2.02E-01   6.32E+03   1.28E+03                             |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   6.15E+02   0.000      Eq. G2-1      1     0.00  |
| Local-Y     0.00E+00   2.07E+02   0.000      Eq. G2-1      1     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                 |
| Local-Z     1.58E-01     1.00     1.20     9.36   6.83E+02             |
| Local-Y     4.78E-02     1.00     0.00    23.59   2.07E+02             |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Major      -2.50E+02   6.49E+02   0.385      Eq. F2-1      1     0.00 |
| Minor       8.00E+01   3.14E+02   0.255      Eq. F6-1      1     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.21E+02   0.00E+00                                       |
| Minor       3.48E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.50E+02   6.42E+02   0.389      Eq. F2-2      1     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       7.14E+02   0.00E+00    1.00    13.13    45.47    14.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -2.50E+02   6.45E+02   0.387      Eq. F3-1      1     0.00 |
| Minor       8.00E+01   3.11E+02   0.257      Eq. F6-2      1     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       7.17E+02   0.00E+00                                       |
| Minor       3.46E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.929      Eq. H1-1a     1     0.00            |
| Flexure Tens           0.646      Eq. H1-1b     1     0.00            |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         6.42E+02  -2.50E+02   1.13E+03                   |
|                      3.11E+02   8.00E+01   4.00E+02                   |
| Flexure Tens         6.42E+02  -2.50E+02   1.31E+03                   |
|                      3.11E+02   8.00E+01   0.00E+00                   |
|-----------------------------------------------------------------------|
    44. LOAD LIST 2
    45. PARAMETER 2
    46. CODE AISC UNIFIED 2005
    47. METHOD ASD
    48. FYLD 7200 ALL
    49. FU 9360 ALL
    50. LZ 14 ALL
    51. LY 14 ALL
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W14X99                   (AISC SECTIONS)
```

```
                            PASS   Eq. H1-1a            0.932        2
                       267.00 C         53.30        167.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     45.200  L/C  :     2                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      2    Ratio     :     0.932(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.670E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   5.330E+01     Mz:   1.670E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.278E+01     Ayy:   6.887E+00     Cw:  1.810E+04             |
| Szz:   1.563E+02     Syy:   5.507E+01                                 |
| Izz:   1.110E+03     Iyy:   4.020E+02     Ix:  5.370E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        14.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.36     N/A     13.49    T.B4.1-3     |
|                Non-Slender    23.59     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact     9.36     9.15    24.08    T.B4.1-1     |
|                Compact        23.59    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   8.71E+02   0.000      Eq. D2-1      2     0.00 |
| Rupture     0.00E+00   9.46E+02   0.000      Eq. D2-2      2     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    2.67E+02   8.25E+02   0.323      Sec. E1       2     0.00 |
| Min Buck    2.67E+02   7.50E+02   0.356      Eq. E3-1      2     0.00 |
| Flexural                                                              |
| Tor Buck    2.67E+02   7.65E+02   0.349      Eq. E4-1      2     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r       Fcr        Fe         Pn          |
| Maj Buck    2.02E-01    27.20   6.82E+03   5.57E+04   1.38E+03        |
| Min Buck    2.02E-01    45.20   6.20E+03   2.02E+04   1.25E+03        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    2.02E-01   6.32E+03   1.28E+03                            |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   4.09E+02   0.000      Eq. G2-1      2     0.00 |
| Local-Y     0.00E+00   1.38E+02   0.000      Eq. G2-1      2     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.58E-01     1.00     1.20     9.36   6.83E+02            |
| Local-Y     4.78E-02     1.00     0.00    23.59   2.07E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -1.67E+02   4.32E+02   0.387      Eq. F2-1      2     0.00 |
| Minor       5.33E+01   2.09E+02   0.256      Eq. F6-1      2     0.00 |
| Intermediate   Mn         My                                          |
| Major       7.21E+02   0.00E+00                                       |
| Minor       3.48E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -1.67E+02   4.27E+02   0.391      Eq. F2-2      2     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major       7.14E+02   0.00E+00    1.00    13.13    45.47    14.00    |
      STAAD SPACE                                            -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major      -1.67E+02   4.29E+02   0.389      Eq. F3-1      2     0.00 |
| Minor       5.33E+01   2.07E+02   0.257      Eq. F6-2      2     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       7.17E+02   0.00E+00                                       |
| Minor       3.46E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.932      Eq. H1-1a     2     0.00             |
| Flexure Tens          0.648      Eq. H1-1b     2     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         4.27E+02  -1.67E+02   7.50E+02                   |
|                      2.07E+02   5.33E+01   2.67E+02                   |
| Flexure Tens         4.27E+02  -1.67E+02   8.71E+02                   |
|                      2.07E+02   5.33E+01   0.00E+00                   |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 W D.1

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.1, pp.D-2,3

Details

From reference (2):

> Select an 8 in. W-shape, ASTM A992, to carry a dead load of 30 kips and a live load of 90 kips in tension. The member is 25 ft long. Verify the member strength by both LRFD and ASD with the bolted end connection shown. Verify that the member satisfies the recommended slenderness limit.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	277	277	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	211	211	none	
Tensile yielding P_n/Ω_t (kips) [ASD]	184	184	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	141	141	none	
Slenderness ratio	238	238.212	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X21
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 30
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 90
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
METHOD LRFD
TMAIN 300 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
METHOD ASD
TMAIN 300 ALL
NSF 0.773 ALL
SLF 0.908 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W8X21                   (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854         3
                      180.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    238.212  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.854(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.800E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02             |
| Szz:   1.819E+01     Syy:   3.708E+00                                |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                          |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1-3     |
|                Non-Slender    27.52     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1-1     |
|                Compact        27.52    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       1.80E+02   2.77E+02   0.649      Eq. D2-1      3     0.00 |
| Rupture     1.80E+02   2.11E+02   0.854      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   1.62E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   2.45E+01   0.000      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.70E+02   0.000      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe          Pn          |
```

```
| Maj Buck    4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02         |
| Min Buck    4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01         |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    4.28E-02   4.41E+03   1.89E+02                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.14E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   6.21E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.93E-02     1.00     1.20     6.59   1.26E+02            |
| Local-Y     1.44E-02     1.00     0.00    27.52   6.21E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   7.65E+01   0.000      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   2.13E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       8.50E+01   0.00E+00                                       |
| Minor       2.37E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   2.52E+01   0.000      Eq. F2-3      3     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       2.80E+01   0.00E+00    1.00     4.45    14.75    25.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                     STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H1-1b     3     0.00             |
| Flexure Tens          0.854      Eq. H1-1a     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         2.52E+01   0.00E+00   2.45E+01                   |
|                      2.13E+01   0.00E+00   0.00E+00                   |
| Flexure Tens         2.52E+01   0.00E+00   2.11E+02                   |
|                      2.13E+01   0.00E+00   1.80E+02                   |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. FU 9360 ALL
    54. FYLD 7200 ALL
    55. METHOD ASD
    56. TMAIN 300 ALL
    57. NSF 0.773 ALL
    58. SLF 0.908 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
```

```
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W8X21                   (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.854         4
                      120.00 T         0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    238.212  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.854(PASS)                |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.200E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.216E+00     Ayy:   2.070E+00     Cw:  1.517E+02              |
| Szz:   1.819E+01     Syy:   3.708E+00                                 |
| Izz:   7.530E+01     Iyy:   9.770E+00     Ix:  2.820E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        25.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    0.77 SLF:    0.91 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.59     N/A     13.49    T.B4.1-3     |
|                Non-Slender    27.52     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.59     9.15    24.08    T.B4.1-1     |
|                Compact        27.52    90.55   137.27    T.B4.1-9     |
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       1.20E+02   1.84E+02   0.651      Eq. D2-1      4     0.00 |
| Rupture     1.20E+02   1.41E+02   0.854      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
```

```
| Maj Buck    0.00E+00   1.08E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   1.63E+01   0.000      Eq. E3-1      4     0.00 |
| Flexural                                                             |
| Tor Buck    0.00E+00   1.13E+02   0.000      Eq. E4-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.28E-02    85.81   4.20E+03   5.60E+03   1.80E+02       |
| Min Buck    4.28E-02   238.21   6.37E+02   7.26E+02   2.72E+01       |
| Flexural       Ag        Fcr        Pn                               |
| Tor Buck    4.28E-02   4.41E+03   1.89E+02                           |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   7.57E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   4.14E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv        Kv      h/tw      Vn             |
| Local-Z     2.93E-02     1.00      1.20     6.59   1.26E+02          |
| Local-Y     1.44E-02     1.00      0.00    27.52   6.21E+01          |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.09E+01   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   1.42E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       8.50E+01   0.00E+00                                      |
| Minor       2.37E+01   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.68E+01   0.000      Eq. F2-3      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb   |
| Major       2.80E+01   0.00E+00    1.00     4.45    14.75    25.00   |
    STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                      RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp         0.000      Eq. H1-1b     4     0.00             |
| Flexure Tens         0.854      Eq. H1-1a     4     0.00             |
| Intermediate        Mcx /      Mrx /      Pc /                      |
|                     Mcy        Mry        Pr                        |
| Flexure Comp        1.68E+01   0.00E+00   1.63E+01                   |
|                     1.42E+01   0.00E+00   0.00E+00                   |
| Flexure Tens        1.68E+01   0.00E+00   1.41E+02                   |
|                     1.42E+01   0.00E+00   1.20E+02                   |
|----------------------------------------------------------------------|
```

V. AISC 360-05 W E.1C

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.1c, p.E-8

Details

From reference (2):

> Calculate the available strength of a W14×132 column with unbraced lengths of 30 ft in both axes. The material properties and loads are as given in Example E.1a.

From Example E.1a (also from Reference (1), p.E-4):

> ...an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	892	893	negligible	
P_n / Ω_c (kips) [ASD]	598	594	negligible	
Slenderness ratio	95.7	95.792	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
```

```
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
KX 1 ALL
KY 1 ALL
METHOD LRFD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FYLD 7200 ALL
FU 9360 ALL
KX 1 ALL
KY 1 ALL
METHOD ASD
PROFILE W14 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   W14X132                  (AISC SECTIONS)
                         PASS   Eq. H1-1a           0.941         3
                    840.00 C          0.00          0.00        0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     95.792  L/C  :     3                |
 | Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.941(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H1-1a                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   8.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04              |
 | Szz:   2.082E+02     Syy:   7.456E+01                                 |
 | Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        30.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1-3     |
 |                Non-Slender    17.74     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         7.14     9.15    24.08    T.B4.1-1     |
 |                Compact        17.74    90.55   137.27    T.B4.1-9     |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield       0.00E+00   1.75E+03   0.000      Eq. D2-1      3     0.00 |
 | Rupture     0.00E+00   1.89E+03   0.000      Eq. D2-2      3     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck    8.40E+02   1.37E+03   0.612      Sec. E1       3     0.00 |
 | Min Buck    8.40E+02   8.93E+02   0.941      Eq. E3-1      3     0.00 |
 | Flexural                                                              |
 | Tor Buck    8.40E+02   1.39E+03   0.602      Eq. E4-1      3     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area    KL/r      Fcr        Fe          Pn           |
 | Maj Buck    2.69E-01    57.33   5.66E+03   1.25E+04   1.53E+03        |
 | Min Buck    2.69E-01    95.79   3.68E+03   4.49E+03   9.92E+02        |
 | Flexural       Ag       Fcr        Pn                                 |
```

```
| Tor Buck    2.69E-01   5.75E+03   1.55E+03                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   8.18E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.84E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     2.10E-01     1.00     1.20     7.14   9.08E+02            |
| Local-Y     6.58E-02     1.00     0.00    17.74   2.84E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   8.78E+02   0.000      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   4.24E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       9.75E+02   0.00E+00                                       |
| Minor       4.71E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   7.48E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb     |
| Major       8.31E+02   0.00E+00    1.00    13.27    56.15    30.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.941      Eq. H1-1a     3     0.00            |
| Flexure Tens           0.000      Eq. H1-1b     3     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          7.48E+02   0.00E+00   8.93E+02                  |
|                       4.24E+02   0.00E+00   8.40E+02                  |
| Flexure Tens          7.48E+02   0.00E+00   1.75E+03                  |
|                       4.24E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2005
    55. FYLD 7200 ALL
    56. FU 9360 ALL
    57. KX 1 ALL
    58. KY 1 ALL
    59. METHOD ASD
    60. PROFILE W14 ALL
    61. TRACK 2 ALL
    62. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X132                  (AISC SECTIONS)
                           PASS   Eq. H1-1a           0.943         4
                      560.00 C         0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     95.792  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.943(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   5.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   3.028E+01     Ayy:   9.481E+00     Cw:  2.560E+04              |
| Szz:   2.082E+02     Syy:   7.456E+01                                 |
| Izz:   1.530E+03     Iyy:   5.480E+02     Ix:  1.230E+01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     7.14     N/A     13.49    T.B4.1-3     |
|                Non-Slender    17.74     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         7.14     9.15    24.08    T.B4.1-1     |
|                Compact        17.74    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                               -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.16E+03   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   1.26E+03   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    5.60E+02   9.14E+02   0.613      Sec. E1       4     0.00 |
| Min Buck    5.60E+02   5.94E+02   0.943      Eq. E3-1      4     0.00 |
| Flexural                                                              |
```

```
| Tor Buck    5.60E+02   9.28E+02   0.603      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    2.69E-01    57.33   5.66E+03   1.25E+04   1.53E+03        |
| Min Buck    2.69E-01    95.79   3.68E+03   4.49E+03   9.92E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    2.69E-01   5.75E+03   1.55E+03                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   5.44E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.90E+02   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.10E-01     1.00     1.20     7.14   9.08E+02            |
| Local-Y     6.58E-02     1.00     0.00    17.74   2.84E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   5.84E+02   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   2.82E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       9.75E+02   0.00E+00                                       |
| Minor       4.71E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   4.98E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       8.31E+02   0.00E+00    1.00    13.27    56.15    30.00    |
     STAAD SPACE                                             -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.943      Eq. H1-1a     4     0.00            |
| Flexure Tens           0.000      Eq. H1-1b     4     0.00            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          4.98E+02   0.00E+00   5.94E+02                  |
|                       2.82E+02   0.00E+00   5.60E+02                  |
| Flexure Tens          4.98E+02   0.00E+00   1.16E+03                  |
|                       2.82E+02   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 W E.1D

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.1d, p.E-9

Details

From reference (2):

> Calculate the available strength of a W14×90 with a strong axis unbraced length of 30 ft and weak axis and torsional unbraced lengths of 15 ft. The material properties and loads are as given in Example E.1b. [which then references Example E.1a]

From Example E.1a (also from Reference (1), p.E-4):

> ...an ASTM A992 (Fy = 50 ksi) W-shape column to carry an axial dead load of 140 kips and live load of 420 kips.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	927	927	none	
P_n/Ω_c (kips) [ASD]	617	617	none	
Slenderness ratio	58.6	58.63	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
```

```
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -140
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -420
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
KX 0.5 ALL
KY 0.5 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FYLD 7200 ALL
FU 9360 ALL
KX 0.5 ALL
KY 0.5 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X90                   (AISC SECTIONS)
```

```
                          PASS   Eq. H1-1a           0.906        3
                     840.00 C          0.00          0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     58.633  L/C  :     3                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.906(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1a                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.400E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   2.059E+01     Ayy:   6.160E+00     Cw:  1.598E+04              |
| Szz:   1.427E+02     Syy:   4.993E+01                                 |
| Izz:   9.990E+02     Iyy:   3.620E+02     Ix:  4.060E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    0.50 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    10.21     N/A     13.49    T.B4.1-3     |
|                Non-Slender    25.86     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact    10.21     9.15    24.08    T.B4.1-1     |
|                Compact        25.86    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   1.19E+03   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   1.29E+03   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    8.40E+02   9.27E+02   0.906      Sec. E1       3     0.00 |
| Min Buck    8.40E+02   1.00E+03   0.838      Eq. E3-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    8.40E+02   1.02E+03   0.821      Eq. E4-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.84E-01    58.63   5.60E+03   1.20E+04   1.03E+03        |
| Min Buck    1.84E-01    48.70   6.05E+03   1.74E+04   1.11E+03        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.84E-01   6.18E+03   1.14E+03                            |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   5.56E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   1.85E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     1.43E-01     1.00     1.20    10.21   6.18E+02            |
| Local-Y     4.28E-02     1.00     0.00    25.86   1.85E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.89E+02   0.000      Eq. F2-1      3     0.00 |
| Minor       0.00E+00   2.84E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                           |
| Major       6.54E+02   0.00E+00                                       |
| Minor       3.15E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.66E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb     |
| Major       5.18E+02   0.00E+00    1.00    13.05    42.60    30.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   5.74E+02   0.000      Eq. F3-1      3     0.00 |
| Minor       0.00E+00   2.73E+02   0.000      Eq. F6-2      3     0.00 |
| Intermediate   Mn        Fcr                                          |
| Major       6.37E+02   0.00E+00                                       |
| Minor       3.03E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.906      Eq. H1-1a     3     0.00             |
| Flexure Tens          0.000      Eq. H1-1b     3     0.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         4.66E+02   0.00E+00   9.27E+02                   |
|                      2.73E+02   0.00E+00   8.40E+02                   |
| Flexure Tens         4.66E+02   0.00E+00   1.19E+03                   |
|                      2.73E+02   0.00E+00   0.00E+00                   |
|-----------------------------------------------------------------------|
    51. LOAD LIST 4
    52. PARAMETER 2
    53. CODE AISC UNIFIED 2005
    54. FYLD 7200 ALL
    55. FU 9360 ALL
    56. KX 0.5 ALL
```

```
   57. KY 0.5 ALL
   58. METHOD ASD
   59. TRACK 2 ALL
   60. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                     STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W14X90                   (AISC SECTIONS)
                           PASS   Eq. H1-1a            0.908         4
                      560.00 C          0.00           0.00       0.00
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :     58.633  L/C  :     4                |
 | Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.908(PASS)                 |
 |          Loc  :    0.00   Condition :   Eq. H1-1a                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   5.600E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   2.059E+01     Ayy:   6.160E+00     Cw:  1.598E+04              |
 | Szz:   1.427E+02     Syy:   4.993E+01                                 |
 | Izz:   9.990E+02     Iyy:   3.620E+02     Ix:  4.060E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        30.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    0.50 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender    10.21     N/A     13.49    T.B4.1-3     |
 |                Non-Slender    25.86     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Non-Compact    10.21     9.15    24.08    T.B4.1-1     |
 |                Compact        25.86    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
10
                     STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                     ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Yield       0.00E+00   7.93E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   8.61E+02   0.000      Eq. D2-2      4     0.00 |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    5.60E+02   6.17E+02   0.908      Sec. E1       4     0.00 |
| Min Buck    5.60E+02   6.67E+02   0.839      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    5.60E+02   6.81E+02   0.822      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.84E-01    58.63   5.60E+03   1.20E+04   1.03E+03        |
| Min Buck    1.84E-01    48.70   6.05E+03   1.74E+04   1.11E+03        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    1.84E-01   6.18E+03   1.14E+03                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   3.70E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.23E+02   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.43E-01     1.00     1.20    10.21   6.18E+02            |
| Local-Y     4.28E-02     1.00     0.00    25.86   1.85E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.92E+02   0.000      Eq. F2-1      4     0.00 |
| Minor       0.00E+00   1.89E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       6.54E+02   0.00E+00                                       |
| Minor       3.15E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.10E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb     |
| Major       5.18E+02   0.00E+00    1.00    13.05    42.60    30.00    |
    STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.82E+02   0.000      Eq. F3-1      4     0.00 |
| Minor       0.00E+00   1.81E+02   0.000      Eq. F6-2      4     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       6.37E+02   0.00E+00                                       |
| Minor       3.03E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.908      Eq. H1-1a     4     0.00           |
```

```
| Flexure Tens          0.000      Eq. H1-1b     4     0.00              |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         3.10E+02   0.00E+00   6.17E+02                  |
|                      1.81E+02   0.00E+00   5.60E+02                  |
| Flexure Tens         3.10E+02   0.00E+00   7.93E+02                  |
|                      1.81E+02   0.00E+00   0.00E+00                  |
|----------------------------------------------------------------------|
```

V. AISC 360-05 W Flex Memb F.1-1

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.1-1, pp.F-6,-7

Details

From reference (2):

> Select an ASTM A992 W-shape beam with a simple span of 35 feet. Limit the member to a maximum nominal depth of 18 in. Limit the live load deflection to L/360. The nominal loads are a uniform dead load of 0.45 kip/ft and a uniform live load of 0.75 kip/ft. Assume the beam is continuously braced.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	379	379	none	
M_n/Ω_b (ft·k) [ASD]	252	252	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Flex Memb F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FU 9360 ALL
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FU 9360 ALL
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.213         3
                        0.00            0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     3.213(FAIL)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
| Szz:   8.889E+01     Syy:   1.069E+01                                 |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3     |
|                Slender        45.23     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1     |
|                Compact        45.23    90.55   137.27    T.B4.1-9     |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   7.17E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   5.09E+02   0.000      Sec. E1       3     0.00 |
```

```
| Min Buck    0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                            |
| Tor Buck    0.00E+00   2.59E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                        |
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02      |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01      |
| Flexural       Ag        Fcr        Pn                              |
| Tor Buck    1.02E-01   2.82E+03   2.88E+02                          |
|---------------------------------------------------------------------|
| CHECK FOR SHEAR                                                     |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00 |
| Intermediate                                                        |
| Results        Aw         Cv        Kv      h/tw      Vn            |
| Local-Z     5.94E-02     1.00      1.20      6.58   2.56E+02         |
| Local-Y     4.44E-02     1.00      0.00     45.23   1.92E+02         |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   3.79E+02   0.703      Eq. F2-1      3    17.50 |
| Minor       0.00E+00   6.23E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                        |
| Major       4.21E+02   0.00E+00                                     |
| Minor       6.92E+01   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                        |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   8.29E+01   3.213      Eq. F2-3      3    17.50 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       9.21E+01   0.00E+00    1.00     5.83    16.97    35.00   |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                        RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp           3.213      Eq. H1-1b     3    17.50          |
| Flexure Tens           3.213      Eq. H1-1b     3    17.50          |
| Intermediate          Mcx /      Mrx /      Pc /                    |
|                       Mcy        Mry        Pr                      |
| Flexure Comp          8.29E+01   2.66E+02   5.14E+01                |
|                       6.23E+01   0.00E+00   0.00E+00                |
| Flexure Tens          8.29E+01   2.66E+02   6.61E+02                |
|                       6.23E+01   0.00E+00   0.00E+00                |
|---------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2005
    59. METHOD ASD
    60. FU 9360 ALL
    61. FYLD 7200 ALL
```

```
   62. TRACK 2 ALL
   63. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W18X50                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            3.331         4
                       0.00            0.00        -183.75       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     3.331(FAIL)                 |
|          Loc  :   17.50   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
| Szz:   8.889E+01     Syy:   1.069E+01                                 |
| Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        35.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3     |
|                Slender        45.23     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1     |
|                Compact        45.23    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
```

```
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    0.00E+00   3.39E+02   0.000      Sec. E1       4     0.00   |
| Min Buck    0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00   |
| Flexural                                                                 |
| Tor Buck    0.00E+00   1.73E+02   0.000      Eq. E7-1      4     0.00   |
| Intermediate                                                             |
| Results     Eff Area     KL/r      Fcr        Fe         Pn             |
| Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02          |
| Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01          |
| Flexural       Ag       Fcr        Pn                                    |
| Tor Buck    1.02E-01   2.82E+03   2.88E+02                               |
|--------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                          |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Local-Z     0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00   |
| Local-Y     2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00   |
| Intermediate                                                             |
| Results        Aw         Cv       Kv      h/tw      Vn                  |
| Local-Z     5.94E-02     1.00     1.20     6.58   2.56E+02               |
| Local-Y     4.44E-02     1.00     0.00    45.23   1.92E+02               |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                               |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       1.84E+02   2.52E+02   0.729      Eq. F2-1      4    17.50   |
| Minor       0.00E+00   4.14E+01   0.000      Eq. F6-1      4     0.00   |
| Intermediate   Mn        My                                              |
| Major       4.21E+02   0.00E+00                                          |
| Minor       6.92E+01   0.00E+00                                          |
|--------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                             |
|                                                                          |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Major       1.84E+02   5.52E+01   3.331      Eq. F2-3      4    17.50   |
| Intermediate   Mn        Me         Cb       Lp       Lr       Lb      |
| Major       9.21E+01   0.00E+00    1.00     5.83    16.97    35.00      |
     STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|--------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                  |
|                        RATIO      CRITERIA    L/C     LOC               |
| Flexure Comp           3.331      Eq. H1-1b     4    17.50              |
| Flexure Tens           3.331      Eq. H1-1b     4    17.50              |
| Intermediate          Mcx /      Mrx /      Pc /                        |
|                       Mcy        Mry        Pr                          |
| Flexure Comp          5.52E+01   1.84E+02   3.42E+01                     |
|                       4.14E+01   0.00E+00   0.00E+00                     |
| Flexure Tens          5.52E+01   1.84E+02   4.40E+02                     |
|                       4.14E+01   0.00E+00   0.00E+00                     |
|--------------------------------------------------------------------------|
```

V. AISC 360-05 W Local Buckling F.3

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.3, pp.F-20 - F-22

Details

From reference (2):

> Select an ASTM A992 W-shape beam with a simple span of 40 feet. The nominal loads are a uniform dead load of 0.05 kip/ft and two equal 18 kip concentrated live loads acting at the third points of the beam. The beam is continuously braced. Also calculate the deflection.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	398	398	none	
M_n / Ω_b (ft·k) [ASD]	265	265	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Local Buckling F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
```

```
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.05
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -18 13.333
1 CON GY -18 26.667
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b           6.470         3
                        0.00            0.00       -395.99       20.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    289.731  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    20.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     6.470(FAIL)                |
|          Loc  :   20.00   Condition :   Eq. H1-1b                     |
```

```
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -3.960E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03             |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        40.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.49    T.B4.1-3     |
|                Slender        53.54     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact     9.47     9.15    24.08    T.B4.1-1     |
|                Compact        53.54    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   6.35E+02   0.000      Eq. D2-1      3     0.00  |
| Rupture     0.00E+00   6.87E+02   0.000      Eq. D2-2      3     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   4.61E+02   0.000      Sec. E1       3     0.00  |
| Min Buck    0.00E+00   3.79E+01   0.000      Eq. E7-1      3     0.00  |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.54E+02   0.000      Eq. E7-1      3     0.00  |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.92E-02    58.20   5.23E+03   1.22E+04   5.12E+02        |
| Min Buck    9.79E-02   289.73   4.31E+02   4.91E+02   4.22E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    9.79E-02   1.75E+03   1.72E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.89E+02   0.000      Eq. G2-1      3     0.00  |
| Local-Y     3.00E+01   2.16E+02   0.139      Eq. G2-1      3     0.00  |
| Intermediate                                                          |
| Results        Aw        Cv        Kv      h/tw      Vn               |
| Local-Z     4.86E-02     1.00      1.20     9.47   2.10E+02           |
| Local-Y     5.01E-02     1.00      0.00    53.54   2.16E+02           |
```

```
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   4.01E+02   0.987      Eq. F2-1      3    20.00 |
| Minor       0.00E+00   5.59E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn        My                                          |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   6.12E+01   6.470      Eq. F2-3      3    20.00 |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb     |
| Major       6.80E+01   0.00E+00    1.00     5.85    16.53    40.00   |
     STAAD SPACE                                               -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       3.96E+02   3.98E+02   0.995      Eq. F3-1      3    20.00 |
| Minor       0.00E+00   5.52E+01   0.000      Eq. F6-2      3     0.00 |
| Intermediate   Mn        Fcr                                         |
| Major       4.42E+02   0.00E+00                                       |
| Minor       6.14E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          6.470      Eq. H1-1b     3    20.00             |
| Flexure Tens          6.470      Eq. H1-1b     3    20.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         6.12E+01   3.96E+02   3.79E+01                  |
|                      5.52E+01   0.00E+00   0.00E+00                  |
| Flexure Tens         6.12E+01   3.96E+02   6.35E+02                  |
|                      5.52E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. FU 9360 ALL
    54. FYLD 7200 ALL
    55. FLX 2 ALL
    56. METHOD ASD
    57. TRACK 2 ALL
    58. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                               -- PAGE NO.
9
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
```

```
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                 (AISC SECTIONS)
                           FAIL   Eq. H1-1b          6.139        4
                        0.00            0.00        -249.99       20.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    289.731  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :    20.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     6.139(FAIL)                 |
|          Loc  :   20.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.500E+02               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03              |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        40.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.49    T.B4.1-3     |
|                Slender        53.54     N/A     35.88    T.B4.1-10    |
| Flexure     :  Non-Compact     9.47     9.15    24.08    T.B4.1-1     |
|                Compact        53.54    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
10
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       0.00E+00   4.22E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   4.58E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   3.07E+02   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   2.52E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   1.03E+02   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.92E-02    58.20   5.23E+03   1.22E+04   5.12E+02        |
```

```
| Min Buck    9.79E-02   289.73   4.31E+02   4.91E+02   4.22E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    9.79E-02   1.75E+03   1.72E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.26E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.90E+01   1.44E+02   0.132      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02            |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   2.67E+02   0.936      Eq. F2-1      4    20.00 |
| Minor       0.00E+00   3.72E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.46E+02   0.00E+00                                       |
| Minor       6.21E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   4.07E+01   6.139      Eq. F2-3      4    20.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       6.80E+01   0.00E+00    1.00     5.85    16.53    40.00   |
    STAAD SPACE                                             -- PAGE NO.
11
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.50E+02   2.65E+02   0.944      Eq. F3-1      4    20.00 |
| Minor       0.00E+00   3.67E+01   0.000      Eq. F6-2      4     0.00 |
| Intermediate   Mn         Fcr                                         |
| Major       4.42E+02   0.00E+00                                       |
| Minor       6.14E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp          6.139      Eq. H1-1b     4    20.00            |
| Flexure Tens          6.139      Eq. H1-1b     4    20.00            |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                        |
| Flexure Comp         4.07E+01   2.50E+02   2.52E+01                  |
|                      3.67E+01   0.00E+00   0.00E+00                  |
| Flexure Tens         4.07E+01   2.50E+02   4.22E+02                  |
|                      3.67E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 W LTB Test F.1-2

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.1-2, pp.F-8,9

Details

From reference (2):

> Verify the strength of the W18×50 beam selected in V. AISC 360-05 W Flex Memb F.1-1 (on page 5642) if the beam is braced at the ends and third points rather than continuously braced.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	305	306	negligible	
M_n / Ω_b (ft·k) [ASD]	203	203	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W LTB Test F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X50
CONSTANTS
```

```
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 FIXED BUT MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.45
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.75
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FU 9360 ALL
FYLD 7200 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FU 9360 ALL
FYLD 7200 ALL
CB 1.01 ALL
LX 11.67 ALL
UNB 11.67 ALL
UNT 11.67 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W18X50                   (AISC SECTIONS)
                          PASS   Eq. H1-1b            0.872         3
                       0.00           0.00        -266.44       17.50
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    254.294  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
```

```
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      3    Ratio     :     0.872(PASS)                 |
 |          Loc  :   17.50   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.664E+02               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
 | Szz:   8.889E+01     Syy:   1.069E+01                                 |
 | Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        35.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3     |
 |                Slender        45.23     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1     |
 |                Compact        45.23    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Yield      0.00E+00   6.61E+02   0.000      Eq. D2-1      3     0.00  |
 | Rupture    0.00E+00   7.17E+02   0.000      Eq. D2-2      3     0.00  |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Maj Buck   0.00E+00   5.09E+02   0.000      Sec. E1       3     0.00  |
 | Min Buck   0.00E+00   5.14E+01   0.000      Eq. E7-1      3     0.00  |
 | Flexural                                                              |
 | Tor Buck   0.00E+00   4.81E+02   0.000      Eq. E7-1      3     0.00  |
 | Intermediate                                                          |
 | Results    Eff Area     KL/r      Fcr        Fe         Pn            |
 | Maj Buck   9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02         |
 | Min Buck   1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01         |
 | Flexural      Ag       Fcr        Pn                                  |
 | Tor Buck   9.97E-02   5.24E+03   5.34E+02                             |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
 | Local-Z    0.00E+00   2.31E+02   0.000      Eq. G2-1      3     0.00  |
 | Local-Y    3.05E+01   1.92E+02   0.159      Eq. G2-1      3     0.00  |
```

```
| Intermediate                                                          |
| Results         Aw         Cv        Kv       h/tw       Vn          |
| Local-Z     5.94E-02     1.00      1.20      6.58   2.56E+02         |
| Local-Y     4.44E-02     1.00      0.00     45.23   1.92E+02         |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   3.79E+02   0.703      Eq. F2-1      3    17.50 |
| Minor       0.00E+00   6.23E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.21E+02   0.00E+00                                       |
| Minor       6.92E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.66E+02   3.06E+02   0.872      Eq. F2-2      3    17.50 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       3.39E+02   0.00E+00    1.01     5.83    16.97    11.67    |
     STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.872      Eq. H1-1b     3    17.50            |
| Flexure Tens           0.872      Eq. H1-1b     3    17.50            |
| Intermediate          Mcx /      Mrx /      Pc /                      |
|                       Mcy        Mry        Pr                        |
| Flexure Comp          3.06E+02   2.66E+02   5.14E+01                  |
|                       6.23E+01   0.00E+00   0.00E+00                  |
| Flexure Tens          3.06E+02   2.66E+02   6.61E+02                  |
|                       6.23E+01   0.00E+00   0.00E+00                  |
|-----------------------------------------------------------------------|
    52. LOAD LIST 4
    53. PARAMETER 2
    54. CODE AISC UNIFIED 2005
    55. METHOD ASD
    56. FU 9360 ALL
    57. FYLD 7200 ALL
    58. CB 1.01 ALL
    59. LX 11.67 ALL
    60. UNB 11.67 ALL
    61. UNT 11.67 ALL
    62. TRACK 2 ALL
    63. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1 ST   W18X50                  (AISC SECTIONS)
                         PASS   Eq. H1-1b           0.904        4
                      0.00            0.00        -183.75       17.50
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    254.294  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :    17.50             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     0.904(PASS)                 |
 |          Loc  :   17.50   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.838E+02               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   8.550E+00     Ayy:   6.390E+00     Cw:  3.046E+03              |
 | Szz:   8.889E+01     Syy:   1.069E+01                                 |
 | Izz:   8.000E+02     Iyy:   4.010E+01     Ix:  1.240E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        35.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     6.58     N/A     13.49    T.B4.1-3     |
 |                Slender        45.23     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         6.58     9.15    24.08    T.B4.1-1     |
 |                Compact        45.23    90.55   137.27    T.B4.1-9     |
      STAAD SPACE                                             -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Yield       0.00E+00   4.40E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   4.78E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Maj Buck    0.00E+00   3.39E+02   0.000      Sec. E1       4     0.00 |
 | Min Buck    0.00E+00   3.42E+01   0.000      Eq. E7-1      4     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   3.20E+02   0.000      Eq. E7-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    9.88E-02    56.93   5.54E+03   1.27E+04   5.65E+02        |
 | Min Buck    1.02E-01   254.29   5.59E+02   6.37E+02   5.71E+01        |
 | Flexural       Ag       Fcr        Pn                                 |
```

```
| Tor Buck    9.97E-02   5.24E+03   5.34E+02                                |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z     0.00E+00   1.54E+02   0.000      Eq. G2-1      4     0.00  |
| Local-Y     2.10E+01   1.28E+02   0.164      Eq. G2-1      4     0.00  |
| Intermediate                                                            |
| Results        Aw        Cv       Kv      h/tw      Vn                  |
| Local-Z     5.94E-02     1.00     1.20     6.58   2.56E+02              |
| Local-Y     4.44E-02     1.00     0.00    45.23   1.92E+02              |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                              |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       1.84E+02   2.52E+02   0.729      Eq. F2-1      4    17.50  |
| Minor       0.00E+00   4.14E+01   0.000      Eq. F6-1      4     0.00  |
| Intermediate   Mn        My                                             |
| Major       4.21E+02   0.00E+00                                        |
| Minor       6.92E+01   0.00E+00                                        |
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                            |
|                                                                         |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Major       1.84E+02   2.03E+02   0.904      Eq. F2-2      4    17.50  |
| Intermediate   Mn        Me         Cb      Lp       Lr       Lb      |
| Major       3.39E+02   0.00E+00    1.01     5.83    16.97    11.67     |
     STAAD SPACE                                                -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                        RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp           0.904      Eq. H1-1b     4    17.50             |
| Flexure Tens           0.904      Eq. H1-1b     4    17.50             |
| Intermediate          Mcx /      Mrx /      Pc /                       |
|                       Mcy        Mry        Pr                         |
| Flexure Comp          2.03E+02   1.84E+02   3.42E+01                   |
|                       4.14E+01   0.00E+00   0.00E+00                   |
| Flexure Tens          2.03E+02   1.84E+02   4.40E+02                   |
|                       4.14E+01   0.00E+00   0.00E+00                   |
|-------------------------------------------------------------------------|
```

V. AISC 360-05 W Memb Selection F.4

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.4, pp.F-23,24

Details

From reference (2):

> Select an ASTM A992 W-shape flexural member by the moment of inertia, to limit the live load deflection to 1 in. The span length is 30 ft. The nominal loads are a uniform dead load of 0.80 kip/ft and a uniform live load of 2 kip/ft. Assume the beam is continuously braced.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
Selected section [LRFD]	W24x55	W24x55	none	
Selected section [ASD]	W24x55	W24x55	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Memb Selection F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 14-Feb-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X192
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
2 FIXED BUT FX MZ
1 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
1 UNI GY -0.8
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -2
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
DEFINE ENVELOPE
3 ENVELOPE 1 TYPE STRENGTH
4 ENVELOPE 2 TYPE SERVICEABILITY
END DEFINE ENVELOPE
LOAD LIST ENV 2
PARAMETER 1
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
BEAM 1 ALL
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
TRACK 2 ALL
FLX 2 ALL
PROFILE W24 ALL
METHOD LRFD
SELECT OPTIMIZED
LOAD LIST ENV 2
PARAMETER 2
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
BEAM 1 ALL
DFF 240 ALL
DJ1 1 ALL
DJ2 2 ALL
TRACK 2 ALL
FLX 2 ALL
PROFILE W24 ALL
METHOD ASD
SELECT OPTIMIZED
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X192                  (AISC SECTIONS)
                           PASS   DEFLECTION           0.186         4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
```

```
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500        |
| Span/Deflection             :  1.29E+03  Deflection :     0.280        |
| L/C                         :     4      LOC        :    15.000        |
| Ratio                       :     0.186  (PASS)                        |
|------------------------------------------------------------------------|
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
4
                     STAAD.PRO MEMBER SELECTION - (AISC-360-05-LRFD)   v3.3a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION          0.864        4
                                                                15.00
|------------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                       |
|                                                                        |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500        |
| Span/Deflection             :  2.78E+02  Deflection :     1.296        |
| L/C                         :     4      LOC        :    15.000        |
| Ratio                       :     0.864  (PASS)                        |
|------------------------------------------------------------------------|
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
5
                     STAAD.PRO MEMBER SELECTION - (AISC-360-05-LRFD)   v3.3a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION          0.864        4
                                                                15.00
|------------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                       |
|                                                                        |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500        |
| Span/Deflection             :  2.78E+02  Deflection :     1.296        |
| L/C                         :     4      LOC        :    15.000        |
| Ratio                       :     0.864  (PASS)                        |
|------------------------------------------------------------------------|
    57. LOAD LIST ENV 2
    58. PARAMETER 2
    59. CODE AISC UNIFIED 2005
    60. FU 9360 ALL
    61. FYLD 7200 ALL
    62. BEAM 1 ALL
    63. DFF 240 ALL
    64. DJ1 1 ALL
    65. DJ2 2 ALL
    66. TRACK 2 ALL
    67. FLX 2 ALL
    68. PROFILE W24 ALL
```

```
   69. METHOD ASD
   70. SELECT OPTIMIZED
 STEEL DESIGN
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864        4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
 STEEL DESIGN
      STAAD SPACE                                               -- PAGE NO.
7
                     STAAD.PRO MEMBER SELECTION - ( AISC-360-05-ASD)   v3.3a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864        4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500       |
| Span/Deflection             :  2.78E+02  Deflection :     1.296       |
| L/C                         :     4      LOC        :    15.000       |
| Ratio                       :     0.864  (PASS)                       |
|-----------------------------------------------------------------------|
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
 STEEL DESIGN
      STAAD SPACE                                               -- PAGE NO.
8
                     STAAD.PRO MEMBER SELECTION - ( AISC-360-05-ASD)   v3.3a
                     ***********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W24X55                   (AISC SECTIONS)
                           PASS   DEFLECTION           0.864        4
                                                               15.00
|-----------------------------------------------------------------------|
| DEFLECTION CHECK:   (UNIT: INCH)                                      |
|                                                                       |
```

```
| Limit Span/Deflection (DFF) :  240.000   Limit      :     1.500        |
| Span/Deflection             :  2.78E+02  Deflection :     1.296        |
| L/C                         :     4      LOC        :    15.000        |
| Ratio                       :     0.864  (PASS)                        |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 W Section Weak Axis Shear G-6

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.6, p.G-11

Details

From reference (2):

Verify the strength of a W21×48 ASTM A992 beam with end shears of 20 kips from dead load and 60 kips from live load in the weak direction.

Validation

Refer to reference.

Table 772: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	189	189	none	
V_n / Ω_v (kips) [ASD]	126	126	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Section Weak Axis Shear G-6.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Oct-12
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.28151e+06
POISSON 0.3
```

```
DENSITY 0.489024
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 9360 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GZ -20 0.665
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GZ -60 0.665
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            1.439        3
                        0.00          -79.80           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     57.946  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
```

```
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     1.439(FAIL)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   1.200E+02               |
| Mx:   0.000E+00     My:  -7.980E+01     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03             |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         8.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.66    T.B4.1-3    |
|                Slender        53.54     N/A     36.33    T.B4.1-10   |
| Flexure     :  Non-Compact     9.47     9.27    24.39    T.B4.1-1    |
|                Compact        53.54    91.69   139.00    T.B4.1-9    |
     STAAD SPACE                                                -- PAGE NO.
4
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield       0.00E+00   6.35E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   6.87E+02   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck    0.00E+00   5.54E+02   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   4.66E+02   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   4.97E+02   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    8.62E-02    11.64   6.28E+03   3.12E+05   6.15E+02        |
| Min Buck    8.94E-02    57.95   5.28E+03   1.26E+04   5.17E+02        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    8.82E-02   5.64E+03   5.52E+02                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Local-Z    -1.20E+02   1.89E+02   0.635      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   2.16E+02   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
```

```
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02           |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.01E+02   0.000      Eq. F2-1      3     0.00 |
| Minor      -7.98E+01   5.59E+01   1.428      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                         |
| Major       4.46E+02   0.00E+00                                      |
| Minor       6.21E+01   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.71E+02   0.000      Eq. F2-2      3     0.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       4.12E+02   0.00E+00    1.00     5.93    16.76     8.00   |
    STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                              |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.99E+02   0.000      Eq. F3-1      3     0.00 |
| Minor      -7.98E+01   5.55E+01   1.439      Eq. F6-2      3     0.00 |
| Intermediate   Mn         Fcr                                        |
| Major       4.44E+02   0.00E+00                                      |
| Minor       6.16E+01   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                       RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp          1.439      Eq. H1-1b     3     0.00            |
| Flexure Tens          1.439      Eq. H1-1b     3     0.00            |
| Intermediate         Mcx /      Mrx /      Pc /                     |
|                      Mcy        Mry        Pr                       |
| Flexure Comp         3.71E+02   0.00E+00   4.66E+02                 |
|                      5.55E+01  -7.98E+01   0.00E+00                 |
| Flexure Tens         3.71E+02   0.00E+00   6.35E+02                 |
|                      5.55E+01  -7.98E+01   0.00E+00                 |
|----------------------------------------------------------------------|
   46. LOAD LIST 4
   47. PARAMETER 2
   48. CODE AISC UNIFIED 2005
   49. METHOD ASD
   50. FYLD 7200 ALL
   51. FU 9360 ALL
   52. TRACK 2 ALL
   53. CHECK CODE ALL
 STEEL DESIGN
    STAAD SPACE                                              -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W21X48                    (AISC SECTIONS)
                           FAIL   Eq. H1-1b            1.442         4
                       0.00          -53.20            0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     57.946  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     1.442(FAIL)                 |
|          Loc  :    0.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   8.000E+01               |
| Mx:   0.000E+00     My:  -5.320E+01     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.000E+00     Ayy:   7.210E+00     Cw:  3.936E+03              |
| Szz:   9.311E+01     Syy:   9.509E+00                                 |
| Izz:   9.590E+02     Iyy:   3.870E+01     Ix:  8.030E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         8.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     9.47     N/A     13.66    T.B4.1-3     |
|                Slender        53.54     N/A     36.33    T.B4.1-10    |
| Flexure     :  Non-Compact     9.47     9.27    24.39    T.B4.1-1     |
|                Compact        53.54    91.69   139.00    T.B4.1-9     |
     STAAD SPACE                                            -- PAGE NO.
7
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Yield      0.00E+00   4.22E+02   0.000      Eq. D2-1      4     0.00  |
| Rupture    0.00E+00   4.58E+02   0.000      Eq. D2-2      4     0.00  |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC   |
| Maj Buck   0.00E+00   3.68E+02   0.000      Sec. E1       4     0.00  |
| Min Buck   0.00E+00   3.10E+02   0.000      Eq. E7-1      4     0.00  |
| Flexural                                                              |
| Tor Buck   0.00E+00   3.31E+02   0.000      Eq. E7-1      4     0.00  |
| Intermediate                                                          |
```

```
| Results     Eff Area     KL/r      Fcr        Fe         Pn         |
| Maj Buck    8.62E-02    11.64   6.28E+03   3.12E+05   6.15E+02      |
| Min Buck    8.94E-02    57.95   5.28E+03   1.26E+04   5.17E+02      |
| Flexural       Ag        Fcr        Pn                              |
| Tor Buck    8.82E-02   5.64E+03   5.52E+02                          |
|---------------------------------------------------------------------|
| CHECK FOR SHEAR                                                     |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z    -8.00E+01   1.26E+02   0.636      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   1.44E+02   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                        |
| Results        Aw        Cv       Kv      h/tw      Vn              |
| Local-Z     4.86E-02     1.00     1.20     9.47   2.10E+02          |
| Local-Y     5.01E-02     1.00     0.00    53.54   2.16E+02          |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                          |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.67E+02   0.000      Eq. F2-1      4     0.00 |
| Minor      -5.32E+01   3.72E+01   1.431      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                        |
| Major       4.46E+02   0.00E+00                                     |
| Minor       6.21E+01   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                        |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.47E+02   0.000      Eq. F2-2      4     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb  |
| Major       4.12E+02   0.00E+00    1.00     5.93    16.76     8.00  |
     STAAD SPACE                                        -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       *******************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                             |
|                                                                     |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.66E+02   0.000      Eq. F3-1      4     0.00 |
| Minor      -5.32E+01   3.69E+01   1.442      Eq. F6-2      4     0.00 |
| Intermediate   Mn         Fcr                                       |
| Major       4.44E+02   0.00E+00                                     |
| Minor       6.16E+01   0.00E+00                                     |
|---------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                             |
|                        RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp           1.442      Eq. H1-1b     4     0.00          |
| Flexure Tens           1.442      Eq. H1-1b     4     0.00          |
| Intermediate         Mcx /      Mrx /      Pc /                    |
|                      Mcy        Mry        Pr                      |
| Flexure Comp         2.47E+02   0.00E+00   3.10E+02                |
|                      3.69E+01  -5.32E+01   0.00E+00                |
| Flexure Tens         2.47E+02   0.00E+00   4.22E+02                |
|                      3.69E+01  -5.32E+01   0.00E+00                |
|---------------------------------------------------------------------|
```

V. AISC 360-05 W Shape Strong Axis Shear G-1

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example G.1, pp.G-3,4

Details

From reference (2):

> Verify the shear strength of a W24×62 ASTM A992 beam with end shears of 48 kips from dead load and 145 kips from live load.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_v V_n$ (kips) [LRFD]	306	306	none	
V_n / Ω_v (kips) [ASD]	204	204	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Shape Strong Axis Shear G-1.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 16 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
```

```
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W24X62
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -100 8
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY -285 8
LOAD COMB 3 LRFD LOAD
1 1.2 2 1.6
LOAD COMB 4 ASD LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FYLD 7200 ALL
FU 9360 ALL
METHOD ASD
STP 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *     1 ST   W24X62                   (AISC SECTIONS)
                           FAIL   Eq. H1-1b            8.014        3
                        0.00            0.00       -2304.00       8.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    139.453  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
```

```
|-------------------------------------------------------------------------|
| STRENGTH CHECKS                                                         |
| Critical L/C  :      3    Ratio     :     8.014(FAIL)                   |
|          Loc  :    8.00   Condition :   Eq. H1-1b                       |
|-------------------------------------------------------------------------|
| DESIGN FORCES                                                           |
| Fx:   0.000E+00(  ) Fy:   2.880E+02     Fz:   0.000E+00                 |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.304E+03                 |
|-------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                        |
| Azz:   8.307E+00     Ayy:   1.019E+01     Cw:  4.606E+03                |
| Szz:   1.308E+02     Syy:   9.801E+00                                   |
| Izz:   1.550E+03     Iyy:   3.450E+01     Ix:  1.710E+00                |
|-------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                     |
| Fyld:        7199.999   Fu:        9359.999                             |
|-------------------------------------------------------------------------|
| Actual Member Length:        16.000                                     |
| Design Parameters                                                       |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00         |
|-------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                                |
| Compression :  Non-Slender     5.97     N/A     13.49    T.B4.1-3      |
|                Slender        50.05     N/A     35.88    T.B4.1-10     |
| Flexure     :  Compact         5.97     9.15    24.08    T.B4.1-1      |
|                Compact        50.05    90.55   137.27    T.B4.1-9      |
     STAAD SPACE                                              -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                 |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Yield       0.00E+00   8.19E+02   0.000      Eq. D2-1      3     0.00   |
| Rupture     0.00E+00   8.87E+02   0.000      Eq. D2-2      3     0.00   |
|-------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                             |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Maj Buck    0.00E+00   7.08E+02   0.000      Sec. E1       3     0.00   |
| Min Buck    0.00E+00   2.11E+02   0.000      Eq. E7-1      3     0.00   |
| Flexural                                                                |
| Tor Buck    0.00E+00   4.45E+02   0.000      Eq. E7-1      3     0.00   |
| Intermediate                                                            |
| Results     Eff Area     KL/r       Fcr        Fe         Pn            |
| Maj Buck    1.12E-01    20.81   6.23E+03   9.52E+04   7.87E+02          |
| Min Buck    1.26E-01   139.45   1.86E+03   2.12E+03   2.35E+02          |
| Flexural       Ag       Fcr        Pn                                   |
| Tor Buck    1.25E-01   3.91E+03   4.94E+02                              |
|-------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                         |
|                                                                         |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Local-Z     0.00E+00   2.24E+02   0.000      Eq. G2-1      3     0.00   |
| Local-Y     2.88E+02   3.06E+02   0.942      Eq. G2-1      3     0.00   |
```

```
| Intermediate                                                              |
| Results        Aw        Cv       Kv      h/tw      Vn                    |
| Local-Z     5.77E-02     1.00     1.20     5.97   2.49E+02                |
| Local-Y     7.08E-02     1.00     0.00    50.05   3.06E+02                |
|---------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                                |
|                                                                           |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major       2.30E+03   5.74E+02   4.016      Eq. F2-1      3     8.00    |
| Minor       0.00E+00   5.88E+01   0.000      Eq. F6-1      3     0.00    |
| Intermediate   Mn         My                                              |
| Major       6.38E+02   0.00E+00                                          |
| Minor       6.53E+01   0.00E+00                                          |
|---------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                              |
|                                                                           |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC     |
| Major       2.30E+03   2.87E+02   8.014      Eq. F2-3      3     8.00    |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb       |
| Major       3.19E+02   0.00E+00    1.00     4.86    14.40    16.00       |
     STAAD SPACE                                              -- PAGE NO.
5
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|---------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                   |
|                       RATIO      CRITERIA    L/C     LOC                  |
| Flexure Comp          8.014      Eq. H1-1b     3     8.00                 |
| Flexure Tens          8.014      Eq. H1-1b     3     8.00                 |
| Intermediate         Mcx /      Mrx /      Pc /                          |
|                      Mcy        Mry        Pr                            |
| Flexure Comp         2.87E+02   2.30E+03   2.11E+02                      |
|                      5.88E+01   0.00E+00   0.00E+00                      |
| Flexure Tens         2.87E+02   2.30E+03   8.19E+02                      |
|                      5.88E+01   0.00E+00   0.00E+00                      |
|---------------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2005
    59. FYLD 7200 ALL
    60. FU 9360 ALL
    61. METHOD ASD
    62. STP 1 ALL
    63. TRACK 2 ALL
    64. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
                      STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 ======================================================================
 *     1 ST   W24X62                   (AISC SECTIONS)
                         FAIL   Eq. H1-1b            8.051         4
                       0.00            0.00      -1540.00        8.00
```

```
 |-----------------------------------------------------------------------|
 | SLENDERNESS                                                           |
 | Actual Slenderness Ratio    :    139.453  L/C  :     4                |
 | Allowable Slenderness Ratio :    300.000  LOC  :     8.00             |
 |-----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                       |
 | Critical L/C  :      4    Ratio     :     8.051(FAIL)                 |
 |          Loc  :    8.00   Condition :   Eq. H1-1b                     |
 |-----------------------------------------------------------------------|
 | DESIGN FORCES                                                         |
 | Fx:   0.000E+00(  ) Fy:   1.925E+02     Fz:   0.000E+00               |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.540E+03               |
 |-----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                      |
 | Azz:   8.307E+00     Ayy:   1.019E+01     Cw:  4.606E+03              |
 | Szz:   1.308E+02     Syy:   9.801E+00                                 |
 | Izz:   1.550E+03     Iyy:   3.450E+01     Ix:  1.710E+00              |
 |-----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                   |
 | Fyld:        7199.999   Fu:        9359.999                           |
 |-----------------------------------------------------------------------|
 | Actual Member Length:        16.000                                   |
 | Design Parameters                                                     |
 | Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
 |-----------------------------------------------------------------------|
 | SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
 |                STIFFENED                                              |
 | Compression :  Non-Slender     5.97     N/A     13.49    T.B4.1-3     |
 |                Slender        50.05     N/A     35.88    T.B4.1-10    |
 | Flexure     :  Compact         5.97     9.15    24.08    T.B4.1-1     |
 |                Compact        50.05    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
  ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL TENSION                                               |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Yield       0.00E+00   5.45E+02   0.000      Eq. D2-1      4     0.00 |
 | Rupture     0.00E+00   5.92E+02   0.000      Eq. D2-2      4     0.00 |
 |-----------------------------------------------------------------------|
 | CHECK FOR AXIAL COMPRESSION                                           |
 |                                                                       |
 |              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
 | Maj Buck    0.00E+00   4.71E+02   0.000      Sec. E1       4     0.00 |
 | Min Buck    0.00E+00   1.41E+02   0.000      Eq. E7-1      4     0.00 |
 | Flexural                                                              |
 | Tor Buck    0.00E+00   2.96E+02   0.000      Eq. E7-1      4     0.00 |
 | Intermediate                                                          |
 | Results     Eff Area     KL/r      Fcr        Fe         Pn           |
 | Maj Buck    1.12E-01    20.81   6.23E+03   9.52E+04   7.87E+02        |
 | Min Buck    1.26E-01   139.45   1.86E+03   2.12E+03   2.35E+02        |
 | Flexural       Ag       Fcr        Pn                                 |
 | Tor Buck    1.25E-01   3.91E+03   4.94E+02                            |
 |-----------------------------------------------------------------------|
 | CHECK FOR SHEAR                                                       |
```

```
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.49E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     1.92E+02   2.04E+02   0.944      Eq. G2-1      4     0.00 |
| Intermediate                                                           |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     5.77E-02     1.00     1.20     5.97   2.49E+02            |
| Local-Y     7.08E-02     1.00     0.00    50.05   3.06E+02            |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.54E+03   3.82E+02   4.034      Eq. F2-1      4     8.00 |
| Minor       0.00E+00   3.91E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                           |
| Major       6.38E+02   0.00E+00                                        |
| Minor       6.53E+01   0.00E+00                                        |
|------------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.54E+03   1.91E+02   8.051      Eq. F2-3      4     8.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       3.19E+02   0.00E+00    1.00     4.86    14.40    16.00    |
     STAAD SPACE                                                -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                         RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp            8.051      Eq. H1-1b     4     8.00            |
| Flexure Tens            8.051      Eq. H1-1b     4     8.00            |
| Intermediate           Mcx /      Mrx /      Pc /                     |
|                        Mcy        Mry        Pr                       |
| Flexure Comp           1.91E+02   1.54E+03   1.41E+02                 |
|                        3.91E+01   0.00E+00   0.00E+00                 |
| Flexure Tens           1.91E+02   1.54E+03   5.45E+02                 |
|                        3.91E+01   0.00E+00   0.00E+00                 |
|------------------------------------------------------------------------|
```

V. AISC 360-05 W Tens and Bending H3

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example H.3, pp.H-6

Details

From reference (2):

Select an ASTM A992 W-shape with a 14-in. nominal depth to carry nominal forces of 29 kips from dead load and 87 kips from live load in axial tension, as well as the following nominal moments:

M_{xD} = 32.0 kip-ft	M_{xL} = 96.0 kip-ft
M_{yD} = 11.3 kip-ft	M_{yL} = 33.8 kip-ft

The unbraced length is 30 ft and the ends are pinned. Assume the connections are made with no holes.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
Interaction ratio per H1-1b [LRFD]	0.874	0.874	negligible	
Interaction ratio per H1-1b [ASD]	0.876	0.876	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 W Tens and Bending H3.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-May-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X82
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
2 FIXED BUT FX MY MZ
1 PINNED
LOAD 1 LOADTYPE None  TITLE FORCE
JOINT LOAD
2 FX 174
1 FX -174
LOAD 2 LOADTYPE None  TITLE MOMENT MAJOR
MEMBER LOAD
1 UNI GY -1.70668
LOAD 3 LOADTYPE None  TITLE MOMENT MINOR
MEMBER LOAD
1 UNI GZ 0.6009
UNIT INCHES KIP
LOAD 4 LOADTYPE None  TITLE FORCE2
JOINT LOAD
1 FX 116
2 FX 116
LOAD 5 LOADTYPE None  TITLE MOMENT MAJOR 2
MEMBER LOAD
1 UNI GY -0.0948
LOAD 6 LOADTYPE None  TITLE MOMENT MINOR 2
MEMBER LOAD
1 UNI GZ 0.0334
UNIT FEET KIP
LOAD COMB 7 LRFD
1 1.0 2 1.0 3 1.0
LOAD COMB 8 ASD
4 1.0 5 1.0 6 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
LOAD LIST 7
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 50 ALL
FU 65 ALL
CB 1.41 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 8
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 50 ALL
FU 65 ALL
CB 1.41 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
```

```
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 ST   W14X82                  (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.874         7
                      174.00 T       -811.21       -2304.02      180.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    144.970  L/C  :     7                |
| Allowable Slenderness Ratio :    300.000  LOC  :   180.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      7    Ratio     :     0.874(PASS)                 |
|          Loc  :  180.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.740E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:  -8.112E+02     Mz:  -2.304E+03              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.727E+01     Ayy:   7.293E+00     Cw:  6.688E+03             |
| Szz:   1.232E+02     Syy:   2.931E+01                                 |
| Izz:   8.810E+02     Iyy:   1.480E+02     Ix:  5.070E+00             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       360.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.91     N/A     13.49    T.B4.1-3     |
|                Non-Slender    22.35     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         5.91     9.15    24.08    T.B4.1-1     |
|                Compact        22.35    90.55   137.27    T.B4.1-9     |
     STAAD SPACE                                               -- PAGE NO.
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       1.74E+02   1.08E+03   0.161      Eq. D2-1      7     0.00 |
| Rupture     1.74E+02   1.17E+03   0.149      Eq. D2-2      7     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   8.34E+02   0.000      Sec. E1       7     0.00 |
| Min Buck    0.00E+00   2.58E+02   0.000      Eq. E3-1      7     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   7.99E+02   0.000      Eq. E4-1      7     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    2.40E+01    59.42   3.86E+01   8.11E+01   9.27E+02        |
```

```
| Min Buck    2.40E+01   144.97   1.19E+01   1.36E+01   2.87E+02         |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    2.40E+01   3.70E+01   8.87E+02                             |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     9.01E+00   4.66E+02   0.019      Eq. G2-1      7     0.00 |
| Local-Y     2.56E+01   2.19E+02   0.117      Eq. G2-1      7     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     1.73E+01     1.00     1.20     5.91   5.18E+02            |
| Local-Y     7.29E+00     1.00     0.00    22.35   2.19E+02            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       2.30E+03   6.26E+03   0.368      Eq. F2-1      7   180.00 |
| Minor      -8.11E+02   2.02E+03   0.402      Eq. F6-1      7   180.00 |
| Intermediate   Mn         My                                          |
| Major       6.95E+03   0.00E+00                                       |
| Minor       2.24E+03   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       2.30E+03   5.89E+03   0.391      Eq. F2-2      7   180.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb      |
| Major       6.54E+03   0.00E+00    1.41   105.26   396.11   360.00    |
     STAAD SPACE                                              -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.794      Eq. H1-1b     7   180.00             |
| Flexure Tens          0.874      Eq. H1-1b     7   180.00             |
| Intermediate         Mcx /      Mrx /      Pc /                      |
|                      Mcy        Mry        Pr                         |
| Flexure Comp         5.89E+03   2.30E+03   2.58E+02                   |
|                      2.02E+03  -8.11E+02   0.00E+00                   |
| Flexure Tens         5.89E+03   2.30E+03   1.08E+03                   |
|                      2.02E+03  -8.11E+02   1.74E+02                   |
|-----------------------------------------------------------------------|
    65. LOAD LIST 8
    66. PARAMETER 2
    67. CODE AISC UNIFIED 2005
    68. METHOD ASD
    69. FYLD 50 ALL
    70. FU 65 ALL
    71. CB 1.41 ALL
    72. TRACK 2 ALL
    73. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
6
```

```
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST   W14X82                  (AISC SECTIONS)
                           PASS   Eq. H1-1b           0.876         8
                      116.00 T       -541.08       -1535.76      180.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    144.970  L/C  :     8                |
| Allowable Slenderness Ratio :    300.000  LOC  :   180.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      8    Ratio     :     0.876(PASS)                 |
|          Loc  :  180.00   Condition :   Eq. H1-1b                     |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.160E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:  -5.411E+02     Mz:  -1.536E+03               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   1.727E+01     Ayy:   7.293E+00     Cw:  6.688E+03              |
| Szz:   1.232E+02     Syy:   2.931E+01                                 |
| Izz:   8.810E+02     Iyy:   1.480E+02     Ix:  5.070E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       360.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender     5.91     N/A     13.49    T.B4.1-3     |
|                Non-Slender    22.35     N/A     35.88    T.B4.1-10    |
| Flexure     :  Compact         5.91     9.15    24.08    T.B4.1-1     |
|                Compact        22.35    90.55   137.27    T.B4.1-9     |
      STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       1.16E+02   7.19E+02   0.161      Eq. D2-1      8     0.00 |
| Rupture     1.16E+02   7.80E+02   0.149      Eq. D2-2      8     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   5.55E+02   0.000      Sec. E1       8     0.00 |
| Min Buck    0.00E+00   1.72E+02   0.000      Eq. E3-1      8     0.00 |
| Flexural                                                              |
```

```
| Tor Buck    0.00E+00   5.31E+02   0.000      Eq. E4-1      8     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe          Pn         |
| Maj Buck    2.40E+01    59.42   3.86E+01   8.11E+01   9.27E+02       |
| Min Buck    2.40E+01   144.97   1.19E+01   1.36E+01   2.87E+02       |
| Flexural       Ag       Fcr        Pn                                |
| Tor Buck    2.40E+01   3.70E+01   8.87E+02                           |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     6.01E+00   3.10E+02   0.019      Eq. G2-1      8     0.00 |
| Local-Y     1.71E+01   1.46E+02   0.117      Eq. G2-1      8     0.00 |
| Intermediate                                                         |
| Results        Aw         Cv       Kv      h/tw      Vn             |
| Local-Z     1.73E+01     1.00     1.20     5.91   5.18E+02           |
| Local-Y     7.29E+00     1.00     0.00    22.35   2.19E+02           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.54E+03   4.16E+03   0.369      Eq. F2-1      8   180.00 |
| Minor      -5.41E+02   1.34E+03   0.403      Eq. F6-1      8   180.00 |
| Intermediate   Mn         My                                         |
| Major       6.95E+03   0.00E+00                                      |
| Minor       2.24E+03   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
|                                                                      |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.54E+03   3.92E+03   0.392      Eq. F2-2      8   180.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       6.54E+03   0.00E+00    1.41   105.26   396.11   360.00    |
    STAAD SPACE                                              -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
|                         RATIO      CRITERIA    L/C     LOC           |
| Flexure Comp            0.795      Eq. H1-1b     8   180.00          |
| Flexure Tens            0.876      Eq. H1-1b     8   180.00          |
| Intermediate           Mcx /      Mrx /      Pc /                  |
|                        Mcy        Mry        Pr                    |
| Flexure Comp           3.92E+03   1.54E+03   1.72E+02              |
|                        1.34E+03  -5.41E+02   0.00E+00              |
| Flexure Tens           3.92E+03   1.54E+03   7.19E+02              |
|                        1.34E+03  -5.41E+02   1.16E+02              |
|----------------------------------------------------------------------|
```

V. AISC 360-05 WT D.3

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example D.3, pp.D-6,7

Details

From reference (2):

> A WT6×20, ASTM A992, member has a length of 30 ft and carries a dead load of 40 kips and a live load of 120 kips in tension. Assume the end connection is fillet welded and has a length of 16 in. Verify the member strength by both LRFD and ASD. Assume that the gusset plate and the weld have been checked and are satisfactory.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
Tensile yielding $\phi_t P_n$ (kips) [LRFD]	263	263	none	
Tensile rupture $\phi_t P_n$ (kips) [LRFD]	266	266	none	
Tensile yielding P_n/Ω_t (kips) [ASD]	175	175	none	
Tensile rupture P_n/Ω_t (kips) [ASD]	177	177	none	
Slenderness ratio	229	230.607	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 WT D.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 21-Jan-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W12X40
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 40
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FX 120
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 7200 ALL
FU 9360 ALL
TMAIN 300 ALL
NSF 1 ALL
SLF 0.932 ALL
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
METHOD ASD
FYLD 7200 ALL
FU 9360 ALL
TMAIN 300 ALL
NSF 1 ALL
SLF 0.932 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
```

```
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 T    W12X40                    (AISC SECTIONS)
                           PASS   Eq. Sec. D2          0.912         3
                      240.00 T          0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    230.607  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.912(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. Sec. D2                   |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   2.400E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.125E+00     Ayy:   1.755E+00     Cw:  6.190E-01              |
| Szz:   2.920E+00     Syy:   5.506E+00                                 |
| Izz:   1.426E+01     Iyy:   2.205E+01     Ix:  4.530E-01              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.93 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.17     N/A     18.06    T.B4.1-8     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         7.78     9.15    24.08    T.B4.1-7     |
|                Non-Compact    20.17     9.15    24.08    T.B4.1-7     |
     STAAD SPACE                                                -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       2.40E+02   2.63E+02   0.912      Eq. D2-1      3     0.00 |
| Rupture     2.40E+02   2.66E+02   0.903      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Maj Buck    0.00E+00   2.49E+01   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   3.84E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   3.82E+01   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
```

```
| Maj Buck    4.06E-02   230.61   6.80E+02   7.75E+02   2.76E+01         |
| Min Buck    4.06E-02   185.43   1.05E+03   1.20E+03   4.27E+01         |
| Flexural       Ag       Fcr        Pn                                  |
| Tor Buck    4.06E-02   1.04E+03   4.24E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Local-Z     0.00E+00   1.11E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   4.74E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                           |
| Results        Aw         Cv       Kv      h/tw      Vn               |
| Local-Z     2.86E-02     1.00     1.20     7.78   1.24E+02            |
| Local-Y     1.22E-02     1.00     1.20    20.17   5.27E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                             |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   1.75E+01   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   3.15E+01   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                           |
| Major       1.95E+01   0.00E+00                                        |
| Minor       3.50E+01   0.00E+00                                        |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                           |
|                                                                        |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       0.00E+00   4.84E+01   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb    |
| Major       5.37E+01   0.00E+00    1.00     0.00     0.00    30.00    |
    STAAD SPACE                                              -- PAGE NO.
8
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                |
|                       RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp          0.000      Eq. H2-1      3     0.00             |
| Flexure Tens          0.912      Eq. H2-1      3     0.00             |
| Intermediate         frbw /     frbz /     fra /                      |
|                      Fcbw       Fcbz       Fca                        |
| Flexure Comp         0.00E+00   0.00E+00   0.00E+00                   |
|                      1.04E+04   9.89E+03   6.12E+02                   |
| Flexure Tens         0.00E+00   0.00E+00   5.91E+03                   |
|                      2.27E+03   9.89E+03   6.48E+03                   |
|-----------------------------------------------------------------------|
    50. LOAD LIST 4
    51. PARAMETER 2
    52. CODE AISC UNIFIED 2005
    53. METHOD ASD
    54. FYLD 7200 ALL
    55. FU 9360 ALL
    56. TMAIN 300 ALL
    57. NSF 1 ALL
    58. SLF 0.932 ALL
    59. TRACK 2 ALL
    60. CHECK CODE ALL
```

```
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
9
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
       1 T    W12X40                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.914         4
                      160.00 T         0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    230.607  L/C  :     4                |
| Allowable Slenderness Ratio :    300.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.914(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   1.600E+02(T ) Fy:   0.000E+00     Fz:   0.000E+00              |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00              |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   4.125E+00     Ayy:   1.755E+00     Cw:  6.190E-01             |
| Szz:   2.920E+00     Syy:   5.506E+00                                 |
| Izz:   1.426E+01     Iyy:   2.205E+01     Ix:  4.530E-01             |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:        30.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    0.93 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        20.17     N/A     18.06    T.B4.1-8     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         7.78     9.15    24.08    T.B4.1-7     |
|                Non-Compact    20.17     9.15    24.08    T.B4.1-7     |
      STAAD SPACE                                                -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Yield       1.60E+02   1.75E+02   0.914      Eq. D2-1      4     0.00 |
| Rupture     1.60E+02   1.77E+02   0.903      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
```

```
| Maj Buck    0.00E+00   1.65E+01   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   2.56E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.54E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    4.06E-02   230.61   6.80E+02   7.75E+02   2.76E+01       |
| Min Buck    4.06E-02   185.43   1.05E+03   1.20E+03   4.27E+01       |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    4.06E-02   1.04E+03   4.24E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   7.41E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   3.15E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     2.86E-02     1.00     1.20     7.78   1.24E+02            |
| Local-Y     1.22E-02     1.00     1.20    20.17   5.27E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   1.17E+01   0.000      Eq. F9-1      4     0.00 |
| Minor       0.00E+00   2.10E+01   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       1.95E+01   0.00E+00                                       |
| Minor       3.50E+01   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.22E+01   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       5.37E+01   0.00E+00    1.00     0.00     0.00    30.00   |
    STAAD SPACE                                              -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                        RATIO      CRITERIA    L/C     LOC             |
| Flexure Comp           0.000      Eq. H2-1      4     0.00            |
| Flexure Tens           0.914      Eq. H2-1      4     0.00            |
| Intermediate          frbw /     frbz /     fra /                     |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   0.00E+00                  |
|                       6.90E+03   6.58E+03   4.07E+02                  |
| Flexure Tens          0.00E+00   0.00E+00   3.94E+03                  |
|                       1.51E+03   6.58E+03   4.31E+03                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 WT E.7

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example E.7, pp. E-31 - E-33

Details

From reference (2):

> Select a WT-shape compression member with a length of 20 ft to support a dead load of 20 kips and live load of 60 kips in axial compression. The ends are pinned.

Validation

Refer to reference.

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_c P_n$ (kips) [LRFD]	128	127	negligible	
P_n / Ω_c (kips) [ASD]	85.0	84.3	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 WT E.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jan-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
```

```
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W14X68
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -20
LOAD 2 LOADTYPE Live  TITLE LOAD CASE 2
JOINT LOAD
2 FY -60
LOAD COMB 3 LRFD
1 1.2 2 1.6
LOAD COMB 4 ASD
1 1.0 2 1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
UNIT INCHES KIP
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
METHOD LRFD
FYLD 50 ALL
FU 65 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FYLD 50 ALL
FU 65 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                      STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                      ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W14X68                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.947        3
                     120.00 C          0.00           0.00       0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
```

```
| Actual Slenderness Ratio    :    133.504  L/C  :     3                 |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00              |
|------------------------------------------------------------------------|
| STRENGTH CHECKS                                                        |
| Critical L/C  :      3    Ratio     :     0.947(PASS)                 |
|          Loc  :    0.00   Condition :   Eq. H2-1                       |
|------------------------------------------------------------------------|
| DESIGN FORCES                                                          |
| Fx:   1.200E+02(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|------------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                       |
| Azz:   7.200E+00     Ayy:   2.905E+00     Cw:  3.173E+00              |
| Szz:   5.635E+00     Syy:   1.210E+01                                 |
| Izz:   3.232E+01     Iyy:   6.050E+01     Ix:  1.505E+00              |
|------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                    |
| Fyld:          50.000   Fu:          65.000                            |
|------------------------------------------------------------------------|
| Actual Member Length:       240.000                                    |
| Design Parameters                                                      |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|------------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE        |
|                STIFFENED                                               |
| Compression :  Non-Slender    16.87     N/A     18.06    T.B4.1-8      |
|                  N/A           N/A      N/A      N/A      N/A          |
| Flexure     :  Compact         6.94     9.15    24.08    T.B4.1-7      |
|                Non-Compact    16.87     9.15    24.08    T.B4.1-7      |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|------------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                                |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Yield       0.00E+00   4.50E+02   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   4.88E+02   0.000      Eq. D2-2      3     0.00 |
|------------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                            |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
| Maj Buck    1.20E+02   1.27E+02   0.947      Sec. E1       3     0.00 |
| Min Buck    1.20E+02   2.24E+02   0.535      Eq. E3-1      3     0.00 |
| Flexural                                                               |
| Tor Buck    1.20E+02   2.21E+02   0.542      Eq. E4-1      3     0.00 |
| Intermediate                                                           |
| Results     Eff Area     KL/r      Fcr        Fe         Pn            |
| Maj Buck    1.00E+01   133.50   1.41E+01   1.61E+01   1.41E+02        |
| Min Buck    1.00E+01    97.57   2.49E+01   3.01E+01   2.49E+02        |
| Flexural       Ag        Fcr        Pn                                 |
| Tor Buck    0.00E+00   2.46E+01   2.46E+02                             |
|------------------------------------------------------------------------|
| CHECK FOR SHEAR                                                        |
|                                                                        |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC    |
```

```
| Local-Z     0.00E+00   1.94E+02   0.000      Eq. G2-1      3     0.00 |
| Local-Y     0.00E+00   7.84E+01   0.000      Eq. G2-1      3     0.00 |
| Intermediate                                                          |
| Results        Aw        Cv       Kv      h/tw      Vn                |
| Local-Z     7.20E+00     1.00     1.20     6.94   2.16E+02            |
| Local-Y     2.90E+00     1.00     1.20    16.87   8.71E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   4.06E+02   0.000      Eq. F9-1      3     0.00 |
| Minor       0.00E+00   8.30E+02   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.51E+02   0.00E+00                                       |
| Minor       9.23E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   3.06E+03   0.000      Eq. F9-4      3     0.00 |
| Intermediate   Mn         Me         Cb      Lp       Lr       Lb    |
| Major       3.40E+03   0.00E+00    1.00     0.00     0.00   240.00    |
     STAAD SPACE                                              -- PAGE NO.
8
                        STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp           0.947      Eq. H2-1      3     0.00            |
| Flexure Tens           0.000      Eq. H2-1      3     0.00            |
| Intermediate          frbw /     frbz /     fra /                     |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   1.20E+01                  |
|                       7.20E+01   6.86E+01   1.27E+01                  |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                  |
|                       1.59E+01   6.86E+01   4.50E+01                  |
|-----------------------------------------------------------------------|
    58. LOAD LIST 4
    59. PARAMETER 2
    60. CODE AISC UNIFIED 2005
    61. FYLD 50 ALL
    62. FU 65 ALL
    63. METHOD ASD
    64. TRACK 2 ALL
    65. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                              -- PAGE NO.
9
                        STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                        ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1 T    W14X68                   (AISC SECTIONS)
                           PASS   Sec. E1             0.949        4
```

```
                       80.00 C         0.00           0.00        0.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    133.504  L/C  :     4                |
| Allowable Slenderness Ratio :    200.000  LOC  :     0.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      4    Ratio     :     0.949(PASS)                 |
|          Loc  :    0.00   Condition :   Sec. E1                       |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   8.000E+01(C ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:   0.000E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   7.200E+00     Ayy:   2.905E+00     Cw:  3.173E+00              |
| Szz:   5.635E+00     Syy:   1.210E+01                                 |
| Izz:   3.232E+01     Iyy:   6.050E+01     Ix:  1.505E+00              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:          50.000   Fu:          65.000                           |
|-----------------------------------------------------------------------|
| Actual Member Length:       240.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Non-Slender    16.87     N/A     18.06    T.B4.1-8     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Compact         6.94     9.15    24.08    T.B4.1-7     |
|                Non-Compact    16.87     9.15    24.08    T.B4.1-7     |
     STAAD SPACE                                             -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   2.99E+02   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   3.25E+02   0.000      Eq. D2-2      4     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    8.00E+01   8.43E+01   0.949      Sec. E1       4     0.00 |
| Min Buck    8.00E+01   1.49E+02   0.536      Eq. E3-1      4     0.00 |
| Flexural                                                              |
| Tor Buck    8.00E+01   1.47E+02   0.544      Eq. E4-1      4     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.00E+01   133.50   1.41E+01   1.61E+01   1.41E+02        |
| Min Buck    1.00E+01    97.57   2.49E+01   3.01E+01   2.49E+02        |
| Flexural       Ag        Fcr        Pn                                |
| Tor Buck    0.00E+00   2.46E+01   2.46E+02                            |
|-----------------------------------------------------------------------|
```

```
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.29E+02   0.000      Eq. G2-1      4     0.00 |
| Local-Y     0.00E+00   5.22E+01   0.000      Eq. G2-1      4     0.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     7.20E+00     1.00     1.20     6.94   2.16E+02            |
| Local-Y     2.90E+00     1.00     1.20    16.87   8.71E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.70E+02   0.000      Eq. F9-1      4     0.00 |
| Minor       0.00E+00   5.52E+02   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                          |
| Major       4.51E+02   0.00E+00                                       |
| Minor       9.23E+02   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       0.00E+00   2.03E+03   0.000      Eq. F9-4      4     0.00 |
| Intermediate   Mn         Me         Cb       Lp       Lr       Lb   |
| Major       3.40E+03   0.00E+00    1.00     0.00     0.00   240.00   |
    STAAD SPACE                                               -- PAGE NO.
11
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                               |
|                         RATIO      CRITERIA    L/C     LOC            |
| Flexure Comp            0.949      Eq. H2-1      4     0.00           |
| Flexure Tens            0.000      Eq. H2-1      4     0.00           |
| Intermediate          frbw /     frbz /     fra /                     |
|                       Fcbw       Fcbz       Fca                       |
| Flexure Comp          0.00E+00   0.00E+00   8.00E+00                  |
|                       4.79E+01   4.57E+01   8.43E+00                  |
| Flexure Tens          0.00E+00   0.00E+00   0.00E+00                  |
|                       1.06E+01   4.57E+01   2.99E+01                  |
|-----------------------------------------------------------------------|
```

V. AISC 360-05 WT Shape F.10

References

1. *Steel Construction Manual*, Version 13.0, American Institute of Steel Construction, 2005
2. *Design Examples*, Version 13.0, American Institute of Steel Construction, 2005, Example F.10, pp.F-37,38

Details

From reference (2):

Select an ASTM A992 WT beam with a simple span of 6 ft. The toe of the stem of the WT is in tension. The nominal loads are a uniform dead load of 0.08 kip/ft and a uniform live load of 0.24 kip/ft. There is no deflection limit for this member. Assume full lateral support.

Validation

Ultimate moment:

$$w_u = 1.2(0.08\text{ k/ft}) + 1.6(0.24\text{ k/ft}) = 0.48\text{ k/ft}$$

$$M_u = \frac{0.48(6)^2}{8} = 2.16\text{ ft-k}$$

Service moment:

$$w_a = 0.08\text{ k/ft} + 0.24\text{ k/ft} = 0.32\text{ k/ft}$$

$$M_a = \frac{0.32(6)^2}{8} = 1.44\text{ ft-k}$$

Try a WT5×6, which has a plastic section modulus, $Z_x = 2.20\ in^3$ and a section modulus, $S_x = 1.22\ in^3$.

Flexural Yielding

$$M_n = M_p = F_y \times Z_x = 50 \times 2.20 = 110\text{ in}\cdot\text{k} < 1.6F_y S_x = 1.6 \times 50 \times 1.22 = 97.6\text{ in-k}\ \text{(F9-2)}$$

Lateral-torsional buckling does not govern since the member is continuously braced.

Flange compactness: $\lambda = {}^{b_f}\!/_{2t_f} = 9.43$

Determine the compact and slender flange ratio limits from Table B4.1b:

$$\lambda_{pf} = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29{,}000}{46}} = 9.15$$

$$\lambda_{rf} = 1.0\sqrt{\frac{E}{F_y}} = 1.0\sqrt{\frac{29{,}000}{46}} = 24.1$$

Thus, the section has a non-compact flange.

$$M_n = \left[M_p - (M_p - 0.7F_y S_{xc})\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)\right] = \ = \left[110 - (110 - 0.7 \times 50 \times 3.20)\left(\frac{9.43 - 9.15}{24.1 - 9.15}\right)\right] = 110\text{ in}\cdot\text{k} >$$

So use $M_n = 97.6\ in \cdot k = 8.13\ ft \cdot k$

The available flexural strength:

LRFD: $\phi_b M_n = 0.90(8.13) = 7.32\ ft \cdot k$

ASD: $M_n / \Omega_b = {}^{8.13}\!/_{1.67} = 4.87\ ft \cdot k$

Result Type	Reference	STAAD.Pro	Difference	Comments
$\phi_b M_n$ (ft·k) [LRFD]	7.32	7.21	1.5%	
M_n/Ω_b (ft·k) [ASD]	4.87	4.80	1.5%	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC 360-05 WT Shape F.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W10X12
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED BUT MZ
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.08
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
MEMBER LOAD
1 UNI GY -0.24
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.2 2 1.6
LOAD COMB 4 COMBINATION LOAD CASE 4
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
```

```
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 2
CODE AISC UNIFIED 2005
FU 9360 ALL
FYLD 7200 ALL
FLX 2 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 T    W10X12                   (AISC SECTIONS)
                           PASS   Eq. H2-1             0.300         3
                        0.00            0.00          -2.16        3.00
|-----------------------------------------------------------------------|
| SLENDERNESS                                                           |
| Actual Slenderness Ratio    :     91.750  L/C  :     3                |
| Allowable Slenderness Ratio :    300.000  LOC  :     3.00             |
|-----------------------------------------------------------------------|
| STRENGTH CHECKS                                                       |
| Critical L/C  :      3    Ratio     :     0.300(PASS)                 |
|          Loc  :    3.00   Condition :   Eq. H2-1                      |
|-----------------------------------------------------------------------|
| DESIGN FORCES                                                         |
| Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00               |
| Mx:   0.000E+00     My:   0.000E+00     Mz:  -2.160E+00               |
|-----------------------------------------------------------------------|
| SECTION PROPERTIES  (UNIT: INCH)                                      |
| Azz:   8.316E-01     Ayy:   9.376E-01     Cw:  2.546E-02              |
| Szz:   1.202E+00     Syy:   5.505E-01                                 |
| Izz:   4.303E+00     Iyy:   1.090E+00     Ix:  2.735E-02              |
|-----------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                   |
| Fyld:        7199.999   Fu:        9359.999                           |
|-----------------------------------------------------------------------|
| Actual Member Length:         6.000                                   |
| Design Parameters                                                     |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00        |
|-----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE       |
|                STIFFENED                                              |
| Compression :  Slender        25.97     N/A     18.06    T.B4.1-8     |
|                  N/A           N/A      N/A      N/A      N/A         |
| Flexure     :  Non-Compact     9.43     9.15    24.08    T.B4.1-7     |
|                Slender        25.97     9.15    24.08    T.B4.1-7     |
     STAAD SPACE                                              -- PAGE NO.
```

```
4
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                               |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   7.96E+01   0.000      Eq. D2-1      3     0.00 |
| Rupture     0.00E+00   8.63E+01   0.000      Eq. D2-2      3     0.00 |
|-----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                           |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   4.31E+01   0.000      Sec. E1       3     0.00 |
| Min Buck    0.00E+00   3.28E+01   0.000      Eq. E7-1      3     0.00 |
| Flexural                                                              |
| Tor Buck    0.00E+00   2.72E+01   0.000      Eq. E7-1      3     0.00 |
| Intermediate                                                          |
| Results     Eff Area     KL/r      Fcr        Fe         Pn           |
| Maj Buck    1.23E-02    46.18   3.89E+03   1.93E+04   4.79E+01        |
| Min Buck    1.23E-02    91.75   2.96E+03   4.90E+03   3.64E+01        |
| Flexural       Ag       Fcr        Pn                                 |
| Tor Buck    1.23E-02   2.46E+03   3.02E+01                            |
|-----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                       |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   2.25E+01   0.000      Eq. G2-1      3     0.00 |
| Local-Y    -1.44E+00   2.53E+01   0.057      Eq. G2-1      3     6.00 |
| Intermediate                                                          |
| Results        Aw         Cv       Kv      h/tw      Vn              |
| Local-Z     5.78E-03     1.00     1.20     9.43   2.49E+01            |
| Local-Y     6.51E-03     1.00     1.20    25.97   2.81E+01            |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                            |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.16E+00   7.21E+00   0.300      Eq. F9-1      3     3.00 |
| Minor       0.00E+00   3.26E+00   0.000      Eq. F6-1      3     0.00 |
| Intermediate   Mn         My                                          |
| Major       8.01E+00   0.00E+00                                       |
| Minor       3.63E+00   0.00E+00                                       |
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                          |
|                                                                       |
|              FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       2.16E+00   2.45E+01   0.088      Eq. F9-4      3     3.00 |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb     |
| Major       2.72E+01   0.00E+00    1.00     0.00     0.00     6.00   |
     STAAD SPACE                                               -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (AISC-360-05-LRFD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-----------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                               |
|                                                                       |
```

```
 |             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
 | Major      2.16E+00   1.18E+01   0.182      Eq. F9-6      3     3.00 |
 | Minor      0.00E+00   3.23E+00   0.000      Eq. F6-2      3     0.00 |
 | Intermediate   Mn        Fcr                                         |
 | Major      1.32E+01   7.16E+03                                       |
 | Minor      3.59E+00   0.00E+00                                       |
 |----------------------------------------------------------------------|
 | CHECK FOR FLEXURE TENS/COMP INTERACTION                              |
 |                      RATIO      CRITERIA    L/C     LOC              |
 | Flexure Comp         0.300      Eq. H2-1      3     3.00             |
 | Flexure Tens         0.300      Eq. H2-1      3     3.00             |
 | Intermediate        frbw /     frbz /     fra /                      |
 |                     Fcbw       Fcbz       Fca                        |
 | Flexure Comp        1.17E+03   0.00E+00   0.00E+00                   |
 |                     3.92E+03   1.01E+04   2.21E+03                   |
 | Flexure Tens        3.11E+03   0.00E+00   0.00E+00                   |
 |                     1.04E+04   1.01E+04   6.48E+03                   |
 |----------------------------------------------------------------------|
    56. LOAD LIST 4
    57. PARAMETER 2
    58. CODE AISC UNIFIED 2005
    59. FU 9360 ALL
    60. FYLD 7200 ALL
    61. FLX 2 ALL
    62. METHOD ASD
    63. TRACK 2 ALL
    64. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
      1 T    W10X12                  (AISC SECTIONS)
                          PASS   Eq. H2-1             0.300        4
                       0.00            0.00          -1.44        3.00
 |----------------------------------------------------------------------|
 | SLENDERNESS                                                          |
 | Actual Slenderness Ratio    :     91.750  L/C  :     4               |
 | Allowable Slenderness Ratio :    300.000  LOC  :     3.00            |
 |----------------------------------------------------------------------|
 | STRENGTH CHECKS                                                      |
 | Critical L/C  :      4    Ratio     :     0.300(PASS)                |
 |          Loc  :    3.00   Condition :   Eq. H2-1                     |
 |----------------------------------------------------------------------|
 | DESIGN FORCES                                                        |
 | Fx:   0.000E+00(  ) Fy:   0.000E+00     Fz:   0.000E+00              |
 | Mx:   0.000E+00     My:   0.000E+00     Mz:  -1.440E+00              |
 |----------------------------------------------------------------------|
 | SECTION PROPERTIES  (UNIT: INCH)                                     |
 | Azz:   8.316E-01     Ayy:   9.376E-01     Cw:  2.546E-02             |
 | Szz:   1.202E+00     Syy:   5.505E-01                                |
 | Izz:   4.303E+00     Iyy:   1.090E+00     Ix:  2.735E-02             |
 |----------------------------------------------------------------------|
 | MATERIAL PROPERTIES                                                  |
```

```
| Fyld:        7199.999   Fu:        9359.999                          |
|----------------------------------------------------------------------|
| Actual Member Length:         6.000                                  |
| Design Parameters                                                    |
| Kz:    1.00 Ky:    1.00 NSF:    1.00 SLF:    1.00 CSP:   12.00       |
|----------------------------------------------------------------------|
| SECTION CLASS  UNSTIFFENED /     λ       λp      λr        CASE      |
|                STIFFENED                                             |
| Compression :  Slender        25.97     N/A     18.06    T.B4.1-8    |
|                  N/A           N/A      N/A      N/A      N/A        |
| Flexure     :  Non-Compact     9.43     9.15    24.08    T.B4.1-7    |
|                Slender        25.97     9.15    24.08    T.B4.1-7    |
     STAAD SPACE                                              -- PAGE NO.
7
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|----------------------------------------------------------------------|
| CHECK FOR AXIAL TENSION                                              |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Yield       0.00E+00   5.30E+01   0.000      Eq. D2-1      4     0.00 |
| Rupture     0.00E+00   5.75E+01   0.000      Eq. D2-2      4     0.00 |
|----------------------------------------------------------------------|
| CHECK FOR AXIAL COMPRESSION                                          |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Maj Buck    0.00E+00   2.87E+01   0.000      Sec. E1       4     0.00 |
| Min Buck    0.00E+00   2.18E+01   0.000      Eq. E7-1      4     0.00 |
| Flexural                                                             |
| Tor Buck    0.00E+00   1.81E+01   0.000      Eq. E7-1      4     0.00 |
| Intermediate                                                         |
| Results     Eff Area     KL/r      Fcr        Fe         Pn          |
| Maj Buck    1.23E-02    46.18   3.89E+03   1.93E+04   4.79E+01       |
| Min Buck    1.23E-02    91.75   2.96E+03   4.90E+03   3.64E+01       |
| Flexural       Ag       Fcr        Pn                                |
| Tor Buck    1.23E-02   2.46E+03   3.02E+01                           |
|----------------------------------------------------------------------|
| CHECK FOR SHEAR                                                      |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Local-Z     0.00E+00   1.49E+01   0.000      Eq. G2-1      4     0.00 |
| Local-Y    -9.60E-01   1.68E+01   0.057      Eq. G2-1      4     6.00 |
| Intermediate                                                         |
| Results        Aw        Cv       Kv      h/tw      Vn               |
| Local-Z     5.78E-03     1.00     1.20     9.43   2.49E+01           |
| Local-Y     6.51E-03     1.00     1.20    25.97   2.81E+01           |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-YIELDING                                           |
|                                                                      |
|             FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC |
| Major       1.44E+00   4.80E+00   0.300      Eq. F9-1      4     3.00 |
| Minor       0.00E+00   2.17E+00   0.000      Eq. F6-1      4     0.00 |
| Intermediate   Mn         My                                         |
| Major       8.01E+00   0.00E+00                                      |
| Minor       3.63E+00   0.00E+00                                      |
|----------------------------------------------------------------------|
| CHECK FOR BENDING-LATERAL TORSIONAL BUCKLING                         |
```

```
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.44E+00   1.63E+01   0.089      Eq. F9-4      4     3.00  |
| Intermediate   Mn         Me        Cb      Lp       Lr       Lb       |
| Major       2.72E+01   0.00E+00    1.00     0.00     0.00     6.00     |
     STAAD SPACE                                               -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - ( AISC-360-05-ASD)   v3.3a
                       ********************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
|-------------------------------------------------------------------------|
| CHECK FOR BENDING-FLANGE LOCAL BUCKLING                                 |
|                                                                         |
|               FORCE     CAPACITY   RATIO      CRITERIA     L/C    LOC  |
| Major       1.44E+00   7.88E+00   0.183      Eq. F9-6      4     3.00  |
| Minor       0.00E+00   2.15E+00   0.000      Eq. F6-2      4     0.00  |
| Intermediate   Mn         Fcr                                           |
| Major       1.32E+01   7.16E+03                                         |
| Minor       3.59E+00   0.00E+00                                         |
|-------------------------------------------------------------------------|
| CHECK FOR FLEXURE TENS/COMP INTERACTION                                 |
|                         RATIO      CRITERIA    L/C     LOC              |
| Flexure Comp            0.300      Eq. H2-1      4     3.00             |
| Flexure Tens            0.300      Eq. H2-1      4     3.00             |
| Intermediate           frbw /     frbz /     fra /                      |
|                        Fcbw       Fcbz       Fca                        |
| Flexure Comp           7.83E+02   0.00E+00   0.00E+00                   |
|                        2.61E+03   6.74E+03   1.47E+03                   |
| Flexure Tens           2.07E+03   0.00E+00   0.00E+00                   |
|                        6.90E+03   6.74E+03   4.31E+03                   |
|-------------------------------------------------------------------------|
```

V. AISC ASD - Column Compression Capacity 1

Determine the axial compression capacity of a W14X132 column.

Reference

AISC Allowable Stress Design, 9th Edition, Example 1, page 3-4.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

K_yL_y =16ft

K_zL_z = 31ft

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output:

17.36 ksi (38.8 in^2) = 673.6 kips

Result Type	Reference	STAAD.Pro	Difference
Allowable load (kips)	679	673.6	0.8%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Column Compression Capacity 1.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-4 OF 9TH ED. AISC-ASD. EXAMPLE 1. FA SHOULD BE 679/38.8 =
* 17.5 KSI. RATIO SHOULD BE APPROXIMATELY 670/679 = 0.987
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 31 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -670
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
LY 192 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO.
3
```

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES   |
   |*************                                       |        IN INCH UNIT |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  38.80  |
   |            *  | ST  W14X132                 |      |   --Z  AY =   8.48  |
   |DESIGN CODE *  |                             |      |        AZ =  20.33  |
   | AISC-1989  *  ===============================   ===|===     SY =  74.56  |
   |            *                                                SZ = 208.16  |
   |            *  |<---LENGTH (FT)=   31.00 --->|                  RY =
3.76  |
   |*************                                                RZ =   6.28  |
   |                                                                          |
   |                 0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
   |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y=  51.09  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA  =  17.36 |
   | KL/R-Z=  59.24  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa  =  17.27 |
   | UNL   = 372.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  21.60 |
   | CB    =   1.00  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  21.60 |
   | CMY   =   0.85  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.00 |
   | CMZ   =   0.85  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.00 |
   | FYLD  =  36.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
   | DFF   =   0.00   0.0                                        Fey =  57.21 |
   | dff=      0.00             ABSOLUTE MZ ENVELOPE             Fez =  42.55 |
   | (KL/R)max =  59.24            (WITH LOAD NO.)               FV  =  14.40 |
   |                                                             fv  =   0.00 |
   |                                                                          |
   |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                     -------------------------                            |
   |                                                                          |
   |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                          |
   |      VALUE     670.0         0.0        0.0         0.0         0.0      |
   |   LOCATION       0.0         0.0        0.0         0.0         0.0      |
   |    LOADING         1           0          0           0           0      |
   |                                                                          |
   |**************************************************************************|
   |*                                                                        *|
   |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
   |*                        --------------                                  *|
   |*                                                                        *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
   |         FX              MY             MZ          LOCATION              |
   |        ======================================================            |
   |          PASS        AISC- H1-1       0.994             1                |
   |        670.00 C         0.00          0.00            0.00               |
   |*                                                                        *|
   |                                                                          |
   |--------------------------------------------------------------------------|
      AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO.
4
    41. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
```

```
          **** DATE= MAR 24,2022   TIME= 10: 4:47 ****
    ************************************************************
```

V. AISC ASD - Column Compression Capacity 2

Determine the axial compression capacity of a W12X106 column.

Reference

AISC Allowable Stress Design, 9th Edition, Example 2, page 3-4.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

K_yL_y =11ft

K_zL_z = 22ft

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = 18.30(31.20) = 571.0.

Table 773: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Allowable load (kips)	577	571.0	1.0%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Column Compression Capacity 2.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-4 OF 9TH ED. AISC-ASD. EXAMPLE 2.
* RATIO SHOULD BE APPROXIMATELY 540/577 = 0.936
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 11 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X106
UNIT INCHES KIP
DEFINE MATERIAL START
```

```
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -540
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
KZ 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO.
3
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y         PROPERTIES  |
   |*************                                       |         IN INCH UNIT |
   |            *  |=============================|   ===|===      ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |         AX =  31.20  |
   |            *  | ST  W12X106                 |      |   --Z   AY =   7.02  |
   |DESIGN CODE *  |                             |      |         AZ =  16.22  |
   | AISC-1989  *  ===============================   ===|===      SY =  49.34  |
   |            *                                                 SZ = 144.65  |
   |            *  |<---LENGTH (FT)=   11.00 --->|                   RY =
3.11  |
   |*************                                                 RZ =   5.47  |
   |                                                                           |
   |                  0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
   |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y=  42.50  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA   =  18.30 |
   | KL/R-Z=  48.28  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa   =  17.31 |
   | UNL    = 132.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  23.76 |
   | CB     =   1.00  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  23.76 |
   | CMY    =   0.85  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.00 |
   | CMZ    =   0.85  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.00 |
   | FYLD   =  36.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
```

```
 | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
 | DFF   =   0.00   0.0                                        Fey =  82.68 |
 | dff=      0.00              ABSOLUTE MZ ENVELOPE            Fez =  64.07 |
 | (KL/R)max =  48.28             (WITH LOAD NO.)              FV  =  14.40 |
 |                                                             fv  =   0.00 |
 |                                                                          |
 |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
 |                     -------------------------                            |
 |                                                                          |
 |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
 |                                                                          |
 |     VALUE     540.0          0.0        0.0         0.0         0.0      |
 |   LOCATION      0.0          0.0        0.0         0.0         0.0      |
 |    LOADING        1            0          0           0           0      |
 |                                                                          |
 |**************************************************************************|
 |*                                                                        *|
 |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
 |*                        --------------                                  *|
 |*                                                                        *|
 |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
 |         FX             MY             MZ           LOCATION             |
 |        ======================================================           |
 |          PASS        AISC- H1-1       0.946            1                |
 |        540.00 C         0.00          0.00           0.00               |
 |*                                                                        *|
 |                                                                          |
 |--------------------------------------------------------------------------|
```

V. AISC ASD - Column Compression Capacity 3

Determine the axial compression capacity of a W12X40 column.

Reference

AISC Allowable Stress Design, 9th Edition, tables on page 3-28.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

$K_yL_y = K_zL_z = 16ft$

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = *12.97(11.70) = 151.7*.

Table 774: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Axial capacity (kips)	154	151.7	1.5%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Column Compression Capacity 3.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-28 OF 9TH ED. AISC-ASD. W12X40, 16 FT LONG.
* FA SHOULD BE APPROXIMATELY 154/11.80 = 13.05 KSI
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 16 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -150
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY -- PAGE NO.
3
                        STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                        ********************************************
   |--------------------------------------------------------------------------|
   |                                                   Y        PROPERTIES    |
   |*************                                      |        IN INCH UNIT  |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  11.70  |
   |            *  | ST  W12X40                  |      |    --Z  AY =   3.25  |
   |DESIGN CODE *  |                             |      |        AZ =   5.51  |
   | AISC-1989  *  ===============================   ===|===     SY =  11.01  |
   |            *                                               SZ =  51.60  |
   |            *  |<---LENGTH (FT)=   16.00 --->|                 RY =
1.94  |
   |*************                                               RZ =   5.12  |
   |                                                                          |
   |                 0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
   |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y=  98.90  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA  =  12.97 |
   | KL/R-Z=  37.48  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa  =  12.82 |
   | UNL   = 192.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  21.60 |
   | CB    =   1.00  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  21.60 |
   | CMY   =   0.85  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.00 |
   | CMZ   =   0.85  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.00 |
   | FYLD  =  36.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
   | DFF   =   0.00   0.0                                        Fey =  15.27 |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE            Fez = 106.29 |
   | (KL/R)max =  98.90             (WITH LOAD NO.)              FV  =  14.40 |
   |                                                             fv  =   0.00 |
   |                                                                          |
   |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                     -------------------------                            |
   |                                                                          |
   |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                          |
   |      VALUE     150.0         0.0        0.0         0.0         0.0      |
   |   LOCATION       0.0         0.0        0.0         0.0         0.0      |
   |    LOADING         1           0          0           0           0      |
   |                                                                          |
   |**************************************************************************|
   |*                                                                        *|
   |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
   |*                        --------------                                  *|
   |*                                                                        *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
   |         FX              MY             MZ          LOCATION              |
   |       ======================================================            |
   |         PASS       AISC- H1-1       0.988            1                   |
   |       150.00 C          0.00        0.00             0.00                |
   |*                                                                        *|
```

```
|                                                                           |
|---------------------------------------------------------------------------|
```

V. AISC ASD - Square Tube Compression Capacity

Determine the axial compression capacity of a square 8x8x5/8 structural tube.

Reference

AISC Allowable Stress Design, 9th Edition, tables on page 3-41.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =46 ksi

$K_yL_y = K_zL_z = 25ft$

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = *14.15(17.40) = 246.2*.

Table 775: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Axial capacity (kips)	247	246.2	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Square Tube Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 03-Sep-18
END JOB INFORMATION
* PAGE 3-41 OF 9TH ED. AISC-ASD. SQUARE TUBE 8X8X5/8, 25 FT LONG.
* FA SHOULD BE APPROX. 247/17.40 = 14.20 KSI
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
TUB808010
```

```
17.4 8 8 0.625 153 153 258 7.48 7.48
END
UNIT FEET KIP
UNIT INCHES KIP
MEMBER PROPERTY
1 UPTABLE 1 TUB808010
*1 TA ST TUB808010
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -240
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 46 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |-----------------------------------------------------------------------|
   |                                                    Y        PROPERTIES |
   |*************                                       |        IN INCH UNIT |
   |            *  |=============================|   |=====|      ------------ |
   |MEMBER   1  *  |            UPT              |   |     |      AX =  17.40  |
   |            *  | UP  TUB808010               |   |     |--Z  AY =   7.48  |
   |DESIGN CODE *  |                             |   |     |      AZ =   7.48  |
   | AISC-1989  *  ===============================   |=====|      SY =  38.25  |
   |            *                                                  SZ =  38.25  |
   |            *  |&lt;---LENGTH (FT)=   25.00 --->|                  RY =
2.97  |
   |*************                                                  RZ =   2.97  |
   |                                                                          |
   |                 0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
   |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y= 101.17  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA   =  14.15 |
   | KL/R-Z= 101.17  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa   =  13.79 |
```

```
| UNL    = 300.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  27.60 |
| CB     =   1.00  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  27.60 |
| CMY    =   0.85  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.60 |
| CMZ    =   0.85  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.60 |
| FYLD   =  46.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
| NSF    =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
| DFF    =   0.00   0.0                                        Fey =  14.59 |
| dff=       0.00              ABSOLUTE MZ ENVELOPE            Fez =  14.59 |
| (KL/R)max = 101.17             (WITH LOAD NO.)               FV  =  18.40 |
|                                                              fv  =   0.00 |
|                                                                           |
|                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
|                     -------------------------                             |
|                                                                           |
|               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
|                                                                           |
|      VALUE    240.0          0.0        0.0         0.0         0.0       |
|   LOCATION      0.0          0.0        0.0         0.0         0.0       |
|    LOADING        1            0          0           0           0       |
|                                                                           |
|***************************************************************************|
|*                                                                         *|
|*                        DESIGN SUMMARY  (KIP-FEET)                       *|
|*                        --------------                                   *|
|*                                                                         *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/            *|
|         FX              MY            MZ            LOCATION             |
|        ======================================================            |
|          PASS       AISC- H1-1       0.975             1                 |
|        240.00 C         0.00         0.00            0.00                |
|*                                                                         *|
|                                                                           |
|---------------------------------------------------------------------------|
```

V. AISC ASD - Rectangular Tube Compression Capacity

Determine the axial compression capacity of a rectangular 7x5x3/8 structural tube.

Reference

AISC Allowable Stress Design, 9th Edition, tables on page 3-49.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =46 ksi

$K_yL_y = K_zL_z$ = 25ft

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output = *6.32(8.08) = 51.06*.

Table 776: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Axial capacity (kips)	51	51.1	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Rectangular Tube Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 03-Sep-18
END JOB INFORMATION
* PAGE 3-49 OF 9TH ED. AISC-ASD. RECTANGULAR TUBE 7X5X3/8, 25 FT LONG.
* FA SHOULD BE APPROX. 51/8.08 = 6.31 KSI
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 25 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
TUB70506
8.08 7 5 0.375 52.2 30.8 64.2 3.97 2.98
END
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 TUB70506
*1 TA ST TUB70506
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -50
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 46 ALL
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y         PROPERTIES  |
   |*************                                       |         IN INCH UNIT |
   |            *  |=============================|   |=====|      ------------ |
   |MEMBER   1  *  |             UPT             |   |     |      AX =   8.08  |
   |            *  | UP  TUB70506                |   |     |--Z   AY =   3.97  |
   |DESIGN CODE *  |                             |   |     |      AZ =   2.98  |
   | AISC-1989  *  ===============================   |=====|      SY =  12.32  |
   |            *                                                 SZ =  14.91  |
   |            *  |&lt;---LENGTH (FT)=   25.00 --->|                  RY =
1.95  |
   |*************                                                 RZ =   2.54  |
   |                                                                          |
   |                 0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
   |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y= 153.66  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA  =   6.32 |
   | KL/R-Z= 118.03  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa  =   6.19 |
   | UNL   = 300.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  27.60 |
   | CB    =   1.00  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  27.60 |
   | CMY   =   0.85  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.60 |
   | CMZ   =   0.85  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.60 |
   | FYLD  =  46.00  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
   | DFF   =   0.00   0.0                                        Fey =   6.32 |
   | dff=      0.00               ABSOLUTE MZ ENVELOPE           Fez =  10.72 |
   | (KL/R)max = 153.66              (WITH LOAD NO.)             FV  =  18.40 |
   |                                                             fv  =   0.00 |
   |                                                                          |
   |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                     -------------------------                            |
   |                                                                          |
   |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                          |
   |      VALUE      50.0         0.0        0.0         0.0         0.0      |
   |    LOCATION      0.0         0.0        0.0         0.0         0.0      |
   |     LOADING        1           0          0           0           0      |
   |                                                                          |
   |**************************************************************************|
   |*                                                                        *|
   |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
   |*                        --------------                                  *|
   |*                                                                        *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
   |         FX                MY           MZ          LOCATION              |
   |        ===================================================               |
   |          PASS        AISC- H1-1       0.978             1                |
   |         50.00 C          0.00         0.00            0.00               |
```

```
|*                                                                          *|
|                                                                            |
|----------------------------------------------------------------------------|
```

V. AISC ASD - Tee Compression Capacity

Determine the axial compression capacity of a WT10.5x25.

Reference

AISC Allowable Stress Design, 9th Edition, tables on page 3-54.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

K_yL_y = 5ft

K_zL_z = 30ft

Comparison

The reported STAAD.Pro value is obtained by multiplying FA times AX from the STAAD.Pro output =*10.09(7.35) = 74.16*.

Table 777: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Axial capacity (kips)	74	74.16	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Tee Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION CAPACITY.
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-93 OF 9TH ED. AISC-ASD. TEE SHAPE WT10.5X25, KL Z-AXIS = 30 FT
* KL Y-AXIS = 5 FT, FA SHOULD BE APPROX. 74/7.35 = 10.07 KSI
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE T W21X50
UNIT INCHES KIP
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -70
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
LZ 360 ALL
LY 60 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES   |
  |*************                                       |        IN INCH UNIT |
  |            *  |=============================|   ===|===     ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =   7.35  |
  |            *  | T   W21X50                  |      |   --Z  AY =   2.83  |
  |DESIGN CODE *  |                             |      |        AZ =   2.63  |
  | AISC-1989  *  |-----------------------------|      |        SY =   3.81  |
  |            *                                                SZ =  10.63  |
  |            *  |&lt;---LENGTH (FT)=   10.00 --->|                  RY =
1.30  |
  |*************                                                RZ =   3.29  |
  |                                                                          |
  |                 0.0 (KIP-FEET)                                           |
  |PARAMETER        |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
  |IN KIP  INCH     |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
  |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
  | KL/R-Y=  85.76  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA   =  10.09 |
  | KL/R-Z= 109.54  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa   =   9.52 |
  | UNL    = 120.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  15.85 |
  | CB     =   1.00 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  21.60 |
  | CMY    =   0.85 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  27.00 |
  | CMZ    =   0.85 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  27.00 |
  | FYLD   =  36.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
```

```
| NSF    =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
| DFF    =   0.00   0.0                                       Fey =  20.30 |
| dff=       0.00              ABSOLUTE MZ ENVELOPE           Fez =  12.45 |
| (KL/R)max = 109.54              (WITH LOAD NO.)             FV  =  14.40 |
|                                                             fv  =   0.00 |
|                                                                          |
|                    MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
|                    -------------------------                             |
|                                                                          |
|              AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
|                                                                          |
|      VALUE     70.0         0.0        0.0         0.0         0.0       |
|    LOCATION     0.0         0.0        0.0         0.0         0.0       |
|     LOADING      1           0          0           0           0        |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                        DESIGN SUMMARY  (KIP-FEET)                      *|
|*                        --------------                                  *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/           *|
|         FX              MY            MZ           LOCATION              |
|       ======================================================            |
|          PASS        AISC- H1-1       0.944             1                |
|         70.00 C          0.00         0.00            0.00               |
|*                                                                        *|
|                                                                          |
|--------------------------------------------------------------------------|
```

V. AISC ASD - Beam Load Capacity 3

Determine the bending (transverse load carrying) capacity of a W10X45 shape.

Reference

AISC Allowable Stress Design, 9th Edition, Example 5, page 2-34.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

L_u = 6ft

Comparison

The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported STAAD.Pro critical ratio = *17/1.002 = 16.97*.

Table 778: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Allowable uniform load (kips/ft)	17	16.97	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Beam Load Capacity 3.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-34 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD EXAMPLE 5. GOVERNING CONDIION IS SHEAR.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W10X45
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -17
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
TRACK 2 ALL
SHEAR 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
```

```
|--------------------------------------------------------------------------|
|                                                    Y        PROPERTIES   |
|*************                                       |        IN INCH UNIT |
|            *  |=============================|   ===|===      ------------ |
|MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  13.30  |
|            *  | ST  W10X45                  |      |   --Z  AY =   3.54  |
|DESIGN CODE *  |                             |      |        AZ =   9.94  |
| AISC-1989  *  ===============================   ===|===      SY =  13.32  |
|            *                                                 SZ =  49.11  |
|            *  |&lt;---LENGTH (FT)=    6.00 --->|                   RY =
2.00  |
|*************                                                 RZ =   4.32  |
|                                                                          |
|                 76.5 (KIP-FEET)                                          |
|PARAMETER        |               L1  L1  L1                  STRESSES     |
|IN KIP  INCH     |                                            IN KIP  INCH |
|---------------  +           L1              L1               -------------|
| KL/R-Y=  35.93  |                                            FA  =  19.51 |
| KL/R-Z=  16.67  +       L1                       L1          fa  =   0.00 |
| UNL   =  72.00  |                                            FCZ =  23.76 |
| CB    =   1.00  +   L1                               L1      FTZ =  23.76 |
| CMY   =   0.85  |                                            FCY =  27.00 |
| CMZ   =   0.85  +                                            FTY =  27.00 |
| FYLD  =  36.00  |L0                                      L0  fbz =   0.00 |
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---|        fby =   0.00 |
| DFF   =   0.00  -4.2                                         Fey = 115.66 |
| dff=      0.00              ABSOLUTE MZ ENVELOPE             Fez = 537.14 |
| (KL/R)max =  35.93             (WITH LOAD NO.)               FV  =  14.40 |
|                                                              fv  =  14.43 |
|                                                                          |
|                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
|                     -------------------------                            |
|                                                                          |
|               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
|                                                                          |
|     VALUE       0.0        51.0        0.0         0.0        76.5       |
|   LOCATION      0.0         0.0        0.0         0.0         3.0       |
|    LOADING        0           1          0           0           1       |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                         DESIGN SUMMARY  (KIP-FEET)                     *|
|*                         --------------                                 *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX               MY             MZ          LOCATION             |
|        ======================================================            |
|          FAIL        SHEAR    -Y       1.002             1               |
|          0.00 T          0.00         0.00             0.00              |
|*                                                                        *|
|                                                                          |
|--------------------------------------------------------------------------|
```

V. AISC ASD - Wide Flange Beam Load Capacity 1

Determine the bending (transverse load carrying) capacity of a W16x40 shape.

Reference

AISC Allowable Stress Design, 9th Edition, Example1, page 2-5.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

L_u = 6ft

Comparison

The reported STAAD.Pro value is obtained by multiplying the FCZ and SZ values from the STAAD.Pro output = *23.76(64.71) = 1,537.5 in·kip = 128.1*.

Table 779: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Bending capacity (ft·kips)	128	128.1	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Wide Flange Beam Load Capacity 1.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-5 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD CODE EXAMPLE 1. RATIO SHOULD BE A LITTLE UNDER 1.0.
* CAPACITY SHOULD BE 128 KIP-FT. (SZ*FCZ SHOULD BE 128 KIP-FT).
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
```

```
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
FYLD 36 ALL
UNL 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES   |
   |*************                                       |        IN INCH UNIT |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  11.80  |
   |            *  | ST  W16X40                  |      |   --Z  AY =   4.39  |
   |DESIGN CODE *  |                             |      |        AZ =   4.73  |
   | AISC-1989  *  ===============================   ===|===     SY =   8.26  |
   |            *                                                SZ =  64.75  |
   |            *  |&lt;---LENGTH (FT)=   10.00 --->|                 RY =
1.56  |
   |*************                                                RZ =   6.63  |
   |                                                                          |
   |                125.0 (KIP-FEET)                                          |
   |PARAMETER        |L1                                         STRESSES     |
   |IN KIP  INCH     |   L1  L1                                  IN KIP  INCH |
   |---------------  +           L1                              -------------|
   | KL/R-Y=  76.68  |               L1                          FA   =  15.57 |
   | KL/R-Z=  18.11  +                                           fa   =   0.00 |
   | UNL    =  72.00  |                       L1                  FCZ =  23.76 |
   | CB     =   1.00  +                                           FTZ =  23.76 |
   | CMY    =   0.85  |                           L1  L1          FCY =  27.00 |
   | CMZ    =   0.85  +                                   L1      FTY =  27.00 |
   | FYLD   =  36.00  |                                       L0  fbz =  23.17 |
   | NSF    =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
   | DFF    =   0.00  -6.9                                        Fey =  25.40 |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE             Fez = 455.24 |
   | (KL/R)max =  76.68              (WITH LOAD NO.)              FV  =  14.40 |
   |                                                              fv  =   2.84 |
   |                                                                          |
   |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
   |                      -------------------------                           |
   |                                                                          |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                          |
   |      VALUE       0.0        12.5        0.0         0.0       125.0      |
   |    LOCATION      0.0         0.0        0.0         0.0         0.0      |
   |     LOADING        0           1          0           0           1      |
   |                                                                          |
   |**************************************************************************|
   |*                                                                        *|
```

```
|*                        DESIGN SUMMARY  (KIP-FEET)                        *|
|*                        --------------                                    *|
|*                                                                          *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/             *|
|         FX              MY              MZ          LOCATION              |
|        ======================================================              |
|          PASS        AISC- H1-3       0.975              1                 |
|          0.00 T          0.00      -125.00            0.00                 |
|*                                                                          *|
|                                                                            |
|----------------------------------------------------------------------------|
```

V. AISC ASD - Wide Flange Beam Load Capacity 2

Determine the bending (transverse load carrying) capacity of a W16x40 shape.

Reference

AISC Allowable Stress Design, 9th Edition, Example1, page 2-5.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

L_{uc} = 9ft (Unbraced length of compression flange)

Comparison

The reported STAAD.Pro value is obtained by multiplying the FCZ and SZ values from the STAAD.Pro output = $21.60(64.71) = 1397.7$ *in·kip* $= 116.5$ *ft·kips*.

Table 780: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Bending capacity (ft·kips)	116.5	116.5	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Wide Flange Beam Load Capacity 2.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-6 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD CODE EXAMPLE 2.
* CAPACITY SHOULD BE 116.5 KIP-FT. (SZ*FCZ SHOULD BE 116.5 KIP-FT).
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 9 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.94
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y         PROPERTIES  |
   |*************                                       |         IN INCH UNIT |
   |            *  |=============================|   ===|===      ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |         AX =  11.80 |
   |            *  | ST  W16X40                  |      |   --Z   AY =   4.39 |
   |DESIGN CODE *  |                             |      |         AZ =   4.73 |
   | AISC-1989  *  ===============================   ===|===      SY =   8.26 |
   |            *                                                 SZ =  64.75 |
   |            *  |&lt;---LENGTH (FT)=    9.00 --->|                   RY =
1.56  |
   |*************                                                 RZ =   6.63 |
   |                                                                          |
   |               116.5 (KIP-FEET)                                           |
   |PARAMETER        |L1                                          STRESSES    |
   |IN KIP  INCH     |   L1                                       IN KIP  INCH |
   |---------------  +       L1  L1                               ------------|
   | KL/R-Y=  69.01  |               L1                           FA  =  16.31 |
   | KL/R-Z=  16.30  +                   L1                       fa  =   0.00 |
   | UNL   = 108.00  |                       L1                   FCZ =  21.60 |
   | CB    =   1.00  +                                            FTZ =  21.60 |
   | CMY   =   0.85  |                           L1               FCY =  27.00 |
   | CMZ   =   0.85  +                               L1  L1       FTY =  27.00 |
   | FYLD  =  36.00  |                                       L0   fbz =  21.58 |
```

```
| NSF    =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
| DFF    =   0.00  -6.5                                        Fey =  31.36 |
| dff=       0.00               ABSOLUTE MZ ENVELOPE           Fez = 562.02 |
| (KL/R)max =  69.01              (WITH LOAD NO.)              FV  =  14.40 |
|                                                              fv  =   2.94 |
|                                                                           |
|                    MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                  |
|                    -------------------------                              |
|                                                                           |
|              AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z        |
|                                                                           |
|     VALUE       0.0        12.9        0.0         0.0       116.5        |
|   LOCATION      0.0         0.0        0.0         0.0         0.0        |
|    LOADING        0           1          0           0           1        |
|                                                                           |
|***************************************************************************|
|*                                                                         *|
|*                        DESIGN SUMMARY  (KIP-FEET)                       *|
|*                        --------------                                   *|
|*                                                                         *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/             *|
|         FX              MY             MZ          LOCATION              |
|        ======================================================            |
|          PASS        AISC- H1-3        0.999            1                |
|          0.00 T          0.00      -116.46            0.00               |
|*                                                                         *|
|                                                                           |
|---------------------------------------------------------------------------|
```

V. AISC ASD - MC Beam Load Capacity

Determine the bending (transverse load carrying) capacity of an MC18x58 channel. The beam is laterally supported along its entire length.

Reference

AISC Allowable Stress Design, 9th Edition, tables, page 2-80.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

L = 25ft

Comparison

The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported STAAD.Pro critical ratio $=43/0.995 = 43.2$.

Table 781: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Total allowable uniform load (kips/ft)	43	43.2	0.5%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - MC Beam Load Capacity.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-80 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD. LATERALLY SUPPORTED MC18X58 CHANNEL, 25FT LONG
* LOAD CAPACITY = 43 KIPS. RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST MC18X58
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
* LOAD INTENSITY = 43/25 = 1.72 KIP/FT
1 UNI GY -1.72
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |-----------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES      |
   |*************                                       |        IN INCH UNIT    |
   |            *  |=============================|    |===       -------------   |
   |MEMBER   1  *  |  AISC SECTIONS              |    |          AX =  17.10    |
   |            *  | ST  MC18X58                 |    |      --Z  AY =   9.98    |
   |DESIGN CODE *  |                             |    |          AZ =   3.20    |
   | AISC-1989  *  ===============================    |===       SY =   5.31    |
   |            *                                               SZ =  75.00    |
   |            *  |<---LENGTH (FT)=   25.00 --->|                  RY =
1.01  |
   |*************                                               RZ =   6.28    |
   |                                                                             |
   |                134.4 (KIP-FEET)                                             |
   |PARAMETER        |               L1  L1  L1                 STRESSES       |
   |IN KIP  INCH     |                                          IN KIP  INCH   |
   |---------------  +           L1              L1             -------------- |
   | KL/R-Y= 295.71  |                                          FA  =   1.71   |
   | KL/R-Z=  47.75  +       L1                       L1        fa  =   0.00   |
   | UNL   =   1.00  |                                          FCZ =  21.60   |
   | CB    =   1.00  +   L1                               L1    FTZ =  21.60   |
   | CMY   =   0.85  |                                          FCY =  21.60   |
   | CMZ   =   0.85  +                                          FTY =  21.60   |
   | FYLD  =  36.00  |L0                                     L0  fbz =  21.50   |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|  fby =   0.00   |
   | DFF   =   0.00  -7.5                                       Fey =   1.71   |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE           Fez =  65.50   |
   | (KL/R)max = 295.71             (WITH LOAD NO.)             FV  =  14.40   |
   |                                                            fv  =   0.00   |
   |                                                                             |
   |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                  |
   |                      -------------------------                              |
   |                                                                             |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z        |
   |                                                                             |
   |      VALUE       0.0        21.5        0.0         0.0       134.4         |
   |    LOCATION      0.0         0.0        0.0         0.0        12.5         |
   |     LOADING        0           1          0           0           1         |
   |                                                                             |
   |*****************************************************************************|
   |*                                                                           *|
   |*                          DESIGN SUMMARY  (KIP-FEET)                       *|
   |*                          --------------                                   *|
   |*                                                                           *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/               *|
   |         FX               MY              MZ          LOCATION              |
   |        ======================================================               |
   |          PASS        AISC- H1-3       0.995             1                   |
   |          0.00 T          0.00      -134.38           12.50                  |
   |*                                                                           *|
   |                                                                             |
   |-----------------------------------------------------------------------------|
```

V. AISC ASD - Wide Flange Beam Load Capacity 3

Determine the bending (transverse load carrying) capacity of a W16x45 shape.

Reference

AISC Allowable Stress Design, 9th Edition, tables, page 2-80.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

L =20ft

L_{uc} = 5ft (Unbraced length of compression flange)

Comparison

The reported STAAD.Pro value is obtained by dividing the applied load (using the table capacity) by the reported STAAD.Pro critical ratio $=58/1.008 = 57.54$.

Table 782: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Total allowable uniform load (kips)	58	57.54	<1%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Wide Flange Beam Load Capacity 3.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-34 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD EXAMPLE 4.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X45
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
```

```
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -2.9
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
TRACK 2 ALL
UNL 60 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES   |
   |*************                                       |        IN INCH UNIT |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  13.30  |
   |            *  | ST  W16X45                  |      |   --Z  AY =   4.98  |
   |DESIGN CODE *  |                             |      |        AZ =   5.31  |
   | AISC-1989  *  ===============================   ===|===     SY =   9.32  |
   |            *                                                SZ =  72.80  |
   |            *  |&lt;---LENGTH (FT)=   20.00 --->|                RY =
1.57  |
   |*************                                                RZ =   6.64  |
   |                                                                          |
   |                145.0 (KIP-FEET)                                          |
   |PARAMETER        |               L1  L1  L1                  STRESSES     |
   |IN KIP  INCH     |                                           IN KIP  INCH |
   |---------------  +           L1              L1              -------------|
   | KL/R-Y= 152.83  |                                           FA  =    6.39 |
   | KL/R-Z=  36.16  +       L1                       L1         fa  =    0.00 |
   | UNL   =  60.00  |                                           FCZ =  23.76 |
   | CB    =   1.00  +   L1                               L1     FTZ =  23.76 |
   | CMY   =   0.85  |                                           FCY =  27.00 |
   | CMZ   =   0.85  +                                           FTY =  27.00 |
   | FYLD  =  36.00  |L0                                     L0  fbz =  23.90 |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---|  fby =    0.00 |
   | DFF   =   0.00  -8.1                                        Fey =    6.39 |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE            Fez = 114.23 |
   | (KL/R)max = 152.83             (WITH LOAD NO.)              FV  =  14.40 |
   |                                                             fv  =    0.00 |
   |                                                                          |
   |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
   |                     -------------------------                            |
   |                                                                          |
```

```
 |                AXIAL      SHEAR-Y     SHEAR-Z     MOMENT-Y     MOMENT-Z   |
 |                                                                           |
 |      VALUE       0.0        29.0         0.0         0.0        145.0    |
 |   LOCATION       0.0         0.0         0.0         0.0         10.0    |
 |    LOADING         0           1           0           0            1    |
 |                                                                           |
 |***************************************************************************|
 |*                                                                         *|
 |*                         DESIGN SUMMARY  (KIP-FEET)                      *|
 |*                         --------------                                  *|
 |*                                                                         *|
 |*       RESULT/      CRITICAL COND/      RATIO/         LOADING/          *|
 |         FX                 MY             MZ           LOCATION           |
 |        ======================================================             |
 |          FAIL        AISC- H1-3        1.006              1               |
 |          0.00 T          0.00      -145.00             10.00              |
 |*                                                                         *|
 |                                                                           |
 |---------------------------------------------------------------------------|
```

V. AISC ASD - Select Wide Flange Beam 1

Determine the optimum W shape that spans 30 ft, and is braced at 7.5 ft intervals.

Reference

AISC Allowable Stress Design, 9th Edition,Example 6, page 2-35.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

Three concentrated loads of 20 kips each at quarter span points. Desired beam depth of 18 in.

F_y =36 ksi

Comparison

Table 783: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Selected section	W18x86	W18x86	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Select Wide Flange Beam 1.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-35 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
```

```
END JOB INFORMATION
* 9TH ED. AISC-ASD EXAMPLE 6.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X35
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 CON GY -20 7.5
1 CON GY -20 15
1 CON GY -20 22.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
DMAX 19 ALL
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 90 ALL
TRACK 2 ALL
SELECT OPTIMIZED
FINISH
```

STAAD Output

```
                     STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION)   v1.0
                     ***********************************************
  |-----------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES      |
  |*************                                       |        IN INCH UNIT    |
  |            *  |=============================|   ===|===     ------------   |
  |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  25.30    |
  |            *  | ST  W18X86                  |      |   --Z  AY =   7.95    |
  |DESIGN CODE *  |                             |      |        AZ =  11.38    |
  | AISC-1989  *  ===============================   ===|===     SY =  31.53    |
  |            *                                                SZ = 166.30    |
  |            *  |&lt;---LENGTH (FT)=   30.00 --->|                 RY =
2.63  |
  |*************                                                RZ =   7.78    |
  |                                                                             |
```

```
 |                300.0 (KIP-FEET)                                             |
 |PARAMETER        |                  L1                       STRESSES        |
 |IN KIP  INCH     |              L1      L1                   IN KIP  INCH    |
 |---------------  +          L1              L1               -------------   |
 | KL/R-Y= 136.88  |                                           FA   =   7.97   |
 | KL/R-Z=  46.29  +                              L1           fa   =   0.00   |
 | UNL    =  90.00 |      L1                                   FCZ =  23.76    |
 | CB     =   1.00 +                                           FTZ =  23.76    |
 | CMY    =   0.85 |  L1                              L1       FCY =  27.00    |
 | CMZ    =   0.85 +                                           FTY =  27.00    |
 | FYLD   =  36.00 |L0                                    L0   fbz =  21.65    |
 | NSF    =   1.00 +---+---+---+---+---+---+---+---+---|       fby =   0.00    |
 | DFF    =   0.00 -16.7                                       Fey =   7.97    |
 | dff=      0.00              ABSOLUTE MZ ENVELOPE            Fez =  69.68    |
 | (KL/R)max = 136.88             (WITH LOAD NO.)              FV   =  14.40   |
 |                                                             fv   =   1.26   |
 |                                                                             |
 |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                   |
 |                     -------------------------                               |
 |                                                                             |
 |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z        |
 |                                                                             |
 |      VALUE       0.0        30.0        0.0         0.0       300.0         |
 |    LOCATION      0.0         0.0        0.0         0.0        15.0         |
 |     LOADING        0           1          0           0           1         |
 |                                                                             |
 |*****************************************************************************|
 |*                                                                           *|
 |*                        DESIGN SUMMARY  (KIP-FEET)                         *|
 |*                        --------------                                     *|
 |*                                                                           *|
 |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/               *|
 |         FX              MY              MZ          LOCATION                |
 |        ======================================================               |
 |          PASS        AISC- H1-3       0.911             1                   |
 |          0.00 T          0.00      -300.00           15.00                  |
 |*                                                                           *|
 |                                                                             |
 |-----------------------------------------------------------------------------|
```

V. AISC ASD - Select Wide Flange Beam 2

Determine a 14in deep section to carry a load of 1 kip/ft over a span of 25 ft.

Reference

AISC Allowable Stress Design, 9th Edition,Example 7, page 2-35.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

The beam is laterally supported throughout its length.

F_y =36 ksi

Comparison

Table 784: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Selected section	W14x30	W14x30	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Select Wide Flange Beam 2.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-35 OF
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* 9TH ED. AISC-ASD EXAMPLE 7. LATERALLY SUPPORTED BEAM.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
* TRIAL SECTION SIZE:
1 TABLE ST W14X22
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
DMAX 15 ALL
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 1 ALL
TRACK 2 ALL
*CHECK CODE ALL
```

```
SELECT OPTIMIZED
FINISH
```

STAAD Output

```
                     STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION)   v1.0
                     ***********************************************
   |-----------------------------------------------------------------------------|
   |                                                   Y         PROPERTIES      |
   |*************                                      |         IN INCH UNIT    |
   |            *  |=============================|   ===|===      ------------   |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =   8.85     |
   |            *  | ST  W14X30                  |      |   --Z  AY =   3.40     |
   |DESIGN CODE *  |                             |      |        AZ =   3.47     |
   | AISC-1989  *  ===============================   ===|===      SY =   5.82     |
   |            *                                                SZ =  42.17     |
   |            *  |&lt;---LENGTH (FT)=   25.00 --->|               RY =
1.49  |
   |*************                                                RZ =   5.73     |
   |                                                                             |
   |                  78.1 (KIP-FEET)                                            |
   |PARAMETER        |               L1  L1  L1                  STRESSES        |
   |IN KIP  INCH     |                                           IN KIP  INCH    |
   |---------------  +           L1              L1              -------------|
   | KL/R-Y= 201.59  |                                           FA  =   3.67    |
   | KL/R-Z=  52.32  +       L1                       L1         fa  =   0.00    |
   | UNL   =   1.00  |                                           FCZ =  23.76    |
   | CB    =   1.00  +   L1                               L1     FTZ =  23.76    |
   | CMY   =   0.85  |                                           FCY =  27.00    |
   | CMZ   =   0.85  +                                           FTY =  27.00    |
   | FYLD  =  36.00  |L0                                     L0  fbz =  22.23    |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|  fby =   0.00    |
   | DFF   =   0.00  -4.3                                      Fey =   3.67    |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE          Fez =  54.56    |
   | (KL/R)max = 201.59             (WITH LOAD NO.)            FV  =  14.40    |
   |                                                           fv  =   0.00    |
   |                                                                             |
   |                       MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
   |                       -------------------------                             |
   |                                                                             |
   |                 AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
   |                                                                             |
   |      VALUE        0.0        12.5        0.0         0.0        78.1        |
   |    LOCATION       0.0         0.0        0.0         0.0        12.5        |
   |     LOADING         0           1          0           0           1        |
   |                                                                             |
   |*****************************************************************************|
   |*                                                                           *|
   |*                          DESIGN SUMMARY  (KIP-FEET)                       *|
   |*                          --------------                                   *|
   |*                                                                           *|
   |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/              *|
   |         FX               MY             MZ          LOCATION                |
   |        ======================================================               |
   |          PASS       AISC- H1-3       0.936              1                   |
   |          0.00 T          0.00       -78.12           12.50                  |
   |*                                                                           *|
```

```
|                                                                              |
|------------------------------------------------------------------------------|
```

V. AISC ASD - Select Wide Flange Beam 3

Determine an optimum section to carry a moment of 220 ft·kips plus its selfweight and spanning 35 ft between supports.

Reference

AISC Allowable Stress Design, 9th Edition,Example 10, page 2-147.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

F_y =36 ksi

C_b = 1.0

Maximum unbraced length of compression flange is 15 ft.

Comparison

Table 785: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Selected section	W24x68	W24x68	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Select Wide Flange Beam 3.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-147
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* OF 9TH ED. AISC-ASD. EXAMPLE 10.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
*TRIAL SECTION
1 TABLE ST W18X35
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
```

```
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
SELFWEIGHT Y -1
MEMBER LOAD
1 CON GY -22 10
1 CON GY -22 25
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
MAIN 1 ALL
BEAM 1 ALL
FYLD 36 ALL
UNL 180 ALL
TRACK 2 ALL
*CHECK CODE ALL
SELECT OPTIMIZED
FINISH
```

STAAD Output

```
                     STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION)   v1.0
                     ***********************************************
   |-----------------------------------------------------------------------|
   |                                                  Y        PROPERTIES  |
   |*************                                     |        IN INCH UNIT |
   |            *  |==============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS               |      |        AX =  20.10 |
   |            *  | ST  W24X68                   |      |   --Z  AY =   8.73 |
   |DESIGN CODE *  |                              |      |        AZ =   7.01 |
   | AISC-1989  *  ===============================   ===|===     SY =  15.70 |
   |            *                                                SZ = 154.43 |
   |            *  |<---LENGTH (FT)=   35.00 --->|                  RY =
1.87  |
   |*************                                                RZ =   9.54  |
   |                                                                        |
   |                232.7 (KIP-FEET)                                         |
   |PARAMETER       |               L1  L1  L1                  STRESSES     |
   |IN KIP  INCH    |           L1               L1             IN KIP  INCH |
   |---------------  +                                          -------------|
   | KL/R-Y= 224.42  |                                          FA   =   2.97 |
   | KL/R-Z=  44.02  +       L1                       L1        fa   =   0.00 |
   | UNL    = 180.00  |                                         FCZ =  18.70 |
   | CB     =   1.00  +   L1                              L1    FTZ =  21.60 |
```

```
 | CMY    =   0.85  |                                          FCY =  27.00 |
 | CMZ    =   0.85  +                                          FTY =  27.00 |
 | FYLD   =  36.00  |L0                                    L0  fbz =  18.08 |
 | NSF    =   1.00  +---+---+---+---+---+---+---+---+---|      fby =   0.00 |
 | DFF    =   0.00 -12.9                                       Fey =   2.97 |
 | dff=       0.00                ABSOLUTE MZ ENVELOPE         Fez =  77.07 |
 | (KL/R)max = 224.42               (WITH LOAD NO.)            FV  =  14.40 |
 |                                                             fv  =   0.00 |
 |                                                                          |
 |                    MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
 |                    -------------------------                             |
 |                                                                          |
 |              AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
 |                                                                          |
 |      VALUE       0.0         23.5        0.0         0.0       232.7     |
 |    LOCATION      0.0          0.0        0.0         0.0        17.5     |
 |     LOADING        0            1          0           0           1     |
 |                                                                          |
 |**************************************************************************|
 |*                                                                        *|
 |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
 |*                        --------------                                  *|
 |*                                                                        *|
 |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/           *|
 |         FX              MY             MZ           LOCATION             |
 |        ======================================================           |
 |          PASS        AISC- H1-3        0.967              1              |
 |          0.00 T          0.00      -232.69            17.50              |
 |*                                                                        *|
 |                                                                          |
 |--------------------------------------------------------------------------|
    AISC VERIFICATION PROBLEM FOR BENDING CAPACITY. PAGE 2-1 -- PAGE NO.
4
    48. FINISH

***************************************************************************
   **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
               IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
               ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
               TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
               FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
               FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
```

V. AISC ASD - Compression and Biaxial Bending

Determine the suitability of a W14x109 shape to resist axial compression and biaxial bending.

Reference

AISC Allowable Stress Design, 9th Edition,Example 4, page 3-11.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

P =200 kips

M_z = 120 ft ·kips (strong axis)

M_y = 40 ft ·kips (weak axis)

KL = 14 ft

C_m= 0.85

Maximum unbraced length of compression flange is 15 ft.

Comparison

Table 786: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Selected section	OK	OK (stress ratio = 0.93)	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Compression and Biaxial Bending.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR COMBINED AXIAL + BENDING.
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
* PAGE 3-11 OF 9TH ED. AISC-ASD. EXAMPLE 4.
* RATIO SHOULD BE APPROXIMATELY 1.0
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 14 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X109
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 50 FU 62 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_50_KSI ALL
SUPPORTS
1 FIXED
```

```
LOAD 1
JOINT LOAD
2 FY -200
2 MZ 120
2 MX 40
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISC
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                     Y        PROPERTIES  |
  |*************                                        |        IN INCH UNIT|
  |            *  |=============================|    ===|===     ------------|
  |MEMBER   1  *  |  AISC SECTIONS              |       |        AX =  32.00 |
  |            *  | ST  W14X109                 |       |    --Z AY =   6.87 |
  |DESIGN CODE *  |                             |       |        AZ =  16.80 |
  | AISC-1989  *  ===============================    ===|===     SY =  61.23 |
  |            *                                                 SZ = 173.43 |
  |            *  |<---LENGTH (FT)=   14.00 --->|                   RY =
3.74  |
  |*************                                                 RZ =   6.22 |
  |                                                                          |
  |               120.0 (KIP-FEET)                                           |
  |PARAMETER        |                            L1  L1  L1  STRESSES        |
  |IN KIP  INCH     |                            L1  L1  L1  IN KIP  INCH    |
  |---------------  +                            L1  L1  L1  -------------   |
  | KL/R-Y=  44.95  |                            L1  L1  L1  FA  =  18.78    |
  | KL/R-Z=  26.99  +                            L1  L1  L1  fa  =   6.25    |
  | UNL   = 168.00  |L1 L1  L1  L1  L1  L1  L1  L1            FCZ =  23.76    |
  | CB    =   1.00  +L1 L1  L1  L1  L1  L1  L1  L1            FTZ =  23.76    |
  | CMY   =   0.85  |L1 L1  L1  L1  L1  L1  L1  L1            FCY =  27.00    |
  | CMZ   =   0.85  +L1 L1  L1  L1  L1  L1  L1  L1            FTY =  27.00    |
  | FYLD  =  36.00  |L1 L1  L1  L1  L1  L1  L1  L1            fbz =   8.30    |
  | NSF   =   1.00  +---+---+---+---+---+---+---+---+---|   fby =   7.84    |
  | DFF   =   0.00 120.0                                     Fey =  73.91    |
  | dff=      0.00              ABSOLUTE MZ ENVELOPE         Fez = 205.02    |
  | (KL/R)max =  44.95             (WITH LOAD NO.)           FV  =  14.40    |
  |                                                          fv  =   0.00    |
  |                                                                          |
  |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
  |                     -------------------------                            |
  |                                                                          |
  |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
```

```
    |                                                                      |
    |     VALUE     200.0          0.0         0.0        40.0       120.0  |
    |  LOCATION       0.0          0.0         0.0         0.0         0.0  |
    |   LOADING         1            0           0           1           1  |
    |                                                                      |
    |**********************************************************************|
    |*                                                                    *|
    |*                        DESIGN SUMMARY  (KIP-FEET)                  *|
    |*                        --------------                              *|
    |*                                                                    *|
    |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/         *|
    |         FX                MY            MZ          LOCATION         |
    |        ======================================================        |
    |          PASS        AISC- H1-2       0.929              1           |
    |        200.00 C          40.00      -120.00            0.00          |
    |*                                                                    *|
    |                                                                      |
    |----------------------------------------------------------------------|
       AISC VERIFICATION PROBLEM FOR COMBINED AXIAL + BENDING.  -- PAGE NO.
6
    47. FINISH
```

V. AISC ASD - Angle in Compression

Determine the allowable compressive load on a single L2x2x1/4 loaded by a gusset plate attached to one leg with eccentricities from the centroid along both axes.

Reference

AISC Allowable Stress Design, 9th Edition,Example 10, page 3-55.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

KL = 40 in

Comparison

Table 787: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Allowable compressive load (kips)	4.5	Using a P of 4.5 kip yields a ratio of 0.898	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - Angle in Compression.STD` is typically installed with the program.

```
STAAD SPACE AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIAL BENDING
* PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGLE 2X2X1/4
* UNBRACED LENGTH IS 40 INCHES.
UNIT KIP FEET
JOINT COORD
1 0 0 0 ; 2 0 5 0
MEMB INCI
1 1 2
UNIT INCH
MEMB PROP AMERICAN
1 TA ST L20204
CONST
E STEEL ALL
POISS STEEL ALL
SUPP
1 FIXED
LOAD 1
JOINT LOAD
2 FY -4.5
* MZ = P * 0.277 = 1.2465 KIP-INCH
* MX = P * 0.854 = 3.843 KIP-INCH
2 MZ 1.2465
2 MX -3.843
PERF ANALY
UNIT INCH
PARAM
CODE AISC
FYLD 50 ALL
UNT 40 ALL
UNB 40 ALL
LY 40 ALL
LZ 40 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                   Y        PROPERTIES    |
  |*************                                      |        IN INCH UNIT  |
  |            *  |=============================|   ==| |==     ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |     | |       AX =   0.94  |
  |            *  | ST  L20204                  |     | |  --Z  AY =   0.33  |
  |DESIGN CODE *  |                             |     | |       AZ =   0.33  |
  | AISC-1989  *  ===============================   ==| |==     SY =   0.39  |
  |            *                                                SZ =   0.17  |
  |            *  |&lt;---LENGTH (FT)=    5.00 --->|                 RY =
0.77  |
```

```
   |*************                                              RZ =   0.39  |
   |                                                                        |
   |                  0.1 (KIP-FEET)                                        |
   |PARAMETER        |                              L1  L1  L1  STRESSES     |
   |IN KIP  INCH     |                              L1  L1  L1  IN KIP  INCH |
   |---------------  +                              L1  L1  L1  -------------|
   | KL/R-Y=  52.22  |                              L1  L1  L1  FA  =  13.92 |
   | KL/R-Z= 103.36  +                              L1  L1  L1  fa  =   4.77 |
   | UNL   =  40.00  |L1 L1  L1  L1  L1  L1  L1  L1             FCZ =  33.00 |
   | CB    =   1.00  +L1 L1  L1  L1  L1  L1  L1  L1             FTZ =  33.00 |
   | CMY   =   0.85  |L1 L1  L1  L1  L1  L1  L1  L1             FCY =  33.00 |
   | CMZ   =   0.85  +L1 L1  L1  L1  L1  L1  L1  L1             FTY =  33.00 |
   | FYLD  =  50.00  |L1 L1  L1  L1  L1  L1  L1  L1             fbz =   7.11 |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|  fby =   9.89 |
   | DFF   =   0.00   0.1                                       Fey =  54.76 |
   | dff=      0.00             ABSOLUTE MZ ENVELOPE            Fez =  13.98 |
   | (KL/R)max = 103.36            (WITH LOAD NO.)              FV  =  20.00 |
   |                                                            fv  =   0.00 |
   |                                                                        |
   |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)             |
   |                      -------------------------                         |
   |                                                                        |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z    |
   |                                                                        |
   |      VALUE       4.5         0.0        0.0         0.3         0.1     |
   |   LOCATION       0.0         0.0        0.0         0.0         0.0     |
   |    LOADING         1           0          0           1           1     |
   |                                                                        |
   |**************************************************************************|
   |*                                                                      *|
   |*                         DESIGN SUMMARY  (KIP-FEET)                    *|
   |*                         --------------                                *|
   |*                                                                      *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/           *|
   |         FX              MY              MZ          LOCATION            |
   |        ======================================================          |
   |          PASS        AISC- H1-1       0.899             1               |
   |          4.50 C         -0.32        -0.10            0.00              |
   |*                                                                      *|
   |                                                                        |
   |--------------------------------------------------------------------------|
      AISC VERIFICATION PROBLEM FOR AXIAL COMPRESSION + BIAXIA -- PAGE NO.
4
    * PAGE 3-55 OF 9TH ED. AISC-ASD. SINGLE ANGL
    35. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME= 10: 4:42 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
```

V. AISC ASD - 2D Frame Validation

Determine the allowable stresses (per 1989 AISC code) for the members of the structure as shown in the figure. Also, perform a code check for these members based on the results of the analysis.

Reference

Attached step by step hand calculation as per 1989 AISC ASD code. Ninth Edition.

Related Links

- *D1.B.1.1 Allowables per AISC Code* (on page 1130)

Problem

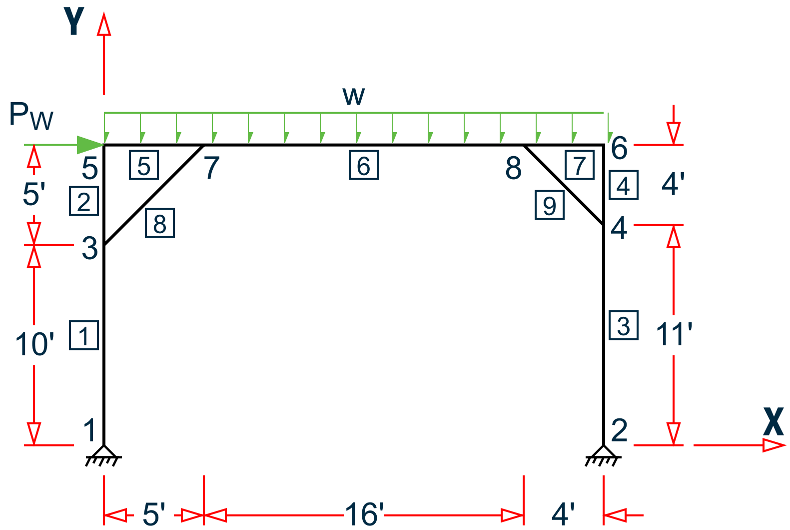


Figure 502: Steel braced frame

Members 1, 2 = W12X26, Members 3, 4 = W14X43

Members 5, 6, 7 = W16X36, Memb 8= L40404, Memb 9 = L50506

F = 15 kip (wind load)

w = 2 kip/ft (dead load + live load)

Steel: Fy = 36ksi

Hand Calculation

Manual / Code refers to AISC Manual of Steel Construction, Allowable Stress Design, ninth edition.

Member 1

Size W12x26, L = 10 ft., a = 7.65 in^2, S_z = 33.39 in^3

Per Section F2 is the lower value for the yielding and lateral-torsional buckling limit states:

Mn = Mp = FyZx = 36ksi × 37.2 in^3 = 1,339 in·k = 111.3 ft·k

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76(1.51\text{in})\sqrt{\frac{29,000\text{ksi}}{36\text{ksi}}} = 75.4\text{in}$$

From clause F1-2, page 5-45 of Manual, L_c = 6.85 ft.

From observation Load case 1 will govern,

F_x = 25.0 kip (compression), M_z = 56.5 ft·kip

Area of compressive flange = 6.49*0.38 = 2.466 in^2

Allowable bending stress, per Clause F1-8 (page 5-47 of Manual)

$$f_b = 12. \cdot 1000 \cdot 1.0/[10 \cdot 12 \cdot (12.22/2.466)] = 20.1817 \text{ ksi}$$

$$(kl/r)y = 120/1.5038 = 79.8, \text{ so } f_a = 15.38 \text{ ksi}$$

Stresses per Table C-36 (page 3-16 of manual)

$$f_a = 25./7.65 = 3.268$$

$$f_b = 56.5 \cdot 12/33.39 = 20.31 \text{ ksi}$$

$$(kl/r)z = 120/5.1639 = 23.238, \text{ so } F'_{ez} = = 276.54 \text{ ksi}$$

Equation H1-1, page 5-54 of Manual

$$3.268/15.38 + 0.85 \cdot 20.31 / [(1-3.268/276.54) \cdot 20.1817] = 1.078$$

Equation H1-2, page 5-54 of Manual.

$$3.268/(0.6 \cdot 36) +20.31/20.1817 = 1.1576$$

Therefore, equation H1-2 governs and ratio = 1.1576

Member 2

Size W12x26, L = 5 ft., a = 7.65 in^2, Sz = 33.39 in^3

From observation load case 1 will govern,

F_x = 8.71 kip (compression), M_z = 56.50 ft·kip

Since L is less than L_c = 6.85ft, per Clauses F1-1 & F1-2 (page 5-45 of Manual)

$$f_b = 0.66 \cdot 36 = 23.76 \text{ ksi}$$

$$(kl/r)y = 60/1.5038 = 39.90, \text{ so } f_a = 19.19 \text{ ksi}$$

Per Table C-36 (page 3-16 of Manual)

$$f_a = 8.71/7.65 = 1.1385, f_b = 56.5 \cdot 12/33.39 = 20.31 \text{ ksi}$$

Since f_a/F_a less than 0.15, apply equation H1-3 (page 5-54 of Manual)

$$1.1385/19.19 + 20.31/23.76 = .0593 + .8548 = 0.9141$$

Member 3

Size W14X43, L = 11ft., a = 12.6 in^2, Sz = 62.7 in^3

From observation load case 3 will govern,

F_x = 25.5 kip (compression), M_z = 112.173 ft·kip

Referring to clause F1-2, page 5-45 of Manual.

L_c = 8.4 ft. Therefore

$f_b = 0.6 \cdot 36 = 21.6$ ksi

$(kl/r)y = 132/1.8941 = 69.69$,

so f_a = 16.46 ksi per Table C-36 (page 3-16 of Manual)

$f_a = 25.5/12.6 = 2.024$

$f_b = 112.173 \cdot 12/62.66 = 21.48$ ksi

since f_a/F_a less than 0.15, use equation H1-3, page 5-54 of Manual

$2.024/16.46 + 21.48/21.6 = 0.123 + 0.994 = 1.117$

Member 4

Size W14x43, L = 4ft, a = 12.6 in^2, Sz = 62.7 in^3

From observation, load case 3 will govern,

F_x = 8.75 kip (tension), M_z = 112.173 ft·kip

Since L is less than L_c = 8.4 ft

$f_b = 0.66 \cdot 36 = 23.76$ ksi

Clause F1-1 (page 5-45 of Manual)

$f_a = 8.75/12.6 = 0.694$

$f_b = 112.73x12/62.66 = 21.48$ ksi

Combined tension and bending, use equation H2-1, page 5-55 of Manual.

$0.694/(0.6 \cdot 36) + 21.48/23.76 = 0.032 + 0.904 = 0.936$

Member 5

Size W16x36, L = 5ft, a = 10.6 in^2, Sz = 56.49 in^3

From observation, load case 3 will govern.

F_x = 14.02 kip (compression), M_z =57.04 ft·kip

Since L is less than L_c = 7.37 (per Clause F1-2, page 5-45 of Manual)

$f_b = 0.66 \cdot 36 = 23.76$ ksi

$(kl/r)y = 60./1.52 = 39.47$

so f_a = 19.23 ksi per Table C-36 (page 3-16 of Manual)

$f_a = 14.02/10.6 = 1.32$

$f_b = 57.04 \cdot 12/56.5 = 12.12$ ksi

Since f_a/F_a less than 0.15, use equation H1-3, page 5-54 of Manual

$1.32/19.23 + 12.12/23.76 = 0.069 + 0.510 = 0.579$

Member 6

Size W16x36, L = 16ft, a = 10.6 in^2, Sz = 56.49 in^3

From observation, load case 1 will govern. Forces at midspan are

F_x = 5.65 kip (compression), M_z = 71.25 ft·kip

From Chapter F of the AISC ASD 9P$^{th\ P}$ ed. specs., with Cb = 1.0,

$(102{,}000Cb/Fy)^{1/2}$ =53.229

$(510{,}000Cb/Fy)^{1/2}$ =119.02

L/rT = 192/1.79 = 107.26

53.229 < 107.26 < 119.02

Therefore F_b (as per F1-6, page 5-47 of Manual)

[(2/3) – 36 · 107.26 · 107.26/(1,530,000)] · 36 = 14.25 ksi

(Kl/r)y = 192/1.5203 = 126.29

so f_a = 9.36 per Table C-36 (page 3-16 of Manual)

f_a = 5.65/10.6 = 0.533

f_b = 71.25x12/56.49 = 15.14 ksi

Since f_a/F_a less than 0.15 use formula [H1-3, page 5-54 of Manual]

0.533/9.36 + 15.14/14.25 = 0.057 + 1.062 = 1.119

Member 7

Size W16x36, L =4ft, a = 10.6in^2, Sz =56.49in^3

L_c = 7.37ft (Clause F1-2 page 5-45 of Manual)

From observation load case 3 will govern, F_x = 24.06 kip (tension), M_z = 62.96 ft·kip

From Clause F1-1, the allowable compressive stress is

f_b = 0.66 Fy = 23.76 ksi

Since section is in tension, per Clause F1-5 (page5-45 of Manual)

f_b = 0.60 · 36 = 21.60 Ksi

Choosing the larger of above 2 values, f_b = 23.76 Ksi

f_a = 24.06/10.6 = 2.2698

f_b = 62.96 · 12/56.49 = 13.37

Since combined tension and bending, use equation H 2-1, page 5-55 of the AISC ASD 9P$^{th\ P}$ ed. specs.

2.2698/(0.6 · 36) + 13.37(23.76) = 0.105 + 0.5627 = 0.6677

Member 8

Size L4x4x1/4, L = 7.071 ft, a = 1.94 in^2

From observation load case 1 will govern, F_x = 23.04 kip (Comp.)

Fa is computed as per page 5-310 of the AISC ASD 9P$^{th\ P}$ ed.specs.

$Q_s = 1.34 – 0.00447 \cdot (4/0.25) \cdot (36)^{1/2} = 0.9108$

Q_a = 1.0

$Q = Q_s \cdot Q_a = 0.9108$

$C_c = 92.0 \cdot \pi^2 \cdot E/(Q \cdot F_y)]^{1/2} = [2.0 \cdot \pi^2 \cdot 29000/(0.9108 \cdot 36)]^{1/2} = 132.1241$

$Kl/r = 7.071 \cdot 12/0.795 = 106.73 < C_c$.

Hence, f_a = 11.6027 ksi (computed per equation 4-1)

Actual compressive stress

$f_a = 23.04/1.94 = 11.876$ ksi

Therefore, Ratio =

$f_a/f_a = 11.876/11.602 = 1.024$

Member 9

Size L5x5x3/8, L = 5.657 ft, a = 3.61 in^2

From observation, load case 1 governs, F_x = 48.44 kip (Comp.)

Fa is computed as per page 5-310 of the AISC ASD 9P$^{th\ P}$ ed. specs.

$Q_s = 1.34 – 0.00447 \cdot (5/0.375) \cdot (36)^{1/2} = 0.9824$

$Q_a = 1.0$

$Q = Q_s \cdot Q_a = 0.9824$

$C_c = 92.0 \cdot \pi^2 \cdot E/(Q \cdot F_y)]^{1/2} = [2.0 \cdot \pi^2 \cdot 29000/(0.9824 \cdot 36)]^{1/2} = 127.2238$

$(Kl/r)min = 5.657 \cdot 12/0.99 = 68.57 < C_c$.

Hence, f_a = 16.301 ksi (computed per equation 4-1)

Actual compressive stress

$f_a = 48.44/3.61 = 13.418$ ksi

Therefore Ratio =

$f_a/f_a = 13.418/16.301 = 0.823$

Comparison

Governing stress ratio at member no.	Hand Calculation	STAAD.Pro	Difference
1	1.158	1.154	negligible
2	0.914	0.913	negligible
3	1.117	1.121	negligible
4	0.936	0.939	negligible
5	0.579	0.580	negligible
6	1.119	1.107	1.1%

Governing stress ratio at member no.	Hand Calculation	STAAD.Pro	Difference
7	0.668	0.670	negligible
8	1.024	1.045	2.1%
9	0.823	0.815	1.0%

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC ASD - 2D Frame validation.STD` is typically installed with the program.

```
STAAD PLANE VERIFICATION PROBLEM NO 13
START JOB INFORMATION
ENGINEER DATE 22-Sep-18
END JOB INFORMATION
*
*  THIS DESIGN EXAMPLE IS VERIFIED BY HAND CALCULATION
*  FOLLOWING AISC-89 CODE.
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0; 3 0 10 0; 4 25 11 0; 5 0 15 0; 6 25 15 0; 7 5 15 0;
8 21 15 0;
MEMBER INCIDENCES
1 1 3; 2 3 5; 3 2 4; 4 4 6; 5 5 7; 6 7 8; 7 8 6; 8 3 7; 9 4 8;
MEMBER PROPERTY AMERICAN
1 2 TABLE ST W12X26
3 4 TABLE ST W14X43
5 TO 7 TABLE ST W16X36
8 TABLE ST L40404
9 TABLE ST L50506
MEMBER TRUSS
8 9
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
LOAD 1 DL + LL
MEMBER LOAD
5 TO 7 UNI Y -2
LOAD 2 WIND FROM LEFT
JOINT LOAD
5 FX 15
LOAD COMBINATION 3
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
```

```
PRINT MEMBER FORCES
PARAMETER 1
CODE AISC
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (AISC 9TH EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  ST   W12X26                   (AISC SECTIONS)
                           FAIL     AISC- H1-2         1.154         1
                      25.00 C          0.00          56.50       10.00
  -----------------------------------------------------------------------
 | MEM=     1, UNIT KIP-INCH, L= 120.0 AX=   7.65 SZ=    33.4 SY=     5.3|
 | KL/R-Y=  79.8  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 20.21 |
 | FTZ= 21.60  FCY= 27.00  FTY= 27.00  FC= 15.38  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
      2  ST   W12X26                   (AISC SECTIONS)
                           PASS     AISC- H1-3         0.913         1
                       8.72 C          0.00          56.50        0.00
  -----------------------------------------------------------------------
 | MEM=     2, UNIT KIP-INCH, L=  60.0 AX=   7.65 SZ=    33.4 SY=     5.3|
 | KL/R-Y=  39.9  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 23.76 |
 | FTZ= 23.76  FCY= 27.00  FTY= 27.00  FC= 19.20  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
 *    3  ST   W14X43                   (AISC SECTIONS)
                           FAIL     AISC- H1-3         1.121         3
                      25.50 C          0.00        -112.20       11.00
  -----------------------------------------------------------------------
 | MEM=     3, UNIT KIP-INCH, L= 132.0 AX=  12.60 SZ=    62.5 SY=    11.3|
 | KL/R-Y=  69.7  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 21.60 |
 | FTZ= 21.60  FCY= 27.00  FTY= 27.00  FC= 16.46  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
      4  ST   W14X43                   (AISC SECTIONS)
                           PASS     AISC- H2-1         0.939         3
                       8.83 T          0.00        -112.20        0.00
  -----------------------------------------------------------------------
 | MEM=     4, UNIT KIP-INCH, L=  48.0 AX=  12.60 SZ=    62.5 SY=    11.3|
 | KL/R-Y=  25.3  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 23.76 |
 | FTZ= 23.76  FCY= 27.00  FTY= 27.00  FC= 19.85  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
      5  ST   W16X36                   (AISC SECTIONS)
                           PASS     AISC- H1-3         0.580         3
                      14.02 C          0.00         -57.00        5.00
      VERIFICATION PROBLEM NO 13                               -- PAGE NO.
6
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
  -----------------------------------------------------------------------
```

```
 | MEM=     5, UNIT KIP-INCH, L=  60.0 AX=  10.60 SZ=    56.4 SY=     7.0|
 | KL/R-Y=  39.5  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 23.76 |
 | FTZ= 23.76  FCY= 27.00  FTY= 27.00  FC= 19.23  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
 *    6  ST   W16X36                   (AISC SECTIONS)
                            FAIL     AISC- H1-3         1.107         1
                         5.65 C         0.00        -71.25        8.00
  -----------------------------------------------------------------------
 | MEM=     6, UNIT KIP-INCH, L= 192.0 AX=  10.60 SZ=    56.4 SY=     7.0|
 | KL/R-Y= 126.3  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 14.45 |
 | FTZ= 21.60  FCY= 27.00  FTY= 27.00  FC=  9.36  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
      7  ST   W16X36                   (AISC SECTIONS)
                            PASS     AISC- H2-1         0.670         3
                        24.13 T         0.00         63.00        0.00
  -----------------------------------------------------------------------
 | MEM=     7, UNIT KIP-INCH, L=  48.0 AX=  10.60 SZ=    56.4 SY=     7.0|
 | KL/R-Y=  31.6  CB=  1.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ= 23.76 |
 | FTZ= 23.76  FCY= 27.00  FTY= 27.00  FC= 18.88  FT= 21.60  FV= 14.40   |
  -----------------------------------------------------------------------
 *    8  ST   L40404                   (AISC SECTIONS)
                            FAIL     AISC- H1-1         1.045         1
                        23.03 C         0.00          0.00        0.00
  -----------------------------------------------------------------------
 | MEM=     8, UNIT KIP-INCH, L=  84.9 AX=   1.93 SZ=     0.8 SY=     1.7|
 | KL/R- = 108.4  CB=  0.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ=  0.00 |
 | FTZ=  0.00  FCY=  0.00  FTY=  0.00  FC= 11.42  FT= 21.60  FV=  0.00   |
  -----------------------------------------------------------------------
      9  ST   L50506                   (AISC SECTIONS)
                            PASS     AISC- H1-1         0.815         3
                        48.55 C         0.00          0.00        0.00
      VERIFICATION PROBLEM NO 13                                 -- PAGE NO.
7
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
  -----------------------------------------------------------------------
 | MEM=     9, UNIT KIP-INCH, L=  67.9 AX=   3.65 SZ=     1.8 SY=     3.9|
 | KL/R- =  68.8  CB=  0.00  YLD= 36.00  ALLOWABLE STRESSES:  FCZ=  0.00 |
 | FTZ=  0.00  FCY=  0.00  FTY=  0.00  FC= 16.32  FT= 21.60  FV=  0.00   |
  -----------------------------------------------------------------------
```

V. AISC ASD - Design of Steel Beam with Web Opening

Design the web reinforcement for a web opening in a steel beam.

Details

Note: The design at locations where openings are not present is not shown in this example.

Design Method : ASD

Beam No : 5

Beam Section : W21X50

d = 20.83 in
t_w = 0.38 in
b_f = 6.53 in
t_f = 0.535 in
Z = 110 in^3
L = 10 ft

Weld Properties : E90XX

F_y,weld = 90 ksi

Weld Stress = 54 ksi

Opening Type: Rectangular

Number of openings : 1

Section location of Opening : 0.6
a_0 = 20 in
h_0 = 10 in
e = 0 in (concentric opening)
F_y = 36 ksi

Capacity check at web hole assuming an unreinforced opening:

Loading :

V_u = 34.46 kip
M_u = 2,510.7 kip-inch

Tee Properties :

s_t = 5.415 in
s_b = 5.415 in

Validation

Check for local buckling of compression flange :

width to thickness ratio $F_1 = \frac{b_f}{2t_f} = 6.1028$

limiting ratio $= \frac{65}{\sqrt{F_y}} = 86.667 > F_1$ Hence O.K

Check for web buckling :

width to thickness ratio of web $W_1 = \frac{\left(d - 2t_f\right)}{t_w} = 52$

limiting ratio $= \frac{520}{\sqrt{F_y}} = 86.667 > W_1$ Hence O.K

Check for opening dimensions to prevent web buckling :

Since $W_1 \le \frac{420}{\sqrt{F_y}} = 70$, $a_0 / h_0 = 2 < 3.0$

$h_0 / d = 0.4801 < 0.7$

Opening parameter $p_0 = \frac{a_0}{h_0} + \frac{6h_0}{d} = 4.8805 < 5.6$ Hence OK

Check for Tee dimension :

$$s_t = \frac{d}{2} - \left(\frac{h_0}{2} + e\right) = 5.415\text{in.} \ge 0.15d = 3.1245\text{in.}$$

$$s_b = \frac{d}{2} - \left(\frac{h_0}{2} - e\right) = 5.415\text{in.} \ge 0.15d = 3.1245\text{in.}$$

Aspect Ratio:

$v_t = \frac{a_0}{s_t} = 3.6934 < 12.0$,

$v_b = \frac{a_0}{s_b} = 3.6934 < 12.0$ Hence OK

Calculation of Maximum Moment Capacity :

For unperforated section $M_p = F_y Z = 3{,}960$ *kip-inch*

$$\Delta A_s = h_0 t_w = 3.8 \text{ in}^2$$

$M_m = M_p\left[1 - \frac{\Delta A_s\left(h_0 / 4 + e\right)}{Z}\right] = 3,\ 618 kip\text{-}in \le M_p$ Hence OK

Calculation of Maximum Shear Capacity

$$V_{pb} = \frac{F_y t_w s_b}{\sqrt{3}} = 42.768\text{kips}$$

$$V_{pt} = \frac{F_y t_w s_t}{\sqrt{3}} = 42.768\text{kips}$$

For unreinforced opening :

$$\mu_b = 0,\ \mu_t = 0$$

Ratio of nominal shear capacity of tees :

$$\alpha_{vb} = \frac{\sqrt{6} + \mu_b}{v_b + \sqrt{3}} = 0.4515 \le 1.0$$

$$\alpha_{vt} = \frac{\sqrt{6} + \mu_t}{v_t + \sqrt{3}} = 0.4515 \le 1.0$$

Hence OK

$$V_{mb} = V_{pb}\alpha_{vb} = 19.309 \text{ kips}$$

$$V_{mb} = V_{pb}\alpha_{vb} = 19.309 \text{ kips}$$

$Vm = V_{mb}+V_{mb} = 38.618$ *kips* Hence OK

Check against Maximum Shear capacity :

$$V_p = \frac{F_y t_w d}{\sqrt{3}} = 164.52\text{kips}$$

Since $W_1 \le \frac{420}{\sqrt{F_y}}$, $V_m \le \frac{2}{3}V_p = 109.68kips$ Hence OK

Check against Moment Shear Interaction:

$$R_1 = \frac{V_u}{\phi V_m} = 0.8923 \le 1.0$$

$$R_2 = \frac{M_u}{\phi M_m} = 0.6939 \le 1.0$$

$$R = \sqrt[3]{R_1^3 + R_2^3} = 1.0147 > 1.0$$

Not OK... Try with a Reinforced web opening .

Capacity check at web hole assuming a reinforced opening:

Reinforcement should be selected to reduce R to 1.0

Let us assume,

Thickness of Reinforcement $t_r = 0.1875\ in$
Width of Reinforcement $b_r = 0.25\ in$

Check for local buckling of compression flange :

width to thickness ratio of web reinforcement $F_2 = \frac{b_r}{t_r} = 1.3333$

limiting ratio $= \frac{65}{\sqrt{F_y}} = 10.833 > F_2$ Hence O.K

Area of Reinforcement $A_r = t_r b_r = 0.0469\ in^2$

Calculation of Maximum Moment Capacity :

For unperforated section $M_p = F_y Z = 3{,}960$ *kip-inch*

$$\Delta A_s = h_0 t_w - 2A_r = 3.7063 \text{ in}^2$$

Since $t_w e = 0 < A_r$

$M_m = M_p\left[1 - \frac{t_w\left(h_0^2 / 4 + h_0 e - e^2\right) - A_r h_0}{Z}\right] = 3,\ 634.88kip\text{-}in \le M_p$ Hence OK

Calculation of Maximum Shear Capacity :

$$V_{pb} = \frac{F_y t_w s_b}{\sqrt{3}} = 42.768\text{kips}$$

$$V_{pt} = \frac{F_y t_w s_t}{\sqrt{3}} = 42.768\text{kips}$$

$$s_{t1} = s_t - \frac{A_r}{2b_f} = 5.4114\text{in}$$

$$s_{b1} = s_b - \frac{A_r}{2b_f} = 5.4114\text{in}$$

$P_r = F_y A_r = 1.6875 \le \frac{F_y t_w a_0}{2\sqrt{3}} = 78.982kips$ Hence Ok

$$d_{rt} = s_t - \frac{1}{2}t_r = 5.3213$$

$$d_{rb} = s_b - \frac{1}{2}t_r = 5.3213$$

$$v_t = \frac{a_0}{s_{t1}} = 3.6959 < 12.0 \text{ , } \mu_t = \frac{2P_r d_{rt}}{V_{pt} s_t} = 0.0775$$

$$v_b = \frac{a_0}{s_{b1}} = 3.6959 < 12.0 \text{ , } \mu_b = \frac{2P_r d_{rb}}{V_{pb} s_b} = 0.0775$$

$$a_{vb} = \frac{\sqrt{6} + \mu_b}{v_b + \sqrt{3}} = 0.4656 \le 1.0$$

$$a_{vt} = \frac{\sqrt{6} + \mu_t}{v_t + \sqrt{3}} = 0.4656 \le 1.0$$

Hence OK

$$V_{mb} = V_{pb}\alpha_{vb} = 19.911 \text{ kips}$$

$$V_{mt} = V_{pt}\alpha_{vt} = 19.911 \text{ kips}$$

$$V_m = V_{mb} + V_{mt} = 39.823 \text{ kips}$$

Check against Maximum Shear capacity :

$$V_p = \frac{F_y t_w d}{\sqrt{3}} = 164.52\text{kips}$$

Since $W_1 \le \frac{420}{\sqrt{F_y}}$, $V_m \le \frac{2}{3}V_p = 109.68kips$ kips Hence OK

Check against Moment Shear Interaction :

$$R_1 = \frac{V_u}{\phi V_m} = 0.8653 \le 1.0$$

$$R_2 = \frac{M_u}{\phi M_m} = 0.6907 \le 1.0$$

$R = \sqrt[3]{R_1^3 + R_2^3} = 0.9924 < 1.0$ Hence OK

Calculation of length of Fillet Weld :

$$A_f = b_f t_f = 3.4936 \text{ in}^2$$

For reinforcing bars on one side of the web :

$A_r \le \frac{A_f}{3} = 1.1645 in^2$ Hence OK

$a_0 / h_0 = 2 \le 2.5$ Hence OK

$V_1 = \frac{s_t}{t_w} = 14.25$, $V_2 = \frac{s_b}{t_w} = 14.25$

$V_1 and V_2 \le \frac{140}{\sqrt{F_y}} = 23.333 kips$ Hence OK

$\frac{M_u}{V_u d} = 3.4977 \le 20$ Hence OK

$R_{w1} = \phi 2 P_r = 3.375 kips$ (strength of weld within the length of the opening)

$L_1 = max\left(\frac{a_0}{4}, \frac{A_r \sqrt{3}}{2t_w}\right) = 5in$ (length extended on each side of the opening)

Thus, Length of bar = $a_0 + 2L_1 = 30\ in$

$R_{w2} = \phi F_y A_r = 1.6875 kips$ (strength of weld for extension on each side of opening)

Strength of Weld $R_{wr} = max(R_{wr1}, R_{wr2}) = 3.375\ kips$

Fillet Weld Size = $0.0044\ in$ (rounded to nearest weld size of 0.0625 in = 1/16 in)

Corner Radii :

Minimum Radii $= max\left(2t_w, \frac{5}{8}\right) = 0.76in$

Results

Table 789: Web opening design for member no. 5

Result Type		Reference	STAAD.Pro	Difference	Comments
Interaction Ratio, R		0.99	1.00	negligible	
Reinforcement bar (required at web opening)	Length, in	30	30	none	
	Width, in	0.25	0.25	none	
	Thickness, in	0.1875	0.1875	none	
Fillet Weld Size, in		1/16	0.0625 (1/16")	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD PLANE Design of Steel Beam with Web Opening
START JOB INFORMATION
```

```
ENGINEER DATE 18-May-05
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 0 20 0; 4 10 20 0; 5 20 20 0; 6 30 20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38 0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4 5; 7 5 6; 8 7 12; 9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14; 16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21 9 11; 22 10 11; 23 8 10;
MEMBER PROPERTY AMERICAN
1 3 4 TABLE ST W14X90
2 TABLE ST W10X49
5 6 7 TABLE ST W21X50
8 TO 13 TABLE ST W18X35
14 TO 23 TABLE ST L40404
MEMBER TRUSS
14 TO 23
MEMBER RELEASE
5 START MZ
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
DENSITY 0.000283
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 3 4
MATERIAL MATERIAL1 MEMB 1 TO 4 6 TO 23
MATERIAL STEEL MEMB 5
UNIT FEET KIP
SUPPORTS
1 FIXED
2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTIES LIST 1 2 5 8 14
LOAD 1 DEAD AND LIVE LOAD
SELFWEIGHT X 1
SELFWEIGHT Y -1
JOINT LOAD
4 5 FY -15
11 FY -35
MEMBER LOAD
8 TO 13 UNI Y -0.9
6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOAD 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6
8 TO 10 UNI Y -1
```

```
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
LOAD COMB 4 75 PERCENT DL LL WL
1 2.75 2 2.75
PERFORM ANALYSIS
LOAD LIST 4
UNIT INCHES KIP
PARAMETER
CODE AISC
*WEB OPENINGS
*********************
RHOLE 0.6 MEMB 5
RDIM 20.0 10.0 MRMB 5
electrode 3
*********************
CHECK CODE MEMB 5
FINISH
```

STAAD.Pro Output

```
                         STAAD.Pro CODE CHECKING - (AISC 9TH EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *    5  ST   W21X50                   (AISC SECTIONS)
                           FAIL     AISC- H2-1         2.268         4
                       50.64 T          0.00       -4151.62      120.00
   OUTPUT FOR WEB OPENING
   ----------------------
   SECTION    LOAD        MZ/           FY/                IR
                      MOM. CAP. (Mm)  SHR. CAP. (Vm) ((MZ/Mm)^3+(FY/
Vm)^3)^0.33
   =======================================================================
    0.600      4        2510.65         34.46
                        3634.88         39.64              1.00
   REINFORCING BARS : TO BE PLACED ON ONE SIDE OF WEB
   BAR DIMENSION    : LENGTH = 30.00 WIDTH =  0.25 THICKNESS = .1875
   FILLET WELD      : SIZE REQD = 0.0625 inch LENGTH REQD =  30.00
   ======================= END WEB OPENING OUTPUT =======================
```

Related Links

- *D1.B.3 Design of Beams with Web Openings* (on page 1157)

V. AISC ASD - Deflection Check for a Steel Beam with Web Opening

Determine the deflection of a beam with a web opening.

Details

Design Method : ASD

Beam No : 5

Beam Section : W21X50

$d = 20.83$ in
$t_w = 0.38$ in
$b_f = 6.53$ in
$t_f = 0.535$ in
$Z = 110\ in^3$
$L = 30$ ft
$E = 29,732$ ksi

Opening Type: Rectangular

Number of openings : 1

Section location of Opening, $s_{open} = 0.3$
$a_0 = 20$ in
$h_0 = 10$ in
$e = 0$ in (concentric opening)
$F_y = 36$ ksi

Loading at critical location:

$V_u = 9.51$ kip
$M_u = 344.36$ kip-inch

Location of maximum deflection from start of the member : $L_{critical} = 120\ in$

Tee Properties :

$s_t = 5.415$ in
$s_b = 5.415$ in
Moment Area = $\frac{b_f t_f^2}{2} + t_w\left(s_t - t_f\right)\left(\frac{s_t t_f}{2} + t_f\right) = 6.4514in^3$
Area = $b_f t_f + t_w(s_t - t_f) = 5.34795\ in^2$
C_{yt} = Moment Area / Area = 1.20632 in.
$I_{zz,t} = I_{zz,b} = \frac{b_f t_f^3}{12} + b_f t_f\left(C_y - \frac{t_f}{2}\right)^2 + \frac{t_w(s_t - t_f)^3}{12} + t_w\left(s_t - t_f\right)\left(c_y - t_f - \frac{s_t - t_f}{2}\right)^2 = 12.644in^3$
(concentric opening)
$I_{zz} = I_{zz,t} + I_{zz,b} = 25.2872\ in^3$

Validation

$$R_t = \frac{I_{zz,b}}{I_{xx}} = 0.5$$

$$R_b = \frac{I_{zz,t}}{I_{xx}} = 0.5$$

$V_t = F_y R_t = 4.755$

$V_b = F_y R_b = 4.755$

$$\Delta_{0t} = \frac{V_t a_0^3}{6EI_{zz,t}} = 0.01686\text{in}$$

$$\Delta_{0b} = \frac{V_b a_0^3}{6EI_{zz,b}} = 0.01686\text{in}$$

$$\Delta_0 = \Delta_{0b} + \Delta_{0b} = 0.03373 \text{ in}$$

L_0 = distance from high moment end of opening to adjacent support $= L - s_{open}L - \frac{a_0}{2} = 314 in$

L_2 = distance from support to the point where deflection is maximum = L_0 - $L_{critical}$ = $240\ in$

$L_3 = L_0 - L_2 = 74\ in$

$$\Delta_{p1} = \Delta_0 \frac{L_2}{L_0} = 0.02578\text{in}$$

$$\theta_H = \frac{\Delta_0}{L_0} = 0.00011$$

$$\theta_T = \frac{V a_0^2}{4EI_{zz}} = 0.00126$$

$$\Delta_{p2} = \frac{2L_2 L_3(\theta_H + \theta_T)}{L_0 + L_2} = 0.08799$$

$\Delta = \Delta_{p1} + \Delta_{p2} = 0.11377\ in$ (extra deflection due to opening)

displacement factor: $\Delta_{factor} = 750$

$$\Delta_{allow} = \frac{L}{\Delta_{factor}} = 0.48\text{in}$$

Ratio of $\Delta_{actual}/\Delta_{allow} = 0.66$ (without web opening)

Thus, $\Delta_{actual} = 0.3168\ in$ (from analysis considering stiffness of unperforated member without web opening)

Total deflection considering the effect of web hole,

$\Delta_{actual,wb} = \Delta + \Delta_{actual} = 0.4306$ in

$$\text{Ratio} = \frac{\Delta_{actual,wb}}{\Delta_{allow}} = 0.897$$

Results

Table 790: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Interaction Ratio, R	0.897	0.899	< 1%	
Total deflection, in	0.4306	0.4315*	< 1%	

* STAAD.Pro does not directly report the deflection, but rather the ratio of deflection with length, dff, for comparison with a limiting displacement factor.

$$\Delta_{actual,wb} = L / dff = 360 \text{ in} / 834.32 = 0.4315 \text{ in}$$

Note: The member length used for deflection calculations is defined using the start deflection joint number, `DJ1`, and end deflection joint number, `DJ2`. These are assigned to the member in the design code parameters. STAAD.Pro uses this to calculate the physical member length rather than the segment length of the analytical member for deflection checks.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\` is typically installed with the program.

```
STAAD PLANE Deflection of Steel Beam with Web Opening
START JOB INFORMATION
ENGINEER DATE 18-May-05
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0; 3 0 20 0; 4 10 20 0; 5 20 20 0; 6 30 20 0; 7 0 35 0;
8 30 35 0; 9 7.5 35 0; 10 22.5 35 0; 11 15 35 0; 12 5 38 0; 13 25 38 0;
14 10 41 0; 15 20 41 0; 16 15 44 0;
MEMBER INCIDENCES
1 1 3; 2 3 7; 3 2 6; 4 6 8; 5 3 4; 6 4 5; 7 5 6; 8 7 12; 9 12 14;
10 14 16; 11 15 16; 12 13 15; 13 8 13; 14 9 12; 15 9 14; 16 11 14;
17 11 15; 18 10 15; 19 10 13; 20 7 9; 21 9 11; 22 10 11; 23 8 10;
MEMBER PROPERTY AMERICAN
1 3 4 TABLE ST W14X90
2 TABLE ST W10X49
5 6 7 TABLE ST W21X50
8 TO 13 TABLE ST W18X35
14 TO 23 TABLE ST L40404
MEMBER TRUSS
14 TO 23
MEMBER RELEASE
5 START MZ
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
DENSITY 0.000283
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 3 4
MATERIAL MATERIAL1 MEMB 1 TO 4 6 TO 23
MATERIAL STEEL MEMB 5
UNIT FEET KIP
SUPPORTS
1 FIXED
```

```
2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTIES LIST 1 2 5 8 14
LOAD 1 DEAD AND LIVE LOAD
SELFWEIGHT X 1
SELFWEIGHT Y -1
JOINT LOAD
4 5 FY -15
11 FY -35
MEMBER LOAD
8 TO 13 UNI Y -0.9
6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOAD 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 0.6
8 TO 10 UNI Y -1
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 3
UNIT INCHES KIP
PARAMETER
CODE AISC
TRACK 2 MEMB 5
DFF 750
DJ1 3 MEMB 5
DJ2 6 MEMB 5
*WEB OPENINGS
*********************
RHOLE 0.3 MEMB 5
RDIM 20.0 10.0 MEMB 5
*********************
CHECK CODE MEMB 5
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (AISC 9TH EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                       Y        PROPERTIES |
   |*************                                          |        IN INCH UNIT |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   5  *  |  AISC SECTIONS              |      |        AX =  14.70  |
   |            *  | ST  W21X50                  |      |   --Z  AY =   6.94  |
   |DESIGN CODE *  |                             |      |        AZ =   4.69  |
   | AISC-1989  *  ===============================   ===|===     SY =   7.63  |
   |            *                                                SZ =  94.62  |
   |            *  |<---LENGTH (FT)=   10.00 --->|                  RY =
1.30  |
   |*************                                                RZ =   8.18  |
   |                                                                          |
   |                 94.4 (KIP-FEET)                                          |
   |PARAMETER        |                                       L3  STESSES      |
   |IN KIP  INCH     |                               L3  L3      IN KIP  INCH |
```

```
 |---------------  +                           L3               -------------|
 | KL/R-Y=  92.20  |                       L3                   FA  =  21.60 |
 | KL/R-Z=  14.67  +                   L3                       fa  =   0.94 |
 | UNL    = 120.00 |               L3                           FCZ =  19.35 |
 | CB     =   1.00 +                                            FTZ =  21.60 |
 | CMY    =   0.85 |       L3                                   FCY =  27.00 |
 | CMZ    =   0.85 +   L3                                       FTY =  27.00 |
 | FYLD   =  36.00 |L0         L0                               fbz =  11.97 |
 | NSF    =   1.00 +---+---+---+---+---+---+---+---+---|        fby =   0.00 |
 | DFF    = 750.00 -5.2                                         Fey =  18.01 |
 | dff=    834.32            ABSOLUTE MZ ENVELOPE               Fez = 711.71 |
 | (KL/R)max =  92.20           (WITH LOAD NO.)                 FV  =  14.40 |
 |                                                              fv  =   1.33 |
 |                                                                           |
 |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
 |                     -------------------------                             |
 |                                                                           |
 |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
 |                                                                           |
 |      VALUE    -14.2         9.6        0.0         0.0        94.4        |
 |   LOCATION      0.0         0.0        0.0         0.0        10.0        |
 |    LOADING        3           3          0           0           3        |
 |                                                                           |
 |***************************************************************************|
 |*                                                                         *|
 |*                        DESIGN SUMMARY  (KIP-FEET)                       *|
 |*                        --------------                                   *|
 |*                                                                         *|
 |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/            *|
 |         FX                MY            MZ           LOCATION             |
 |        ======================================================             |
 |          PASS        DEFLECTION       0.899              3                |
 |         13.81 T          0.00       -94.36            10.00               |
 |*                                                                         *|
 |                                                                           |
 |---------------------------------------------------------------------------|
      DEFLECTION OF STEEL BEAM WITH WEB OPENING                -- PAGE NO.
7
   OUTPUT FOR WEB OPENING
   ----------------------
   SECTION    LOAD        MZ/            FY/                 IR
                      MOM. CAP. (Mm)  SHR. CAP. (Vm)  ((MZ/Mm)^3+(FY/
Vm)^3)^0.33
   =======================================================================
    0.300       3         344.36          9.51
                         3618.00         38.44              0.25
   ====================== END WEB OPENING OUTPUT =======================
    79. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME= 10: 4:57 ****
      DEFLECTION OF STEEL BEAM WITH WEB OPENING                -- PAGE NO.
8
         ************************************************************
```

Related Links

- *D1.B.3 Design of Beams with Web Openings* (on page 1157)

V. AISC LRFD - Wide Flange Tension Capacity

To check the adequacy of a W8X24 American section to carry a tensile design load of 245 Kips.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 3.1, case (a), page 3-5.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

Assume the reduction in area due to bolt holes to be equivalent to a net section factor of 0.72.

$F_y = 50$ ksi

L =10ft

Hand Calculation

Net section = $0.72(7.08 in.^2) = 5.11\ in.^2$

$$\phi P_n = \begin{cases} \phi_t F_y A_g = 0.9(50ksi)(7.08in.^2) = 318.6k \\ \phi_t F_u A_e = 0.75(65ksi)(5.11in.^2) = 248.5k \end{cases}$$

Comparison

The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output.

Table 791: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	248.5	248.51	none

Note: The assumed NSF of 0.72 is comparable to the term Ae/Ag which evaluates to 5.11/7.08.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Tension Capacity.STD` is typically installed with the program.

```
STAAD SPACE TENSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  OBJECTIVE : CHECKING THE DESIGN AXIAL TENSILE STRENGTH OF A
*  W8X24 AMERICAN SECTION. CAPACITY SHOULD BE 245 KIPS ACCORDING TO
*  EXAMPLE 3-1 ON PAGE 3-5 OF 3RD EDITION LRFD CODE.
```

```
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W8X24
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FX 245
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
* NSF = 5.11/7.08 = 0.72
NSF 0.72 ALL
FYLD 50 ALL
FU 65 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                        **********************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W8X24                       (AISC SECTIONS)
                           PASS     LRFD-H1-1A-T       0.986         1
                      245.00 T          0.00           0.00        0.00
 +-----------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN                |
 | PNC=  200.22  PNT=  248.51  MNZ=  930.59  MNY=  380.08  VN=   52.46 |
 +-----------------------------------------------------------------------+
```

V. AISC LRFD - Angle Section Tension Capacity

To check the adequacy of a L4X4X1/2 American single angle to carry a tensile design load of 120 Kips.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 3.2, case (a), page 3-7.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

Assume the reduction in area due to bolt holes to be equivalent to a net section factor of 0.795.

F_y =36 ksi

Hand Calculation

Net section = $0.795(3.75\ in.^2) = 2.98\ in.^2$

$$\phi P_n = \begin{cases} \phi_t F_y A_g = 0.9(36ksi)(3.75in.^2) = 121.5k \\ \phi_t F_u A_e = 0.75(58ksi)(2.98in.^2) = 129.6k \end{cases}$$

Comparison

The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output.

Table 792: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	121.5	121.5	none

Note: The assumed NSF of 0.795 is comparable to the term Ae/Ag which evaluates to 2.98/3.75.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Angle Section Tension Capacity.STD` is typically installed with the program.

```
STAAD SPACE TENSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  OBJECTIVE : CHECKING THE DESIGN AXIAL TENSILE STRENGTH OF A
*  L4X4X1/2 AMERICAN SECTION. CAPACITY SHOULD BE 122 KIPS ACCORDING TO
*  EXAMPLE 3-2 ON PAGE 3-7 OF 3RD EDITION LRFD CODE.
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST L40408
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 120
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FU 58 ALL
* NSF = 2.98/3.75 = 0.795
NSF 0.795 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  ST   L40408                   (AISC SECTIONS)
                           PASS     LRFD-H1-1A-T       0.988         1
                     120.00 T          0.00          0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN              |
| PNC=   34.36  PNT=  121.50  MNZ=   96.57  MNY=  134.69  VN=   25.92 |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Wide Flange Compression Capacity 1

To determine the capacity of W14X132 column in axial compression.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.1, case (a), page 4-7.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

The column is braced at its ends for both axes.

F_y =50 ksi

Length = 30 ft

Hand Calculation

Net section = $0.795(3.75\ in.^2) = 2.98\ in.^2$

$$\phi P_n = \begin{cases} \phi_t F_y A_g = 0.9(36ksi)(3.75in.^2) = 121.5k \\ \phi_t F_u A_e = 0.75(58ksi)(2.98in.^2) = 129.6k \end{cases}$$

Comparison

The reported STAAD.Pro value is based on the term PNT in the STAAD.Pro output..

Table 793: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	844	843.02	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Compression Capacity 1.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.1, CASE (A), PAGE 4-7, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 844 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
```

```
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -840
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W14X132                  (AISC SECTIONS)
                           PASS     LRFD-H1-1A-C       0.996         1
                       840.00 C          0.00          0.00        0.00
 +-----------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN                |
 | PNC=  843.02  PNT= 1746.00  MNZ= 9138.93  MNY= 5032.65  VN=  256.00 |
 +-----------------------------------------------------------------------+
```

V. AISC LRFD - Wide Flange Compression Capacity 2

To determine the capacity of W14X132 column in axial compression. The column is braced at its ends for its major axis, and at ends and mid-height for the minor axis.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.1, case (b), page 4-7.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

F_y =50 ksi

Length = 30 ft

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output..

Table 794: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	1,300	1,296.75	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Compression Capacity 2.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.1, CASE (B), PAGE 4-7, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 1300 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 30 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X132
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -840
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
LY 15 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
```

```
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX             MY             MZ       LOCATION
=======================================================================
     1  ST   W14X132                  (AISC SECTIONS)
                          PASS     LRFD-H1-1A-C       0.648         1
                     840.00 C          0.00           0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN              |
| PNC= 1296.75  PNT= 1746.00  MNZ= 9138.93  MNY= 5032.65  VN=  256.00 |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Angle Section Compression Capacity

To determine the design strength of an 8ft long single angle L4X3.5X5/16 in axial compression.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.3, page 4-12.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

The column is pinned at its ends and no intermediate bracing.

F_y =36 ksi

Length = 8 ft

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output..

Table 795: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
Design strength (kips)	28.7	28.66	Negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Angle Section Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.3, PAGE 4-12, AISC LRFD 3RD ED.
```

```
*  CAPACITY (PNC) SHOULD BE ABOUT 28.7 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 8 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST L40355
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -25
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 36 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                        ***********************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  ST   L40355                   (AISC SECTIONS)
                          PASS     LRFD-H1-1A-C       0.872         1
                      25.00 C          0.00           0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN               |
| PNC=   28.66  PNT=   72.90  MNZ=   60.87  MNY=   67.97  VN=   16.23 |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Tee Section Compression Capacity

To find a suitable American T-section with an axial compressive strength of 100 Kips.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.4, page 4-13.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

The member ends are pinned, and the member is braced at the ends only for both axes.

$F_y = 50$ ksi

Length = 20 ft

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 796: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	102	101.56	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Tee Section Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.4, PAGE 4-13, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 102 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE T W12X26
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
```

```
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -100
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
TRACK 1 ALL
SELECT ALL
FINISH
```

STAAD Output

```
                       STAAD.Pro MEMBER SELECTION - (LRFD 3RD EDITION)   v1.0
                       **************************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  T    W14X53                   (AISC SECTIONS)
                          PASS     LRFD-H1-1A-C       0.985         1
                     100.00 C          0.00           0.00       0.00
+----------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN              |
| PNC=  101.56  PNT=  351.00  MNZ=  209.89  MNY=  322.15  VN=   69.43 |
+----------------------------------------------------------------------+
```

V. AISC LRFD - Rectangular HSS Compression Strength

To validate that an American ASTM A500 grade B HSS12X10X1/2 compression member has a design compressive strength of over 500 kips.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.8, page 4-13 & Table 4-6, Page 4-50.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

F_y = 46 ksi

Length = 20 ft

KY = KZ = 0.8

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 797: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength in Axial Compression (kips)	634 (*)	634.26	none

Note: * In the solved example, on page 4-14, the capacity is reported as ϕ×Pn = 580 kips. Looking at Table 4-6 on page 4-50, this happens to be the capacity for a 20 ft effective length, not a 16 ft effective length. As per the table, the capacity corresponding to a 16 ft effective length is 634 kips.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Rectangular HSS Compression Strength.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.5, PAGE 4-13, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 634 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST12X10X0.5
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -500
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 46 ALL
```

```
KY 0.8 ALL
KZ 0.8 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         **********************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  ST   HSST12X10X0.5           (AISC SECTIONS)
                           PASS     HSS COMPRESS       0.788         1
                      500.00 C         0.00          0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1                   UNITS - KIP IN |
| PNC=  634.26      PNT=  786.60      MNZ= 3262.32      MNY= 2881.44 |
| VNZ=  231.01      VNY=  277.21      TN= 2533.68                    |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Double Angle Compression Capacity

To determine the design compressive strength of a long-leg-back-to-back double angle made of American L4X3.5X1/2 angles with 3/4 in. separation.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 4.7, page 4-14.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Member is 8 ft long. Ends are pinned. Unbraced length should be assumed as the distance between ends.

Problem

$F_y = 36$ ksi

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 798: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	101	101.22	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Double Angle Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 4.7, PAGE 4-14, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 101 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 8 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TABLE LD L40355 SP 0.75
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -100
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
FYLD 36 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  LD   L40355                   (AISC SECTIONS)
                            PASS     LRFD-H1-1A-C       0.988         1
                       100.00 C          0.00          0.00       0.00
 +---------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN              |
```

```
| PNC=  101.22  PNT=  145.80  MNZ=   81.97  MNY=  107.03  VN=   48.68 |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Round HSS Compression Capacity

To validate that an American Round HSS 10.000X0.5 compression member has a design compressive strength of about 364 kips.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Table 4-7, Page 4-68.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

F_y = 42 ksi

Length = 20 ft

Comparison

The reported STAAD.Pro value is based on the term PNC in the STAAD.Pro output.

Table 799: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design strength (kips)	364	364.22	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Round HSS Compression Capacity.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  DESIGN COMPRESSIVE STRENGTH OF A ROUND HSS 10.000X0.5
*  TABLE 4-7, PAGE 4-68, AISC LRFD 3RD ED.
*  CAPACITY (PNC) SHOULD BE ABOUT 364 KIPS
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 20 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST HSSP10X0.5
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1
JOINT LOAD
2 FY -350
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 42 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     1  ST   HSSP10X0.5               (AISC SECTIONS)
                           PASS     HSS COMPRESS       0.961         1
                     350.00 C          0.00           0.00        0.00
+---------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     1                   UNITS - KIP IN |
| PNC=  364.22      PNT=  525.42      MNZ= 1598.94      MNY= 1598.94 |
| VNZ=  157.63      VNY=  157.63      TN= 1440.18                    |
+---------------------------------------------------------------------+
```

V. AISC LRFD - Wide Flange Flexural Strength 1

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 5.1, Case (a), page 5-12.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

The beam is braced such that $L_b = 2\ ft$. The deflection at mid-span due to a uniformly distributed service load of 1 kip/ft is also to be calculated.

$F_y = 50$ ksi

Length = 35 ft

Validation

The beam is continuously braced, so by observation $L_b < L_p$.

The design flexural strength is:

$$\phi_b M_n = \phi_b [F_y Z_x] = 0.9 \times [50 \text{ ksi} \times 78.4 \text{ in}^3] = 3{,}528 \text{ k}\cdot\text{in}$$

The maximum deflection (at midpsan) due to a uniform load of 1 kip/ft is:

$$\Delta = \frac{5w \times l^4}{384EI} = \frac{5(1)(35)^4(12)^3}{384(29{,}000)(612)} = 1.90 \text{ in.}$$

Results

Table 800: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	3,528	3,528	none	
Deflection (in)	1.90	1.8918	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Flexural Strength 1.STD` is typically installed with the program.

Note: The UNT parameter is set to 1.0 in. in order to approximate a fully-braced condition.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.1, CASE (A), PAGE 5-12, AISC LRFD 3RD ED.
*  CAPACITY (MNZ) SHOULD BE ABOUT 294 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST W18X40
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1
LOAD COMBINATION 2
1 1.8
PERFORM ANALYSIS
LOAD LIST 1
SECTION 0.5 ALL
PRINT SECTION DISPL
LOAD LIST 2
UNIT FEET KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
UNT 2 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
  SECTION  DISPL
      BENDING CAPACITY PER AISC LRFD 3RD ED                    -- PAGE NO.
3
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - INCH
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     1       0.0000    0.0000    0.0000    0.0000   -0.4973    0.0000
                     0.0000   -0.9571    0.0000    0.0000   -1.3475    0.0000
                     0.0000   -1.6441    0.0000    0.0000   -1.8290    0.0000
                     0.0000   -1.8918    0.0000    0.0000   -1.8290    0.0000
                     0.0000   -1.6441    0.0000    0.0000   -1.3475    0.0000
                     0.0000   -0.9571    0.0000    0.0000   -0.4973    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    1.89185   AT     210.00  LOAD    1   L/DISP=    222
         ************ END OF SECT DISPL RESULTS ***********
    37. LOAD LIST 2
    38. UNIT FEET KIP
    39. PARAMETER 1
    40. CODE LRFD
    41. MAIN 1 ALL
    42. FYLD 7200 ALL
```

```
    43. UNT 2 ALL
    44. TRACK 1 ALL
    45. CHECK CODE ALL
 STEEL DESIGN
      BENDING CAPACITY PER AISC LRFD 3RD ED                    -- PAGE NO.
4
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1  ST   W18X40                  (AISC SECTIONS)
                           PASS     LRFD-H1-1B-C       0.938         2
                        0.00 C         0.00        -275.62       17.50
 +-----------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN                |
 | PNC=   23.10  PNT=  531.00  MNZ= 3528.00  MNY=  428.32  VN=  152.24 |
 +-----------------------------------------------------------------------+
```

V. AISC LRFD - Wide Flange Flexural Strength 2

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 5.1, Case (b), page 5-12.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

The beam is braced at the ends and at third points such that $L_b = 140\ in$.

$F_y = 50$ ksi

Length = 35 ft

Validation

The plastic limiting unbraced length is given by:

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76(1.27)\sqrt{\frac{29{,}000}{50}} = 53.8 \text{ in.}$$

The inelastic limiting unbraced length is given by:

$$L_r = \frac{r_y X_1}{F_L}\sqrt{1 + \sqrt{1 + X_2 F_L{}^2}} \qquad \text{(F1-6)}$$

where

F_L = $F_{yf} - F_r = 50\ ksi - 10\ ksi = 40\ ksi$

$$X_1 = \frac{\pi}{S_x}\sqrt{\frac{EGJA}{2}} = \frac{\pi}{68.4}\sqrt{\frac{29,000(11,200)(0.810)(11.85)}{2}} = 1,813$$

$$X_2 = 4\frac{C_w}{I_y}\left(\frac{S_x}{GJ}\right)^2 = 4\left(\frac{1,440}{19.1}\right)\left[\frac{68.4}{11,200(0.810)}\right]^2 = 0.0171$$

$$L_r = \frac{1.27 \times 1,813}{40}\sqrt{1 + \sqrt{1 + 0.0171(40)^2}} = 144.9 \text{ in.}$$

$M_r = F_L S_x = 40 \times 68.4 = 2,736$ in·k (F1-7)

$L_p = 53.8$ in.

$L_r = 144.9$ in.

$L_p < L_b < L_r$; thus, the section is inelastic.

The nominal flexural strength is:

$$M_n = C_b\left[M_p - \left(M_p - M_r\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] < M_p \tag{F1-2}$$

$M_p = F_y Z_x = 50 \times 78.4 = 3,920$ in·k

Conservatively assume $C_b = 1.0$.

$$M_n = 3,920 - \left(3,920 - 2,736\right)\left(\frac{140 - 53.8}{144.9 - 53.8}\right) = 2,800 \text{ in·k}$$

The lateral-torsional buckling limit state capacity:

$\phi_b M_n = 0.9 \times (2,800) = 2,520$ in·k

Results

Table 801: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	2,520	2,514	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Flexural Strength 2.STD` is typically installed with the program.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.1, CASE (B), PAGE 5-12, AISC LRFD 3RD ED.
*  CAPACITY (MNZ) SHOULD BE ABOUT 211 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X40
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1.3
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
MAIN 1 ALL
CB 1 ALL
FYLD 7200 ALL
UNT 11.7 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y         PROPERTIES  |
  |*************                                       |         IN INCH UNIT |
  |            *  |=============================|   ===|===      ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |      |         AX=0.1180E+2 |
  |            *  | ST  W18X40                  |      |    --Z  AY=0.5638E+1 |
  |DESIGN CODE *  |                             |      |         AZ=0.4214E+1 |
  | LRFD 2001  *  ===============================   ===|===      PY=0.1000E+2 |
  |            *                                                  PZ=0.7840E+2 |
  |            *  |<---LENGTH (FT)=   35.00 --->|                   RY=0.1272E
+1 |
  |*************                                                  RZ=0.7202E+1 |
  |                                                                           |
  |                199.1 (KIP-FEET)                                           |
  |PARAMETER        |               L1  L1  L1                  CAPACITIES  |
  |IN KIP  INCH     |                                           IN KIP  INCH |
  |---------------  +           L1              L1              -------------|
  | KL/R-Y= 330.12  |                                           PNC=0.2310E+2|
  | KL/R-Z=  58.32  +       L1                       L1         pnc=0.0000E+0|
  | UNL    = 140.40  |                                           PNT=0.5310E+3|
  | CB     =   1.00  +   L1                               L1     pnt=0.0000E+0|
  | PHIC   =   0.85  |                                           MNZ=0.2514E+4|
  | PHIB   =   0.90  +                                           mnz=0.2389E+4|
  | FYLD   =  50.00  |L0                                    L0   MNY=0.4283E+3|
```

```
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   mny=0.0000E+0|
| DFF =     0.00 -11.1                                       VN =0.1522E+3|
| dff =     0.00               ABSOLUTE MZ ENVELOPE          vn =0.0000E+0|
|                                 (WITH LOAD NO.)                         |
|                                                                         |
|                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)              |
|                      -------------------------                          |
|                                                                         |
|                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z    |
|                                                                         |
|      VALUE       0.0         22.7        0.0         0.0       199.1    |
|   LOCATION       0.0          0.0        0.0         0.0        17.5    |
|    LOADING         0            1          0           0           1    |
|                                                                         |
|**************************************************************************|
|*                                                                        *|
|*                        DESIGN SUMMARY  (KIP-FEET)                      *|
|*                        --------------                                  *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX              MY             MZ          LOCATION             |
|        ======================================================           |
|          PASS        LRFD-H1-1B-C     0.950             1                |
|          0.00 C          0.00      -199.06            17.50              |
|*                                                                        *|
|**************************************************************************|
|                                                                          |
|--------------------------------------------------------------------------|
```

V. AISC LRFD - Wide Flange Flexural Strength 3

To determine the design flexural strength of an ASTM A992 W18X40 bent about its strong axis on a 35-ft span with a uniformly distributed load.

Reference

Hand calculation using the following reference:

AISC Load Factor Resistance Design, 3rd Edition, Example 5.1, Case (c), page 5-12.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

The beam is braced at the ends only and so, $L_b = 420\ in$.

F_y = 50 ksi

Length = 35 ft

Validation

The inelastic limiting unbraced length is given by:

$$L_r = \frac{r_y X_1}{F_L}\sqrt{1+\sqrt{1+X_2 F_L{}^2}} \tag{F1-6}$$

where

F_L = F_{yf} - F_r = 50 ksi - 10 ksi = 40 ksi

X_1 = $\frac{\pi}{S_x}\sqrt{\frac{EGJA}{2}} = \frac{\pi}{68.4}\sqrt{\frac{29{,}000(11{,}200)(0.810)(11.85)}{2}} = 1{,}813$

X_2 = $4\frac{C_w}{I_y}\left(\frac{S_x}{GJ}\right)^2 = 4\left(\frac{1{,}440}{19.1}\right)\left[\frac{68.4}{11{,}200(0.810)}\right]^2 = 0.0171$

$$L_r = \frac{1.27 \times 1{,}813}{40}\sqrt{1+\sqrt{1+0.0171(40)^2}} = 144.9 \text{ in.}$$

$L_b > L_r$, thus the section is subject to elastic lateral-torsional buckling.

The modification factor, C_b *is given by:*

$$C_b = \frac{12.5M_{\max}}{2.5M_{\max} + 3M_A + 4M_B + 3M_C} \tag{F1-3}$$

Since the moment diagram is symmetric about the mid-span point of the beam, the maximum moment is the same as the mid-span moment and the quarter-point moments are equal.

$$M_{\max} = M_B = \frac{wl^2}{8}$$

The quarter-point moments of a uniformly loaded beam are likewise equal:

$$M_A = M_C = \frac{w(l/4)}{2}\left(l - \frac{l}{4}\right) = \frac{3wl^2}{32}$$

The wl^2 terms factor out and the constants simply as follows:

$$C_b = \frac{12.5(1/8)}{2.5(1/8) + 3(3/32) + 4(1/8) + 3(3/32)} = \frac{12.5(1/8)}{6.5(1/8) + 6(3/32)} = 1.14$$

The critical buckling moment is given by:

$$M_{cr} = \frac{C_b S_x X_1 \sqrt{2}}{L_b / r_y}\sqrt{1 + \frac{X_1{}^2 X_2}{2(L_b / r_y)^2}} \tag{F1-13}$$

$$M_{cr} = \frac{1.14(68.4)(1{,}813)\sqrt{2}}{420/1.27}\sqrt{1+\frac{(1{,}813)^2(0.0171)}{2(420/1.27)^2}} = 675.6 \text{ in}\cdot\text{k}$$

The lateral-torsional buckling limit state capacity:

$$\phi_b M_n = 0.9 \times (675.6) = 608.1 \text{ in}\cdot\text{k}$$

Results

Table 802: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Modification factor, C_b	1.14	1.14	none	

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	608.1	607.2	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Flexural Strength 3.STD` is typically installed with the program.

The input parameter `CB 0` directs the program to calculate the value of the modification factor.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.1, CASE (C), PAGE 5-12, AISC LRFD 3RD ED.
*  CAPACITY (MNZ) SHOULD BE ABOUT 50.8 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 35 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W18X40
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -0.3
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
MAIN 1 ALL
CB 0 ALL
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
```

```
|--------------------------------------------------------------------------|
|                                                     Y        PROPERTIES  |
|*************                                       |        IN INCH UNIT |
|            *  |=============================|   ===|===      ------------ |
|MEMBER   1  *  |  AISC SECTIONS              |      |        AX=0.1180E+2 |
|            *  | ST  W18X40                  |      |    --Z AY=0.5638E+1 |
|DESIGN CODE *  |                             |      |        AZ=0.4214E+1 |
| LRFD 2001  *  ==============================   ===|===      PY=0.1000E+2 |
|            *                                                PZ=0.7840E+2 |
|            *  |<---LENGTH (FT)=   35.00 --->|                  RY=0.1272E
+1 |
|*************                                                RZ=0.7202E+1 |
|                                                                          |
|                 45.9 (KIP-FEET)                                          |
|PARAMETER        |               L1  L1  L1                 CAPACITIES   |
|IN KIP  INCH     |                                          IN KIP  INCH |
|---------------  +           L1              L1             -------------|
| KL/R-Y= 330.12  |                                          PNC=0.2310E+2|
| KL/R-Z=  58.32  +       L1                      L1         pnc=0.0000E+0|
| UNL   = 420.00  |                                          PNT=0.5310E+3|
| CB    =   1.14  +   L1                              L1     pnt=0.0000E+0|
| PHIC  =   0.85  |                                          MNZ=0.6072E+3|
| PHIB  =   0.90  +                                          mnz=0.5512E+3|
| FYLD  =  50.00  |L0                                    L0  MNY=0.4283E+3|
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|  mny=0.0000E+0|
| DFF =     0.00  -2.6                                       VN =0.1522E+3|
| dff =     0.00               ABSOLUTE MZ ENVELOPE          vn =0.0000E+0|
|                                  (WITH LOAD NO.)                         |
|                                                                          |
|                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
|                      -------------------------                           |
|                                                                          |
|                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
|                                                                          |
|      VALUE       0.0         5.2        0.0         0.0        45.9      |
|   LOCATION       0.0         0.0        0.0         0.0        17.5      |
|    LOADING         0           1          0           0           1      |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                        DESIGN SUMMARY  (KIP-FEET)                      *|
|*                        --------------                                  *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX              MY             MZ          LOCATION              |
|        ======================================================            |
|          PASS        LRFD-H1-1B-C     0.908            1                 |
|          0.00 C          0.00       -45.94           17.50               |
|*                                                                        *|
|**************************************************************************|
|                                                                          |
|--------------------------------------------------------------------------|
```

V. AISC LRFD - Select Wide Flange 1

To select the optimum ASTM A992 W-shape with a design flexural strength of 500 kip-ft.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Table 5.2, , page 5-50.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

Beam is bent about its strong axis, and is continuously braced.

F_y = 50 ksi

Length = 30 ft

w = 4.44 kip/ft (equiv. of 500 ft·kip design moment)

Comparison

Table 803: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Optimum section	W24x55	W24x55	none
Design strength, $\phi_b M_z$ (ft·kips)	506	502.5 (*)	<1%

* Calculated as MZ envelope value divided by the MZ ratio:

499.95 ft·kips/0.995 = 502.5 ft·kips

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Select Wide Flange 1.STD` is typically installed with the program.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  TABLE 5.2, PAGE 5-40, AISC LRFD 3RD ED.
*  OBJECTIVE : TO SELECT THE MOST OPTIMUM SECTION WITH A DESIGN
*  FLEXURAL STRENGTH OF 500 KIP-FT.
*  FOR A FULLY BRACED BEAM, PER ABOVE REFERENCE, IT SHOULD BE
*  A W24X55
*
* 500 KIP-FT IS EQUAL TO AN APPLIED LOAD OF 4.444 KIP/FT ON
* A SIMPLY SUPPORTED BEAM
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X26
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -4.444
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
MAIN 1 ALL
FYLD 50 ALL
CB 0 ALL
TRACK 2 ALL
SELECT ALL
FINISH
```

STAAD Output

```
                     STAAD.PRO MEMBER SELECTION - (LRFD 3RD EDITION)   v1.0
                     ***********************************************
   |--------------------------------------------------------------------------|
   |                                                  Y        PROPERTIES     |
   |*************                                     |        IN INCH UNIT   |
   |             *  |=============================|   ===|===    ------------ |
   |MEMBER   1   *  |  AISC SECTIONS              |      |       AX=0.1620E+2 |
   |             *  | ST  W24X55                  |      |   --Z  AY=0.9322E+1 |
   |DESIGN CODE *   |                             |      |       AZ=0.4720E+1 |
   | LRFD 2001  *   ==============================   ===|===     PY=0.1330E+2 |
   |             *                                               PZ=0.1340E+3 |
   |             *  |&lt;---LENGTH (FT)=   30.00 --->|               RY=0.1340E
+1 |
   |*************                                                RZ=0.9129E+1 |
   |                                                                          |
   |                499.9 (KIP-FEET)                                          |
   |PARAMETER        |              L1  L1  L1                   CAPACITIES   |
   |IN KIP  INCH     |                                           IN KIP  INCH |
   |---------------  +          L1              L1               -------------|
   | KL/R-Y= 268.60  |                                           PNC=0.4791E+2|
   | KL/R-Z=  39.44  +      L1                      L1           pnc=0.0000E+0|
   | UNL    =   1.00 |                                           PNT=0.7290E+3|
   | CB     =   1.14 +   L1                              L1      pnt=0.0000E+0|
```

```
| PHIC  =   0.85  |                                             MNZ=0.6030E+4|
| PHIB  =   0.90  +                                             mnz=0.5999E+4|
| FYLD  =  50.00  |L0                                       L0  MNY=0.5604E+3|
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|    mny=0.0000E+0|
| DFF =     0.00 -27.8                                          VN =0.2517E+3|
| dff =     0.00               ABSOLUTE MZ ENVELOPE             vn =0.0000E+0|
|                                 (WITH LOAD NO.)                            |
|                                                                            |
|                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                 |
|                      -------------------------                             |
|                                                                            |
|                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
|                                                                            |
|      VALUE       0.0         66.7        0.0         0.0       499.9       |
|   LOCATION       0.0          0.0        0.0         0.0        15.0       |
|    LOADING        0             1          0           0           1       |
|                                                                            |
|****************************************************************************|
|*                                                                          *|
|*                         DESIGN SUMMARY  (KIP-FEET)                       *|
|*                         --------------                                   *|
|*                                                                          *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/             *|
|         FX             MY             MZ           LOCATION                |
|        ======================================================              |
|          PASS        LRFD-H1-1B-C      0.995             1                 |
|          0.00 C         0.00       -499.95           15.00                 |
|*                                                                          *|
|****************************************************************************|
|                                                                            |
|----------------------------------------------------------------------------|
```

V. AISC LRFD - Non Compact Wide Flange 1

To check the adequacy of a non-compact fully braced W shape to carry a uniformly distributed load of 5.5 kips/ft.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Table 5.4, , page 5-65.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

W14x90 shape. Beam is bent about its strong axis.

$F_y = 50$ ksi

Length = 30 ft

Validation

Check flange slenderness:

$$\lambda_p = 0.38\sqrt{E/F_y} = 0.38\sqrt{\tfrac{29{,}000}{50}} = 9.152 < \tfrac{b}{2t_f} = 10.2$$

$$F_L = 50 \text{ ksi} - 10 \text{ ksi} = 40 \text{ ksi}$$

$$\lambda_r = 0.83\sqrt{E/F_L} = 0.83\sqrt{\tfrac{29{,}000}{40}} = 22.35 > \tfrac{b}{2t_f} = 10.2$$

$$M_r = F_L S_x = 40 \times 143 = 5{,}720 \text{ in·k} \qquad \text{(F1-7)}$$

Flanges are non-compact. The nominal moment capacity for the limit state of flange local buckling is:

$$M_n = M_p - \left(M_p - M_r\right)\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right) \qquad \text{(A-F1-3)}$$

$$M_p = F_y Z_x = 50 \times 157 = 7{,}850 \text{ in·k}$$

$$M_n = 7{,}850 - \left(7{,}850 - 5{,}720\right)\left(\tfrac{10.2 - 9.152}{22.35 - 9.152}\right) = 7{,}681 \text{ in·k}$$

Note: As the flanges are fully braced, lateral-torsional buckling does not govern the capacity.

The flange local buckling limit state capacity:

$$\phi_b M_n = 0.9 \times (7{,}681) = 6{,}913 \text{ in·k}$$

The ultimate moment is:

$$M_u = \frac{w_u l^2}{8} = \frac{5.5(30)^2 12}{8} = 7{,}425 \text{ in-k}$$

The capacity ratio is: *7,425 / 6,913 = 1.074*

Results

Table 804: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	6,913	6,910	negligible	
Capacity ratio	1.074	1.074	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Non Compact Wide Flange 1.STD` is typically installed with the program.

Note: The UNT parameter is set to 1.0 in. in order to approximate a fully-braced condition.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
```

```
*
*  TABLE 5-4, PAGE 5-65, AISC LRFD 3RD ED.
*  OBJECTIVE : TO CHECK THE ADEQUACY OF A NON-COMPACT W SHAPE.
*
*  BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
*  CAPACITY OF A 30 FT LONG W14X90 IS 154/30=5.133 KIP/FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 30 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -5.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
FYLD 50 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                   Y        PROPERTIES    |
   |*************                                      |        IN INCH UNIT  |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX=0.2650E+2 |
   |            *  | ST  W14X90                  |      |   --Z  AY=0.6160E+1 |
   |DESIGN CODE *  |                             |      |        AZ=0.1373E+2 |
   | LRFD 2001  *  ===============================   ===|===     PY=0.7560E+2 |
   |            *                                                PZ=0.1570E+3 |
   |            *  |&lt;---LENGTH (FT)=   30.00 --->|                  RY=0.3696E
+1 |
   |*************                                                RZ=0.6140E+1 |
```

```
  |                                                                       |
  |               618.7 (KIP-FEET)                                        |
  |PARAMETER        |               L1  L1  L1                CAPACITIES  |
  |IN KIP  INCH     |                                         IN KIP  INCH |
  |---------------  +           L1              L1            -------------|
  | KL/R-Y=  97.40  |                                         PNC=0.5628E+3|
  | KL/R-Z=  58.63  +       L1                      L1        pnc=0.0000E+0|
  | UNL    =   1.00 |                                         PNT=0.1192E+4|
  | CB     =   1.00 +   L1                              L1    pnt=0.0000E+0|
  | PHIC   =   0.85 |                                         MNZ=0.6910E+4|
  | PHIB   =   0.90 +                                         mnz=0.7425E+4|
  | FYLD   =  50.00 |L0                                   L0  MNY=0.3280E+4|
  | NSF    =   1.00 +---+---+---+---+---+---+---+---+---+---|  mny=0.0000E+0|
  | DFF =      0.00 -34.4                                     VN =0.1663E+3|
  | dff =      0.00              ABSOLUTE MZ ENVELOPE         vn =0.0000E+0|
  |                                (WITH LOAD NO.)                         |
  |                                                                        |
  |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)              |
  |                     -------------------------                          |
  |                                                                        |
  |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z    |
  |                                                                        |
  |      VALUE       0.0        82.5        0.0         0.0       618.8    |
  |   LOCATION       0.0         0.0        0.0         0.0        15.0    |
  |    LOADING         0           1          0           0           1    |
  |                                                                        |
  |************************************************************************|
  |*                                                                      *|
  |*                        DESIGN SUMMARY  (KIP-FEET)                    *|
  |*                        --------------                                *|
  |*                                                                      *|
  |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/         *|
  |         FX              MY              MZ          LOCATION           |
  |        ======================================================          |
  |          FAIL        LRFD-H1-1B-C     1.074             1              |
  |          0.00 C           0.00      -618.75           15.00            |
  |*                                                                      *|
  |************************************************************************|
  |                                                                        |
  |------------------------------------------------------------------------|
```

V. AISC LRFD - Channel Shape Capacity

To check the adequacy of a fully braced C shape to carry a uniformly distributed ultimate load of 1.5 kips/ft.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Table 5.9, page 5-120.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

C12x25 shape. Beam is bent about its strong axis.

F_y = 36 ksi

Length = 20 ft

Validation

The beam is continuously braced, so by observation $L_b < L_p$.

The yielding limit state capacity in bending:

$$\phi_b M_n = \phi_b [F_y Z_x] = 0.9 \times [36 \text{ ksi} \times 29.4 \text{ in}^3] = 952.6 \text{ k·in}$$

The ultimate moment is:

$$M_u = \frac{w_u l^2}{8} = \frac{1.5(12)^2}{8} = 75 \text{ k-ft} = 900 \text{ k-in}$$

The capacity ratio is: $900 / 952.6 = 0.945$

Results

Table 805: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	952.6	952.6	none	
Capacity ratio	0.945	0.945	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Channel Shape Capacity.STD` is typically installed with the program.

Note: The UNT parameter is set to 1.0 in. in order to approximate a fully-braced condition.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  TABLE 5-9, PAGE 5-120, AISC LRFD 3RD ED.
*  OBJECTIVE : TO CHECK THE ADEQUACY OF A C SHAPE.
*
*  BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
*  CAPACITY OF A 20 FT LONG C12X25 IS 31.8/20=1.59 KIP/FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST C12X25
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -1.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD. Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y         PROPERTIES  |
  |*************                                       |         IN INCH UNIT |
  |            *  |=============================|    |===       ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |    |          AX=0.7340E+1 |
  |            *  | ST  C12X25                  |    |      --Z AY=0.4644E+1 |
  |DESIGN CODE *  |                             |    |          AZ=0.2037E+1 |
  | LRFD 2001  *  ===============================    |===       PY=0.3820E+1 |
  |            *                                              PZ=0.2940E+2 |
  |            *  |&lt;---LENGTH (FT)=   20.00 --->|               RY=0.7786E
+0 |
  |*************                                              RZ=0.4429E+1 |
  |                                                                          |
  |                75.0 (KIP-FEET)                                           |
  |PARAMETER        |               L1  L1  L1                CAPACITIES   |
  |IN KIP  INCH     |                                         IN KIP  INCH |
  |---------------  +           L1              L1            -------------|
  | KL/R-Y= 308.23  |                                         PNC=0.1636E+2|
  | KL/R-Z=  54.18  +       L1                      L1        pnc=0.0000E+0|
  | UNL    =   1.00 |                                         PNT=0.2378E+3|
  | CB     =   1.00 +   L1                              L1    pnt=0.0000E+0|
  | PHIC   =   0.85 |                                         MNZ=0.9526E+3|
  | PHIB   =   0.90 +                                         mnz=0.9000E+3|
  | FYLD   =  36.00 |L0                                  L0  MNY=0.9102E+2|
  | NSF    =   1.00 +---+---+---+---+---+---+---+---+---+---|   mny=0.0000E+0|
```

```
| DFF =     0.00  -4.2                                          VN =0.9028E+2|
| dff =     0.00                 ABSOLUTE MZ ENVELOPE           vn =0.0000E+0|
|                                   (WITH LOAD NO.)                          |
|                                                                            |
|                        MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
|                        -------------------------                           |
|                                                                            |
|                  AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
|                                                                            |
|        VALUE      0.0         15.0        0.0         0.0        75.0     |
|     LOCATION      0.0          0.0        0.0         0.0        10.0     |
|      LOADING        0            1          0           0           1     |
|                                                                            |
|***************************************************************************|
|*                                                                         *|
|*                            DESIGN SUMMARY  (KIP-FEET)                   *|
|*                            --------------                               *|
|*                                                                         *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/            *|
|         FX             MY              MZ           LOCATION              |
|        ======================================================             |
|          PASS        LRFD-H1-1B-C     0.945              1                |
|          0.00 C          0.00       -75.00            10.00               |
|*                                                                         *|
|***************************************************************************|
|                                                                            |
|----------------------------------------------------------------------------|
```

V. AISC LRFD - MC Shape Capacity

To check the adequacy of a fully braced MC shape to carry a uniformly distributed ultimate load of 1.34 kips/ft.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Table 5.10, page 5-124.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

MC10x41.1 shape. Beam is bent about its strong axis.

$F_y = 36$ ksi

Length = 25 ft

Validation

The beam is continuously braced, so by observation $L_b < L_p$.

The yielding limit state capacity in bending:

$$\phi_b M_n = \phi_b[F_y Z_x] = 0.90\times[36 \text{ ksi} \times 39.3 \text{ in}^3] = 1{,}273 \text{ k}\cdot\text{in}$$

The ultimate moment is:

$$M_u = \frac{w_u l^2}{8} = \frac{1.34(25)^2 12}{8} = 1{,}256 \text{ k-in}$$

The capacity ratio is: *1,256 / 1,273 = 0.987*

Results

Table 806: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	1,273	1,273	none	
Capacity ratio	0.987	0.987	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - MC Shape Capacity.STD` is typically installed with the program.

Note: The UNT parameter is set to 1.0 in. in order to approximate a fully-braced condition.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  TABLE 5-10, PAGE 5-124, AISC LRFD 3RD ED.
*  OBJECTIVE : TO CHECK THE ADEQUACY OF AN MC SHAPE.
*
*  BEAM IS FULLY BRACED. ACCORDING TO ABOVE REFERENCE,
*  CAPACITY OF A 25 FT LONG MC10X41 IS APPROX. 34/25=1.36 KIP/FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 25 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST MC10X41
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
```

```
LOAD 1
MEMBER LOAD
1 UNI GY -1.34
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
* FULLY BRACED CONDITION IS ACHIEVED WITH A UNT OF 1.0 INCH
UNT 1 ALL
FYLD 36 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
   |-----------------------------------------------------------------------|
   |                                                    Y        PROPERTIES |
   |*************                                       |        IN INCH UNIT |
   |            *  |=============================|    |===       ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |    |          AX=0.1210E+2 |
   |            *  | ST  MC10X41                 |    |      --Z  AY=0.7960E+1 |
   |DESIGN CODE *  |                             |    |          AZ=0.3312E+1 |
   | LRFD 2001  *  ===============================    |===       PY=0.9490E+1 |
   |            *                                               PZ=0.3930E+2 |
   |            *  |<---LENGTH (FT)=   25.00 --->|                 RY=0.1139E
+1 |
   |*************                                               RZ=0.3602E+1 |
   |                                                                         |
   |                104.7 (KIP-FEET)                                         |
   |PARAMETER        |               L1  L1  L1                 CAPACITIES   |
   |IN KIP  INCH     |                                          IN KIP  INCH |
   |---------------  +           L1              L1              -------------|
   | KL/R-Y= 263.37  |                                          PNC=0.3693E+2|
   | KL/R-Z=  83.28  +       L1                       L1        pnc=0.0000E+0|
   | UNL    =   1.00  |                                          PNT=0.3920E+3|
   | CB     =   1.00  +   L1                               L1     pnt=0.0000E+0|
   | PHIC   =   0.85  |                                          MNZ=0.1273E+4|
   | PHIB   =   0.90  +                                          mnz=0.1256E+4|
   | FYLD   =  36.00  |L0                                   L0  MNY=0.2362E+3|
   | NSF    =   1.00  +---+---+---+---+---+---+---+---+---|  mny=0.0000E+0|
   | DFF =      0.00  -5.8                                     VN =0.1547E+3|
   | dff =      0.00              ABSOLUTE MZ ENVELOPE           vn =0.0000E+0|
   |                                (WITH LOAD NO.)                         |
   |                                                                         |
   |                         MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)           |
   |                         -------------------------                       |
   |                                                                         |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
   |                                                                         |
   |      VALUE       0.0         16.8        0.0         0.0       104.7     |
   |    LOCATION      0.0          0.0        0.0         0.0        12.5     |
   |     LOADING        0            1          0           0           1     |
   |                                                                         |
```

```
|*************************************************************************|
|*                                                                       *|
|*                     DESIGN SUMMARY  (KIP-FEET)                        *|
|*                     --------------                                    *|
|*                                                                       *|
|*      RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|        FX                 MY            MZ          LOCATION           |
|       ======================================================           |
|         PASS        LRFD-H1-1B-C     0.987            1                |
|         0.00 C          0.00      -104.69           12.50              |
|*                                                                       *|
|*************************************************************************|
|                                                                         |
|-------------------------------------------------------------------------|
```

V. AISC LRFD - Non Compact Wide Flange 2

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis. The deflection at mid-span due to two 10 kip concentrated loads acting at the third points along the beam is also to be calculated.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 5.2, case (a), page 5-16.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

W21x48 shape

$F_y = 50$ ksi

Length = 40 ft

$L_b = 60$ in.

Validation

Evaluate section for lateral-torsional buckling:

The plastic limiting unbraced length is given by:

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76(1.66)\sqrt{\frac{29{,}000}{50}} = 70.4 \text{ in.}$$

$L_b < L_p$; thus, the member is plastic.

Check flange slenderness:

$$\lambda_p = 0.38\sqrt{E / F_y} = 0.38\sqrt{\frac{29{,}000}{50}} = 9.152 < \frac{b}{2t_f} = 9.47$$

$F_L = 50$ ksi - 10 ksi = 40 ksi

$$\lambda_r = 0.83\sqrt{E/F_L} = 0.83\sqrt{\frac{29{,}000}{40}} = 22.35 > \frac{b}{2t_f} = 9.47$$

$$M_r = F_L S_x = 40 \times 93.0 = 3{,}720 \text{ in}\cdot\text{k} \quad \text{(F1-7)}$$

Flanges are non-compact. The nominal moment capacity for the limit state of flange local buckling is:

$$M_n = M_p - \left(M_p - M_r\right)\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right) \quad \text{(A-F1-3)}$$

$$M_p = F_y Z_x = 50 \times 107 = 5{,}350 \text{ in}\cdot\text{k}$$

$$M_n = 5{,}350 - \left(5{,}350 - 3{,}720\right)\left(\frac{9.47 - 9.152}{22.35 - 9.152}\right) = 5{,}311 \text{ in}\cdot\text{k}$$

The flange local buckling limit state capacity:

$$\phi_b M_n = 0.9 \times (5{,}311) = 4{,}780 \text{ in}\cdot\text{k}$$

The ultimate moment is:

$$M_u = P_u\left(l/3\right) = 10\left(\frac{30 \times 12}{3}\right) = 1{,}600 \text{ in-k}$$

The capacity ratio is: *1,600 / 4,780 = 0.335*

The deflection at mid-span for a beam loaded at third points:

$$\Delta_{\max} = \frac{Pl^3}{28EI} = \frac{10 \times (40 \times 12)^3}{28 \times 29{,}000 \times 959} = 1.42 \text{ in.}$$

Results

Table 807: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	4,780	4,780.24	negligible	
Capacity ratio	0.335	0.335	none	
Deflection (in)	1.42	1.4114	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Non Compact Wide Flange 2.STD` is typically installed with the program.

The `UNT 5` parameter specifies an unbraced length of 5'.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.2, CASE (A), PAGE 5-16, AISC LRFD 3RD ED.
```

```
*  CAPACITY (MNZ) SHOULD BE ABOUT 398 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 PINNED
2 FIXED BUT MZ
*
LOAD 1
MEMBER LOAD
1 CON GY -10 13.333
1 CON GY -10 26.667
*
PERFORM ANALYSIS
*
SECTION 0.5 ALL
PRINT SECTION DISPL
*
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
UNT 5 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - INCH
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     1       0.0000    0.0000    0.0000    0.0000   -0.3644    0.0000
                     0.0000   -0.7057    0.0000    0.0000   -1.0010    0.0000
                     0.0000   -1.2273    0.0000    0.0000   -1.3654    0.0000
                     0.0000   -1.4114    0.0000    0.0000   -1.3654    0.0000
                     0.0000   -1.2273    0.0000    0.0000   -1.0010    0.0000
                     0.0000   -0.7057    0.0000    0.0000   -0.3644    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    1.41141   AT     240.00  LOAD    1   L/DISP=    340
```

```
        ************ END OF SECT DISPL RESULTS ***********
   40. *
   41. PARAMETER 1
   42. CODE LRFD
   43. MAIN 1 ALL
   44. FYLD 7200 ALL
   45. UNT 5 ALL
   46. TRACK 1 ALL
   47. CHECK CODE ALL
 STEEL DESIGN
      BENDING CAPACITY PER AISC LRFD 3RD ED                    -- PAGE NO.
4
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W21X48                   (AISC SECTIONS)
                           PASS     LRFD-H1-1B-C       0.335         1
                        0.00 C          0.00        -133.33       13.33
 +---------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN              |
 | PNC=   35.84  PNT=  634.50  MNZ= 4780.24  MNY=  636.75  VN=  194.67 |
 +---------------------------------------------------------------------+
```

V. AISC LRFD - Non Compact Wide Flange 3

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 5.2, case (b), page 5-16.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

W21x48 shape

$F_y = 50$ ksi

Length = 40 ft

$L_b = 160$ in.

Validation

The plastic limiting unbraced length is given by:

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76\left(1.66\right)\sqrt{\frac{29{,}000}{50}} = 70.4 \text{ in.}$$

The inelastic limiting unbraced length is given by:

$$L_r = \frac{r_y X_1}{F_L}\sqrt{1+\sqrt{1+X_2 F_L^{\ 2}}} \tag{F1-6}$$

where

F_L = $F_{yf} - F_r = 50\ ksi - 10\ ksi = 40\ ksi$

X_1 = $\frac{\pi}{S_x}\sqrt{\frac{EGJA}{2}} = \frac{\pi}{93.0}\sqrt{\frac{29{,}000(11{,}200)(0.803)(14.1)}{2}} = 1{,}449$

X_2 = $4\frac{C_w}{I_y}\left(\frac{S_x}{GJ}\right)^2 = 4\left(\frac{3{,}950}{38.7}\right)\left[\frac{93.0}{11{,}200(0.803)}\right]^2 = 0.0437$

$$L_r = \frac{1.66 \times 1{,}449}{40}\sqrt{1+\sqrt{1+0.0437(40)^2}} = 184.6 \text{ in.}$$

$L_p < L_b < L_r$; thus, the section is inelastic.

The nominal flexural strength is:

$$M_n = C_b\left[M_p - \left(M_p - M_r\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] < M_p \tag{F1-2}$$

$$M_r = F_L S_x = 40 \times 93.0 = 3{,}720 \text{ in}\cdot\text{k} \tag{F1-7}$$

$$M_p = F_y Z_x = 50 \times 107 = 5{,}350 \text{ in}\cdot\text{k}$$

Conservatively assume $C_b = 1.0$ (this is the default behavior in STAAD.Pro when the parameter is not directly specified).

$$M_n = 5{,}350 - \left(5{,}350 - 3{,}720\right)\left(\frac{160 - 70.4}{184.6 - 70.4}\right) = 4{,}071 \text{ in}\cdot\text{k}$$

The lateral-torsional buckling limit state capacity:

$$\phi_b M_n = 0.9\times(4{,}071) = 3{,}664 \text{ in}\cdot\text{k}$$

Comparison

The reported STAAD.Pro value is based on the term `MNZ` in the STAAD.Pro output.

Table 808: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design flexural strength, $\phi_b M_n$ (in·kips)	3,664	3,658.52	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Non Compact Wide Flange 3.STD` is typically installed with the program.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.2, CASE (B), PAGE 5-16, AISC LRFD 3RD ED.
```

```
*  CAPACITY (MNZ) SHOULD BE ABOUT 306 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 PINNED
2 FIXED BUT MZ
*
LOAD 1
MEMBER LOAD
1 CON GY -10 13.333
1 CON GY -10 26.667
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
UNT 13.333 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                        ***********************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W21X48                     (AISC SECTIONS)
                           PASS     LRFD-H1-1B-C       0.437        1
                        0.00 C          0.00       -133.33      13.33
 +---------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN               |
 | PNC=   35.84  PNT=  634.50  MNZ= 3658.52  MNY=  636.75  VN=  194.67 |
 +---------------------------------------------------------------------+
```

V. AISC LRFD - Wide Flange Compression Capacity 4

To determine the design bending strength of a non-compact American wide flange section bent about its strong axis.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 5.2, case (c), page 5-16.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

W21x48 shape

$F_y = 50$ ksi

Length = 40 ft

$L_b = 480$ in. (braced at end points only)

Validation

The inelastic limiting unbraced length is given by:

$$L_r = \frac{r_y X_1}{F_L}\sqrt{1 + \sqrt{1 + X_2 F_L{}^2}} \qquad \text{(F1-6)}$$

where

$$F_L = F_{yf} - F_r = 50 \text{ ksi} - 10 \text{ ksi} = 40 \text{ ksi}$$

$$X_1 = \frac{\pi}{S_x}\sqrt{\frac{EGJA}{2}} = \frac{\pi}{93.0}\sqrt{\frac{29{,}000(11{,}200)(0.803)(14.1)}{2}} = 1{,}449$$

$$X_2 = 4\frac{C_w}{I_y}\left(\frac{S_x}{GJ}\right)^2 = 4\left(\frac{3{,}950}{38.7}\right)\left[\frac{93.0}{11{,}200(0.803)}\right]^2 = 0.0437$$

$$L_r = \frac{1.66 \times 1{,}449}{40}\sqrt{1 + \sqrt{1 + 0.0437(40)^2}} = 184.6 \text{ in.}$$

$L_b > L_r$, thus the section is subject to elastic lateral-torsional buckling.

The modification factor, C_b *is given by:*

$$C_b = \frac{12.5M_{\max}}{2.5M_{\max} + 3M_A + 4M_B + 3M_C} \qquad \text{(F1-3)}$$

Since the moment diagram is symmetric about the mid-span point of the beam, the maximum moment is the same as the mid-span moment and the quarter-point moments are equal.

$$M_{\max} = M_B = \frac{wl^2}{8}$$

The quarter-point moments of a uniformly loaded beam are likewise equal:

$$M_A = M_C = \frac{w(l/4)}{2}\left(l - \frac{l}{4}\right) = \frac{3wl^2}{32}$$

The wl^2 terms factor out and the constants simply as follows:

$$C_b = \frac{12.5(1/8)}{2.5(1/8) + 3(3/32) + 4(1/8) + 3(3/32)} = \frac{12.5(1/8)}{6.5(1/8) + 6(3/32)} = 1.14$$

The critical buckling moment is given by:

$$M_{cr} = \frac{C_b S_x X_1 \sqrt{2}}{L_b / r_y} \sqrt{1 + \frac{{X_1}^2 X_2}{2(L_b / r_y)^2}} \quad \text{(F1-13)}$$

$$M_{cr} = \frac{1.14(93.0)(1{,}449)\sqrt{2}}{480 / 1.66} \sqrt{1 + \frac{(1{,}449)^2(0.0437)}{2(480 / 1.66)^2}} = 935.0 \text{ in·k}$$

The lateral-torsional buckling limit state capacity:

$$\phi_b M_n = 0.9 \times (935.0) = 841.5 \text{ in·k}$$

Results

Table 809: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Modification factor, C_b	1.14	1.14	none	
Design flexural strength, $\phi_b M_n$ (in·kips)	841.5	834.4	<1%	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Wide Flange Compression Capacity 4.STD` is typically installed with the program.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 5.2, CASE (C), PAGE 5-16, AISC LRFD 3RD ED.
*  CAPACITY (MNZ) SHOULD BE ABOUT 70.2 KIP-FT
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W21X48
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
```

```
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 PINNED
2 FIXED BUT MZ
*
LOAD 1
MEMBER LOAD
1 CON GY -10 13.333
1 CON GY -10 26.667
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
CB 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y         PROPERTIES  |
  |*************                                       |         IN INCH UNIT |
  |            *  |=============================|   ===|===     ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |      |       AX=0.1410E+2 |
  |            *  | ST  W21X48                  |      |   --Z  AY=0.7210E+1 |
  |DESIGN CODE *  |                             |      |        AZ=0.4667E+1 |
  | LRFD 2001  *  =============================   ===|===      PY=0.1490E+2 |
  |            *                                               PZ=0.1070E+3 |
  |            *  |<---LENGTH (FT)=   40.00 --->|                 RY=0.1657E
+1 |
  |*************                                               RZ=0.8247E+1 |
  |                                                                          |
  |               133.3 (KIP-FEET)                                           |
  |PARAMETER        |              L1  L1  L1                CAPACITIES     |
  |IN KIP  INCH     |                                        IN KIP  INCH   |
  |---------------  +          L1              L1            -------------|
  | KL/R-Y= 289.73  |                                        PNC=0.3584E+2|
  | KL/R-Z=  58.20  +      L1                      L1        pnc=0.0000E+0|
  | UNL   = 480.00  |                                        PNT=0.6345E+3|
  | CB    =   1.14  +                                        pnt=0.0000E+0|
  | PHIC  =   0.85  |   L1                             L1    MNZ=0.8344E+3|
  | PHIB  =   0.90  +                                        mnz=0.1600E+4|
  | FYLD  =  50.00  |L0                                    L0  MNY=0.6367E+3|
  | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|  mny=0.0000E+0|
  | DFF =     0.00  -7.4                                     VN =0.1947E+3|
  | dff =     0.00              ABSOLUTE MZ ENVELOPE         vn =0.1000E+2|
  |                                (WITH LOAD NO.)                        |
  |                                                                       |
```

```
|                    MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
|                    -------------------------                            |
|                                                                         |
|              AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
|                                                                         |
|     VALUE      0.0        10.0        0.0         0.0       133.3       |
|  LOCATION      0.0         0.0        0.0         0.0        13.3       |
|   LOADING        0           1          0           0           1       |
|                                                                         |
|*************************************************************************|
|*                                                                       *|
|*                       DESIGN SUMMARY  (KIP-FEET)                      *|
|*                       --------------                                  *|
|*                                                                       *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/           *|
|         FX              MY             MZ          LOCATION            |
|        ======================================================          |
|          FAIL       LRFD-H1-1B-C     1.917              1              |
|          0.00 C         0.00      -133.33            13.33             |
|*                                                                       *|
|*************************************************************************|
|                                                                         |
|-------------------------------------------------------------------------|
```

V. AISC LRFD - Select Wide Flange 2

To find the optimum W shape with a flexural design strength of 150 ft·kips. Beam is braced at ends and at mid-span point.

Reference

AISC. *Load and Resistance Factor Design Specification*, 3rd Edition. 1999. Example 5.4, page 5-21.

AISC. *Manual of Steel Construction*, 3rd Edition. 1999.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

The beam should also satisfy a deflection limit of 1.0 inches under service load.

F_y = 50 ksi

Length = 20 ft

L_b = 10 ft

Service load of 2.0 kip/ft. Deflection limit under service load = 1.0 in.

Validation

From the chart on p.4-160 of the *Manual of Steel Construction*, the most efficient wide-flange section to resist 150 kip-ft of moment with an unbraced length of 10 ft is a W16x31. From this same chart, the design bending strength can be taken as 150 kip-ft (the curve crosses at approximately the same point as the demand load).

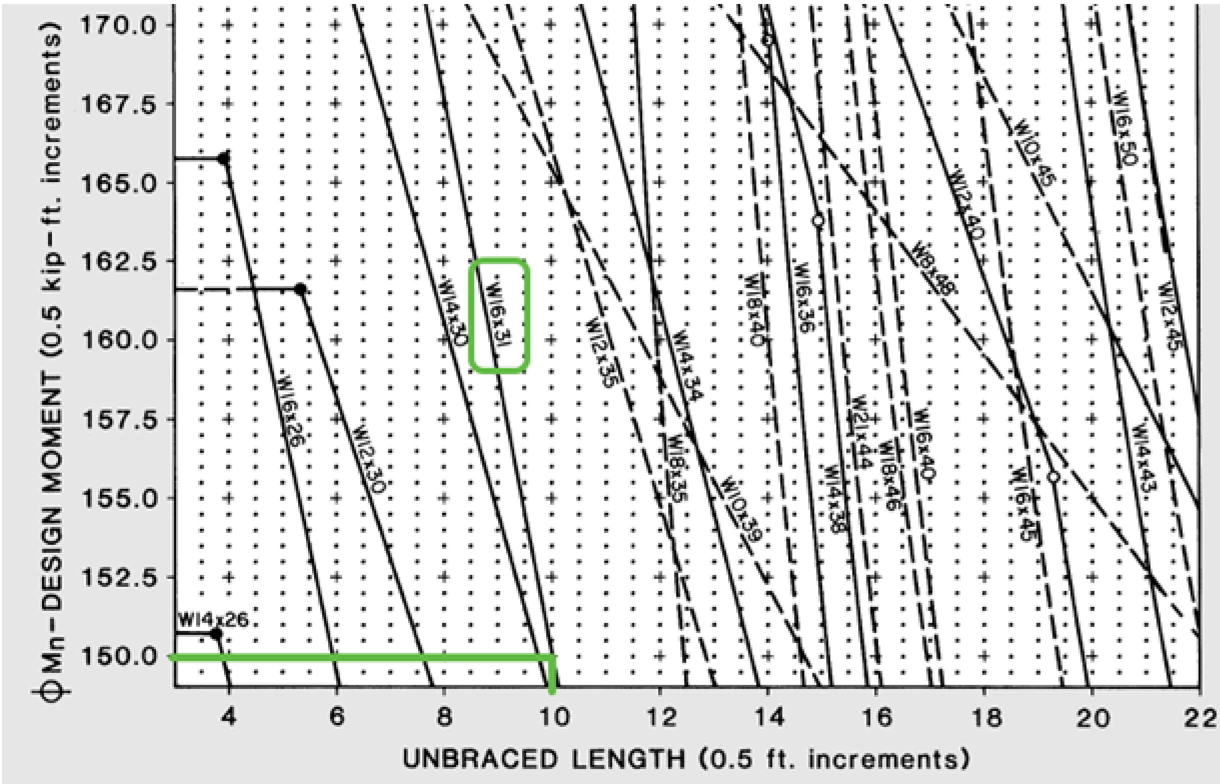


Figure 503: Design flexural strength of beams with unbraced length greater than L_p

The deflection limit under service conditions can be met by determining the minimum required moment of inertia needed.

$$1 \text{ in.} = \frac{5(2.0 \text{ k/ft})(20 \text{ ft})^4(12 \text{ in/ft})^3}{384(29,\ 000 \text{ ksi})I}$$

Solving for the minimum $I = 248\ in^4$. A W16x31 has an $I_x = 375\ in^4$, so it will have less deflection than the limit under service conditions.

Results

Table 810: Comparison of results

Result Type	Theory	STAAD.Pro	Difference	Comments
Optimum Section	W16x31	W16x31	none	
Design bending strength, $\phi_b M_n$ (in·kips)	1,800 (150 ft·kips)	1,801.66	negligible	

Result Type	Theory	STAAD.Pro	Difference	Comments
Maximum permissible deflection (in)	1.0	0.6584	N/A	The STAAD.Pro reported maximum deflection is less than 1.0 in; therefore this section satisfies the requirement. However, this value is not to be directly compared with the limit itself.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Select Wide Flange 2.STD` is typically installed with the program.

```
STAAD SPACE BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE 5.4, PAGE 5-21, AISC LRFD 3RD ED.
*  OBJECTIVE : TO FIND THE OPTIMUM W SHAPE WITH A FLEXURAL
*  DESIGN STRENGTH OF 150 KIP-FT.
*
*  ACCORDING TO ABOVE REFERENCE, THE SECTION SHOULD BE W16X31
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 20 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W12X26
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 PINNED
2 FIXED BUT MZ
LOAD 1
MEMBER LOAD
1 UNI GY -2
LOAD COMBINATION 2
1 1.5
```

```
PERFORM ANALYSIS
LOAD LIST 2
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
MAIN 1 ALL
UNT 120 ALL
FYLD 50 ALL
TRACK 1 ALL
SELECT ALL
LOAD LIST ALL
PERFORM ANALYSIS
LOAD LIST 1
SECTION 0.5 ALL
PRINT SECTION DISPL
LOAD LIST 2
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.Pro MEMBER SELECTION - (LRFD 3RD EDITION)   v1.0
                       **************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W16X31                   (AISC SECTIONS)
                           PASS     LRFD-H1-1B-C       0.999         2
                        0.00 C          0.00       -1800.00      120.00
 +-----------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN                |
 | PNC=   45.93  PNT=  410.85  MNZ= 1801.66  MNY=  302.71  VN=  118.06 |
 +-----------------------------------------------------------------------+
   ************** END OF TABULATED RESULT OF DESIGN **************
    46. LOAD LIST ALL
    47. PERFORM ANALYSIS
    48. LOAD LIST 1
    49. SECTION 0.5 ALL
    50. PRINT SECTION DISPL
  SECTION  DISPL
      BENDING PER AISC LRFD 3RD ED                             -- PAGE NO.
4
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - INCH
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     1       0.0000    0.0000    0.0000    0.0000   -0.1731    0.0000
                     0.0000   -0.3331    0.0000    0.0000   -0.4690    0.0000
                     0.0000   -0.5722    0.0000    0.0000   -0.6365    0.0000
                     0.0000   -0.6584    0.0000    0.0000   -0.6365    0.0000
                     0.0000   -0.5722    0.0000    0.0000   -0.4690    0.0000
                     0.0000   -0.3331    0.0000    0.0000   -0.1731    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    0.65839   AT     120.00  LOAD    1   L/DISP=    364
         ************ END OF SECT DISPL RESULTS ***********
```

V. AISC LRFD - Load Capacity of 3 Wide Flange Beams

To find the design flexural strength capacity of 3 beams of different spans using the data in Table 5-4 of AISC LRFD 3rd edition.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 5.5, page 5-21.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Details

W21x62 section.

$$F_y = 50 \text{ ksi}$$

40 ft, 30 ft, and 8 ft spans are investigated. All beams are fully braced.

Validation

Bending Capacity

The beam is fully braced so it can reach full plastic capacity:

$$\phi M_n = 0.9 \times 50 \times 144 = 6{,}480 \text{ in-k}$$

The total load capacity on the beam is then calculated by setting the ultimate capacity equal to the load, $\phi M_n = M_u = W \times l\ /\ 8$ and then solving for the total load:

$$W_1 = 8\left(\phi M_n / l\right) = 8\left(\frac{6{,}480}{40 \times 12}\right) = 108 \text{ kips}$$

$$W_2 = 8\left(\phi M_n / l\right) = 8\left(\frac{6{,}480}{30 \times 12}\right) = 144 \text{ kips}$$

Shear Capacity

Check for web slenderness:

$$\frac{h}{t_w} = 46.9 < 2.45\sqrt{E / F_{yw}} = 2.45\sqrt{29{,}000 / 50} = 59.0$$

The nominal shear capacity of a W21x62 section:

$$V_n = 0.6F_{yw}A_w = 0.6 \times 50 \times 0.4 \times 21.0 = 252 \text{ kips}$$

$$\phi V_n = 0.9 \times 252 = 226.8 \text{ kips}$$

The total load capacity on the beam is then calculated by setting the ultimate capacity equal to the load, $\phi V_n = V_u = W\ /\ 2$ and then solving for the total load:

$$W_3 = 2\left(\phi V_n\right) = 2\left(226.8\right) = 453.6 \text{ kips}$$

This governs the total load capacity for the short span (8') case.

Using the total weight values from the reference, the following distributed loads are calculated:

Member	W (kips)	l (ft)	w (kips/ft)
1	108	40	2.7
2	144	30	4.8
3	454	8	56.75

Results

There is no direct comparison of the results from STAAD.Pro with the Reference, as the program does not directly compute the total member load. However, when the total member load is applied as a distributed load (see previous table), the Utilization ratio will then nominally be unity (1.0) for a beam loaded at its capacity.

Table 811: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
Beam #1: 30 ft span	Bending capacity, ϕM_n (in·k)	6,480	6,480	none	
	Utilization ratio	1.0	1.0	none	
Beam #2: 20 ft span	Bending capacity, ϕM_n (in·k)	6,480	6,480	none	
	Utilization ratio	1.0	1.0	none	
Beam #3: 8 ft span	Shear capacity, ϕV_n (kips)	226.8	226.8	none	
	Utilization ratio	1.0	1.001	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Load Capacity of 3 Wide Flange Beams.STD` is typically installed with the program.

Note: The UNT parameter is set to 1.0 in. in order to approximate a fully-braced condition.

```
STAAD SPACE BENDING CAPACITY PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  OBJECTIVE : TO DETERMINE THE MAXIMUM UNIFORMLY
*  DISTRIBUTED LOAD CARRYING CAPACITY OF 3 BEAMS.
*
*  EXAMPLE PROBLEM 5.5 PAGE 5-21, AISC LRFD 3RD ED.
```

```
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 40 0 0; 3 0 10 0; 4 30 10 0; 5 0 20 0; 6 8 20 0;
MEMBER INCIDENCES
1 1 2; 2 3 4; 3 5 6;
MEMBER PROPERTY AMERICAN
1 TO 3 TABLE ST W21X62
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 3 5 PINNED
2 4 6 FIXED BUT MZ
*
LOAD 1
MEMBER LOAD
* ON MEMBER 1, W=108/40=2.7
1 UNI GY -2.7
* ON MEMBER 2, W=144/30
2 UNI GY -4.8
* ON MEMBER 3, W=454/8
3 UNI GY -56.75
*
PERFORM ANALYSIS
PARAMETER 1
CODE LRFD
MAIN 1 ALL
FYLD 7200 ALL
* FULLY BRACED CONDITION CAN BE ACHIEVED BY SETTING UNT TO 1.0 INCH
UNT 0.1 ALL
TRACK 1 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                         ***********************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
 *    1  ST   W21X62                   (AISC SECTIONS)
                           FAIL     LRFD-H1-1B-C       1.000         1
                        0.00 C          0.00        -540.00       20.00
 +-----------------------------------------------------------------------+
 | DESIGN STRENGTHS FOR MEMBER     1       UNITS - KIP IN                |
 | PNC=   53.25  PNT=  823.50  MNZ= 6480.00  MNY=  942.05  VN=  226.80 |
 +-----------------------------------------------------------------------+
      2  ST   W21X62                   (AISC SECTIONS)
                           PASS     LRFD-H1-1B-C       1.000         1
                        0.00 C          0.00        -540.00       15.00
```

```
+------------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     2       UNITS - KIP IN                 |
| PNC=   94.66  PNT=  823.50  MNZ= 6480.00  MNY=  942.05  VN=  226.80 |
+------------------------------------------------------------------------+
*    3  ST   W21X62                   (AISC SECTIONS)
                          FAIL     SHEAR-Y           1.001         1
                      0.00 C          0.00          0.00        0.00
+------------------------------------------------------------------------+
| DESIGN STRENGTHS FOR MEMBER     3       UNITS - KIP IN                 |
| PNC=  608.18  PNT=  823.50  MNZ= 6161.00  MNY=  942.05  VN=  226.80 |
+------------------------------------------------------------------------+
  ************** END OF TABULATED RESULT OF DESIGN **************
```

V. AISC LRFD - Tension and Strong Axis Bending

To check the adequacy of an ASTM A992 W10X22 subject to axial tension and flexure about the strong axis.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 6.1 case(b), page 6-5.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

W10x22 section.

$F_y = 50$ ksi

$F_u = 65$ ksi

$P_u = 55$ kips

$M_{uy} = 0$, $M_{uz} = 55$ ft·kips

$L_b = 4$ ft

Comparison

The reported STAAD.Pro value is based on the terms PNT and MNZ in the STAAD.Pro output.

Table 812: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design tensile strength, $\phi_t P_n$ (kips)	292.0	292	none
Design bending strength, $\phi_b M_{ny}$ (ft·kips)	97.5	97.5 (1,170 in·kips)	none
Utilization ratio	0.98	0.98	none

Note: Fully braced condition can be achieved by setting the UNT parameter equal to 1.0 inch.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Tension and Strong Axis Bending.STD` is typically installed with the program.

```
STAAD SPACE COMBINED AXIAL TENSION + FLEXURE PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 6.1, CASE (B), PAGE 6-5, AISC LRFD 3RD ED.
*  INTERACTION RATIO SHOULD BE ABOUT 0.98
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W10X22
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 29000
POISSON 0.3
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 140
2 MZ 55
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE LRFD
FYLD 50 ALL
FU 65 ALL
UNT 48 ALL
UNB 48 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES   |
```

```
   |*************                                     |        IN INCH UNIT |
   |            *  |=============================|   ===|===     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX=0.6490E+1 |
   |            *  | ST  W10X22                  |      |   --Z  AY=0.2448E+1 |
   |DESIGN CODE *  |                             |      |        AZ=0.2760E+1 |
   | LRFD 2001  *  ===============================   ===|===     PY=0.6100E+1 |
   |            *                                                PZ=0.2600E+2 |
   |            *  |&lt;---LENGTH (FT)=   10.00 --->|                RY=0.1325E
+1 |
   |*************                                                RZ=0.4264E+1 |
   |                                                                          |
   |                 55.0 (KIP-FEET)                                          |
   |PARAMETER        |                              L1  L1  L1  CAPACITIES    |
   |IN KIP  INCH     |                              L1  L1  L1  IN KIP  INCH  |
   |---------------  +                              L1  L1  L1  -------------|
   | KL/R-Y=  90.54  |                              L1  L1  L1  PNC=0.1515E+3|
   | KL/R-Z=  28.14  +                              L1  L1  L1  pnc=0.0000E+0|
   | UNL    =  48.00  |L1 L1  L1  L1  L1  L1  L1  L1              PNT=0.2920E+3|
   | CB     =   1.00  +L1 L1  L1  L1  L1  L1  L1  L1              pnt=0.1400E+3|
   | PHIC   =   0.85  |L1 L1  L1  L1  L1  L1  L1  L1              MNZ=0.1170E+4|
   | PHIB   =   0.90  +L1 L1  L1  L1  L1  L1  L1  L1              mnz=0.6600E+3|
   | FYLD   =  50.00  |L1 L1  L1  L1  L1  L1  L1  L1              MNY=0.2677E+3|
   | NSF    =   1.00  +---+---+---+---+---+---+---+---+---|   mny=0.0000E+0|
   | DFF =      0.00  55.0                                      VN =0.6610E+2|
   | dff =      0.00             ABSOLUTE MZ ENVELOPE           vn =0.0000E+0|
   |                               (WITH LOAD NO.)                            |
   |                                                                          |
   |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
   |                      -------------------------                           |
   |                                                                          |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
   |                                                                          |
   |      VALUE    -140.0         0.0        0.0         0.0        55.0     |
   |   LOCATION       0.0         0.0        0.0         0.0         0.0     |
   |    LOADING         1           0          0           0           1     |
   |                                                                          |
   |**************************************************************************|
   |*                                                                        *|
   |*                         DESIGN SUMMARY  (KIP-FEET)                     *|
   |*                         --------------                                 *|
   |*                                                                        *|
   |*      RESULT/     CRITICAL COND/     RATIO/        LOADING/             *|
   |        FX              MY             MZ          LOCATION              |
   |       ======================================================            |
   |         PASS        LRFD-H1-1A-T     0.980             1                |
   |       140.00 T          0.00       -55.00            0.00               |
   |*                                                                        *|
   |**************************************************************************|
   |                                                                          |
   |--------------------------------------------------------------------------|
```

V. AISC LRFD - Compression and Biaxial Bending

To check the adequacy of an ASTM A992 W14X176 beam subject to axial compression and biaxial bending.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 6.2, page 6-6.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

W14x176 section.

$F_y = 50$ ksi

$P_u = 1{,}400$ kips

$M_{uy} = 70$ ft ·kips, $M_{uz} = 200$ ft ·kips

$L_b = 14$ ft

Reverse curvature bending with equal end moments for both axis.

Hand Calculations

According to page 16.1 - 97 of the LRFD 3rd ed. code,

$$M_p = F_y \cdot Z \leq 1.5F_y \cdot S$$

For a W14x176,

$$0.9\ (1.5F_y \cdot S) = 0.9\ [1.5\ (50)(107)] = 7{,}222 \text{ kip-in} = 601.8 \text{ kip-ft},$$

which is lower than $(0.9\ Fy\ .\ Z) = 611$. Hence, STAAD.Pro uses this limit state.

Comparison

The reported STAAD.Pro value is based on the terms `PNC`, `MNY`, and `MNZ` in the STAAD.Pro output.

Table 813: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Design compressive strength, $\phi_c P_n$ (kips)	1,940	1,938	negligible
Design bending strength, $\phi_b M_{ny}$ (ft·kips)	611	600.5 (7,206 in·kips)	1.7%
Design bending strength, $\phi_b M_{nz}$ (ft·kips)	1,200	1,200 (14,400 in·kips)	none
Utilization ratio	0.973	0.974	negligible

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Compression and Biaxial Bending.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 6.2, PAGE 6-6, AISC LRFD 3RD ED.
*  INTERACTION RATIO SHOULD BE ABOUT 0.973
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 14 0 0;
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X176
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 FIXED BUT FX MX MY MZ
2 FIXED BUT MY MZ
*
LOAD 1
JOINT LOAD
1 FX 1400
*
LOAD 2
JOINT LOAD
1 MZ 200
2 MZ 200
*
LOAD 3
JOINT LOAD
1 MY 70
2 MY 70
*
LOAD COMBINATION 4
1 1.0 2 1.0 3 1.0
*
PERFORM ANALYSIS
LOAD LIST 4
PRINT MEMBER FORCES
*
PARAMETER 1
```

```
CODE LRFD
FYLD 7200 ALL
TRACK 2 ALL
CHECK CODE ALL
*
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (LRFD 3RD EDITION)   v1.0
                       ********************************************
  |-----------------------------------------------------------------------|
  |                                                        Y        PROPERTIES |
  |*************                                           |        IN INCH UNIT |
  |            *  |=============================|   ===|===      ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |      |          AX=0.5180E+2 |
  |            *  | ST  W14X176                 |      |   --Z    AY=0.1262E+2 |
  |DESIGN CODE *  |                             |      |          AZ=0.2742E+2 |
  | LRFD 2001  *  ==============================   ===|===        PY=0.1630E+3 |
  |            *                                                  PZ=0.3200E+3 |
  |            *  |&lt;---LENGTH (FT)=   14.00 --->|                   RY=0.4022E
+1 |
  |*************                                                  RZ=0.6427E+1 |
  |                                                                          |
  |                200.0 (KIP-FEET)                                          |
  |PARAMETER        |L4                                       L4  CAPACITIES |
  |IN KIP  INCH     |                                     L4      IN KIP  INCH |
  |---------------  +   L4                                       -------------|
  | KL/R-Y=  41.77  |       L4                      L4           PNC=0.1938E+4|
  | KL/R-Z=  26.14  +                                            pnc=0.1400E+4|
  | UNL   = 168.00  |           L4              L4               PNT=0.2331E+4|
  | CB    =   1.00  +               L4      L4                   pnt=0.0000E+0|
  | PHIC  =   0.85  |                                            MNZ=0.1440E+5|
  | PHIB  =   0.90  +                                            mnz=0.2400E+4|
  | FYLD  =  50.00  |                   L0                       MNY=0.7206E+4|
  | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|    mny=0.8400E+3|
  | DFF =     0.00 -11.1                                         VN =0.3406E+3|
  | dff =     0.00              ABSOLUTE MZ ENVELOPE             vn =0.2857E+2|
  |                                (WITH LOAD NO.)                           |
  |                                                                          |
  |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
  |                     -------------------------                            |
  |                                                                          |
  |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
  |                                                                          |
  |      VALUE    1400.0        28.6       10.0        70.0       200.0      |
  |    LOCATION      0.0         0.0        0.0         0.0         0.0      |
  |     LOADING        4           4          4           4           4      |
  |                                                                          |
  |**************************************************************************|
  |*                                                                        *|
  |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
  |*                        --------------                                  *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
  |         FX              MY             MZ          LOCATION              |
  |        ======================================================            |
```

```
|          PASS        LRFD-H1-1A-C      0.974              4          |
|       1400.00 C         70.00       200.00              0.00          |
|*                                                                     *|
|***********************************************************************|
|                                                                       |
|-----------------------------------------------------------------------|
```

V. AISC LRFD - Select Compression and Biaxial Bending

To select the lightest ASTM A992 W14 section capable of carrying an axial compression plus biaxial bending.

Reference

AISC Load Factor Resistance Design, 3rd Edition, Example 6.4, page 6-9.

Related Links

- *D1.C.5 Limit States* (on page 1165)

Problem

F_y = 50 ksi

P_u = 400 kips

M_{uy} = 80 ft·kips, M_{uz} = 250 ft·kips

L_b = 14 ft

Reverse curvature bending with equal end moments for both axis.

Comparison

Table 814: Comparison of results

Result Type	Theory	STAAD.Pro	Difference
Selection section	W14x99	W14x99	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC LRFD - Select Compression and Biaxial Bending.STD` is typically installed with the program.

```
STAAD SPACE AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
*  EXAMPLE PROBLEM 6.4, PAGE 6-9, AISC LRFD 3RD ED.
*  THE LIGHTEST SUITABLE SECTION SHOULD BE A W14X99
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 14 0 0;
```

```
*
MEMBER INCIDENCES
1 1 2;
*
MEMBER PROPERTY AMERICAN
1 TABLE ST W14X90
*
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 4.176e+06
POISSON 0.3
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
*
SUPPORTS
1 FIXED
*
LOAD 1
JOINT LOAD
2 FX -400
2 MZ 250
2 MY 80
*
PERFORM ANALYSIS
*
PARAMETER 1
CODE LRFD
FYLD 7200 ALL
TRACK 2 ALL
SELECT ALL
*
FINISH
```

STAAD Output

```
      AXIAL COMPRESSION + BIAXIAL BENDING PER AISC LRFD 3RD ED -- PAGE NO.
3
                     STAAD.PRO MEMBER SELECTION - (LRFD 3RD EDITION)   v1.0
                     ***********************************************
   |--------------------------------------------------------------------------|
   |                                                   Y        PROPERTIES    |
   |*************                                      |        IN INCH UNIT  |
   |            *  |=============================|  ===|===     ------------  |
   |MEMBER   1  *  |  AISC SECTIONS              |     |        AX=0.2910E+2  |
   |            *  | ST  W14X99                  |     |   --Z  AY=0.6887E+1  |
   |DESIGN CODE *  |                             |     |        AZ=0.1518E+2  |
   | LRFD 2001  *  ===============================  ===|===     PY=0.8360E+2  |
   |            *                                               PZ=0.1730E+3  |
   |            *  |<---LENGTH (FT)=   14.00 --->|                 RY=0.3717E
+1 |
   |*************                                               RZ=0.6176E+1  |
   |                                                                          |
   |                250.0 (KIP-FEET)                                          |
   |PARAMETER        |                              L1  L1  L1  CAPACITIES    |
   |IN KIP  INCH     |                              L1  L1  L1  IN KIP  INCH  |
   |---------------  +                              L1  L1  L1  -------------|
```

```
| KL/R-Y=  45.20  |                                 L1  L1  L1  PNC=0.1065E+4|
| KL/R-Z=  27.20  +                                 L1  L1  L1  pnc=0.4000E+3|
| UNL   = 168.00  |L1 L1  L1  L1  L1  L1  L1  L1              PNT=0.1310E+4|
| CB    =   1.00  +L1 L1  L1  L1  L1  L1  L1  L1              pnt=0.0000E+0|
| PHIC  =   0.85  |L1 L1  L1  L1  L1  L1  L1  L1              MNZ=0.7717E+4|
| PHIB  =   0.90  +L1 L1  L1  L1  L1  L1  L1  L1              mnz=0.3000E+4|
| FYLD  =  50.00  |L1 L1  L1  L1  L1  L1  L1  L1              MNY=0.3698E+4|
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---|   mny=0.9600E+3|
| DFF =     0.00 250.0                                      VN =0.1859E+3|
| dff =     0.00              ABSOLUTE MZ ENVELOPE          vn =0.0000E+0|
|                                (WITH LOAD NO.)                           |
|                                                                          |
|                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
|                     -------------------------                            |
|                                                                          |
|               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
|                                                                          |
|      VALUE    400.0         0.0        0.0        80.0       250.0       |
|   LOCATION      0.0         0.0        0.0         0.0         0.0       |
|    LOADING        1           0          0           1           1       |
|                                                                          |
|**************************************************************************|
|*                                                                        *|
|*                        DESIGN SUMMARY  (KIP-FEET)                      *|
|*                        --------------                                  *|
|*                                                                        *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
|         FX               MY             MZ          LOCATION             |
|        ======================================================            |
|          PASS        LRFD-H1-1A-C     0.951             1                |
|        400.00 C        -80.00      -250.00            0.00               |
|*                                                                        *|
|**************************************************************************|
|                                                                          |
|--------------------------------------------------------------------------|
```

V. AISC N690 1994 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the AISC N690 1994 code.

Details

The horizontal load is 7.5 kips and the vertical load is 10 kips.

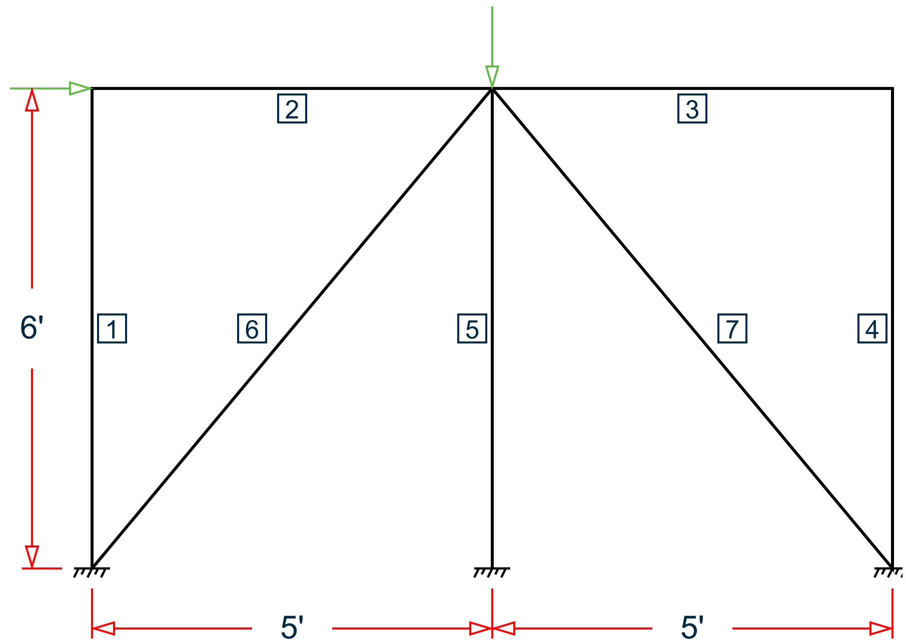


Figure 504: Space frame subjected to static loads

The forces due to loading in member 6 is 2.861 kips (tension) and in member 7 is 7.328 kips (compression). Members 6 and 7 are "truss" members (axial-only).

Both member 6 and 7 are L 80x60x10, austenitic stainless steel ($F_y = 36\ ksi$, $F_u = 58\ ksi$).

Assume a net section factor of 0.9 and an effective length factor of 0.85.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

For members in compression (member 7), the equivalent slenderness ratio per Eqn. 4-4 of AISC ASD p5-311 must also be determined:

$$(KL/r)_{equiv} = \pi\times\sqrt{(E/F_e)}$$

where

F_e = the elastic buckling strength for flexural torsional buckling taken to be 60.348 ksi. This is defined as the lowest root of the Eqn in AISC ASD (C4-2) page 5-317

Thus, $(KL/r)_{equiv} = \pi\times\sqrt{(29{,}000/60.348)} = 68.86$

<u>Axial Tension: Member 6</u>

Actual tensile stress (gross) = $P/A = 2.862 / 8.41 = 0.340$ ksi

Allowable tensile stress (Tension capacity) = $0.6 \times F_y = 0.6 \times 36 = 21.6$ ksi

Critical ratio = 0.340/21.6 = 0.016

<u>Axial Compression: Member 7</u>

Actual compressive stress = $P/A = 7.328 / 8.41 = 0.871$ ksi

As this member is made from austenitic stainless steel the allowable compression stress should be determined according to clause Q.1.5.9. As $KL/r < 120$, then the value of allowable axial stress, F_a, is determined from Eqn. Q1.5-11:

$$Fa = \frac{Fy}{2.15} - \left(\frac{\frac{Fy}{2.15} - 6}{120} \right) \times Kl \Big/ r = 16.74 - \left(\frac{16.71 - 6}{120} \right) 68.86 = 10.57 \text{ ksi}$$

Check cross-section meets requirements of Q1.9, which sets a limit for any unstiffened element clause Q1.9.1.2 of $75/\sqrt{(F_y)} = 12.67$

Short leg = $B/t = 6/0.625 = 9.5 < 12.67$, OK
Long leg = $D/t = 8/0.625 = 12.8 > 12.67$, leg is slender

Since $75/\sqrt{(F_y)} = 12.67 < D/t = 12.8 < 155/\sqrt{(F_y)} = 25.83$, Q_s is determined by Eqn. QC2-1:

$$Q_s = 1.340 - 0.00447 \times (b/t) \times \sqrt{(F_y)} = 0.9967$$

Allowable compressive stress, $F_a = 0.9967 \times 10.57 = 10.54$ ksi

Critical ratio = $0.871 / 10.54 = 0.083$

Results

Table 815: Verification comparison for AISC_N690_1994_Angle

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	negligible	
	Actual tensile stress (ksi)	340	340	none	
	Allowable tensile stress (ksi)	21.6	21.6	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Critical slenderness ratio	68.86	68.87	negligible	
	Actual tensile stress (ksi)	0.871	0.87	negligible	

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
	Allowable compressive stress (ksi)	10.54	10.55	negligible	
	Critical ratio	0.083	0.083	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1994 Angle.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
*************************************************************
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
PRINT MEMBER PROPERTIES all
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
***********************************************************
PARAMETER 1
CODE AISC N690 1994
* Material strength
FU 58000  MEMB 6 7
FYLD 36000  MEMB 6 7
```

```
* Net area reduced by 10%
NSF 0.9 MEMB 6 7
* Austenitic stainless steel
STYPE 1  MEMB 6 7
* Effective length factors 0.85 in both axes
LX 1 MEMB 6 7
KY 0.85  MEMB 6 7
KZ 0.85  MEMB 6 7
*
TRACK 2 ALL
CHECK CODE MEMB 6 7
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                        Y        PROPERTIES |
   |*************                                           |        IN INCH UNIT |
   |            *  |=============================|   ==| |==        ------------ |
   |MEMBER   6  *  |  AISC SECTIONS              |     | |          AX =   8.41 |
   |            *  | ST  L806010                 |     | |   --Z   AY =   3.33 |
   |DESIGN CODE *  |                             |     | |          AZ =   2.50 |
   |ANSI N690-94*  ==============================   ==| |==        SY =  11.99 |
   |            *                                                   SZ =   5.16 |
   |            *  |&lt;---LENGTH (FT)=    7.81 --->|                     RY =
2.81  |
   |*************                                                   RZ =   1.29 |
   |                                                                            |
   |                  0.0 (KIP-FEET)                                            |
   |PARAMETER        |L0                                    L0  STRESSES        |
   |IN KIP  INCH     |L0                                    L0  IN KIP  INCH    |
   |---------------  +L0                                    L0  -------------|
   | KL/R-Y=  28.34  |L0                                    L0  FA   =  21.60 |
   | KL/R-Z=  61.76  +L0                                    L0  fa   =   0.34 |
   | UNL    =  93.72  |L0                                   L0  FCZ =   0.00 |
   | CB     =   0.00  +L0                                   L0  FTZ =   0.00 |
   | CMY    =   0.00  |L0                                   L0  FCY =   0.00 |
   | CMZ    =   0.00  +L0                                   L0  FTY =   0.00 |
   | FYLD   =  36.00  |L0                                   L0  fbz =   0.00 |
   | NSF    =   0.90  +---+---+---+---+---+---+---+---+---|  fby =   0.00 |
   | DFF    =   0.00   0.0                                       Fey = 165.72 |
   | dff=      0.00             ABSOLUTE MZ ENVELOPE             Fez =  34.91 |
   | (KL/R)max =  68.87            (WITH LOAD NO.)               FV   =   0.00 |
   |                                                             fv   =   0.00 |
   |                                                                            |
   |                       MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                       -------------------------                            |
   |                                                                            |
   |                 AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                            |
   |      VALUE       -2.9         0.0        0.0         0.0         0.0       |
   |    LOCATION       0.0         0.0        0.0         0.0         0.0       |
   |     LOADING         1           0          0           0           0       |
   |                                                                            |
   |**************************************************************************|
```

```
   |*                                                                          *|
   |*                         DESIGN SUMMARY  (KIP-FEET)                       *|
   |*                         --------------                                   *|
   |*                                                                          *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/              *|
   |         FX               MY            MZ          LOCATION               |
   |        ======================================================            |
   |          PASS        ANSI Q1.6-2      0.016            1                  |
   |          2.86 T          0.00         0.00            0.00                |
   |*                                                                          *|
   |                                                                           |
   |---------------------------------------------------------------------------|
       STAAD SPACE                                                -- PAGE NO.
5
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
   |---------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES    |
   |*************                                       |        IN INCH UNIT  |
   |            *  |=============================|   ==| |==     ------------  |
   |MEMBER   7  *  |  AISC SECTIONS              |     | |       AX =   8.41   |
   |            *  | ST  L806010                 |     | |  --Z  AY =   3.33   |
   |DESIGN CODE *  |                             |     | |       AZ =   2.50   |
   |ANSI N690-94*  ===============================   ==| |==     SY =  11.99   |
   |            *                                                SZ =   5.16   |
   |            *  |<---LENGTH (FT)=    7.81 --->|                  RY =
2.81  |
   |*************                                                RZ =   1.29   |
   |                                                                           |
   |                  0.0 (KIP-FEET)                                           |
   |PARAMETER        |L0                                    L0  STRESSES       |
   |IN KIP  INCH     |L0                                    L0  IN KIP  INCH   |
   |---------------  +L0                                    L0  -------------|
   | KL/R-Y=  28.34  |L0                                    L0  FA   =  10.55 |
   | KL/R-Z=  61.76  +L0                                    L0  fa   =   0.87 |
   | UNL   =  93.72  |L0                                    L0  FCZ =   0.00 |
   | CB    =   0.00  +L0                                    L0  FTZ =   0.00 |
   | CMY   =   0.00  |L0                                    L0  FCY =   0.00 |
   | CMZ   =   0.00  +L0                                    L0  FTY =   0.00 |
   | FYLD  =  36.00  |L0                                    L0  fbz =   0.00 |
   | NSF   =   0.90  +---+---+---+---+---+---+---+---+---+---|  fby =   0.00 |
   | DFF   =   0.00   0.0                                       Fey = 165.72 |
   | dff=      0.00              ABSOLUTE MZ ENVELOPE           Fez =  34.91 |
   | (KL/R)max =  68.87             (WITH LOAD NO.)             FV   =   0.00 |
   |                                                            fv   =   0.00 |
   |                                                                           |
   |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                      -------------------------                            |
   |                                                                           |
   |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
   |                                                                           |
   |      VALUE       7.3          0.0        0.0         0.0         0.0      |
   |    LOCATION      0.0          0.0        0.0         0.0         0.0      |
   |     LOADING        1            0          0           0           0      |
   |                                                                           |
   |***************************************************************************|
   |*                                                                          *|
   |*                         DESIGN SUMMARY  (KIP-FEET)                       *|
```

```
|*                   --------------                                     *|
|*                                                                      *|
|*      RESULT/     CRITICAL COND/     RATIO/         LOADING/          *|
|        FX               MY            MZ           LOCATION           |
|       ======================================================          |
|         PASS        ANSI Q1.6-2      0.083             1               |
|         7.33 C          0.00         0.00            0.00              |
|*                                                                      *|
|                                                                        |
|-----------------------------------------------------------------------|
```

Related Links

- *D1.K.1. ANSI/AISC N690-1994 Code* (on page 1296)

V. AISC N690 1994 Channel

Verify the design of channel section used for a beam member using the AISC N690 1994 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,000}{(d / A_f) F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0) / (2 \times 404) = 15.41\ \text{ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

<u>Shear</u>

Actual shear:

$$F_y = 51.375 \text{ kips}$$

STAAD.Pro calculates the shear are based on the general shear stress equation, $\tau = V \times Q/(I \times t)$ from Timoshenko. Thus the shear area, $A_y = I \times t/Q = 8.47\ in^2$

$$f_v = 51.375 / 8.47 = 6.07 \text{ ksi}$$

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$$t = t_w = 0.72 \text{ in}$$

If

$$Cv = 17.2$$

$$F_v = \min \begin{vmatrix} F_y/2.89 \times C_v = 25.6\ ksi \\ 0.4 \times F_y = 14.4\ ksi \end{vmatrix}$$

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v/F_v = 6.07\ /\ 14.4 = 0.422$

Results

Table 816: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.41	15.41	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	6.07	6.06	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1994 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1994
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
```

```
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                    Y        PROPERTIES   |
  |*************                                       |        IN INCH UNIT |
  |            *  |=============================|    |===       ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |    |          AX =  14.70  |
  |            *  | ST  C15X50                  |    |     --Z  AY =   8.47  |
  |DESIGN CODE *  |                             |    |          AZ =   2.67  |
  |ANSI N690-94*  ===============================    |===       SY =   3.83  |
  |            *                                                SZ =  53.87  |
  |            *  |&lt;---LENGTH (FT)=    6.67 --->|                 RY =
0.87  |
  |*************                                                RZ =   5.24  |
  |                                                                          |
  |                 69.2 (KIP-FEET)                                          |
  |PARAMETER        |L1                                         STRESSES     |
  |IN KIP  INCH     |                                           IN KIP  INCH |
  |---------------  +                                           -------------|
  | KL/R-Y=  78.61  |   L1                                      FA  =   9.71 |
  | KL/R-Z=  12.97  +                   L1  L1                  fa  =   0.00 |
  | UNL   =  80.00  |               L1          L1              FCZ =  21.60 |
  | CB    =   1.00  +       L1                      L1          FTZ =  21.60 |
  | CMY   =   1.00  |                                   L1      FCY =  21.60 |
  | CMZ   =   1.00  +                                           FTY =  21.60 |
  | FYLD  =  36.00  |           L1                          L0  fbz =  15.41 |
  | NSF   =   0.90  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
  | DFF   =   0.00  -3.8                                        Fey =  21.54 |
  | dff=      0.00              ABSOLUTE MZ ENVELOPE            Fez = 791.24 |
  | (KL/R)max =  78.61             (WITH LOAD NO.)              FV  =  14.40 |
  |                                                             fv  =   6.06 |
  |                                                                          |
  |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)               |
  |                      -------------------------                           |
  |                                                                          |
  |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z     |
  |                                                                          |
  |      VALUE       0.0         51.4        0.0         0.0        69.2     |
  |   LOCATION       0.0          0.0        0.0         0.0         0.0     |
  |    LOADING         0            1          0           0           1     |
  |                                                                          |
  |**************************************************************************|
  |*                                                                        *|
  |*                          DESIGN SUMMARY  (KIP-FEET)                    *|
  |*                          --------------                                *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
  |         FX              MY             MZ          LOCATION              |
  |        ======================================================            |
  |          PASS        ANSI Q1.6-2      0.713             1                |
  |          0.00 T          0.00       69.17            0.00                |
```

```
|*                                                                    *|
|                                                                      |
```

Related Links

- *D1.K.1. ANSI/AISC N690-1994 Code* (on page 1296)

V. AISC N690 1994 Pipe Section With SLC

Verify the design of a pipe section used for a beam member per the AISC N690 1994 code.

References

Gere, J. M. and S. P. Timoshenko. 1984. *Mechanics of Materials, 2nd Edition*. Boston, MA: PWS Publishers. p.238.

Related Links

- *D1.K.1. ANSI/AISC N690-1994 Code* (on page 1296)

Details

The 10 ft long span is a propped cantilever which is free to move axially at the fixed end. The beam is subject to the following four different load cases:

1. a 10 kip axial tension force,
2. a 10 kip axial compression force,
3. a uniformly distributed load of 0.3 kip/ft in the Global Y directions, the member selfweight (acts in Global Y), and the tension load from case 1
4. a uniformly distributed load of 0.3 kip/ft in the Global Y directions, the member selfweight (acts in Global Y), and the compression load from case 2

The member is Pipe 5-Sched. 40 (standard) pipe section with Grade A36 steel. The self-weight is 14.6 lbs/ft.

Validation

<u>Slenderness Ratio</u>

The slenderness ratio is the same about either local z or y axes: $KL/r = 120\ in\ /\ 1.888\ in = 63.56 < 300$, OK

<u>Axial Tension</u>

From section Q1.5.1.1, the allowable tensile stress:

$$F_t = \min \left| \begin{array}{l} 0.6F_y = 0.6 \times 36 = 21.6 \text{ ksi} \\ 0.5F_u = 0.5 \times 58 = 29 \text{ ksi} \end{array} \right.$$

The actual tensile stress from load case 1:

$$f_t = \frac{P}{A} = \frac{10}{4.01} = 2.49 \text{ ksi} < F_t$$

Stress ratio = $2.49\ /\ 21.6 = 0.115$, hence OK.

<u>Axial Compression</u>

Allowable compressive stress per Cl Q1.9.2.3:

$$\frac{\mathrm{OD}}{t} = \frac{5.56}{0.24} = 23.17 < \frac{3{,}300}{F_y} = \frac{3{,}300}{36} = 91.67$$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = 126.1$$

Since $Kl/r < C_c$, per eq. Q1.5-1, the allowable axial compressive stress:

$$F_a = \frac{1 - \frac{\left(kl/r\right)^2}{2 \times C_c^2}}{\frac{5}{3} + 3 \times \frac{kl/r}{8 \times C_c} - \frac{\left(kl/r\right)^3}{8 \times C_c^3}} F_y = 17.08 \text{ ksi}$$

The actual compressive stress from load case 2:

$$f_a = \frac{P}{A} = \frac{10}{4.01} = 2.49 \text{ ksi} < F_a$$

Stress ratio = $2.49 / 17.08 = 0.146$, hence OK.

Bending

Section is compact per Cl Q1.5.1.4.1: $\frac{OD}{t} = 23.17 < \frac{3{,}300}{F_y} = 91.67$

Allowable bending stress about both axes: $F_b = 0.66 \times F_y = 23.76$ ksi

Moment due to load case 3 or 4 on a propped cantilever (ref. AISC Steel Construction Manual):

$$M_z = \frac{w \times l^2}{8} = \frac{0.314 \times 10^2}{8} = 3.93 \text{ ft·k} = 47.2 \text{ in·k}$$

Actual bending stress about the major axis: $f_{bz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{47.2}{14.3}\left(\frac{5.56}{2}\right) = 9.17$ ksi

(actual bending about the minor axis: $f_{by} = 0$)

Stress ratio: $9.17 / 23.76 = 0.386$, hence OK.

Combined Bending and Axial Tension

Interaction per Cl Q1.6.2 in load case 3 (tension + bending):

$$\frac{f_a}{\texttt{SFT} \times 0.6F_y} + \frac{f_{bz}}{\texttt{SMZ} \times F_{bz}} = \frac{2.49}{1.3 \times 21.6} + \frac{9.17}{1.4 \times 23.76} = 0.364 \qquad \text{(Eq. Q1.6-1b)}$$

Combined Bending and Axial Compression

Interaction per Cl Q1.6 in load case 4 (compression + bending):

$$\frac{f_a}{F_a} = 0.145 < 0.15 \qquad \text{(Eq. Q1.6-2)}$$

$$\frac{f_a}{\texttt{SFC} \times F_a} + \frac{f_{by}}{\texttt{SMY} \times F_{by}} + \frac{f_{bz}}{\texttt{SMZ} \times F_{bz}} = \frac{2.49}{1.3 \times 17.08} + \frac{0}{1.4 \times 23.76} + \frac{9.17}{1.4 \times 23.76} = 0.388$$

Shear

Actual shear (at supports):

$$f_{v,y} = \frac{5w \times l}{8} = \frac{5 \times 0.315 \times 10}{8} = 1.97 \text{ kip}$$

Shear area (calculated as per Gere & Timoshenko reference):

$$A_z = \frac{\frac{\pi}{4}\left(r_2{}^4 - r_1{}^4\right) \times 2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2{}^3 - r_2{}^3\right)}$$

where

r_1	=	the inner radius of the pipe = $5.56\ /\ 2 - 0.24 = 2.54\ in$
r_2	=	the outer radius of the pipe = $5.56\ /\ 2 = 2.78\ in$

Thus, the shear area along both axes = $2.008\ in^2$.

$$F_{vy} = \frac{F_{v,y}}{A_y} = \frac{1.97}{2.008} = 0.98 \text{ ksi}$$

(shear stress in major axis = 0)

The allowable shear stress:

$$F_v = 0.4F_y = 14.4 \text{ ksi}$$

Stress ratio in Y (minor axis): $0.98/14.4 = 0.068$, hence OK.

Results

Table 817: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	63.56	63.55	negligible	
Actual axial tensile stress (ksi)	2.49	2.49	none	
Allowable axial tensile stress (ksi)	21.6	21.6	none	
Tensile stress ratio (LC1)	0.115	0.115	none	
Actual axial compressive stress (ksi)	2.49	2.49	none	
Allowable axial compressive stress (ksi)	17.08	17.08	none	
Compressive stress ratio (LC2)	0.146	0.146	none	
Allowable bending stress (ksi)	23.76	23.8	negligible	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Actual bending stress (ksi)	9.17	9.15	negligible	
Combined axial tension and bending stress ratio (LC3)	0.364	0.363	negligible	
Combined axial compression and bending stress ratio (LC4)	0.388	0.386	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	0.98	0.97	negligible	
Shear stress ratio, Y	0.068	0.067	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1994 Pipe Section With SLC.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS50
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
```

```
2 FIXED BUT FX
LOAD 1 TENSION ONLY
JOINT LOAD
2 FX 10
LOAD 2 COMPRESSION ONLY
JOINT LOAD
2 FX -10
LOAD 3 TENSION+BENDING
SELFWEIGHT Y -1
MEMBER LOAD
1 UNI GY -0.3
JOINT LOAD
2 FX 10
LOAD 4 COMPRESSION+BENDING
SELFWEIGHT Y -1
MEMBER LOAD
1 UNI GY -0.3
JOINT LOAD
2 FX -10
PERFORM ANALYSIS PRINT LOAD DATA
PRINT MEMBER PROPERTIES ALL
UNIT INCHES KIP
*
LOAD LIST 1
PARAMETER 1
CODE AISC N690 1994
CB 1 ALL
CMY 1 ALL
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
*
LOAD LIST 2
PARAMETER 2
CODE AISC N690 1994
CB 1 ALL
CMY 1 ALL
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
```

```
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
*
LOAD LIST 3
PARAMETER 3
CODE AISC N690 1994
CB 1 ALL
CMY 1 ALL
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
SMZ 1.4 ALL
SMY 1.4 ALL
SFT 1.3 ALL
SFC 1.3 ALL
TMAIN 300 ALL
TRACK 2 ALL
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
*
LOAD LIST 4
PARAMETER 4
CODE AISC N690 1994
CB 1 ALL
CMY 1 ALL
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
SMZ 1.4 ALL
SMY 1.4 ALL
SFT 1.3 ALL
SFC 1.3 ALL
TMAIN 300 ALL
TRACK 2 ALL
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
*
LOAD LIST ALL
PARAMETER 5
CODE AISC N690 1994
CB 1 ALL
CMY 1 ALL
```

```
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
SMZ 1.4 ALL
SMY 1.4 ALL
SFT 1.3 ALL
SFC 1.3 ALL
TMAIN 300 ALL
TRACK 2 ALL
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                   Y         PROPERTIES   |
   |*************                                      |         IN INCH UNIT |
   |            *  |=============================|   ==| |==     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |     | |       AX =   4.01  |
   |            *  | ST  PIPS50                  |     | |  --Z  AY =   2.02  |
   |DESIGN CODE *  |                             |     | |       AZ =   2.02  |
   |ANSI N690-94*  ===============================   ==| |==     SY =   5.14  |
   |            *                                                SZ =   5.14  |
   |            *  |<---LENGTH (FT)=   10.00 --->|                  RY =
1.89  |
   |*************                                                RZ =   1.89  |
   |                                                                          |
   |                 0.0 (KIP-FEET)                                           |
   |PARAMETER       |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES      |
   |IN KIP  INCH    |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH  |
   |---------------  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
   | KL/R-Y=  63.55  |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA   =  21.60 |
   | KL/R-Z=  63.55  +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa   =   2.49 |
   | UNL    =  30.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  23.76  |
   | CB     =   1.00 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  23.76  |
   | CMY    =   1.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  23.76  |
   | CMZ    =   1.00 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  23.76  |
   | FYLD   =  36.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00  |
   | NSF    =   1.00 +---+---+---+---+---+---+---+---+---+---|  fby =   0.00  |
   | DFF    =   0.00  0.0                                       Fey =  36.98  |
   | dff=       0.00             ABSOLUTE MZ ENVELOPE           Fez =  36.98  |
   | (KL/R)max =  63.55             (WITH LOAD NO.)             FV   =  14.40 |
   |                                                            fv   =   0.00 |
   |                                                                          |
   |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
   |                     -------------------------                            |
   |                                                                          |
```

```
  |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
  |                                                                          |
  |      VALUE     -10.0         0.0        0.0         0.0         0.0      |
  |   LOCATION       0.0         0.0        0.0         0.0         0.0      |
  |    LOADING         1           0          0           0           0      |
  |                                                                          |
  |**************************************************************************|
  |*                                                                        *|
  |*                         DESIGN SUMMARY  (KIP-FEET)                     *|
  |*                         --------------                                 *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
  |         FX             MY             MZ          LOCATION              |
  |       ======================================================            |
  |         PASS        ANSI Q1.6-3      0.115              1               |
  |        10.00 T          0.00         0.00             0.00              |
  |*                                                                        *|
  |                                                                          |
  |--------------------------------------------------------------------------|
      STAAD SPACE                                                 -- PAGE NO.
7
    70. *
    71. LOAD LIST 2
    72. PARAMETER 2
    73. CODE AISC N690 1994
    74. CB 1 ALL
    75. CMY 1 ALL
    76. CMZ 1 ALL
    77. FYLD 36 ALL
    78. FU 58 ALL
    79. KY 1 ALL
    80. KZ 1 ALL
    81. MAIN 200 ALL
    82. NSF 1 ALL
    83. RATIO 1 ALL
    84. TMAIN 300 ALL
    85. TRACK 2 ALL
    86. UNB 30 ALL
    87. UNT 30 ALL
    88. CT 0.75 ALL
    89. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                 -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                        Y        PROPERTIES |
  |*************                                           |        IN INCH UNIT |
  |            *  |=============================|   ==| |==      ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |     | |        AX =   4.01  |
  |            *  | ST  PIPS50                  |     | |   --Z  AY =   2.02  |
  |DESIGN CODE *  |                             |     | |        AZ =   2.02  |
  |ANSI N690-94*  ===============================   ==| |==      SY =   5.14  |
  |            *                                                 SZ =   5.14  |
  |            *  |<---LENGTH (FT)=   10.00 --->|                   RY =
1.89  |
  |*************                                                 RZ =   1.89  |
```

```
 |                                                                          |
 |                 0.0 (KIP-FEET)                                           |
 |PARAMETER       |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  STRESSES     |
 |IN KIP  INCH    |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  IN KIP  INCH |
 |--------------- +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  -------------|
 | KL/R-Y=  63.55 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FA  =  17.08 |
 | KL/R-Z=  63.55 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fa  =   2.49 |
 | UNL   =  30.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCZ =  23.76 |
 | CB    =   1.00 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTZ =  23.76 |
 | CMY   =   1.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FCY =  23.76 |
 | CMZ   =   1.00 +L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  FTY =  23.76 |
 | FYLD  =  36.00 |L0 L0  L0  L0  L0  L0  L0  L0  L0  L0  L0  fbz =   0.00 |
 | NSF   =   1.00 +---+---+---+---+---+---+---+---+---+---|   fby =   0.00 |
 | DFF   =   0.00   0.0                                       Fey =  36.98 |
 | dff=      0.00             ABSOLUTE MZ ENVELOPE            Fez =  36.98 |
 | (KL/R)max =  63.55            (WITH LOAD NO.)              FV  =  14.40 |
 |                                                            fv  =   0.00 |
 |                                                                          |
 |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
 |                     -------------------------                            |
 |                                                                          |
 |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
 |                                                                          |
 |      VALUE      10.0         0.0        0.0         0.0         0.0      |
 |    LOCATION      0.0         0.0        0.0         0.0         0.0      |
 |     LOADING        2           0          0           0           0      |
 |                                                                          |
 |**************************************************************************|
 |*                                                                        *|
 |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
 |*                        --------------                                  *|
 |*                                                                        *|
 |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
 |         FX              MY             MZ          LOCATION              |
 |        ======================================================            |
 |          PASS       ANSI Q1.6-2      0.146             2                 |
 |         10.00 C         0.00         0.00             0.00               |
 |*                                                                        *|
 |                                                                          |
 |--------------------------------------------------------------------------|
      STAAD SPACE                                               -- PAGE NO.
9
    90. *
    91. LOAD LIST 3
    92. PARAMETER 3
    93. CODE AISC N690 1994
    94. CB 1 ALL
    95. CMY 1 ALL
    96. CMZ 1 ALL
    97. FYLD 36 ALL
    98. FU 58 ALL
    99. KY 1 ALL
   100. KZ 1 ALL
   101. MAIN 200 ALL
   102. NSF 1 ALL
   103. RATIO 1 ALL
   104. SMZ 1.4 ALL
   105. SMY 1.4 ALL
```

```
   106. SFT 1.3 ALL
   107. SFC 1.3 ALL
   108. TMAIN 300 ALL
   109. TRACK 2 ALL
   110. UNB 30 ALL
   111. UNT 30 ALL
   112. CT 0.75 ALL
   113. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
10
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
   |--------------------------------------------------------------------------|
   |                                                    Y         PROPERTIES  |
   |*************                                       |         IN INCH UNIT |
   |            *  |=============================|   ==| |==     ------------ |
   |MEMBER   1  *  |  AISC SECTIONS              |     | |        AX =   4.01  |
   |            *  | ST  PIPS50                  |     | |  --Z   AY =   2.02  |
   |DESIGN CODE *  |                             |     | |        AZ =   2.02  |
   |ANSI N690-94*  ==============================   ==| |==      SY =   5.14  |
   |            *                                                 SZ =   5.14  |
   |            *  |<---LENGTH (FT)=   10.00 --->|                    RY =
1.89  |
   |*************                                                 RZ =   1.89  |
   |                                                                           |
   |                 3.9 (KIP-FEET)                                            |
   |PARAMETER        |                                       L3  STRESSES      |
   |IN KIP  INCH     |                                           IN KIP  INCH  |
   |---------------  +                                           ------------- |
   | KL/R-Y=  63.55  |                                   L3      FA  =  21.60  |
   | KL/R-Z=  63.55  +               L3                          fa  =   2.49  |
   | UNL   =  30.00  |           L3       L3                     FCZ =  23.76  |
   | CB    =   1.00  +       L3              L3                  FTZ =  23.76  |
   | CMY   =   1.00  |   L3                          L3          FCY =  23.76  |
   | CMZ   =   1.00  +                                           FTY =  23.76  |
   | FYLD  =  36.00  |L0                       L0                fbz =   9.15  |
   | NSF   =   1.00  +---+---+---+---+---+---+---+---+---+---|   fby =   0.00  |
   | DFF   =   0.00  -0.2                                        Fey =  36.98  |
   | dff=      0.00             ABSOLUTE MZ ENVELOPE             Fez =  36.98  |
   | (KL/R)max =  63.55            (WITH LOAD NO.)               FV  =  14.40  |
   |                                                             fv  =   0.97  |
   |                                                                           |
   |                         MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)             |
   |                         -------------------------                         |
   |                                                                           |
   |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z       |
   |                                                                           |
   |      VALUE     -10.0         2.0        0.0         0.0         3.9       |
   |   LOCATION       0.0        10.0        0.0         0.0        10.0       |
   |    LOADING         3           3          0           0           3       |
   |                                                                           |
   |***************************************************************************|
   |*                                                                         *|
   |*                            DESIGN SUMMARY  (KIP-FEET)                   *|
   |*                            --------------                               *|
   |*                                                                         *|
   |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/             *|
```

```
 |         FX               MY              MZ           LOCATION          |
 |        =====================================================            |
 |          PASS        ANSI Q1.6-3      0.364              3              |
 |         10.00 T          0.00         3.92             10.00            |
 |*                                                                       *|
 |                                                                         |
 |                                                                         |
 |-------------------------------------------------------------------------|
      STAAD SPACE                                              -- PAGE NO.
11
   114. *
   115. LOAD LIST 4
   116. PARAMETER 4
   117. CODE AISC N690 1994
   118. CB 1 ALL
   119. CMY 1 ALL
   120. CMZ 1 ALL
   121. FYLD 36 ALL
   122. FU 58 ALL
   123. KY 1 ALL
   124. KZ 1 ALL
   125. MAIN 200 ALL
   126. NSF 1 ALL
   127. RATIO 1 ALL
   128. SMZ 1.4 ALL
   129. SMY 1.4 ALL
   130. SFT 1.3 ALL
   131. SFC 1.3 ALL
   132. TMAIN 300 ALL
   133. TRACK 2 ALL
   134. UNB 30 ALL
   135. UNT 30 ALL
   136. CT 0.75 ALL
   137. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
12
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
 |-------------------------------------------------------------------------|
 |                                                        Y        PROPERTIES |
 |*************                                           |        IN INCH UNIT |
 |            *  |=============================|   ==| |==       ------------ |
 |MEMBER   1  *  |  AISC SECTIONS              |     | |         AX =   4.01 |
 |            *  | ST  PIPS50                  |     | |  --Z    AY =   2.02 |
 |DESIGN CODE *  |                             |     | |         AZ =   2.02 |
 |ANSI N690-94*  ==============================   ==| |==       SY =   5.14 |
 |            *                                                  SZ =   5.14 |
 |            *  |<---LENGTH (FT)=   10.00 --->|                    RY =
1.89  |
 |*************                                                  RZ =   1.89 |
 |                                                                          |
 |                  3.9 (KIP-FEET)                                          |
 |PARAMETER         |                                        L4  STRESSES    |
 |IN KIP  INCH      |                                            IN KIP  INCH |
 |---------------  +                                             -------------|
 | KL/R-Y=  63.55  |                                   L4       FA   =  17.08 |
 | KL/R-Z=  63.55  +               L4                            fa   =   2.49 |
 | UNL    =  30.00  |           L4      L4                        FCZ =  23.76 |
```

```
 | CB    =   1.00  +       L4              L4                    FTZ =  23.76 |
 | CMY   =   1.00  |   L4                          L4            FCY =  23.76 |
 | CMZ   =   1.00  +                                             FTY =  23.76 |
 | FYLD  =  36.00  |L0                        L0                 fbz =   9.15 |
 | NSF   =   1.00  +---+---+---+---+---+---+---+---+---|         fby =   0.00 |
 | DFF   =   0.00  -0.2                                          Fey =  36.98 |
 | dff=      0.00              ABSOLUTE MZ ENVELOPE              Fez =  36.98 |
 | (KL/R)max =  63.55             (WITH LOAD NO.)                FV  =  14.40 |
 |                                                               fv  =   0.97 |
 |                                                                            |
 |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                  |
 |                     -------------------------                              |
 |                                                                            |
 |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z        |
 |                                                                            |
 |      VALUE      10.0         2.0        0.0         0.0         3.9        |
 |    LOCATION      0.0        10.0        0.0         0.0        10.0        |
 |     LOADING        4           4          0           0           4        |
 |                                                                            |
 |****************************************************************************|
 |*                                                                          *|
 |*                         DESIGN SUMMARY  (KIP-FEET)                       *|
 |*                         --------------                                   *|
 |*                                                                          *|
 |*       RESULT/     CRITICAL COND/     RATIO/         LOADING/             *|
 |         FX            MY              MZ           LOCATION              |
 |        ======================================================            |
 |          PASS        ANSI Q1.6-2      0.387              4               |
 |         10.00 C         0.00          3.92            10.00              |
 |*                                                                          *|
 |                                                                            |
 |----------------------------------------------------------------------------|
    STAAD SPACE                                                   -- PAGE NO.
13
   138. *
   139. LOAD LIST ALL
   140. PARAMETER 5
   141. CODE AISC N690 1994
   142. CB 1 ALL
   143. CMY 1 ALL
   144. CMZ 1 ALL
   145. FYLD 36 ALL
   146. FU 58 ALL
   147. KY 1 ALL
   148. KZ 1 ALL
   149. MAIN 200 ALL
   150. NSF 1 ALL
   151. RATIO 1 ALL
   152. SMZ 1.4 ALL
   153. SMY 1.4 ALL
   154. SFT 1.3 ALL
   155. SFC 1.3 ALL
   156. TMAIN 300 ALL
   157. TRACK 2 ALL
   158. UNB 30 ALL
   159. UNT 30 ALL
   160. CT 0.75 ALL
   161. CHECK CODE ALL
```

```
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
14
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                     Y        PROPERTIES  |
  |*************                                        |        IN INCH UNIT |
  |            *  |=============================|   ==| |==     ------------ |
  |MEMBER   1  *  |  AISC SECTIONS              |     | |      AX =   4.01  |
  |            *  | ST  PIPS50                  |     | |  --Z AY =   2.02  |
  |DESIGN CODE *  |                             |     | |      AZ =   2.02  |
  |ANSI N690-94*  ===============================   ==| |==    SY =   5.14  |
  |            *                                               SZ =   5.14  |
  |            *  |<---LENGTH (FT)=   10.00 --->|                 RY =
1.89  |
  |*************                                               RZ =   1.89  |
  |                                                                          |
  |                  3.9 (KIP-FEET)                                          |
  |PARAMETER        |                                      L3  STRESSES     |
  |IN KIP  INCH     |                                          IN KIP  INCH |
  |---------------  +                                          -------------|
  | KL/R-Y=  63.55  |                                  L3      FA  =  17.08 |
  | KL/R-Z=  63.55  +              L3                          fa  =   2.49 |
  | UNL   =  30.00  |          L3      L3                      FCZ =  23.76 |
  | CB    =   1.00  +      L3              L3                  FTZ =  23.76 |
  | CMY   =   1.00  |   L3                         L3          FCY =  23.76 |
  | CMZ   =   1.00  +                                          FTY =  23.76 |
  | FYLD  =  36.00  |L0                        L0              fbz =   9.15 |
  | NSF   =   1.00  +---+---+---+---+---+---+---+---+---|  fby =   0.00 |
  | DFF   =   0.00  -0.2                                       Fey =  36.98 |
  | dff=      0.00              ABSOLUTE MZ ENVELOPE           Fez =  36.98 |
  | (KL/R)max =  63.55             (WITH LOAD NO.)             FV  =  14.40 |
  |                                                            fv  =   0.97 |
  |                                                                          |
  |                     MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                |
  |                     -------------------------                            |
  |                                                                          |
  |               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z      |
  |                                                                          |
  |      VALUE    -10.0         2.0        0.0         0.0         3.9       |
  |   LOCATION      0.0        10.0        0.0         0.0        10.0       |
  |    LOADING        1           3          0           0           3       |
  |                                                                          |
  |**************************************************************************|
  |*                                                                        *|
  |*                        DESIGN SUMMARY  (KIP-FEET)                      *|
  |*                        --------------                                  *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
  |         FX              MY             MZ          LOCATION              |
  |        ==================================================               |
  |          PASS        ANSI Q1.6-2      0.387             4                |
  |         10.00 C         0.00         3.92            10.00               |
  |*                                                                        *|
  |                                                                          |
  |--------------------------------------------------------------------------|
```

V. AISC N690 1994 Pipe

Verify the design of a pipe section used for a beam member per the AISC N690 1994 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,\ 300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2{}^4 - r_1{}^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2{}^3 - r_1{}^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 818: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.76	23.76	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1994 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
```

```
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
  |--------------------------------------------------------------------------|
  |                                                   Y        PROPERTIES    |
  |*************                                      |        IN INCH UNIT  |
  |            *  |=============================|   ==| |==    ------------  |
  |MEMBER   1  *  |  AISC SECTIONS              |     | |      AX =  13.70   |
  |            *  | ST  PIPS120                 |     | |  --Z  AY =   6.80   |
  |DESIGN CODE *  |                             |     | |      AZ =   6.80   |
  |ANSI N690-94*  ===============================   ==| |==    SY =  41.10   |
  |            *                                               SZ =  41.10   |
  |            *  |<---LENGTH (FT)=    4.17 --->|                 RY =
4.37  |
  |*************                                               RZ =   4.37   |
  |                                                                          |
  |                 27.6 (KIP-FEET)                                          |
  |PARAMETER        |               L1  L1  L1                 STRESSES      |
  |IN KIP  INCH     |                                          IN KIP  INCH  |
  |---------------  +           L1              L1             -------------|
  | KL/R-Y=   9.72  |                                          FA  =  15.87 |
  | KL/R-Z=   9.72  +       L1                       L1        fa  =   0.00 |
  | UNL   =  25.00  |                                          FCZ =  23.76 |
  | CB    =   1.00  +   L1                               L1    FTZ =  23.76 |
  | CMY   =   1.00  |                                          FCY =  23.76 |
  | CMZ   =   1.00  +                                          FTY =  23.76 |
  | FYLD  =  36.00  |L0                                    L0  fbz =   8.06 |
  | NSF   =   0.90  +---+---+---+---+---+---+---+---+---+---|  fby =   0.00 |
  | DFF   =   0.00  -1.5                                       Fey =1409.49 |
  | dff=      0.00              ABSOLUTE MZ ENVELOPE           Fez =1409.49 |
  | (KL/R)max =   9.72             (WITH LOAD NO.)             FV  =  14.40 |
  |                                                            fv  =   0.11 |
```

```
|                                                                           |
|                  MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                    |
|                  -------------------------                                |
|                                                                           |
|            AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z          |
|                                                                           |
|    VALUE     0.0        25.8        0.0        0.0         27.6           |
|  LOCATION    0.0         0.0        0.0        0.0          2.1           |
|   LOADING       0           1          0          0            1          |
|                                                                           |
|***************************************************************************|
|*                                                                         *|
|*                     DESIGN SUMMARY  (KIP-FEET)                          *|
|*                     --------------                                      *|
|*                                                                         *|
|*       RESULT/     CRITICAL COND/     RATIO/        LOADING/             *|
|         FX              MY             MZ          LOCATION              |
|        ======================================================            |
|          PASS        ANSI Q1.6-2      0.339             1                |
|          0.00 T          0.00       -27.60            2.08               |
|*                                                                         *|
|                                                                           |
```

Related Links

- *D1.K.1. ANSI/AISC N690-1994 Code* (on page 1296)

V. AISC N690 1994 W Shaped

Verify the design of a wide-flange section used for a cantilever beam using the AISC N690 1994 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in{\cdot}kips$

<u>Slenderness Ratio</u>

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

<u>Bending</u>

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 6.862 < 65/\sqrt{(F_y)}$$

Thus, allowable bending tensile stress = $F_{btz} = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress About Major Axis

$f_{btz} = M_z D / 2 \times I_z = 23.2$ ksi

Stress ratio = 23.2 ksi / 23.76 kis = 0.975

<u>Shear</u>

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = *120 inch*

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.91$ *inch*

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$

$t = t_w = 0.3$ inch

Cv =1.475

$F_v = (Fy/2.89)Cv \geq 0.4 \times F_y = 14.4$ *ksi* Hence allowable=14.4ksi

Actual Shear Stress

STAAD.Pro calculates the shear are based on the general shear stress equation, $\tau = V \times Q/(I \times t)$ from Timoshenko. Thus the shear area, $A_y = I \times t/Q = 4.401\ in^2$

$f_v = F_y/A_y = 12.5$ kip / 4.401 = 2.84 ksi

Stress ratio = $f_v/F_v = 0.18$

Results

Table 819: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	23.76	23.76	none	
Actual bending stress (ksi)	23.17	23.17	none	
Critical stress ratio	0.975	0.975	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.84	2.84	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1994 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC N690 1994
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ANSI N690-1994)   v1.0
                       ********************************************
   |-----------------------------------------------------------------------------|
   |                                                    Y        PROPERTIES      |
   |*************                                       |        IN INCH UNIT    |
   |            *  |=============================|   ===|===     ------------    |
   |MEMBER   1  *  |  AISC SECTIONS              |      |        AX =  11.80     |
   |            *  | ST  W16X40                  |      |    --Z  AY =   4.39    |
   |DESIGN CODE *  |                             |      |        AZ =   4.73     |
   |ANSI N690-94*  ===============================   ===|===     SY =   8.26     |
   |            *                                                SZ =  64.75     |
   |            *  |<---LENGTH (FT)=   10.00 --->|                  RY =
1.56  |
   |*************                                                RZ =   6.63     |
```

```
  |                                                                         |
  |                125.0 (KIP-FEET)                                         |
  |PARAMETER        |L1                                        STRESSES     |
  |IN KIP  INCH     |   L1  L1                                 IN KIP  INCH |
  |---------------  +           L1                             -------------|
  | KL/R-Y=  76.68  |               L1                         FA  =  15.57 |
  | KL/R-Z=  18.11  +                                          fa  =   0.00 |
  | UNL    =  72.00 |                       L1                 FCZ =  23.76 |
  | CB     =   1.00 +                                          FTZ =  23.76 |
  | CMY    =   0.85 |                           L1  L1         FCY =  27.00 |
  | CMZ    =   0.85 +                                   L1     FTY =  27.00 |
  | FYLD   =  36.00 |                                       L0 fbz =  23.17 |
  | NSF    =   1.00 +---+---+---+---+---+---+---+---+---+---|  fby =   0.00 |
  | DFF    =   0.00  -6.9                                      Fey =  25.40 |
  | dff=       0.00              ABSOLUTE MZ ENVELOPE          Fez = 455.24 |
  | (KL/R)max =  76.68              (WITH LOAD NO.)            FV  =  14.40 |
  |                                                            fv  =   2.84 |
  |                                                                         |
  |                      MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)              |
  |                      -------------------------                          |
  |                                                                         |
  |                AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z    |
  |                                                                         |
  |      VALUE       0.0        12.5        0.0         0.0       125.0     |
  |   LOCATION       0.0         0.0        0.0         0.0         0.0     |
  |    LOADING       0           1          0           0           1       |
  |                                                                         |
  |**************************************************************************|
  |*                                                                        *|
  |*                          DESIGN SUMMARY  (KIP-FEET)                    *|
  |*                          --------------                                *|
  |*                                                                        *|
  |*       RESULT/     CRITICAL COND/     RATIO/        LOADING/            *|
  |         FX                MY            MZ          LOCATION             |
  |        ======================================================           |
  |          PASS        ANSI Q1.6-2       0.975             1              |
  |          0.00 T          0.00       -125.00            0.00             |
  |*                                                                        *|
  |                                                                         |
```

Related Links

- *D1.K.1. ANSI/AISC N690-1994 Code* (on page 1296)

V. AISC N690 1984 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the AISC N690 1984 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

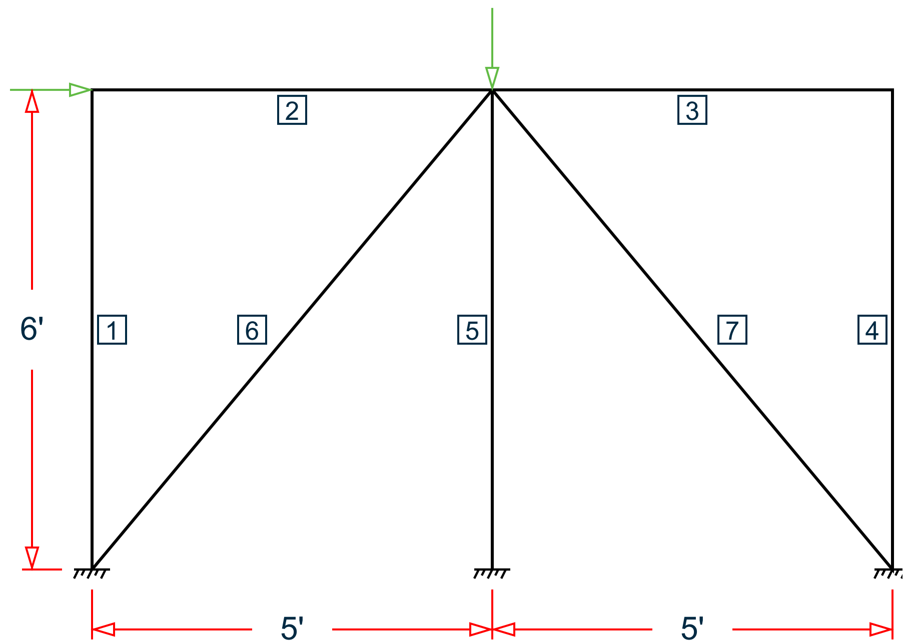


Figure 505: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = *36 ksi*, F_u = *58 ksi*).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

Axial Tension: Member 6

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21600 = 0.016

Axial Compression: Member 7

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 820: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
|-----------------------------------------------------------------------------
|
      6  ST   L806010                  (AISC SECTIONS)
                           PASS      Q1.5.1.1          0.016         1
                    -2666.18 T          0.00           0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |
```

```
|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AX.TENS: 3.52E+02                    COMPRESS:0.00E
+00                     |

|--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:   8.41  AYY:   5.00  AZZ:   3.75  RZZ:  1.29  RYY:
2.81                 |
 | SZZ:       5.16  SYY:
11.99                                            |

|--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                               |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:
1.0                                                 |

|--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    1    2.67E+03       -         -         -
-   |
 | COMP&BEND    0.000    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    1        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    1        -          -    0.00E+00       -
-   |

|--------------------------------------------------------------------------------
|
      7  ST   L806010                  (AISC SECTIONS)
                           PASS      Q1.5.1.3.1        0.049         1
                     7104.23 C          0.00          0.00        0.00

|--------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                     |
```

```
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                            |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                                |
 | AXX:   8.41  AYY:   5.00  AZZ:   3.75  RZZ:  1.29  RYY:
2.81                |
 | SZZ:       5.16  SYY:
11.99                                                |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                   |
 | KL/R-Z: 61.76  KL/R-Y: 28.34  UNL:   60.0  CMZ:  1.00  CMY:
1.00        |
 | CB: 0.00  FYLD: 36000.00  FU: 58000.00  NET SECTION FACTOR:
0.90         |
 | CT:  0.90  STEEL TYPE:
1.0                                                  |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD      FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    1    7.10E+03       -         -         -
-   |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-   |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.000    1    7.10E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    1        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    1        -         -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

V. AISC N690 1984 Channel

Verify the design of channel section used for a beam member using the AISC N690 1984 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0)\ /\ (2 \times 404) = 15.41\ \text{ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$F_y = 51.375$ kips

$V_y = D \times t_w = 10.8\ \text{in}^2$

fv,y = Fy/Vy = 4.76 ksi

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

$Cv = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4$ ksi

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v / F_v = 0.330$

Results

Table 821: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  AISC N690 1984)   v1.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                     (AISC SECTIONS)
                           PASS      Q1.6-Eqn 2        0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                           |
 | ACTUAL STRESSES: (UNIT - POUN
```

```
INCH)                                          |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  14.70  AYY:  10.74  AZZ:   4.84  RZZ:  5.24  RYY:
0.87                |
 | SZZ:      53.87  SYY:
3.77                                             |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.75   STEEL TYPE:
0.0                                                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    1    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    1    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.713    1    0.00E+00       -         -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    1    0.00E+00       -         -    8.30E+05 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04       -         -
-   |
 | SHEAR-Z      0.000    1        -         -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

V. AISC N690 1984 Pipe Section With SLC

Verify the design of a pipe section used for a beam member per the AISC N690 1984 code.

References

Gere, J. M. and S. P. Timoshenko. 1984. *Mechanics of Materials, 2nd Edition*. Boston, MA: PWS Publishers. p.238.

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

Details

The 10 ft long span is a propped cantilever which is free to move axially at the fixed end. The beam is subject to the following four different load cases:

1. a 10 kip axial tension force,
2. a 10 kip axial compression force,
3. a uniformly distributed load of 0.3 kip/ft in the Global Y and Z directions, the member selfweight (acts in Global Y), and the tension load from case 1
4. a uniformly distributed load of 0.3 kip/ft in the Global Y and Z directions, the member selfweight (acts in Global Y), and the compression load from case 2

The member is Pipe 5-Sched. 40 (standard) pipe section with Grade A36 steel. The self-weight is 14.6 lbs/ft.

Validation

Slenderness Ratio

The slenderness ratio is the same about either local z or y axes: $KL/r = 120\ in\ /\ 1.888\ in = 63.56 < 300$, OK

Axial Tension

From section Q1.5.1.1, the allowable tensile stress:

$$F_t = \min \left| \begin{array}{l} 0.6F_y = 0.6 \times 36 = 21.6 \text{ ksi} \\ 0.5F_u = 0.5 \times 58 = 29 \text{ ksi} \end{array} \right.$$

The actual tensile stress from load case 1:

$$f_t = \frac{P}{A} = \frac{10}{4.01} = 2.49 \text{ ksi} < F_t$$

Stress ratio = $2.49\ /\ 21.6 = 0.115$, hence OK.

Axial Compression

Allowable compressive stress per Cl Q1.9.2.3:

$$\frac{\text{OD}}{t} = \frac{5.56}{0.24} = 23.17 < \frac{3{,}300}{F_y} = \frac{3{,}300}{36} = 91.67$$

$$C_c = \sqrt{\frac{2\pi^2 E}{F_y}} = 126.1$$

Since $Kl/r < C_c$, per eq. Q1.5-1, the allowable axial compressive stress:

$$F_a = \frac{1 - \frac{\left(kl/r\right)^2}{2 \times C_c^2}}{\frac{5}{3} + 3 \times \frac{kl/r}{8 \times C_c} - \frac{\left(kl/r\right)^3}{8 \times C_c^3}} F_y = 17.08 \text{ ksi}$$

The actual compressive stress from load case 2:

$$f_a = \frac{P}{A} = \frac{10}{4.01} = 2.49 \text{ ksi} < F_a$$

Stress ratio = *2.49 / 17.08 = 0.146*, hence OK.

Bending

Section is compact per Cl Q1.5.1.4.1: $\frac{OD}{t} = 23.17 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress about both axes: $F_b = 0.66 \times F_y = 23.76\ ksi$

Moment due to load case 3 or 4 on a propped cantilever (ref. AISC Steel Construction Manual):

$$M_z = \frac{w \times l^2}{8} = \frac{0.314 \times 10^2}{8} = 3.93 \text{ ft}\cdot\text{k} = 47.2 \text{ in}\cdot\text{k}$$

$$M_y = \frac{w \times l^2}{8} = \frac{0.3 \times 10^2}{8} = 3.75 \text{ ft}\cdot\text{k} = 45 \text{ in}\cdot\text{k}$$

Actual bending stress about the major axis: $f_{bz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{47.2}{14.3}\left(\frac{5.56}{2}\right) = 9.17\ ksi$

Actual bending stress about the minor axis: $f_{by} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{45.0}{14.3}\left(\frac{5.56}{2}\right) = 8.75\ ksi$

Stress ratio: *9.17 / 23.76 = 0.386*, hence OK.

Combined Bending and Axial Tension

Interaction per Cl Q1.6.2 in load case 3 (tension + bending):

$$\frac{f_a}{\texttt{SFT} \times 0.6F_y} + \frac{f_{by}}{\texttt{SMY} \times F_{by}} + \frac{f_{bz}}{\texttt{SMZ} \times F_{bz}} = \frac{2.49}{1.3 \times 21.6} + \frac{8.75}{1.4 \times 23.76} + \frac{9.17}{1.4 \times 23.76} = 0.627 \quad \text{(Eq. Q1.6-1b)}$$

Combined Bending and Axial Compression

Interaction per Cl Q1.6 in load case 4 (compression + bending):

$$\frac{f_a}{F_a} = 0.145 < 0.15 \quad \text{(Eq. Q1.6-2)}$$

$$\frac{f_a}{\texttt{SFC} \times F_a} + \frac{f_{by}}{\texttt{SMY} \times F_{by}} + \frac{f_{bz}}{\texttt{SMZ} \times F_{bz}} = \frac{2.49}{1.3 \times 17.08} + \frac{8.75}{1.4 \times 23.76} + \frac{9.17}{1.4 \times 23.76} = 0.651$$

Shear

Actual shear (at supports):

$$f_{\text{v,y}} = \frac{5w \times l}{8} = \frac{5 \times 0.315 \times 10}{8} = 1.97 \text{ kip}$$

$$f_{\text{v,z}} = \frac{5w \times l}{8} = \frac{5 \times 0.3 \times 10}{8} = 1.88 \text{ kip}$$

Shear area (calculated as per Gere & Timoshenko reference):

$$\text{A}_z = \frac{\frac{\pi}{4}\left(\text{r}_2{}^4 - \text{r}_1{}^4\right) \times 2\left(\text{r}_2 - \text{r}_1\right)}{\frac{2}{3}\left(\text{r}_2{}^3 - \text{r}_2{}^3\right)}$$

where

r_1 = the inner radius of the pipe = *5.56 / 2 - 0.24 = 2.54 in*

r_2 = the outer radius of the pipe = $5.56\ /\ 2 = 2.78\ in$

Thus, the shear area along both axes = $2.008\ in^2$.

$$F_{vy} = \frac{F_{v,y}}{A_y} = \frac{1.97}{2.008} = 0.98 \text{ ksi}$$

$$F_{vz} = \frac{F_{v,z}}{A_y} = \frac{1.88}{2.008} = 0.94 \text{ ksi}$$

The allowable shear stress:

$$F_v = 0.4F_y = 14.4 \text{ ksi}$$

Stress ratio in Y (minor axis): $0.98/14.4 = 0.068$, hence OK.

Stress ratio in Z (major axis): $0.94/14.4 = 0.065$, hence OK.

Results

Table 822: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	63.56	63.55	negligible	
Actual axial tensile stress (ksi)	2.49	2.49	none	
Allowable axial tensile stress	21.6	21.6	none	
Tensile stress ratio (LC1)	0.115	0.115	none	
Actual axial compressive stress (ksi)	2.49	2.49	none	
Allowable axial compressive stress	17.08	17.1	negligible	
Compressive stress ratio (LC2)	0.146	0.146	none	
Allowable bending stress (ksi)	23.76	23.8	negligible	
Actual bending stress (ksi)	9.17	9.15	negligible	
Combined axial tension and bending stress ratio (LC3)	0.627	0.625	negligible	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Combined axial compression and bending stress ratio (LC4)	0.651	0.649	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	0.98	0.97	negligible	
Shear stress ratio in major axis	0.065	0.064	negligible	
Shear stress ratio in minor axis	0.068	0.067	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 Pipe Section With SLC.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 05-Apr-19
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+006
POISSON 0.3
DENSITY 0.489024
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS50
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX
LOAD 1 TENSION ONLY
JOINT LOAD
```

```
2 FX 10
LOAD 2 COMPRESSION ONLY
JOINT LOAD
2 FX -10
LOAD 3 TENSION+BENDING
SELFWEIGHT Y -1
MEMBER LOAD
1 UNI GY -0.3
JOINT LOAD
2 FX 10
MEMBER LOAD
1 UNI GZ 0.3
LOAD 4 COMPRESSION+BENDING
SELFWEIGHT Y -1
MEMBER LOAD
1 UNI GY -0.3
JOINT LOAD
2 FX -10
MEMBER LOAD
1 UNI GZ 0.3
PERFORM ANALYSIS PRINT LOAD DATA
PRINT MEMBER PROPERTIES ALL
UNIT INCHES KIP
PARAMETER 1
CODE AISC N690 1984
CB 1 ALL
CMY 1 ALL
CMZ 1 ALL
FYLD 36 ALL
FU 58 ALL
KY 1 ALL
KZ 1 ALL
MAIN 200 ALL
NSF 1 ALL
RATIO 1 ALL
SMZ 1.4 ALL
SMY 1.4 ALL
SFT 1.3 ALL
SFC 1.3 ALL
TMAIN 300 ALL
TRACK 2 ALL
UNB 30 ALL
UNT 30 ALL
CT 0.75 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      STAAD SPACE                                               -- PAGE NO.
6
                       STAAD.PRO CODE CHECKING - (  AISC N690 1984)   v1.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1  ST   PIPS50                   (AISC SECTIONS)
                          PASS      Q1.6-Eqn 2        0.651         4
                      10.00 C         45.00          47.04      120.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  63.55 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                 |
 | AXIAL: 1.71E+01  FCZ: 2.38E+01  FCY: 2.38E+01  FTZ: 2.38E+01  FTY: 2.38E
+01 |
 | SHEAR: 1.44E
+01                                                          |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                      |
 | AXIAL: 2.49E+00  FBZ: 9.15E+00  FBY: 8.75E+00 SHEAR:
9.72E-01               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                       |
 | AXX:   4.01  AYY:   2.02  AZZ:   2.02  RZZ:  1.89  RYY:
1.89               |
 | SZZ:       5.14  SYY:
5.14                                               |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                          |
 | KL/R-Z: 63.55   KL/R-Y: 63.55   UNL:   30.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD:    36.00   FU:    58.00   NET SECTION FACTOR:
1.00          |
 | CT:  0.75   STEEL TYPE:
0.0                                                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                        |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.115    1    1.00E+01       -         -         -
-    |
 | COMPRESSION  0.146    2    1.00E+01       -         -         -
-    |
 | COMP&BEND    0.651    4    1.00E+01       -         -    4.70E+01 4.50E
+01   |
 | TEN&BEND     0.627    3    1.00E+01       -         -    4.70E+01 4.50E
+01   |
 | SHEAR-Y      0.067    3        -     1.96E+00       -         -
-    |
 | SHEAR-Z      0.065    3        -         -    1.88E+00       -
-    |
```

```
|-----------------------------------------------------------------------------
|
```

V. AISC N690 1984 Pipe

Verify the design of a pipe section used for a beam member per the AISC N690 1984 code.

Reference

Timoshenko & Gere. *Mechanics of Materials*. 2nd Edition. p.238

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76$ ksi

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2{}^4 - r_1{}^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2{}^3 - r_1{}^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78$ ksi

Actual shear stress (at midspan): $f_v = 0.75 / 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 823: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress ratio	0.263	0.263	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
```

```
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
STYPE 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  AISC N690 1984)   v1.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                  (AISC SECTIONS)
                           PASS      Q1.6-Eqn 2        0.339         1
                        0.00 T          0.00     -331249.97       25.00

 |-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                    |
 | AXIAL: 1.59E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                            |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
```

```
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 1.10E
+02               |

 |-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                               |
 | AXX:  13.70  AYY:   6.80  AZZ:   6.80  RZZ:  4.37  RYY:
4.37                |
 | SZZ:      41.10  SYY:
41.10                                                |

 |-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                   |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:
1.0                                                   |

 |-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    1    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    1    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.339    1    0.00E+00       -         -    3.31E+05 0.00E
+00  |
 | TEN&BEND     0.000    1    0.00E+00       -         -    3.31E+05 0.00E
+00  |
 | SHEAR-Y      0.263    1        -     2.57E+04       -         -
-   |
 | SHEAR-Z      0.000    1        -          -    0.00E+00       -
-   |

 |-----------------------------------------------------------------------------
|
```

V. AISC N690 1984 Tee

Verify the design of a pipe section used for a beam member per the AISC N690 1984 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

<u>Slenderness Ratio</u>

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

<u>Bending</u>

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8\text{ ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93\ ksi < 23.8\ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7\ ksi > 23.8\ ksi$

Critical stress ratio = $38.7\ ksi\ /\ 23.8\ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01\ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.65$

$$t = t_w = 0.44\text{ inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4\ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$$Vy = Dt_w = 10.68\text{ in}^2$$

$$fv = Fy/Vy = 5.805\text{ ksi}$$

Stress ratio = fv/Fv = 0.403

Allowable bending stress about minor axis, $f_{btz} = 0.75 \times F_y = 27\ ksi$

Results

Table 824: Verification comparison for AISC_N690_1984_Tee

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
INPUT WIDTH 79
SET SHEAR
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE AISC N690 1984
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
STYPE 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  AISC N690 1984)   v1.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                  (AISC SECTIONS)
                           FAIL      Q1.6-Eqn 2        1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                    |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.70E+04  FTZ: 2.38E+04  FTY: 2.70E
+04 |
 | SHEAR: 1.44E
+04                                                            |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
```

```
INCH)                                                 |
 | AXX:  19.95  AYY:  10.68  AZZ:   9.48  RZZ:  5.66  RYY:
2.37              |
 | SZZ:      49.69  SYY:
18.75                                                  |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                     |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:
0.0                                                      |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                         |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    1    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    1    0.00E+00       -         -         -
-   |
 | COMP&BEND    1.620    1    0.00E+00       -         -    1.92E+06 0.00E
+00   |
 | TEN&BEND     0.000    1    0.00E+00       -         -    1.92E+06 0.00E
+00   |
 | SHEAR-Y      0.403    1       -     6.20E+04       -         -
-   |
 | SHEAR-Z      0.000    1       -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

V. AISC N690 1984 W Shaped

Verify the design of a wide-flange section used for a cantilever beam using the AISC N690 1984 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in \cdot kips$

<u>Slenderness Ratio</u>

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

<u>Bending</u>

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 6.862 < 65/\sqrt{}(F_y)$$

Thus, allowable bending tensile stress = $F_{btz} = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress About Major Axis

$$f_{btz} = M_z D / 2 \times I_z = 23.2 \text{ ksi}$$

Stress ratio = 23.2 ksi / 23.76 kis = 0.975

<u>Shear</u>

Allowable Shear Stress

$$F_v = 0.4 \times F_y = 14.4\ ksi$$

Actual Shear Stress

$$V_y = D \times t_w = 4.8 \text{ in}^2$$

$$f_v = F_y/V_y = 2.6 \text{ ksi}$$

Stress ratio = f_v/F_v = 0.18

Results

Table 825: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.975	0.975	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.60	2.56	1.5%	
Shear stress ratio	0.180	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISC\AISC N690 1984 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISC N690 1984
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  AISC N690 1984)   v1.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1  ST   W16X40                    (AISC SECTIONS)
                          PASS      Q1.6-Eqn 2        0.975         1
                       0.00 T          0.00       -1500.00        0.00

 |-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                   |
 | AXIAL: 1.49E+01  FCZ: 2.38E+01  FCY: 2.70E+01  FTZ: 2.38E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                         |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                        |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00                |

 |-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  11.80  AYY:   4.88  AZZ:   7.07  RZZ:  6.63  RYY:
1.56             |
 | SZZ:      64.75  SYY:
8.26                                              |

 |-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85        |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00          |
 | CT:  0.75   STEEL TYPE:
0.0                                                  |

 |-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    1    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    1    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.975    1    0.00E+00       -         -    1.50E+03 0.00E
+00  |
 | TEN&BEND     0.000    1    0.00E+00       -         -    1.50E+03 0.00E
+00  |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-   |
 | SHEAR-Z      0.000    1        -          -    0.00E+00       -
-   |
```

```
|--------------------------------------------------------------------------------
|
```

Related Links

- *D1.K.2. ANSI/AISC N690-1984 Code* (on page 1303)

V. AISI

V. AISI 2016 2CU F2F Compressive Strength

Verify the axial compressive strength of a double cold-formed channel section according to the AISI S100 2016 specification using both the LRFD and ASD methods.

References

1. American Iron and Steel Institute. *AISI Manual: Cold-formed Steel Design - Vol. 1.*. 2018. Milwaukee, WI.
2. Young, Warren C. and Richard G. Budynas. *Roark's Formulas for Stress and Strain.* Seventh Edition. New York, NY: McGraw Hill, 2002. p. 414

Details

The member is 10 ft long cantilever. The section used is a pair of 600T125-54 channels in a front-to-front configuration (no spacing between channel toes).

Section Properties

Properties of a 600T125-54 (listed as a 600CU125-54 in STAAD.Pro):

Depth, $D = 6.198\ in$
Flange width, $B = 1.250\ in$

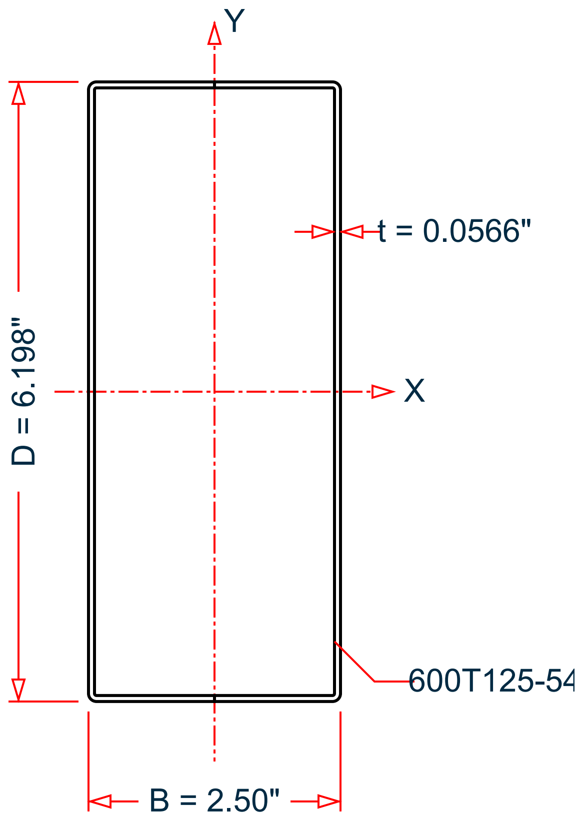
Thickness, $t = 0.0566\ in$
Inner radius, $R = 0.0849\ in.$
Cross-section area, $A = 0.48\ in^2$
Moment of inertia about the Z axis, $I_z = 2.34\ in^4$
Moment of inertia about the Y axis, $I_y = 0.0539\ in^4$
Heel to C.G. distance, $x = 0.204\ in$
Torsional constant, $J = 0.000513\ in^4$
Warping constant, $C_w = 0.348\ in^6$

Material Properties

$E = 29{,}000\ ksi$
$G = 11{,}200\ ksi$
$F_y = 36\ ksi$
$F_u = 58\ ksi$

Calculations

Section Properties



Average radius, $r = R + t/2 = 0.0849 + 0.0566\ /\ 2 = 0.113\ in.$
Clear depth of web, $a = D = (2\times r + t) = 6.198 - (2\times 0.113 + 0.0566) = 5.915\ in.$
Clear width of the flanges, $b = B - (r + t/2) = 1.25 - (0.113 + 0.0556\ /\ 2) = 1.109\ in.$
Cross-section area, $A = 2\times 0.48 = 0.96\ in^2$
Moment of inertia about the Z axis, $I_z = 2\times 2.34 = 4.68\ in^4$
Moment of inertia about the Y axis, $I_y = 2[0.0539 + 0.48\times(1.25 - 0.204)^2] = 1.158\ in^4$

Radius of gyration about the Z axis, $r_z = \sqrt{I_z/A} = \sqrt{\frac{4.68}{0.96}} = 2.21\ in$

Radius of gyration about the Y axis, $r_y = \sqrt{I_y/A} = \sqrt{\frac{1.158}{0.96}} = 1.10\ in$

Torsional constant, $J = 2\times 0.000513 = 0.00103\ in^4$

Center-to-center depth, $\bar{a} = D - t = 6.198 - 0.0566 = 6.141\ in$

Center-to-edge flange width, $\bar{b} = B - t/2 = 1.25 - 0.0566/2 = 1.222\ in$

Warping constant, $C_w = \frac{t\bar{a}^3}{24}\left(56\bar{b}^3 + 4\bar{a}\bar{b}^2\right) = \frac{0.0566(6.141)^3}{24}\left[56(1.222)^3 + 4(6.141)(1.222)^2\right] = 12.344\ in^6$
(Taken from Table 10.2, Case 4 of Reference 2; simplified for the case where $h = 2\times h_1$)

<u>Compression Capacity for Yielding and Global Buckling</u>

About the major axis:

The elastic flexural buckling stress

$$F_{crez} = \frac{\pi^2 E}{\left(\frac{K_z L_z}{r_z}\right)^2} = \frac{\pi^2 \times 29{,}000}{\left(\frac{1.0 \times 120}{2.21}\right)^2} = 96.9 \text{ ksi}$$ (Eq. E2.1-1)

$$\lambda_{cz} = \sqrt{\frac{F_y}{F_{crez}}} = \sqrt{\frac{36}{96.9}} = 0.610$$ (Eq. E2-4)

Since $\lambda_{cz} < 1.5$, the compressive strength is given by:

$$F_{nz} = 0.658^{\lambda_{cz}^2} \times F_y = 0.658^{(0.610)^2} \times 36 = 30.82 \text{ ksi}$$ (Eq. 2-2)

About the minor axis:

The elastic flexural buckling stress

$$F_{crey} = \frac{\pi^2 E}{\left(\frac{K_y L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29{,}000}{\left(\frac{1.0 \times 120}{1.10}\right)^2} = 24.0 \text{ ksi}$$ (Eq. E2.1-1)

$$\lambda_{cy} = \sqrt{\frac{F_y}{F_{crey}}} = \sqrt{\frac{36}{24.0}} = 1.225$$ (Eq. E2-4)

Since $\lambda_{cy} < 1.5$, the compressive strength is given by:

$$F_{ny} = 0.658^{\lambda_{cy}^2} \times F_y = 0.658^{(1.225)^2} \times 36 = 19.20 \text{ ksi}$$ (Eq. 2-2)

The controlling compressive stress is the minimum of F_{nz} and F_{ny}: $F_n = 19.20$ *ksi*

The nominal axial compressive strength, $P_{ne} = A \times F_n = 0.96 \times 19.20 = 18.43$ *kips*

LRFD: $\phi_c P_{ne} = 0.85 \times 18.43 = 15.67$ *kips*
ASD: P_{ne}/Ω_c = 18.43 / 1.8 = 10.24 kips

Compression Capacity in Local Buckling

Global column stress about major axis (as defined in Section E2):

$$F_n = F_{ny} = 19.20 \text{ ksi}$$

Determine the effective area, A_e as per Appendix 1. The flanges of a front-to-front channel are considered stiffened elements.

Compression flange effective width:

w = b = 1.109 in (AISC Manual 3.3.2)

$$F_{cr\ell} = k \frac{\pi^2 E}{12\left(1 - \mu^2\right)}\left(\frac{t}{w}\right)^2$$ (E1. 1.1-4)

where

k	=	Plate bucking coefficient, = *4*
μ	=	Poisson's ratio of steel, = *0.3*

$$F_{cr\ell} = 4\frac{\pi^2 \times 29{,}000}{12\left[1 - (0.3)^2\right]}\left(\frac{0.0566}{1.1097}\right)^2 = 273.1 \text{ ksi}$$

The slenderness factor:

$$\lambda = \sqrt{\frac{F_n}{F_{cr\ell}}} = \sqrt{\frac{19.20}{273.1}} = 0.265 \quad \text{(Eq. 1.1-3)}$$

As $\lambda < 0.673$, the local reduction factor, $\rho = 1$. Therefore, the effective width of the compression flange is $b_f = \rho \times w = 1.109\ in$.

Web effective width:

$$w = a = 5.915 \text{ in} \quad \text{(AISC Manual 3.3.2)}$$

$$F_{cr\ell} = 4\frac{\pi^2 \times 29{,}000}{12\left[1 - (0.3)^2\right]}\left(\frac{0.0566}{5.915}\right)^2 = 9.60 \text{ ksi}$$

$$\lambda = \sqrt{\frac{F_n}{F_{cr\ell}}} = \sqrt{\frac{19.20}{9.60}} = 1.414$$

As $\lambda > 0.673$, the local reduction factor:

$$\rho = \left(1 - {}^{0.22}\!/_{\lambda}\right)/\lambda = \left(1 - {}^{0.22}\!/_{1.414}\right)/\,1.414 = 0.597$$

Therefore, the effective width of the compression flange is $b_w = \rho \times w = 0.597 \times 5.915 = 3.532\ in$

The total effective area is the gross area minus the difference in nominal widths and effective widths:

$$A_e = A - 4\times(b - b_f)t - 2\times(a - b_w)t = 0.96 - 4\times(1.109 - 1.109)\times0.0566 - 2\times(5.915 - 3.532)\times0.0566 = 0.69 \text{ in.}^2$$

$$P_{n\ell} = A_e F_n = 0.69 \times 19.20 = 13.25 < P_{ne} \quad \text{(Eq. E3.1-1)}$$

LRFD: $\phi_c P_{n\ell} = 0.85 \times 11.26 = kips$
ASD: $P_{n\ell}/\Omega_c = 13.26 / 1.8 = 7.36$ kips

Comparison

Table 826: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Axial comp. per section E2 (kips)	15.67	15.671	negligible	
	Local comp. per section E3 (kips)	11.26	11.283	negligible	
ASD	Axial comp. per section E2 (kips)	10.24	10.242	negligible	
	Local comp. per section E3 (kips)	7.36	7.375	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 2CU F2F Compressive Strength.STD` is typically installed with the program.

The following D1.E.5 Design Parameters (on page 1192) are used:

- The material properties for design are specified by the parameters `FYLD 36` and `FU 58`.
- A member with unreinforced webs is specified using `TSA 0`

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jan-22
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
G 1.6128e+06
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE FR 600CU125-54
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 10
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:FR  600CU125-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.322             REF: Sec. D2       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. D2              LC:           1
|
|  Fx:(T)      -10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.96000       Az:     0.25096            Ay:     0.66958
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         4.68000       Iy:     1.15815             J:     0.00104
|
|  Sz:         1.51016       Sy:     0.92652
|
|  Rz:         2.20794       Ry:     1.09837            Cw:    12.34418
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000       G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  109.253   Allowable :  300.000   Ratio :   0.364
```

```
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     17.085   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |    104.505   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States          | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |     1 |   0.00 |  -10.000 |   31.104 |0.322|Sec. D2
|
|   Yielding          |     1 |   0.00 |  -10.000 |   31.104 |0.322|Sec. D2
|
|   Rupture           |     1 |   0.00 |  -10.000 |   41.760 |0.239|Sec. D3
|
| Compression         |   -   |    -   |     -    |   11.283 |  -  |Sec. E3
|
|   Local Comp        |   -   |    -   |     -    |   11.283 |  -  |Sec. E3
|
|   Axial Comp        |   -   |    -   |     -    |   15.671 |  -  |Sec. E2
|
| Bending             |   -   |    -   |     -    |   14.721 |  -  |Sec. F2
|
|   Major Local Bend  |   -   |    -   |     -    |   14.756 |  -  |Sec. F3
|
|   Minor Local Bend  |   -   |    -   |     -    |   14.846 |  -  |Sec. F3
|
|   Major Sectional   |   -   |    -   |     -    |   48.929 |  -  |Sec. F2
|
|   Minor Sectional   |   -   |    -   |     -    |   30.019 |  -  |Sec. F2
|
|   LTB Major         |   -   |    -   |     -    |   14.721 |  -  |Sec. F2
|
|   LTB Minor         |   -   |    -   |     -    |   23.956 |  -  |Sec. F2
|
| Shear               |   -   |    -   |     -    |    5.150 |  -  |Sec. G2
|
```

```
|   Major Shear (Z)  |    -   |    -    |      -    |    5.150 |  -  |Sec. G2
|
|   Minor Shear (Y)  |    -   |    -    |      -    |    8.154 |  -  |Sec. G2
|
| Interaction        |      1 |   0.00 |      -    |      -   |0.322|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |      1 |   0.00 |      -    |      -   |0.322|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
    43. PARAMETER 2
    44. CODE AISI 2016
    45. METHOD ASD
    46. FYLD 36 ALL
    47. FU 58 ALL
    48. FLX 0 ALL
    49. TSA 0 ALL
    50. TRACK 2 ALL
    51. CHECK CODE ALL
 STEEL DESIGN
   WARNING # 14 ~ Channel front to front member #     1 will be considered
welded throughout length.
   Spacing will not be considered.
      STAAD SPACE                                              -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:FR  600CU125-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.483             REF: Sec. D2       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. D2              LC:           1
|
|  Fx:(T)      -10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.96000       Az:     0.25096            Ay:     0.66958
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         4.68000       Iy:     1.15815             J:     0.00104
|
|  Sz:         1.51016       Sy:     0.92652
|
|  Rz:         2.20794       Ry:     1.09837            Cw:    12.34418
|
```

```
|--------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000       G:  11200.000
|
|--------------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|--------------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|--------------------------------------------------------------------------------
|
|  LC :     1   Actual :  109.253   Allowable :  300.000   Ratio :   0.364
|
|--------------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|--------------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     17.085   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |    104.505   |    200.000   |      PASS
|
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                               - MEMBER:       1 (Contd.....)
|
|
|
|--------------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |     1 |   0.00 |  -10.000 |   20.695 |0.483|Sec. D2
|
|   Yielding         |     1 |   0.00 |  -10.000 |   20.695 |0.483|Sec. D2
|
|   Rupture          |     1 |   0.00 |  -10.000 |   27.840 |0.359|Sec. D3
```

```
|
| Compression        |   -  |   -   |     -   |    7.375 |  -  |Sec. E3
|
|   Local Comp       |   -  |   -   |     -   |    7.375 |  -  |Sec. E3
|
|   Axial Comp       |   -  |   -   |     -   |   10.242 |  -  |Sec. E2
|
| Bending            |   -  |   -   |     -   |    9.795 |  -  |Sec. F2
|
|   Major Local Bend |   -  |   -   |     -   |    9.818 |  -  |Sec. F3
|
|   Minor Local Bend |   -  |   -   |     -   |    9.878 |  -  |Sec. F3
|
|   Major Sectional  |   -  |   -   |     -   |   32.554 |  -  |Sec. F2
|
|   Minor Sectional  |   -  |   -   |     -   |   19.973 |  -  |Sec. F2
|
|   LTB Major        |   -  |   -   |     -   |    9.795 |  -  |Sec. F2
|
|   LTB Minor        |   -  |   -   |     -   |   15.939 |  -  |Sec. F2
|
| Shear              |   -  |   -   |     -   |    3.388 |  -  |Sec. G2
|
|   Major Shear (Z)  |   -  |   -   |     -   |    3.388 |  -  |Sec. G2
|
|   Minor Shear (Y)  |   -  |   -   |     -   |    5.364 |  -  |Sec. G2
|
| Interaction        |     1 |  0.00 |     -   |     -    |0.483|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     1 |  0.00 |     -   |     -    |0.483|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 2CU F2F Tensile Strength

Verify the axial tensile strength of a double cold-formed channel section according to the AISI S100 2016 specification using both the LRFD and ASD methods.

Reference

1. American Iron and Steel Institute. *AISI Manual: Cold-formed Steel Design - Vol. 1.*. 2018. Milwaukee, WI.

Details

The member is 10 ft long cantilever. The section used is a pair of 600T125-54 channels in a front-to-front configuration (no spacing between channel toes). Assume a net section area of $0.90\times$ the gross section area.

<u>Section Properties</u>

Properties of a 600T125-54 (listed as a 600CU125-54 in STAAD.Pro):

Cross-section area, $A = 0.48\ in^2$

<u>Material Properties</u>

$E = 29{,}000\ ksi$
$G = 11{,}200\ ksi$
$F_y = 36\ ksi$
$F_u = 58\ ksi$

Calculations

Section Properties

Cross-section area, $A = 2\times\ 0.48 = 0.96\ in^2$

The net area of the cross section:

$$A_n = 0.9\times\ (0.96) = 0.864\ \text{in}^2$$

Tensile Yielding of Gross Section

$$T_n = A_g \times F_y = 0.96 \times 36 = 34.56 \quad \text{(Eq. D2-1)}$$

LRFD: $\phi_t T_n = 0.90 \times 34.56 = 31.10\ kips$
ASD: T_n/Ω_t = 34.56 / 1.67 = 20.69 kips

Tensile Rupture of Net Section

$$T_n = A_n \times F_y = 0.864 \times 58 = 50.11 \quad \text{(Eq. D3-1)}$$

LRFD: $\phi_t T_n = 0.75 \times 50.11 = 37.58\ kips$
ASD: T_n/Ω_t = 50.11 / 2.00 = 25.06 kips

Comparison

Table 827: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Tensile Yielding of Gross Section, $\phi_t T_n$, (kN)	31.10	31.104	negligible	
	Tensile Rupture of Net Section, $\phi_t T_n$, (kN)	37.58	37.584	negligible	
ASD	Tensile Yielding of Gross Section, T_n/Ω_t, (kN)	20.69	20.695	negligible	
	Tensile Rupture of Net Section, T_n/Ω_t, (kN)	25.06	25.056	negligible	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 2CU F2F Tensile Strength.STD` is typically installed with the program.

The following design parameters are used:

- The net section factor is specified using the `NSF 0.9` parameter.
- Unreinforced webs are specified using the `TSA 0` parameter.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jan-22
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
G 1.6128e+06
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT INCHES KIP
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE FR 600CU125-54
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 10
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
NSF 0.9 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
FYLD 36 ALL
```

```
FU 58 ALL
FLX 0 ALL
NSF 0.9 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
              *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:FR  600CU125-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.322             REF: Sec. D2       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. D2              LC:           1
|
|  Fx:(T)      -10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.96000       Az:     0.25096            Ay:     0.66958
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         4.68000       Iy:     1.15815             J:     0.00104
|
|  Sz:         1.51016       Sy:     0.92652
|
|  Rz:         2.20794       Ry:     1.09837            Cw:    12.34418
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000      G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
```

```
||  Kz:   1.000    Ky:   1.000    NSF:  0.900
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  109.253   Allowable :  300.000   Ratio :   0.364
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     17.085   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |    104.505   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                               - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |     1 |   0.00 |  -10.000 |   31.104 |0.322|Sec. D2
|
|   Yielding          |     1 |   0.00 |  -10.000 |   31.104 |0.322|Sec. D2
|
|   Rupture           |     1 |   0.00 |  -10.000 |   37.584 |0.266|Sec. D3
|
| Compression         |   -   |   -    |     -    |   11.283 |  -  |Sec. E3
|
|   Local Comp        |   -   |   -    |     -    |   11.283 |  -  |Sec. E3
|
|   Axial Comp        |   -   |   -    |     -    |   15.671 |  -  |Sec. E2
|
| Bending             |   -   |   -    |     -    |   14.721 |  -  |Sec. F2
|
|   Major Local Bend  |   -   |   -    |     -    |   14.756 |  -  |Sec. F3
|
|   Minor Local Bend  |   -   |   -    |     -    |   14.846 |  -  |Sec. F3
|
|   Major Sectional   |   -   |   -    |     -    |   48.929 |  -  |Sec. F2
```

```
|
|   Minor Sectional  |   -   |   -    |     -    |   30.019 |  -  |Sec. F2
|
|   LTB Major        |   -   |   -    |     -    |   14.721 |  -  |Sec. F2
|
|   LTB Minor        |   -   |   -    |     -    |   23.956 |  -  |Sec. F2
|
| Shear              |   -   |   -    |     -    |    5.150 |  -  |Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |    5.150 |  -  |Sec. G2
|
|   Minor Shear (Y)  |   -   |   -    |     -    |    8.154 |  -  |Sec. G2
|
| Interaction        |     1 |   0.00 |     -    |     -    |0.322|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     1 |   0.00 |     -    |     -    |0.322|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
    44. PARAMETER 2
    45. CODE AISI 2016
    46. METHOD ASD
    47. FYLD 36 ALL
    48. FU 58 ALL
    49. FLX 0 ALL
    50. NSF 0.9 ALL
    51. TSA 0 ALL
    52. TRACK 2 ALL
    53. CHECK CODE ALL
 STEEL DESIGN
   WARNING # 14 ~ Channel front to front member #     1 will be considered
welded throughout length.
   Spacing will not be considered.
      STAAD SPACE                                              -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:FR  600CU125-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.483            REF: Sec. D2       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. D2              LC:           1
|
|  Fx:(T)      -10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
```

```
|  Ag:         0.96000      Az:     0.25096           Ay:     0.66958
|
|  Cz:         0.00000      Cy:     0.00000
|
|  Iz:         4.68000      Iy:     1.15815            J:     0.00104
|
|  Sz:         1.51016      Sy:     0.92652
|
|  Rz:         2.20794      Ry:     1.09837           Cw:    12.34418
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000      G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  0.900
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  109.253   Allowable :  300.000   Ratio :   0.364
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     17.085   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |    104.505   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                               -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|
|-----------------------------------------------------------------------------
```

```
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |     1 |   0.00 | -10.000 |   20.695 |0.483|Sec. D2
|
|   Yielding         |     1 |   0.00 | -10.000 |   20.695 |0.483|Sec. D2
|
|   Rupture          |     1 |   0.00 | -10.000 |   25.056 |0.399|Sec. D3
|
| Compression        |   -   |   -    |    -     |    7.375 |  -  |Sec. E3
|
|   Local Comp       |   -   |   -    |    -     |    7.375 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |    -     |   10.242 |  -  |Sec. E2
|
| Bending            |   -   |   -    |    -     |    9.795 |  -  |Sec. F2
|
|   Major Local Bend |   -   |   -    |    -     |    9.818 |  -  |Sec. F3
|
|   Minor Local Bend |   -   |   -    |    -     |    9.878 |  -  |Sec. F3
|
|   Major Sectional  |   -   |   -    |    -     |   32.554 |  -  |Sec. F2
|
|   Minor Sectional  |   -   |   -    |    -     |   19.973 |  -  |Sec. F2
|
|   LTB Major        |   -   |   -    |    -     |    9.795 |  -  |Sec. F2
|
|   LTB Minor        |   -   |   -    |    -     |   15.939 |  -  |Sec. F2
|
| Shear              |   -   |   -    |    -     |    3.388 |  -  |Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |    -     |    3.388 |  -  |Sec. G2
|
|   Minor Shear (Y)  |   -   |   -    |    -     |    5.364 |  -  |Sec. G2
|
| Interaction        |     1 |   0.00 |    -     |    -     |0.483|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     1 |   0.00 |    -     |    -     |0.483|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 Channel section

Determine the flexural strength and shear strength of a cold-formed channel section without lips according to AISI S100 2016 Specification using both LRFD and ASD methods.

Details

The member is a simply-supported, 6 ft span. The member is subject to a uniform load of 194.4 lbs/ft. The section used is a 550CU125-54.

Section Properties

Depth of section, $d = 5.698\ in$
Width of section, $b = 1.25\ in$
Thickness, $t = 0.0566\ in$
Fillet radius, $r = 0.0849\ in$
Area, $A = 0.452\ in^2$
Moment of inertia major axis, $I_{zz} = 1.9\ in^4$
Moment of inertia minor axis, $I_{yy} = 0.053\ in^4$
Section modulus major axis, $Z_{zz} = 0.667\ in^3$
Section modulus minor axis, $Z_{yy} = 0.0512\ in^3$
Distance between centroid and shear center: $x_0 = -0.539\ in$
Radius of gyration about major axis: $r_x = \sqrt{I_x / A} = 2.053\ in.$
Radius of gyration about minor axis: $r_y = \sqrt{I_t / A} = 0.342\ in.$
Polar radius of gyration: $r_0 = \sqrt{r_x^2 + r_y^2 + x_0^2} = 2.15\ in.$
Torsion constant, $J = 0.00048\ in^4$

Material Properties

$E = 29{,}500\ ksi$
$G = 11{,}300\ ksi$
$F_y = 33\ ksi$

Design Forces

$M_z = 10.5\ ft \cdot k$
$V_y = 0.583\ kips$

Validation

Lateral-Torsional Buckling Strength

$$\sigma_{ey} = \frac{\pi^2 \times E}{\left(K_y \times L_y / r_y\right)^2} = \frac{\pi^2 \times 29{,}000}{\left(1 \times 72 / 0.342\right)^2} = 6.586 \text{ ksi}$$

$$\sigma_t = \frac{1}{A \times r_0^2} \times \left[G \times J + \frac{\pi^2 \times E \times C_w}{(Kt \times Lt)^2}\right] = \frac{1}{0.452(2.15)^2} \times \left[11{,}300 \times 0.000483 + \frac{\pi^2 \times 29{,}000 \times 0.315}{(1 \times 72)^2}\right] = 11.12 \text{ k}$$

$$C_b = 1.136$$

For singly symmetric sections bent about the axis of symmetry:

$$F_{cre} = \frac{C_b r_0 A}{S_f}\sqrt{\sigma_{ey}\sigma_t} = \frac{1.136(2.15)(0.452)}{0.668}\sqrt{6.586 \times 11.12} = 14.15 \text{ ksi}$$

where

S_f = the elastic section modulus for the unreduced section relative to the extreme compression fiber per AISI S100 2016. Hence, $S_f = S_z$.

For $F_{cre} < 0.56F_y = 0.56 \times 33 = 18.58$, $F_n = F_{cre}$.

$$M_{ne} = F_n \times S_f = 14.15 \times 0.667 = 9.433 \text{ in-k}$$

Allowable nominal lateral-torsional buckling strength (ASD):

$$\frac{M_{ne}}{\Omega_b} = \frac{9.433}{1.67} = 5.649 \text{ in}\cdot\text{kips}$$

Design nominal lateral-torsional buckling strength (LRFD):

$$\phi_b M_{ne} = 0.9 \times 9.433 = 8.49 \text{ in}\cdot\text{kips}$$

<u>Bending About Major Axis</u>

$$M_y = F_y \times S_f = 33 \times 0.667 = 22.01 \text{ in-k}$$

Allowable nominal bending strength (ASD):

$$\frac{M_y}{\Omega_b} = \frac{21.01}{1.67} = 13.18 \text{ in}\cdot\text{kips}$$

Design nominal bending strength (LRFD):

$$\phi_b M_y = 0.9 \times 22.01 = 19.81 \text{ in}\cdot\text{kips}$$

<u>Local Buckling Interaction with Yield and Global Buckling</u>

From Section F3:

$$S_e = 0.665 \text{ in}^3$$

$$M_{nl} = S_e \times F_n = 0.665 \times 14.15 = 9.412 \text{ in-k}$$

Allowable nominal bending strength (ASD):

$$\frac{M_{nl}}{\Omega_b} = \frac{9.412}{1.67} = 5.636 \text{ in}\cdot\text{kips}$$

Design nominal bending strength (LRFD):

$$\phi_b M_{nl} = 0.9 \times 9.412 = 8.471 \text{ in}\cdot\text{kips}$$

Governing strength ratio:

LRFD: 10.498 / 8.471 = 1.239

ASD: 10.498 / 5.636 = 1.862

<u>Shear Strength along Minor Axis</u>

$$h = D - 2\times(t - r) = 5.698 - 2\times(0.0566 - 0.0849) = 5.145 \text{ in}$$

$$h/t = 5.415 / 0.0566 = 95.67$$

$$A_w = h \times t = 5.415 \times 0.0566 = 0.307 \text{ in}^2$$

$$V_y = 0.6 \times A_w \times F_y = 0.6 \times 0.307 \times 33 = 6.069 \text{ kip}$$

$$F_{cr} = \frac{\pi^2 \times EK_v}{12\left(1 - \mu^2\right)\left(\frac{h}{t}\right)^2} = \frac{\pi^2 \times 29{,}000 \times 5.34}{12\left(1 - 0.3^2\right)(95.67)^2} = 15.56 \text{ ksi}$$

$$V_{cr} = A_w \times F_{cr} = 0.307 \times 15.56 = 4.75 \text{ kip}$$

$$\lambda_v = \sqrt{\frac{V_y}{V_{cr}}} = \sqrt{\frac{6.069}{4.75}} = 1.128$$

For $0.815 < \lambda_v \leq 1.227$,

$$V_n = 0.815\sqrt{V_{cr} \times V_y} = 0.815\sqrt{4.75 \times 6.069} \times 6.069 = 4.384 \text{ kip}$$

Allowable nominal shear strength (ASD):

$$\frac{V_n}{\Omega_v} = \frac{4.384}{1.60} = 2.739 \text{ kips}$$

Design nominal shear strength (LRFD):

$$\phi_v V_n = 0.95 \times 4.384 = 4.615 \text{ kips}$$

Governing strength ratio:

LRFD: 0.583 / 4.165 = 0.14

ASD: 0.583 / 2.74 = 0.213

Table 828: Comparison of results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Major bending (in-k)	19.81	19.807	negligible	
	LTB bending (in-k)	8.49	8.49	none	
	Major local bending (in-k)	8.471	8.477	negligible	
	Bending ratio	1.239	1.238	negligible	
	Shear strength (kips)	4.165	4.163	negligible	
	Shear ratio	0.14	0.14	none	
ASD	Major bending (in-k)	13.18	13.178	negligible	
	LTB bending (in-k)	5.649	5.649	none	
	Major local bending (in-k)	5.636	5.64	negligible	
	Bending ratio	1.862	1.861	negligible	
	Shear strength (kips)	2.739	2.739	none	
	Shear ratio	0.213	0.213	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 Channel section.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME AISI C-Section Without Lips Braced at Mid-Span
JOB CLIENT BENTLEY SYSTEMS Inc.
ENGINEER DATE 04-Mar-20
JOB REV 1.0
JOB REF AISI Manual Cold-Formed Steel Design - Vol. 1
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 72 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29500
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 550CU125-54
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
1 UNI GY -0.0162
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD LRFD
BEAM 1 ALL
FYLD 33 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 1
CODE AISI 2016
METHOD ASD
BEAM 1 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1(*)  SECTION:ST  550CU125-54     LEN:   72.000     LOC:
36.000 |
| STATUS: FAIL       RATIO:  1.238            REF: Sec. F3       LC:      2
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   36.000  Criteria:   Sec. F3             LC:           2
|
|  Fx:(C)        0.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -10.498
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.45200       Az:     0.12548            Ay:     0.30649
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         1.90000       Iy:     0.05300             J:     0.00048
|
|  Sz:         0.66690       Sy:     0.05121
|
|  Rz:         2.05025       Ry:     0.34243            Cw:     0.31500
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    33.000        Fu:    58.000      E:  29500.000       G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     2   Actual :  210.263   Allowable :  300.000   Ratio :   0.701
```

```
|
|--------------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|--------------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     19.585   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |     95.671   |    200.000   |      PASS
|
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                                 -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|--------------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   13.424 |  -  |Sec. D2
|
|   Yielding         |   -   |   -    |     -    |   13.424 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   19.662 |  -  |Sec. D3
|
| Compression        |   -   |   -    |     -    |    2.185 |  -  |Sec. E3
|
|   Local Comp       |   -   |   -    |     -    |    2.185 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |     -    |    2.219 |  -  |Sec. E2
|
| Bending            |     2 |  36.00 |  -10.498 |    8.477 |1.238|Sec. F3
|
|   Major Local Bend |     2 |  36.00 |  -10.498 |    8.477 |1.238|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |    1.518 |  -  |Sec. F3
|
|   Major Sectional  |     2 |  36.00 |  -10.498 |   19.807 |0.530|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |    1.521 |  -  |Sec. F2
|
|   LTB Major        |     2 |  36.00 |  -10.498 |    8.490 |1.236|Sec. F2
|
| Shear              |     2 |   0.00 |    0.583 |    4.163 |0.140|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |    2.360 |  -  |Sec. G2
|
```

```
|   Minor Shear (Y)  |     2 |   0.00 |    0.583 |    4.163 |0.140|Sec. G2
|
| Interaction        |     2 |  36.00 |     -    |     -    |1.238|Eq. H1.1-2
|
|   (Bend+Ten) Int1  |     2 |  36.00 |     -    |     -    |0.530|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |  36.00 |     -    |     -    |1.238|Eq. H1.1-2
|
|   (Bend+Comp)      |     2 |  36.00 |     -    |     -    |1.238|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     2 |  36.00 |     -    |     -    |0.591|Sec. H2
|
|-----------------------------------------------------------------------------
|
    43. PARAMETER 1
    44. CODE AISI 2016
    45. METHOD ASD
    46. BEAM 1 ALL
    47. TRACK 2 ALL
    48. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                              -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1(*)  SECTION:ST  550CU125-54     LEN:   72.000     LOC:
36.000 |
| STATUS: FAIL       RATIO:  1.861             REF: Sec. F3       LC:      2
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   36.000  Criteria:   Sec. F3              LC:           2
|
|  Fx:(C)        0.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -10.498
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.45200       Az:     0.12548            Ay:     0.30649
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         1.90000       Iy:     0.05300             J:     0.00048
|
|  Sz:         0.66690       Sy:     0.05121
|
|  Rz:         2.05025       Ry:     0.34243            Cw:     0.31500
|
|-----------------------------------------------------------------------------
|
```

```
| MATERIAL INFO:
|
|  Fy:    33.000        Fu:    58.000      E:  29500.000       G:  11300.000
|
|----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|----------------------------------------------------------------------------
|
|  LC :     2   Actual :  210.263   Allowable :  300.000   Ratio :   0.701
|
|----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     19.585   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |     95.671   |    200.000   |      PASS
|
|----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|----------------------------------------------------------------------------
|
| Limit States         | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension              |   -   |   -    |     -    |    8.932 |  -  |Sec. D2
|
|   Yielding           |   -   |   -    |     -    |    8.932 |  -  |Sec. D2
|
|   Rupture            |   -   |   -    |     -    |   13.108 |  -  |Sec. D3
|
| Compression          |   -   |   -    |     -    |    1.428 |  -  |Sec. E3
```

```
|
|   Local Comp       |   -   |   -    |     -    |    1.428 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |     -    |    1.450 |  -  |Sec. E2
|
| Bending            |     2 |  36.00 |  -10.498 |    5.640 |1.861|Sec. F3
|
|   Major Local Bend |     2 |  36.00 |  -10.498 |    5.640 |1.861|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |    1.010 |  -  |Sec. F3
|
|   Major Sectional  |     2 |  36.00 |  -10.498 |   13.178 |0.797|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |    1.012 |  -  |Sec. F2
|
|   LTB Major        |     2 |  36.00 |  -10.498 |    5.649 |1.858|Sec. F2
|
| Shear              |     2 |   0.00 |    0.583 |    2.739 |0.213|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |    1.553 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     2 |   0.00 |    0.583 |    2.739 |0.213|Sec. G2
|
| Interaction        |     2 |  36.00 |     -    |     -    |1.861|Eq. H1.1-2
|
|   (Bend+Ten) Int1  |     2 |  36.00 |     -    |     -    |0.797|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |  36.00 |     -    |     -    |1.861|Eq. H1.1-2
|
|   (Bend+Comp)      |     2 |  36.00 |     -    |     -    |1.861|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     2 |  36.00 |     -    |     -    |0.888|Sec. H2
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 CS Compressive Strength

Verify the compressive strength of a cold-formed channel section with lips per the AISI S100 2016 specification.

Details

A 10' long, simply supported beam is subjected to a 10 kip axial load (compression). The section used is a 362CS162-54, Grade 50 steel.

Section Properties

Taken from AISI Manual, Cold-Formed Steel Design - Vol. 1, 2017 Table I-2.

Area, $A_g = 0.422\ in^2$
Radius of gyration z axis, $r_z = 1.44in$
Radius of gyration y axis, $r_y = 0.605\ in$
Distance from centroid to shear center, $x_o = -1.28\ in$
Elastic section modulus y axis, $S_{fz} = 0.482\ in^3$
Elastic section modulus z axis, $S_{fy} = 0.142\ in^3$

Torsional constant, $J = 0.000451\ in^4$
Warping constant, $C_w = 0.457\ in^6$
$j = 2.02\ in$

Material Properties

$E = 29{,}000\ ksi$
$G = 11{,}300\ ksi$
$F_y = 50\ ksi$

Validation

The elastic flexural buckling stress about the z axis:

$$F_{\text{crez1}} = \frac{\pi^2 E}{(K_z L_z / r_z)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 120 / 1.44)^2} = 41.22 \text{ ksi} \quad \text{(Eq. E2.1-1)}$$

Polar radius of gyration of cross-section about shear center:

$$r_o = \sqrt{r_z^2 + r_y^2 + x_o^2} = 2.019 \text{ in} \quad \text{(Eq. E2.2-4)}$$

$$\beta = 1 - (x_o / r_o)^2 = 1 - \left(\frac{-1.28}{2.019}\right)^2 = 0.598 \quad \text{(Eq. E2.2-3)}$$

$$\sigma_t = \frac{1}{A_g r_o^2}\left[GJ + \frac{\pi^2 E C_w}{(K_t L_t)^2}\right] = \frac{1}{0.422 \times 2.019^2}\left[11{,}200 \times 0.000451 + \frac{\pi^2 \times 29{,}000 \times 0.457}{(1 \times 120)^2}\right] = 8.243 \text{ ksi} \quad \text{(Eq. E2.2-5)}$$

$$\sigma_{\text{ez}} = F_{\text{crez}} = 41.22 \text{ ksi} \quad \text{(Eq. E2.2-6)}$$

The flexural-torsional buckling stress:

$$F_{\text{cre}} = \frac{1}{2\beta}\left[\left(\sigma_{\text{ez}} + \sigma_t\right) - \sqrt{(\sigma_{\text{ez}} + \sigma_t)^2 - 4\beta\sigma_{\text{ez}}\sigma_t}\right] \quad \text{(Eq. E2.2-1)}$$

$$= \frac{1}{2 \times 0.598}\left[\left(41.22 + 8.197\right) - \sqrt{(41.22 + 8.197)^2 - 4 \times 0.598 \times 41.22 \times 8.197}\right] = 7.560 \text{ ksi}$$

The elastic global buckling stress about the z axis is the minimum of F_{cre} and F_{crez1}; so $F_{crez} = 7.560\ ksi$.

The compressive stress capacity is:

$$\lambda_{\text{cz}} = \sqrt{\frac{F_y}{F_{\text{cre}}}} = \sqrt{\frac{50}{7.560}} = 2.572 > 1.5 \quad \text{(Eq. E2-4)}$$

$$F_{\text{nz}} = \left(\frac{0.877}{\lambda_{\text{cz}}^2}\right)F_y = \frac{0.877}{(2.579)^2}50 = 6.630 \text{ ksi}$$

The elastic flexural buckling stress about the y axis:

$$F_{\text{crey1}} = \frac{\pi^2 E}{(K_y L_y / r_y)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 120 / 0.536)^2} = 7.275 \text{ ksi} \quad \text{(Eq. E2.1-1)}$$

The elastic global buckling stress about the z axis is the minimum of F_{cre} and F_{crey1}; so $F_{crey} = 7.275\ ksi$.

$$\lambda_{\text{cy}} = \sqrt{\frac{F_y}{F_{\text{crey}}}} = \sqrt{\frac{50}{7.275}} = 2.622 > 1.5 \quad \text{(Eq. E2-4)}$$

$$F_{ny} = \left(\frac{0.877}{\lambda_{cy}{}^2}\right)F_y = \frac{0.877}{(2.622)^2}50 = 6.380 \text{ ksi} \quad \text{(Eq. E2-2)}$$

Global buckling about the Y axis governs.

Nominal capacity against global buckling:

$$P_n = A_g F_n = 0.422 \times 6.380 = 2.692 \text{ kips}$$

The available axial strength:

LRFD: $\phi_c P_n = 0.85 \times 2.692 = 2.289$ *kips*
ASD: P_n/Ω_c = 2.692 / 1.8 = 1.496 kips

Determine the compression capacity for local buckling interacting with yielding and global buckling as per E3:

The effective area, A_e, is equal to the gross area, $A_g = 0.422\ in^2$. By inspection, the local capacity, $P^{n\ell}$ is equal to the global buckling capacity in this case.

LRFD: $\phi_c P_{n\ell} = 0.85 \times 2.692 = 2.289$ *kips*
ASD: $P_{n\ell}/\Omega_c$ = 2.692 / 1.8 = 1.496 kips

Determine the compression capacity for distortional buckling as per E4:

The elastic distortional buckling stress, $F_{crd} = 149.8$ *ksi*.

The elastic distortional buckling load, $P_{crd} = A_g \times F_{crd} = 0.422 \times 149.8 = 63.21$ *kips*.

$$P_y = A_g \times F_y = 0.422 \times 50 = 21.1 \text{ kips} \quad \text{(Eq. E4.1-4)}$$

$$\lambda_d = \sqrt{\frac{P_y}{P_{crd}}} = \sqrt{\frac{21.1}{63.21}} = 0.578 > 0.561 \quad \text{(Eq. E4.1-3)}$$

$$P_{nd} = \left[1 - 0.25\left(\frac{P_{crd}}{P_y}\right)^{0.6}\right]\left(\frac{P_{crd}}{P_y}\right)^{0.6} P_y \quad \text{(Eq. 4.1-2)}$$

$$= \left[1 - 0.25\left(\frac{63.21}{21.1}\right)^{0.6}\right]\left(\frac{63.21}{21.1}\right)^{0.6} 21.1 = 21.08 \text{ ksi}$$

The available axial strength:

LRFD: $\phi_c P_{nd} = 0.85 \times 21.08 = 17.91$ *kips*
ASD: P_{nd}/Ω_c = 21.08 / 1.8 = 11.71 kips

Results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Local compression capacity (kips)	2.289	2.282	negligible	
	Distorional buckling capacity (kips)	17.91	17.914	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Axial compression capacity (kips)	2.289	2.282	negligible	
ASD	Local compression capacity (kips)	1.496	1.491	negligible	
	Distorional buckling capacity (kips)	11.71	11.709	negligible	
	Axial compression capacity (kips)	1.496	1.491	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\09 Steel Design\US\AISI\AISI 2016 CS Compressive Strength.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
G 1.6128e+06
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 362CS162-54
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX 10
PERFORM ANALYSIS
```

```
UNIT INCHES KIP
PARAMETER 1
CODE AISI 2016
METHOD LRFD
FYLD 50 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
FYLD 50 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  362CS162-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.545             REF: Sec. D3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. D3              LC:           1
|
|  Fx:(T)      -10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.42200       Az:     0.15191            Ay:     0.18916
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         0.87300       Iy:     0.15400             J:     0.00045
|
|  Sz:         0.48166       Sy:     0.14141
|
|  Rz:         1.43830       Ry:     0.60409            Cw:     0.45700
|
|-----------------------------------------------------------------------------
```

```
 |
 | MATERIAL INFO:
 |
 |  Fy:    50.000        Fu:    58.000      E:  29000.000       G:  11200.000
 |
 |-----------------------------------------------------------------------------
 |
 | DESIGN PROPERTIES:
 |
 |  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
 |
 | DESIGN PARAMETERS:
 |
 |  Kz:   1.000    Ky:   1.000    NSF:  1.000
 |
 |-----------------------------------------------------------------------------
 |
 | CRITICAL SLENDERNESS:
 |
 |-----------------------------------------------------------------------------
 |
 |  LC :     1   Actual :  198.645   Allowable :  300.000   Ratio :   0.662
 |
 |-----------------------------------------------------------------------------
 |
 |  Section Dimension Check      |    Ratio     |    Limit     |     Status
 |
 |-----------------------------------------------------------------------------
 |
 |  Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
 |
 |  Width To Thickness Ratio     |     23.710   |     60.000   |      PASS
 |
 |  Web Depth To Thickness Ratio |     59.046   |    200.000   |      PASS
 |
 |-----------------------------------------------------------------------------
 |
      STAAD SPACE                                                -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
 |                                               - MEMBER:       1 (Contd.....)
 |
 |
 |
 |
 |-----------------------------------------------------------------------------
 |
 | Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
 |
 |--------------------|-------|--------|----------|----------|-----|-----------
 |
 | Tension            |     1 |   0.00 |  -10.000 |   18.357 |0.545|Sec. D3
 |
 |   Yielding         |     1 |   0.00 |  -10.000 |   18.990 |0.527|Sec. D2
 |
 |   Rupture          |     1 |   0.00 |  -10.000 |   18.357 |0.545|Sec. D3
 |
```

```
| Compression         |   -  |   -   |     -    |    2.282 |  -  |Sec. E2
|
|   Local Comp        |   -  |   -   |     -    |    2.282 |  -  |Sec. E3
|
|   Distortional      |   -  |   -   |     -    |   17.914 |  -  |Sec. E4
|
|   Axial Comp        |   -  |   -   |     -    |    2.282 |  -  |Sec. E2
|
| Bending             |   -  |   -   |     -    |    5.920 |  -  |Sec. F2
|
|   Major Local Bend  |   -  |   -   |     -    |    5.924 |  -  |Sec. F3
|
|   Minor Local Bend  |   -  |   -   |     -    |    6.239 |  -  |Sec. F3
|
|   Distortional      |   -  |   -   |     -    |   21.674 |  -  |Sec. F4
|
|   Major Sectional   |   -  |   -   |     -    |   21.674 |  -  |Sec. F2
|
|   Minor Sectional   |   -  |   -   |     -    |    6.364 |  -  |Sec. F2
|
|   LTB Major         |   -  |   -   |     -    |    5.920 |  -  |Sec. F2
|
| Shear               |   -  |   -   |     -    |    4.330 |  -  |Sec. G2
|
|   Major Shear (Z)   |   -  |   -   |     -    |    4.330 |  -  |Sec. G2
|
|   Minor Shear (Y)   |   -  |   -   |     -    |    5.081 |  -  |Sec. G2
|
| Interaction         |     1 |  0.00 |     -    |     -    |0.545|Eq. H1.1-1
|
|   (Bend+Ten) Int1   |     1 |  0.00 |     -    |     -    |0.545|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
    42. PARAMETER 2
    43. CODE AISI 2016
    44. METHOD ASD
    45. FYLD 50 ALL
    46. FU 58 ALL
    47. FLX 0 ALL
    48. TSA 0 ALL
    49. TRACK 2 ALL
    50. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  362CS162-54     LEN:  120.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.817            REF: Sec. D3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
```

```
||
| Location:    0.000  Criteria:   Sec. D3              LC:           1
|
| Fx:(T)      -10.000       Fy:          0.000          Fz:       0.000
|
| Mx:           0.000       My:          0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
| Ag:         0.42200       Az:      0.15191            Ay:     0.18916
|
| Cz:         0.00000       Cy:      0.00000
|
| Iz:         0.87300       Iy:      0.15400             J:     0.00045
|
| Sz:         0.48166       Sy:      0.14141
|
| Rz:         1.43830       Ry:      0.60409            Cw:     0.45700
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
| Fy:    50.000        Fu:    58.000      E:  29000.000       G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
| Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
| Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
| LC :     1   Actual :  198.645   Allowable :  300.000   Ratio :   0.662
|
|-----------------------------------------------------------------------------
|
| Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
| Radius To Thickness Ratio    |      1.500   |     10.000   |      PASS
|
| Width To Thickness Ratio     |     23.710   |     60.000   |      PASS
|
| Web Depth To Thickness Ratio |     59.046   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
```

```
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |     1 |   0.00 | -10.000 |   12.238 |0.817|Sec. D3
|
|   Yielding          |     1 |   0.00 | -10.000 |   12.635 |0.791|Sec. D2
|
|   Rupture           |     1 |   0.00 | -10.000 |   12.238 |0.817|Sec. D3
|
| Compression         |   -   |   -    |    -     |    1.491 |  -  |Sec. E2
|
|   Local Comp        |   -   |   -    |    -     |    1.491 |  -  |Sec. E3
|
|   Distortional      |   -   |   -    |    -     |   11.709 |  -  |Sec. E4
|
|   Axial Comp        |   -   |   -    |    -     |    1.491 |  -  |Sec. E2
|
| Bending             |   -   |   -    |    -     |    3.939 |  -  |Sec. F2
|
|   Major Local Bend |   -   |   -    |    -     |    3.941 |  -  |Sec. F3
|
|   Minor Local Bend |   -   |   -    |    -     |    4.151 |  -  |Sec. F3
|
|   Distortional      |   -   |   -    |    -     |   14.421 |  -  |Sec. F4
|
|   Major Sectional   |   -   |   -    |    -     |   14.421 |  -  |Sec. F2
|
|   Minor Sectional   |   -   |   -    |    -     |    4.234 |  -  |Sec. F2
|
|   LTB Major         |   -   |   -    |    -     |    3.939 |  -  |Sec. F2
|
| Shear               |   -   |   -    |    -     |    2.848 |  -  |Sec. G2
|
|   Major Shear (Z)   |   -   |   -    |    -     |    2.848 |  -  |Sec. G2
|
|   Minor Shear (Y)   |   -   |   -    |    -     |    3.343 |  -  |Sec. G2
|
| Interaction         |     1 |   0.00 |    -     |     -    |0.817|Eq. H1.1-1
|
|   (Bend+Ten) Int1   |     1 |   0.00 |    -     |     -    |0.817|Eq. H1.1-1
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 CS Flexural Strength

Verify the flexural strength of a cold-formed channel section according to the AISI S100 2016 specification using both the LRFD and ASD methods.

Details

A 5' long, simply supported beam is subjected to a concentrated load of 1 kip/in at midspan in both the global Y and Z directions. The section used is a 12CS4x105, Gr. 36 steel.

Section Properties

Taken from AISI Manual, Cold-Formed Steel Design - Vol. 1, 2017 Table I-1.

Area, $A_g = 2.20\ in^2$
Radius of gyration z axis, $r_z = 4.65in$
Radius of gyration y axis, $r_y = 1.39\ in$
Distance from centroid to shear center, $x_o = -2.63\ in$
Elastic section modulus y axis, $S_{fy} = 1.45\ in^3$
Elastic section modulus z axis, $S_{fz} = 7.91\ in^3$
Torsional constant, $J = 0.00808\ in^4$
Warping constant, $C_w = 122\ in^6$
$j = 6.74\ in$

Material Properties

$E = 29{,}000\ ksi$
$G = 11{,}300\ ksi$
$F_y = 36\ ksi$

Validation

The modification factor, C_b, for a simply-supported beam under a concentrated load at midspan:

$$C_b = \frac{12.5 \times M_{\text{max}}}{2.5 \times M_{\text{max}} + 3 \times M_{\text{a}} + 4 \times M_{\text{b}} + 3 \times M_{\text{c}}} = 1.32 \quad \text{(Eq. F2.1.1-2)}$$

Polar radius of gyration of cross-section about shear center:

$$r_o = \sqrt{{r_z}^2 + {r_y}^2 + {x_o}^2} = 5.52 \text{ in} \quad \text{(Eq. F2.1.1-3)}$$

Major axis bending:

$$\sigma_{\text{ey}} = \frac{\pi^2 E}{(K_y L_y / r_y)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 60 / 1.39)^2} = 153.6 \text{ ksi} \quad \text{(Eq. F2.1.1-4)}$$

$$\sigma_t = \frac{1}{A{r_o}^2}\left[GJ + \frac{\pi^2 E C_w}{(K_t L_t)^2}\right] = \frac{1}{2.20 \times 5.52^2}\left[11{,}200 \times 0.00808 + \frac{\pi^2 \times 29{,}000 \times 122}{(1 \times 60)^2}\right] = 146.0 \text{ ksi} \quad \text{(Eq. F2.1.1-5)}$$

$$F_{\text{cre}} = \frac{C_b r_o A}{S_f}\sqrt{\sigma_{\text{ey}} \times \sigma_t} = \frac{1.32 \times 5.52 \times 2.20}{7.91}\sqrt{153.6 \times 146.0} = 299.4 \text{ ksi} \quad \text{(Eq. F2.1.1-1)}$$

Per F2.1, where $F_{cre} > 2.78F_y = 2.78(36) = 100.1\ ksi$, $F_n = F_y$ (Eq. F2.1-3).

Nominal capacity against lateral-torsional buckling about the major axis:

$$M_{nez} = S_{fz}F_n = 7.91 \times 36 = 284.8 \text{ in·kips}$$

The available flexural strength about the major axis:

LRFD: $\phi_b M_{nez} = 0.9 \times 284.8 = 256.3 \text{ in·kips}$
ASD: $M_{nez}/\Omega_b = 284.8 / 1.67 = 170.5$ in·kips

Minor axis bending:

$$\sigma_{ez} = \frac{\pi^2 E}{(K_z L_z / r_z)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 60 / 4.65)^2} = 1{,}719 \text{ ksi} \quad \text{(Eq. F2.1.2-2)}$$

Simply-supported beam, so $C_{TF} = 1$; assume compression on shear center side of centroid, so $C_S = +1$.

$$F_{cre} = \frac{C_S A \sigma_{ez}}{C_{TF} S_f}\left[j + C_S \sqrt{j^2 + r_o^2 (\sigma_t / \sigma_{ez})} \right] = \frac{1.0 \times 2.20 \times 1{,}719}{1.0 \times 1.45}\left[6.74 + 1.0\sqrt{(6.74)^2 + (5.52)^2 (146.0 / 1{,}719)} \right] =$$

Per F2.1, where $F_{cre} > 2.78F_y = 2.78(36) = 100.1 \text{ ksi}, F_n = F_y$ (Eq. F2.1-3).

Nominal capacity against lateral-torsional buckling about the major axis:

$$M_{ney} = S_{fy}F_n = 1.45 \times 36 = 52.2 \text{ in·kips}$$

The available flexural strength about the major axis:

LRFD: $\phi_b M_{ney} = 0.9 \times 52.2 = 47.0 \text{ in·kips}$
ASD: $M_{ney}/\Omega_b = 52.2 / 1.67 = 31.3$ in·kips

Results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Flexural capacity about major axis (in·kips)	256.3	256.5	negligible	
	Flexural capacity about minor axis (in·kips)	47.0	46.898	negligible	
ASD	Flexural capacity about major axis (in·kips)	170.5	170.659	negligible	
	Flexural capacity about minor axis (in·kips)	31.3	31.203	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 CS Flexural Strength.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
```

```
ENGINEER DATE 11-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 5 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
G 1.6128e+06
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 12CS4X105
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
UNIT INCHES KIP
MEMBER LOAD
1 CON GY -1
1 CON GZ 1
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD LRFD
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
```

```
|-------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12CS4X105       LEN:   60.000     LOC:
30.000 |
| STATUS: PASS       RATIO:  0.400             REF: Eq. H1.1-2    LC:      1
|
|-------------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   30.000  Criteria:   Eq. H1.1-2          LC:           1
|
|  Fx:(C)        0.000       Fy:         0.500          Fz:      -0.500
|
|  Mx:           0.000       My:       -15.000          Mz:     -15.000
|
|-------------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:          2.20000       Az:     0.71715            Ay:     1.19857
|
|  Cz:          0.00000       Cy:     0.00000
|
|  Iz:        47.50000       Iy:     4.27000             J:     0.00808
|
|  Sz:          7.91667       Sy:     1.44746
|
|  Rz:          4.64660       Ry:     1.39317            Cw:   122.00000
|
|-------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000      G:  11200.000
|
|-------------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    60.000   Lz:    60.000   Ly:    60.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-------------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-------------------------------------------------------------------------------
|
|  LC :     1   Actual :   43.067   Allowable :  300.000   Ratio :   0.144
|
|-------------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
```

```
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     32.524   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States         | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension              |   -   |   -    |     -    |   71.280 |  -  |Sec. D2
|
|   Yielding           |   -   |   -    |     -    |   71.280 |  -  |Sec. D2
|
|   Rupture            |   -   |   -    |     -    |   95.700 |  -  |Sec. D3
|
| Compression          |   -   |   -    |     -    |   39.758 |  -  |Sec. E3
|
|   Local Comp         |   -   |   -    |     -    |   39.758 |  -  |Sec. E3
|
|   Distortional       |   -   |   -    |     -    |   50.926 |  -  |Sec. E4
|
|   Axial Comp         |   -   |   -    |     -    |   60.583 |  -  |Sec. E2
|
| Bending              |     1 |  30.00 |  -15.000 |   44.537 |0.337|Sec. F3
|
|   Major Local Bend |     1 |  30.00 |  -15.000 |  235.933 |0.064|Sec. F3
|
|   Minor Local Bend |     1 |  30.00 |  -15.000 |   44.537 |0.337|Sec. F3
|
|   Distortional       |     1 |  30.00 |  -15.000 |  242.008 |0.062|Sec. F4
|
|   Major Sectional    |     1 |  30.00 |  -15.000 |  256.500 |0.058|Sec. F2
|
|   Minor Sectional    |     1 |   0.00 |  -15.000 |   46.898 |0.320|Sec. F2
|
| Shear                |     1 |   0.00 |    0.500 |   13.487 |0.037|Sec. G2
|
|   Major Shear (Z)    |     1 |   0.00 |   -0.500 |   14.716 |0.034|Sec. G2
|
|   Minor Shear (Y)    |     1 |   0.00 |    0.500 |   13.487 |0.037|Sec. G2
|
| Interaction          |     1 |  30.00 |     -    |     -    |0.400|Eq. H1.1-2
```

```
|
|   (Bend+Ten) Int1  |     1 |  30.00 |     -    |     -    |0.378|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     1 |  30.00 |     -    |     -    |0.400|Eq. H1.1-2
|
|   (Bend+Comp)      |     1 |  30.00 |     -    |     -    |0.400|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     1 |  30.00 |     -    |     -    |0.074|Sec. H2
|
|   (Bend+Shear) Y   |     1 |  30.00 |     -    |     -    |0.339|Sec. H2
|
|-----------------------------------------------------------------------------
|
    44. PARAMETER 2
    45. CODE AISI 2016
    46. METHOD ASD
    47. FYLD 36 ALL
    48. FU 58 ALL
    49. FLX 0 ALL
    50. TSA 0 ALL
    51. TRACK 2 ALL
    52. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12CS4X105       LEN:   60.000     LOC:
30.000 |
| STATUS: PASS       RATIO:  0.602            REF: Eq. H1.1-2    LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   30.000  Criteria:   Eq. H1.1-2           LC:           1
|
|  Fx:(C)        0.000       Fy:          0.500         Fz:      -0.500
|
|  Mx:           0.000       My:        -15.000         Mz:     -15.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.20000       Az:     0.71715            Ay:     1.19857
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:        47.50000       Iy:     4.27000             J:     0.00808
|
|  Sz:         7.91667       Sy:     1.44746
|
|  Rz:         4.64660       Ry:     1.39317            Cw:   122.00000
|
```

```
|--------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000       G:  11200.000
|
|--------------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    60.000   Lz:    60.000   Ly:    60.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|--------------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|--------------------------------------------------------------------------------
|
|  LC :     1   Actual :   43.067   Allowable :  300.000   Ratio :   0.144
|
|--------------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|--------------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     32.524   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|--------------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                               - MEMBER:       1 (Contd.....)
|
|
|
|--------------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   47.425 |  -  |Sec. D2
|
|   Yielding         |   -   |   -    |     -    |   47.425 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   63.800 |  -  |Sec. D3
```

```
|
| Compression         |  -  |   -   |    -    |   25.986 |  -  |Sec. E3
|
|   Local Comp        |  -  |   -   |    -    |   25.986 |  -  |Sec. E3
|
|   Distortional      |  -  |   -   |    -    |   33.285 |  -  |Sec. E4
|
|   Axial Comp        |  -  |   -   |    -    |   39.597 |  -  |Sec. E2
|
| Bending             |   1 | 30.00 | -15.000 |   29.632 |0.506|Sec. F3
|
|   Major Local Bend  |   1 | 30.00 | -15.000 |  156.974 |0.096|Sec. F3
|
|   Minor Local Bend  |   1 | 30.00 | -15.000 |   29.632 |0.506|Sec. F3
|
|   Distortional      |   1 | 30.00 | -15.000 |  161.016 |0.093|Sec. F4
|
|   Major Sectional   |   1 | 30.00 | -15.000 |  170.659 |0.088|Sec. F2
|
|   Minor Sectional   |   1 |  0.00 | -15.000 |   31.203 |0.481|Sec. F2
|
| Shear               |   1 |  0.00 |   0.500 |    8.873 |0.056|Sec. G2
|
|   Major Shear (Z)   |   1 |  0.00 |  -0.500 |    9.682 |0.052|Sec. G2
|
|   Minor Shear (Y)   |   1 |  0.00 |   0.500 |    8.873 |0.056|Sec. G2
|
| Interaction         |   1 | 30.00 |    -    |     -    |0.602|Eq. H1.1-2
|
|   (Bend+Ten) Int1   |   1 | 30.00 |    -    |     -    |0.569|Eq. H1.1-1
|
|   (Bend+Ten) Int2   |   1 | 30.00 |    -    |     -    |0.602|Eq. H1.1-2
|
|   (Bend+Comp)       |   1 | 30.00 |    -    |     -    |0.602|Eq. H1.2-1
|
|   (Bend+Shear) Z    |   1 | 30.00 |    -    |     -    |0.111|Sec. H2
|
|   (Bend+Shear) Y    |   1 | 30.00 |    -    |     -    |0.509|Sec. H2
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 CU Nominal and Local Axial Capacity

Determine the nominal axial capacity and local axial capacity of a 72-Inch-long member subjected to axial compression load.

Details

The member is a 6' simply-supported span subject to a 1 kip axial load. The section used is a 800CU125-68 channel without lips.

Section properties:

Depth: $D = 8.25\ in$
Width: $B = 1.25\ in$

Thickness: $t = 0.0713\ in$
Inner radius: $R = 0.1069\ in$
Area: $A = 0.748\ in^2$
Moment of inertia about major axis: $I_x = 6.0\ in^4$
Moment of inertia about minor axis: $I_y = 0.0703\ in^4$
Torsional constant: $J = 0.00127\ in^4$
Distance between centroid and shear center: $x_0 = -0.427\ in$
Radius of gyration about major axis: $r_x = \sqrt{I_x / A} = 2.832\ in.$
Radius of gyration about minor axis: $r_y = \sqrt{I_t / A} = 0.307\ in.$
Polar radius of gyration: $r_0 = \sqrt{r_x^2 + r_y^2 + x_0^2} = 2.881\ in.$
Warping constant: $C_w = 0.92\ in^6$

Material properties:

$E = 29{,}500\ ksi$
$G = 11{,}300\ ksi$
$F_y = 50\ ksi$

Validation

Section Dimension Checks

Radius to thickness ratio:

$$\frac{R}{t} = \frac{0.1069}{0.0713} = 1.499 < 10 \ \ \text{(OK)}$$

Width to thickness ratio:

$$W = (B - R - t) = 1.072 \text{ in}$$

$$\frac{W}{t} = \frac{1.072}{0.0713} = 15.03 < 60 \ \ \text{(OK)}$$

Depth to thickness ratio:

$$d = (D - 2R - 2t) = 7.849 \text{ in}$$

$$\frac{d}{t} = \frac{7.849}{0.0713} = 110.7 < 200 \ \ \text{(OK)}$$

Nominal Strength of Global Buckling

From Cl. E.2 of AISI S100-16:

$$\frac{K_y \times L_y}{r_y} = \frac{1 \times 72}{0.307} = 234.9$$

$$\sigma_x = \frac{\pi^2 \times E}{\left(\frac{K_x \times L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29000}{\left(1 \times 72 / 2.832\right)^2} = 442.9 \text{ ksi}$$

$$F_{crey} = \frac{\pi^2 \times E}{\left(\frac{K_y \times L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29000}{(234.9)^2} = 5.189 \text{ ksi}$$

$$\sigma_t = \frac{1}{A \times r_0^2} \times \left[G \times J + \frac{\pi^2 \times E \times C_w}{(Kt \times Lt)^2} \right] = \frac{1}{0.748(2.881)^2} \times \left[11{,}300 \times 0.001 + \frac{\pi^2 \times 29{,}000 \times 0.92}{(1 \times 72)^2} \right] = 10.50 \text{ ksi}$$

$$\beta = 1 - \frac{x_0}{r_0}^2 = 1 - \left(\frac{-0.427}{2.881} \right)^2 = 0.978$$

$$F_{(cre)TF} = \frac{1}{2 \times \beta} \times \left[\sigma_{ex} + \sigma_t - \sqrt{(\sigma_{ex} + \sigma_t)^2 - 4 \times \beta \times \sigma_{ex} \times \sigma_t} \right)$$

$$= \frac{1}{2 \times 0.978} \times \left[442.9 + 10.50 - \sqrt{(442.9 + 10.50)^2 - 4 \times 0.976 \times 442.9 \times 10.50} \right) = 10.49 \text{ ksi}$$

Since: $F_{(cre)TF} > F_{crey}$, $F_{(cre)} = 5.189\ ksi$

$$\lambda_c = \sqrt{\frac{F_y}{F_{(cre)}}} = 3.104 > 1.5$$

$$F_n = \frac{0.877}{\lambda_c^2} \times F_y = \frac{0.877}{(3.104)^2} \times 50 = 4.551 \text{ ksi}$$

$$P_{ne} = A \times F_n = 0.748 \times 4.551 = 3.404 \text{ kips}$$

Allowable nominal axial strength (ASD):

$$\frac{P_{ne}}{\Omega_c} = \frac{3.404}{1.80} = 1.891 \text{ kips}$$

Design nominal axial strength (LRFD):

$$P_{ne} \times \phi_c = 3.404 \times 0.85 = 2.893 \text{ kips}$$

<u>Local Buckling Interacting with Lateral-Torsional Buckling</u>

From Cl. E.3.1 of AISI S100-16:

Flange elements:

$$s = 1.28 \times \sqrt{\frac{E}{F_n}} = 1.28 \times \sqrt{\frac{29{,}000}{4.551}} = 102.2$$ (Eq 1.3-7, Appendix 1 of AISI S100-16)

$$\frac{W}{t} = 15.03 < 0.328 \times s = 33.51$$

Therefore, calculation of the effective width is not required.

Web element:

$$\frac{d}{t} = 110.7 < 500 \text{ (OK)}$$

$$k = 4.0$$

$$F_{cr} = \frac{k \times \pi^2 \times E}{12 \times \left(1 - \mu^2\right)} \times \left(\frac{t}{d}\right)^2 = \frac{4 \times \pi^2 \times 29{,}000}{12 \times \left(1 - 0.3^2\right)} \times \left(\frac{0.0713}{7.849}\right)^2 = 8.553 \text{ ksi}$$

$$\lambda = \sqrt{\frac{F_n}{F_{cr}}} = \sqrt{\frac{4.551}{8.553}} = 0.729$$

$$\rho = \frac{1 - \frac{0.22}{\lambda}}{\lambda} = 0.958$$

$$D_e = \rho \times d = 0.958 \times 7.894 = 7.562 \text{ in}$$

Effective area, $A_e = A - (d - D_e) \times t = 0.748 - (7.894 - 7.562) \times 0.0713 = 0.724\ in^2$

$$P_{nl} = A_e \times F_n = 0.724 \times 4.551 = 3.295 \text{ kips}$$

Allowable local axial strength (ASD):

$$\frac{P_{nl}}{\Omega_b} = \frac{3.295}{1.80} = 1.830 \text{ kips}$$

Design local axial strength (LRFD):

$$P_{nl} \times \phi_b = 3.295 \times 0.85 = 2.801 \text{ kips}$$

Results

Table 829: AISI 2016 Cylindrical Tubular Section comparison

Result Type	Reference	STAAD.Pro	Difference	Comments
Allowable nominal axial strength, ASD P_{ne}/Ω_c (kips)	1.891	1.891	none	
Allowable local axial strength, ASD P_{nl}/Ω_c (kips)	2.893	2.893	none	
Design nominal axial strength, LRFD $\phi_b P_n$ (kips)	1.830	1.830	none	
Design local axial strength, LRFD $\phi_b P_{nl}$ (kips)	2.801	2.800	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 CU Nominal and Local axial Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 02-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 6 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 800CU125-68
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GX -1 0
1 CON GX 1 72
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
FYLD 50 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
FYLD 50 ALL
METHOD ASD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
      STAAD SPACE                                              -- PAGE NO.
3
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  800CU125-68     LEN:   72.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.357            REF: Sec. E3       LC:      1
|
|------------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. E3             LC:           1
```

```
|
| Fx:(C)       1.000      Fy:         0.000          Fz:       0.000
|
| Mx:          0.000      My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
| Ag:        0.74800      Az:     0.15284            Ay:     0.56281
|
| Cz:        0.00000      Cy:     0.00000
|
| Iz:        6.00000      Iy:     0.07030             J:     0.00127
|
| Sz:        1.45455      Sy:     0.06552
|
| Rz:        2.83221      Ry:     0.30657            Cw:     0.92000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
| Fy:    50.000       Fu:    58.000      E:  29000.000       G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
| Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
| Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
| LC :     1   Actual :  234.858   Allowable :  200.000   Ratio :   1.174
|
|-----------------------------------------------------------------------------
|
| Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
| Radius To Thickness Ratio    |      1.499   |     10.000   |      PASS
|
| Width To Thickness Ratio     |     15.032   |     60.000   |      PASS
|
| Web Depth To Thickness Ratio |    110.710   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
    STAAD SPACE                                                -- PAGE NO.
```

```
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|------------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   32.538 |  -  |Sec. D3
|
|   Yielding         |   -   |   -    |     -    |   33.660 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   32.538 |  -  |Sec. D3
|
| Compression        |     1 |   0.00 |   -1.000 |    2.800 |0.357|Sec. E3
|
|   Local Comp       |     1 |   0.00 |   -1.000 |    2.800 |0.357|Sec. E3
|
|   Axial Comp       |     1 |   0.00 |   -1.000 |    2.893 |0.346|Sec. E2
|
| Bending            |   -   |   -    |     -    |    2.913 |  -  |Sec. F3
|
|   Major Local Bend |   -   |   -    |     -    |   14.316 |  -  |Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |    2.913 |  -  |Sec. F3
|
|   Major Sectional  |   -   |   -    |     -    |   65.455 |  -  |Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |    2.948 |  -  |Sec. F2
|
|   LTB Major        |   -   |   -    |     -    |   14.311 |  -  |Sec. F2
|
| Shear              |   -   |   -    |     -    |    4.356 |  -  |Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |    4.356 |  -  |Sec. G2
|
|   Minor Shear (Y)  |   -   |   -    |     -    |    6.107 |  -  |Sec. G2
|
| Interaction        |     1 |   0.00 |     -    |     -    |0.357|Eq. H1.2-1
|
|   (Bend+Comp)      |     1 |   0.00 |     -    |     -    |0.357|Eq. H1.2-1
|
|------------------------------------------------------------------------------
|
    40. PARAMETER 2
    41. CODE AISI 2016
    42. FYLD 50 ALL
    43. METHOD ASD
    44. TRACK 2 ALL
    45. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
```

```
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  800CU125-68     LEN:   72.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.546             REF: Sec. E3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. E3              LC:           1
|
|  Fx:(C)        1.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:          0.74800       Az:     0.15284           Ay:     0.56281
|
|  Cz:          0.00000       Cy:     0.00000
|
|  Iz:          6.00000       Iy:     0.07030            J:     0.00127
|
|  Sz:          1.45455       Sy:     0.06552
|
|  Rz:          2.83221       Ry:     0.30657           Cw:     0.92000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    50.000        Fu:    58.000      E:  29000.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  234.858   Allowable :  200.000   Ratio :   1.174
|
```

```
||-----------------------------------------------------------------------------
||
||  Section Dimension Check      |    Ratio     |    Limit     |     Status
||
||-----------------------------------------------------------------------------
||
||  Radius To Thickness Ratio    |      1.499   |     10.000   |      PASS
||
||  Width To Thickness Ratio     |     15.032   |     60.000   |      PASS
||
||  Web Depth To Thickness Ratio |    110.710   |    200.000   |      PASS
||
||-----------------------------------------------------------------------------
||
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                               - MEMBER:       1 (Contd.....)
|
|
|
||-----------------------------------------------------------------------------
||
|| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
||
||--------------------|-------|--------|----------|----------|-----|-----------
||
|| Tension             |   -   |   -    |     -    |   21.692 |  -  |Sec. D3
||
||   Yielding          |   -   |   -    |     -    |   22.395 |  -  |Sec. D2
||
||   Rupture           |   -   |   -    |     -    |   21.692 |  -  |Sec. D3
||
|| Compression         |     1 |   0.00 |   -1.000 |    1.830 |0.546|Sec. E3
||
||   Local Comp        |     1 |   0.00 |   -1.000 |    1.830 |0.546|Sec. E3
||
||   Axial Comp        |     1 |   0.00 |   -1.000 |    1.891 |0.529|Sec. E2
||
|| Bending             |   -   |   -    |     -    |    1.938 |  -  |Sec. F3
||
||   Major Local Bend  |   -   |   -    |     -    |    9.525 |  -  |Sec. F3
||
||   Minor Local Bend  |   -   |   -    |     -    |    1.938 |  -  |Sec. F3
||
||   Major Sectional   |   -   |   -    |     -    |   43.549 |  -  |Sec. F2
||
||   Minor Sectional   |   -   |   -    |     -    |    1.962 |  -  |Sec. F2
||
||   LTB Major         |   -   |   -    |     -    |    9.522 |  -  |Sec. F2
||
|| Shear               |   -   |   -    |     -    |    2.866 |  -  |Sec. G2
||
||   Major Shear (Z)   |   -   |   -    |     -    |    2.866 |  -  |Sec. G2
||
||   Minor Shear (Y)   |   -   |   -    |     -    |    4.018 |  -  |Sec. G2
```

```
|
| Interaction         |     1 |   0.00 |      -     |      -     |0.546|Eq. H1.2-1
|
|   (Bend+Comp)       |     1 |   0.00 |      -     |      -     |0.546|Eq. H1.2-1
|
|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.E.3 Design Procedure* (on page 1190)

V. AISI 2016 CU Nominal Moment Capacity

Determine the nominal moment axial capacity of a simply-supported beam subject to a concentrated load at mid-span.

Reference

Yu, Wei-Wen and Roger A. LaBoube. 2010 *Cold-Formed Steel Design, Fourth Edition*. Hoboken, NJ: Wiley. Example 4.3.

Related Links

- *D1.E.3 Design Procedure* (on page 1190)

Details

The member is a 10' simply-supported span subject to a 1 kip load applied at midspan. The section used is a CU_8X2_0.06 channel without lips. The member is laterally supported at the one-fourth points along the span.

Section properties:

Depth: $D = 8\ in$
Width: $B = 2\ in$
Thickness: $t = 0.06\ in$
Inner radius: $R = 3/32\ in$
Area: $A = 0.706\ in^2$
Moment of inertia about major axis: $I_x = 6.129\ in^4$
Moment of inertia about minor axis: $I_y = 0.229\ in^4$
Major axis section modulus: $S_x = 1.532\ in^3$
Minor axis section modulus: $S_y = 0.137\ in^3$
Torsional constant: $J = 0.00085\ in^4$
Distance between centroid and shear center: $x_0 = -0.919\ in$
Radius of gyration about major axis: $r_x = \sqrt{I_x / A} = 2.946\ in.$
Radius of gyration about minor axis: $r_y = \sqrt{I_t / A} = 0.569\ in.$
Polar radius of gyration: $r_0 = \sqrt{r_x^2 + r_y^2 + x_0^2} = 3.138\ in.$
Warping constant: $C_w = 2.568\ in^6$

Material properties:

$E = 29{,}500\ ksi$

G = 11,300 ksi
F_y *= 33 ksi*
F_u *= 58 ksi*

Validation

Elastic Critical Lateral-Torsional Stress Per Cl. F2.1.1 of AISI S100-16:

$$\sigma_{ey} = \frac{\pi^2 \times E}{\left(\frac{K_y \times L_y}{r_y}\right)^2}$$

where

K_y	=	Effective length factor for bending about y axis = *1*
L_y	=	Unbraced length of member for bending about y axis = *30 in*

So, $\sigma_{ey} = \frac{\pi^2 \times 29{,}000}{\left(\frac{1 \times 30}{0.569}\right)^2} = 103.1\ ksi$

$$C_b = \frac{12.5 \times M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C}$$

where

M_{max}	=	Absolute value of maximum moment in unbraced segment = *30 kip·in*
M_A	=	Absolute value of moment at quarter point of unbraced segment = *18.75 kip·in*
M_B	=	Absolute value of moment at mid-point of unbraced segment = *22.50 kip·in*
M_C	=	Absolute value of moment at three-quarter point of unbraced segment = *26.25 kip·in*

$$C_b = \frac{12.5 \times 30}{2.5 \times 30 + 3 \times 18.75 + 4 \times 22.5 + 3 \times 26.25} = 1.25 \quad \text{(Eq. F2.1.1-2)}$$

$$\sigma_t = \frac{1}{A \times r_0^2} \times \left[G \times J + \frac{\pi^2 \times E \times C_w}{(Kt \times Lt)^2}\right] = \frac{1}{0.706 \times 3.138^2} \times \left[11{,}300 \times 0.00085 + \frac{\pi^2 \times 29{,}000 \times 2.658}{(1 \times 30)^2}\right] = 12$$

The elastic buckling stress:

$$F_{cre} = C_b \times r_0 \times \frac{A}{S_x} \times \sqrt{\sigma_{ey} \times \sigma_t} = 1.25 \times 3.138 \times \frac{0.706}{1.532} \times \sqrt{103.1 \times 122.9} = 203.5\text{ ksi} \quad \text{(Eq. F2.1.1-1)}$$

Calculate the nominal flexural strength, F_n, per Cl. F2.1 of S100-16:

Since $F_{cre} > 2.78 \times F_y = 2.78 \times 33 = 97.74\ ksi$, $F_n = F_y = 33\ ksi$

The nominal flexural strength:

$$M_n = S_x \times F_y = 1.532 \times 33 = 50.57\text{ kip·in} \quad \text{(Eq. F2.1-1)}$$

Allowable flexural strength (ASD):

$$\frac{M_n}{\Omega_b} = \frac{50.57}{1.67} = 30.278\text{ kip·in}$$

Design flexural strength (LRFD):

$$M_n \times \phi_b = 50.57 \ \times 0.9 \ = 45.508 \text{ kip} \cdot \text{in}$$

Results

Table 830: AISI 2016 Cylindrical Tubular Section comparison

Result Type	Reference	STAAD.Pro	Difference	Comments
Flexural strength, ASD $^{M_{ne}}/_{\Omega_b}$ (in·kips)	30.278	30.278	none	
Flexural strength, LRFD $\phi_b M_{ne}$ (in·kips)	45.508	45.508	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 CU Nominal Moment Capacity.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 01-Jun-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
CHANNEL
CU_8X2_0.06
0.706 8 0.06 2 0.06 6.129 0.229 0.00085 0.33 0.462 0.1143
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 UPTABLE 1 CU_8X2_0.06
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
```

```
1 2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
FYLD 33 ALL
METHOD LRFD
TRACK 2 ALL
LY 30 ALL
UNB 30 ALL
UNT 30 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
FYLD 33 ALL
METHOD ASD
TRACK 2 ALL
LY 30 ALL
UNB 30 ALL
UNT 30 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:UPT CU_8X2_0.06     LEN:  120.000     LOC:
60.000 |
| STATUS: PASS       RATIO:  0.895             REF: Sec. F3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Sec. F3             LC:           1
|
|  Fx:(C)        0.000       Fy:         0.500          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -30.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.70600       Az:     0.11430            Ay:     0.46200
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         6.12900       Iy:     0.22900             J:     0.00085
|
|  Sz:         1.53225       Sy:     0.13713
```

```
|
|  Rz:         2.94641       Ry:      0.56953           Cw:      2.65754
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    33.000        Fu:    58.000      E:  29000.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:    30.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  210.701   Allowable :  300.000   Ratio :   0.702
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Width To Thickness Ratio     |     32.333   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    131.333   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |   -   |   -    |     -    |   20.968 |  -  |Sec. D2
|
|   Yielding          |   -   |   -    |     -    |   20.968 |  -  |Sec. D2
|
```

```
|   Rupture           |   -   |    -    |     -     |   30.711 |  -  |Sec. D3
|
| Compression         |   -   |    -    |     -     |    3.623 |  -  |Sec. E3
|
|   Local Comp        |   -   |    -    |     -     |    3.623 |  -  |Sec. E3
|
|   Axial Comp        |   -   |    -    |     -     |    4.702 |  -  |Sec. E2
|
| Bending             |     1 |  60.00 |  -30.000 |   33.522 |0.895|Sec. F3
|
|   Major Local Bend |     1 |  60.00 |  -30.000 |   33.522 |0.895|Sec. F3
|
|   Minor Local Bend |   -   |    -    |     -     |    3.998 |  -  |Sec. F3
|
|   Major Sectional  |     1 |  60.00 |  -30.000 |   45.508 |0.659|Sec. F2
|
|   Minor Sectional  |   -   |    -    |     -     |    4.073 |  -  |Sec. F2
|
|   LTB Major         |     1 |  60.00 |  -30.000 |   42.559 |0.705|Sec. F2
|
| Shear               |     1 |   0.00 |    0.500 |    3.562 |0.140|Sec. G2
|
|   Major Shear (Z)  |   -   |    -    |     -     |    2.150 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     1 |   0.00 |    0.500 |    3.562 |0.140|Sec. G2
|
| Interaction         |     1 |  60.00 |     -     |     -    |0.895|Eq. H1.1-2
|
|   (Bend+Ten) Int1  |     1 |  60.00 |     -     |     -    |0.659|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     1 |  60.00 |     -     |     -    |0.895|Eq. H1.1-2
|
|   (Bend+Comp)       |     1 |  60.00 |     -     |     -    |0.895|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     1 |  60.00 |     -     |     -    |0.856|Sec. H2
|
|-----------------------------------------------------------------------------
|
    48. PARAMETER 2
    49. CODE AISI 2016
    50. FYLD 33 ALL
    51. METHOD ASD
    52. TRACK 2 ALL
    53. LY 30 ALL
    54. UNB 30 ALL
    55. UNT 30 ALL
    56. CHECK CODE ALL
 STEEL DESIGN
   NOTE # 1 ~ The assigned section for the member #      1
                  may not be a cold formed section.
      STAAD SPACE                                              -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1(*)  SECTION:UPT CU_8X2_0.06     LEN:  120.000     LOC:
```

```
60.000 |
| STATUS: FAIL       RATIO:  1.345            REF: Sec. F3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Sec. F3              LC:           1
|
|  Fx:(C)        0.000       Fy:         0.500          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -30.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         0.70600       Az:     0.11430            Ay:     0.46200
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         6.12900       Iy:     0.22900             J:     0.00085
|
|  Sz:         1.53225       Sy:     0.13713
|
|  Rz:         2.94641       Ry:     0.56953            Cw:     2.65754
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    33.000        Fu:    58.000      E:  29000.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:    30.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  210.701   Allowable :  300.000   Ratio :   0.702
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Width To Thickness Ratio     |     32.333   |     60.000   |      PASS
```

```
|
|  Web Depth To Thickness Ratio |    131.333   |    200.000   |      PASS
|
|------------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|------------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand   | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |   -   |   -    |     -    |   13.951 |  -  |Sec. D2
|
|   Yielding          |   -   |   -    |     -    |   13.951 |  -  |Sec. D2
|
|   Rupture           |   -   |   -    |     -    |   20.474 |  -  |Sec. D3
|
| Compression         |   -   |   -    |     -    |    2.368 |  -  |Sec. E3
|
|   Local Comp        |   -   |   -    |     -    |    2.368 |  -  |Sec. E3
|
|   Axial Comp        |   -   |   -    |     -    |    3.073 |  -  |Sec. E2
|
| Bending             |     1 |  60.00 |  -30.000 |   22.303 |1.345|Sec. F3
|
|   Major Local Bend  |     1 |  60.00 |  -30.000 |   22.303 |1.345|Sec. F3
|
|   Minor Local Bend  |   -   |   -    |     -    |    2.660 |  -  |Sec. F3
|
|   Major Sectional   |     1 |  60.00 |  -30.000 |   30.278 |0.991|Sec. F2
|
|   Minor Sectional   |   -   |   -    |     -    |    2.710 |  -  |Sec. F2
|
|   LTB Major         |     1 |  60.00 |  -30.000 |   28.316 |1.059|Sec. F2
|
| Shear               |     1 |   0.00 |    0.500 |    2.344 |0.213|Sec. G2
|
|   Major Shear (Z)   |   -   |   -    |     -    |    1.414 |  -  |Sec. G2
|
|   Minor Shear (Y)   |     1 |   0.00 |    0.500 |    2.344 |0.213|Sec. G2
|
| Interaction         |     1 |  60.00 |     -    |     -    |1.345|Eq. H1.1-2
|
|   (Bend+Ten) Int1   |     1 |  60.00 |     -    |     -    |0.991|Eq. H1.1-1
|
|   (Bend+Ten) Int2   |     1 |  60.00 |     -    |     -    |1.345|Eq. H1.1-2
|
|   (Bend+Comp)       |     1 |  60.00 |     -    |     -    |1.345|Eq. H1.2-1
|
```

```
|   (Bend+Shear) Z    |      1 |  60.00 |      -     |      -     |1.287|Sec. H2
|
|-------------------------------------------------------------------------------
|
```

V. AISI 2016 Cylindrical Tubular Section

Determine the flexural strength of a closed cylindrical tubular member subjected to axial compression and bending moment according to AISI S100 2016 specification.

Reference

Cold-Formed Steel Design: AISI Manual. 2013 ed. Washington, D.C.: American Iron and Steel Institute, 2014. Example II-8.

Related Links

- *D1.E.3 Design Procedure* (on page 1190)

Details

The member is a 10' simply-supported span. The section is a 8" diameter closed cylindrical tube with a wall thickness of 0.1".

Axial loads:

Dead load: $P_{DL} = 7.5\ kips$
Roof live load: $P_{LR} = 20\ kips$

Transverse loads:

Wind load: $P_W = 50\ kips$ (concentrated load at mid-span)

Section properties:

Area: $A = 2.482\ in^2$
Moment of inertia: $I = 19.37\ in^4$
Section modulus: $S_f = 4.841\ in^3$
Radius of gyration: $r = \sqrt{I/A} = 2.794\ in.$

Material properties:

$E = 29{,}500\ ksi$
$G = 11{,}300\ ksi$
$F_y = 42\ ksi$
$F_u = 58\ ksi$

Validation

Design forces:

Governing load combination is DL + RLL + WL

$$Mu = 1.0\ (5\ kips)\ (120\ in)/4 = 150\ in \cdot kips$$

Section Dimension Checks

Check the outside diameter to wall thickness ratio:

$$D / t = 8.0 / 0.1 = 80$$

$$\frac{D}{t}_{Limit} \frac{0.441 \times E}{F_y} = \frac{0.441 \times 29500}{42} = 310 > 80$$

Hence, OK.

Determine Axial Capacity

Determine the governing equation for F_n:

$$\frac{0.0714 \times E}{F_y} = \frac{0.0714 \times 29500}{42} = 50.2$$

$$\frac{0.318 \times E}{F_y} = \frac{0.318 \times 29500}{42} = 223$$

Since: $\frac{0.0714 \times E}{F_y} < \frac{D}{t} < \frac{0.318 \times E}{F_y}$

$$F_n = \left[0.970 + 0.020\frac{E/(F_y)}{D/T}\right]F_y = \left[0.970 + 0.020\frac{29500/42)}{80}\right]42 = 48.1 \text{ ksi}$$

Determine Flexural Capacity

Nominal flexural strength (resistance) for yielding and global buckling:

$$M_{ne} = S_f \times F_n = 4.841 \times 48.1 = 233 \text{ in·kip}$$

Allowable flexural strength (ASD):

$$\frac{M_{ne}}{\Omega_b} = \frac{233}{1.67} = 140 \text{ in·kip}$$

Design flexural strength (LRFD):

$$M_{ne} \times \phi_b = 233 \times 0.95 = 221 \text{ in·kip}$$

Results

Table 831: AISI 2016 Cylindrical Tubular Section comparison

Result Type	Reference	STAAD.Pro	Difference	Comments
Flexural strength, ASD M_{ne}/Ω_b (in·kips)	140	139.481	negligible	
Flexural strength, LRFD $\phi_b M_{ne}$ (in·kips)	221	221.286	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 Cylindrical Tubular Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-Apr-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29500
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST PIPE OD 8 ID 7.8
CONSTANTS
MATERIAL STEEL ALL
UNIT FEET KIP
SUPPORTS
2 FIXED BUT FY MX MZ
1 FIXED BUT MX MZ
UNIT INCHES KIP
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY 7.5 0
1 CON GY -7.5 120
LOAD 2 LOADTYPE Roof Live  TITLE LOAD CASE 2
MEMBER LOAD
1 CON GY 20 0
1 CON GY -20 120
LOAD 3 LOADTYPE Wind  TITLE LOAD CASE 3
MEMBER LOAD
1 CON GX 5
LOAD COMB 4 ULC D+0.45W+0.75L
1 1.0 3 0.45 2 0.75
LOAD COMB 7 ULC, 1 DEAD + 1 ROOF LIVE
1 1.0 2 1.0
LOAD COMB 8 ULC,  DEAD + 0.6 WIND
1 1.0 3 0.6
PERFORM ANALYSIS
LOAD LIST 4 7 8
PARAMETER 1
CODE AISI 2016
METHOD LRFD
```

```
FYLD 42 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
FYLD 42 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  PIP E           LEN:  120.000     LOC:
60.000 |
| STATUS: PASS       RATIO:  0.657            REF: Eq. H1.2-1    LC:      4
|
|------------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Eq. H1.2-1           LC:           4
|
|  Fx:(C)       22.500       Fy:         1.125          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -67.500
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.48186       Az:     1.24093            Ay:     1.24093
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:        19.36467       Iy:    19.36467             J:    38.72935
|
|  Sz:         4.84117       Sy:     4.84117
|
|  Rz:         2.79330       Ry:     2.79330            Cw:     0.00000
|
|------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    42.000        Fu:    58.000      E:  29500.000       G:  11300.000
|
|------------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
```

```
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     4   Actual :   42.960   Allowable :  300.000   Ratio :   0.143
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   93.814 |  -  |Sec. D2
|
|   Yielding         |   -   |   -    |     -    |   93.814 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |  107.961 |  -  |Sec. D3
|
| Compression        |     7 |  10.00 |  -27.500 |   78.700 |0.349|Sec. E3
|
|   Local Comp       |     7 |  10.00 |  -27.500 |   78.700 |0.349|Sec. E3
|
|   Axial Comp       |     7 |  10.00 |  -27.500 |   79.259 |0.347|Sec. E2
|
| Bending            |     8 |  60.00 |  -90.000 |  181.828 |0.495|Sec. F3
|
|   Major Local Bend |     8 |  60.00 |  -90.000 |  181.828 |0.495|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |  181.828 |  -  |Sec. F3
|
|   CYL TUB Major    |     8 |  60.00 |   90.000 |  221.286 |0.407|Sec. F2.3
|
| Shear              |     8 |   0.00 |    1.500 |   29.708 |0.050|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   29.708 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     8 |   0.00 |    1.500 |   29.708 |0.050|Sec. G2
|
| Interaction        |     4 |  60.00 |     -    |     -    |0.657|Eq. H1.2-1
|
```

```
|   (Bend+Ten) Int1  |     8 |  60.00 |     -    |     -    |0.492|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     8 |  60.00 |     -    |     -    |0.495|Eq. H1.1-2
|
|   (Bend+Comp)      |     4 |  60.00 |     -    |     -    |0.657|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     8 |  60.00 |     -    |     -    |0.498|Sec. H2
|
|------------------------------------------------------------------------------
|
    56. PARAMETER 2
    57. CODE AISI 2016
    58. METHOD ASD
    59. FYLD 42 ALL
    60. TRACK 2 ALL
    61. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  PIP E           LEN:  120.000     LOC:
60.000 |
| STATUS: PASS       RATIO:  0.995             REF: Eq. H1.2-1    LC:      4
|
|------------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Eq. H1.2-1           LC:           4
|
|  Fx:(C)       22.500       Fy:         1.125          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -67.500
|
|------------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:          2.48186       Az:     1.24093             Ay:     1.24093
|
|  Cz:          0.00000       Cy:     0.00000
|
|  Iz:         19.36467       Iy:    19.36467              J:    38.72935
|
|  Sz:          4.84117       Sy:     4.84117
|
|  Rz:          2.79330       Ry:     2.79330             Cw:     0.00000
|
|------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    42.000       Fu:    58.000      E:  29500.000      G:  11300.000
|
```

```
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     4   Actual :   42.960   Allowable :  300.000   Ratio :   0.143
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   62.418 |  -  |Sec. D2
|
|   Yielding         |   -   |   -    |     -    |   62.418 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   71.974 |  -  |Sec. D3
|
| Compression        |     7 |  10.00 |  -27.500 |   51.438 |0.535|Sec. E3
|
|   Local Comp       |     7 |  10.00 |  -27.500 |   51.438 |0.535|Sec. E3
|
|   Axial Comp       |     7 |  10.00 |  -27.500 |   51.804 |0.531|Sec. E2
|
| Bending            |     8 |  60.00 |  -90.000 |  120.977 |0.744|Sec. F3
|
|   Major Local Bend |     8 |  60.00 |  -90.000 |  120.977 |0.744|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |  120.977 |  -  |Sec. F3
|
|   CYL TUB Major    |     8 |  60.00 |   90.000 |  139.481 |0.645|Sec. F2.3
|
| Shear              |     8 |   0.00 |    1.500 |   19.545 |0.077|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   19.545 |  -  |Sec. G2
```

```
|
|   Minor Shear (Y)   |     8 |   0.00 |    1.500 |   19.545 |0.077|Sec. G2
|
| Interaction         |     4 |  60.00 |     -    |     -    |0.995|Eq. H1.2-1
|
|   (Bend+Ten) Int1   |     8 |  60.00 |     -    |     -    |0.739|Eq. H1.1-1
|
|   (Bend+Ten) Int2   |     8 |  60.00 |     -    |     -    |0.744|Eq. H1.1-2
|
|   (Bend+Comp)       |     4 |  60.00 |     -    |     -    |0.995|Eq. H1.2-1
|
|   (Bend+Shear) Z    |     8 |  60.00 |     -    |     -    |0.748|Sec. H2
|
|------------------------------------------------------------------------------
|
```

V. AISI 2016 Hat Section

Determine the flexural strength of a cold-formed hat section according to the AISI S100 2016 specification.

Reference

Cold-Formed Steel Design: AISI Manual. 2013 ed. Washington, D.C.: American Iron and Steel Institute, 2014. Example II-7.

Related Links

- *D1.E.3 Design Procedure* (on page 1190)

Details

The member is 6' simply supported span. The section is a 3HU4.5X135. The top flange is in compression and is fully braced. Inelastic reserve is not considered.

Member loads:

Dead load: $w_{DL} = 120\ plf$
Live load: $w_{LL} = 700\ plf$

Section properties:

$t = 0.135\ in$ (Note that STAAD.Pro uses a thickness of 0.14")
$R = 0.1875\ in$ (Note that STAAD.Pro uses a bend radius of 0.19")

Area: $A = 1.74\ in^2$
Moment of inertia, weak axis: $I_z = 2.47\ in^4$
Moment of inertia, strong axis: $I_y = 8.29\ in^4$
Section modulus, strong axis: $S_y = 1.52\ in^3$

Material properties:

$E = 29{,}500\ ksi$
$F_y = 50\ ksi$

Validation

Design forces

$$M_y = (0.12)(6)^2/8 + (0.7)(6)^2/8 = 3.69 \text{ ft·kips} = 44.3 \text{ in·kips}$$

$$M_{uy} = 1.2[(0.12)(6)^2/8] + 1.6[(0.7)(6)^2/8] = 5.69 \text{ ft·kips} = 68.3 \text{ in·kips}$$

Bending Capacity

Calculate nominal strength:

$$M_n = S_e \times F_y = 1.52 \times 50 = 76.0 \text{ in·kips}$$

Allowable strength:

$$\frac{M_n}{\Omega_b} = \frac{76.0}{1.67} = 45.5 \text{ in·kip}$$

Ultimate strength:

$$\phi_b M_{ne} = 0.90 \times 76.0 = 68.4 \text{ in·kip}$$

Results

Table 832:

Result Type	Reference	STAAD.Pro	Difference	Comments
Flexural strength, ASD M_{ne}/Ω_b (in·kips)	45.5	45.451	negligible	
Flexural strength, LRFD $\phi_b M_{ne}$ (in·kips)	68.4	68.313	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 Hat Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Fully Braced Hat Section
JOB CLIENT BENTLEY SYSTEMS Inc.
ENGINEER DATE 04-Mar-20
JOB REV 1.0
JOB REF AISI Manual Cold-Formed Steel Design - Vol. 1
ENGINEER NAME Poulami Raha
CHECKER NAME Aksha Saha
APPROVED NAME Carlos Aguera
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 72 0 0;
```

```
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL_50_AISI
E 29500
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11153.8
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 3HU4.5x135
CONSTANTS
MATERIAL STEEL_50_AISI ALL
SUPPORTS
1 2 FIXED BUT MY MZ
UNIT FEET POUND
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
1 UNI GY -120
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
1 UNI GY -700
UNIT INCHES KIP
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD LRFD
BEAM 1 ALL
FLX 0 ALL
FU 58 ALL
FYLD 50 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
BEAM 1 ALL
FLX 0 ALL
FU 58 ALL
FYLD 50 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
```

```
|
| MEMBER:     1     SECTION:ST  3HU4.5X135      LEN:   72.000     LOC:
36.000 |
| STATUS: PASS       RATIO:  0.384             REF: Sec. F2       LC:      2
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   36.000  Criteria:   Sec. F2              LC:           2
|
|  Fx:(C)        0.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -37.800
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         1.74000       Az:     1.04085            Ay:     0.31793
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         8.28000       Iy:     2.47000             J:     0.01050
|
|  Sz:         2.19000       Sy:     1.51609
|
|  Rz:         2.18143       Ry:     1.19144            Cw:     5.66000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    50.000        Fu:    58.000      E:  29500.000      G:  11153.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :   60.431   Allowable :  300.000   Ratio :   0.201
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
```

```
|
|  Radius To Thickness Ratio    |      1.389   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     28.556   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |     17.444   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand   | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   75.690 |  -  |Sec. D3
|
|   Yielding         |   -   |   -    |     -    |   78.300 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   75.690 |  -  |Sec. D3
|
| Compression        |   -   |   -    |     -    |   56.800 |  -  |Sec. E3
|
|   Local Comp       |   -   |   -    |     -    |   56.800 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |     -    |   56.877 |  -  |Sec. E2
|
| Bending            |     2 |  36.00 |  -37.800 |   98.550 |0.384|Sec. F2
|
|   Major Local Bend |     2 |  36.00 |  -37.800 |   98.568 |0.383|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |   68.313 |  -  |Sec. F3
|
|   Major Sectional  |     2 |  36.00 |  -37.800 |   98.550 |0.384|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |   68.224 |  -  |Sec. F2
|
| Shear              |     2 |   0.00 |    2.100 |    9.061 |0.232|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   29.664 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     2 |   0.00 |    2.100 |    9.061 |0.232|Sec. G2
|
| Interaction        |     2 |  36.00 |     -    |     -    |0.384|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     2 |  36.00 |     -    |     -    |0.384|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |  36.00 |     -    |     -    |0.384|Eq. H1.1-2
|
```

```
|   (Bend+Comp)       |     2 |  36.00 |     -    |     -    |0.384|Eq. H1.2-1
|
|   (Bend+Shear) Z    |     2 |  36.00 |     -    |     -    |0.383|Sec. H2
|
|--------------------------------------------------------------------------------
|
    54. PARAMETER 2
    55. CODE AISI 2016
    56. METHOD ASD
    57. BEAM 1 ALL
    58. FLX 0 ALL
    59. FU 58 ALL
    60. FYLD 50 ALL
    61. TSA 0 ALL
    62. TRACK 2 ALL
    63. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|--------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  3HU4.5X135      LEN:   72.000     LOC:
36.000 |
| STATUS: PASS       RATIO:  0.576             REF: Sec. F2       LC:      2
|
|--------------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   36.000  Criteria:   Sec. F2              LC:           2
|
|  Fx:(C)        0.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -37.800
|
|--------------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         1.74000       Az:     1.04085            Ay:     0.31793
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         8.28000       Iy:     2.47000             J:     0.01050
|
|  Sz:         2.19000       Sy:     1.51609
|
|  Rz:         2.18143       Ry:     1.19144            Cw:     5.66000
|
|--------------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    50.000       Fu:    58.000      E:  29500.000      G:  11153.000
|
```

```
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    72.000   Lz:    72.000   Ly:    72.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :   60.431   Allowable :  300.000   Ratio :   0.201
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.389   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     28.556   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |     17.444   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   50.460 |  -  |Sec. D3
|
|   Yielding         |   -   |   -    |     -    |   52.096 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   50.460 |  -  |Sec. D3
|
| Compression        |   -   |   -    |     -    |   37.124 |  -  |Sec. E3
|
|   Local Comp       |   -   |   -    |     -    |   37.124 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |     -    |   37.175 |  -  |Sec. E2
```

```
|
| Bending            |     2 |  36.00 |  -37.800 |   65.569 |0.576|Sec. F2
|
|   Major Local Bend |     2 |  36.00 |  -37.800 |   65.581 |0.576|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |   45.451 |  -  |Sec. F3
|
|   Major Sectional  |     2 |  36.00 |  -37.800 |   65.569 |0.576|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |   45.392 |  -  |Sec. F2
|
| Shear              |     2 |   0.00 |    2.100 |    5.961 |0.352|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   19.516 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     2 |   0.00 |    2.100 |    5.961 |0.352|Sec. G2
|
| Interaction        |     2 |  36.00 |     -    |     -    |0.576|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     2 |  36.00 |     -    |     -    |0.576|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |  36.00 |     -    |     -    |0.576|Eq. H1.1-2
|
|   (Bend+Comp)      |     2 |  36.00 |     -    |     -    |0.576|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     2 |  36.00 |     -    |     -    |0.576|Sec. H2
|
|----------------------------------------------------------------------------
|
```

V. AISI 2016 SHS section

Determine the flexural Strength and axial strength of a square hollow section according to AISI S100 2016 Specification using both LRFD and ASD methods.

Details

The member is a simply-supported, 10 ft span. The member is subject to a transverse uniform dead load of 100 lbs/ft. Additionally, the member is subject to a 7,500 lb axial dead load and a 37,500 lb axial live load. The section used is a HSS 6x6x0.125. The beam-column is *not* subject to moment or axial loading due to lateral translation of the structure.

Section Properties

Depth of section, $d = 6\ in$
Width of section, $b = 6\ in$
Thickness, $t = 0.125\ in$
Area, $A = 2.7\ in^2$
Moment of inertia, $I_{zz} = 15.5\ in^4$
Section modulus, $Z_{zz} = 5.167\ in^3$

Radius of gyration of cross-section: $r_x = \sqrt{I_x / A} = 2.393\ in.$

Torsion constant, $J = 23.63\ in^4$

Material Properties

$E = 29{,}500\ ksi$
$G = 11{,}300\ ksi$
$F_y = 46\ ksi$
$F_u = 58\ ksi$

Design Forces

ASD: $M_z = 15\ in \cdot k, P = 45\ kip$
LRFD $M_{uz} = 1.2\ (15) = 18\ in \cdot k, P_u = 1.2\ (\ 7.5) + 1.6\ (37.5) = 69\ kip$

Validation

Axial Compressive Strength

Nominal Flexural Buckling Stress

$$\frac{K_x \times L_x}{r_x} = \frac{K_y \times L_y}{r_y} = \frac{1.0 \times 120}{2.393} = 50.15$$

$$F_{\text{cre}} = \frac{\pi^2 \times E}{(K \times L\ / r)^2} = \frac{\pi^2 \times 29{,}500}{(50.15)^2} = 115.8 \text{ ksi}$$

$$\lambda_c = \sqrt{\frac{F_y}{F_{\text{cre}}}} = \sqrt{\frac{46}{115.8}} = 0.630 < 1.5$$

$$F_n = 0.658^{\lambda_c^2} \times F_y = 0.658^{(0.630)^2} \times 46 = 38.95 \text{ ksi}$$

$$P_n = A \times F_n = 2.7 \times 38.95 = 105.2 \text{ kips}$$

Allowable nominal axial strength (ASD):

$$\frac{P_n}{\Omega_c} = \frac{105.2}{1.80} = 58.43 \text{ kips}$$

Design nominal axial strength (LRFD):

$$\phi_c \times P_n = 0.85 \times 105.2 = 89.40 \text{ kips}$$

Nominal Sectional Flexural Strength

According to Eq. F2.1.4-1 in the specification, the minimum unbraced length of a closed box member subject to lateral-torsional buckling, L_u , is calculated as:

$$L_u = \frac{0.36 \times C_b \times \pi}{F_y \times S_f}\sqrt{EGJI_y} = 138 \text{ ft}$$

Since $L_u > 10\ ft$, global buckling does not need to be considered and $F_n = F_y = 46\ ksi$

$$M_y = S_f \times F_n = 5.167 \times 46 = 237.7 \text{ ksi}$$

Allowable nominal axial strength (ASD):

$$\frac{M_y}{\Omega_b} = \frac{237.7}{1.67} = 142.3 \text{ in} \cdot \text{kips}$$

Design nominal axial strength (LRFD):

$$\phi_b \times M_y = 0.90 \times 237.79 = 213.9 \text{ in} \cdot \text{kips}$$

Governing strength ratio:

LRFD: 18 / 213.9 = 0.084

ASD: 15 / 142.3 = 0.105

Table 833: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Design local axial strength, LRFD $\phi_c P_n$ (kips)	89.40	89.434	negligible	
Allowable local axial strength, ASD P_n / Ω_c (kips)	58.43	58.454	negligible	
Design section flexural strength, LRFD $\phi_b M_y$ (in·kips)	213.9	213.91	negligible	
Allowable section flexural strength, ASD M_y / Ω_b (in·kips)	142.3	142.315	negligible	
Strength ratio in major sectional bending (LRFD)	0.084	0.084	none	
Strength ratio in major sectional bending (ASD)	0.105	0.105	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 SHS section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 31-Mar-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
```

```
1 0 0 0; 2 0 10 0;
MEMBER INCIDENCES
1 1 2;
START USER TABLE
TABLE 1
UNIT INCHES KIP
TUBE
HS6X6X18
2.913 6 6 0.125 16.688 16.688 25.3 4.413 4.413
END
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29500
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST HSST6X6X0.125
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FY MX MZ
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
1 CON GY -7.5 0
1 CON GY 7.5 120
1 UNI GX 0.00833
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
1 CON GY -37.5 0
1 CON GY 37.5 120
LOAD COMB 3 LRFD 1.2 D+1.6 L
1 1.2 2 1.6
LOAD COMB 4 ASD D + L
1 1.0 2 1.0
PERFORM ANALYSIS
LOAD LIST 3
PARAMETER 1
CODE AISI 2016
METHOD LRFD
TRACK 2 ALL
FYLD 46 ALL
FU 58 ALL
CHECK CODE ALL
LOAD LIST 4
PARAMETER 1
CODE AISI 2016
METHOD ASD
TRACK 2 ALL
FYLD 46 ALL
FU 58 ALL
```

```
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  HSST6X6X0.125   LEN:  120.000     LOC:
60.000 |
| STATUS: PASS       RATIO:  0.701             REF: Eq. H1.1-1    LC:      3
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Eq. H1.1-1           LC:           3
|
|  Fx:(T)      -69.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:     -17.993
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.70000       Az:     1.33818            Ay:     1.33818
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:        15.50000       Iy:    15.50000             J:    23.63067
|
|  Sz:         5.16667       Sy:     5.16667
|
|  Rz:         2.39598       Ry:     2.39598            Cw:     0.00000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    46.000        Fu:    58.000      E:  29500.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
```

```
|
|--------------------------------------------------------------------------
|
|  LC :     3   Actual :   50.084   Allowable :  200.000   Ratio :   0.250
|
|--------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|--------------------------------------------------------------------------
|
|  Width To Thickness Ratio     |     49.724   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |     49.724   |    200.000   |      PASS
|
|--------------------------------------------------------------------------
|
      STAAD SPACE                                             -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|--------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |     3 |  10.00 |  -69.000 |  111.780 |0.617|Sec. D2
|
|   Yielding         |     3 |  10.00 |  -69.000 |  111.780 |0.617|Sec. D2
|
|   Rupture          |     3 |  10.00 |  -69.000 |  117.450 |0.587|Sec. D3
|
| Compression        |   -   |   -    |     -    |   73.089 |  -  |Sec. E3
|
|   Local Comp       |   -   |   -    |     -    |   73.089 |  -  |Sec. E3
|
|   Axial Comp       |   -   |   -    |     -    |   89.434 |  -  |Sec. E2
|
| Bending            |     3 |  60.00 |  -17.993 |  185.281 |0.097|Sec. F3
|
|   Major Local Bend |     3 |  60.00 |  -17.993 |  185.281 |0.097|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |  185.281 |  -  |Sec. F3
|
|   Major Sectional  |     3 |  60.00 |  -17.993 |  213.900 |0.084|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |  213.900 |  -  |Sec. F2
|
| Shear              |     3 | 120.00 |   -0.600 |   35.087 |0.017|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   35.087 |  -  |Sec. G2
|
```

```
|   Minor Shear (Y)  |     3 | 120.00 |   -0.600 |   35.087 |0.017|Sec. G2
|
| Interaction        |     3 |  60.00 |     -    |     -    |0.701|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     3 |  60.00 |     -    |     -    |0.701|Eq. H1.1-1
|
|   (Bend+Comp)      |     3 |  60.00 |     -    |     -    |0.097|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     3 |  60.00 |     -    |     -    |0.097|Sec. H2
|
|-----------------------------------------------------------------------------
|
    58. LOAD LIST 4
    59. PARAMETER 1
    60. CODE AISI 2016
    61. METHOD ASD
    62. TRACK 2 ALL
    63. FYLD 46 ALL
    64. FU 58 ALL
    65. CHECK CODE ALL
 STEEL DESIGN
      STAAD SPACE                                                -- PAGE NO.
5
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  HSST6X6X0.125   LEN:  120.000     LOC:
60.000 |
| STATUS: PASS       RATIO:  0.710             REF: Eq. H1.1-1    LC:      4
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   60.000  Criteria:   Eq. H1.1-1           LC:           4
|
|  Fx:(T)      -45.000       Fy:          0.000          Fz:       0.000
|
|  Mx:           0.000       My:          0.000          Mz:     -14.994
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.70000       Az:     1.33818             Ay:     1.33818
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:        15.50000       Iy:    15.50000              J:    23.63067
|
|  Sz:         5.16667       Sy:     5.16667
|
|  Rz:         2.39598       Ry:     2.39598             Cw:     0.00000
|
|-----------------------------------------------------------------------------
|
```

```
| MATERIAL INFO:
|
|  Fy:    46.000        Fu:    58.000      E:  29500.000       G:  11300.000
|
|---------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   120.000   Lz:   120.000   Ly:   120.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|---------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|---------------------------------------------------------------------------
|
|  LC :     4   Actual :   50.084   Allowable :  200.000   Ratio :   0.250
|
|---------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|---------------------------------------------------------------------------
|
|  Width To Thickness Ratio     |     49.724   |    500.000   |      PASS
|
|  Web Depth To Thickness Ratio |     49.724   |    200.000   |      PASS
|
|---------------------------------------------------------------------------
|
     STAAD SPACE                                                -- PAGE NO.
6
           STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|---------------------------------------------------------------------------
|
| Limit States         | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension              |     4 |  10.00 |  -45.000 |   74.371 |0.605|Sec. D2
|
|   Yielding           |     4 |  10.00 |  -45.000 |   74.371 |0.605|Sec. D2
|
|   Rupture            |     4 |  10.00 |  -45.000 |   78.300 |0.575|Sec. D3
|
| Compression          |   -   |   -    |     -    |   47.770 |  -  |Sec. E3
|
|   Local Comp         |   -   |   -    |     -    |   47.770 |  -  |Sec. E3
```

```
|
|   Axial Comp       |   -   |   -    |     -    |   58.454 |  -  |Sec. E2
|
| Bending            |     4 |  60.00 |  -14.994 |  123.274 |0.122|Sec. F3
|
|   Major Local Bend |     4 |  60.00 |  -14.994 |  123.274 |0.122|Sec. F3
|
|   Minor Local Bend |   -   |   -    |     -    |  123.274 |  -  |Sec. F3
|
|   Major Sectional  |     4 |  60.00 |  -14.994 |  142.315 |0.105|Sec. F2
|
|   Minor Sectional  |   -   |   -    |     -    |  142.315 |  -  |Sec. F2
|
| Shear              |     4 | 120.00 |   -0.500 |   23.084 |0.022|Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |     -    |   23.084 |  -  |Sec. G2
|
|   Minor Shear (Y)  |     4 | 120.00 |   -0.500 |   23.084 |0.022|Sec. G2
|
| Interaction        |     4 |  60.00 |     -    |     -    |0.710|Eq. H1.1-1
|
|   (Bend+Ten) Int1  |     4 |  60.00 |     -    |     -    |0.710|Eq. H1.1-1
|
|   (Bend+Comp)      |     4 |  60.00 |     -    |     -    |0.122|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     4 |  60.00 |     -    |     -    |0.122|Sec. H2
|
|----------------------------------------------------------------------------
|
```

V. AISI 2016 Zee Section

Determine the axial strength and flexural strength of a cold-formed zee section without lips according to AISI S100 2016 Specification using the LRFD method.

Details

The member is a simply-supported, 2m span. The member is subject to a uniform load of 60 lbs/ft. The section used is an 8ZU 12.5x105.

Section Properties

Depth of section, $d = 8\ in$
Width of section, $b = 1.25\ in$
Thickness, $t = 0.105\ in$
Fillet radius, $r = 0.188\ in$
Area, $A = 1.059\ in^2$
Moment of inertia major axis, $I_{zz} = 7.88\ in^4$
Moment of inertia minor axis, $I_{yy} = 0.120\ in^4$
Section modulus major axis, $Z_{zz} = 1.97\ in^3$
Section modulus minor axis, $Z_{yy} = 0.096\ in^3$
Torsion constant, $J = 0.004\ in^4$
Warping constant, $C_w = 1.546\ in^6$

Radius of gyration about major axis: $r_x = \sqrt{I_x / A} = 2.728\ in.$

Radius of gyration about minor axis: $r_y = \sqrt{I_t / A} = 0.337\ in.$

Polar radius of gyration: $r_0 = \sqrt{r_x^2 + r_y^2 + x_0^2} = 2.749\ in.$

Material Properties

$E = 29{,}000\ ksi$
$G = 11{,}300\ ksi$
$F_y = 36\ ksi$

Design Forces

$M_z = 3.975\ ft \cdot k$
$V_y = 0.197\ kips$

Validation

Section Dimension Checks

Radius to thickness ratio:

$$\frac{R}{t} = \frac{0.188}{0.105} = 1.79 < 10 \ \ \text{(OK)}$$

Width to thickness ratio:

$$W = (B - R - t) = 0.957 \text{ in}$$

$$\frac{W}{t} = \frac{0.957}{0.105} = 9.11 < 60 \ \ \text{(OK)}$$

Depth to thickness ratio:

$$d = (D - 2R - 2t) = 7.414 \text{ in}$$

$$\frac{d}{t} = \frac{7.414}{0.105} = 70.60 < 200 \ \ \text{(OK)}$$

Calculate Axial Strength

Along Z axis:

$$\frac{K_x \times L_x}{r_x} = \frac{1 \times 78.74}{2.728} = 28.86$$

$$\sigma_x = \frac{\pi^2 \times E}{\left(\frac{K_x \times L_x}{r_x}\right)^2} = \frac{\pi^2 \times 29{,}000}{(28.86)^2} = 343.6 \text{ ksi}$$

$$\sigma_t = \frac{1}{A \times r_0^2} \times \left[G \times J + \frac{\pi^2 \times E \times C_w}{(Kt \times Lt)^2}\right] = \frac{1}{1.059(2.798)^2} \times \left[11{,}200 \times 0.004 + \frac{\pi^2 \times 29{,}000 \times 1.546}{(1 \times 78.74)^2}\right] = 14.37 \text{ ksi}$$

F^{crex} is the lesser of σ_x and σ_t ; $F^{crex} = 14.37\ ksi$

$$\lambda_{cz} = \sqrt{\frac{F_y}{F_{(crex)}}} = 1.583 > 1.5$$

$$F_{\text{nz}} = \frac{0.877}{\lambda_{\text{cz}}{}^2} \times F_y = \frac{0.877}{(1.583)^2} \times 36 = 12.60 \text{ ksi}$$

$$P_{nz} = A \times F_{\text{nz}} = 1.059 \times 12.60 = 13.34 \text{ kips}$$

Along Y axis:

$$\frac{K_y \times L_y}{r_y} = \frac{1 \times 78.74}{0.337} = 233.7$$

$$\sigma_y = \frac{\pi^2 \times E}{\left(\frac{K_y \times L_y}{r_y}\right)^2} = \frac{\pi^2 \times 29{,}000}{(233.7)^2} = 5.244 \text{ ksi}$$

F^{crey} is the lesser of σ_y and σ_t ; $F^{crey} = 5.244\ ksi$

$$\lambda_{\text{cy}} = \sqrt{\frac{F_y}{F_{(crey)}}} = 2.62 > 1.5$$

$$F_{\text{ny}} = \frac{0.877}{\lambda_{\text{cy}}{}^2} \times F_y = \frac{0.877}{(2.62)^2} \times 36 = 4.599 \text{ ksi}$$

$$P_{ny} = A \times F_{\text{nz}} = 1.059 \times 4.599 = 4.869 \text{ kips (Governs)}$$

Design nominal axial strength (LRFD):

$$P_{ny} \times \phi_c = 4.869 \times 0.85 = 4.140 \text{ kips}$$

Calculate Bending Strength

Major axis nominal moment capacity:

$$\mathrm{M_z} = \mathrm{S_z} \times \mathrm{F_y} = 1.97 \times 36 = 70.92 \text{ in} \cdot \text{kips}$$

Major axis ultimate moment capacity:

$$\phi_b M_z = 0.90 \times 70.92 = 63.83 \text{ in} \cdot \text{kip}$$

Minor axis nominal moment capacity:

$$\mathrm{M_y} = \mathrm{S_y} \times \mathrm{F_y} = 0.096 \times 36 = 3.456 \text{ in} \cdot \text{kips}$$

Minor axis ultimate moment capacity:

$$\phi_b M_y = 0.90 \times 3.456 = 3.110 \text{ in} \cdot \text{kip}$$

Lateral-Torsional Buckling Strength

$$\sigma_{\text{ey}} = \sigma_y = 5.244 \text{ ksi}$$

$$C_b = \frac{12.5 M_{\text{max}}}{2.5 M_{\text{max}} + 3M_A + 4M_B + 3M_C} = 1.136$$

For singly symmetric sections bent about the axis of symmetry:

$$F_{\text{crez}} = \frac{C_b r_0 A}{2 S_f} \sqrt{\sigma_{\text{ey}} \sigma_t} = \frac{1.136(2.749)(1.059)}{2(1.97)} \sqrt{5.244 \times 14.37} = 7.283 \text{ ksi}$$

where

S_f = the elastic section modulus for the unreduced section relative to the extreme compression fiber per AISI S100 2016. Hence, $S_f = S_z$.

For $F_{cre} < 0.56F_y = 0.56 \times 36 = 20.16, F_n = F_{crez}$.

$$M_{ne} = F_n \times S_f = 7.283 \times 1.97 = 14.35 \text{ in-k}$$

Design nominal lateral-torsional buckling strength (LRFD):

$$\phi_b M_{ne} = 0.9 \times 14.35 = 12.92 \text{ in·kips}$$

Table 834: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Ultimate axial strength (kips)	4.140	4.130	negligible	
Major axis bending strength (in·kips)	63.83	63.828	negligible	
Minor axis bending strength (in·kips)	3.110	3.110	none	
Lateral-torsional buckling strength (in·kips)	12.92	12.914	negligible	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 Zee Section.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 08-Mar-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 78.74 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11200
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 8ZU1.25X105
CONSTANTS
MATERIAL STEEL ALL
```

```
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -0.004
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
AXIS 1 ALL
METHOD LRFD
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  8ZU1.25X105     LEN:   78.740     LOC:    
39.370 |
| STATUS: PASS       RATIO:  0.994             REF: Sec. F2       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:   39.370  Criteria:   Sec. F2             LC:           1
|
|  Fx:(C)        0.000       Fy:         0.000         Fz:       0.000
|
|  Mx:           0.000       My:         0.000         Mz:      -3.100
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         1.06000       Az:     0.20107           Ay:     0.77857
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:         7.88000       Iy:     0.12000            J:     0.00389
|
|  Sz:         1.97000       Sy:     0.09600
|
|  Rz:         2.72653       Ry:     0.33646           Cw:     1.55000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000      G:  11200.000
|
|-----------------------------------------------------------------------------
```

```
|
| DESIGN PROPERTIES:
|
|  Member Length:    78.740   Lz:    78.740   Ly:    78.740   Lb:      0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  234.023   Allowable :  300.000   Ratio :   0.780
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |      9.119   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |     70.619   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
          ******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |   -   |   -    |     -    |   34.344 |  -  |Sec. D2
|
|   Yielding          |   -   |   -    |     -    |   34.344 |  -  |Sec. D2
|
|   Rupture           |   -   |   -    |     -    |   46.110 |  -  |Sec. D3
|
| Compression         |   -   |   -    |     -    |    4.127 |  -  |Sec. E3
|
|   Local Comp        |   -   |   -    |     -    |    4.127 |  -  |Sec. E3
|
|   Axial Comp        |   -   |   -    |     -    |    4.130 |  -  |Sec. E2
|
```

```
| Bending              |     1 |   0.00 |   -3.091 |    3.110 |0.994|Sec. F2
|
|   Major Local Bend   |     1 |  39.37 |   -0.237 |   12.927 |0.018|Sec. F3
|
|   Minor Local Bend   |     1 |  39.37 |   -3.091 |    3.275 |0.944|Sec. F3
|
|   Major Sectional    |     1 |  39.37 |   -0.237 |   63.828 |0.004|Sec. F2
|
|   Minor Sectional    |     1 |   0.00 |   -3.091 |    3.110 |0.994|Sec. F2
|
|   LTB Major          |     1 |  39.37 |   -0.237 |   12.914 |0.018|Sec. F2
|
| Shear                |     1 |   0.00 |   -0.157 |    4.126 |0.038|Sec. G2
|
|   Major Shear (Z)    |     1 |   0.00 |   -0.157 |    4.126 |0.038|Sec. G2
|
|   Minor Shear (Y)    |     1 |   0.00 |    0.012 |   14.838 |0.001|Sec. G2
|
| Interaction          |     1 |  39.37 |     -    |     -    |0.240|Eq. H1.1-2
|
|   (Bend+Ten) Int1    |     1 |  39.37 |     -    |     -    |0.049|Eq. H1.1-1
|
|   (Bend+Ten) Int2    |     1 |  39.37 |     -    |     -    |0.240|Eq. H1.1-2
|
|   (Bend+Comp)        |     1 |  39.37 |     -    |     -    |0.240|Eq. H1.2-1
|
|   (Bend+Shear) Z     |     1 |  39.37 |     -    |     -    |0.049|Sec. H2
|
|-----------------------------------------------------------------------------
|
```

V. AISI 2016 ZS Compressive Strength

Verify the compressive strength of a cold-formed zee section with lips per the AISI S100 2016 specification.

Details

A 15' long, simply supported beam is subjected to a 10 kip axial load (compression). The section used is a 12ZS3.25x105, Grade 36 steel.

Section Properties

Taken from AISI Manual, Cold-Formed Steel Design - Vol. 1, 2017 Table I-1.

Area, $A_g = 2.09\ in^2$
Radius of gyration z axis, $r_z = 4.57\ in$
Radius of gyration y axis, $r_y = 1.49\ in$
Elastic section modulus y axis, $S_{fy} = 1.22\ in^3$
Elastic section modulus z axis, $S_{fz} = 7.29\ in^3$
Torsional constant, $J = 0.00769\ in^4$
Warping constant, $C_w = 123\ in^6$

Material Properties

$E = 29{,}000\ ksi$

G = 11,300 ksi
F_y = *36 ksi*

Validation

The elastic flexural buckling stress about the z axis:

$$F_{\text{crez1}} = \frac{\pi^2 E}{(K_z L_z / r_z)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 180 / 4.57)^2} = 184.5 \text{ ksi} \quad \text{(Eq. E2.1-1)}$$

Polar radius of gyration of cross-section about shear center:

$$r_o = \sqrt{r_z^2 + r_y^2} = 4.81 \text{ in} \quad \text{(Eq. E2.2-4)}$$

$$\sigma_t = \frac{1}{A_g r_o^2}\left[GJ + \frac{\pi^2 E C_w}{(K_t L_t)^2}\right] = \frac{1}{2.09 \times 4.81^2}\left[11{,}200 \times 0.00769 + \frac{\pi^2 \times 29{,}000 \times 123}{(1 \times 180)^2}\right] = 24.14 \text{ ksi} \quad \text{(Eq. E2.2-5)}$$

$$\sigma_{\text{ez}} = F_{\text{crez}} = 184.5 \text{ ksi} \quad \text{(Eq. E2.2-6)}$$

Per E2.3, the flexural-torsional buckling stress is the lesser of σ_{ez} and σ_t; so $F_{crez} = 24.14$ *ksi*

The compressive stress capacity is:

$$\lambda_{\text{cz}} = \sqrt{\frac{F_y}{F_{\text{cre}}}} = \sqrt{\frac{36}{24.14}} = 1.221 < 1.5 \quad \text{(Eq. E2-4)}$$

$$F_{\text{nz}} = \left(0.658^{\lambda_{\text{cz}}^2}\right) F_y = 0.658^{(1.221)^2} 36 = 19.28 \text{ ksi}$$

The elastic flexural buckling stress about the y axis:

$$F_{\text{crey1}} = \frac{\pi^2 E}{(K_y L_y / r_y)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 180 / 1.49)^2} = 19.61 \text{ ksi} \quad \text{(Eq. E2.1-1)}$$

The elastic global buckling stress about the z axis is the minimum of F_{crey1} and σ_t; so $F_{crey} = 19.61$ *ksi*.

$$\lambda_{\text{cy}} = \sqrt{\frac{F_y}{F_{\text{crey}}}} = \sqrt{\frac{36}{19.61}} = 1.355 < 1.5 \quad \text{(Eq. E2-4)}$$

$$F_{\text{ny}} = \left(0.658^{\lambda_{\text{cz}}^2}\right) F_y = 0.658^{(1.355)^2} 36 = 16.69 \text{ ksi} \quad \text{(Eq. E2-2)}$$

Global buckling about the Y axis governs.

Nominal capacity against global buckling:

$$P_n = A_g F_n = 2.09 \times 16.69 = 34.89 \text{ kips}$$

The available axial strength:

LRFD: $\phi_c P_n = 0.85 \times 34.89 = 29.66$ *kips*
ASD: P_n/Ω_c = 34.89 / 1.8 = 19.38 kips

Determine the compression capacity for local buckling interacting with yielding and global buckling as per E3:

For local buckling about the Z axis, the effective area, $A_e = 1.586\ in^2$.

$$P_{n\ell z} = A_e \times F_{nz} = 1.586 \times 19.28 = 30.56 \text{ kips}$$

For local buckling about the Y axis, the effective area, $A_e = 1.627\ in^2$.

$$P_{n\ell y} = A_e \times F_{nz} = 1.627 \times 16.69 = 27.15 \text{ kips}$$

$P_{n\ell}$ is taken as the minimum of $P_{n\ell z}$, $P_{n\ell y}$, and P_{ne}; so $P_{n\ell} = 27.15\ kips$.

LRFD: $\phi_c P_{n\ell} = 0.85 \times 27.15 = 23.08\ kips$
ASD: $P_{n\ell}/\Omega_c = 27.15 / 1.8 = 15.06$ kips

Determine the compression capacity for distortional buckling as per E4:

The elastic distortional buckling stress, $F^{crd} = 30.84\ ksi$.

The elastic distortional buckling load, $P_{crd} = A_g \times F^{crd} = 2.09 \times 30.84 = 64.46\ kips$.

$$P_y = A_g \times F^y = 2.09 \times 36 = 75.24 \text{ kips} \qquad \text{(Eq. E4.1-4)}$$

$$\lambda_d = \sqrt{\frac{P_y}{P_{crd}}} = \sqrt{\frac{75.24}{64.46}} = 1.08 > 0.561 \qquad \text{(Eq. E4.1-3)}$$

$$P_{nd} = \left[1 - 0.25\left(\frac{P_{crd}}{P_y}\right)^{0.6}\right]\left(\frac{P_{crd}}{P_y}\right)^{0.6} P_y \qquad \text{(Eq. 4.1-2)}$$

$$= \left[1 - 0.25\left(\frac{64.46}{75.24}\right)^{0.6}\right]\left(\frac{64.46}{75.24}\right)^{0.6} 75.24 = 52.95 \text{ ksi}$$

The available axial strength:

LRFD: $\phi_c P_{nd} = 0.85 \times 52.95 = 45.01\ kips$
ASD: $P_{nd}/\Omega_c = 52.95 / 1.8 = 29.42$ kips

Results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Local compression capacity (kips)	23.08	23.212	negligible	
	Distorional buckling capacity (kips)	45.01	45.006	negligible	
	Axial compression capacity (kips)	29.66	29.809	negligible	
ASD	Local compression capacity (kips)	15.06	15.171	negligible	
	Distorional buckling capacity (kips)	29.42	29.416	negligible	

Result Type		Reference	STAAD.Pro	Difference	Comments
	Axial compression capacity (kips)	19.38	19.483	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 ZS Compressive Strength.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 11-Jan-21
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 15 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
G 1.6128e+06
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 12ZS3.25X105
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FX -10
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE AISI 2016
METHOD LRFD
AXIS 1 ALL
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD ASD
```

```
AXIS 1 ALL
FYLD 36 ALL
FU 58 ALL
FLX 0 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12ZS3.25X105    LEN:  180.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.431             REF: Sec. E3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. E3              LC:           1
|
|  Fx:(C)       10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.09000       Az:     0.59243            Ay:     1.19857
|
|  Cz:         0.00000       Cy:     0.00000
|
|  Iz:        43.70000       Iy:     4.67000             J:     0.00769
|
|  Sz:         7.28333       Sy:     1.20164
|
|  Rz:         4.57265       Ry:     1.49481            Cw:   123.00000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000      G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   180.000   Lz:   180.000   Ly:   180.000   Lb:       0.000
|
| DESIGN PARAMETERS:
```

```
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  120.417   Allowable :  200.000   Ratio :   0.602
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     26.868   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                                - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension             |   -   |   -    |     -    |   67.716 |  -  |Sec. D2
|
|   Yielding          |   -   |   -    |     -    |   67.716 |  -  |Sec. D2
|
|   Rupture           |   -   |   -    |     -    |   90.915 |  -  |Sec. D3
|
| Compression         |     1 |   0.00 |  -10.000 |   23.212 |0.431|Sec. E3
|
|   Local Comp        |     1 |   0.00 |  -10.000 |   23.212 |0.431|Sec. E3
|
|   Distortional      |     1 |   0.00 |  -10.000 |   45.006 |0.222|Sec. E4
|
|   Axial Comp        |     1 |   0.00 |  -10.000 |   29.809 |0.335|Sec. E2
|
| Bending             |   -   |   -    |     -    |   38.933 |  -  |Sec. F2
|
|   Major Local Bend |   -   |   -    |     -    |   99.087 |  -  |Sec. F3
|
```

```
|   Minor Local Bend |    -   |    -    |      -    |   39.498 |  -  |Sec. F3
|
|   Distortional     |    -   |    -    |      -    |  222.225 |  -  |Sec. F4
|
|   Major Sectional  |    -   |    -    |      -    |  235.980 |  -  |Sec. F2
|
|   Minor Sectional  |    -   |    -    |      -    |   38.933 |  -  |Sec. F2
|
|   LTB Major        |    -   |    -    |      -    |   98.978 |  -  |Sec. F2
|
| Shear              |    -   |    -    |      -    |   12.157 |  -  |Sec. G2
|
|   Major Shear (Z)  |    -   |    -    |      -    |   12.157 |  -  |Sec. G2
|
|   Minor Shear (Y)  |    -   |    -    |      -    |   13.487 |  -  |Sec. G2
|
| Interaction        |      1 |   0.00 |      -    |     -    |0.431|Eq. H1.2-1
|
|   (Bend+Comp)      |      1 |   0.00 |      -    |     -    |0.431|Eq. H1.2-1
|
|-----------------------------------------------------------------------------
|
    43. PARAMETER 2
    44. CODE AISI 2016
    45. METHOD ASD
    46. AXIS 1 ALL
    47. FYLD 36 ALL
    48. FU 58 ALL
    49. FLX 0 ALL
    50. TSA 0 ALL
    51. TRACK 2 ALL
    52. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                                -- PAGE NO.
5
           STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12ZS3.25X105    LEN:  180.000     LOC:
0.000 |
| STATUS: PASS       RATIO:  0.659             REF: Sec. E3       LC:      1
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    0.000  Criteria:   Sec. E3              LC:           1
|
|  Fx:(C)       10.000       Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000       My:         0.000          Mz:       0.000
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
```

```
|  Ag:          2.09000       Az:      0.59243            Ay:      1.19857
|
|  Cz:          0.00000       Cy:      0.00000
|
|  Iz:         43.70000       Iy:      4.67000             J:      0.00769
|
|  Sz:          7.28333       Sy:      1.20164
|
|  Rz:          4.57265       Ry:      1.49481            Cw:    123.00000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    36.000        Fu:    58.000      E:  29000.000       G:  11200.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:   180.000   Lz:   180.000   Ly:   180.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     1   Actual :  120.417   Allowable :  200.000   Ratio :   0.602
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     26.868   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                              -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|
|-----------------------------------------------------------------------------
```

```
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |    -     |   45.054 |  -  |Sec. D2
|
|   Yielding         |   -   |   -    |    -     |   45.054 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |    -     |   60.610 |  -  |Sec. D3
|
| Compression        |     1 |   0.00 |  -10.000 |   15.171 |0.659|Sec. E3
|
|   Local Comp       |     1 |   0.00 |  -10.000 |   15.171 |0.659|Sec. E3
|
|   Distortional     |     1 |   0.00 |  -10.000 |   29.416 |0.340|Sec. E4
|
|   Axial Comp       |     1 |   0.00 |  -10.000 |   19.483 |0.513|Sec. E2
|
| Bending            |   -   |   -    |    -     |   25.904 |  -  |Sec. F2
|
|   Major Local Bend |   -   |   -    |    -     |   65.926 |  -  |Sec. F3
|
|   Minor Local Bend |   -   |   -    |    -     |   26.280 |  -  |Sec. F3
|
|   Distortional     |   -   |   -    |    -     |  147.854 |  -  |Sec. F4
|
|   Major Sectional  |   -   |   -    |    -     |  157.006 |  -  |Sec. F2
|
|   Minor Sectional  |   -   |   -    |    -     |   25.904 |  -  |Sec. F2
|
|   LTB Major        |   -   |   -    |    -     |   65.854 |  -  |Sec. F2
|
| Shear              |   -   |   -    |    -     |    7.998 |  -  |Sec. G2
|
|   Major Shear (Z)  |   -   |   -    |    -     |    7.998 |  -  |Sec. G2
|
|   Minor Shear (Y)  |   -   |   -    |    -     |    8.873 |  -  |Sec. G2
|
| Interaction        |     1 |   0.00 |    -     |    -     |0.659|Eq. H1.2-1
|
|   (Bend+Comp)      |     1 |   0.00 |    -     |    -     |0.659|Eq. H1.2-1
|
|---------------------------------------------------------------------------
|
```

V. AISI 2016 ZS Flexural Strength

Verify the flexural strength in yielding of a cold-formed zee section according to the AISI S100 2016 specification using both the LRFD and ASD methods.

Details

A 1.5' long, simply supported beam is subjected to a uniform load in the global Y direction. The section used is a 12ZS3.25x105, Gr. 50 steel.

Section Properties

Taken from AISI Manual, Cold-Formed Steel Design - Vol. 1, 2017 Table I-1.

Area, $A_g = 2.09\ in^2$
Radius of gyration z axis, $r_z = 4.57\ in$
Radius of gyration y axis, $r_y = 1.49\ in$
Elastic section modulus y axis, $S_{fy} = 1.22\ in^3$
Elastic section modulus z axis, $S_{fz} = 7.29\ in^3$
Torsional constant, $J = 0.00769\ in^4$
Warping constant, $C_w = 123\ in^6$

Material Properties

$E = 29{,}000\ ksi$
$G = 11{,}300\ ksi$
$F_y = 50\ ksi$

Validation

The modification factor, C_b, for a simply-supported beam under a uniformly distributed load:

$$C_b = \frac{12.5 \times M_{\mathrm{max}}}{2.5 \times M_{\mathrm{max}} + 3 \times M_{\mathrm{a}} + 4 \times M_{\mathrm{b}} + 3 \times M_{\mathrm{c}}} = 1.14 \qquad \text{(Eq. F2.1.1-2)}$$

Polar radius of gyration of cross-section about shear center:

$$r_o = \sqrt{{r_z}^2 + {r_y}^2} = 4.81 \text{ in} \qquad \text{(Eq. F2.1.1-3)}$$

Major axis bending:

$$\sigma_{\mathrm{ey}} = \frac{\pi^2 E}{(K_y L_y / r_y)^2} = \frac{\pi^2 \times 29{,}000}{(1 \times 18 / 1.49)^2} = 1{,}961 \text{ ksi} \qquad \text{(Eq. F2.1.1-4)}$$

$$\sigma_t = \frac{1}{A{r_o}^2}\left[GJ + \frac{\pi^2 E C_w}{(K_t L_t)^2}\right] = \frac{1}{2.09 \times 4.81^2}\left[11{,}200 \times 0.00769 + \frac{\pi^2 \times 29{,}000 \times 123}{(1 \times 18)^2}\right] = 2{,}249 \text{ ksi} \qquad \text{(Eq. F2.1.1-5)}$$

$$F_{\mathrm{cre}} = \frac{C_b r_o A}{2S_f}\sqrt{\sigma_{\mathrm{ey}} \times \sigma_t} = \frac{1.14 \times 4.81 \times 2.09}{2 \times 7.29}\sqrt{1{,}961 \times 2{,}249} = 1{,}651 \text{ ksi} \qquad \text{(Eq. F2.1.1-1)}$$

Per F2.1, where $F_{cre} > 2.78F_y = 2.78(50) = 139\ ksi$, $F_n = F_y$ (Eq. F2.1-3).

Nominal capacity against lateral-torsional buckling about the major axis:

$$\mathrm{M_{nez} = S_{fz}F_n} = 7.29 \times 50 = 364.5 \text{ in·kips}$$

The available flexural strength about the major axis:

LRFD: $\phi_b M_{nez} = 0.9 \times 364.5 = 328.1\ in{\cdot}kips$
ASD: M_{nez}/Ω_b = 364.5 / 1.67 = 218.3 in·kips

Minor axis bending:

$$F_{\mathrm{cre}} = \frac{C_b r_o A}{2S_f}\sqrt{\sigma_{\mathrm{ey}} \times \sigma_t} = \frac{1.14 \times 4.81 \times 2.09}{2 \times 7.29}\sqrt{1{,}961 \times 2{,}249} = 1{,}651 \text{ ksi} \qquad \text{(Eq. F2.1.1-1)}$$

Per F2.1, where $F_{cre} > 2.78F_y = 2.78(50) = 139\ ksi$, $F_{ny} = F_y$ (Eq. F2.1-3).

Elastic section modulus of full unreduced cross-section relative to extreme fiber in first yielding: $S_{fyy} = S_{fy}$.

Nominal capacity against lateral-torsional buckling about the major axis:

$$M_{ney} = S_{fy}F_n = 1.22 \times 50 = 61.0 \text{ in·kips}$$

The available flexural strength about the major axis:

LRFD: $\phi_b M_{ney} = 0.9 \times 61.0 = 54.9\ in \cdot kips$
ASD: $M_{ney}/\Omega_b = 61.0 / 1.67 = 36.5$ in·kips

Results

Result Type		Reference	STAAD.Pro	Difference	Comments
LRFD	Flexural capacity about major axis (in·kips)	328.1	327.750	negligible	
	Flexural capacity about minor axis (in·kips)	54.9	54.074	negligible	
ASD	Flexural capacity about major axis (in·kips)	218.3	218.064	negligible	
	Flexural capacity about minor axis (in·kips)	36.5	35.977	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\AISI\AISI 2016 ZS Flexural Strength.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
JOB NAME Fully Braced Hat Section
JOB CLIENT BENTLEY SYSTEMS Inc.
ENGINEER DATE 04-Mar-20
JOB REV 1.0
JOB REF AISI Manual Cold-Formed Steel Design - Vol. 1
ENGINEER NAME Abrar Ahmed
CHECKER NAME Binith Anto
APPROVED NAME Ahmed A
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 18 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL_50_AISI
E 29500
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
G 11153.8
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY COLDFORMED AMERICAN
1 TABLE ST 12ZS3.25X105
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 2 FIXED BUT MY MZ
UNIT FEET POUND
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
1 UNI GY -700
UNIT INCHES KIP
PERFORM ANALYSIS
PARAMETER 1
CODE AISI 2016
METHOD ASD
AXIS 1 ALL
FLX 0 ALL
FU 58 ALL
FYLD 50 ALL
TSA 0 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE AISI 2016
METHOD LRFD
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|--------------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12ZS3.25X105    LEN:   18.000     LOC:
9.000 |
| STATUS: PASS       RATIO:  0.067             REF: Sec. F3       LC:      2
```

```
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    9.000  Criteria:   Sec. F3             LC:            2
|
|  Fx:(C)        0.000       Fy:          0.000           Fz:       0.000
|
|  Mx:           0.000       My:          0.000           Mz:      -2.363
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:          2.09000       Az:     0.59243             Ay:     1.19857
|
|  Cz:          0.00000       Cy:     0.00000
|
|  Iz:         43.70000       Iy:     4.67000              J:     0.00769
|
|  Sz:          7.28333       Sy:     1.20164
|
|  Rz:          4.57265       Ry:     1.49481             Cw:   123.00000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    50.000        Fu:    58.000      E:  29000.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    18.000   Lz:    18.000   Ly:    18.000   Lb:       0.000
|
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     2   Actual :   12.042   Allowable :  300.000   Ratio :   0.040
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     26.868   |     60.000   |      PASS
```

```
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                  -- PAGE NO.
4
            STAAD.Pro CODE CHECKING - (AISI S100-16) [ASD ] (v1.0)
           ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States        | Load# |Location|  Demand  | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   60.610 |  -  |Sec. D3
|
|   Yielding         |   -   |   -    |     -    |   62.575 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   60.610 |  -  |Sec. D3
|
| Compression        |   -   |   -    |     -    |   33.801 |  -  |Sec. E4
|
|   Local Comp       |   -   |   -    |     -    |   33.826 |  -  |Sec. E3
|
|   Distortional     |   -   |   -    |     -    |   33.801 |  -  |Sec. E4
|
|   Axial Comp       |   -   |   -    |     -    |   57.443 |  -  |Sec. E2
|
| Bending            |     2 |   9.00 |   -2.295 |   34.319 |0.067|Sec. F3
|
|   Major Local Bend |     2 |   9.00 |   -0.562 |  202.771 |0.003|Sec. F3
|
|   Minor Local Bend |     2 |   9.00 |   -2.295 |   34.319 |0.067|Sec. F3
|
|   Distortional     |     2 |   9.00 |   -0.562 |  182.264 |0.003|Sec. F4
|
|   Major Sectional  |     2 |   9.00 |   -0.562 |  218.064 |0.003|Sec. F2
|
|   Minor Sectional  |     2 |   0.00 |   -2.295 |   35.977 |0.064|Sec. F2
|
| Shear              |     2 |   0.00 |   -0.510 |   11.108 |0.046|Sec. G2
|
|   Major Shear (Z)  |     2 |   0.00 |   -0.510 |   11.108 |0.046|Sec. G2
|
|   Minor Shear (Y)  |     2 |   0.00 |    0.125 |    8.873 |0.014|Sec. G2
|
| Interaction        |     2 |   0.00 |     -    |     -    |0.059|Sec. H2
|
|   (Bend+Ten) Int1  |     2 |   9.00 |     -    |     -    |0.011|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |   9.00 |     -    |     -    |0.013|Eq. H1.1-2
|
```

```
|   (Bend+Comp)       |     2 |   9.00 |     -    |     -    |0.013|Eq. H1.2-1
|
|   (Bend+Shear) Z    |     2 |   0.00 |     -    |     -    |0.059|Sec. H2
|
|-----------------------------------------------------------------------------
|
    59. PARAMETER 2
    60. CODE AISI 2016
    61. METHOD LRFD
    62. CHECK CODE ALL
 STEEL DESIGN
     STAAD SPACE                                                 -- PAGE NO.
5
           STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
          ********************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|-----------------------------------------------------------------------------
|
|  MEMBER:     1     SECTION:ST  12ZS3.25X105    LEN:   18.000     LOC:
9.000 |
| STATUS: PASS       RATIO:  0.044             REF: Sec. F3       LC:      2
|
|-----------------------------------------------------------------------------
|
| DESIGN FORCES:
|
|  Location:    9.000  Criteria:   Sec. F3              LC:           2
|
|  Fx:(C)        0.000      Fy:         0.000          Fz:       0.000
|
|  Mx:           0.000      My:         0.000          Mz:      -2.363
|
|-----------------------------------------------------------------------------
|
| SECTION PROPERTIES:
|
|  Ag:         2.09000      Az:     0.59243            Ay:     1.19857
|
|  Cz:         0.00000      Cy:     0.00000
|
|  Iz:        43.70000      Iy:     4.67000             J:     0.00769
|
|  Sz:         7.28333      Sy:     1.20164
|
|  Rz:         4.57265      Ry:     1.49481            Cw:   123.00000
|
|-----------------------------------------------------------------------------
|
| MATERIAL INFO:
|
|  Fy:    50.000        Fu:    58.000      E:  29000.000      G:  11300.000
|
|-----------------------------------------------------------------------------
|
| DESIGN PROPERTIES:
|
|  Member Length:    18.000   Lz:    18.000   Ly:    18.000   Lb:       0.000
|
```

```
| DESIGN PARAMETERS:
|
|  Kz:   1.000    Ky:   1.000    NSF:  1.000
|
|-----------------------------------------------------------------------------
|
| CRITICAL SLENDERNESS:
|
|-----------------------------------------------------------------------------
|
|  LC :     2   Actual :   12.042   Allowable :  300.000   Ratio :   0.040
|
|-----------------------------------------------------------------------------
|
|  Section Dimension Check      |    Ratio     |    Limit     |     Status
|
|-----------------------------------------------------------------------------
|
|  Radius To Thickness Ratio    |      1.786   |     10.000   |      PASS
|
|  Width To Thickness Ratio     |     26.868   |     60.000   |      PASS
|
|  Web Depth To Thickness Ratio |    108.714   |    200.000   |      PASS
|
|-----------------------------------------------------------------------------
|
      STAAD SPACE                                                -- PAGE NO.
6
            STAAD.Pro CODE CHECKING - (AISI S100-16) [LRFD] (v1.0)
           *******************************************************
ALL UNITS ARE IN "INCH" AND "KIP " (U.N.O.)
|                                              - MEMBER:       1 (Contd.....)
|
|
|
|-----------------------------------------------------------------------------
|
| Limit States       | Load# |Location|  Demand   | Capacity |Ratio| Reference
|
|--------------------|-------|--------|----------|----------|-----|-----------
|
| Tension            |   -   |   -    |     -    |   90.915 |  -  |Sec. D3
|
|   Yielding         |   -   |   -    |     -    |   94.050 |  -  |Sec. D2
|
|   Rupture          |   -   |   -    |     -    |   90.915 |  -  |Sec. D3
|
| Compression        |   -   |   -    |     -    |   51.715 |  -  |Sec. E4
|
|   Local Comp       |   -   |   -    |     -    |   51.753 |  -  |Sec. E3
|
|   Distortional     |   -   |   -    |     -    |   51.715 |  -  |Sec. E4
|
|   Axial Comp       |   -   |   -    |     -    |   87.888 |  -  |Sec. E2
|
| Bending            |     2 |   9.00 |   -2.295 |   51.581 |0.044|Sec. F3
|
|   Major Local Bend |     2 |   9.00 |   -0.562 |  304.765 |0.002|Sec. F3
```

```
|
|   Minor Local Bend |     2 |   9.00 |   -2.295 |   51.581 |0.044|Sec. F3
|
|   Distortional     |     2 |   9.00 |   -0.562 |  273.942 |0.002|Sec. F4
|
|   Major Sectional  |     2 |   9.00 |   -0.562 |  327.750 |0.002|Sec. F2
|
|   Minor Sectional  |     2 |   0.00 |   -2.295 |   54.074 |0.042|Sec. F2
|
| Shear              |     2 |   0.00 |   -0.510 |   16.884 |0.030|Sec. G2
|
|   Major Shear (Z)  |     2 |   0.00 |   -0.510 |   16.884 |0.030|Sec. G2
|
|   Minor Shear (Y)  |     2 |   0.00 |    0.125 |   13.487 |0.009|Sec. G2
|
| Interaction        |     2 |   0.00 |     -    |     -    |0.039|Sec. H2
|
|   (Bend+Ten) Int1  |     2 |   9.00 |     -    |     -    |0.007|Eq. H1.1-1
|
|   (Bend+Ten) Int2  |     2 |   9.00 |     -    |     -    |0.009|Eq. H1.1-2
|
|   (Bend+Comp)      |     2 |   9.00 |     -    |     -    |0.009|Eq. H1.2-1
|
|   (Bend+Shear) Z   |     2 |   0.00 |     -    |     -    |0.039|Sec. H2
|
|------------------------------------------------------------------------------
|
```

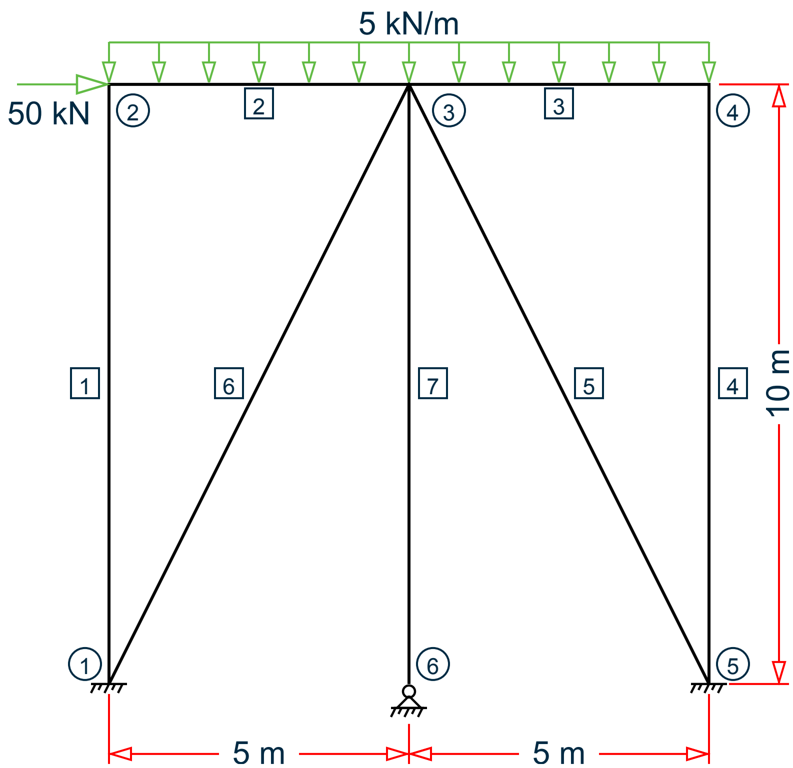
V. API

V.API K Joint

Verify the design of a simple K joint with axial load and in-plane bending.

Details

Design of node 3 in the following model.



Chord member (2 & 3): D = 500 mm, T = 20 mm (A_c = 30,159 mm^2, Z_c = 3,480,380 mm^3)

Brace member (5 & 7): d = 400 mm, t = 20 mm (A_b = 23,876 mm^2, Z_b = 2,160,790 mm^3)

Table 835: Loads

Member	Fx (kN)	MIP (kN·m)	MOP (kN·m)
Chord	49.99 (T)	17.58	0
Brace	72.51 (C)	0.727	0

Validation

Design Parameters

$$\beta = d/D = 0.8$$

So, $0.2 < \beta < 1.0$ (OK)

$$\gamma = D/2T = 12.5$$

So, $10 < \gamma < 50$ (OK)

$$\theta = \tan^{-1}(10/5) = 63.4°$$

So, $30° < \theta < 90°$ (OK)

$F_y = 500\ MPa$ (OK)

$$g = 20 \text{ mm}$$

$g/D = 20 / 500 = 0.04$

<u>Determination of Q_u</u>

First, evaluate Q_g. Since $-0.05 \le g/D \le 0.05$, use linear interpolation.

For $g/D = 0.05$: $Q_g = 1 + 0.2\times(1 - 2.8\times g/D)^3 = 1 + 0.2\times[1 - 2.8\times(0.05)]^3 = 1.127$

For $g/D = -0.05$: $Q_g = 0.13 + 0.65\times\phi\times\gamma^{0.5} = 0.13 + 0.65\times(1)\times(12.5)^{0.5} = 2.428$

For $g/D = 0.04$: $1.127 + (2.428 - 1.127)\times(0.05 - 0.04) / (0.05 - -0.05) = 1.257$

$$Q_{u\_ax} = (16 + 1.2\times\gamma)\times\beta^{1.2}\times Q_g = [16 + 1.2(12.5)](0.8)^{1.2}(1.257) = 29.815$$

$$Q_{u\_ipb} = (5 + 0.7\times\gamma)\times\beta^{1.2} = [5 + 0.7(12.5)](0.8)^{1.2} = 10.52$$

<u>Determination of Q_f</u>

From Table 4.3.2:

$C1_{ax} = 0.2$, $C2_{axi} = 0.2$, $C3_{ax} = 0.3$

$C1_b = 0.2$, $C2_b = 0$, $C3_b = 0.4$

$$P_{yc} = A_c\times F_{yc} = 15{,}079.5 \text{ kN}$$

$$M_{pc} = Z_c\times F_{yc} = 1{,}740.19 \text{ kN}\cdot\text{m}$$

$$P_c = 49.99 \text{ kN}$$

$$M_c = \sqrt{(MIPC^2+MOPC^2)} = 17.58 \text{ kN}\cdot\text{m}$$

$$A = \sqrt{\left(\frac{FS \times P_c}{P_{yc}}\right)^2 + \left(\frac{FS \times M_c}{M_{pc}}\right)^2} = \sqrt{\left(\frac{1.6 \times 49.99}{17,\ 079.5}\right)^2 + \left(\frac{1.6 \times 17.58}{1,\ 740.19}\right)^2} = 0.017$$

$$Q_{f\_\text{ax}} = 1 + C1_{ax}\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_{ax}\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_{ax}A^2$$

$$Q_{f\_\text{ax}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0.2\left(\frac{1.6 \times 17.58}{1,\ 740.19}\right) - 0.3(0.017)^2 = 0.9977$$

$$Q_{f\_\text{bend}} = 1 + C1_b\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_b\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_bA^2$$

$$Q_{f\_\text{bend}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.4(0.017)^2 = 1.001$$

<u>Axial Capacity</u>

$$P_a = Q_uQ_f\frac{F_{yc}T^2}{FS\sin\theta} = 29.815(0.9977)\frac{500(20)^2}{1.6\sin\left(63.4^\circ\right)10^3} = 4,\ 158.5 \text{ kN}$$

<u>Moment Capacity</u>

$$M_a = Q_uQ_f\frac{F_{yc}T^2d}{FS\sin\theta} = 10.52(1.001)\frac{500(20)^2 400}{1.6\sin\left(63.4^\circ\right)10^6} = 588.85 \text{ kN}\cdot\text{m}$$

Out-of-plane bending is zero so moment capacity is not calculated.

<u>Interaction Ratio</u>

$$P/P_a + (M/M_a)^2 = (72.511 / 4{,}158.5) + (0.727 / 588.85)^2 = 0.017$$

Results

Table 836: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Q_{u_ax}	29.815	29.824	negligible	
Q_{u_ipb}	10.52	10.523	negligible	
Q_{f_ax}	0.9977	0.999	negligible	
Q_{f_ipb}	1.001	1.001	none	
Axial capacity (kN)	4,158.5	4,158.06	negligible	
In-plane Bending Capacity (kN·m)	588.86	588.1	negligible	
Interaction ratio	0.017	0.017	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\API\API KJoint.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 5 10 0; 4 10 10 0; 5 10 0 0; 6 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 3 5; 6 3 1; 7 3 6;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
5 6 TABLE ST PIPE OD 0.4 ID 0.36
2 3 TABLE ST PIPE OD 0.5 ID 0.46
MEMBER PROPERTY AMERICAN
1 4 7 TABLE ST HSSP12.750X0.25
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
```

```
1 5 FIXED
6 PINNED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -50
MEMBER LOAD
2 3 UNI GY -5
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE API
*FSJ 2.6 ALL
FYLD 500000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

Note: The generated .PUN file is required to generated this output for K or X joints. Otherwise, the program assumes all joints are Y joints. Refer to D1.J.8.2 Joint File Format (on page 1295) for details.

```
======================================================================

NODE NO :      3   CHORD NO:     2   BRACE NO:
6
======================================================================

DESIGN DATA : (Units    : N ,
mm  )
  Chord Memb :      D =  500.00  T =
19.99
  Brace Memb :      d =  400.00  t =
19.99
  Angle (THETA) =  63.4 deg  GAP   =   20.00   Fyc =
499.9
  BETA          =  0.80      GAMMA =   12.51   TAU =
1.00
      ***WARNING - JNT      3:
GAP
      GAP PROVIDED = 20.00 IS LESS THAN THE NOMINAL VALUE OF 2 inches (50
mm).
      STAAD SPACE                                                -- PAGE NO.
7
JOINT CLASS :
KG

FACTORS :

--------------------------------------------------------------------
  Joint   Load   Strength     Chord Load    C1         C2
C3
  Class   Cond   factor (Qu)  factor
(Qf)
```

```
-------------------------------------------------------------------
    K     AX      29.824        0.999       0.200    0.200
0.300
    K     IPB     10.523        1.001       0.200    0.000
0.400
---------------------------------------------------------------------

CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO
STATUS
   (Cl. 4.3)             (KN, m )         (KN,
m )
---------------------------------------------------------------------

AXIAL         :             -70.33      1     4158.06    0.017
PASS
IP BENDING    :               0.76      1      588.10    0.001
PASS
INTERACTION   :                         1                0.017
PASS
---------------------------------------------------------------------

CRITICAL      :                         1                0.017
PASS
---------------------------------------------------------------------
```

Related Links

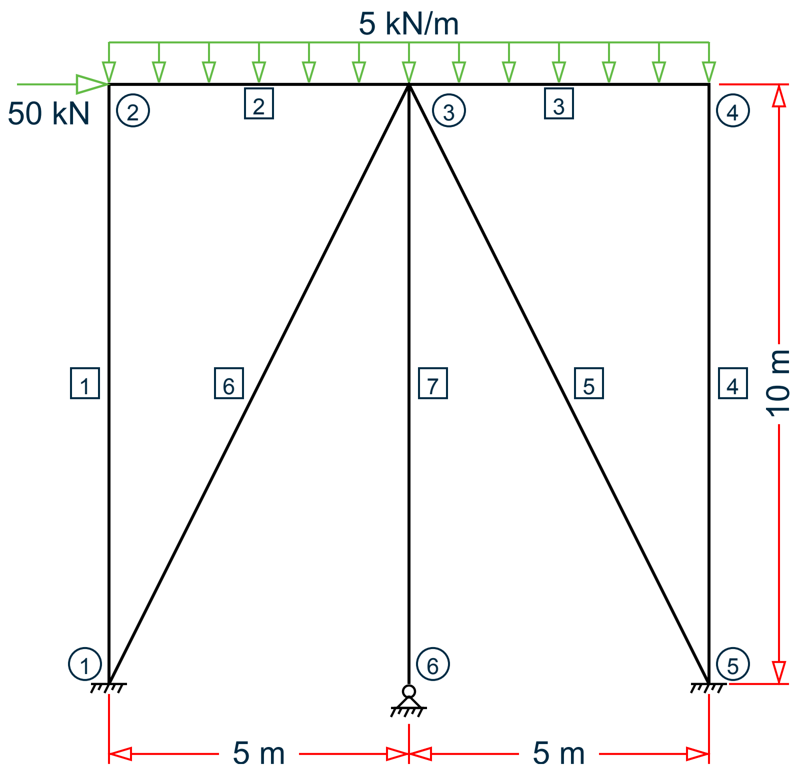
- *D1.J.8.1 Joint Checking* (on page 1293)

V.API Overlapping KJoint - Comp and Bend

Verify the design of an overlapping K joint with axial load, in-plane bending, and out-of-plane bending.

Details

Design of node 3 in the following model.



Chord member (2 & 3): D = 500 mm, T = 20 mm (A_c = 30,159 mm^2, Z_c = 3,480,380 mm^3)

Brace member (5 & 7): d = 400 mm, t = 20 mm (A_b = 23,876 mm^2, Z_b = 2,160,790 mm^3)

Table 837: Loads

Member	Fx (kN)	MIP (kN·m)	MOP (kN·m)
Chord	49.99 (T)	17.58	4.96
Brace	72.51 (C)	0.727	6.254

Validation

<u>Design Parameters</u>

$$\beta = d/D = 0.8$$

So, $0.2 < \beta < 1.0$ (OK)

$$\gamma = D/2T = 12.5$$

So, $10 < \gamma < 50$ (OK)

$$\theta = \tan^{-1}(10/5) = 63.4°$$

So, $30° < \theta < 90°$ (OK)

$F_y = 500\ MPa$ (OK)

$$g = -76.2\text{ mm}$$

$g/D = -76.2 / 500 = -0.1524$

<u>Determination of Q_u</u>

First, evaluate Q_g. Since $g/D < -0.50$, use linear interpolation.

For $g/D = -0.05$: $Q_g = 0.13 + 0.65 \times \phi \times \gamma^{0.5} = 0.13 + 0.65 \times (1) \times (12.5)^{0.5} = 2.428$

$$Q_{u\_ax} = (16 + 1.2 \times \gamma) \times \beta^{1.2} \times Q_g = [16 + 1.2(12.5)](0.8)^{1.2}(2.428) = 57.586$$

$$Q_{u\_ipb} = (5 + 0.7 \times \gamma) \times \beta^{1.2} = [5 + 0.7(12.5)](0.8)^{1.2} = 10.52$$

$$Q_{u\_opb} = 2.5 + (4.5 + 0.2 \times \gamma) \times \beta^{2.6} = 2.5 + [4.5 + 0.2(12.5)](0.8)^{2.6} = 6.419$$

<u>Determination of Q_f</u>

From Table 4.3.2:

$C1_{ax} = 0.2$, $C2_{axi} = 0.2$, $C3_{ax} = 0.3$

$C1_{ipb} = 0.2$, $C2_{ipb} = 0$, $C3_{ipb} = 0.4$

$C1_{opb} = 0.2$, $C2_{opb} = 0$, $C3_{opb} = 0.4$

$$P_{yc} = A_c \times F_{yc} = 15{,}079.5 \text{ kN}$$

$$M_{pc} = Z_c \times F_{yc} = 1{,}740.19 \text{ kN} \cdot \text{m}$$

$$P_c = 49.99 \text{ KN}$$

$$M_c = \sqrt{(MIPC^2 + MOPC^2)} = 18.267 \text{ kN} \cdot \text{m}$$

$$A = \sqrt{\left(\frac{FS \times P_c}{P_{yc}}\right)^2 + \left(\frac{FS \times M_c}{M_{pc}}\right)^2} = \sqrt{\left(\frac{1.6 \times 49.99}{17,\ 079.5}\right)^2 + \left(\frac{1.6 \times 18.267}{1,\ 740.19}\right)^2} = 0.0176$$

$$Q_{f\_ax} = 1 + C1_{ax}\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_{ax}\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_{ax}A^2$$

$$Q_{f\_ax} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0.2\left(\frac{1.6 \times 18.267}{1,\ 740.19}\right) - 0.3(0.0176)^2 = 0.9989$$

$$Q_{\text{f\_ipb}} = 1 + \text{C1}_{\text{ipb}}\left(\frac{FS \times P_c}{P_{yc}}\right) - \text{C2}_{\text{ipb}}\left(\frac{FS \times M_c}{M_{pc}}\right) - \text{C3}_{\text{ipb}}A^2$$

$$Q_{\text{f\_ipb}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.4(0.0176)^2 = 1.001$$

$$Q_{\text{f\_opb}} = 1 + \text{C1}_{\text{opb}}\left(\frac{FS \times P_c}{P_{yc}}\right) - \text{C2}_{\text{opb}}\left(\frac{FS \times M_c}{M_{pc}}\right) - \text{C3}_{\text{opb}}A^2$$

$$Q_{\text{f\_ipb}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.4(0.0176)^2 = 1.001$$

<u>Axial Capacity</u>

$$P_a = Q_u Q_f \frac{F_{yc}T^2}{FS\sin\theta} = 57.586(0.9977)\frac{500(20)^2}{1.6\sin(63.4\circ)10^3} = 8,\ 041.5 \text{ kN}$$

<u>Moment Capacity</u>

In-plane bending:

$$M_a = Q_u Q_f \frac{F_{yc}T^2 d}{FS\sin\theta} = 10.52(1.001)\frac{500(20)^2 400}{1.6\sin(63.4\circ)10^6} = 588.85 \text{ kN} \cdot \text{m}$$

Out-of-plane bending:

$$M_a = Q_u Q_f \frac{F_{yc} T^2 d}{FS \sin\theta} = 6.419(1.001)\frac{500(20)^2 400}{1.6\sin\left(63.4^\circ\right)10^6} = 359.31 \text{ kN·m}$$

<u>Brace Loads</u>

As axial loads for through and overlapping brace are of different sign, resultant axial load will be that of through brace. (Page 65, API code).

Axial load = 72.511 kN (C)

According to Page 65 of the API code, the resultant in-plane and out-plane bending moments will be algebraic summation of moments of the through brace and overlapping brace.

In-plane moment = 0.727 + (-0.89) = -0.163 kN·m

Out-of-plane moment = 6.254 + 6.254 = 12.508 kN·m

<u>Interaction Ratio</u>

$P/P_a + (M/M_a)_{ip}^2 + (M/M_a)_{op} = (72.511 / 8{,}014.5) + (0.163 / 588.85)^2 + (12.508 / 359.31) = 0.044$

Results

Table 838: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Q_{u_ax}	57.586	57.617	negligible	
Q_{u_ipb}	10.52	10.523	negligible	
Q_{u_opb}	6.419	6.419	none	
Q_{f_ax}	0.9989	0.999	negligible	
Q_{f_ipb}	1.001	1.001	none	
Q_{f_opb}	1.001	1.001	none	
Axial capacity (kN)	8,041.5	8,029.9	negligible	
In-plane Bending Capacity (kN·m)	588.86	588.1	negligible	
Out-of-plane Bending Capacity (kN·m)	359.31	358.74	negligible	
Interaction ratio	0.044	0.044	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\API\API Overlapping KJoint - Comp and Bend.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 5 10 0; 4 10 10 0; 5 10 0 0; 6 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 3 5; 6 3 1; 7 3 6;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 4 7 TABLE ST W6X12
5 6 TABLE ST PIPE OD 0.4 ID 0.36
2 3 TABLE ST PIPE OD 0.5 ID 0.46
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 FIXED
6 PINNED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -50
MEMBER LOAD
2 3 UNI GY -5
2 3 CMOM Y 5
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE API
*FSJ 2.6 ALL
FYLD 500000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

Note: The generated .PUN file is required to generated this output for K or X joints. Otherwise, the program assumes all joints are Y joints. Refer to D1.J.8.2 Joint File Format (on page 1295) for details.

```
======================================================================

NODE NO :      3   CHORD NO:     2   BRACE NO:
6
======================================================================

DESIGN DATA : (Units    : N ,
mm  )
  Chord Memb :      D =  500.00  T =
19.99
  Brace Memb :      d =  400.00  t =
19.99
  Angle (THETA) =  63.4 deg  GAP   =  -76.20   Fyc =
499.9
  BETA          =  0.80      GAMMA =   12.51   TAU =
1.00
      STAAD SPACE                                              -- PAGE NO.
7
      ***WARNING - JNT      3:
OVERLAP
      OVERLAP PROVIDED =  76.20 IS LESS THAN THE NOMINAL VALUE OF
0.25*BETA*D.
JOINT CLASS :
KO
  Overlap Brace :       5   Angle =   63.43 deg.   Diam. =
400.00

FACTORS :

--------------------------------------------------------------------
  Joint   Load   Strength     Chord Load    C1        C2
C3
  Class   Cond   factor (Qu)  factor
(Qf)

--------------------------------------------------------------------
    K     AX      57.617        0.999       0.200    0.200
0.300
    K     IPB     10.523        1.001       0.200    0.000
0.400
    K     OPB      6.419        1.001       0.200    0.000
0.400
----------------------------------------------------------------------

CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO
STATUS
   (Cl. 4.3)             (KN, m )         (KN,
m )
----------------------------------------------------------------------

AXIAL        :            -72.51      1     8029.90    0.009
```

```
PASS
IP BENDING   :            -0.16      1      588.09    0.000
PASS
OP BENDING   :            12.51      1      358.74    0.035
PASS
INTERACTION  :                       1                0.044
PASS
----------------------------------------------------------------------

CRITICAL     :                       1                0.044
PASS
----------------------------------------------------------------------
```

Related Links

- *D1.J.8.1 Joint Checking* (on page 1293)

V.API Overlapping K Joint - Tens and Bend

Verify the design of an overlapping K joint with axial load and in-plane bending.

References

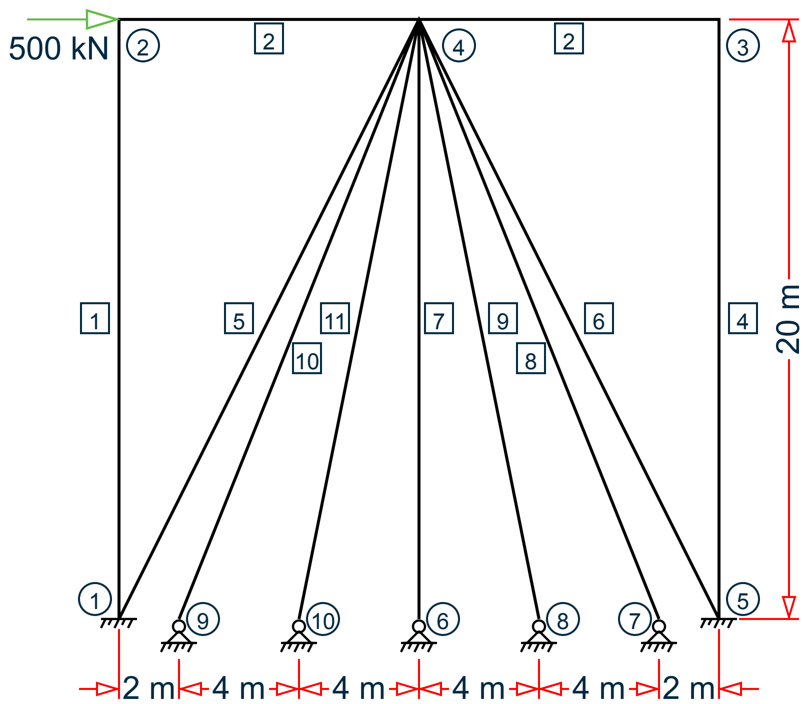
Recommended Practice for Planning, Design and Constructing Fixed Offshore Platforms-Working Stress Design. American Petroleum Institute. 21st Edition (December 2000).

Related Links

- *D1.J.8.1 Joint Checking* (on page 1293)

Details

Design of node 4 (between members 2 and 5) in the following model.



Chord member (2): D = 20 in, T = 1 in (A_c = 59.69 in^2, Z_c = 361.3 in^3)

Brace member (5): d = 18 in, t = 1 in (A_b = 53.41^2, Z_b = 289.3^3)

Brace member 10 overlaps with member 5.

Table 839: Loads

Member	Fx (kN)	MIP (kN·m)	MOP (kN·m)
Chord	499.21 (C)	268.18	0
Brace	322.34 (T)	101.72	0

Validation

Design Parameters

$$\beta = d/D = 0.9$$

So, $0.2 < \beta < 1.0$ (OK)

$$\gamma = D/2T = 10$$

So, $10 < \gamma < 50$ (OK)

$$\theta = \tan^{-1}(10/5) = 63.4°$$

So, $30° < \theta < 90°$ (OK)

$F_y = 50\ ksi$ (OK)

$$g = -4 \text{ mm}$$

$g/D = -4 / 20 = -0.2$

Determination of Q_u

First, evaluate Q_g:

For $g/D < -0.05$: $Q_g = 0.13 + 0.65 \times \phi \times \gamma^{0.5} = 0.13 + 0.65 \times (1) \times (10)^{0.5} = 2.185$

$$Q_{u\_ax} = (16 + 1.2 \times \gamma) \times \beta^{1.2} \times Q_g = [16 + 1.2(10)](0.9)^{1.2}(2.185) = 53.914$$

$$Q_{u\_ipb} = (5 + 0.7 \times \gamma) \times \beta^{1.2} = [5 + 0.7(10)](0.9)^{1.2} = 10.575$$

Determination of Q_f

From Table 4.3.2:

$C1_{ax} = 0.2$, $C2_{axi} = 0.2$, $C3_{ax} = 0.3$

$C1_b = 0.2$, $C2_b = 0$, $C3_b = 0.4$

$$P_{yc} = A_c \times F_{yc} = 2{,}984.5 \text{ kips}$$

$$M_{pc} = Z_c \times F_{yc} = 18{,}067 \text{ in·k}$$

$$P_c = -499.21 \text{ kips}$$

$$M_c = \sqrt{(MIPC^2 + MOPC^2)} = 268.18 \text{ in·k}$$

$$A = \sqrt{\left(\frac{FS \times P_c}{P_{yc}}\right)^2 + \left(\frac{FS \times M_c}{M_{pc}}\right)^2} = \sqrt{\left(\frac{2 \times 499.21}{2,\ 984.5}\right)^2 + \left(\frac{2 \times 268.18}{18,\ 067}\right)^2} = 0.3359$$

$$Q_{f\_ax} = 1 + C1_{ax}\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_{ax}\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_{ax}A^2$$

$$Q_{f\_ax} = 1 + 0.2\left(\frac{2 \times 499.21}{2,\ 984.5}\right) - 0.2\left(\frac{2 \times 268.18}{18,\ 067}\right) - 0.3(0.3359)^2 = 0.893$$

$$Q_{f\_bend} = 1 + C1_b\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_b\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_bA^2$$

$$Q_{f\_ax} = 1 + 0.2\left(\frac{2 \times 499.21}{2,\ 984.5}\right) - 0 - 0.4(0.3359)^2 = 0.888$$

Axial Capacity

$$P_a = Q_u Q_f \frac{F_{yc}T^2}{FS\sin\theta} = 53.914(0.893)\frac{50(1)^2}{2\sin\left(63.4\,°\right)10^3} = 1,\ 346.1 \text{ kips}$$

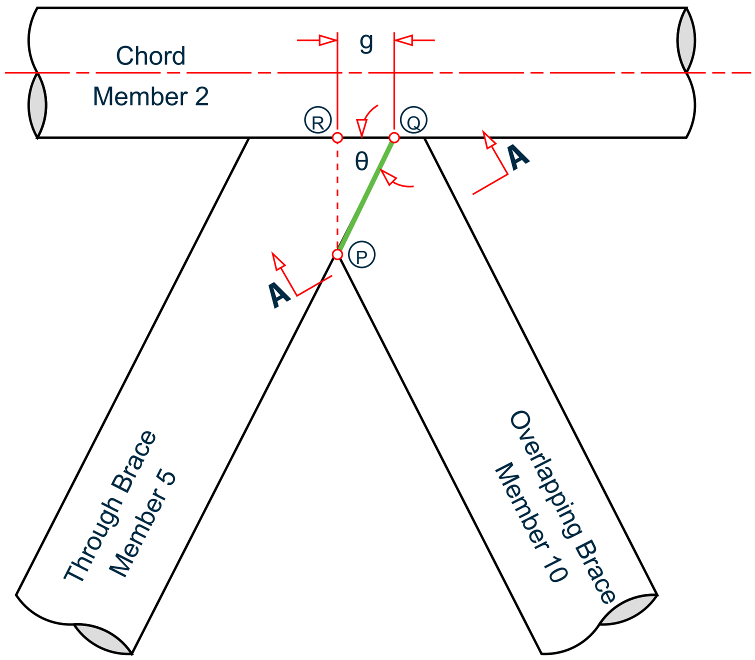
Moment Capacity

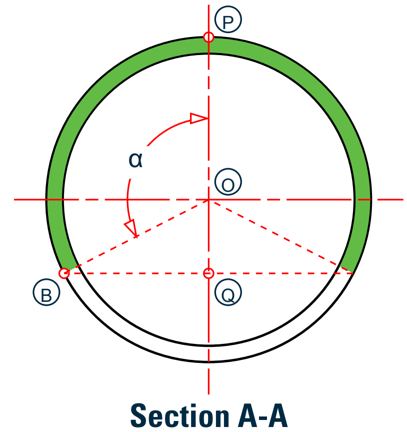
$$M_a = Q_u Q_f \frac{F_{yc}T^2 d}{FS\sin\theta} = 10.52(1.001)\frac{50(1)^2 18}{2\sin\left(63.4\,°\right)10^6} = 4,\ 726 \text{ in·k}$$

Out-of-plane bending is zero so moment capacity is not calculated.

Brace Loads

As axial loads for through and overlapping braces are of different signs, the resultant axial load will be the sum of both brace loads (reference p.65).





The gap distance, $g = 4\ in$ (segment QR).

Angle PQR = $\theta = 63.4°$

The loaded segment length (green line PQ) = $g / cos(\theta) = 4 / cos(63.4°) = 8.93\ in$

To determine the overlapping portion of the cross section of member 10 (i.e., the green portion of Section A-A), node that line segments OP and OB are equal to the radius of the pipe (use the nominal outer radius), $r = 6\ in$.

$$\cos(\alpha) = (r - PQ)/r = (6 - 8.93) / 6 = -0.489$$

$$\alpha = 119.3° = 2.082 \text{ rad}$$

The arc length (green portion of Section A-A) = $2 \times \alpha \times r = 2(2.082)(6) = 25\ in$

The proportion of this arc to the total circumference = $25\ /\ (2\pi \times r) = 25\ /\ (2\pi \times 6) = 0.663$

So the total axial load is the axial load of the through brace plus this ratio times the axial load of the overlapping brace (axial force values taken from STAAD.Pro post-processing workflow):

$$322.39 \text{ kip} + 0.663 \times 181.5 \text{ kip} = 442.7 \text{ kip}$$

According to p.65 of the API code, the resultant in-plane and out-plane bending moments will be algebraic summation of moments of the through brace and overlapping brace. Therefore, the total in-plane moment:

$$101.72 \text{ in·k} + 12.61 \text{ in·k} = 114.33 \text{ in·k}$$

Interaction Ratio

$$P/P_a + (M/M_a)^2 = (442.7 / 1{,}346.1) + (114.33 / 4{,}726)^2 = 0.329$$

Results

Table 840: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Q_{u_ax}	53.91	53.926	negligible	
Q_{u_ipb}	10.575	10.575	none	
Q_{f_ax}	0.893	0.893	none	
Q_{f_ipb}	0.888	0.888	none	
Axial capacity (kip)	1,346.1	1,346.18	negligible	
In-plane Bending Capacity (in·k)	4,726	4,724.22	negligible	
Brace Load, Axial (kip)	442.7	442.72	negligible	
Brace Load, In-plane Bending (in·k)	114.33	114.33	none	
Interaction ratio	0.329	0.329	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\API\API Overlapping KJoint - Tens and Bend.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 787.402 0; 3 787.402 787.402 0; 4 393.701 787.402 0;
5 787.402 0 0; 6 393.701 0 0; 7 708.661 0 0; 8 551.181 0 0; 9 78.7402 0 0;
10 236.22 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 4; 3 4 3; 4 5 3; 5 1 4; 6 4 5; 7 4 6; 8 4 7; 9 4 8; 10 9 4; 11 10
4;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36.7236 FU 59.1464 RY 1.5 RT 1.2
```

```
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 4 7 TABLE ST PIPE OD 20 ID 18
5 6 8 9 11 TABLE ST PIPE OD 18 ID 16
10 TABLE ST PIPE OD 12 ID 10
CONSTANTS
MATERIAL STEEL ALL
*START USER TABLE
*TABLE 1
*UNIT INCHES KIP
*PRISMATIC
*UPRISMATIC-PIPE(1)
*6.09218 280.735 280.735 561.471 3.04623 3.04623 20 20
*TABLE 2
*UNIT INCHES KIP
*PRISMATIC
*UPRISMATIC-PIPE(2)
*5.48931 205.369 205.369 410.738 2.74481 2.74481 18 18
*END
*DEFINE MATERIAL START
*ISOTROPIC ASTM_STEEL
*E 29000
*POISSON 0.3
*DENSITY 8.80613e-006
*ALPHA 2.16e-005
*DAMP 0.03
*G 11153.8
*END DEFINE MATERIAL
*MEMBER PROPERTY AMERICAN
*1 TO 4 7 UPTABLE 1 UPRISMATIC-PIPE(1)
*5 6 8 TO 11 UPTABLE 2 UPRISMATIC-PIPE(2)
*CONSTANTS
*MATERIAL ASTM_STEEL ALL
SUPPORTS
1 5 FIXED
6 TO 10 PINNED
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE LOAD GROUP 1
JOINT LOAD
2 FX 500
END DEFINE REFERENCE LOADS
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
REFERENCE LOAD
R1 1.0
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE API
FYLD 50 ALL
FSJ 2 MEMB 2 3 5 6 8 10
TRACK 2 MEMB 2 3 5 6 8 10
CHECK CODE MEMB 2 3 5 6 8 10
FINISH
```

STAAD.Pro Output

Note: The generated .PUN file is required to generated this output for K or X joints. Otherwise, the program assumes all joints are Y joints. Refer to D1.J.8.2 Joint File Format (on page 1295) for details.

```
          STAAD.Pro - API JOINT CHECKS TO 21st edition.
          ---------------------------------------------------
                     (Includes Supplements 1-3)
          ---------------------------------------------------
======================================================================

NODE NO :      4   CHORD NO:     2   BRACE NO:
5
======================================================================

DESIGN DATA : (Units    :
kip,inch)
  Chord Memb :      D =   20.00  T =
1.00
  Brace Memb :      d =   18.00  t =
1.00
  Angle (THETA) =  63.4 deg  GAP   =   -4.00   Fyc =
50.0
  BETA         =  0.90      GAMMA =   10.00   TAU =
1.00
      ***WARNING - JNT      4:
OVERLAP
      OVERLAP PROVIDED =   4.00 IS LESS THAN THE NOMINAL VALUE OF
0.25*BETA*D.
JOINT CLASS :
KO
  Overlap Brace :      10   Angle =   68.20 deg.   Diam. =
12.00

FACTORS :

--------------------------------------------------------------------
  Joint   Load   Strength     Chord Load    C1        C2
C3
  Class   Cond   factor (Qu)  factor
(Qf)

--------------------------------------------------------------------
    K     AX      53.926        0.893       0.200    0.200
0.300
    K     IPB     10.575        0.888       0.200    0.000
0.400
----------------------------------------------------------------------

CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO
STATUS
   (Cl. 4.3)             (kip,in)
(kip,in)
----------------------------------------------------------------------

AXIAL        :            442.72      1     1346.18    0.329
```

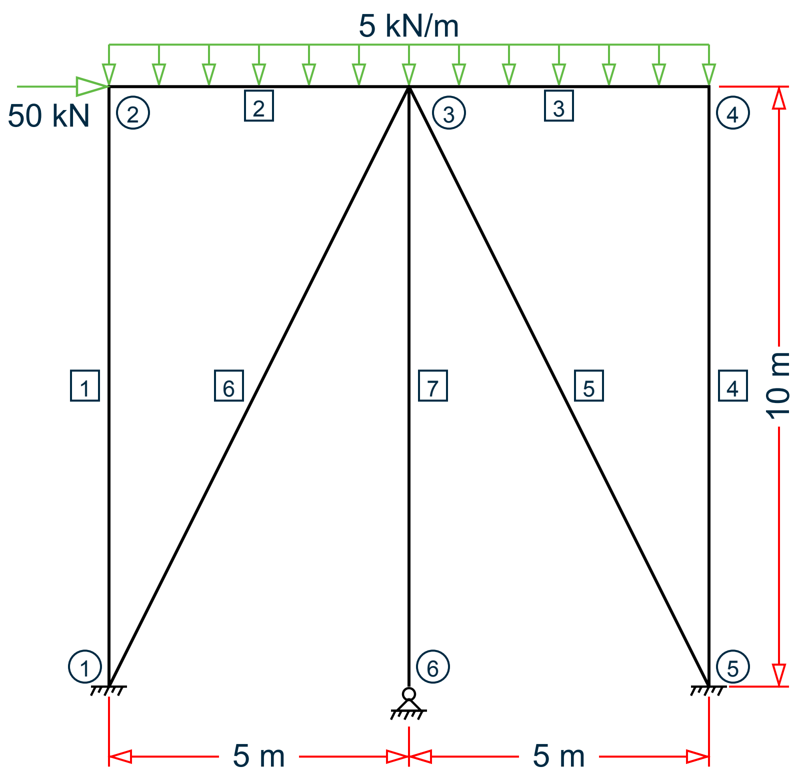
```
PASS
IP BENDING    :           114.33      1     4724.22    0.024
PASS
INTERACTION   :                       1                0.329
PASS
----------------------------------------------------------------------

CRITICAL      :                       1                0.329
PASS
----------------------------------------------------------------------
```

V.API X Joint

Verify the design of a simple X joint with axial load and in-plane bending.

Details

Design of node 3 between members 2 and 6 in the following model.



Chord member (2 & 3): D = 500 mm, T = 20 mm (A_c = 30,159 mm^2, Z_c = 3,480,380 mm^3)

Brace member (6): d = 475 mm, t = 15 mm (A_b = 21,677 mm^2, Z_b = 2,416,700 mm^3)

Table 841: Loads

Member	Fx (kN)	MIP (kN·m)	MOP (kN·m)
Chord	49.99 (T)	17.34	4.97
Brace	66.082 (C)	0.921	6.255

Validation

Design Parameters

$$\beta = d/D = 0.95$$

So, $0.2 < \beta < 1.0$ (OK)

$$\gamma = D/2T = 12.5$$

So, $10 < \gamma < 50$ (OK)

$$\theta = \tan^{-1}(10/5) = 63.4°$$

So, $30° < \theta < 90°$ (OK)

$F_y = 500\ MPa$ (OK)

Determination of Q_u

$$Q_\beta = 0.3/[\beta(1 - 0.833\times\beta)] = 1.513$$

$$Q_{u\_ax} = [2.8 + (12 + 0.1\times\gamma)\times\beta]\times Q_\beta = [2.8 + (12 + 0.1\times12.5)\times0.95](1.513) = 23.289$$

$$Q_{u\_ipb} = (5 + 0.7\times\gamma)\times\beta^{1.2} = [5 + 0.7(12.5)](0.95)^{1.2} = 12.929$$

$$Q_{u\_opb} = 2.5 + (4.5 + 0.2\times\gamma)\times\beta^{2.6} = 2.5 + [4.5 + 0.2(12.5)](0.95)^{2.6} = 8.626$$

Determination of Q_f

From Table 4.3.2:

$C1_{ax} = 0.2$, $C2_{axi} = 0$, $C3_{ax} = 0.35$

For C1, C2, and C3, interpolate between values for β ≤ 0.9 and β = 1.0.

$C1_b = 0.2$, $C2_b = 0$, $C3_b = 0.4$

$$P_{yc} = A_c\times F_{yc} = 15{,}079.5 \text{ kN}$$

$$M_{pc} = Z_c\times F_{yc} = 1{,}740.19 \text{ kN·m}$$

$$P_c = 49.99 \text{ kN}$$

$$M_c = \sqrt{(MIPC^2+MOPC^2)} = 18.04 \text{ kN·m}$$

$$A = \sqrt{\left(\frac{FS \times P_c}{P_{yc}}\right)^2 + \left(\frac{FS \times M_c}{M_{pc}}\right)^2} = \sqrt{\left(\frac{1.6 \times 49.99}{17,\ 079.5}\right)^2 + \left(\frac{1.6 \times 18.04}{1,\ 740.19}\right)^2} = 0.0174$$

$$Q_{f\_ax} = 1 + C1_{ax}\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_{ax}\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_{ax}A^2$$

$$Q_{f\_ax} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.35(0.0174)^2 = 1.0$$

$$Q_{f\_\text{ipb}} = 1 + C1_b\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_b\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_b A^2$$

$$Q_{f\_\text{ipb}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.4(0.0174)^2 = 1.001$$

$$Q_{f\_\text{opb}} = 1 + C1_b\left(\frac{FS \times P_c}{P_{yc}}\right) - C2_b\left(\frac{FS \times M_c}{M_{pc}}\right) - C3_b A^2$$

$$Q_{f\_\text{opb}} = 1 + 0.2\left(\frac{1.6 \times 49.99}{15,\ 079.5}\right) - 0 - 0.4(0.0174)^2 = 1.001$$

Axial Capacity

$$P_a = Q_u Q_f \frac{F_{yc} T^2}{FS \sin\theta} = 23.289(1.0)\frac{500(20)^2}{1.6\sin\left(63.4\,^\circ\right)10^3} = 3,\ 255.7\ \text{kN}$$

Moment Capacity

In-plane bending:

$$M_a = Q_u Q_f \frac{F_{yc} T^2 d}{FS \sin\theta} = 12.929(1.001)\frac{500(20)^2 475}{1.6\sin\left(63.4\,^\circ\right)10^6} = 858.53\ \text{kN}\cdot\text{m}$$

Out-of-plane bending:

$$M_a = Q_u Q_f \frac{F_{yc} T^2 d}{FS \sin\theta} = 8.626(1.001)\frac{500(20)^2 475}{1.6\sin\left(63.4\,^\circ\right)10^6} = 572.8\ \text{kN}\cdot\text{m}$$

Interaction Ratio

$P/P_a + (M/M_a)_{ipb}^2 + (M/M_a)_{opb} = (66.082 / 3{,}255.7) + (0.921 / 858.53)^2 + (6.255 / 572.8) = 0.031$

Results

Table 842: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Q_{u_ax}	23.829	23.921	negligible	
Q_{u_ipb}	12.929	12.934	negligible	
Q_{u_opb}	8.626	8.627	negligible	
Q_{f_ax}	1.0	1.0	none	
Q_{f_ipb}	1.001	1.001	none	
Q_{f_opb}	1.001	1.001	none	
Axial capacity (kN)	3,255.7	3,250,61	negligible	
In-plane Bending Capacity (kN·m)	858.53	858.33	negligible	

Result Type	Reference	STAAD.Pro	Difference	Comments
Out-of-plane Bending Capacity (kN·m)	572.8	572.55	negligible	
Interaction ratio	0.031	0.031	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\API\API XJoint.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 5 10 0; 4 10 10 0; 5 10 0 0; 6 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 3 5; 6 3 1; 7 3 6;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 4 TABLE ST W6X12
*5 6 7 TABLE ST HSSP16X0.625
*2 3 TABLE ST HSSP20x0.5
2 3 TABLE ST PIPE OD 0.5 ID 0.46
5 TO 7 TABLE ST PIPE OD 0.475 ID 0.445
*MEMBER PROPERTY EUROPEAN
*2 3 TABLE ST 610x20CHS
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 FIXED
6 PINNED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX -50
MEMBER LOAD
2 3 UNI GY -5
2 3 CMOM Y 5
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
```

```
CODE API
*FSJ 2.6 ALL
FYLD 500000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

Note: The generated **.PUN** file is required to generated this output for K or X joints. Otherwise, the program assumes all joints are Y joints. Refer to D1.J.8.2 Joint File Format (on page 1295) for details.

```
======================================================================

NODE NO :      3   CHORD NO:     2   BRACE NO:
6
======================================================================

DESIGN DATA : (Units    : N ,
mm  )
  Chord Memb :      D =  500.00  T =
19.99
  Brace Memb :      d =  475.01  t =
15.01
  Angle (THETA) =  63.4 deg  GAP   =    0.00   Fyc =
499.9
  BETA          =  0.95      GAMMA =   12.51   TAU =
0.75
      STAAD SPACE                                            -- PAGE NO.
7
JOINT CLASS :
X

FACTORS :

--------------------------------------------------------------------
  Joint   Load   Strength     Chord Load    C1        C2
C3
  Class   Cond   factor (Qu)  factor
(Qf)

--------------------------------------------------------------------
    X     AX      23.291        1.000      -0.000    0.000
0.350
    X     IPB     12.934        1.001       0.200    0.000
0.400
    X     OPB      8.627        1.001       0.200    0.000
0.400
----------------------------------------------------------------------

CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO
STATUS
   (Cl. 4.3)             (KN, m )        (KN,
m )
----------------------------------------------------------------------
```

```
AXIAL        :          -66.08      1     3250.61    0.020
PASS
IP BENDING   :            0.92      1      858.33    0.001
PASS
OP BENDING   :            6.26      1      572.55    0.011
PASS
INTERACTION  :                      1                0.031
PASS
----------------------------------------------------------------------

CRITICAL     :                      1                0.031
PASS
----------------------------------------------------------------------
```

Related Links

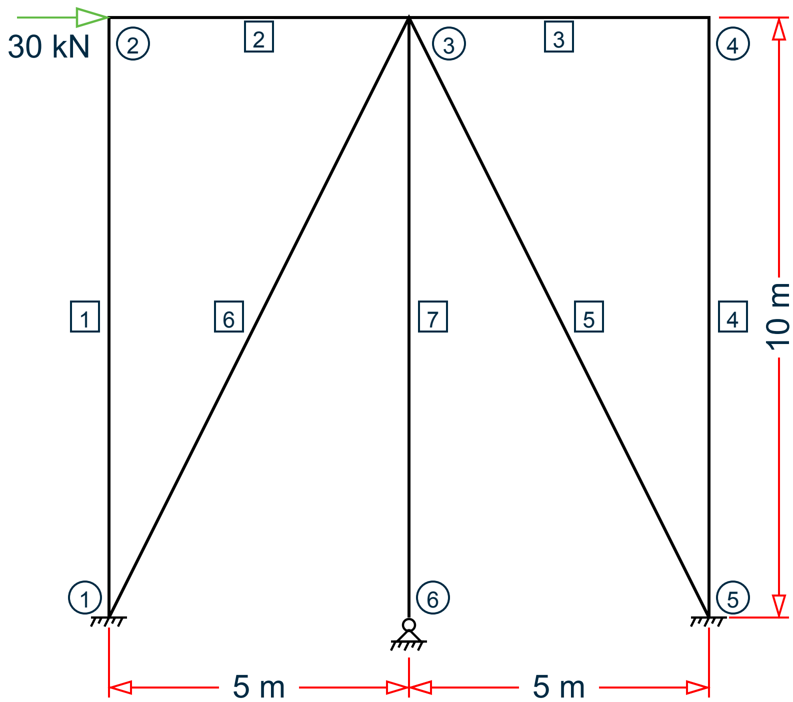
- *D1.J.8.1 Joint Checking* (on page 1293)

V.API Y Joint

Verify the design of a simple Y joint with axial load and in-plane bending.

Details

Consider the Y joint at node 3 between members 2 and 6.



Chord member (2 & 3): D = 500 mm, T = 20 mm (A_c = 30,159 mm^2, Z_c = 3,480,380 mm^3)

Brace member (5 & 7): d = 400 mm, t = 20 mm (A_b = 23,876 mm^2, Z_b = 2,160,790 mm^3)

Table 843: Loads

Member	Fx (kN)	MIP (kN·m)	MOP (kN·m)
Chord	30 (C)	0.49	0
Brace	33.34 (T)	0.485	0

Validation

Design Parameters

$$\beta = d/D = 0.8$$

So, $0.2 < \beta < 1.0$ (OK)

$$\gamma = D/2T = 12.5$$

So, $10 < \gamma < 50$ (OK)

$$\theta = \tan^{-1}(10/5) = 63.4°$$

So, $30° < \theta < 90°$ (OK)

$F_y = 500\ MPa$ (OK)

$$g = 0$$

Determination of Q_u

$$Q_{u\_ax} = 30\times\beta = 30(0.8) = 24$$

$$Q_{u\_ipb} = (5 + 0.7\times\gamma)\times\beta^{1.2} = [5 + 0.7(12.5)](0.8)^{1.2} = 10.52$$

Determination of Q_f

From Table 4.3.2:

$C1_{ax} = 0.3$, $C2_{axi} = 0$, $C3_{ax} = 0.8$

$C1_b = 0.2$, $C2_b = 0$, $C3_b = 0.4$

$$P_{yc} = A_c\times F_{yc} = 15{,}079.5\ kN$$

$$M_{pc} = Z_c\times F_{yc} = 1{,}740.19\ kN\cdot m$$

$$P_c = -30\ kN$$

$$M_c = \sqrt{(MIPC^2+MOPC^2)} = 0.49\ kN\cdot m$$

$$A = \sqrt{\left(\frac{FS\times P_c}{P_{yc}}\right)^2 + \left(\frac{FS\times M_c}{M_{pc}}\right)^2} = \sqrt{\left(\frac{1.6\times 30}{17,\ 079.5}\right)^2 + \left(\frac{1.6\times 0.49}{1,\ 740.19}\right)^2} = 0.003$$

$$Q_{f\_\text{ax}} = 1 + C1_{ax}\left(\frac{FS\times P_c}{P_{yc}}\right) - C2_{ax}\left(\frac{FS\times M_c}{M_{pc}}\right) - C3_{ax}A^2$$

$$Q_{f\_\text{ax}} = 1 + 0.3\left(\frac{1.6\times 30}{15,\ 079.5}\right) - 0 - 0.8(0.003)^2 = 0.999$$

$$Q_{f\_\text{bend}} = 1 + C1_b\left(\frac{FS\times P_c}{P_{yc}}\right) - C2_b\left(\frac{FS\times M_c}{M_{pc}}\right) - C3_bA^2$$

$$Q_{f\_\mathrm{bend}} = 1 + 0.2\left(\frac{1.6 \times 30}{15,\ 079.5}\right) - 0 - 0.4(0.003)^2 = 0.999$$

Axial Capacity

$$P_a = Q_u Q_f \frac{F_{yc} T^2}{FS \sin\theta} = 24(0.999)\frac{500(20)^2}{1.6\sin\left(63.4\,^\circ\right)10^3} = 3,\ 351.8\ \mathrm{kN}$$

Moment Capacity

$$M_a = Q_u Q_f \frac{F_{yc} T^2 d}{FS \sin\theta} = 10.52(0.999)\frac{500(20)^2 400}{1.6\sin\left(63.4\,^\circ\right)10^6} = 587.9\ \mathrm{kN{\cdot}m}$$

Out-of-plane bending is zero so moment capacity is not calculated.

Interaction Ratio

$$P/P_a + (M/M_a)^2 = (33.34 / 3{,}351.8) + (0.49 / 587.9)^2 = 0.010$$

Results

Table 844: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Q_{u_ax}	24	24	none	
Q_{u_ipb}	10.52	10.52	none	
Q_{f_ax}	0.999	0.999	none	
Q_{f_ipb}	0.999	0.999	none	
Axial capacity (kN)	3,351.8	3,350.87	negligible	
In-plane Bending Capacity (kN·m)	587.86	587.7	negligible	
Interaction ratio	0.010	0.010	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\API\API YJoint.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 12-Mar-19
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 5 10 0; 4 10 10 0; 5 10 0 0; 6 5 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 3 5; 6 3 1; 7 3 6;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 253200 FU 407800 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 4 7 TABLE ST W6X12
5 6 TABLE ST PIPE OD 0.4 ID 0.36
2 3 TABLE ST PIPE OD 0.5 ID 0.46
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 5 6 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FX 30
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES ALL
PARAMETER 1
CODE API
FSJ 1.6 ALL
FYLD 500000 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
======================================================================

NODE NO :      3   CHORD NO:     2   BRACE NO:
6
======================================================================

DESIGN DATA : (Units    : N ,
mm  )
  Chord Memb :      D =  500.00  T =
19.99
  Brace Memb :      d =  400.00  t =
19.99
  Angle (THETA) =  63.4 deg  GAP   =    0.00   Fyc =
499.9
  BETA          =  0.80      GAMMA =   12.51   TAU =
1.00
JOINT CLASS :
Y
      STAAD SPACE                                         -- PAGE NO.
7

FACTORS :

--------------------------------------------------------------------
```

```
  Joint   Load   Strength     Chord Load    C1        C2
C3
  Class   Cond   factor (Qu)  factor
(Qf)

--------------------------------------------------------------------
    Y     AX      24.000        0.999       0.300    0.000
0.800
    Y     IPB     10.523        0.999       0.200    0.000
0.400
----------------------------------------------------------------------

CAPACITY CHECKS        BRACE LOAD    LC   CAPACITY   RATIO
STATUS
   (Cl. 4.3)             (KN, m )         (KN,
m )
----------------------------------------------------------------------

AXIAL        :             33.34      1     3346.58    0.010
PASS
IP BENDING   :             -0.48      1      587.14    0.001
PASS
INTERACTION  :                        1                0.010
PASS
----------------------------------------------------------------------

CRITICAL     :                        1                0.010
PASS
----------------------------------------------------------------------
```

Related Links

- *D1.J.8.1 Joint Checking* (on page 1293)

V. ASME NF 3000 Codes

V. ASME NF 3000 1974 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1974 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

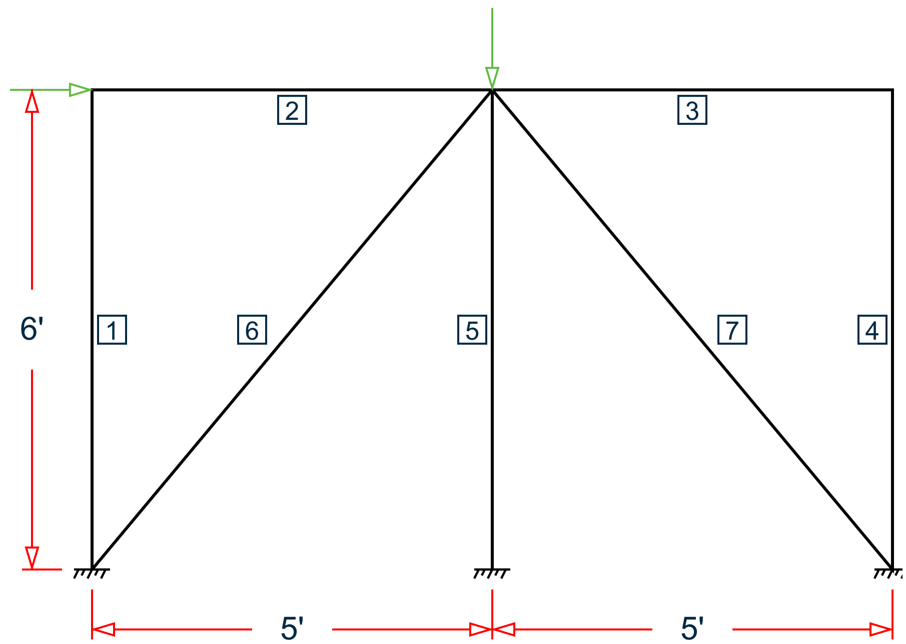


Figure 506: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = *36 ksi*, F_u = *58 ksi*).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

Axial Tension: Member 6 (per NF-74.2211)

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

Axial Compression: Member 7 (NF-74.2213):

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 845: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1974 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                          PASS      NF-74.2211        0.016         1
                    -2666.18 T          0.00          0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                    |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
```

```
 | AX.TENS: 3.52E+02                      COMPRESS:0.00E
+00                     |

 |--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81              |
 | SZZ:       5.16  SYY:
11.99                                             |

 |--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                               |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00       |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90         |
 | KS:1.000  KV:1.000  KBK:
1.000                                               |

 |--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                     |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

 |--------------------------------------------------------------------------------
|
      7  ST   L806010                 (AISC SECTIONS)
                       PASS      NF-74.2213       0.049         1
                    7104.23 C          0.00           0.00        0.00

 |--------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AX.TENS: 2.16E+04                      COMPRESS:1.72E
+04                     |

 |
```

```
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                  |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81                 |
 | SZZ:       5.16  SYY:
11.99                                   |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                      |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                      |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-   |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1974 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1974 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0)\ /\ (2 \times 404) = 15.41\ \text{ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$F_y = 51.375$ kips

$V_y = D \times t_w = 10.8\ in^2$

fv,y = Fy/Vy = 4.76 ksi

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

Cv =17.2

$$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi}$$

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v / F_v = 0.330$

Results

Table 846: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1974 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-74)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
                           PASS      NF-74-EQN-21      0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                          |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03                |
```

```
|--------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                |
 | SZZ:      53.87  SYY:
3.77                                             |

|--------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                                |

|--------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.713    1    0.00E+00       -         -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04       -         -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

|--------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1974 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1974 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2^4 - r_1^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2^3 - r_1^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 847: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress ratio	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1974 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
```

```
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-74)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                  (AISC SECTIONS)
                           PASS      NF-74-EQN-21      0.339         1
                        0.00 T          0.00     -331249.97       25.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                    |
 | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                            |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
+01               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37               |
 | SZZ:      41.10  SYY:
41.10                                          |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                             |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                            |
```

```
|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                         |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00        -          -          -
-    |
 | COMPRESSION  0.000    0    0.00E+00        -          -          -
-    |
 | COMP&BEND    0.339    1    0.00E+00        -          -    3.31E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00        -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.201    1        -      2.57E+04        -          -
-    |
 | SHEAR-Z      0.000    0        -          -     0.00E+00       -
-    |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1974 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1974 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in{\cdot}kips$

<u>Slenderness Ratio</u>

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

<u>Bending</u>

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8 \text{ ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93 \ ksi < 23.8 \ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7 \ ksi > 23.8 \ ksi$

Critical stress ratio = $38.7 \ ksi \ / \ 23.8 \ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01 \ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/ \ (a/h)^2] = 5.65$

$$t = t_w = 0.44 \text{ inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4 \ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$$Vy = Dt_w = 10.68 \text{ in}^2$$

$$fv = Fy/Vy = 5.805 \text{ ksi}$$

Stress ratio = fv/Fv = 0.403

Results

Table 848: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1974 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1974
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-74)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                  (AISC SECTIONS)
                           FAIL      NF-74-EQN-21      1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                             |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37                |
 | SZZ:      49.69  SYY:
18.75                                              |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                               |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
```

```
 | TENSION      0.000    0    0.00E+00        -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00        -         -         -
-   |
 | COMP&BEND    1.620    1    0.00E+00        -         -    1.92E+06 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00        -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.403    1        -     6.20E+04        -         -
-   |
 | SHEAR-Z      0.000    0        -         -    0.00E+00         -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1974 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1974 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in{\cdot}kips$

Slenderness Ratio

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)

Allowable Bending Stress

$$b_f/2t_f = 6.862 < 65/(Fy)^{0.5} \text{ [Ref. NF3322.1(d)(2)]}$$

Thus, allowable bending tensile stress = $F_{btz} = Fy\times[0.79\text{-}0.002\times(b_f/2t_f)\times\sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_zD/2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

Cv =1.475

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

Fy = 12.5 kip, $Vy = Dt_w = 4.8\ inch^2$

fv = Fy/Vy = 2.6 ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 849: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	24.3	4.8%	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.954	4.6%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1974 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1974
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-74)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W16X40                   (AISC SECTIONS)
                           PASS      NF-74-EQN-21      0.954         1
                        0.00 T          0.00      -1500.00        0.00

 |-----------------------------------------------------------------------------
 |
  | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
  | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                   |
  | AXIAL: 1.51E+01  FCZ: 2.43E+01  FCY: 2.70E+01  FTZ: 2.43E+01  FTY: 2.70E
+01 |
  | SHEAR: 1.44E
+01                                                             |
  | ACTUAL STRESSES: (UNIT - KIP
INCH)                                        |
```

```
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00                 |

 |--------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                              |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56             |
 | SZZ:      64.75  SYY:
8.26                                               |

 |--------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                  |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85        |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00          |
 | KS:1.000  KV:1.000  KBK:
1.000                                                |

 |--------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.954    1    0.00E+00       -         -    1.50E+03 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

 |--------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1977 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1977 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

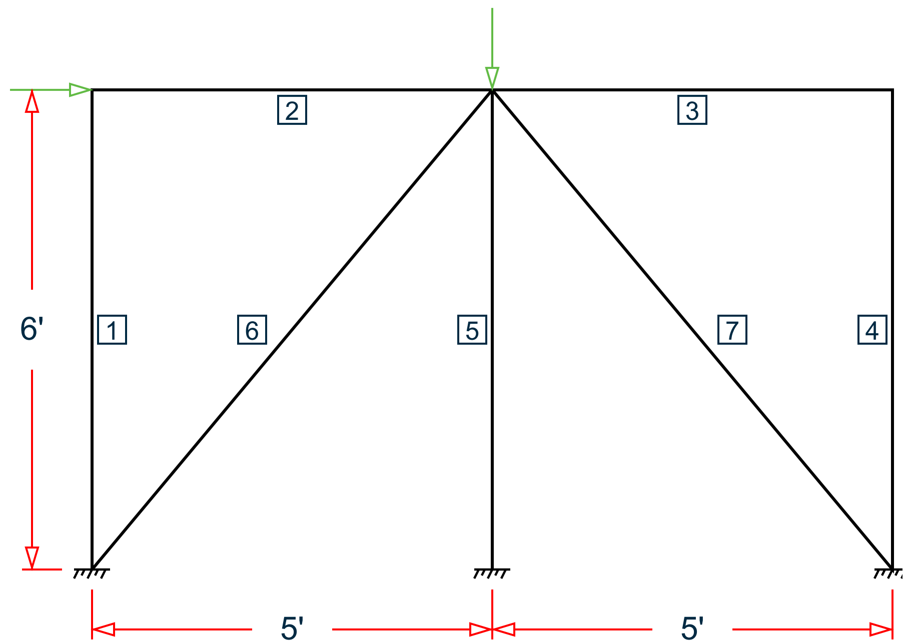


Figure 507: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = 36 ksi, F_u = 58 ksi).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

<u>Axial Tension: Member 6 (per NF-77.2211)</u>

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

<u>Axial Compression: Member 7 (NF-77.2213):</u>

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 850: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1977 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
```

```
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                           PASS      NF-77.2211        0.016         1
                    -2666.18 T          0.00          0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                  |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |
```

```
|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
 | AX.TENS: 3.52E+02                    COMPRESS:0.00E
+00                     |

|--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81              |
 | SZZ:       5.16  SYY:
11.99                                           |

|--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                             |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                              |

|--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                       |
 | CLAUSE        RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|--------------------------------------------------------------------------------
|
      7  ST   L806010                  (AISC SECTIONS)
                           PASS      NF-77.2213        0.049         1
                      7104.23 C          0.00          0.00        0.00

|--------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
```

```
INCH)                                    |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                      |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                      |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81             |
 | SZZ:       5.16  SYY:
11.99                                            |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                              |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-    |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1977 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1977 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0)\ /\ (2 \times 404) = 15.41\text{ ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$$F_y = 51.375\text{ kips}$$

$$V_y = D \times t_w = 10.8\text{ in}^2$$

$$fv,y = Fy/Vy = 4.76\text{ ksi}$$

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

$Cv = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4$ ksi

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v / F_v = 0.330$

Results

Table 851: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1977 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-77)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
                           PASS      NF-77-EQN-21      0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                               |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                              |
 | ACTUAL STRESSES: (UNIT - POUN
```

```
INCH)                                           |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                             |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                |
 | SZZ:      53.87  SYY:
3.77                                              |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                 |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                         |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.713    1    0.00E+00       -         -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04       -         -
-    |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -
-    |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1977 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1977 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,\ 300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2{}^4 - r_1{}^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2{}^3 - r_1{}^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 852: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress at mid-span (ksi)	0.084	0.0843	negligible	
Shear stress ratio at supports	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1977 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
```

```
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-77)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                   (AISC SECTIONS)
                           PASS      NF-77-EQN-21      0.339         1
                        0.00 T          0.00     -331249.97       25.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                     |
 | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                           |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                          |
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
+01               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37                |
```

```
 | SZZ:      41.10  SYY:
41.10                                            |

 |---------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                               |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00         |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                               |

 |---------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -          -          -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -          -          -
-   |
 | COMP&BEND    0.339    1    0.00E+00       -          -    3.31E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.201    1        -     2.57E+04       -          -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

 |---------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1977 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1977 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

Slenderness Ratio

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

Bending

Allowable Bending Stress About Major Axis (Z Axis)

$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$

$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$

Thus allowable bending stress is obtained by treating the section as compact.

$F_b = 0.66 \times F_Y = 23.8$ ksi

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93\ ksi < 23.8\ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7\ ksi > 23.8\ ksi$

Critical stress ratio = $38.7\ ksi\ /\ 23.8\ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01\ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.65$

$t = t_w = 0.44$ inch

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \le 0.4F_y = 14.4\ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$Vy = Dt_w = 10.68\ in^2$

$fv = Fy/Vy = 5.805$ ksi

Stress ratio = fv/Fv = 0.403

Results

Table 853: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1977 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1977
CMY 1 ALL
```

```
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-77)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                 (AISC SECTIONS)
                           FAIL      NF-77-EQN-21      1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                          |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03              |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37               |
 | SZZ:      49.69  SYY:
18.75                                            |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
```

```
1.00         |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | KS:1.000  KV:1.000  KBK:
1.000                                                    |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                        |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    1.620    1    0.00E+00       -         -    1.92E+06 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.403    1        -     6.20E+04       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1977 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1977 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in{\cdot}kips$

<u>Slenderness Ratio</u>

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

<u>Bending about Major Axis (Z Axis)</u>

Allowable Bending Stress

$b_f/2t_f = 6.862 < 65/(Fy)^{0.5}$ [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = $F_{btz} = Fy \times [0.79 - 0.002 \times (b_f/2t_f) \times \sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_z D/2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D-2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

$Cv = 1.475$

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

$Fy = 12.5$ kip, $Vy = Dt_w = 4.8$ inch^2

$fv = Fy/Vy = 2.6$ ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 854: Verification comparison for ASME_NF_3000_1977_WShaped

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	24.3	4.8%	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.954	4.6%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1977 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Mar-19
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1977
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (   ASME NF3000-77)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W16X40                   (AISC SECTIONS)
```

```
                           PASS      NF-77-EQN-21      0.954         1
                        0.00 T          0.00       -1500.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                   |
 | AXIAL: 1.51E+01  FCZ: 2.43E+01  FCY: 2.70E+01  FTZ: 2.43E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                            |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                        |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00              |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56               |
 | SZZ:      64.75  SYY:
8.26                                              |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                              |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85       |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00         |
 | KS:1.000  KV:1.000  KBK:
1.000                                            |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.954    1    0.00E+00       -         -    1.50E+03 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |
```

```
|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.1. ASME NF 3000 - 1974 & 1977 Codes* (on page 1310)

V. ASME NF 3000 1989 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1989 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

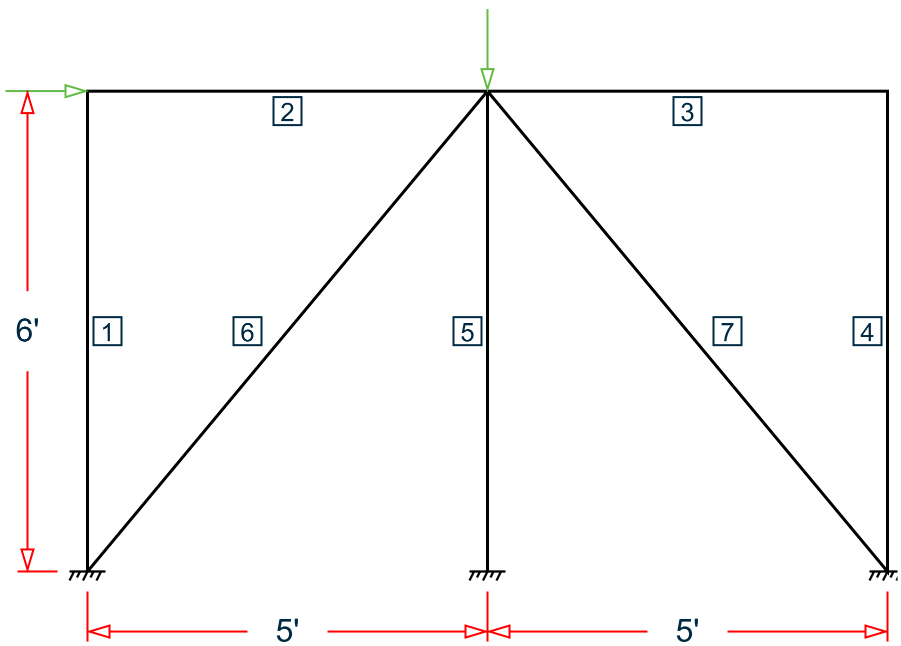


Figure 508: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = *36 ksi*, F_u = *58 ksi*).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

<u>Axial Tension: Member 6 (per NF-89.1.a)</u>

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

<u>Axial Compression: Member 7 (NF-89.1.c):</u>

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 855: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1989 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
```

```
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                         PASS      NF-89.1.a        0.016        1
                  -2666.18 T          0.00          0.00       0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                    |
 | AX.TENS: 2.16E+04                    COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AX.TENS: 3.52E+02                    COMPRESS:0.00E
+00                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81                |
 | SZZ:       5.16  SYY:
11.99                                        |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
```

```
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|--------------------------------------------------------------------------------
|
      7  ST   L806010                  (AISC SECTIONS)
                           PASS      NF-89.1.c         0.049         1
                     7104.23 C          0.00           0.00        0.00

|--------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                     |

|--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81           |
 | SZZ:       5.16  SYY:
11.99                                          |

|--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                  |

|--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-   |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
```

```
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.2. ASME NF 3000 - 1989 Code* (on page 1319)

V. ASME NF 3000 1989 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1989 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\surd(Fy)$ and $95/\surd(Fy)$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

830.0 (15.0) / (2 × 404) = 15.41 ksi

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$F_y = 51.375$ kips

$V_y = D \times t_w = 10.8\ in^2$

$fv,y = Fy/Vy = 4.76$ ksi

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

$Cv = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4$ ksi

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v/F_v = 0.330$

Results

Table 856: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1989 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
```

```
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-89)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
```

```
                           PASS      NF-89-Eqn-22      0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                  |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                            |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                       |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                 |
 | SZZ:      53.87  SYY:
3.77                                              |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.713    1    0.00E+00       -         -    8.30E+05 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.332    1        -     5.14E+04       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |
```

```
|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.2. ASME NF 3000 - 1989 Code* (on page 1319)

V. ASME NF 3000 1989 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1989 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

Bending About Major Axis (Z-Axis)

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

Shear

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2^4 - r_1^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2^3 - r_1^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 857: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress ratio	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1989 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
```

```
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-89)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                  (AISC SECTIONS)
                           PASS      NF-89-Eqn-22      0.339         1
                        0.00 T          0.00     -331249.97       25.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                         |
 | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                           |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                       |
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
+01               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
```

```
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37              |
 | SZZ:      41.10  SYY:
41.10                                            |

 |-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00          |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90            |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                    |

 |-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                        |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -          -          -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -          -          -
-   |
 | COMP&BEND    0.339    1    0.00E+00       -          -    3.31E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.201    1        -     2.57E+04        -          -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

 |-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.2. ASME NF 3000 - 1989 Code* (on page 1319)

V. ASME NF 3000 1989 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1989 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

<u>Slenderness Ratio</u>

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

<u>Bending</u>

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8\ \text{ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93\ ksi < 23.8\ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7\ ksi > 23.8\ ksi$

Critical stress ratio = $38.7\ ksi\ /\ 23.8\ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01\ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.65$

$$t = t_w = 0.44\ \text{inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4\ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$$Vy = Dt_w = 10.68\ \text{in}^2$$

$$fv = Fy/Vy = 5.805\ \text{ksi}$$

Stress ratio = fv/Fv = 0.403

Results

Table 858: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1989 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
```

```
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1989
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-89)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                 (AISC SECTIONS)
                           FAIL      NF-89-Eqn-22      1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                             |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37                |
 | SZZ:      49.69  SYY:
18.75                                           |

|-----------------------------------------------------------------------------
```

```
|
 | PARAMETER: (UNIT - POUN
INCH)                                                     |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00         |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90           |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -          -          -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -          -          -
-    |
 | COMP&BEND    1.620    1    0.00E+00       -          -    1.92E+06 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.403    1        -     6.20E+04        -          -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00         -
-    |

|------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.2. ASME NF 3000 - 1989 Code* (on page 1319)

V. ASME NF 3000 1989 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1989 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in{\cdot}kips$

<u>Slenderness Ratio</u>

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

<u>Bending about Major Axis (Z Axis)</u>

Allowable Bending Stress

$b_f/2t_f = 6.862 < 65/(Fy)^{0.5}$ [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = $F_{btz} = Fy \times [0.79 - 0.002 \times (b_f / 2t_f) \times \sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_z D / 2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

<u>Shear</u>

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D - 2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

$Cv = 1.475$

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

Fy = 12.5 kip, $Vy = Dt_w = 4.8$ inch^2

fv = Fy/Vy = 2.6 ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 859: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	25.4	0.3%	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.910	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1989 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1989
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-89)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
      1  ST   W16X40                   (AISC SECTIONS)
                           PASS      NF-89-Eqn-22      0.910         1
                        0.00 T          0.00       -1500.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                    |
 | AXIAL: 1.49E+01  FCZ: 2.54E+01  FCY: 2.70E+01  FTZ: 2.54E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                             |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                         |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56                |
 | SZZ:      64.75  SYY:
8.26                                               |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                   |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85        |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                  |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.910    1    0.00E+00       -         -    1.50E+03 0.00E
```

```
+00   |
  | TEN&BEND     0.000    0    0.00E+00       -         -     0.00E+00 0.00E
+00   |
  | SHEAR-Y      0.178    1        -      1.25E+01       -          -
-    |
  | SHEAR-Z      0.000    0        -          -     0.00E+00        -
-    |

 |----------------------------------------------------------------------------
 |
```

Related Links

- *D1.L.2. ASME NF 3000 - 1989 Code* (on page 1319)

V. ASME NF 3000 1998 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 1998 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

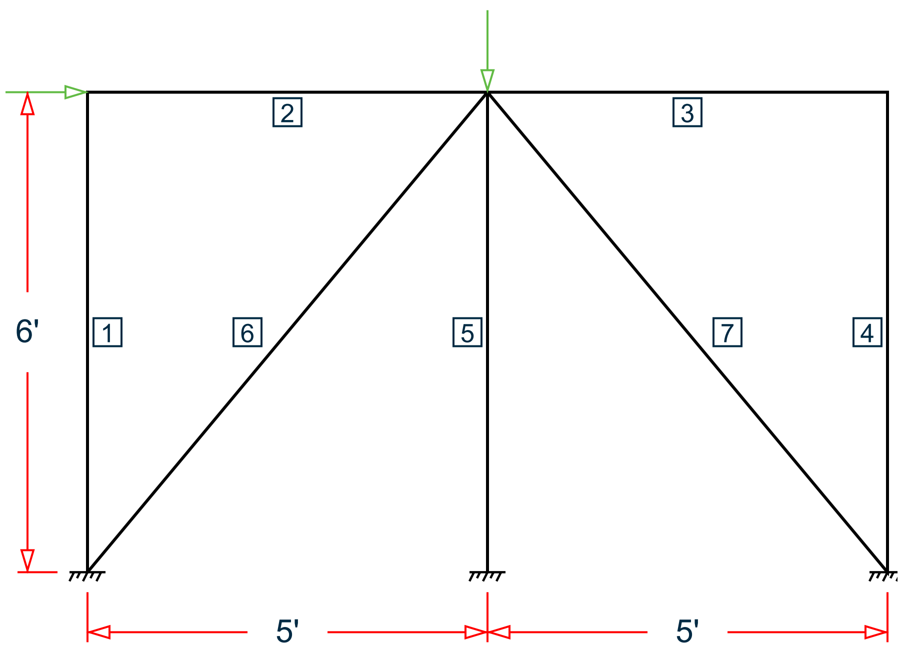


Figure 509: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = 36 ksi, F_u = 58 ksi).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

Axial Tension: Member 6 (per NF-98.1.a)

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

Axial Compression: Member 7 (NF-98.1.c):

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 860: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1998 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
```

```
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                         PASS      NF-98.1.a         0.016         1
                  -2666.18 T          0.00           0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AX.TENS: 2.16E+04                    COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AX.TENS: 3.52E+02                    COMPRESS:0.00E
+00                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81               |
 | SZZ:       5.16  SYY:
11.99                                            |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
```

```
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
      7  ST   L806010                  (AISC SECTIONS)
                            PASS      NF-98.1.c         0.049         1
                      7104.23 C          0.00           0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                             |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                  |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                    |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81               |
 | SZZ:       5.16  SYY:
11.99                                        |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                            |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
```

```
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-   |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.3. ASME NF 3000 - 1998 Code* (on page 1329)

V. ASME NF 3000 1998 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 1998 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$830.0\ (15.0)\ /\ (2 \times 404) = 15.41$ ksi

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$F_y = 51.375$ kips

$V_y = D \times t_w = 10.8\ in^2$

$fv,y = Fy/Vy = 4.76$ ksi

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

$Cv = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4$ ksi

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v/F_v = 0.330$

Results

Table 861: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1998 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE NF3000 2004
CAN 0 ALL
```

```
CB 1 ALL
CMY 0.6 ALL
CMZ 0.6 ALL
CT 0.8 ALL
DFF 0 ALL
DJ1 1 ALL
DJ2 2 ALL
DMAX 39370.1 ALL
DMIN 0 ALL
FU 60000.4 ALL
FYLD 36000.2 ALL
KY 1 ALL
KZ 1 ALL
LY 0 ALL
LZ 0 ALL
MAIN 200 ALL
NSF 0.7 ALL
PROFILE W14 ALL
RATIO 1 ALL
STIFF 5 ALL
STYPE 0 ALL
TMAIN 301 ALL
TRACK 2 ALL
UNB 0 ALL
UNT 0 ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-98)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
                           PASS      NF-98-EQN-22      0.713         1
                        0.00 T          0.00      829999.94        0.00

 |-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                  |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                        |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                       |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |

 |-----------------------------------------------------------------------------
|
```

```
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                |
 | SZZ:      53.87  SYY:
3.77                                             |

|----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -          -          -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -          -          -
-   |
 | COMP&BEND    0.713    1    0.00E+00       -          -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04        -          -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.3. ASME NF 3000 - 1998 Code* (on page 1329)

V. ASME NF 3000 1998 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1998 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,\ 300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2{}^4 - r_1{}^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2{}^3 - r_1{}^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 862: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress ratio	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1998 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
```

```
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-98)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY              MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                  (AISC SECTIONS)
                           PASS      NF-98-EQN-22      0.339         1
                        0.00 T          0.00     -331249.97       25.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                          |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
+01               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37               |
 | SZZ:      41.10  SYY:
41.10                                        |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                           |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |
```

```
|-------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                        |
 | CLAUSE        RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00        -          -          -
-    |
 | COMPRESSION  0.000    0    0.00E+00        -          -          -
-    |
 | COMP&BEND    0.339    1    0.00E+00        -          -    3.31E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00        -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.201    1        -      2.57E+04        -          -
-    |
 | SHEAR-Z      0.000    0        -           -    0.00E+00       -
-    |

|-------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.3. ASME NF 3000 - 1998 Code* (on page 1329)

V. ASME NF 3000 1998 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 1998 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

Slenderness Ratio

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

Bending

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8 \text{ ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93$ ksi < 23.8 ksi

Bending Compressive Stress = $f_{bcz} = 38.7$ ksi > 23.8 ksi

Critical stress ratio = 38.7 ksi / 23.8 ksi = 1.626

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01$ in

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/ (a/h)^2] = 5.65$

$$t = t_w = 0.44 \text{ inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4$ ksi

Hence allowable shear =14.4ksi

Actual Shear

$$Vy = Dt_w = 10.68 \text{ in}^2$$

$$fv = Fy/Vy = 5.805 \text{ ksi}$$

Stress ratio = fv/Fv = 0.403

Results

Table 863: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1998 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 1998
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-98)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY            MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                   (AISC SECTIONS)
                           FAIL      NF-98-EQN-22      1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                  |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                             |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                       |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03               |

|-------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37                |
 | SZZ:      49.69  SYY:
18.75                                              |

|-------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                  |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                   |

|-------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
```

```
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    1.620    1    0.00E+00       -         -    1.92E+06 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.403    1       -      6.20E+04       -         -
-   |
 | SHEAR-Z      0.000    0       -          -     0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.3. ASME NF 3000 - 1998 Code* (on page 1329)

V. ASME NF 3000 1998 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 1998 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in \cdot kips$

Slenderness Ratio

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)

Allowable Bending Stress

$b_f/2t_f = 6.862 < 65/(Fy)^{0.5}$ [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = $F_{btz} = Fy \times [0.79 - 0.002 \times (b_f/2t_f) \times \sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_z D/2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

Cv =1.475

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

Fy = 12.5 kip, $Vy = Dt_w = 4.8\ inch^2$

fv = Fy/Vy = 2.6 ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 864: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	25.4	0.3%	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.910	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 1998 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
```

```
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 1998
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-98)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W16X40                   (AISC SECTIONS)
                           PASS      NF-98-EQN-22      0.910         1
                        0.00 T          0.00       -1500.00        0.00

 |-----------------------------------------------------------------------------
 |
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                |
 | AXIAL: 1.49E+01  FCZ: 2.54E+01  FCY: 2.70E+01  FTZ: 2.54E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                          |
 | ACTUAL STRESSES: (UNIT - KIP
```

```
INCH)                                               |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                                 |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56                |
 | SZZ:      64.75  SYY:
8.26                                                  |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                     |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85         |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00           |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                            |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.910    1    0.00E+00       -         -    1.50E+03 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.3. ASME NF 3000 - 1998 Code* (on page 1329)

V. ASME NF 3000 2001 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 2001 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

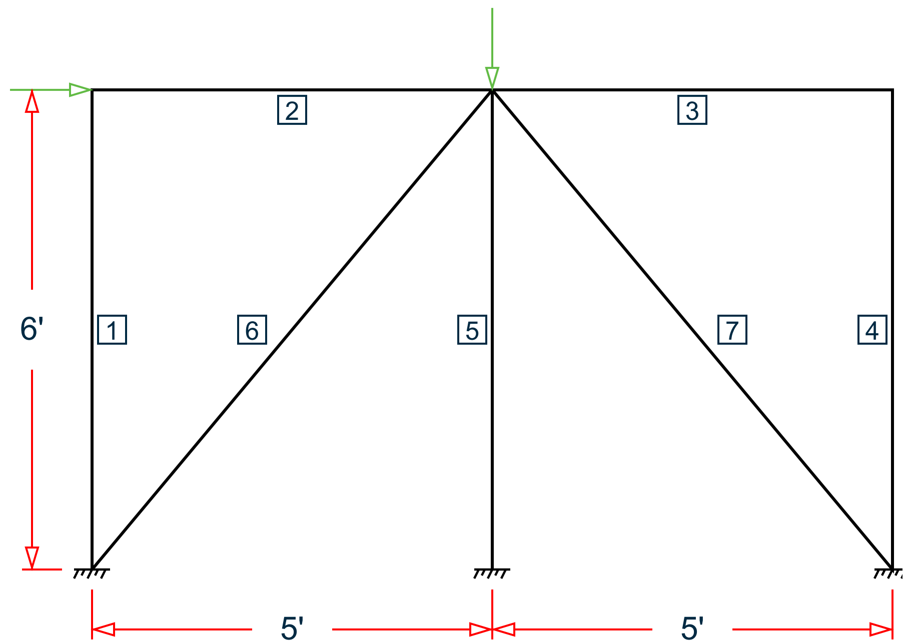


Figure 510: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel ($F_y = 36\ ksi$, $F_u = 58\ ksi$).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L\ = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

<u>Axial Tension: Member 6</u>

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

<u>Axial Compression: Member 7</u>

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 865: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2001 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
```

```
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                           PASS      NF-3322.1(a)      0.016         1
                   -2666.18 T          0.00           0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |
```

```
|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
 | AX.TENS: 3.52E+02                    COMPRESS:0.00E
+00                     |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                          |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81                 |
 | SZZ:       5.16  SYY:
11.99                                           |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00       |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90         |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                  |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|-----------------------------------------------------------------------------
|
      7  ST   L806010                 (AISC SECTIONS)
                           PASS      NF-3322.1(c)      0.049         1
                     7104.23 C         0.00           0.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00           |
 | ALLOWABLE STRESSES:   (UNIT - POUN
```

```
INCH)                                    |
 | AX.TENS: 2.16E+04                    COMPRESS:1.72E
+04                     |

 |
 |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                         |
 | AX.TENS: 0.00E+00                    COMPRESS:8.45E
+02                     |

 |-----------------------------------------------------------------------------
 |
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81                |
 | SZZ:       5.16  SYY:
11.99                                             |

 |-----------------------------------------------------------------------------
 |
 | PARAMETER: (UNIT - POUN
INCH)                                               |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

 |-----------------------------------------------------------------------------
 |
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                        |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -          -          -
-    |
 | COMPRESSION  0.049    1    7.10E+03       -          -          -
-    |
 | COMP&BEND    0.049    1    7.10E+03       -          -    0.00E+00 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.000    0        -     0.00E+00       -          -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

 |-----------------------------------------------------------------------------
 |
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2001 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 2001 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\surd(Fy)$ and $95/\surd(Fy)$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6{\times}F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0)\ /\ (2 \times 404) = 15.41\ \text{ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$$F_y = 51.375\ \text{kips}$$

$$V_y = D{\times}t_w = 10.8\ \text{in}^2$$

$$fv,y = Fy/Vy = 4.76\ \text{ksi}$$

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$t = t_w = 0.72$ inch

$Cv = 17.2$

$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4$ ksi

Hence allowable=14.4 ksi

Therefore, Ratio = $f_v / F_v = 0.330$

Results

Table 866: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2001 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PARAMETER 2
CODE NF3000 2004
CAN 0 ALL
CB 1 ALL
CMY 0.6 ALL
CMZ 0.6 ALL
CT 0.8 ALL
DFF 0 ALL
DJ1 1 ALL
DJ2 2 ALL
DMAX 39370.1 ALL
DMIN 0 ALL
FU 60000.4 ALL
FYLD 36000.2 ALL
KY 1 ALL
KZ 1 ALL
LY 0 ALL
LZ 0 ALL
MAIN 200 ALL
NSF 0.7 ALL
PROFILE W14 ALL
RATIO 1 ALL
STIFF 5 ALL
STYPE 0 ALL
TMAIN 301 ALL
TRACK 2 ALL
UNB 0 ALL
```

```
UNT 0 ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-01)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
                           PASS      NF-EQN-22         0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                            |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                |
 | SZZ:      53.87  SYY:
3.77                                            |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                        |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
```

```
 | TENSION      0.000    0    0.00E+00        -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00        -         -         -
-   |
 | COMP&BEND    0.713    1    0.00E+00        -         -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00        -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04        -         -
-   |
 | SHEAR-Z      0.000    0        -         -     0.00E+00        -
-   |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2001 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2001 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

Bending About Major Axis (Z-Axis)

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

Shear

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2^4 - r_1^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2^3 - r_1^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 867: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress at mid-span (ksi)	0.084	0.0843	negligible	
Shear stress ratio at supports	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2001 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-01)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                   (AISC SECTIONS)
                           PASS      NF-EQN-22         0.339         1
                        0.00 T          0.00     -331249.97       25.00

|-----------------------------------------------------------------------------
|
```

```
 | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                     |
 | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
+04 |
 | SHEAR: 1.44E
+04                                                         |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                          |
 | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
+01                 |

 |------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37                 |
 | SZZ:      41.10  SYY:
41.10                                               |

 |------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                   |
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                     |

 |------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.339    1    0.00E+00       -         -    3.31E+05 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.201    1        -     2.57E+04       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

 |------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2001 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2001 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

Slenderness Ratio

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

Bending

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8\ \text{ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93\ ksi < 23.8\ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7\ ksi > 23.8\ ksi$

Critical stress ratio = $38.7\ ksi\ /\ 23.8\ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01\ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.65$

$$t = t_w = 0.44\ \text{inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4\ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$Vy = Dt_w = 10.68\ in^2$

$fv = Fy/Vy = 5.805\ ksi$

Stress ratio = fv/Fv = 0.403

Results

Table 868: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2001 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2001
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-01)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                 (AISC SECTIONS)
                           FAIL      NF-EQN-22         1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                             |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
```

```
+03                 |

 |--------------------------------------------------------------------------
 |
  | SECTION PROPERTIES: (UNIT -
INCH)                                                 |
  | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37               |
  | SZZ:      49.69  SYY:
18.75                                                  |

 |--------------------------------------------------------------------------
 |
  | PARAMETER: (UNIT - POUN
INCH)                                                     |
  | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00         |
  | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90           |
  | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

 |--------------------------------------------------------------------------
 |
  | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
  | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
  | TENSION      0.000    0    0.00E+00       -         -         -
-    |
  | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
  | COMP&BEND    1.620    1    0.00E+00       -         -    1.92E+06 0.00E
+00   |
  | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
  | SHEAR-Y      0.403    1        -     6.20E+04       -         -
-    |
  | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

 |--------------------------------------------------------------------------
 |
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2001 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 2001 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in \cdot kips$

<u>Slenderness Ratio</u>

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

<u>Bending about Major Axis (Z Axis)</u>

Allowable Bending Stress

$b_f/2t_f = 6.862 < 65/(Fy)^{0.5}$ [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = $F_{btz} = Fy\times[0.79-0.002\times(b_f/2t_f)\times\sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_zD/2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

<u>Shear</u>

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

$Cv = 1.475$

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

$Fy = 12.5$ kip, $Vy = Dt_w = 4.8$ inch^2

$fv = Fy/Vy = 2.6$ ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 869: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	25.4	0.3%	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.910	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2001 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 2001
```

```
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-01)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   W16X40                  (AISC SECTIONS)
                           PASS      NF-EQN-22         0.910         1
                        0.00 T          0.00      -1500.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                |
 | AXIAL: 1.49E+01  FCZ: 2.54E+01  FCY: 2.70E+01  FTZ: 2.54E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                         |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                     |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00               |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                       |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56               |
 | SZZ:      64.75  SYY:
8.26                                           |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                              |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85       |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00         |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
```

```
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.910    1    0.00E+00       -         -    1.50E+03 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-    |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -
-    |

|------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2004 Angle

Verify the allowable stress and critical ratio of the brace members in a frame using the ASME NF 3000 2004 code.

Details

The forces due to loading in member 6 is 2,666.18 lbs (tension) and in member 7 is 7,104.23 lbs (compression). Members 6 and 7 are “truss” members (axial-only).

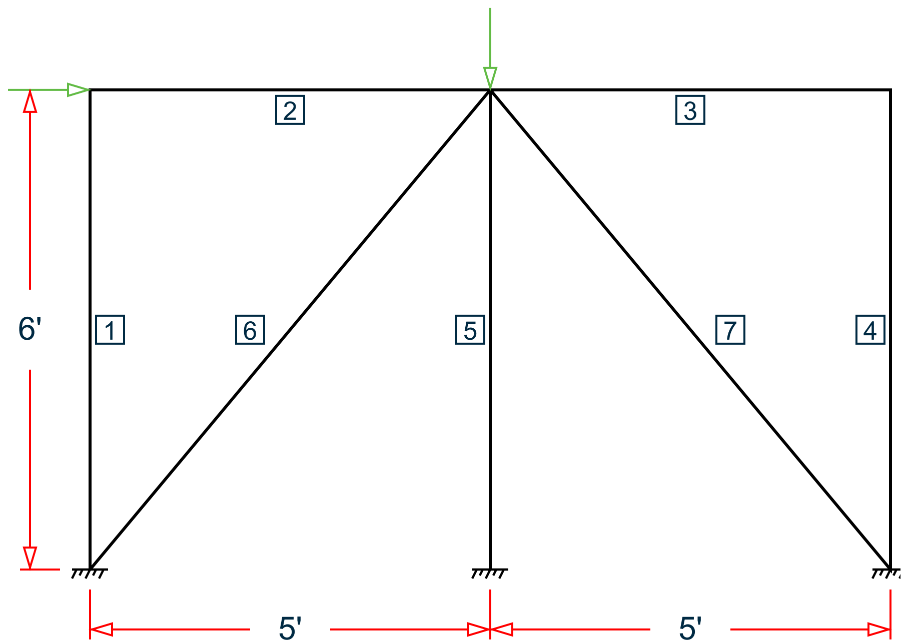


Figure 511: Space frame subjected to static loads

Both member 6 and 7 are L 80x60x10, grade A36 steel (F_y = *36 ksi*, F_u = *58 ksi*).

Assume a net section factor of 0.9, an effective length factor of 0.85, and C_t = 0.9.

Validation

$$L = \sqrt{5^2 + 6^2}(12\text{in/ft}) = 93.726\text{in}$$

Slenderness ratio (same for both members) using the minimum value of r: $kL/r_z = 0.85(93.726)/1.29 = 61.76$

For member 6, $kL/r_z = 61.76 < 300$

For member 7, $kL/r_z = 61.76 < 200$

<u>Axial Tension: Member 6</u>

Corrected area, $A_c = C_t \times A_x = 7.57\ in^2$

Actual tensile stress = $P/A = 2{,}666.18\ /\ 7.57 = 352.2\ psi$

Allowable tensile stress (Tension capacity) = $0.6 \times Fy = 0.6 \times 36 = 21.6\ ksi$

Critical ratio = 352.2/21,600 = 0.016

<u>Axial Compression: Member 7</u>

Actual compressive stress = $P/A = 7{,}104.23\ /\ 8.41 = 844.74\ psi$

$$C_c' = \sqrt{\frac{2\pi^2 E}{Q_s Q_a F_y}} = 126.2 \qquad \text{(NF-3322.2(e)(5))}$$

$(kL/r)min < C_c'$, allowable compression stress, $F_a = 17.2\ ksi$

Critical ratio = $844.74\ /\ 17{,}200 = 0.049$

Results

Table 870: Comparison of results

Parameter		Hand Calculation	STAAD.Pro	Difference	Comments
Member 6 (tension)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	352	352	none	
	Allowable tensile stress (psi)	21,600	21,600	none	
	Critical ratio	0.016	0.016	none	
Member 7 (compression)	Slenderness ratio	61.76	61.76	none	
	Actual tensile stress (psi)	845	845	none	
	Allowable tensile stress (psi)	17,200	17,200	none	
	Critical ratio	0.049	0.049	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 Angle.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 26-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0; 3 120 0 0; 4 0 72 0; 5 120 72 0; 6 60 72 0;
MEMBER INCIDENCES
1 1 4; 2 4 6; 3 6 5; 4 5 3; 5 6 2; 6 1 6; 7 6 3;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+07
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-05
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
1 TO 5 TABLE ST W8X48
6 7 TABLE ST L806010
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 TO 3 FIXED
MEMBER TRUSS
6 7
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
4 FX 7500
6 FY -10000
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES LIST 6 7
SECTION 0 0.25 0.5 0.75 1 MEMB 6 7
PRINT MEMBER SECTION FORCES LIST 6 7
PRINT MEMBER FORCES LIST 6 7
PRINT MEMBER STRESSES LIST 6
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
CT 0.9 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
      6  ST   L806010                  (AISC SECTIONS)
                          PASS      NF-3322.1(a)      0.016         1
                    -2666.18 T          0.00           0.00        0.00

|-------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  72.65 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

|
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                        |
```

```
 | AX.TENS: 3.52E+02                     COMPRESS:0.00E
+00                     |

 |--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                         |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81              |
 | SZZ:       5.16  SYY:
11.99                                         |

 |--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                             |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00       |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90         |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

 |--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.016    1    2.67E+03       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | TEN&BEND     0.016    1    2.67E+03       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

 |--------------------------------------------------------------------------------
|
      7  ST   L806010                  (AISC SECTIONS)
                           PASS      NF-3322.1(c)      0.049         1
                     7104.23 C          0.00           0.00        0.00

 |--------------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  61.76 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                   |
 | AX.TENS: 2.16E+04                     COMPRESS:1.72E
+04                     |

 |
```

```
|
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                              |
 | AX.TENS: 0.00E+00                     COMPRESS:8.45E
+02                     |

|--------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                                |
 | AXX:   8.41  AYY:   3.33  AZZ:   2.50  RZZ:  1.29  RYY:
2.81                 |
 | SZZ:       5.16  SYY:
11.99                                                |

|--------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                    |
 | KL/R-Z: 61.76   KL/R-Y: 28.34   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 0.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.90   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|--------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.049    1    7.10E+03       -         -         -
-   |
 | COMP&BEND    0.049    1    7.10E+03       -         -    0.00E+00 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.000    0        -     0.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -
-   |

|--------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2004 Channel

Verify the design of channel section used for a beam member using the ASME NF 3000 2004 code.

Details

The beam is a propped cantilever 80 inches long. The beam is loaded with a 2 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is C15x50 channel section with Grade A36 steel.

Validation

Maximum shear force (at fixed end): V = 51.375 kips (taken from STAAD.Pro analysis)

Maximum moment (at fixed end): M = 830.0 in-kip (taken from STAAD.Pro analysis)

Slenderness Ratio

About z-axis: $(KL/r)_z = 0.85(80\ in)/(5.25\ in) = 12.95$

About y-axis: $(KL/r)_y = 0.85(80\ in)/(0.865\ in) = 78.61$

Max. slenderness ratio < 300, OK.

Bending About Major Axis (Z Axis)

$$\frac{b_f}{t_f} = 4.62 < \frac{65}{\sqrt{F_y}} = 10.83$$

The laterally unsupported length of the compression flange, $L_b = 80 < \frac{76 b_f}{\sqrt{F_y}} = 120$

and $< \frac{20,\ 000}{(d\ /\ A_f)F_y} = 89.56$

As b_f/t_f is not in the range of $65/\sqrt{(Fy)}$ and $95/\sqrt{(Fy)}$, the value of allowable bending compressive and tensile stress = $F_{bty} = F_{bcy} = 0.6 \times F_y = 21.6\ ksi$

Actual bending stress:

Bending Tensile Stress = bending compressive stress = $f_{btz} = f_{bcz} = MzD\ /\ (2I_z) = 15.4\ ksi$

$$830.0\ (15.0)\ /\ (2 \times 404) = 15.41\ \text{ksi}$$

Critical ratio = 15.41 ksi / 21.6 ksi = 0.713

Shear

Actual shear:

$$F_y = 51.375\ \text{kips}$$

$$V_y = D \times t_w = 10.8\ \text{in}^2$$

$$fv,y = Fy/Vy = 4.76\ \text{ksi}$$

Allowable shear:

a = clear distance between transverse stiffeners (here taken as length) = 80 in

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.35\ in$

Thus, $a/h = 5.57$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.46$

$$t = t_w = 0.72\ \text{inch}$$

$$Cv = 17.2$$

$$F_v = (F_y/2.89)C_v \geq 0.4 \times F_y = 14.4 \text{ ksi}$$

Hence allowable=14.4 ksi

Therefore, Ratio = f_v / F_v = *0.330*

Results

Table 871: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	78.61	78.61	none	
Allowable bending stress (ksi)	21.6	21.6	none	
Actual bending stress (ksi)	15.4	15.4	none	
Critical ratio	0.713	0.713	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	4.76	4.78	0.4%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 Channel.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 80 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
1 TABLE ST C15X50
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
2 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 40
1 UNI GY -1000 0 80
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 80 ALL
UNB 80 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      1  ST   C15X50                   (AISC SECTIONS)
                           PASS      NF-EQN-22         0.713         1
                        0.00 T          0.00      829999.94        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  78.61 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                |
 | AXIAL: 1.55E+04  FCZ: 2.16E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                           |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                     |
 | AXIAL: 0.00E+00  FBZ: 1.54E+04  FBY: 0.00E+00 SHEAR: 4.78E
+03               |
```

```
|--------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                              |
 | AXX:  14.70  AYY:  10.74  AZZ:   3.22  RZZ:  5.24  RYY:
0.87                 |
 | SZZ:      53.87  SYY:
3.77                                               |

|--------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                                  |
 | KL/R-Z: 12.97   KL/R-Y: 78.61   UNL:   80.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|--------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX         VY         VZ         MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -          -          -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -          -          -
-   |
 | COMP&BEND    0.713    1    0.00E+00       -          -    8.30E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -          -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.332    1        -     5.14E+04        -          -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|--------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2004 Pipe

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2004 code.

Details

The simply supported beam is 50 inches long. The beam is loaded with a 1.5 kip concentrated load at midspan and a uniformly distributed load of 1 kip/in over the entire length.

The member is PIPS 120 pipe section with Grade A36 steel.

Validation

Maximum shear force, $V = \frac{P}{2} + \frac{wL}{2} = \frac{1.5}{2} + \frac{1.0(50)}{2} = 25.75 kips$

Maximum moment, $M = \frac{PL}{4} + \frac{wL^2}{8} = \frac{1.5(50)}{4} + \frac{1.0(50)^2}{8} = 331.25 in \cdot kip$

Slenderness ratio = (KL/r) = 9.72 < 300

<u>Bending About Major Axis (Z-Axis)</u>

Section is compact: $\frac{OD}{t} = 36.43 < \frac{3,\ 300}{F_y} = 91.67$

Allowable bending stress: $F_b = 0.66 \times F_y = 23.76\ ksi$

Actual bending stress: $f_{btz} = f_{bcz} = \frac{M_z}{I_z}\left(\frac{OD}{2}\right) = \frac{331.25}{262}\left(\frac{12.75}{2}\right) = 8.06 ksi$

Stress ratio: 0.339

<u>Shear</u>

From the reference, the shear area:

$$A_y = \frac{0.25\pi\left(r_2^4 - r_1^4\right)2\left(r_2 - r_1\right)}{\frac{2}{3}\left(r_2^3 - r_1^3\right)}$$

where

r_1	=	inner radius
r_2	=	outer radius

Shear area: $A_y = 6.8\ in^2$

Actual shear stress (at supports): $f_v = 25.75\ /\ 6.8 = 3.78\ ksi$

Actual shear stress (at midspan): $f_v = 0.75\ /\ 6.8 = 0.110\ ksi$

Allowable shear stress, $F_v = 0.4 \times F_y = 14.4\ ksi$

Stress ratio: 3.78/14.4 = 0.263

Results

Table 872: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	9.72	9.72	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	8.06	8.06	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Critical stress ratio	0.339	0.339	none	
Allowable shear stress (ksi)	14.4	14.4	none	
Shear stress at mid-span (ksi)	0.084	0.0843	negligible	
Shear stress ratio at supports	0.201	0.201	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 Pipe.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 17-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 50 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MY MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -1500 25 0
1 UNI GY -1000 0 50
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
```

```
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 0.85 ALL
KZ 0.85 ALL
UNT 50 ALL
UNB 50 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
UNB 25 ALL
UNT 25 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
      1  ST   PIPS120                   (AISC SECTIONS)
                           PASS      NF-EQN-22         0.339         1
                        0.00 T          0.00     -331249.97       25.00

 |--------------------------------------------------------------------------------
 |
  | SLENDERNESS CHECK:    ACTUAL RATIO:   9.72 ALLOWABLE RATIO:
 300.00          |
  | ALLOWABLE STRESSES:   (UNIT - POUN
 INCH)                                  |
  | AXIAL: 2.12E+04  FCZ: 2.38E+04  FCY: 2.38E+04  FTZ: 2.38E+04  FTY: 2.38E
 +04 |
  | SHEAR: 1.44E
 +04                                                             |
  | ACTUAL STRESSES: (UNIT - POUN
 INCH)                                       |
  | AXIAL: 0.00E+00  FBZ: 8.06E+03  FBY: 0.00E+00 SHEAR: 8.43E
 +01               |

 |--------------------------------------------------------------------------------
 |
  | SECTION PROPERTIES: (UNIT -
 INCH)                                         |
  | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
 4.37             |
  | SZZ:      41.10  SYY:
 41.10                                         |

 |--------------------------------------------------------------------------------
 |
  | PARAMETER: (UNIT - POUN
 INCH)                                             |
```

```
 | KL/R-Z:  9.72   KL/R-Y:  9.72   UNL:   25.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90         |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                      |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    0.339    1    0.00E+00       -         -    3.31E+05 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.201    1        -     2.57E+04       -         -
-    |
 | SHEAR-Z      0.000    0        -         -    0.00E+00       -
-    |

|-----------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2004 Tee

Verify the design of a pipe section used for a beam member per the ASME NF 3000 2004 code.

Details

A cantilever beam is 60 in long. The beam is loaded at the free end with a 2 kip load and a uniform load of 1 kip/in along the entire length.

The member is a WT18x67.5 section with Grade A36 steel.

Validation

Maximum shear force (at support) = $V = wL + P = (1\ kip/in) \times 60\ in + 2\ kip = 62\ kips$

Maximum moment (at support) = $M = 0.5 \times w \times L^2 + P \times L = 0.5\ (1\ kip/in) \times (60\ in)^2 + (2\ kip) \times (60\ in) = 1{,}920\ in \cdot kips$

Slenderness Ratio

Slenderness Ratio along Z-Axis = $(KL/r)_z = 21.22$

Slenderness Ratio along Y-Axis = $(KL/r)_y = 50.53$

Maximum Slenderness Ratio = $50.53 < 300$

Bending

Allowable Bending Stress About Major Axis (Z Axis)

$$b_f/2t_f = 4.65 > 65/\sqrt{(F_Y)} = 10.833$$

$$d/t_w = 28.3 < 640/\sqrt{(F_Y)} = 106.67$$

Thus allowable bending stress is obtained by treating the section as compact.

$$F_b = 0.66 \times F_Y = 23.8 \text{ ksi}$$

Actual Bending Stress

Bending Tensile Stress = $f_{btz} = (Mz/Iz) \times Y_{bar} = 14.93\ ksi < 23.8\ ksi$

Bending Compressive Stress = $f_{bcz} = 38.7\ ksi > 23.8\ ksi$

Critical stress ratio = $38.7\ ksi\ /\ 23.8\ ksi = 1.626$

Shear

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = 60 in

h = clear distance flanges of the girder = $D\text{-}t_f = 17.01\ in$

Thus, a/h = 3.527 Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.65$

$$t = t_w = 0.44 \text{ inch}$$

If Cv > 0.8, $Cv = [190/(h/t)]\sqrt{(k/Sy)} = 78.4$ which is true. Thus, the value of Cv is taken as 78.4

Allowable shear stress = $Fv = (FY/2.89)Cv \leq 0.4F_y = 14.4\ ksi$

Hence allowable shear =14.4ksi

Actual Shear

$$Vy = Dt_w = 10.68 \text{ in}^2$$

$$fv = Fy/Vy = 5.805 \text{ ksi}$$

Stress ratio = fv/Fv = 0.403

Results

Table 873: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	50.53	50.53	none	
Allowable bending stress (ksi)	23.8	23.8	none	
Actual bending stress (ksi)	38.7	38.5	0.5%	
Critical shear ratio	1.626	1.620	0.4%	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	5.81	5.81	none	
Shear stress ratio	0.403	0.403	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 Tee.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 03-Aug-18
END JOB INFORMATION
SET SHEAR
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 60 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.9e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
TYPE STEEL
STRENGTH FY 36000 FU 58000 RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TABLE T W36X135
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
1 CON GY -2000 60
1 UNI GY -1000 0 60
PERFORM ANALYSIS
PARAMETER 1
CODE NF3000 2004
CMY 1 ALL
CMZ 1 ALL
CT 0.85 ALL
FU 58000 ALL
FYLD 36000 ALL
KY 2 ALL
```

```
KZ 2 ALL
UNT 60 ALL
UNB 60 ALL
MAIN 200 ALL
NSF 0.9 ALL
RATIO 1 ALL
TMAIN 300 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************
 ALL UNITS ARE - POUN INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
 *    1  T    W36X135                  (AISC SECTIONS)
                           FAIL      NF-EQN-22         1.620         1
                        0.00 T          0.00     1919999.75        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  50.53 ALLOWABLE RATIO:
300.00          |
 | ALLOWABLE STRESSES:   (UNIT - POUN
INCH)                                 |
 | AXIAL: 1.16E+04  FCZ: 2.38E+04  FCY: 2.16E+04  FTZ: 2.16E+04  FTY: 2.16E
+04 |
 | SHEAR: 1.44E
+04                                                             |
 | ACTUAL STRESSES: (UNIT - POUN
INCH)                                      |
 | AXIAL: 0.00E+00  FBZ: 3.85E+04  FBY: 0.00E+00 SHEAR: 5.81E
+03              |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                        |
 | AXX:  19.95  AYY:   9.61  AZZ:   6.32  RZZ:  5.66  RYY:
2.37                |
 | SZZ:      49.69  SYY:
18.75                                               |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - POUN
INCH)                                              |
 | KL/R-Z: 21.22   KL/R-Y: 50.53   UNL:   60.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD: 36000.00   FU: 58000.00   NET SECTION FACTOR:
0.90          |
 | CT:  0.85   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                 |
```

```
|--------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS POUN-
INCH)                          |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY    |
 | TENSION      0.000    0    0.00E+00       -         -         -
-    |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-    |
 | COMP&BEND    1.620    1    0.00E+00       -         -     1.92E+06 0.00E
+00   |
 | TEN&BEND     0.000    0    0.00E+00       -         -     0.00E+00 0.00E
+00   |
 | SHEAR-Y      0.403    1        -     6.20E+04       -         -
-    |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-    |

|--------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V. ASME NF 3000 2004 WShaped

Verify the design of a wide-flange section used for a cantilever beam using the ASME NF 3000 2004 code.

Details

The beam is a 10 ft cantilever with a 12.5 kip concentrated load at the free end.

The member is a W16X40 section with Grade A36 steel.

Validation

Maximum shear force (constant shear): $V = 12.5\ kips$

Maximum moment (at fixed end): $M = 12.5\ kips \times 120\ in = 1{,}500\ in \cdot kips$

Slenderness Ratio

$(KL/r)_z$ = Slenderness Ratio along Z-Axis = 18.11

$(KL/r)_y$ = Slenderness Ratio along Y-Axis = 76.68

Maximum Slenderness Ratio = 76.68 < 240 (Hence OK)

Bending about Major Axis (Z Axis)

Allowable Bending Stress

$b_f/2t_f = 6.862 < 65/(Fy)^{0.5}$ [Ref. NF3322.1(d)(2)]

Thus, allowable bending tensile stress = $F_{btz} = Fy \times [0.79 - 0.002 \times (b_f/2t_f) \times \sqrt{Fy}] = 25.47\ ksi$

Actual Bending Stress

Bending Tensile Stress = Bending Compressive Stress = $f_{btz} = f_{bcz} = M_z D / 2I_z = 23.17\ ksi$

Stress ratio = $23.16\ ksi\ /\ 25.47\ ksi = 0.909$

<u>Shear</u>

Allowable Shear Stress

a = clear distance between transverse stiffeners (here taken as length) = $120\ in$

h = clear distance flanges of the girder = $D\text{-}2t_f = 14.91\ in$

Thus, $a/h = 8.05$, Therefore, $k = 5.34 + [4.00/\ (a/h)^2] = 5.36$ [Ref. NF3322.6(e)(2)]

$t = t_w = 0.3$ inch

Cv =1.475

Allowable shear stress = $Fv = (Fy/2.89)Cv => 0.4Fy = 14.4\ ksi$ Hence allowable=14.4ksi

Actual Shear Stress

Fy = 12.5 kip, $Vy = Dt_w = 4.8\ \text{inch}^2$

fv = Fy/Vy = 2.6 ksi

Stress ratio = $fv/Fv = 0.18$

Results

Table 874: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Slenderness ratio	76.68	76.68	none	
Allowable bending stress (ksi)	25.47	25.4	0.3%	
Actual bending stress (ksi)	23.2	23.2	none	
Critical stress ratio	0.909	0.910	negligible	
Allowable shear stress (ksi)	14.4	14.4	none	
Actual shear stress (ksi)	2.6	2.56	1.5%	
Shear stress ratio	0.18	0.178	1.1%	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 W Shaped.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 18-Jun-18
END JOB INFORMATION
SET SHEAR
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
MEMBER PROPERTY AMERICAN
1 TABLE ST W16X40
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC STEEL_36_KSI
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FEET KIP
CONSTANTS
MATERIAL STEEL_36_KSI ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY 12.5
PERFORM ANALYSIS
UNIT INCHES KIP
PARAMETER 1
CODE NF3000 2004
MAIN 2 ALL
FYLD 36 ALL
UNB 72 ALL
UNT 72 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
```

```
      1  ST   W16X40                      (AISC SECTIONS)
                          PASS      NF-EQN-22         0.910         1
                       0.00 T          0.00       -1500.00        0.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  76.68 ALLOWABLE RATIO:
240.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                    |
 | AXIAL: 1.49E+01  FCZ: 2.54E+01  FCY: 2.70E+01  FTZ: 2.54E+01  FTY: 2.70E
+01 |
 | SHEAR: 1.44E
+01                                                             |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                         |
 | AXIAL: 0.00E+00  FBZ: 2.32E+01  FBY: 0.00E+00 SHEAR: 2.56E
+00                |

|-----------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                           |
 | AXX:  11.80  AYY:   4.88  AZZ:   4.71  RZZ:  6.63  RYY:
1.56                 |
 | SZZ:      64.75  SYY:
8.26                                               |

|-----------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                  |
 | KL/R-Z: 18.11   KL/R-Y: 76.68   UNL:   72.0   CMZ:  0.85   CMY:
0.85        |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00          |
 | CT:  0.75   STEEL TYPE:  0.0  KS:1.000  KV:1.000  KBK:
1.000                  |

|-----------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                       |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.000    0    0.00E+00       -         -         -
-   |
 | COMP&BEND    0.910    1    0.00E+00       -         -    1.50E+03 0.00E
+00  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.178    1        -     1.25E+01       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |
```

```
|--------------------------------------------------------------------------------
|
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V.ASME NF 3000 2004 STYPE 1 Pipe

Verify the design results an austenitic stainless steel pipe section, designed as per ASME NF 3000 2004.

Details

A beam (beam no. 7 in the model) of length 10 ft is loaded with a uniformly distributed load of 1 kip/ft over the whole span, and axial compressive loads of 100 kip at both ends. The beam is assigned with a steel pipe section of PIPS120 and is designed in accordance with ASME NF3000 2004.

The austenitic stainless steel has a yield strength of 36 ksi, an ultimate rupture strength of 58 ksi, and a modulus of elasticity of 29,000 ksi.

Validation

From the analysis:

$F_x = 100.145\ kip$
$F_y = 5\ kip$ (at ends)
$M_z = 97.683\ in \cdot k$ (at mid-span)
$M_y = 10.798\ in \cdot k$

(KL/r)z = Slenderness Ratio along Z-Axis = $27.44 = (KL/r)_y$ = Slenderness Ratio along Y-Axis < 200 Hence OK (*Refer NF-3322.2(c)(1)*)

<u>Check Against Axial Compression</u>

As per clause NF 3322.2(d)(2)(c),

$$OD/t_w = 12.75/0.35 = 36.429 < 3{,}300/F_y = 91.667$$

Since $(KL/r)_y = 27.44 = (kl/r)_z < 120$

As per NF 3322.1(c)(2)(a),

Allowable compressive stress, $F_a = F_y\left(0.47 - \frac{kl\ /\ r}{444}\right) = 14.7\ ksi$

Actual compressive stress $f_a = F_x / A_x = 7.31\ ksi$

$$\text{Ratio} = 0.497$$

<u>Check Against Bending</u>

Allowable Stress along Major Axis (Z-Axis):

Since as per clause NF 3322.1(d)(1)(a)(7),

$$OD/t_w = 12.75/0.35 = 36.429 < 3{,}300/F_y = 91.67$$

Thus, allowable bending stress is obtained by treating the section as compact.

$\text{Fbcz} = 0.66\times F_y = 23.76\ \text{ksi} = \text{Fbtz}$

Allowable Stress along Minor Axis (Y-Axis):

Here also, allowable stress along minor axis is given as *Fbty=0.66xFYLD = 23.8 ksi = Fbcy*

Actual Bending Stress along Major Axis (Z-Axis):

$\text{Mz} = 97.683\ \text{in}\cdot\text{k}$

Bending Tensile Stress = *fbtz = (Mz/Iz)×(OD/2) = 2.38 ksi < 23.8 ksi*, Hence OK.

Bending Compressive Stress = *fbcz = 2.38 ksi < 23.8 ksi*, Hence safe.

Actual Bending Stress along Minor Axis (Y-Axis):

$\text{My} = 10.798\ \text{in}\cdot\text{k}$

Bending Tensile Stress = *fbty = (Mz/Iz)×(OD/2) = 0.263 ksi < 23.8 ksi*, Hence OK.

Bending Compressive Stress = *fbcy = 0.263 ksi < 23.8 ksi* , Hence safe.

<u>Check for Combined Axial Compression and Bending</u>

Cmz = Cmy = 1 (as provided in the input)

Now, *fa = 7.31 ksi*, *Fa = 14.7 ksi*, *fbcz = 2.38 ksi*, *Fbcz = 23.8 ksi*, *fbcy = 0.263 ksi*, *Fbcy = 23.8 ksi*

Since *fa/ Fa = 7.31 / 14.7 = 0.497 > 0.15*

$$F_{ez}' = \frac{\pi^2 \times E}{2.15 \times \left(\frac{kl}{r}\right)_z^2} = 176.804\ \text{ksi} \qquad \text{(NF 3322.1(e)(1))}$$

$$F_{ey}' = \frac{\pi^2 \times E}{2.15 \times \left(\frac{kl}{r}\right)_y^2} = 176.804\ \text{ksi} \qquad \text{(NF 3322.1(e)(1))}$$

From Eqn. 20:

$$\frac{f_a}{F_a} + \frac{C_{mz} \times f_{bz}}{\left(1 - \frac{fa}{F'_{ez}}\right) \times F_{bz}} + \frac{C_{my} \times f_{by}}{\left(1 - \frac{fa}{F'_{ey}}\right) \times F_{by}} = 0.613 < 1$$, Hence safe (critical ratio)

From Eqn. 21:

$$\frac{f_a}{0.6F_y} + \frac{f_{bz}}{F_{bz}} + \frac{f_{by}}{F_{by}} = 0.449 < 1$$, Hence safe

<u>Check Against Shear</u>

Actual Shear along Major Axis:

$V_y = 2\times \text{OD} \times t_w = 8.925\ \text{in}^2$

$f_{vy} = 5\ \text{kip} / 8.925\ \text{in}^2 = 0.56\ \text{ksi}$

(No shear along minor axis; $f_{vz} = 0$)

Allowable Shear:

As per NV 3322.1(b)(1), allowable shear stress, $F_v = 0.4\times F_y = 14.4\ ksi$

Ratio = 0.039

Results

Table 875: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Slenderness ratio	27.44	27.44	none	
Actual compressive stress (ksi)	7.31	7.31	none	
Allowable compressive stress (ksi)	14.7	14.7	none	
Actual bending stress; major axis (ksi)	2.38	2.38	none	
Allowable bending stress, major axis (ksi)	23.8	23.8	none	
Actual bending stress, minor axis (ksi)	0.263	0.263	none	
Allowable bending stress, minor axis (ksi)	23.8	23.8	none	
Combined interaction ratio	0.613	0.613	none	
Shear stress, major axis (ksi)	0.039	0.039	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\09 Steel Design\US\ASME\ASME NF 3000 2004 STYPE 1 Pipe.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Mar-19
END JOB INFORMATION
****************************************************************************
*This problem has been created to validate the program calculated design
*results of ASME NF 3000 2004 pipe section with STYPE 1 parameter
****************************************************************************
INPUT WIDTH 79
SET SHEAR
UNIT FEET KIP
```

```
JOINT COORDINATES
1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10 10;
8 10 0 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 TO 8 TABLE ST PIPS120
CONSTANTS
MATERIAL STEEL ALL
SUPPORTS
1 4 5 8 FIXED
LOAD 1 LOADTYPE Dead  TITLE DL-1
JOINT LOAD
6 FX 100
7 FX -100
LOAD 2 LOADTYPE Dead  TITLE DL-2
MEMBER LOAD
2 4 5 7 UNI GY -1
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.0 2 1.0
PERFORM ANALYSIS
UNIT INCHES KIP
PRINT MEMBER FORCES LIST 7
SECTION 0.25 0.5 0.75
PRINT MEMBER SECTION FORCES LIST 7
PRINT MATERIAL PROPERTIES LIST 7
PRINT MEMBER PROPERTIES LIST 7
LOAD LIST 3
PARAMETER 1
CODE NF3000 2004
STYPE 1 ALL
CMZ 1 ALL
CMY 1 ALL
TRACK 2 ALL
CHECK CODE MEMB 7
FINISH
```

STAAD.Pro Output

```
                                                                  PAGE NO.
1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
```

```
             *           Date=    MAR 24, 2022                 *
             *           Time=    10: 9:42                     *
             *                                                 *
             *  Licensed to: Bentley Systems Inc               *
             ***************************************************
     1. STAAD SPACE
INPUT FILE: ASME NF 3000 2004 STYPE 1 Pipe.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 20-MAR-19
     4. END JOB INFORMATION
     6.
*****************************************************************************
     7. *THIS PROBLEM HAS BEEN CREATED TO VALIDATE THE PROGRAM CALCULATED
DESIGN
     8. *RESULTS OF ASME NF 3000 2004 PIPE SECTION WITH STYPE 1 PARAMETER
     9.
*****************************************************************************
    11. INPUT WIDTH 79
    12. SET SHEAR
    13. UNIT FEET KIP
    14. JOINT COORDINATES
    15. 1 0 0 0; 2 0 10 0; 3 10 10 0; 4 10 0 0; 5 0 0 10; 6 0 10 10; 7 10 10
10
    16. 8 10 0 10
    17. MEMBER INCIDENCES
    18. 1 1 2; 2 2 3; 3 3 4; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 7 8
    19. DEFINE MATERIAL START
    20. ISOTROPIC STEEL
    21. E 4.176E+06
    22. POISSON 0.3
    23. DENSITY 0.489024
    24. ALPHA 6.5E-06
    25. DAMP 0.03
    26. TYPE STEEL
    27. STRENGTH RY 1.5 RT 1.2
    28. END DEFINE MATERIAL
    29. MEMBER PROPERTY AMERICAN
    30. 1 TO 8 TABLE ST PIPS120
    31. CONSTANTS
    32. MATERIAL STEEL ALL
    33. SUPPORTS
    34. 1 4 5 8 FIXED
    35. LOAD 1 LOADTYPE DEAD  TITLE DL-1
    36. JOINT LOAD
    37. 6 FX 100
    38. 7 FX -100
    39. LOAD 2 LOADTYPE DEAD  TITLE DL-2
    40. MEMBER LOAD
      STAAD SPACE                                              -- PAGE NO.
2
    41. 2 4 5 7 UNI GY -1.
    42. LOAD COMB 3 COMBINATION LOAD CASE 3
    43. 1 1.0 2 1.0
    44. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          8  NUMBER OF MEMBERS       8
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
```

```
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      24
   TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
    45. UNIT INCHES KIP
    46. PRINT MEMBER FORCES LIST 7
  MEMBER   FORCES   LIST     7
      STAAD SPACE                                              -- PAGE NO.
3
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KIP  INCH     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      7    1     6     99.31     -0.00    -0.00      0.00     10.80     -14.22
                 7    -99.31      0.00     0.00     -0.00    -10.80      14.22
           2     6      0.83      5.00    -0.00     -0.00      0.00      66.53
                 7     -0.83      5.00     0.00      0.00      0.00     -66.53
           3     6    100.14      5.00    -0.00      0.00     10.80      52.32
                 7   -100.14      5.00     0.00     -0.00    -10.80     -52.32
   ************** END OF LATEST ANALYSIS RESULT **************
    47. SECTION 0.25 0.5 0.75
    48. PRINT MEMBER SECTION FORCES LIST 7
  MEMBER   SECTION  FORCES   LIST
      STAAD SPACE                                              -- PAGE NO.
4
   MEMBER FORCES AT INTERMEDIATE SECTIONS
   --------------------------------------
   ALL UNITS ARE -- KIP  INCH
 MEMB  LOAD  SEC       AXIAL   SHEAR-Y   SHEAR-Z       MOM-X       MOM-
Y       MOM-Z
     7    1  0.25      99.31     -0.00     -0.00        0.00       10.80
-14.22
             0.50      99.31     -0.00     -0.00        0.00       10.80
-14.22
             0.75      99.31     -0.00     -0.00        0.00       10.80
-14.22
          2  0.25       0.83      2.50     -0.00       -0.00        0.00
-45.97
             0.50       0.83      0.00     -0.00       -0.00       -0.00
-83.47
             0.75       0.83     -2.50     -0.00       -0.00       -0.00
-45.97
          3  0.25     100.14      2.50     -0.00        0.00       10.80
-60.18
             0.50     100.14     -0.00     -0.00        0.00       10.80
-97.68
             0.75     100.14     -2.50     -0.00        0.00       10.80
-60.18
   ************** END OF LATEST ANALYSIS RESULT **************
    49. PRINT MATERIAL PROPERTIES LIST 7
  MATERIAL PROPERTI LIST     7
      STAAD SPACE                                              -- PAGE NO.
5
   MATERIAL PROPERTIES.
   --------------------
   ALL UNITS ARE - KIP  INCH
  MATERIAL
```

```
           KIND          E           POIS        DENS       ALPHA
DAMP        G
   STEEL
             1D    2.90000E+04     0.300   2.83000E-04 6.50000E-06
0.030   1.11538E+04
      STAAD SPACE                                          -- PAGE NO.
6
   MATERIAL PROPERTIES.
   --------------------
   ALL UNITS ARE - KIP  INCH
   MEMBER           E            G          DEN         ALPHA
      7          29000.0      11153.8    0.00028300  0.00000650
   ************ END OF DATA FROM INTERNAL STORAGE ************
    50. PRINT MEMBER PROPERTIES LIST 7
  MEMBER   PROPERTI LIST     7
      STAAD SPACE                                          -- PAGE NO.
7
   MEMBER PROPERTIES. UNIT - INCH
   -----------------
   MEMB   PROFILE             AX/          IZ/          IY/        IX/
                              AY           AZ           SZ         SY
     7  ST  PIPS120          13.70       262.00       262.00      524.00
                              7.01         7.01        41.10       41.10
   ************ END OF DATA FROM INTERNAL STORAGE ************
    51. LOAD LIST 3
    52. PARAMETER 1
    53. CODE NF3000 2004
    54. STYPE 1 ALL
    55. CMZ 1 ALL
    56. CMY 1 ALL
    57. TRACK 2 ALL
    58. CHECK CODE MEMB 7
 STEEL DESIGN
      STAAD SPACE                                          -- PAGE NO.
8
                       STAAD.PRO CODE CHECKING - (  ASME NF3000-04)   v2.0
                       ********************************************
 ALL UNITS ARE - KIP  INCH (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
      7  ST   PIPS120                   (AISC SECTIONS)
                           PASS      NF-EQN-20         0.613         3
                      100.14 C         10.80        -97.68       60.00

|-----------------------------------------------------------------------------
|
 | SLENDERNESS CHECK:    ACTUAL RATIO:  27.44 ALLOWABLE RATIO:
200.00          |
 | ALLOWABLE STRESSES:   (UNIT - KIP
INCH)                                    |
 | AXIAL: 1.47E+01  FCZ: 2.38E+01  FCY: 2.38E+01  FTZ: 2.38E+01  FTY: 2.38E
+01 |
 | SHEAR: 1.44E
+01                                                             |
 | ACTUAL STRESSES: (UNIT - KIP
INCH)                                         |
 | AXIAL: 7.31E+00  FBZ: 2.38E+00  FBY: 2.63E-01 SHEAR: 0.00E
```

```
+00               |

|------------------------------------------------------------------------------
|
 | SECTION PROPERTIES: (UNIT -
INCH)                                            |
 | AXX:  13.70  AYY:   7.01  AZZ:   7.01  RZZ:  4.37  RYY:
4.37                 |
 | SZZ:      41.10  SYY:
41.10                                             |

|------------------------------------------------------------------------------
|
 | PARAMETER: (UNIT - KIP
INCH)                                                |
 | KL/R-Z: 27.44   KL/R-Y: 27.44   UNL:  120.0   CMZ:  1.00   CMY:
1.00        |
 | CB: 1.00   FYLD:    36.00   FU:    58.02   NET SECTION FACTOR:
1.00          |
 | CT:  0.75   STEEL TYPE:  1.0  KS:1.000  KV:1.000  KBK:
1.000                |

|------------------------------------------------------------------------------
|
 | CRITICAL LOADS FOR EACH CLAUSE CHECK (UNITS KIP -
INCH)                           |
 | CLAUSE       RATIO  LOAD       FX        VY        VZ        MZ
MY   |
 | TENSION      0.000    0    0.00E+00       -         -         -
-   |
 | COMPRESSION  0.497    3    1.00E+02       -         -         -
-   |
 | COMP&BEND    0.613    3    1.00E+02       -         -    9.77E+01 1.08E
+01  |
 | TEN&BEND     0.000    0    0.00E+00       -         -    0.00E+00 0.00E
+00  |
 | SHEAR-Y      0.039    3        -     5.00E+00       -         -
-   |
 | SHEAR-Z      0.000    0        -          -    0.00E+00       -
-   |

|------------------------------------------------------------------------------
|
    59. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME= 10: 9:43 ****
      STAAD SPACE                                             -- PAGE NO.
9
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
```

```
        *                http://www.bentley.com                        *
        ****************************************************************
```

Related Links

- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)
- *D1.L.4. ASME NF 3000 - 2001 & 2004 Codes* (on page 1338)

V.10 Concrete Design

V. India

V. IS456 2000-Axially Loaded Rectangular Column

Verify the designed results produced by the program with hand calculated design results for an axially loaded rectangular column section as per IS456:2000.

Details

The fixed-base, 2.5 m column is subjected to axial force of 3,300 kN. The section is 450mm x 600mm. The concrete strength is 25 N/mm^2 and the yield strength of steel is 415 N/mm^2. Use a clear cover of 40 mm. The main reinforcement is 20 mm and the lateral ties are 8 mm.

Validation

Slenderness Ratio

From Table. 28 Clause E-3 of IS 456 : 2000, the effective length of the compression member:

$$(L_{eff}) = 2 \times L = 2 \times 2{,}500 = 5{,}000 \text{ mm}$$

Least lateral dimension of column = (5,000/450) = 11.11 < 12

Large lateral dimension of column = (5,000/600) = 8.33 < 12

So, we can consider and design as a short column.

Check for Minimum Eccentricity of Column

As per Clause 25.4 of IS 456 : 2000, calculate the minimum eccentricity:

For Depth, $e_{min} = (2{,}500/500) + (600/30) = 25\ mm > 20\ mm$. (OK)

For Width $e_{min} = (2{,}500/500) + (450/30) = 20\ mm$. (OK)

Hence Minimum Eccentricity is ok.

Check for Maximum Eccentricity of Column

As per Clause 39.3 of IS 456 : 2000 calculate the maximum eccentricity:

For Depth $e_{max} = 0.05 \times 600 = 30\ mm$. Hence, $e_{min} < e_{max}$ (OK)

For Width $e_{max} = 0.05 \times 450 = 22.5\ mm$. Hence, $e_{min} < e_{max}$ (OK)

Hence Maximum Eccentricity is ok.

Design for Flexure

Gross Area of Column, $A_g = 450 \times 600 = 27 \times 10^4\ mm^2$.

Area of Concrete, $A_c = A_g - A_{sc} = 27 \times 10^4 - A_{sc}$

Area of Longitudinal Reinforcement, $A_{sc} = P_u = 0.4 \times f_{ck} \times Ac + 0.67 \times f_y \times A_{sc}$

$$3300 \times 10^3 = 0.4 \times 25\ [\ A_g - A_{sc}\ ] + 0.67 \times 415 \times A_{sc}$$

$$3300 \times 10^3 = 0.4 \times 25\ [\ 27 \times 10^4 - A_{sc}\ ] + 0.67 \times 415 \times A_{sc} = 2{,}238\ mm^2\ .$$

Let assume 20 mm dia bar. So, $A_{st} = (3.14/4) \times 20^2 = 314.15\ mm^2$.

Nos. of Bars will be = $(2238.3/314.15) = 7.12 \approx 8$ Nos

So, the actual reinforcement will be = $8 \times (3.14/4) \times 20^2 = 2513.2\ mm^2$

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Bar Diameter of Lateral Ties:

$$¼ \times \text{Main Bar dia} = ¼ \times 20 = 5\ mm$$

So, Provide 8 mm dia. bar for lateral ties.

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Pitch of Lateral Ties:

Least lateral dimension = 450 mm
16 × 20 = 320 mm
300 mm

Provide 8 mm dia. bar @ 300 mm for lateral ties.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Reqd. area of steel (mm^2)	2,238	2,238.39	negligible	
Spacing of rectangular ties reinforcement (mm)	300	300	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Axially Loaded Rectangular Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Aug-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 2.5 0;
MEMBER INCIDENCES
```

```
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17184e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 0.6 ZD 0.45
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -3300
PERFORM ANALYSIS PRINT ALL
START CONCRETE DESIGN
CODE INDIAN
CLEAR 0.04 ALL
ELZ 2 ALL
ELY 2 ALL
FC 25000 ALL
FYMAIN 415000 ALL
MAXMAIN 20 ALL
MAXSEC 8 ALL
MINMAIN 20 ALL
MINSEC 8 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
            =========================================================
            IS 456 - 2000   C O L U M N   D E S I G N   R E S U L T S
            =========================================================

============================================================================
                IS-456    L I M I T   S T A T E   D E S I G N
            C O L U M N   N O.     1  D E S I G N   R E S U L T S
         M25                 Fe415 (Main)              Fe415 (Sec.)
   LENGTH:  2500.0 mm   CROSS SECTION:  450.0 mm X  600.0 mm  COVER: 40.0 mm
   ** GUIDING LOAD CASE:    1 END JOINT:     1  SHORT COLUMN
   DESIGN FORCES (KNS-MET)
   -----------------------
   DESIGN AXIAL FORCE (Pu)             :    3300.00
                                            About Z            About Y
   INITIAL MOMENTS                     :       0.00               0.00
   MOMENTS DUE TO MINIMUM ECC.         :       0.00               0.00
   SLENDERNESS RATIOS                  :     -                  -
   MOMENTS DUE TO SLENDERNESS EFFECT   :     -                  -
```

```
   MOMENT REDUCTION FACTORS           :     -                  -
   ADDITION MOMENTS (Maz and May)     :     -                  -
   TOTAL DESIGN MOMENTS               :      0.00               0.00
   REQD. STEEL AREA   :     2238.39 Sq.mm.
   REQD. CONCRETE AREA:   267761.62 Sq.mm.
   MAIN REINFORCEMENT : Provide   8 - 20 dia. (0.93%,   2513.27 Sq.mm.)
                     (Equally distributed)
   TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 300 mm c/c
   SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS)
   ------------------------------------------------------
   Puz :   3300.00 KNS
      STAAD SPACE                                           -- PAGE NO.
6
   SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS)
   ------------------------------------------------------
   END JOINT:     1 Puz :   3373.68 KNS   IR:0.98
   END JOINT:     2 Puz :   3373.68 KNS   IR:0.98

============================================================================
   ********************END OF COLUMN DESIGN RESULTS********************
```

Related Links

- *D8.E.6 Column Design* (on page 1680)

V. IS456 2000-Axially Loaded Square Column

Verify the designed results produced by the program with hand calculated design results for an axially loaded square column section as per IS456:2000.

Details

The fixed-base, 2.5 m column is subjected to axial force of 3,300 kN. The section is 500mm x 500mm. The concrete strength is 25 N/mm^2 and the yield strength of steel is 415 N/mm^2. Use a clear cover of 40 mm. The main reinforcement is 20 mm and the lateral ties are 8 mm.

Validation

<u>Slenderness Ratio</u>

From Table. 28 Clause E-3 of IS 456 : 2000, the effective length of the compression member:

$$(L_{eff}) = 2 \times L = 2 \times 2{,}500 = 5{,}000 \text{ mm}$$

Least lateral dimension of column = (5,000/500) = 10 < 12

Large lateral dimension of column = (5,000/500) = 10 < 12

So, we can consider and design as a short column.

<u>Check for Minimum Eccentricity of Column</u>

As per Clause 25.4 of IS 456 : 2000, calculate the minimum eccentricity:

For Depth, $e_{min} = (2{,}500/500) + (500/30) = 21.7\ mm > 20\ mm$. (OK)

For Width $e_{min} = (2{,}500/500) + (500/30) = 21.7\ mm > 20\ mm$. (OK)

Hence Minimum Eccentricity is ok.

Check for Maximum Eccentricity of Column

As per Clause 39.3 of IS 456 : 2000 calculate the maximum eccentricity:

For Depth $e_{max} = 0.05 \times 500 = 25\ mm$. Hence, $e_{min} < e_{max}$ (OK)

For Width $e_{max} = 0.05 \times 500 = 25\ mm$. Hence, $e_{min} < e_{max}$ (OK)

Hence Maximum Eccentricity is ok.

Design for Flexure

Gross Area of Column, $A_g = 500 \times 500 = 25 \times 10^4\ mm^2$.

Area of Concrete, $A_c = A_g - A_{sc} = 25 \times 10^4 - A_{sc}$

Area of Longitudinal Reinforcement, $A_{sc} = P_u = 0.4 \times f_{ck} \times Ac + 0.67 \times f_y \times A_{sc}$

$$3{,}300 \times 10^3 = 0.4 \times 25\ [\ A_g - A_{sc}\ ] + 0.67 \times 415 \times A_{sc}$$

$$3{,}300 \times 10^3 = 0.4 \times 25\ [\ 25 \times 10^4 - A_{sc}\ ] + 0.67 \times 415 \times A_{sc}$$

$$2{,}985\ \text{mm}^2\ .$$

Let assume 25 mm dia bar. So, $A_{st} = (3.14/4) \times 25^2 = 490.3\ mm^2$.

Nos. of Bars will be = $(2{,}985/490.3) = 6.1 \approx 8$ Nos

So, the actual reinforcement will be = $8 \times (3.14/4) \times 25^2 = 3{,}925\ mm^2$

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Bar Diameter of Lateral Ties:

$$\tfrac{1}{4} \times \text{Main Bar dia} = \tfrac{1}{4} \times 25 = 6.3\ \text{mm}$$

So, Provide 8 mm dia. bar for lateral ties.

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Pitch of Lateral Ties:

Least lateral dimension = 500 mm
16 × 25 = 400 mm
300 mm

Provide 8 mm dia. bar @ 300 mm for lateral ties.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Reqd. area of steel (mm^2)	2,985	2,984.52	negligible	
Spacing of rectangular ties reinforcement (mm)	300	300	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Axially Loaded Square Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 16-Aug-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 2.5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17184e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 0.5 ZD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -3300
PERFORM ANALYSIS PRINT ALL
START CONCRETE DESIGN
CODE INDIAN
CLEAR 0.04 ALL
ELZ 2 ALL
ELY 2 ALL
FC 25000 ALL
FYMAIN 415000 ALL
MAXMAIN 25 ALL
MAXSEC 8 ALL
MINMAIN 25 ALL
MINSEC 8 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
            =========================================================
            IS 456 - 2000   C O L U M N   D E S I G N   R E S U L T S
            =========================================================
```

```
============================================================================
                IS-456    L I M I T    S T A T E    D E S I G N
            C O L U M N   N O.     1  D E S I G N   R E S U L T S
         M25                    Fe415 (Main)                Fe415 (Sec.)
   LENGTH:  2500.0 mm   CROSS SECTION:  500.0 mm X  500.0 mm  COVER: 40.0 mm
   ** GUIDING LOAD CASE:    1 END JOINT:     1  SHORT COLUMN
   DESIGN FORCES (KNS-MET)
   -----------------------
   DESIGN AXIAL FORCE (Pu)            :    3300.00
                                           About Z            About Y
   INITIAL MOMENTS                    :       0.00               0.00
   MOMENTS DUE TO MINIMUM ECC.        :       0.00               0.00
   SLENDERNESS RATIOS                 :     -                  -
   MOMENTS DUE TO SLENDERNESS EFFECT  :     -                  -
   MOMENT REDUCTION FACTORS           :     -                  -
   ADDITION MOMENTS (Maz and May)     :     -                  -
   TOTAL DESIGN MOMENTS               :       0.00               0.00
   REQD. STEEL AREA   :     2984.52 Sq.mm.
   REQD. CONCRETE AREA:   247015.48 Sq.mm.
   MAIN REINFORCEMENT : Provide   8 - 25 dia. (1.57%,   3926.99 Sq.mm.)
                        (Equally distributed)
   TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 300 mm c/c
   SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS)
   ------------------------------------------------------
   Puz :   3300.00 KNS
      STAAD SPACE                                                -- PAGE NO.
6
   SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS)
   ------------------------------------------------------
   END JOINT:     1 Puz :   3552.63 KNS   IR:0.93
   END JOINT:     2 Puz :   3552.63 KNS   IR:0.93

============================================================================
   *******************END OF COLUMN DESIGN RESULTS********************
```

Related Links

- *D8.E.6 Column Design* (on page 1680)

V. IS456 2000-Circular Column

Verify the designed results produced by the program with hand calculated design results for an axially loaded circular column section as per IS456:2000.

References

Bureau of Indian Standards. *SP 16 - Design aids for reinforced concrete to IS: 456-1978.* 1980. New Dehli: Indian Standards Institution. p. 141.

Details

The fixed-base, 2.5 m cantilever column is subjected to axial force of 800 kN and a moment at the top of 162.5 kN·m. The section is 500 mm in diameter. The concrete strength is 20 N/mm^2 and the yield strength of steel is 415 N/mm^2. Use a clear cover of 50 mm. The main reinforcement is 20 mm and the lateral ties are 10 mm.

Validation

Slenderness Ratio

From Table. 28 Clause E-3 of IS 456 : 2000, the effective length of the compression member:

$$(L_{eff}) = 2 \times L = 2 \times 2{,}500 = 5{,}000 \text{ mm}$$

Least lateral dimension of column = (5,000/500) = 10 < 12

Large lateral dimension of column = (5,000/500) = 10 < 12

So, we can consider and design as a short column.

Check for Minimum Eccentricity of Column

As per Clause 25.4 of IS 456 : 2000, calculate the minimum eccentricity:

For Width $e_{min} = (2{,}500/500) + (500/30) = 21.7\ mm > 20\ mm$. (OK)

Hence Minimum Eccentricity is ok.

Check for Maximum Eccentricity of Column

As per Clause 39.3 of IS 456 : 2000 calculate the maximum eccentricity:

For Width $e_{max} = 0.05 \times 500 = 25\ mm$. Hence, $e_{min} < e_{max}$ (OK)

Hence Maximum Eccentricity is ok.

Design for Flexure

Determine the non-dimenstional parameters to use with interaction chart 56 of SP 16:

$$P_u / f_{ck} D^2 = 800(10^3) / \left(20 \times 500^2\right) = 0.16$$

$$M_u / f_{ck} D^3 = 162.5(10^6) / \left(20 \times 500^3\right) = 0.065$$

From the chart, $P / f_{ck} = 0.06$, and thus $P = 20 \times 0.06 = 1.2$

The area of longitudinal reinforcement required then is given by:

$$A_{sc} = \frac{P \times \pi \times D^2}{400} = \frac{1.2 \times \pi \times 500^2}{400} = 2{,}355 \text{ mm}^2$$

Let assume 20 mm dia bar. So, $A_{st} = (3.14/4) \times 25^2 = 314\ mm^2$.

Number of bars = $(2{,}355/314) = 7.5 \approx 8$ bars
So, the actual reinforcement will be = $8 \times 314\ mm^2 = 2{,}513\ mm^2$

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Bar Diameter of Lateral Ties:

¼ × Main Bar dia = ¼ × 20 = 5 mm

or, 6 mm

So, Provide 10 mm dia. bar for lateral ties.

As per Clause 26.5.3.2 (c) of IS 456 : 2000 Pitch of Lateral Ties:

Least lateral dimension = 500 mm
16 × 20 = 320 mm
300 mm

Provide 10 mm dia. bar @ 300 mm for lateral ties.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Reqd. area of steel (mm^2)	2,355	2,367.71	negligible	
Spacing of rectangular ties reinforcement (mm)	300	300	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Circular Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 09-Dec-20
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 2.5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY
1 PRIS YD 0.5
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -800
LOAD 2 LOADTYPE None  TITLE LOAD CASE 2
JOINT LOAD
2 MZ 162.5
LOAD COMB 3 COMBINATION LOAD CASE 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
LOAD LIST 3
START CONCRETE DESIGN
CODE INDIAN
FC 20000 ALL
FYMAIN 415000 ALL
```

```
FYSEC 415000 ALL
CLEAR 0.050 ALL
MAXMAIN 20 ALL
MAXSEC 10 ALL
MINMAIN 20 ALL
MINSEC 10 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

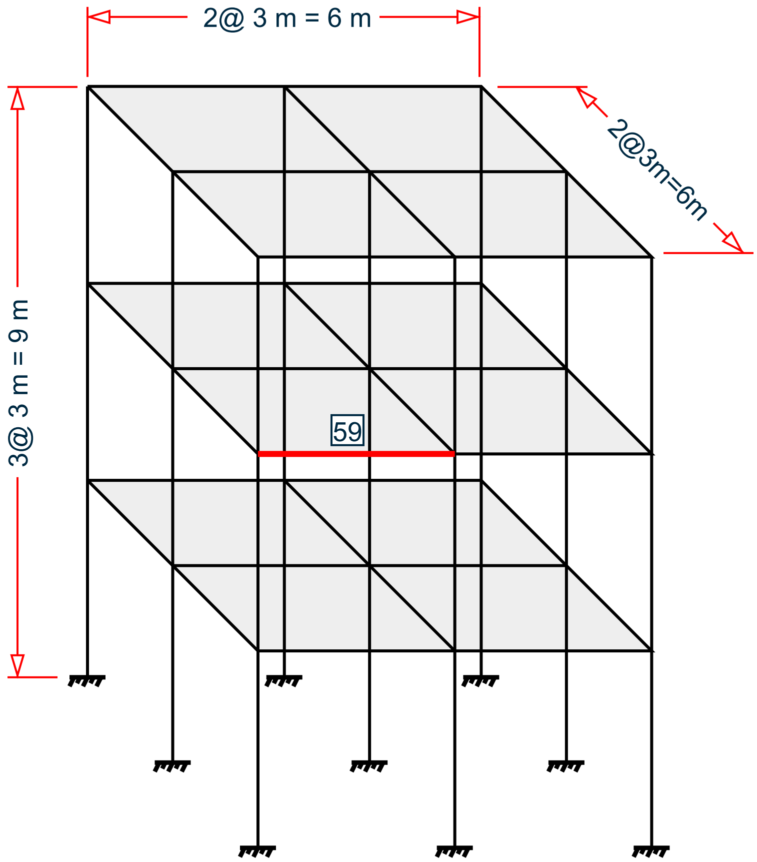
```
 COLUMN
           =========================================================
           IS 456 - 2000   C O L U M N   D E S I G N   R E S U L T S
           =========================================================

============================================================================
               IS-456    L I M I T    S T A T E    D E S I G N
           C O L U M N   N O.     1  D E S I G N   R E S U L T S
        M20                  Fe415 (Main)               Fe415 (Sec.)
   LENGTH:  2500.0 mm   CROSS SECTION:  500.0 mm dia.  COVER: 50.0 mm
   ** GUIDING LOAD CASE:    3 END JOINT:     1  SHORT COLUMN
   DESIGN FORCES (KNS-MET)
   -----------------------
   DESIGN AXIAL FORCE (Pu)             :     800.00
                                            About Z            About Y
   INITIAL MOMENTS                     :     162.50               0.00
   MOMENTS DUE TO MINIMUM ECC.         :      17.33              17.33
   SLENDERNESS RATIOS                  :     -                  -
   MOMENTS DUE TO SLENDERNESS EFFECT   :     -                  -
   MOMENT REDUCTION FACTORS            :     -                  -
   ADDITION MOMENTS (Maz and May)      :     -                  -
   TOTAL DESIGN MOMENTS                :     162.50              17.33
   REQD. STEEL AREA   :     2367.71 Sq.mm.
   REQD. CONCRETE AREA:   193981.83 Sq.mm.
   MAIN REINFORCEMENT : Provide   8 - 20 dia. (1.28%,   2513.27 Sq.mm.)
                        (Equally distributed)
   TIE REINFORCEMENT  : Provide 10 mm dia. circular ties @ 300 mm c/c
   SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
   ----------------------------------------------------------
   Puz :   2482.79   Muz1 :    173.76   Muy1 :    173.76
   INTERACTION RATIO: 0.98 (as per Cl. 39.6, IS456:2000)
      STAAD SPACE                                            -- PAGE NO.
7
   SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
   ----------------------------------------------------------
   WORST LOAD CASE:     3
   END JOINT:     2 Puz :   2526.78   Muz :    176.84   Muy :    176.84   IR:
0.97

============================================================================
   ********************END OF COLUMN DESIGN RESULTS********************
```

V. IS456 2000-Doubly Reinforced Rectangular Beam

Verify the design of a rectangular beam per IS 456:2000.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:



Section properties:

Length of beam, $L = 3{,}000\ mm$ (clear span)
Width of beam, $B = 250\ mm$
Depth of beam, $D = 300\ mm$
Clear cover = $30\ mm$
Diameter of main reinforcement = $16\ mm$
Diameter of stirrups = $8\ mm$

Material properties:

Characteristic strength of concrete, $f_{ck} = 20\ N/mm^2$
Yield strength of steel, $f_y = 415\ N/mm^2$

Loads: total uniform force = $50\ kN/m$

Validation

Effective Depth, $d = 300 – 30 – 8 – (16/2) = 254\ mm$

Depth of Compression Reinforcement, $d' = 30 + (16/2) + 8 = 46\ mm$

Design for Flexure

Limiting moment value, $M_{z,limit} = 0.138 \times f_{ck} \times B \times d^2 = 44.52\ kN \cdot m$

From the STAAD.Pro analysis, $M_z = 55.44\ kN \cdot m > M_{z,limit}$ at the start of the member (x = 0).

So, the actual moment is greater than limiting moment then the section will be design as doubly reinforced section.

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section:

Percentage of tension reinforcement $\rho_t = 1.212$
Percentage of compression reinforcement $\rho_c = 0.288$

<u>Reinforcement at Start of Member (x = 0): top bars</u>

$$A_{st,top\_a} = 1.212 \times B \times d/100 = 769.6\ mm^2$$

Provide 4 - 16 mm diameter bars. Actual $A_{st,\ top} = 4 \times (\pi/4) \times (16)^2 = 804.2\ mm^2$
<u>Reinforcement at Start of Member (x = 0): bottom bars</u>

$$A_{st,bot\_a} = 0.288 \times B \times d/100 = 182.9\ mm^2$$

Provide 2 - 16 mm diameter bars. Actual $A_{st,\ bot} = 2 \times (\pi/4) \times (16)^2 = 402.1\ mm^2$

From the STAAD.Pro analysis, $M_z = 54.11\ kN \cdot m > M_{z,limit}$ at the end of the member (x = 3 m).

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section:

Percentage of tension reinforcement $\rho_t = 1.183$
Percentage of compression reinforcement $\rho_c = 0.256$

<u>Reinforcement at End of Member (x = 3 m): top bars</u>

$$A_{st,top\_b} = 1.183 \times B \times d/100 = 751.2\ mm^2$$

Provide 4 - 16 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (16)^2 = 804.2\ mm^2$
<u>Reinforcement at End of Member (x = 3 m): bottom bars</u>

$$A_{st,bot\_b} = 0.256 \times B \times d/100 = 162.6\ mm^2$$

Provide 2 - 16 mm diameter bars. Actual $A_{st,\ bot} = 2 \times (\pi/4) \times (16)^2 = 402.1\ mm^2$

Design for Shear

Provide percentage of tension steel, $\rho_t = 1.266$.

From Table 19 of IS 456:2000, for M20 grade concrete, shear stress: $T_c = 0.673\ N/mm^2$.
Factored shear force as per linear structural analysis in STAAD.Pro, $F_y = 117.9\ kN$.

The nominal shear stress, $T_v = 117.9\ /\ (B \times d) = 117.9 \times 10^3\ /\ (250 \times 254) = 1.86\ N/mm^2$.

The maximum shear stress from Table 20 of IS 456:2000 for M20 grade concrete, $T_{c,max} = 2.8\ N/mm^2$.

Shear strength of the reinforcement: $V_{us} = F_y - T_c \times B \times d = 117.9 - 0.673 \times 250 \times 254 \times 10^3 = 75.16\ kN$.

Provide 2 legged 8 mm diameter bar. So, the area of shear reinforcement, $A_{sv} = \pi / 4 \times 8^2 \times 2 = 100.5\ mm^2$.

Using formula of IS 456:2000 spacing for vertical stirrups: $s_v = 0.87 \times (f_y \times A_{sv} \times d)\ /\ V_{us} = 122.8\ mm$.

So, provide 2 legged 8 mm diameter bar stirrups at 100 mm center to center.

Results

Table 876: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Required area of steel at top of start section (mm^2)	769.4	772.79	negligible	
Required area of steel at bottom of start section (mm^2)	182.6	186.11	negligible	
Required area of steel at top of end section (mm^2)	751.2	754.73	negligible	
Required area of steel at bottom of end section (mm^2)	162.3	166.44	negligible	
Spacing of shear reinforcement at ends (mm)	100	100	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Doubly Reinforced Rectangular Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
******************************************************************************
*
*This Problem is created to verify the design results produced by the program
*
*with hand calculated design results for a rectangular(300 mm * 250 mm)
Doubly*
*reinforced  beam section as per IS456:2000
*
*
*
******************************************************************************
*
```

```
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD
0.25
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -45
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
1 1.5
LOAD COMB 6 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 4
```

```
1 0.9
LOAD COMB 7 COMBINATION LOAD CASE 7
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
START CONCRETE DESIGN
CODE INDIAN
CLB 0.03 ALL
CLT 0.03 ALL
FC 20000 MEMB 59
FYMAIN 413688 MEMB 59
FYSEC 413688 MEMB 59
MAXMAIN 16 MEMB 59
MAXSEC 8 MEMB 59
MINMAIN 16 MEMB 59
MINSEC 8 MEMB 59
TRACK 2 MEMB 59
DESIGN BEAM 59
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
               ====================================================
               IS 456 - 2000  B E A M   D E S I G N   R E S U L T S
               ====================================================

============================================================================
                  IS-456    L I M I T   S T A T E   D E S I G N
                 B E A M  N O.     59  D E S I G N  R E S U L T S
         M20                     Fe415 (Main)                Fe415 (Sec.)
       LENGTH:  3000.0 mm      SIZE:   250.0 mm X  300.0 mm   COVER: 40.0 mm
                        DESIGN LOAD SUMMARY (KN MET)

--------------------------------------------------------------------------------
----
 SECTION |FLEXURE  (Maxm. Sagg./Hogg./Eqv. moments) |          SHEAR
 (in mm) |    P       MZ       MX       ME   Load # |    VY       MX
VE  Load #

--------------------------------------------------------------------------------
----
     0.0 |    0.00     0.00     0.00     0.00     1 |  112.94     0.77
117.85     3
         |    0.00   -55.44     0.77   -56.44     3 |
   250.0 |    0.00     0.00     0.00     0.00     1 |   94.19     0.77
99.10     3
         |    0.00   -29.55     0.77   -30.54     3 |
   500.0 |    0.00     0.00     0.00     0.00     1 |   75.44     0.77
80.35     3
         |    0.00    -8.35     0.77    -9.34     3 |
   750.0 |    0.00     8.17     0.77     9.16     3 |   56.69     0.77
61.60     3
         |    0.00     0.00     0.00     0.00     1 |
  1000.0 |    0.00    20.00     0.77    20.99     3 |   37.94     0.77
42.85     3
         |    0.00     0.00     0.00     0.00     1 |
  1250.0 |    0.00    27.14     0.77    28.13     3 |   19.19     0.77
```

```
24.10     3
         |    0.00     0.00     0.00     0.00     1 |
  1500.0 |    0.00    29.60     0.77    30.59     3 |    0.44     0.77
5.35     3
         |    0.00     0.00     0.00     0.00     1 |
  1750.0 |    0.00    27.36     0.77    28.36     3 |  -18.31     0.77
23.21     3
         |    0.00     0.00     0.00     0.00     1 |
  2000.0 |    0.00    20.44     0.77    21.44     3 |  -37.06     0.77
41.96     3
         |    0.00     0.00     0.00     0.00     1 |
  2250.0 |    0.00     8.84     0.77     9.83     3 |  -55.81     0.77
60.71     3
         |    0.00     0.00     0.00     0.00     1 |
  2500.0 |    0.00     0.00     0.00     0.00     1 |  -74.56     0.77
79.46     3
         |    0.00    -7.46     0.77    -8.45     3 |
  2750.0 |    0.00     0.00     0.00     0.00     1 |  -93.31     0.77
98.21     3
         |    0.00   -28.44     0.77   -29.43     3 |
  3000.0 |    0.00     0.00     0.00     0.00     1 | -112.06     0.77
116.96     3
         |    0.00   -54.11     0.77   -55.11     3 |

----------------------------------------------------------------------------
      STAAD SPACE                                              -- PAGE NO.
11
                        SUMMARY OF REINF. AREA (Sq.mm)

----------------------------------------------------------------------------
   SECTION |             TOP           |          BOTTOM           |   STIRRUPS
   (in mm) | Reqd./Provided reinf.     | Reqd./Provided reinf.     |  (2 legged)

----------------------------------------------------------------------------
       0.0 |   772.79/ 804.25( 4-16d )|  186.11/ 402.12( 2-16d )|  8d  @ 100 mm
     250.0 |   384.93/ 402.12( 2-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 100 mm
     500.0 |   153.61/ 402.12( 2-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 100 mm
     750.0 |     0.00/ 402.12( 2-16d )|  129.04/ 402.12( 2-16d )|  8d  @ 100 mm
    1000.0 |     0.00/ 402.12( 2-16d )|  251.83/ 402.12( 2-16d )|  8d  @ 100 mm
    1250.0 |     0.00/ 402.12( 2-16d )|  349.91/ 402.12( 2-16d )|  8d  @ 100 mm
    1500.0 |     0.00/ 402.12( 2-16d )|  385.60/ 402.12( 2-16d )|  8d  @ 100 mm
    1750.0 |     0.00/ 402.12( 2-16d )|  353.09/ 402.12( 2-16d )|  8d  @ 100 mm
    2000.0 |     0.00/ 402.12( 2-16d )|  257.70/ 402.12( 2-16d )|  8d  @ 100 mm
    2250.0 |     0.00/ 402.12( 2-16d )|  129.04/ 402.12( 2-16d )|  8d  @ 100 mm
    2500.0 |   153.61/ 402.12( 2-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 100 mm
    2750.0 |   368.69/ 402.12( 2-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 100 mm
    3000.0 |   754.73/ 804.25( 4-16d )|  166.44/ 402.12( 2-16d )|  8d  @ 100 mm

----------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT DISTANCE d (EFFECTIVE DEPTH) FROM FACE OF THE
SUPPORT

-----------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT   452.0 mm AWAY FROM START SUPPORT
     VY =    79.04 MX =     0.77 LD=    3
     Provide 2 Legged  8d  @ 100 mm c/c
   SHEAR DESIGN RESULTS AT   452.0 mm AWAY FROM END SUPPORT
     VY =   -78.16 MX =     0.77 LD=    3
```

```
    Provide 2 Legged  8d  @ 100 mm c/c
=========================================================================
```

Related Links

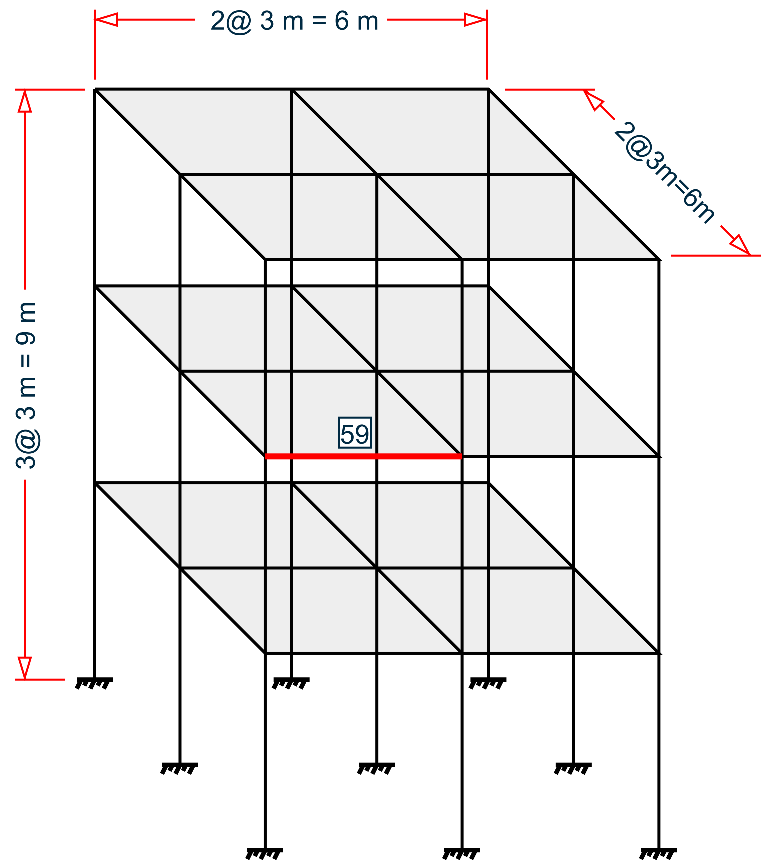
- *D8.E.5 Beam Design* (on page 1677)

V. IS456 2000-Singly Reinforced Rectangular Beam

Verify the designed results produced by the program with hand calculated design results for a rectangular beam section as per IS456:2000.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:



Section properties:

Length of beam, $L = 3,000\ mm$ (clear span)
Width of beam, $B = 250\ mm$
Depth of beam, $D = 300\ mm$
Clear cover = $30\ mm$
Diameter of main reinforcement = $12\ mm$

Diameter of stirrups = $8\ mm$

Material properties:

Characteristic strength of concrete, $f_{ck} = 20\ N/mm^2$
Yield strength of steel, $f_y = 415\ N/mm^2$

Loads: total uniform force = $25\ kN/m$

Validation

Effective Depth, $d = 300 – 30 – 8 – (12/2) = 256\ mm$

Design for Flexure

Limiting moment value, $M_{z,limit} = 0.138 \times f_{ck} \times B \times d^2 = 0.138 \times 20 \times 250 \times 256^2 \times 10^6 = 45.22\ kN{\cdot}m$

From the STAAD.Pro analysis, $M_z = 30.26\ kN{\cdot}m < M_{z,limit}$ at the start of the member (x = 0).

So, the actual moment is less than limiting moment then the section will be design as singly reinforced section.

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $\rho_t = 0.583$

Reinforcement at Start of Member (x = 0): top bars

$$A_{st} = 0.583 \times B \times d/100 = 0.583 \times 250 \times 256 / 100 = 373.0\ mm^2$$

Provide 4 - 12 mm diameter bars. Actual $A_{st,\ top} = 4 \times (\pi/4) \times (12)^2 = 452.4\ mm^2$

From the STAAD.Pro analysis, $M_z = 24.57\ kN{\cdot}m < M_{z,limit}$ at the end of the member (x = 3 m).

The actual moment is less than the limiting moment. Thus, the section is designed as a singly reinforced section.

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $\rho_t = 0.46$

Reinforcement at End of Member (x = 3 m): top bars

$$A_{st} = 0.46 \times B \times d/100 = 0.46 \times 250 \times 256 / 100 = 294.4\ mm^2$$

Provide 3 - 12 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (12)^2 = 393.3\ mm^2$

Shear Strength of Concrete

Provide percentage of tension steel, $\rho_t = 0.707$.

From Table 19 of IS 456:2000, for M20 grade concrete, shear stress: $T_c = 0.546\ N/mm^2$.

Factored shear force as per linear structural analysis in STAAD.Pro, $F_y = 58.15\ kN$.

The nominal shear stress, $T_v = F_y\ /\ (B \times d) = 58.15\ /\ (250 \times 256) = 0.909\ N/mm^2$.

The maximum shear stress from Table 20 of IS 456:2000 for M20 grade concrete, $T_{c,max} = 2.8\ N/mm^2$.

$T_c < T_v < T_{c,max}$, so provide shear reinforcement.

Shear strength of the reinforcement: $V_{us} = F_y - T_c \times B \times d = 58.15 - 0.546 \times 250 \times 256 \times 10^3 = 23.19\ kN$.

Provide 2 legged 8 mm diameter bar. So, the area of shear reinforcement, $A_{sv} = \pi/4 \times 8^2 \times 2 = 100.5\ mm^2$.

Using the formula of IS 456:200, the spacing for vertical stirrups:

$$S_v = 0.87(F_y \times A_{sv} \times d / V_{us}) = 0.87(250 \times 100.5 \times 256 / 23.19) = 400.7 \text{ mm}$$

Per Cl. 26.5.1.5 of IS 456:2000, the maximum spacing for of shear reinforcement is the least of:

$0.75 \times d = 0.75 \times 256 = 192\ mm$
$300\ mm$
$(0.87\times f_y \times A_{sv}) / (0.4 \times B) = 393\ mm$
$S_v = 401\ mm$

So, provide 2 legged 8 mm diameter bar stirrups at 190 mm center to center.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Reqd. area of steel at start section (mm^2)	373	376.80	negligible	
Reqd. area of steel at end section (mm^2)	294.4	297.10	negligible	
Spacing of shear reinforcement (mm)	192	190	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Singly Reinforced Rectangular Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
```

```
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD
0.25
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
MEMBER RELEASE
59 START MX
59 END MX
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -20
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
1 1.5
LOAD COMB 6 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 4
1 0.9
LOAD COMB 7 COMBINATION LOAD CASE 7
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
LOAD LIST 3 TO 6
START CONCRETE DESIGN
CODE INDIAN
CLB 0.03 ALL
CLT 0.03 ALL
FC 20000 MEMB 59
FYMAIN 415000 MEMB 59
FYSEC 415000 MEMB 59
MAXMAIN 12 MEMB 59
MAXSEC 8 MEMB 59
MINMAIN 12 MEMB 59
```

```
MINSEC 8 MEMB 59
TRACK 2 MEMB 59
DESIGN BEAM 59
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
               ====================================================
               IS 456 - 2000  B E A M   D E S I G N   R E S U L T S
               ====================================================

============================================================================
                   IS-456    L I M I T    S T A T E    D E S I G N
                  B E A M  N O.     59   D E S I G N  R E S U L T S
         M20                   Fe415 (Main)                Fe415 (Sec.)
       LENGTH:  3000.0 mm      SIZE:   250.0 mm X  300.0 mm   COVER: 40.0 mm
                        DESIGN LOAD SUMMARY (KN MET)

--------------------------------------------------------------------------------
----
 SECTION |FLEXURE  (Maxm. Sagg./Hogg./Eqv. moments) |          SHEAR
 (in mm) |    P        MZ       MX       ME   Load # |    VY       MX
VE  Load #

--------------------------------------------------------------------------------
----
     0.0 |    0.00     0.00     0.00     0.00     3 |   58.15     0.00
58.15     3
         |    0.00   -30.26     0.00   -30.26     3 |
   250.0 |    0.00     0.00     0.00     0.00     3 |   48.77     0.00
48.77     3
         |    0.00   -16.90     0.00   -16.90     3 |
   500.0 |    0.00     0.00     0.00     0.00     3 |   39.40     0.00
39.40     3
         |    0.00    -5.88     0.00    -5.88     3 |
   750.0 |    0.00     2.80     0.00     2.80     3 |   30.02     0.00
30.02     3
         |    0.00     0.00     0.00     0.00     3 |
  1000.0 |    0.00     9.13     0.00     9.13     3 |   20.65     0.00
20.65     3
         |    0.00     0.00     0.00     0.00     3 |
  1250.0 |    0.00    13.12     0.00    13.12     3 |   11.27     0.00
11.27     3
         |    0.00     0.00     0.00     0.00     3 |
  1500.0 |    0.00    14.77     0.00    14.77     3 |    1.90     0.00
1.90     3
         |    0.00     0.00     0.00     0.00     3 |
  1750.0 |    0.00    14.07     0.00    14.07     3 |   -7.48     0.00
7.48     3
         |    0.00     0.00     0.00     0.00     3 |
  2000.0 |    0.00    11.03     0.00    11.03     3 |  -16.85     0.00
16.85     3
         |    0.00     0.00     0.00     0.00     3 |
  2250.0 |    0.00     5.64     0.00     5.64     3 |  -26.23     0.00
26.23     3
         |    0.00     0.00     0.00     0.00     3 |
```

```
  2500.0 |    0.00     0.00     0.00     0.00     3 |  -35.60     0.00
35.60     3
         |    0.00    -2.09     0.00    -2.09     3 |
  2750.0 |    0.00     0.00     0.00     0.00     3 |  -44.98     0.00
44.98     3
         |    0.00   -12.16     0.00   -12.16     3 |
  3000.0 |    0.00     0.00     0.00     0.00     3 |  -54.35     0.00
54.35     3
         |    0.00   -24.57     0.00   -24.57     3 |

-----------------------------------------------------------------------------
      STAAD SPACE                                                -- PAGE NO.
35
                        SUMMARY OF REINF. AREA (Sq.mm)

-----------------------------------------------------------------------------
   SECTION |           TOP            |          BOTTOM          |   STIRRUPS
   (in mm) | Reqd./Provided reinf.    | Reqd./Provided reinf.    |  (2 legged)

-----------------------------------------------------------------------------
       0.0 |  376.80/ 452.39( 4-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm
     250.0 |  197.12/ 226.19( 2-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm
     500.0 |  153.61/ 226.19( 2-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm
     750.0 |    0.00/ 226.19( 2-12d )|  130.06/ 226.19( 2-12d )|  8d  @ 190 mm
    1000.0 |    0.00/ 226.19( 2-12d )|  130.06/ 226.19( 2-12d )|  8d  @ 190 mm
    1250.0 |    0.00/ 226.19( 2-12d )|  150.62/ 226.19( 2-12d )|  8d  @ 190 mm
    1500.0 |    0.00/ 226.19( 2-12d )|  170.70/ 226.19( 2-12d )|  8d  @ 190 mm
    1750.0 |    0.00/ 226.19( 2-12d )|  162.15/ 226.19( 2-12d )|  8d  @ 190 mm
    2000.0 |    0.00/ 226.19( 2-12d )|  130.06/ 226.19( 2-12d )|  8d  @ 190 mm
    2250.0 |    0.00/ 226.19( 2-12d )|  130.06/ 226.19( 2-12d )|  8d  @ 190 mm
    2500.0 |  153.61/ 226.19( 2-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm
    2750.0 |  153.61/ 226.19( 2-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm
    3000.0 |  297.10/ 339.29( 3-12d )|    0.00/ 226.19( 2-12d )|  8d  @ 190 mm

-----------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT DISTANCE d (EFFECTIVE DEPTH) FROM FACE OF THE
SUPPORT

------------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT   454.0 mm AWAY FROM START SUPPORT
     VY =    41.20 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @ 190 mm c/c
   SHEAR DESIGN RESULTS AT   454.0 mm AWAY FROM END SUPPORT
     VY =   -37.40 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @ 190 mm c/c

=============================================================================
   *******************END OF BEAM DESIGN RESULTS*******************
```

Related Links

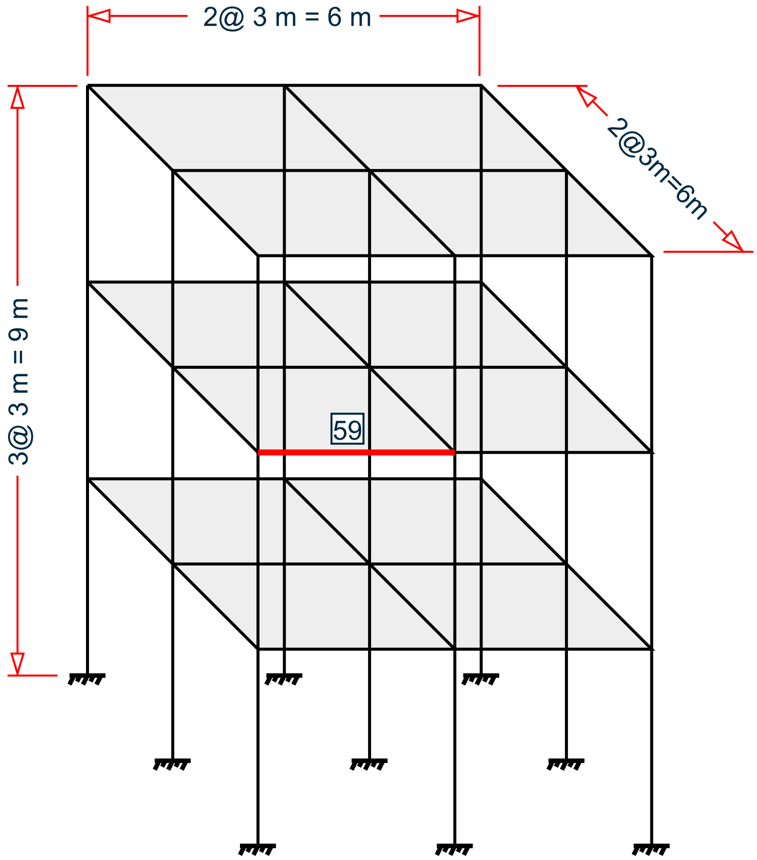
- *D8.E.5 Beam Design* (on page 1677)

V. IS456 2000-Singly Reinforced Square Beam

Verify the designed results produced by the program with hand calculated design results for a square beam section as per IS456:2000.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:



Section properties:

> Length of beam, $L = 3{,}000\ mm$ (clear span)
> Width of beam, $B = 300\ mm$
> Depth of beam, $D = 300\ mm$
> Clear cover = $30\ mm$
> Diameter of main reinforcement = $16\ mm$
> Diameter of stirrups = $8\ mm$

Material properties:

> Characteristic strength of concrete, $f_{ck} = 20\ N/mm^2$
> Yield strength of steel, $f_y = 415\ N/mm^2$

Loads: total uniform force = $35\ kN/m$

Validation

Effective Depth, $d = 300 - 30 - 8 - (16/2) = 254\ mm$

Design for Flexure

Limiting moment value, $M_{z,limit} = 0.138 \times f_{ck} \times B \times d^2 = 0.138 \times 20 \times 300 \times 254^2 \times 10^6 = 53.42\ kN \cdot m$

From the STAAD.Pro analysis, $M_z = 40.46\ kN \cdot m < M_{z,limit}$ at the start of the member (x = 0).

So, the actual moment is less than limiting moment then the section will be design as singly reinforced section.

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $\rho_t = 0.674$

Reinforcement at Start of Member (x = 0): top bars

$$A_{st} = 0.674 \times B \times d / 100 = 0.674 \times 300 \times 254 / 100 = 513.6\ mm^2$$

Provide 3 - 16 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (16)^2 = 603.2\ mm^2$

From the STAAD.Pro analysis, $M_z = 35.84\ kN \cdot m < M_{z,limit}$ at the end of the member (x = 3 m).

The actual moment is less than the limiting moment. Thus, the section is designed as a singly reinforced section.

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $\rho_t = 0.585$

Reinforcement at Start of Member (x = 3): top bars

$$A_{st} = 0.585 \times B \times d / 100 = 0.585 \times 300 \times 254 / 100 = 445.6\ mm^2$$

Provide 3 - 16 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (16)^2 = 603.2\ mm^2$

<u>Shear Strength of Concrete</u>

Provide percentage of tension steel, $\rho_t = 0.792$.

From Table 19 of IS 456:2000, for M20 grade concrete, shear stress: $T_c = 0.57\ N/mm^2$.

Factored shear force as per linear structural analysis in STAAD.Pro, $F_y = 80.29\ kN$.

The nominal shear stress, $T_v = F_y\ /\ (B \times d) = 80.29\ /\ (300 \times 254) = 1.054\ N/mm^2$.

The maximum shear stress from Table 20 of IS 456:2000 for M20 grade concrete, $T_{c,max} = 2.8\ N/mm^2$.

$T_c < T_v < T_{c,max}$, so provide shear reinforcement.

Shear strength of the reinforcement: $V_{us} = F_y - T_c \times B \times d = 80.29 - 0.57 \times 300 \times 254 \times 10^3 = 36.85\ kN$.

Provide 2 legged 8 mm diameter bar. So, the area of shear reinforcement, $A_{sv} = \pi/4 \times 8^2 \times 2 = 100.5\ mm^2$.

Using the formula of IS 456:200, the spacing for vertical stirrups:

$$S_v = 0.87\left(F_y \times A_{sv} \times d / V_{us}\right) = 0.87\left(250 \times 100.5 \times 256 / 36.85\right) = 250.2\ mm$$

Per Cl. 26.5.1.5 of IS 456:2000, the maximum spacing for of shear reinforcement is the least of:

$0.75 \times d = 0.75 \times 256 = 191\ mm$
$300\ mm$
$(\ 0.87 \times f_y \times A_{sv}\ )\ /\ (0.4 \times B) = 302.5\ mm$
$S_v = 250.2\ mm$

So, provide 2 legged 8 mm diameter bar stirrups at 190 mm center to center.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Reqd. area of steel at start section (mm^2)	513.6	519.32	negligible	
Reqd. area of steel at end section (mm^2)	445.6	449.98	negligible	
Spacing of shear reinforcement (mm)	190	185	negligible	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS456 2000-Singly Reinforced Square Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
```

```
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD 0.3
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
MEMBER RELEASE
59 START MX
59 END MX
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -30
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
1 1.5
LOAD COMB 6 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 4
1 0.9
LOAD COMB 7 COMBINATION LOAD CASE 7
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
LOAD LIST 3 TO 6
START CONCRETE DESIGN
CODE INDIAN
CLB 0.03 ALL
CLT 0.03 ALL
FC 20000 MEMB 59
FYMAIN 415000 MEMB 59
FYSEC 415000 MEMB 59
MAXMAIN 16 MEMB 59
MAXSEC 8 MEMB 59
MINMAIN 16 MEMB 59
MINSEC 8 MEMB 59
TRACK 2 MEMB 59
DESIGN BEAM 59
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
               ====================================================
               IS 456 - 2000   B E A M   D E S I G N   R E S U L T S
               ====================================================
```

```
===========================================================================
                   IS-456    L I M I T    S T A T E    D E S I G N
                  B E A M  N O.     59   D E S I G N  R E S U L T S
         M20                    Fe415 (Main)              Fe415 (Sec.)
       LENGTH:  3000.0 mm      SIZE:   300.0 mm X  300.0 mm   COVER: 40.0 mm
                        DESIGN LOAD SUMMARY (KN MET)

-----------------------------------------------------------------------------
----
 SECTION |FLEXURE  (Maxm. Sagg./Hogg./Eqv. moments) |          SHEAR
 (in mm) |    P        MZ       MX       ME   Load # |    VY       MX
VE  Load #

-----------------------------------------------------------------------------
----
     0.0 |    0.00     0.00     0.00     0.00      3 |   80.29     0.00
80.29     3
         |    0.00   -40.46     0.00   -40.46      3 |
   250.0 |    0.00     0.00     0.00     0.00      3 |   67.16     0.00
67.16     3
         |    0.00   -22.03     0.00   -22.03      3 |
   500.0 |    0.00     0.00     0.00     0.00      3 |   54.04     0.00
54.04     3
         |    0.00    -6.88     0.00    -6.88      3 |
   750.0 |    0.00     4.99     0.00     4.99      3 |   40.91     0.00
40.91     3
         |    0.00     0.00     0.00     0.00      3 |
  1000.0 |    0.00    13.58     0.00    13.58      3 |   27.79     0.00
27.79     3
         |    0.00     0.00     0.00     0.00      3 |
  1250.0 |    0.00    18.89     0.00    18.89      3 |   14.66     0.00
14.66     3
         |    0.00     0.00     0.00     0.00      3 |
  1500.0 |    0.00    20.91     0.00    20.91      3 |    1.54     0.00
1.54     3
         |    0.00     0.00     0.00     0.00      3 |
  1750.0 |    0.00    19.66     0.00    19.66      3 |  -11.59     0.00
11.59     3
         |    0.00     0.00     0.00     0.00      3 |
  2000.0 |    0.00    15.12     0.00    15.12      3 |  -24.71     0.00
24.71     3
         |    0.00     0.00     0.00     0.00      3 |
  2250.0 |    0.00     7.30     0.00     7.30      3 |  -37.84     0.00
37.84     3
         |    0.00     0.00     0.00     0.00      3 |
  2500.0 |    0.00     0.00     0.00     0.00      3 |  -50.96     0.00
50.96     3
         |    0.00    -3.80     0.00    -3.80      3 |
  2750.0 |    0.00     0.00     0.00     0.00      3 |  -64.09     0.00
64.09     3
         |    0.00   -18.18     0.00   -18.18      3 |
  3000.0 |    0.00     0.00     0.00     0.00      3 |  -77.21     0.00
77.21     3
         |    0.00   -35.84     0.00   -35.84      3 |

---------------------------------------------------------------------------
      STAAD SPACE                                                -- PAGE NO.
35
```

```
                    SUMMARY OF REINF. AREA (Sq.mm)

--------------------------------------------------------------------------
   SECTION |         TOP              |         BOTTOM           |   STIRRUPS
   (in mm) | Reqd./Provided reinf.    | Reqd./Provided reinf.    |  (2 legged)

--------------------------------------------------------------------------
       0.0 |  519.32/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm
     250.0 |  261.02/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm
     500.0 |  184.34/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm
     750.0 |    0.00/ 402.12( 2-16d )|  154.84/ 603.19( 3-16d )|  8d  @ 185 mm
    1000.0 |    0.00/ 402.12( 2-16d )|  156.06/ 603.19( 3-16d )|  8d  @ 185 mm
    1250.0 |    0.00/ 402.12( 2-16d )|  221.19/ 603.19( 3-16d )|  8d  @ 185 mm
    1500.0 |    0.00/ 402.12( 2-16d )|  246.77/ 603.19( 3-16d )|  8d  @ 185 mm
    1750.0 |    0.00/ 402.12( 2-16d )|  230.86/ 603.19( 3-16d )|  8d  @ 185 mm
    2000.0 |    0.00/ 402.12( 2-16d )|  174.69/ 603.19( 3-16d )|  8d  @ 185 mm
    2250.0 |    0.00/ 402.12( 2-16d )|  154.84/ 603.19( 3-16d )|  8d  @ 185 mm
    2500.0 |  184.34/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm
    2750.0 |  212.33/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm
    3000.0 |  449.98/ 603.19( 3-16d )|    0.00/ 402.12( 2-16d )|  8d  @ 185 mm

--------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT DISTANCE d (EFFECTIVE DEPTH) FROM FACE OF THE
SUPPORT

---------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT   452.0 mm AWAY FROM START SUPPORT
     VY =    56.56 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @ 185 mm c/c
   SHEAR DESIGN RESULTS AT   452.0 mm AWAY FROM END SUPPORT
     VY =   -53.48 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @ 185 mm c/c

==========================================================================
   ********************END OF BEAM DESIGN RESULTS********************
```

Related Links

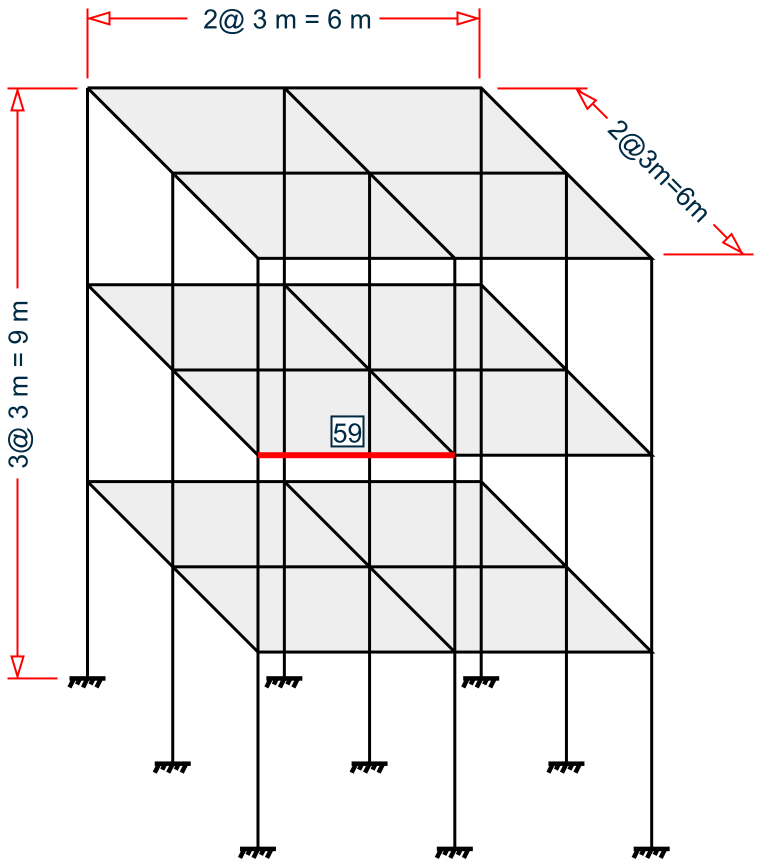
- *D8.E.5 Beam Design* (on page 1677)

V. IS13920 2016-Singly Reinforced Rectangular Beam

Verify the design of a rectangular beam per IS 13920:2016.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:



Section properties:

Length of beam, $L = 3{,}000\ mm$ (clear span)
Width of beam, $B = 250\ mm$
Depth of beam, $D = 300\ mm$
Clear cover = $30\ mm$
Diameter of main reinforcement = $12\ mm$
Diameter of stirrups = $8\ mm$

Material properties:

Characteristic strength of concrete, $f_{ck} = 20\ N/mm^2$
Yield strength of steel, $f_y = 415\ N/mm^2$

Loads: total uniform force = $25\ kN/m$

Validation

Effective Depth, $d = 300 - 30 - 8 - (12/2) = 256\ mm$

Depth of Compression Reinforcement, $d' = 30 + (12/2) + 8 = 44\ mm$

Check for Member Size

Width to Depth ratio of the member, $B/D = 250/300 = 0.833$ (Clause 6.1.1; IS 13920:2016)

Width of Beam, $B = 250\ mm > 200\ mm$ (Clause 6.1.2; IS 13920:2016)

As per clause 6.1.3 of IS 13920:2016, depth of beam shall not have more than 1/4 of clear span. So, the allowable depth not more than, $L/4 = 3000/4 = 750\ mm > 300\ mm$. Hence ok. (Clause 6.1.3; IS 13920:2016)

Width of the supporting member, C2 = 400 mm and Breadth of the supporting member, C1 = 400 mm. So, the width of member, B, shall not exceed smaller of (C2) and (C1) = 400 mm or C2 + smaller of (C1) and (C2) = 800 mm. Hence ok. (Clause 6.1.4; IS 13920:2016)

<u>Check for Axial Stress</u>

Axial force from STAAD.Pro analysis, $F_x = 0.299\ kN$

Factored axial compressive stress: $F_x\ /\ (B \times D) = 0.004 < 0.08 f_{ck} = 1.6$, Hence OK (Clause 6.1; IS 13920:2016)

Design for Flexure

Minimum longitudinal reinforcement, $p_{min} = 0.24 \times \sqrt{(f_{ck})} / f_y \times B \times d = 165.5\ mm^2$

Maximum longitudinal reinforcement, $p_{max} = 0.025 \times B \times d = 1{,}600\ mm^2$

Limiting moment value, $M_{z,limit} = 0.138 f_{ck} \times B \times d^2 = 45.22\ kN \cdot m$

From the STAAD.Pro analysis, $M_z = 30.28\ kN \cdot m < M_{z,limit}$ at the start of the member (x = 0).

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $p_t = 0.583$
Percentage of compression reinforcement $p_c = 0.292$

<u>Reinforcement at Start of Member (x = 0): top bars</u>

$$A_{st,top\_a} = 0.583 \times B \times d/100 = 373.1.\ mm^2$$

Provide 4 - 12 mm diameter bars. Actual $A_{st,\ top} = 4 \times (\pi/4) \times (12)^2 = 452.4\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

<u>Reinforcement at Start of Member (x = 0): bottom bars</u>

$$A_{st,bot\_a} = 0.292 \times B \times d/100 = 186.9\ mm^2$$

Provide 2 - 12 mm diameter bars. Actual $A_{st,\ bot} = 2 \times (\pi/4) \times (12)^2 = 226.2\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

From the STAAD.Pro analysis, $M_z = 24.56\ kN \cdot m < M_{z,limit}$ at the end of the member (x = 3 m).

From SP16 table 2 flexure, reinforcement percentage for singly reinforced section:

Percentage of tension reinforcement $p_t = 0.451$
Percentage of compression reinforcement $p_c = 0.225$.

<u>Reinforcement at End of Member (x = 3 m): top bars</u>

$$A_{st,top\_b} = 0.451 \times B \times d/100 = 288.6\ mm^2$$

Provide 3 - 12 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (12)^2 = 339.3\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

<u>Reinforcement at End of Member (x = 3 m): bottom bars</u>

$$A_{st,bot\_b} = 0.225 \times B \times d/100 = 144\ mm^2$$

Provide 2 - 12 mm diameter bars. Actual $A_{st,\ bot} = 2×(\pi / 4)×(12)^2 = 226.2\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

As per clause 6.2.4 of IS 13920:2016, any intermediate section, on top or bottom face should have at least 452.4 mm^2 /4 = 113.1 mm^2 of steel as the main reinforcement. So the provided reinforcement satisfies this condition.

Design for Shear

Calculation of resistance moment based on the area of steel provided:

Sagging resistance moment at start:

$$M_{z,as} = 0.87×f_y×A_{st,bot\_a}×d×[1 - A_{st,bot\_a}×f_y / (B×d×f_{ck})] = 19.37\ kN·m$$

Hogging resistance moment at start:

$$M_{z,ah} = 0.87×f_y×A_{st,top\_a}×d×[1 - A_{st,top\_a}×f_y / (B×d×f_{ck})] = 35.68\ kN·m$$

Sagging resistance moment at end:

$$M_{z,bs} = 0.87×f_y×A_{st,bot\_b}×d×[1 - A_{st,bot\_b}×f_y / (B×d×f_{ck})] = 19.37\ kN·m$$

Hogging resistance moment at end:

$$M_{z,bh} = 0.87×f_y×A_{st,top\_b}×d×[1 - A_{st,top\_b}×f_y / (B×d×f_{ck})] = 27.91\ kN·m$$

Simple shear from gravity load on the span:

$$V_a = 1.2×25\ kN/m × 3/2 = 45\ kN$$

Factor shear force as per linear structural analysis in STAAD.Pro, $F_y = 58.16\ kN$

Calculation of shear force due to formation of plastic hinge at both ends of the beam plus the factored gravity load on the span.

For sway to right:

$$V_{u,ar} = V_a - 1.4×(M_{z,as} + M_z×b×h/3) = 22.93\ kN$$

$$V_{u,br} = V_a + 1.4×(M_{z,as} + M_z×b×h/3) = 67.01\ kN$$

For sway to left:

$$V_{u,al} = V_a + 1.4×(M_{z,as} + M_z×b×h/3) = 70.69\ kN$$

$$V_{u,bl} = V_a - 1.4×(M_{z,as} + M_z×b×h/3) = 19.31\ kN$$

Design shear force at start: $V_{ua} = 70.69\ kN$

Design shear force at end: $V_{ub} = 67.01\ kN$

Design shear force, $V_{ua} = 70.69\ kN$ (Considering clause 6.3.4(c) of IS13920:2016)

Provide 2 legged 8 mm diameter bar. Thus the area of shear reinforcement:

$$A_v = 2×(\pi/4)×(8)^2 = 100.5\ mm^2$$

Using the formula from IS 456:2000, the spacing for vertical stirrups is:

$$S_v = 0.87×f_v×A_{sv}×d / V_{ua}) = 131.4\ mm$$

Per clause 6.3.5 of IS13920:2016, the spacing of the links over a length of $2d$ at either end of the beam shall be the least of:

a. $d / 4 = 64\ mm$

b. $6\times$ bar diameter of the smallest bars = $6\times12 = 72\ mm$
c. $100\ mm$
d. $S_v = 131\ mm$

Thus, it should be not greater than 64 mm. Therefore, provide 2 legged - 8 mm links @ 60 mm center to center at left and right end over a length of $2d = 512\ mm$.

Per Clause 6.3.5.2 of IS13920:2016, the spacing of the stirrups in the mid-span shall not exceed $d/2 = 128\ mm$. Thus, use a spacing of 125 mm elsewhere.

Results

Table 877: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Required area of steel at top of start section (mm^2)	373.1	377.0	1.0%	
Required area of steel at bottom of start section (mm^2)	186.9	188.5	0.6%	
Required area of steel at top of end section (mm^2)	288.6	296.92	2.9%	negligible difference that give same bar selection
Required area of steel at bottom of end section (mm^2)	144.0	148.46	3.1%	negligible difference that give same bar selection
Spacing of shear reinforcement at ends (mm)	60	60	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS13920 2016-Singly Reinforced Rectangular Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
****************************************************************************
*
*This Problem is created to verify the design results produced by the program
*
*with hand calculated design results for a rectangular(300 mm * 250 mm)
singly*
*reinforced  beam section as per IS13920:2016
```

```
*
******************************************************************************
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD
0.25
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -20
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
```

```
1 1.5
LOAD COMB 6 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 4
1 0.9
LOAD COMB 7 COMBINATION LOAD CASE 7
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
START CONCRETE DESIGN
CODE IS13920 2016
GLD 7 MEMB 59
CLB 0.03 ALL
CLT 0.03 ALL
FC 20000 MEMB 59
TORSION 1 ALL
FYMAIN 415000 MEMB 59
FYSEC 415000 MEMB 59
MAXMAIN 12 MEMB 59
MAXSEC 8 MEMB 59
MINMAIN 12 MEMB 59
MINSEC 8 MEMB 59
TRACK 2 MEMB 59
DESIGN BEAM 59
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
                ====================================================
                IS-13920:2016  B E A M   D E S I G N   R E S U L T S
                ====================================================
   EUDL CONSIDERED ON MEMBER #     59 IS  25.00 N/MM.

============================================================================
                   IS-13920   L I M I T    S T A T E    D E S I G N
                  B E A M  N O.      59   D E S I G N  R E S U L T S
         M20                   Fe415 (Main)                Fe415 (Sec.)
       LENGTH:  3000.0 mm      SIZE:   250.0 mm X  300.0 mm   COVER: 40.0 mm
                          DESIGN LOAD SUMMARY (KN MET)

----------------------------------------------------------------------------
   SECTION |FLEXURE  (Maxm. Sagging/Hogging moments)|          SHEAR
   (in mm) |     P        MZ         MX    Load Case |    VY        MX  Load
Case

----------------------------------------------------------------------------
       0.0 |     0.00      0.00       0.00     1      |  58.16     0.00      3
           |     0.00    -30.28       0.00     3      |
     250.0 |     0.00      0.00       0.00     1      |  48.78     0.00      3
           |     0.00    -16.91       0.00     3      |
     500.0 |     0.00      0.00       0.00     1      |  39.41     0.00      3
           |     0.00     -5.89       0.00     3      |
     750.0 |     0.00      2.79       0.00     3      |  30.03     0.00      3
           |     0.00      0.00       0.00     1      |
    1000.0 |     0.00      9.13       0.00     3      |  20.66     0.00      3
           |     0.00      0.00       0.00     1      |
    1250.0 |     0.00     13.12       0.00     3      |  11.28     0.00      3
```

```
          |    0.00      0.00      0.00     1      |
   1500.0 |    0.00     14.77      0.00     3      |     1.91     0.00      3
          |    0.00      0.00      0.00     1      |
   1750.0 |    0.00     14.07      0.00     3      |    -7.47     0.00      3
          |    0.00      0.00      0.00     1      |
   2000.0 |    0.00     11.03      0.00     3      |   -16.84     0.00      3
          |    0.00      0.00      0.00     1      |
   2250.0 |    0.00      5.65      0.00     3      |   -26.22     0.00      3
          |    0.00      0.00      0.00     1      |
   2500.0 |    0.00      0.00      0.00     1      |   -35.59     0.00      3
          |    0.00     -2.08      0.00     3      |
   2750.0 |    0.00      0.00      0.00     1      |   -44.97     0.00      3
          |    0.00    -12.15      0.00     3      |
   3000.0 |    0.00      0.00      0.00     1      |   -54.34     0.00      3
          |    0.00    -24.56      0.00     3      |
   *** DESIGN SHEAR FORCE AT SECTION     0.0 IS    70.44 KN.
                                             - CLAUSE 6.3.3 OF IS-13920
   *** DESIGN SHEAR FORCE AT SECTION  3000.0 IS    66.86 KN.
                                             - CLAUSE 6.3.3 OF IS-13920
   Note :
   MOMENT OF RESISTANCE IS CALCULATED BASED ON THE AREA OF STEEL PROVIDED.
   IF AREA OF STEEL PROVIDED IS MUCH HIGHER COMPARED TO AREA OF STEEL
   REQUIRED MOMENT OF RESISTANCE WILL INCREASE WHICH MAY INCREASE DESIGN
   SHEAR FORCE.

----------------------------------------------------------------------------
      STAAD SPACE                                                -- PAGE NO.
35
                        SUMMARY OF REINF. AREA (Sq.mm)

----------------------------------------------------------------------------
   SECTION |           TOP            |          BOTTOM          |   STIRRUPS
   (in mm) | Reqd./Provided reinf.    | Reqd./Provided reinf.    |  (2 legged)

----------------------------------------------------------------------------
       0.0 |  377.00/ 452.39( 4-12d )|  188.50/ 226.19( 2-12d )|  8d  @  60 mm
     250.0 |  197.26/ 226.19( 2-12d )|   94.25/ 226.19( 2-12d )|  8d  @  60 mm
     500.0 |  164.23/ 226.19( 2-12d )|   94.25/ 226.19( 2-12d )|  8d  @  60 mm
     750.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    1000.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    1250.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    1500.0 |   94.25/ 226.19( 2-12d )|  170.69/ 226.19( 2-12d )|  8d  @ 120 mm
    1750.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    2000.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    2250.0 |   94.25/ 226.19( 2-12d )|  164.23/ 226.19( 2-12d )|  8d  @ 120 mm
    2500.0 |  164.23/ 226.19( 2-12d )|   94.25/ 226.19( 2-12d )|  8d  @  60 mm
    2750.0 |  164.23/ 226.19( 2-12d )|   94.25/ 226.19( 2-12d )|  8d  @  60 mm
    3000.0 |  296.92/ 339.29( 3-12d )|  148.46/ 226.19( 2-12d )|  8d  @  60 mm

----------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT DISTANCE 2d (TWICE EFFECTIVE DEPTH) FROM FACE OF
THE SUPPORT
   SHEAR DESIGN RESULTS AT   708.0 mm AWAY FROM START SUPPORT
     VY =    31.76 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @  60 mm c/c
   SHEAR DESIGN RESULTS AT   708.0 mm AWAY FROM END SUPPORT
     VY =   -27.94 MX =     0.00 LD=    3
     Provide 2 Legged  8d  @  60 mm c/c
```

```
CONFINING REINFORCEMENT  : Provide reinforcement as per CL 9.2.1 b)
                           of IS-13920:2016 at member start.
CONFINING REINFORCEMENT  : Provide reinforcement as per CL 9.2.1 b)
                           of IS-13920:2016 at member end.
********************END OF BEAM DESIGN RESULTS********************
```

Related Links

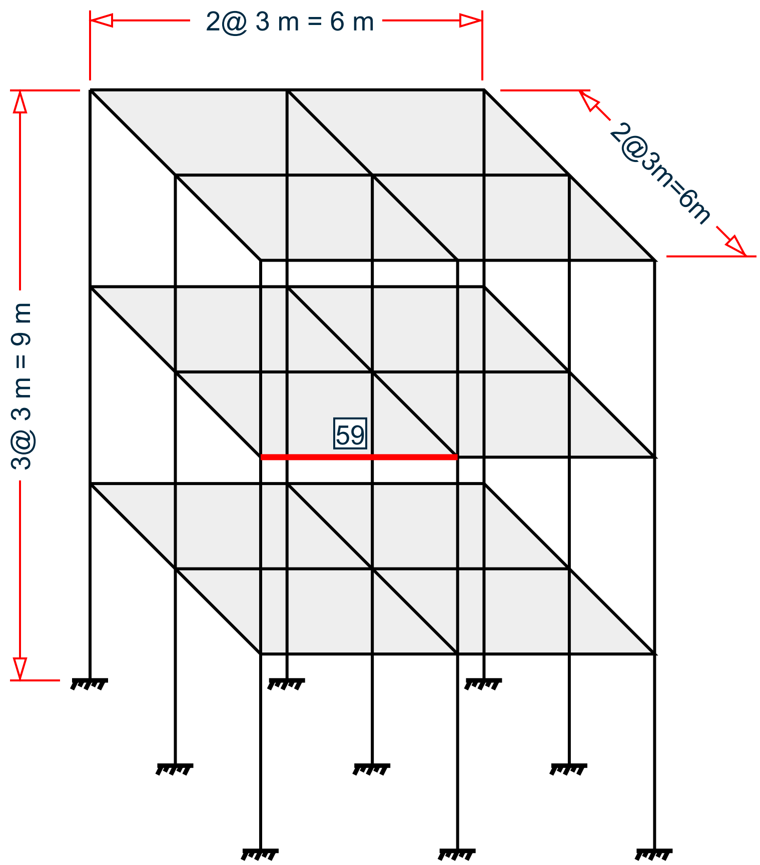
- *D8.F.2 Beam Design* (on page 1684)

V. IS13920 2016-Doubly Reinforced Rectangular Beam

Verify the design of a rectangular beam per IS 13920:2016.

Details

A three story, concrete structure is subject to gravity loads only. Beam 59 is to be designed:



Section properties:

Length of beam, $L = 3{,}000\ mm$ (clear span)
Width of beam, $B = 250\ mm$
Depth of beam, $D = 300\ mm$
Clear cover = $30\ mm$
Diameter of main reinforcement = $16\ mm$

Diameter of stirrups = $8\ mm$

Material properties:

Characteristic strength of concrete, $f_{ck} = 20\ N/mm^2$
Yield strength of steel, $f_y = 415\ N/mm^2$

Loads: total uniform force = $50\ kN/m$

Validation

Effective Depth, $d = 300 – 30 – 8 – (16/2) = 254\ mm$

Depth of Compression Reinforcement, $d' = 30 + (16/2) + 8 = 46\ mm$

Check for Member Size

Width to Depth ratio of the member, $B/D = 250/300 = 0.833$ (Clause 6.1.1; IS 13920:2016)

Width of Beam, $B = 250\ mm > 200\ mm$ (Clause 6.1.2; IS 13920:2016)

As per clause 6.1.3 of IS 13920:2016, depth of beam shall not have more than 1/4 of clear span. So, the allowable depth not more than, $L/4 = 3000/4 = 750\ mm > 300\ mm$. Hence ok. (Clause 6.1.3; IS 13920:2016)

Width of the supporting member, C2 = 400 mm and Breadth of the supporting member, C1 = 400 mm. So, the width of member, B, shall not exceed smaller of (C2) and (C1) = 400 mm or C2 + smaller of (C1) and (C2) = 800 mm. Hence ok. (Clause 6.1.4; IS 13920:2016)

Axial force from STAAD.Pro analysis, $F_x = 6.805\ kN$

Factored axial compressive stress: $F_x\ /\ (B \times D) = 0.09 < 0.08 f_{ck} = 1.6$, Hence OK (Clause 6.1; IS 13920:2016)

Design for Flexure

Minimum longitudinal reinforcement, $p_{min} = 0.24 \times \sqrt{(f_{ck})}/f_y \times B \times d = 164.2\ mm^2$

Maximum longitudinal reinforcement, $p_{max} = 0.025 \times B \times d = 1,588\ mm^2$

Limiting moment value, $M_{z,limit} = 0.138 f_{ck} \times B \times d^2 = 44.52\ kN \cdot m$

From the STAAD.Pro analysis, $M_z = 55.44\ kN \cdot m > M_{z,limit}$ at the start of the member (x = 0).

So, the actual moment is greater than limiting moment then the section will be design as doubly reinforced section.

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section:

Percentage of tension reinforcement $p_t = 1.19$
Percentage of compression reinforcement $p_c = 0.595$

Reinforcement at Start of Member (x = 0): top bars

$$A_{st,top\_a} = 1.19 \times B \times d/100 = 755.7.\ mm^2$$

Provide 4 - 16 mm diameter bars. Actual $A_{st,\ top} = 4 \times (\pi/4) \times (16)^2 = 804.2\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

Reinforcement at Start of Member (x = 0): bottom bars

$$A_{st,bot\_a} = 0.595 \times B \times d/100 = 377.8\ mm^2$$

Provide 2 - 16 mm diameter bars. Actual $A_{st,\ bot} = 2 \times (\pi/4) \times (16)^2 = 402.1\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

From the STAAD.Pro analysis, $M_z = 54.11\ kN{\cdot}m > M_{z,limit}$ at the end of the member (x = 3 m).

From SP16 table 50 flexure, reinforcement percentage for doubly reinforced section Percentage of tension reinforcement $p_t = 1.161$ and compression reinforcement $p_c = 0.581$.

Reinforcement at End of Member (x = 3 m): top bars

$$A_{st,top\_b} = 1.161 \times B \times d/100 = 737.4\ mm^2$$

Provide 4 - 16 mm diameter bars. Actual $A_{st,\ top} = 3 \times (\pi/4) \times (16)^2 = 804.2\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

Reinforcement at End of Member (x = 3 m): bottom bars

$$A_{st,bot\_b} = 0.225 \times B \times d/100 = 368.7\ mm^2$$

Provide 2 - 16 mm diameter bars. Actual $A_{st,\ bot} = 2 \times (\pi/4) \times (16)^2 = 402.1\ mm^2$

Thus, clause 6.2.1(b) and 6.2.2 of IS 13920:2016 are satisfied.

As per clause 6.2.4 of IS 13920:2016, any intermediate section, on top or bottom face should have at least 804.2 mm^2 /4 = 201.1 mm^2 of steel as the main reinforcement. So the provided reinforcement satisfies this condition.

Design for Shear

Calculation of resistance moment based on the area of steel provided:

Sagging resistance moment at start:

$$M_{z,as} = 0.87 \times f_y \times A_{st,bot\_a} \times d \times [1 - A_{st,bot\_a} \times f_y / (B \times d \times f_{ck})] = 32.03\ kN{\cdot}m$$

Hogging resistance moment at start:

$$M_{z,ah} = 0.87 \times f_y \times A_{st,top\_a} \times d \times [1 - A_{st,top\_a} \times f_y / (B \times d \times f_{ck})] = 54.37\ kN{\cdot}m$$

Sagging resistance moment at end:

$$M_{z,bs} = 0.87 \times f_y \times A_{st,bot\_b} \times d \times [1 - A_{st,bot\_b} \times f_y / (B \times d \times f_{ck})] = 32.03\ kN{\cdot}m$$

Hogging resistance moment at end:

$$M_{z,bh} = 0.87 \times f_y \times A_{st,top\_b} \times d \times [1 - A_{st,top\_b} \times f_y / (B \times d \times f_{ck})] = 54.37\ kN{\cdot}m$$

Simple shear from gravity load on the span:

$$V_a = 1.2 \times 50\ kN/m \times 3/2 = 90\ kN$$

Factor shear force as per linear structural analysis in STAAD.Pro, $F_y = 112.9\ kN$

Calculation of shear force due to formation of plastic hinge at both ends of the beam plus the factored gravity load on the span.

For sway to right:

$$V_{u,ar} = V_a - 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 49.68\ kN$$

$$V_{u,br} = V_a + 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 130.3\ kN$$

For sway to left:

$$V_{u,al} = V_a + 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 130.3\ kN$$

$$V_{u,bl} = V_a - 1.4 \times (M_{z,as} + M_z \times b \times h/3) = 49.68\ kN$$

Design shear force at start: $V_{ua} = 130.3\ kN$

Design shear force at end: $V_{ub} = 130.3\ kN$

Design shear force, $V_{ua} = 130.3\ kN$ (Considering clause 6.3.4(c) of IS13920:2016)

Provide 2 legged 8 mm diameter bar. Thus the area of shear reinforcement:

$$A_v = 2\times(\pi/4)\times(8)^2 = 100.5\ mm^2$$

Using the formula from IS 456:2000, the spacing for vertical stirrups is:

$$S_v = 0.87\times f_v\times A_{sv}\times d\ /\ V_{ua}) = 70.7\ mm$$

Per clause 6.3.5 of IS13920:2016, the spacing of the links over a length of $2d$ at either end of the beam shall be the least of:

a. $d/4 = 64\ mm$

b. $6\times$ bar diameter of the smallest bars = $6\times 16 = 96\ mm$

c. $100\ mm$

d. $S_v = 71\ mm$

Thus, it should be not greater than 64 mm. Therefore, provide 2 legged - 8 mm links @ 60 mm center to center at left and right end over a length of $2d = 508\ mm$.

Per Clause 6.3.5.2 of IS13920:2016, the spacing of the stirrups in the mid-span shall not exceed $d/2 = 127\ mm$. Thus, use a spacing of 125 mm elsewhere.

Results

Table 878: Comparison of results

Result Type	Reference	STAAD.Pro	Difference	Comments
Required area of steel at top of start section (mm^2)	769.4	772.79	negligible	
Required area of steel at bottom of start section (mm^2)	384.7	386.39	negligible	
Required area of steel at top of end section (mm^2)	751.2	754.73	negligible	
Required area of steel at bottom of end section (mm^2)	375.6	377.37	negligible	
Spacing of shear reinforcement at ends (mm)	60	60	none	

STAAD.Pro Input File

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\10 Concrete Design\India\IS13920 2016-Doubly Reinforced Rectangular Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 25-Jul-18
END JOB INFORMATION
INPUT WIDTH 79
******************************************************************************
*
*This Problem is created to verify the design results produced by the program
*
*with hand calculated design results for a rectangular(300 mm * 250 mm)
doubly*
*reinforced  beam section as per IS13920:2016
*
*
*
******************************************************************************
*
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 3 0; 3 0 6 0; 4 0 9 0; 5 3 9 0; 6 6 9 0; 7 6 6 0; 8 6 3 0;
9 6 0 0; 10 3 6 0; 11 3 3 0; 12 3 0 0; 13 0 0 3; 14 0 3 3; 15 0 6 3; 16 0 9 3;
17 3 9 3; 18 6 9 3; 19 6 6 3; 20 6 3 3; 21 6 0 3; 22 3 6 3; 23 3 3 3; 24 3 0
3;
25 0 0 6; 26 0 3 6; 27 0 6 6; 28 0 9 6; 29 3 9 6; 30 6 9 6; 31 6 6 6; 32 6 3
6;
33 6 0 6; 34 3 6 6; 35 3 3 6; 36 3 0 6;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 7; 7 7 8; 8 8 9; 9 3 10; 10 10 7;
11 2 11; 12 11 8; 13 5 10; 14 10 11; 17 11 12; 18 2 14; 19 3 15; 20 4 16;
21 5 17; 22 6 18; 23 7 19; 24 8 20; 25 10 22; 26 11 23; 27 13 14; 28 14 15;
29 15 16; 30 16 17; 31 17 18; 32 18 19; 33 19 20; 34 20 21; 35 15 22; 36 22
19;
37 14 23; 38 23 20; 39 17 22; 40 22 23; 41 23 24; 42 14 26; 43 15 27; 44 16
28;
45 17 29; 46 18 30; 47 19 31; 48 20 32; 49 22 34; 50 23 35; 51 25 26; 52 26
27;
53 27 28; 54 28 29; 55 29 30; 56 30 31; 57 31 32; 58 32 33; 59 27 34; 60 34
31;
61 26 35; 62 35 32; 63 29 34; 64 34 35; 65 35 36;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 27579
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 PRIS YD 0.3 ZD
0.25
```

```
1 TO 3 6 TO 8 13 14 17 27 TO 29 32 TO 34 39 TO 41 51 TO 53 56 TO 58 -
63 TO 65 PRIS YD 0.4 ZD 0.4
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 9 12 13 21 24 25 33 36 FIXED
LOAD 1 LOADTYPE Dead  TITLE DL
MEMBER LOAD
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -3.125
4 5 9 TO 12 18 TO 26 30 31 35 TO 38 42 TO 50 54 55 59 TO 62 UNI GY -1.875
LOAD 2 LOADTYPE Live  TITLE LL
MEMBER LOAD
4 5 20 22 44 46 54 55 UNI GY -7.875
21 30 31 45 UNI GY -15.75
9 TO 12 18 19 23 24 42 43 47 48 59 TO 62 UNI GY -45
25 26 35 TO 38 49 50 UNI GY -90
LOAD COMB 3 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 1
1 1.5 2 1.5
LOAD COMB 4 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 2
1 1.2 2 1.2
LOAD COMB 5 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 3
1 1.5
LOAD COMB 6 GENERATED IS:456/IS:800 GENRAL_STRUCTURES 4
1 0.9
LOAD COMB 7 COMBINATION LOAD CASE 7
1 1.0 2 1.0
PERFORM ANALYSIS PRINT ALL
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
START CONCRETE DESIGN
CODE IS13920 2016
GLD 7 MEMB 59
CLB 0.03 ALL
CLT 0.03 ALL
FC 20000 MEMB 59
FYMAIN 415000 MEMB 59
FYSEC 415000 MEMB 59
MAXMAIN 16 MEMB 59
MAXSEC 8 MEMB 59
MINMAIN 16 MEMB 59
MINSEC 8 MEMB 59
TRACK 2 MEMB 59
DESIGN BEAM 59
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
                ====================================================
                IS-13920:2016  B E A M   D E S I G N   R E S U L T S
                ====================================================
   EUDL CONSIDERED ON MEMBER #     59 IS  50.00 N/MM.

====================================================================
                   IS-13920   L I M I T    S T A T E    D E S I G N
                  B E A M  N O.     59   D E S I G N  R E S U L T S
         M20                    Fe415 (Main)              Fe415 (Sec.)
```

```
      LENGTH:  3000.0 mm      SIZE:   250.0 mm X  300.0 mm   COVER: 40.0 mm
                          DESIGN LOAD SUMMARY (KN MET)

-----------------------------------------------------------------------------
   SECTION |FLEXURE  (Maxm. Sagging/Hogging moments)|         SHEAR
   (in mm) |     P         MZ         MX   Load Case |    VY       MX  Load
Case

-----------------------------------------------------------------------------
       0.0 |     0.00      0.00      0.00     1      |  112.94     0.77      3
           |     0.00    -55.44      0.77     3      |
     250.0 |     0.00      0.00      0.00     1      |   94.19     0.77      3
           |     0.00    -29.55      0.77     3      |
     500.0 |     0.00      0.00      0.00     1      |   75.44     0.77      3
           |     0.00     -8.35      0.77     3      |
     750.0 |     0.00      8.17      0.77     3      |   56.69     0.77      3
           |     0.00      0.00      0.00     1      |
    1000.0 |     0.00     20.00      0.77     3      |   37.94     0.77      3
           |     0.00      0.00      0.00     1      |
    1250.0 |     0.00     27.14      0.77     3      |   19.19     0.77      3
           |     0.00      0.00      0.00     1      |
    1500.0 |     0.00     29.60      0.77     3      |    0.44     0.77      3
           |     0.00      0.00      0.00     1      |
    1750.0 |     0.00     27.36      0.77     3      |  -18.31     0.77      3
           |     0.00      0.00      0.00     1      |
    2000.0 |     0.00     20.44      0.77     3      |  -37.06     0.77      3
           |     0.00      0.00      0.00     1      |
    2250.0 |     0.00      8.84      0.77     3      |  -55.81     0.77      3
           |     0.00      0.00      0.00     1      |
    2500.0 |     0.00      0.00      0.00     1      |  -74.56     0.77      3
           |     0.00     -7.46      0.77     3      |
    2750.0 |     0.00      0.00      0.00     1      |  -93.31     0.77      3
           |     0.00    -28.44      0.77     3      |
    3000.0 |     0.00      0.00      0.00     1      | -112.06     0.77      3
           |     0.00    -54.11      0.77     3      |
   *** DESIGN SHEAR FORCE AT SECTION     0.0 IS   133.30 KN.
                                                - CLAUSE 6.3.3 OF IS-13920
   *** DESIGN SHEAR FORCE AT SECTION  3000.0 IS   133.30 KN.
                                                - CLAUSE 6.3.3 OF IS-13920
   Note :
   MOMENT OF RESISTANCE IS CALCULATED BASED ON THE AREA OF STEEL PROVIDED.
   IF AREA OF STEEL PROVIDED IS MUCH HIGHER COMPARED TO AREA OF STEEL
   REQUIRED MOMENT OF RESISTANCE WILL INCREASE WHICH MAY INCREASE DESIGN
   SHEAR FORCE.

-----------------------------------------------------------------------------
      STAAD SPACE                                                -- PAGE NO.
35
                        SUMMARY OF REINF. AREA (Sq.mm)

-----------------------------------------------------------------------------
   SECTION |          TOP              |          BOTTOM           |   STIRRUPS
   (in mm) | Reqd./Provided reinf.     | Reqd./Provided reinf.     |  (2 legged)

-----------------------------------------------------------------------------
       0.0 |  772.79/ 804.25( 4-16d )|  386.39/ 402.12( 2-16d )|  8d  @  60 mm
     250.0 |  384.93/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @  60 mm
     500.0 |  193.20/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @  60 mm
```

```
     750.0 |  193.20/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @ 100 mm
    1000.0 |  193.20/ 402.12( 2-16d )|  251.83/ 402.12( 2-16d )|  8d  @ 100 mm
    1250.0 |  193.20/ 402.12( 2-16d )|  349.91/ 402.12( 2-16d )|  8d  @ 100 mm
    1500.0 |  193.20/ 402.12( 2-16d )|  385.60/ 402.12( 2-16d )|  8d  @ 100 mm
    1750.0 |  193.20/ 402.12( 2-16d )|  353.09/ 402.12( 2-16d )|  8d  @ 100 mm
    2000.0 |  193.20/ 402.12( 2-16d )|  257.70/ 402.12( 2-16d )|  8d  @ 100 mm
    2250.0 |  193.20/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @ 100 mm
    2500.0 |  193.20/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @  60 mm
    2750.0 |  368.69/ 402.12( 2-16d )|  193.20/ 402.12( 2-16d )|  8d  @  60 mm
    3000.0 |  754.73/ 804.25( 4-16d )|  377.37/ 402.12( 2-16d )|  8d  @  60 mm

---------------------------------------------------------------------------
   SHEAR DESIGN RESULTS AT DISTANCE 2d (TWICE EFFECTIVE DEPTH) FROM FACE OF
THE SUPPORT
   SHEAR DESIGN RESULTS AT   704.0 mm AWAY FROM START SUPPORT
     VY =    60.14 MX =     0.77 LD=    3
     Provide 2 Legged  8d  @  60 mm c/c
   SHEAR DESIGN RESULTS AT   704.0 mm AWAY FROM END SUPPORT
     VY =   -59.26 MX =     0.77 LD=    3
     Provide 2 Legged  8d  @  60 mm c/c
   CONFINING REINFORCEMENT  : Provide reinforcement as per CL 9.2.1 b)
                              of IS-13920:2016 at member start.
   CONFINING REINFORCEMENT  : Provide reinforcement as per CL 9.2.1 b)
                              of IS-13920:2016 at member end.
   ********************END OF BEAM DESIGN RESULTS********************
```

Related Links

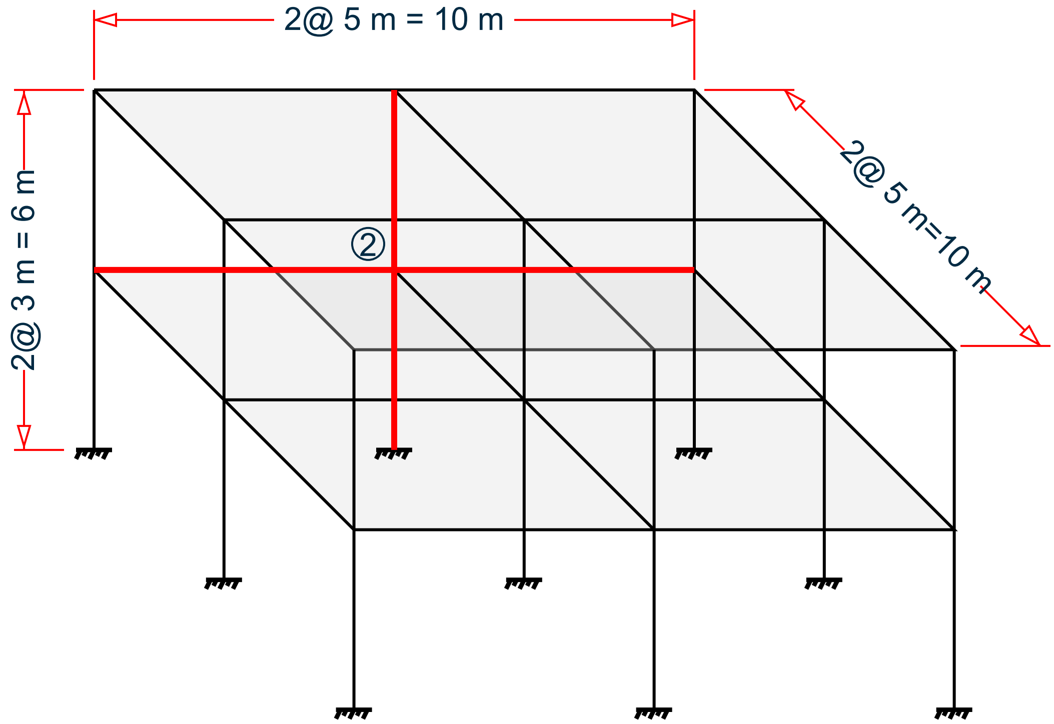
- *D8.F.2 Beam Design* (on page 1684)

V. IS13920 2016-Relative Strength Check

Verify the relative strength of beams and columns at a joint as per clause 7.2 of IS 13920:2016.

Details

Check the relative strength of beams and columns at joint 2 for the two-story structure shown.



The beams on each side of the joint are 400 mm × 400 mm square sections and are 5 m long. The columns above and below the joint are 400 mm × 400 mm square sections and are 3 m tall. The clear cover for both beams and columns is 30 mm on all faces.

Material properties:

- Characteristic strength of concrete, $f_{ck} = 30\ N/mm^2$
- Yield strength of steel, $f_y = 415\ N/mm^2$

Validation

Columns Strength

From Output file , Moment of Resistance based on reinforcement provided for column no. 22 along global X direction $(M_{uz1}) = 129.99\ kN\text{-}m$.

From Output file, Moment of Resistance based on reinforcement provided for column no. 1 along global X direction $(M_{uz2}) = 149.74\ kN\text{-}m$.

Total Moment of Resistance for column no. 1 and 22 along global X direction

$$M_{uz} = (M_{uz1}) + (M_{uz2}) = 129.99 + 149.74 = 279.73 \text{ KN-m.}$$

From Output file, Moment of Resistance based on reinforcement provided for column no. 22 along global Z direction $(M_{uy1}) = 129.99\ kN\text{-}m$.

From Output file, Moment of Resistance based on reinforcement provided for column no. 1 along global Z direction $(M_{uy2}) = 149.74\ kN\text{-}m$.

Total Moment of Resistance for column no. 1 and 22 along global Z direction

$$M_{uy} = (M_{uy1}) + (M_{uy2}) = 129.99 + 149.74 = 279.73 \text{ KN-m.}$$

Beams Strength

From output file , Moment of Resistance of Beam no. 2 due to hogging moment at start *55.48 kN-m*.

From output file , Moment of Resistance of Beam no. 2 due to sagging moment at start *55.48 kN-m*.

From output file , Moment of Resistance of Beam no. 9 due to hogging moment at start *208.14 kN-m*.

From output file , Moment of Resistance of Beam no. 9 due to sagging moment at start *117.32 kN-m*.

Total Moment of Resistance of Beam at joint no.2 along global X direction

Max *{(Beam no.2 hogging moment + Beam no.9 sagging moment),(Beam no.2 sagging moment + Beam no.9 hogging moment)}*

$$\text{Max } \{(55.48 + 117.32),(55.48 + 208.14)\} = 263.62 \text{ kN-m}$$

.

From output file , Moment of Resistance of Beam no. 4 due to hogging moment at start *55.48 kN-m*.

From output file , Moment of Resistance of Beam no. 4 due to sagging moment at start *55.48 kN-m*.

Beam is not extended in global (-Z) direction. So, taking zero hogging and sagging moment at global (-Z) direction

Total Moment of Resistance of Beam at joint no.2 along global Z direction

Max {(Beam no.4 hogging moment + 0),(Beam no.4 sagging moment + 0)}

$$= \text{Max } \{(55.48 + 0),(55.48 + 0)\} = 55.48 \text{ kN-m}$$

Checking Joint status for global X direction

As per Clause 7.2 of IS 13920 : 2016

Total Column Strength at joint no.2 along global X direction should be greater than or equal to 1.4 times Maximum Beam strength at joint no.2 along global X direction

$$M_{uz} = 279.73 < 1.4 \times 263.62 = 369.068$$

So , the joint no. 2 is *falling* along global X direction as per Clause 7.2 of IS 13920 : 2016.

Checking Joint status for global Z direction

As per Clause 7.2 of IS 13920 : 2016

Total Column Strength at joint no.2 along global Z direction should be greater than or equal to 1.4 times Maximum Beam strength at joint no.2 along global Z direction

$$M_{uy} = 279.73 < 1.4 \times 55.48 = 77.672$$

So , the joint no. 2 is *passing* along global Z direction as per Clause 7.2 of IS 13920 : 2016.

Results

Result Type	Reference	STAAD.Pro	Difference	Comments
Cl. 7.2 check at joint 2, X direction	Fails	Fails (“F”)	none	
Cl. 7.2 check at joint 2, Z direction	Passes	Passes (“P”)	none	

STAAD.Pro Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\India\IS13920 2016-Relative Strength check.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 13-Aug-20
END JOB INFORMATION
INPUT WIDTH 79
*****************************************************************************
*
*This Problem is created to verify the clause 7.2 of IS13920 2016 produced by
*
*the program
*
*Checking Relative Strength of Beam and Column at Joint no.2
*                                                            *
*****************************************************************************
*
UNIT METER KN
JOINT COORDINATES
1 0 2 0; 2 0 5 0; 3 5 5 0; 4 5 2 0; 5 0 2 5; 6 0 5 5; 7 5 5 5; 8 5 2 5;
9 -5 2 0; 10 -5 5 0; 11 -5 2 5; 12 -5 5 5; 13 0 2 10; 14 0 5 10; 15 5 5 10;
16 5 2 10; 17 -5 2 10; 18 -5 5 10; 19 0 8 0; 20 5 8 0; 21 0 8 5; 22 5 8 5;
23 -5 8 0; 24 -5 8 5; 25 0 8 10; 26 5 8 10; 27 -5 8 10;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 4 3; 4 2 6; 5 3 7; 6 5 6; 7 6 7; 8 8 7; 9 2 10; 10 6 12;
11 9 10; 12 10 12; 13 11 12; 14 6 14; 15 7 15; 16 12 18; 17 13 14; 18 14 15;
19 16 15; 20 14 18; 21 17 18; 22 2 19; 23 3 20; 24 6 21; 25 7 22; 26 10 23;
27 12 24; 28 14 25; 29 15 26; 30 18 27; 31 19 20; 32 19 21; 33 20 22; 34 21
22;
35 19 23; 36 21 24; 37 23 24; 38 21 25; 39 22 26; 40 24 27; 41 25 26; 42 25
27;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+07
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-05
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 30000
G 9.28141e+06
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
```

```
2 4 5 7 9 10 12 14 TO 16 18 20 31 TO 42 PRIS YD 0.4 ZD 0.4
MEMBER PROPERTY AMERICAN
1 3 8 11 13 22 23 25 TO 27 PRIS YD 0.4 ZD 0.4
MEMBER PROPERTY AMERICAN
6 17 19 24 28 29 PRIS YD 0.5 ZD 0.5
MEMBER PROPERTY
21 30 PRIS YD 0.3 ZD 0.3
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 4 5 8 9 11 13 16 17 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
MEMBER LOAD
2 4 5 7 9 10 12 14 TO 16 18 20 31 TO 42 UNI GY -50
JOINT LOAD
10 12 18 23 24 27 FX 100
2 3 10 19 20 23 FZ 50
6 15 FY 1500
21 26 FY -1500
2 3 10 12 18 FY 300
15 19 20 23 24 27 FY -300
25 FY -500
PERFORM ANALYSIS
PRINT MEMBER FORCES
START CONCRETE DESIGN
CODE IS13920 2016
CLT 0.03 ALL
CLB 0.03 ALL
FC 30000 ALL
FYSEC 415000 ALL
FYMAIN 415000 ALL
MAXMAIN 12 ALL
MINMAIN 12 ALL
GLD 1 ALL
TRACK 9 ALL
TRACK 4
DESIGN BEAM 2 4 5 7 9 10 12 14 TO 16 18 20 31 TO 42
DESIGN COLUMN 1 3 6 8 11 13 17 19 21 TO 30
END CONCRETE DESIGN
FINISH
```

STAAD.Pro Output

```
 JOINT
   RELATIVE STRENGTHS OF BEAMS AND COLUMNS AT A JOINT
   ==================================================
     COLUMN     JOINT   CONNECTED COLUMN   CONNECTED             BEAM
STATUS
     -------    -----   ----------------   --------------------------
----------
                                             X                  Z
X      Z
          1          2          22        2        9         4       -
F      P
         3          3          23        2        -         5       -
F      P
         6          6          24        7       10         4      14
```

```
F       F
          8           7           25          7         -          5       15
F       F
      STAAD SPACE                                                -- PAGE NO.
112
```

Related Links

- *D8.F.3 Column Design* (on page 1685)

V. United States

V. ACI 318-14 Circular Column

Calculate the axial capacity of a circular column subjected to axial compressive force, given Ast

Details

A circular, cantilever column of length 5ft with an ultimate axial load of 500 kip has the following concrete and steel properties.

Compressive strength of concrete (normal weight concrete), $f_c = 4{,}000\ psi$
Yield strength of steel, $f_y = 60{,}000\ psi$
Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5\ in$
Diameter of column, $D = 20\ in$
Strength reduction factor, $\phi = 0.65$
Number of #8 bars, $N = 6$
Rebar diameter for #8 bar, $d = 1\ in$

Factored axial load (axial demand), $P_u = 500\ kip$

Validation

Gross area of concrete section, $A_g = \pi D^2/4 = 314.159\ in^2$

Total area of steel reinforcement, $A_{st} = \left(\pi \frac{d^2}{4}\right) N = 4.712\ in^2$

Nominal axial strength at zero eccentricity:

$$P_0 = 0.85 \times f_c (A_g - A_{st}) + f_y \times A_{st} = 1.335 \times 10^3\ \text{kip}$$

For tied reinforced column:

$$P_{n(max)} = 0.8 \cdot P_0 = 1.068 \times 10^3\ \text{kip}$$

Capacity = $\phi P_{n(max)} = 694.129\ kip$

Ratio = $P_u / \phi P_{n(max)} = 0.72$

Results

Table 879: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Axial Capacity (kip)	694.129	694.079	negligible	
Ratio $P_u/\phi P_{n(max)}$	0.72	0.72	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-14 Circular Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Aug-17
JOB NAME Circular Column Design
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.66667
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -500
PERFORM ANALYSIS
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
CODE ACI 318 14
MAXMAIN 9 ALL
MINMAIN 8 ALL
TRACK 2 ALL
DESIGN COLUMN ALL
```

```
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0
                       ***********************************************
Units: KIP   , FEET   (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length:    5.000 |
| Critical Ratio :   0.720        Criteria: Axial                            |
| Critical Clause:   10.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------
| Shape: Circular    | Dia:    1.67 |
-------------------------------------
DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc         576.000 |                     | Ec     0.519E+06 |
| Steel    | Fy(main)  8639.999 | Fy(trans)  8639.999 | Es     0.418E+07 |
| Cover    | Top          0.125 | Bottom        0.125 | Sides      0.125 |
-------------------------------------------------------------------------
CRITICAL STRENGTH RESULTS
-----------------------------------------------------------------------
| Category |    Demand     |  Min Capacity  |  Max Capacity  |  Ratio  |
-----------------------------------------------------------------------
| Axial    |       -500.000|       -694.079 |        255.960 |   0.720 |
| Flexure  |          0.000|       -148.011 |        148.011 |   0.000 |
| Shear Y  |          0.000|          0.000 |          0.000 |   0.000 |
| Shear Z  |          0.000|          0.000 |          0.000 |   0.000 |
| Torsion  |          0.000|          0.000 |          0.000 |   0.000 |
-----------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-----------------------------------------------------------------------
| Distance |  Ast-reqd  |  Ast-prov  | No(s)bars| Size | No of Layers |
-----------------------------------------------------------------------
|    0.000 |      0.022 |      0.033 |     6    |  # 8 |       1      |
|    1.250 |      0.022 |      0.033 |     6    |  # 8 |       1      |
|    2.500 |      0.022 |      0.033 |     6    |  # 8 |       1      |
|    3.750 |      0.022 |      0.033 |     6    |  # 8 |       1      |
|    5.000 |      0.022 |      0.033 |     6    |  # 8 |       1      |
-----------------------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |      Location       | Distance |  Anchor  |
|        Position | Nums | Size | Start  |    End     | From Face | Start End |
---------------------------------------------------------------------------
|   Evenly Dist. |     6 | # 8  |    0.00|     5.00 |     0.17  |  Yes  Yes |
---------------------------------------------------------------------------
      STAAD SPACE                                               -- PAGE NO.
6
TRANSVERSE BAR LAYOUT
-----------------------------------------------------------------------------
```

```
------
|                                 |        Asv        |      Rebar
Specification       |
| Zone | Dir.|  From    |    To    |  Reqd.  |   Prov.   | Nums | Size | Spacing
| Legs |
-------------------------------------------------------------------------------
------
|    1 |  Y  |    0.00 |    5.00 |     0.00 |     0.00 |     5 |  # 4 |    1.25
|   2  |
|    1 |  Z  |    0.00 |    5.00 |     0.00 |     0.00 |     5 |  # 4 |    1.25
|   2  |
-------------------------------------------------------------------------------
------
------------------------ Member:     1 Design Ends ------------------------
```

Related Links

- *D1.F.5 Column Design* (on page 1229)

V. ACI 318-14 Rectangular Column

Calculate the axial capacity of a rectangular column subjected to axial compressive force, given Ast

Details

A rectangular, cantilever column of length 5ft with an ultimate axial load of 1,538 kip has the following concrete and steel properties.

Compressive strength of concrete (normal weight concrete), $f_c = 5{,}000\ psi$
Yield strength of steel, $f_y = 60{,}000\ psi$
Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5\ in$
Width of column, $B = 20\ in$
Depth of column, $D = 30\ in$
Strength reduction factor, $\phi = 0.65$
Number of #9 bars = 10
Rebar diameter for #9 bar, $Ød = 1.128\ in$

Factored axial load (axial demand), $P_u = 1{,}538\ kip$

Validation

Gross area of concrete section, $A_g = B \times D = 600\ in^2$

Total area of steel reinforcement, $A_{st} = 10\ in^2$

Nominal axial strength at zero eccentricity:

$$P_0 = 0.85 \times f_c (A_g - A_{st}) + f_y \times A_{st} = 3.108 \times 10^3 \text{ kip}$$

For tied reinforced column:

$$P_{n(max)} = 0.8 \cdot P_0 = 2.486 \times 10^3 \text{ kip}$$

Capacity = $\phi P_{n(max)} = 1{,}615.9\ kip$

Ratio = $P_u / \phi P_{n(max)} = 0.952$

Results

Table 880: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Axial Capacity (kip)	1,615.9	1,615.9	none	
Ratio $P_u/\phi P_{n(max)}$	0.952	0.952	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-14 Rectangular Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 01-Jan-18
JOB CLIENT Bentley Systems Inc.
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 30 ZD 20
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET KIP
SUPPORTS
1 FIXED
UNIT INCHES KIP
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -1538
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 2014
FC 5 ALL
MXMT 9 ALL
MXMB 9 ALL
MXMS 9 ALL
```

```
MIMT 9 ALL
MIMB 9 ALL
MIMS 9 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0
                       **********************************************
Units: KIP   , INCHES (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length:   72.000 |
| Critical Ratio :   0.952        Criteria: Axial                            |
| Critical Clause:   10.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   20.00 | Depth:   30.00 |
-------------------------------------------------------
DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc           5.000 |                      | Ec     0.403E+04 |
| Steel    | Fy(main)    60.000 | Fy(trans)    60.000 | Es     0.290E+05 |
| Cover    | Top          1.500 | Bottom       1.500 | Sides      1.500 |
-------------------------------------------------------------------------
CRITICAL STRENGTH RESULTS
-----------------------------------------------------------------------
| Category |    Demand      |  Min Capacity  |  Max Capacity  |  Ratio  |
-----------------------------------------------------------------------
| Axial    |      -1538.000|      -1615.900 |        540.000 |   0.952 |
| Flexure  |          0.000|      -4780.603 |       4780.603 |   0.000 |
| Shear Y  |          0.000|          0.000 |          0.000 |   0.000 |
| Shear Z  |          0.000|          0.000 |          0.000 |   0.000 |
| Torsion  |          0.000|          0.000 |          0.000 |   0.000 |
-----------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-----------------------------------------------------------------------
| Distance |  Ast-reqd  |  Ast-prov  | No(s)bars| Size | No of Layers |
-----------------------------------------------------------------------
|    0.000 |      8.660 |     10.000 |    10    |  # 9 |       1      |
|   18.000 |      8.660 |     10.000 |    10    |  # 9 |       1      |
|   36.000 |      8.660 |     10.000 |    10    |  # 9 |       1      |
|   54.000 |      8.660 |     10.000 |    10    |  # 9 |       1      |
|   72.000 |      8.660 |     10.000 |    10    |  # 9 |       1      |
-----------------------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                |     Bars    |      Location       | Distance  |  Anchor  |
|       Position | Nums | Size | Start  |    End     | From Face | Start End |
---------------------------------------------------------------------------
|            Top |    3 | # 9  |     0.00|    72.00 |      2.06  |  Yes  Yes |
```

```
|          Bottom |     3 | # 9   |     0.00|     72.00 |        2.06  |  Yes  Yes |
|            Left |     2 | # 9   |     0.00|     72.00 |        2.06  |  Yes  Yes |
|           Right |     2 | # 9   |     0.00|     72.00 |        2.06  |  Yes  Yes |
      STAAD SPACE                                                    -- PAGE NO.
5
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------
------
|                                |        Asv         |      Rebar
Specification        |
| Zone | Dir.|  From    |     To    | Reqd.   |  Prov.   | Nums | Size | Spacing
| Legs |
------------------------------------------------------------------------------
------
|    1 |  Y  |    0.00 |   72.00 |    0.02 |    0.02 |     5 |  # 4 |  18.00
|   2  |
|    1 |  Z  |    0.00 |   72.00 |    0.02 |    0.02 |     5 |  # 4 |  18.00
|   2  |
------------------------------------------------------------------------------
------
------------------------ Member:     1 Design Ends ------------------------
```

Related Links

- *D1.F.5 Column Design* (on page 1229)

V. ACI 318-14 Square Column

Calculate the axial capacity of a square column subjected to an axial compressive force, given Ast

Details

A square, cantilever column of length 5ft with an ultimate axial load of 998 kip has the following concrete and steel properties.

Compressive strength of concrete (normal weight concrete), $f_c = 5{,}000\ psi$
Yield strength of steel, $f_y = 60{,}000\ psi$
Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5\ in$
Width of column, $B = 20\ in$
Depth of column, $D = 20\ in$
Strength reduction factor, $\phi = 0.65$
Number of #9 bars = 4
Rebar diameter for #9 bar, $Ød = 1.128\ in$

Factored axial load (axial demand), $P_u = 998\ kip$

Validation

Gross area of concrete section, $A_g = B \times D = 400\ in^2$

Total area of steel reinforcement, $A_{st} = 4\ in^2$

Nominal axial strength at zero eccentricity:

$$P_0 = 0.85 \times f_c(A_g - A_{st}) + f_y \times A_{st} = 1.923 \times 10^3 \text{ kip}$$

For tied reinforced column:

$$P_{n(max)} = 0.8 \cdot P_0 = 1.5384 \times 10^3 \text{ kip}$$

Capacity = $\phi P_{n(max)}$ = *999.96 kip*

Ratio = $P_u / \phi P_{n(max)}$ = *0.998*

Results

Table 881: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Axial Capacity (kip)	999.96	999.96	none	
Ratio $P_u / \phi P_{n(max)}$	0.998	0.998	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-14 Square Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 29-Aug-18
JOB CLIENT Bentley Systems Inc.
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 20 ZD 20
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET KIP
SUPPORTS
1 FIXED
UNIT INCHES KIP
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -998
```

```
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 2014
FC 5 ALL
MXMT 9 ALL
MXMB 9 ALL
MXMS 9 ALL
MIMT 9 ALL
MIMB 9 ALL
MIMS 9 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0
                       ***********************************************
Units: KIP   , INCHES (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Column          Length:   72.000 |
| Critical Ratio :   0.998        Criteria: Axial                            |
| Critical Clause:   10.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   20.00 | Depth:   20.00 |
-------------------------------------------------------
DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc           5.000 |                      | Ec     0.403E+04 |
| Steel    | Fy(main)    60.000 | Fy(trans)     60.000 | Es     0.290E+05 |
| Cover    | Top          1.500 | Bottom         1.500 | Sides      1.500 |
-------------------------------------------------------------------------
CRITICAL STRENGTH RESULTS
-----------------------------------------------------------------------
| Category |    Demand      |  Min Capacity  |  Max Capacity  |  Ratio  |
-----------------------------------------------------------------------
| Axial    |        -998.000|       -999.960 |        216.000 |   0.998 |
| Flexure  |           0.000|      -1740.458 |       1740.458 |   0.000 |
| Shear Y  |           0.000|          0.000 |          0.000 |   0.000 |
| Shear Z  |           0.000|          0.000 |          0.000 |   0.000 |
| Torsion  |           0.000|          0.000 |          0.000 |   0.000 |
-----------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-----------------------------------------------------------------------
| Distance |  Ast-reqd  |  Ast-prov  | No(s)bars| Size | No of Layers |
-----------------------------------------------------------------------
|    0.000 |      4.000 |      4.000 |     4    |  # 9 |       1      |
|   18.000 |      4.000 |      4.000 |     4    |  # 9 |       1      |
|   36.000 |      4.000 |      4.000 |     4    |  # 9 |       1      |
|   54.000 |      4.000 |      4.000 |     4    |  # 9 |       1      |
|   72.000 |      4.000 |      4.000 |     4    |  # 9 |       1      |
```

```
-----------------------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                |    Bars     |       Location         | Distance   |   Anchor  |
|       Position | Nums | Size | Start  |    End        | From Face  | Start End |
---------------------------------------------------------------------------
|            Top |    2 | # 9  |    0.00|    72.00 |       2.06  |  Yes  Yes |
|         Bottom |    2 | # 9  |    0.00|    72.00 |       2.06  |  Yes  Yes |
---------------------------------------------------------------------------
      STAAD SPACE                                                   -- PAGE NO.
5
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------
------
|                                |        Asv         |      Rebar
Specification       |
| Zone | Dir.|  From    |    To    | Reqd.  |   Prov.  | Nums | Size | Spacing
| Legs |
------------------------------------------------------------------------------
------
|    1 |  Y  |    0.00 |   72.00 |    0.02 |    0.02 |    5 |  # 4 |  18.00
|   2  |
|    1 |  Z  |    0.00 |   72.00 |    0.02 |    0.02 |    5 |  # 4 |  18.00
|   2  |
------------------------------------------------------------------------------
------
------------------------ Member:     1 Design Ends ------------------------
```

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

V. ACI 318-14 Tee Beam

Calculate the flexural capacity of a T-beam subjected to major axis moment, given Ast

References

1. ACI 318-14: Sections 19.2.1, 20.2.2, 22.3, and 22.4; Tables 22.2.2.4.3, 22.4.2.1, 27.3.2.1
2. PCA Notes 2011: Tables 6.1, 7.1

Related Links

- *D1.F.4 Beam Design* (on page 1215)

Details

A T-beam of length 5ft is subjected to a major axis moment has the following concrete and steel properties.

Width of T flange, $B_f = 30\ in$
Depth of T, $D = 26\ in$
Thickness of flange, $T_f = 4\ in$
Thickness of web, $T_w = 10\ in$
Steel modulus of elasticity, $E_s = 29{,}000\ ksi$
Ultimate strain, $\varepsilon_u = 0.003$
Yield strength of reinforcing steel, $f_y = 60\ ksi$

Compressive strength of concrete (normal weight concrete), $f_c = 4{,}000\ psi$
Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5\ in$
Strength reduction factor, $\phi = 0.9$
Rebar diameter for #10 bar, $Ød = 1.27\ in$

Validation

$$d = D - C_c - \frac{Ød}{2} = 23.865$$

$$d' = C_c + \frac{Ød}{2} = 2.135$$

$$R_n = \frac{6,\ 400}{\phi \times 0.85 \times f_c \times B_f \times d^2} = 0.122$$

From Table 7.1 of PCA (ref. 2):

$$w = \frac{(0.1726 - 0.1648)}{(0.19 - 0.18)} \times (R_n - 0.1648) + 0.18 = 0.147$$

$a = 1.18 \times w \times d = 4.138 > T_f$, hence beam acts as a T

Compression strength of the flange:

$$C_f = 0.85 \times f_c \times (B_f + T_w) \times T_f = 136$$

$$A_{sf} = C_f/f_y = 2.267\ in^2$$

this is the required area to equilibrate C_f.

Flexural strength of the flange:

$$M_{nw} = \frac{6,\ 400}{\phi} - M_{nf} = 4.001 \times 10^{-3} in \cdot k$$

To check whether this section is tension controlled, refer to PCA Table 7.1 (ref. 2):

$$R_n = \frac{M_{nw}}{f_c \times T_w \times d^2} = 0.176$$

$$w = \frac{(0.1802 - 0.1726)}{(0.2 - 0.19)} \times (R_n - 0.1726) + 0.19 = 0.192$$

$$\rho = w_n \frac{f_c}{f_y} = 0.013 < 0.01806$$

based on PCA Table 6.1 (ref. 2), thus the section is tension controlled.

$$A_{sw} = \rho \times T_w \times d = 3.06\ in^2$$

Assume 6- #10 bars, divided between top and bottom: $A_{st,\ tot} = 6 \times 1.27\ in^2 = 7.62\ in^2$

$$a_w = \left(A_{st,tot} - A_{sf}\right) \frac{f_y}{0.85 \times f_c \times T_w} = 9.447 in$$

$$M_n = (A_{st,tot} - A_{sf}) \times f_y \left(d - \frac{a_w}{2}\right) + A_{sf} \times f_y \times \left(d - \frac{T_f}{2}\right) = 9,\ 258 in \cdot k$$

$$\phi M_n = 8{,}332\ in \cdot k = 694.4\ ft \cdot k$$

Results

Table 882: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Flexural capacity (ft·k)	694.4	698.938	<1%	OK

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-14 Tee Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 23-Feb-18
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 4 0; 2 5 4 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 2.16667 ZD 2.5 YB 1.83333 ZB 0.833333
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 533.33333
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI
MAXMAIN 10 ALL
MINMAIN 9 ALL
MXMS 5 ALL
MIMS 4 ALL
TRACK 2 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0
                       ***********************************************
Units: KIP   , FEET   (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length:    5.000 |
| Critical Ratio :   0.763        Criteria: Flexure                         |
| Critical Clause:   9.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
------------------------------------------------------------------------------
| Shape: Tee-Shape   | YD:    2.17 | ZD:    2.50 | ZB:    0.83 | YB:    1.83 |
------------------------------------------------------------------------------
DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc         576.000 |                     | Ec     0.519E+06 |
| Steel    | Fy(main)  8639.999 | Fy(trans)  8639.999 | Es     0.418E+07 |
| Cover    | Top          0.125 | Bottom        0.125 | Sides      0.125 |
-------------------------------------------------------------------------
CRITICAL STRENGTH RESULTS
-----------------------------------------------------------------------
| Category |    Demand     |  Min Capacity  |  Max Capacity  |  Ratio  |
-----------------------------------------------------------------------
| Axial    |          0.000|       -825.395 |        411.480 |   0.000 |
| Flexure  |        533.333|        -23.984 |        698.938 |   0.763 |
| Shear Y  |          0.000|        -17.485 |         17.485 |   0.000 |
| Shear Z  |          0.000|         -8.467 |          8.467 |   0.000 |
| Torsion  |          0.000|          0.000 |          0.000 |   0.000 |
-----------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
------------------------------------------------------------------------------
---
| Distance | Position |  Ast-reqd  |  Ast-prov  | No(s)bars| Size | No of
Layers |
------------------------------------------------------------------------------
---
|    0.000 |  Bottom  |      0.039 |      0.053 |     6    |  #10 |
2      |
|    1.250 |  Bottom  |      0.039 |      0.053 |     6    |  #10 |
2      |
|    2.500 |  Bottom  |      0.039 |      0.053 |     6    |  #10 |
2      |
|    3.750 |  Bottom  |      0.039 |      0.053 |     6    |  #10 |
2      |
|    5.000 |  Bottom  |      0.039 |      0.053 |     6    |  #10 |
2      |
------------------------------------------------------------------------------
---
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |   Bars      |       Location      | Distance |  Anchor  |
|        Position | Nums | Size | Start  |    End     | From Face | Start End |
---------------------------------------------------------------------------
|          Bottom |    3 | #10  |    0.00|      5.00 |     0.18  |  Yes  Yes |
```

```
|        Bottom |    3 | #10  |     0.00|     5.00 |     0.37  |  Yes  Yes |
---------------------------------------------------------------------------
      STAAD SPACE                                                -- PAGE NO.
5
TRANSVERSE BAR LAYOUT
----------------------------------------------------------------------------
|                            | Asv Density  |     Rebar Specification      |
| Zone | Dir.|  From |  To   | Reqd. | Prov. | Nums | Size | Spacing | Legs |
----------------------------------------------------------------------------
    Minimum transverse reinforcement is not required as per Cl.9.6.3.1
------------------------ Member:     1 Design Ends ------------------------
```

V. ACI 318-14 Rectangular Beam without Torsion

Design of a doubly reinforced rectangular beam.

Reference

1. ACI 318-14
2. PCA Notes 2011 Example 6.2

Related Links

- *D1.F.4 Beam Design* (on page 1215)

Details

To determine the Required Area of Tension and Compression Reinforcement for given Concrete and Steel Properties with Given Factored Moment.

Design of Rectangular Beam with Compression Reinforcement-Doubly Reinforced

A beam cross-section is limited to size 14in X 23 in . Determine the required area of reinforcement for a factored moment Mu= 516 ft-kips. fc= 4000psi, fy=60,000 psi.,

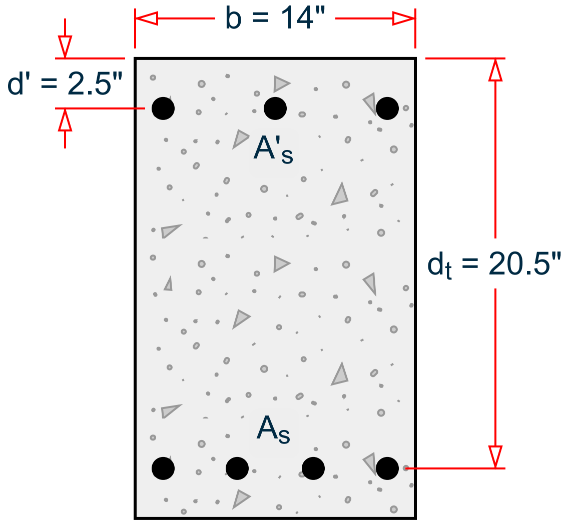
Design Properties, available:

Validation

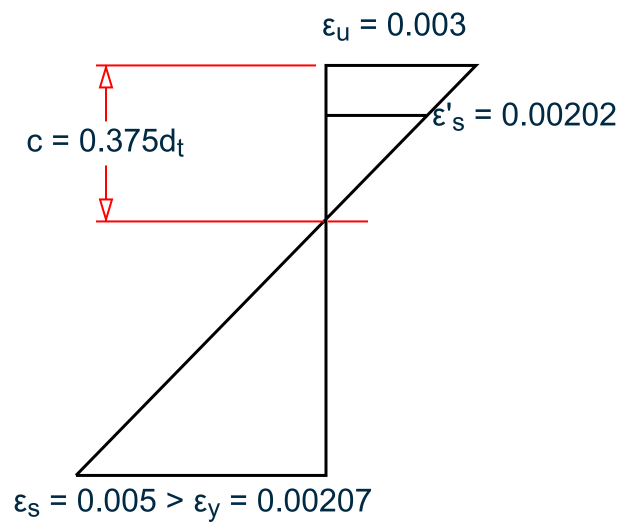
d_{cover} = 2.5 in

Step 1: Check if compression reinforcement is required, using $Z = 0.9$.

See table 6-1 of PCA Notes (ref. 2).



Section



Stress Block

b = 14 in

d = 20.5 in

Note: Assume d = dt

$M_n = M_u/Z = 573.33$ ft-kip

$R_n = M_n / (b*d^2) = 1.169(10)^3$ psi

$R_{nt} = 911$ psi

Note: R_{nt} = *Maximum Rn for tension controlled section for that particular grade for concrete.*

$R_n > R_{nt}$

Hence, both compression and tension reinforcement is required.

<u>Step 2: Determine maximum moment with compression reinforcement,</u>

<u>M_{nt}</u>

This condition corresponds to tension controlled limit ($Z = 0.9$).

Depth of neutral axis, $c = 0.375dt = 0.375d = 7.687\ in$

$\beta_1 = 0.85$ (from table 6.1 (ref. 2)

$a = \beta_1 * c = 6.534\ in$

Compressive force, $C = 0.85 f_c \cdot a \cdot b = 311.036\ kips$

Tension force, $T = C = 311.036\ kips$

Nominal moment strength resisted by the concrete section, without compression reinforcement, $M_{nt} = T(d - a/2) = 446.669\ ft\text{-}kips$

Nominal moment strength resisted by the concrete section, with compression reinforcement, $M_{nc} = M_n - M_{nt} = 126.664\ ft\text{-}kips$

Step 3: Determine area of tension steel

Area of tension steel required, $A_{s,nt}$, to develop M_{nt}

$$A_{s,nt} = T / f_y = 5.184 \text{ in}^2$$

Step 4: Determine additional area of tension steel required

Area of tension steel required, $A_{s,additional}$, required to counteract additional moment due to T-C couple.

$$A_{s,additional} = (M_n - M_{nt}) / [f_y \cdot (d - d_{cover})] = 1.407 \text{ in}^2$$

Total tension steel required:

$$A_s = A_s + A_{s,nt} = 6.591 \text{ in}^2$$

Comparison

Table 883: Comparison of results

Parameter	STAAD.Pro	Reference	Difference	Comments
Area Required (compression) (in^2)	1.441	1.61	11.0	See note (*) below.
Area Required (tension) (in^2)	6.591	6.355	3.6%	See note (*) below.

Note: (*) The required steel calculation above assumes a single layer of bottom bars and the effective depth = 0.375*dt based on the single bar layer. The reference (ref. 2) calculations only considers a strength design and does not check for the minimum spacing requirements. STAAD.Pro however considers two layers of bars in the design to account for the minimum criteria and hence uses a neutral axis position based on the two bar layers. Hence the minor difference in the steel areas calculated.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-14 Rectangular Beam Without Torsion.std` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
***********************************************************************
* This example is a replica of example no 6.2 of PCA Notes 318-11.
***********************************************************************
ENGINEER DATE 29-Aug-18
JOB NAME PCA Example 6.2
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 8 0 0;
MEMBER INCIDENCES
1 1 2;
```

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.91667 ZD 1.16667
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 MZ 464.417
PERFORM ANALYSIS
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
CODE ACI
UNIT INCHES KIP
MAXMAIN 8 ALL
MINMAIN 8 ALL
*MIMB 6 ALL
*MXMB 6 ALL
CLT 2 ALL
CLB 2 ALL
TRACK 2 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)
v2.0
                       ***********************************************
Units: KIP   , INCHES (Unless Noted Otherwise)
 Member :      1
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam            Length:   96.000 |
| Critical Ratio :   0.910        Criteria: Flexure                          |
| Critical Clause:   9.5.2                                                   |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:   14.00 | Depth:   23.00 |
-------------------------------------------------------
DESIGN INPUTS
-------------------------------------------------------------------------
| Concrete | Fc           4.000 |                     | Ec     0.360E+04 |
| Steel    | Fy(main)    60.000 | Fy(trans)    60.000 | Es     0.290E+05 |
| Cover    | Top          2.000 | Bottom        2.000 | Sides      1.500 |
```

```
--------------------------------------------------------------------------
CRITICAL STRENGTH RESULTS
------------------------------------------------------------------------
| Category |    Demand     |  Min Capacity  |  Max Capacity  |  Ratio  |
------------------------------------------------------------------------
| Axial    |          0.000|       -871.565 |        554.580 |  0.000 |
| Flexure  |       5573.004|      -2586.136 |       6122.188 |  0.910 |
| Shear Y  |          0.000|        -21.019 |         21.019 |  0.000 |
| Shear Z  |          0.000|        -22.720 |         22.720 |  0.000 |
| Torsion  |          0.000|          0.000 |          0.000 |  0.000 |
------------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-------------------------------------------------------------------------------
---
| Distance | Position |  Ast-reqd   |  Ast-prov  | No(s)bars| Size | No of
Layers |
-------------------------------------------------------------------------------
---
|    0.000 |     Top  |      1.610 |      2.370 |     3    |  # 8 |
1       |
|          |  Bottom  |      6.355 |      7.900 |    10    |  # 8 |
2       |
|   24.000 |     Top  |      1.610 |      2.370 |     3    |  # 8 |
1       |
|          |  Bottom  |      6.355 |      7.900 |    10    |  # 8 |
2       |
|   48.000 |     Top  |      1.610 |      2.370 |     3    |  # 8 |
1       |
|          |  Bottom  |      6.355 |      7.900 |    10    |  # 8 |
2       |
|   72.000 |     Top  |      1.610 |      2.370 |     3    |  # 8 |
1       |
|          |  Bottom  |      6.355 |      7.900 |    10    |  # 8 |
2       |
|   96.000 |     Top  |      1.610 |      2.370 |     3    |  # 8 |
1       |
|          |  Bottom  |      6.355 |      7.900 |    10    |  # 8 |
2       |
-------------------------------------------------------------------------------
---
      STAAD SPACE                                                -- PAGE NO.
6
LONGITUDINAL BAR LAYOUT
----------------------------------------------------------------------------
|                 |    Bars     |       Location      | Distance  |  Anchor  |
|       Position  | Nums | Size | Start  |    End     | From Face | Start End |
----------------------------------------------------------------------------
|            Top  |    3 | # 8  |     0.00|    96.00 |      2.50  |  Yes  Yes |
|         Bottom  |    5 | # 8  |     0.00|    96.00 |      2.50  |  Yes  Yes |
|         Bottom  |    5 | # 8  |     0.00|    96.00 |      4.50  |  Yes  Yes |
----------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
-------------------------------------------------------------------------------
------
|                                 |        Asv        |      Rebar
Specification       |
| Zone | Dir.|  From    |    To    |  Reqd.  |  Prov.   | Nums | Size | Spacing
| Legs |
```

```
--------------------------------------------------------------------------------
------
|    1 |  Y  |    0.00 |   96.00 |     0.03 |     0.03 |     8 |  # 4 |  13.71
|   2  |
|    1 |  Z  |    0.00 |   96.00 |     0.03 |     0.03 |     8 |  # 4 |  13.71
|   2  |
--------------------------------------------------------------------------------
------
------------------------ Member:     1 Design Ends ------------------------
```

V. ACI 318-11 Circular Column

Calculate the axial capacity of a circular column subjected to axial compressive force, given Ast

References

ACI 318-11

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

Details

A circular, cantilever column of length 5ft with an ultimate axial load of 500 kip has the following concrete and steel properties.

Compressive strength of concrete (normal weight concrete), $f_c = 4{,}000\ psi$
Yield strength of steel, $f_y = 60{,}000\ psi$
Clear spacing for outermost bottom reinforcement to bottom face, $C_c = 1.5\ in$
Diameter of column, $D = 20\ in$
Strength reduction factor, $\phi = 0.65$
Number of #8 bars, $N = 6$
Rebar diameter for #8 bar, $d = 1\ in$

Factored axial load (axial demand), $P_u = 500\ kip$

Validation

Gross area of concrete section, $A_g = \pi D^2/4 = 314.159\ in^2$

Total area of steel reinforcement, $A_{st} = \left(\pi \frac{d^2}{4}\right) N = 4.712\ in^2$

Nominal axial strength at zero eccentricity:

$$P_0 = 0.85 \times f_c (A_g - A_{st}) + f_y \times A_{st} = 1.335 \times 10^3\ \text{kip}$$

For tied reinforced column:

$$P_{n(max)} = 0.8 \cdot P_0 = 1.068 \times 10^3\ \text{kip}$$

Capacity = $\phi P_{n(max)} = 694.129\ kip$

Ratio = $P_u/\phi P_{n(max)} = 0.72$

Results

Table 884: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
P_0 (kip)	1,335	1,336.43	negligible	
P_n (kip)	1,068	1,069.14	negligible	
Bar configuration	6- #8 bars	6- #8 bars	none	
Column size	20" dia. tied column	20" dia. tied column	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\10 Concrete Design\US\ACI 318-11 Circular Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 15-May-2020
ENGINEER NAME Design of Circular Column
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 5 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.66667
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE Dead  TITLE LOAD CASE 1
JOINT LOAD
2 FY -500
PERFORM ANALYSIS
SECTION 0 0.25 0.5 0.75 1 ALL
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
```

```
CODE ACI 2011
MAXMAIN 8 ALL
MINMAIN 8 ALL
TRACK 2 ALL
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
        ACI 318-11   COLUMN  NO.     1   DESIGN RESULTS
        ===============================================
   FY - 60000  FC - 4000 PSI,  CIRC SIZE  20.00 INCHES DIAMETER  TIED
           ONLY MINIMUM STEEL IS REQUIRED.
           AREA OF STEEL REQUIRED =   3.142  SQ. IN.
   BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
   ----------------------------------------------------------
     6 - NUMBER  8           1.509        1      END     0.650
   (EQUALLY SPACED)
   TIE BAR NUMBER    4 SPACING  16.00 IN
       COLUMN INTERACTION: MOMENT ABOUT Z/Y -AXIS (KIP-FT)
    --------------------------------------------------------
      P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
    1336.43   1069.14    475.51    247.89    6.26
       M0       P-tens.   Des.Pn    Des.Mn     e/h
     153.92   -284.40    769.23      0.00    0.00000
    --------------------------------------------------------
                           Pn        Mn        Pn        Mn
            |             986.90   144.54   493.45   246.76
         P0 |*            904.66   176.00   411.21   248.11
            | *           822.42   201.19   328.97   246.46
      Pn,max|__*          740.18   220.34   246.73   239.25
            |   *         657.93   233.38   164.48   222.95
      Pn    |    *        575.69   242.15    82.24   190.02
     NOMINAL|     *
       AXIAL|      *
 COMPRESSION|       *
          Pb|-------*Mb
            |      *
 ___________|____*_______
            |  * M0   Mn,
            | *   BENDING
      P-tens|*     MOMENT
            |
   ********************END OF COLUMN DESIGN RESULTS********************
```

V. ACI 318-11 Square Column

Calculate the axial capacity of a square column subjected to an axial compressive force, given Ast

Details

A square, cantilever column of length 5ft with an ultimate axial load of 998 kip has the following concrete and steel properties.

Compressive strength of concrete (normal weight concrete), f_c = *5,000 psi*
Yield strength of steel, f_y = *60,000 psi*
Clear spacing for outermost bottom reinforcement to bottom face, C_c = *1.5 in*
Width of column, B = *20 in*
Depth of column, D = *20 in*
Strength reduction factor, ϕ = *0.65*
Number of #9 bars = 4
Rebar diameter for #9 bar, $Ød$ = *1.128 in*

Factored axial load (axial demand), P_u = *998 kip*

Validation

Gross area of concrete section, $A_g = B \times D = 400\ in^2$

Total area of steel reinforcement, $A_{st} = 4\ in^2$

Nominal axial strength at zero eccentricity:

$$P_0 = 0.85 \times f_c(A_g - A_{st}) + f_y \times A_{st} = 1.923 \times 10^3 \text{ kip}$$

For tied reinforced column:

$$P_{n(max)} = 0.8 \cdot P_0 = 1.5384 \times 10^3 \text{ kip}$$

Capacity = $\phi P_{n(max)} = 999.96\ kip$

Ratio = $P_u / \phi P_{n(max)} = 0.998$

Results

Table 885: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
P_0 (kip)	1,923	1,923	none	
P_n (kip)	1,583	1,583.4	negligible	
Bar configuration	4- #9 bars	4- #9 bars	none	
Column size	20"×20" tied column	20"×20" tied column	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-11 Square Column.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
**********************************************************************
* This example is a replica of example no 6.3 of PCA Notes 318-11.
**********************************************************************
ENGINEER DATE 21-June-2020
```

```
JOB NAME PCA Example 6.3
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 6 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 20 ZD 20
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET KIP
SUPPORTS
1 FIXED
UNIT INCHES KIP
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -998
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 2011
FC 5 ALL
MXMT 9 ALL
MXMB 9 ALL
MXMS 9 ALL
MIMT 9 ALL
MIMB 9 ALL
MIMS 9 ALL
TRACK 2 ALL
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
        ACI 318-11   COLUMN  NO.     1   DESIGN RESULTS
        ===============================================
   FY - 60000  FC - 5000 PSI,  SQRE SIZE - 20.00 X 20.00 INCHES, TIED
           ONLY MINIMUM STEEL IS REQUIRED.
           AREA OF STEEL REQUIRED =   4.000  SQ. IN.
   BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI-Z  PHI-Y
   --------------------------------------------------------------------
     4 - NUMBER  9           1.000        1      END     0.000  0.000
```

```
   (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
   TIE BAR NUMBER    9 SPACING  18.05 IN
       COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)
    --------------------------------------------------------
      P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
    1923.00   1538.40    716.26    446.51    7.48
       M0        P-tens.   Des.Pn    Des.Mn     e/h
     156.73   -240.00  Infinity       NaN    0.00000
    --------------------------------------------------------
       COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)
    --------------------------------------------------------
      P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
    1923.00   1538.40    716.26    446.51    7.48
       M0        P-tens.   Des.Pn    Des.Mn     e/h
     156.73   -240.00  Infinity       NaN    0.00000
    --------------------------------------------------------
                              Pn        Mn        Pn        Mn     (@ Z )
              |             1420.06   279.88   710.03   446.27
          P0  |*            1301.72   332.82   591.69   436.82
              | *           1183.38   372.72   473.35   420.15
      Pn,max  |__*          1065.05   403.75   355.02   373.59
              |   *          946.71   425.39   236.68   309.55
      Pn      |    *         828.37   438.88   118.34   236.06
     NOMINAL  |     *         Pn        Mn        Pn        Mn     (@ Y )
       AXIAL  |      *      1420.06   279.88   710.03   446.27
 COMPRESSION  |       *     1301.72   332.82   591.69   436.82
          Pb  |-------*Mb   1183.38   372.72   473.35   420.15
              |      *      1065.05   403.75   355.02   373.59
 _____________|____*_______  946.71   425.39   236.68   309.55
              |  * M0   Mn,  828.37   438.88   118.34   236.06
              | *   BENDING
      P-tens  |*     MOMENT
              |
   *******************END OF COLUMN DESIGN RESULTS*******************
```

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

V. ACI 318-11 Rectangular Singly Reinforced Beam

Design of a singly reinforced concrete beam per the ACI 318-2011 code.

References

1. ACI 318-11
2. PCA Notes 2011: Example 7.1

Related Links

- *D1.F.4.6 Beam Design per ACI 318-11 and Earlier* (on page 1224)

Details

Determine the Required Area of Tension Reinforcement for given Concrete and Steel Properties with Given Factored Moment.

Select a rectangular beam size and required reinforcement to carry service load moments M_{DL} = *56 ft-kips* and M_{LL}= *35 ft-kips*. Select reinforcement to control flexural cracking.

A 10' simply supported beam is subjected to an ultimate uniform load of 9.736 k/ft.

f'_c = 4,000 psi
f_y = 60,000 psi
cover = 2 in

Validation

This is a complete design procedure for rectangular sections with tension reinforcement only, a minimum beam depth will be computed using the maximum reinforcement permitted for tension-controlled flexural members, ρ_t.

d_{cover} = dist. from bottom of beam up to center of bottom bar = C_c + 1/2(bar diameter) = 2.5 in = 63.5 mm

Determine the maximum tension-controlled reinforcement ratio fro material strengths from Table 6-1 of PCA Notes on ACI 318-11.

$$\rho_{t.} = 0.01806$$

Compute bd^2 required:

$$M_u = 1.2 \cdot M_{DL} + 1.6 \cdot M_{LL} = 123 \text{ ft} \cdot \text{kips}$$

$$R_n = \rho_t f_y \left(1 - 0.5\rho_t \frac{f_y}{0.85 f'_c}\right) = 910.9 \text{ psi}$$

$$bd^2{}_{req'd} = \frac{M_u}{\phi R_n} = 1,\ 803 \text{in}^3$$

Size members so that $bd^2_{(prov)} \geq bd^2_{(req'd)}$.

b = 10 in (column width)

$$d = \sqrt{\frac{1,\ 803}{10}} = 13.43 \text{in}$$

Total beam depth, $D > d + d_{cover} = 15.93\ in$; use a $D = 16\ in$; $d = 13.5\ in$.

$$\rho_{prov} = \frac{0.85 f'_c}{f_y}\left(1 - \sqrt{1 - \frac{2R_{n,prov}}{085 f'_c}}\right) = 0.018$$

$$A_s = \rho_{\text{prov}} \cdot b \cdot d_{\text{prov}} = 2.406 \text{ in}^2$$

<u>Verify the design.</u>

$$\underline{T = A_s f_y = 2.406 \times 60 = 144.36 \text{ kips}}$$

Design moment strength, $\phi M_n = D_m$
Strength reduction factor, $\phi = 0.9$

$$D_m = \phi\left[A_s f_y \left(d_{\text{prov}} - \frac{a}{2}\right)\right] = 123.2 \text{ ft} \cdot \text{kip}$$

$D_m > M_u$, hence design OK.

Compute minimum reinforcement

$$A_{s,\min} = \max \begin{vmatrix} \frac{200}{f_y} bd = 0.45 \text{ in}^2 \\ \frac{3\sqrt{f'_c}}{f_y} bd = 0.427 \text{ in}^2 \end{vmatrix}$$

Since $A_s > A_{s,min}$, the design satisfies the minimum reinforcement criteria.

Compute the reinforcement provided

Use 2-#10 bars: $A_s = 2.53\ in^2$

Check Bar Spacing

End bar diameter, □ = $1.27\ in$, two bars in a layer

$$f_s = \frac{2}{3} f_y = 40(10)^3 \text{ psi}$$

Maximum spacing allowed:

$$s_{\max} = \min \begin{vmatrix} 15\left(\frac{40{,}000}{f_s}\right) - 2.5C_c \\ 12\left(\frac{40{,}000}{f_s}\right) \end{vmatrix} = 10 \text{ in}$$

Spacing provided:

$$s_{prov} = \frac{b - \left(2C_c + 2 \times 0.5\varnothing\right)}{N - 1} = 4.73 \text{ in}$$

Results

Table 886: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Area of steel required (in^2)	2.406	2.44	negligible	
Maximum bar spacing (in)	10	10	none	
Bar spacing (in)	4.73	4.73	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI-318-11 Singly Reinforced Rectangular Beam.std` is typically installed with the program.

```
STAAD PLANE RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 2011
START JOB INFORMATION
ENGINEER DATE 18-December-19
JOB NAME PCA Example 7.1
JOB CLIENT Bentley Systems Inc.
```

```
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 16 ZD 10
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -4.48
LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI GY -2.8
LOAD COMB 3 ACI 318R-02
1 1.2 2 1.6
PERFORM ANALYSIS
UNIT INCHES KIP
START CONCRETE DESIGN
CODE ACI 2011
*MINMAIN 8 ALL
*MAXMAIN 9 ALL
TRACK 2 ALL
DESIGN BEAM 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
          ACI 318-11   BEAM  NO.     1   DESIGN RESULTS
          ============================================
   LEN -  10.00FT.  FY - 60000.  FC - 4000.  SIZE - 10.00 X 16.00 INCHES
   LEVEL    HEIGHT     BAR INFO       FROM            TO          ANCHOR
          FT.    IN.              FT.     IN.     FT.     IN.   STA  END
   _____________________________________________________________________
     1    0 + 2-3/4    2-NUM.10     0 + 0-0/0     10 + 0-0/0    YES  YES
    |----------------------------------------------------------------|
    |   CRITICAL POS MOMENT=    123.20 KIP-FT  AT  5.00 FT, LOAD    3|
    |   REQD STEEL=  2.44 IN2, RHO=0.0183, RHOMX=0.0214 RHOMN=0.0033 |
    |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 2.54/ 4.73 INCH           |
    |   REQD. DEVELOPMENT LENGTH = 48.52 INCH                        |
    |----------------------------------------------------------------|
     Cracked Moment of Inertia Iz at above location =    1817.34 inch^4
   REQUIRED REINF. STEEL SUMMARY :
```

```
   -------------------------------
   SECTION     REINF STEEL(+VE/-VE)     MOMENTS(+VE/-VE)    LOAD(+VE/-VE)
   (FEET)         (SQ. INCH)                (KIP-FEET)
     0.00         0.000/    0.000          0.00/      0.00        3/    1
     0.83         0.650/    0.000         37.64/      0.00        3/    0
     1.67         1.231/    0.000         68.44/      0.00        3/    0
     2.50         1.723/    0.000         92.40/      0.00        3/    0
     3.33         2.101/    0.000        109.51/      0.00        3/    0
     4.17         2.341/    0.000        119.78/      0.00        3/    0
     5.00         2.423/    0.000        123.20/      0.00        3/    0
     5.83         2.340/    0.000        119.78/      0.00        3/    0
     6.67         2.101/    0.000        109.51/      0.00        3/    0
     7.50         1.723/    0.000         92.40/      0.00        3/    0
     8.33         1.231/    0.000         68.44/      0.00        3/    0
     9.17         0.650/    0.000         37.64/      0.00        3/    0
    10.00         0.000/    0.000          0.00/      0.00        3/    1
          B E A M  N O.     1 D E S I G N  R E S U L T S - SHEAR
    AT START SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     3
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
      RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 2011        -- PAGE NO.
4
    AT END   SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     3
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
  ___    1J____________________  120.X  10.X  16_____________________   2J____
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |  8#4 C/C  7                                                 8#4 C/C  7     |
 | 2#10H 2.63   0.TO  120.                                                    |
 ||========================================================================|  |
 |                                                                            |
 |____________________________________________________________________________|

  _________    _________    _________    _________    _________     _________
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |   | 2#10    |
 |   OO    |  |   OO    |  |   OO    |  |   OO    |  |   OO    |   |   OO    |
 |         |  |         |  |         |  |         |  |         |   |         |
 |_________|  |_________|  |_________|  |_________|  |_________|   |_________|
   *******************END OF BEAM DESIGN**************************
```

V. ACI 318-08 Rectangular Singly Reinforced Beam

Flexural and shear reinforcement design of a singly reinforced concrete beam per the ACI 318-2008 code.

Details

A 10' simply supported beam is subjected to an ultimate uniform load of 9.736 k/ft.

f'_c = 4,000 psi
f_y = 60,000 psi
f_{yt} = 60,000 psi
cover = 2.5 in
D = 16 in
b_w = 10 in

Validation

$$Ag = D \times b_w = 16 \times 10 = 160 \text{ in}^2$$

$$d = D - cover = 16 - 2.5 = 13.5 \text{ in}$$

Bending Reinforcement $\phi = 0.9$

Minimum reinforcement of flexural members [*Cl 10.5*]

$$A_{s,\min} = \max \begin{vmatrix} \frac{200}{f_y} b_w d = 0.45 \text{ in}^2 \\ \\ \frac{\sqrt{f'_c}}{f_y} b_w d = 0.427 \text{ in}^2 \end{vmatrix}$$

$$\beta_1 = 0.85 - 0.05\left(\frac{f'_c - 4,000}{1,000}\right) = 0.85 \qquad \text{[Cl 10.2.7]}$$

$$C_{\max} = \frac{0.003}{0.003 + 0.005} d = 5.063 \text{ in}$$

$$a_{\max} = \beta_1 C_{\max} = 4.303 \text{ in}$$

The ultimate moment:

$$M_u = \frac{w_u l^2}{8} = \frac{9.736(12)^2}{8} = 121.7 \text{ft} \cdot \text{k}$$

The depth of the compression block is:

$$a = d - \sqrt{d^2 - 2\frac{M_u}{0.85 f'_c \varphi b_w}} = 4.183 \text{ in}$$

Hence, $a < a_{max}$.

The area of the tensile steel reinforcement required is:

$$A_{s,req'd} = \frac{M_u}{\varphi f_y (d - a/2)} = 2.371 \text{ in}^2$$

Use 2-#10 bars: $A_s = 2.53 \ in^2$

Shear Reinforcement $\phi = 0.75$

Limit of concrete strength: $\sqrt{f'_c} = 63.2 < 100$, hence OK. *[Cl 11.1.2]*

$$\rho_w = \frac{A_s}{b_w d} = 0.018$$

$\lambda = 1$

Shear and moment at a distance, d, from the support:

$$V_{u1} = w\left(\frac{l}{2} - d\right) = 9.736\left(\frac{10}{2} - 1.125\right) = 37.73 \text{ kip}$$

$$M_{u1} = \frac{wd}{2}(l - d) = \frac{9.736 \times 1.125}{2}(10 - 1.125) = 48.6 \text{ ft·k} = 583.2 \text{ in·k}$$

Per Cl 11.2.2.1 equation 11.5, the unfactored shear capacity of concrete is:

$$V_c = \left(1.9\lambda\sqrt{f'_c} + 2,\ 500\rho_w V_{u1}\frac{d}{M_{u1}}\right)b_w d = 21.48 \text{ kip}$$

$$\phi V_c = 0.75 \times 21.48 = 16.11 \text{ kip}$$

The maximum shear capacity of the reinforcement is:

$$\varphi V_s = 8\sqrt{f'_c}b_w d = 0.75 \times 68.31 = 51.23 \text{ kip}$$ [Cl 11.4.7.9]

The minimum shear reinforcement , Av/s, for any load combination is:

[Cl 11.4.6.3 and R11.4.7]

$$A_{vs,\min} = \max\left|\begin{array}{l} 50\frac{b_w}{f_y} = 0.008 \text{ in}^2\text{/in} \\ \\ \frac{b_w}{f_y}0.75\sqrt{f'_c} = 0.0003 \text{ in}^2\text{/in} \end{array}\right.$$

$$A_{vs,req'd} = \frac{V_{u1} - \varphi V_c}{\varphi f_{yt} d} = 0.036 \text{ in}^2\text{/in}$$

Use 2-legged #4 stirrups ($A_v = 0.39\ in^2$ per stirrup).

$$s_{max} = d/2 = 13.5 /2 = 6.75 \text{ in}$$

Use maximum spacing of 6.75 in: $A_{vs}/s = 0.39\ /\ 6 = 0.058\ in^2/in$

$$V_s = \frac{V_{u1} - \varphi V_c}{\varphi} = 28.83 \text{ kip}$$

Results

Table 887: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Design moment, M_u (ft·k)	121.7	121.7	none	
Tension reinforcement required, $A_{s,re'd}$ (in^2)	2.371	2.41	1.6%	OK
Design shear, V_{u1} (kip)	37.73	37.73	none	

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Shear reinforcement, A_v/s (in^2/in)	0.058	0.058*	none	
V_c (kip)	21.48	21.34	negligible	
V_s (kip)	28.83	28.97	negligible	

*Shear reinforcement provided by STAAD: $A_{vs}/s = 0.39\ /\ 6.7 = 0.058\ in^2/in$

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-08 Rectangular Singly Reinforced Beam.STD` is typically installed with the program.

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 06-May-19
JOB CLIENT Bentley Systems Inc.
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
1 PRIS YD 1.333 ZD 0.8333
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 PINNED
2 FIXED BUT FX MZ
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -9.736
PERFORM ANALYSIS
PRINT MEMBER PROPERTIES
PRINT SUPPORT REACTION ALL
START CONCRETE DESIGN
CODE ACI 2008
UNIT INCHES KIP
TRACK 2 ALL
```

```
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
   ======================================================================
          BEAM  NO.     1 DESIGN RESULTS - FLEXURE PER CODE ACI 318-08
   LEN -  10.00FT.  FY - 60000.  FC - 4000.  SIZE - 10.00 X 16.00 INCHES
   LEVEL    HEIGHT     BAR INFO       FROM           TO          ANCHOR
          FT.    IN.               FT.     IN.     FT.     IN.   STA  END
   _______________________________________________________________________
     1    0 + 2-3/4    2-NUM.10     0 + 0-0/0     10 + 0-0/0    YES  YES
    |----------------------------------------------------------------|
    |   CRITICAL POS MOMENT=    121.70 KIP-FT  AT  5.00 FT, LOAD    1|
    |   REQD STEEL=  2.41 IN2, RHO=0.0180, RHOMX=0.0214 RHOMN=0.0033 |
    |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 2.54/ 4.73 INCH           |
    |   REQD. DEVELOPMENT LENGTH = 48.53 INCH                        |
    |----------------------------------------------------------------|
     Cracked Moment of Inertia Iz at above location =    1816.05 inch^4
   REQUIRED REINF. STEEL SUMMARY :
   -------------------------------
   SECTION     REINF STEEL(+VE/-VE)     MOMENTS(+VE/-VE)    LOAD(+VE/-VE)
   (FEET)         (SQ. INCH)               (KIP-FEET)
     0.00        0.000/    0.000         0.00/      0.00        1/    0
     0.83        0.642/    0.000        37.19/      0.00        1/    0
     1.67        1.215/    0.000        67.61/      0.00        1/    0
     2.50        1.700/    0.000        91.28/      0.00        1/    0
     3.33        2.071/    0.000       108.18/      0.00        1/    0
     4.17        2.307/    0.000       118.32/      0.00        1/    0
     5.00        2.388/    0.000       121.70/      0.00        1/    0
     5.83        2.307/    0.000       118.32/      0.00        1/    0
     6.67        2.071/    0.000       108.18/      0.00        1/    0
     7.50        1.700/    0.000        91.28/      0.00        1/    0
     8.33        1.215/    0.000        67.61/      0.00        1/    0
     9.17        0.642/    0.000        37.19/      0.00        1/    0
    10.00        0.000/    0.000         0.00/      0.00        1/    0
          B E A M  N O.     1 D E S I G N  R E S U L T S - SHEAR
    AT START SUPPORT - Vu=   37.73 KIP  Vc=   21.34 KIP  Vs=   28.97 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     1
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   47. IN.
      STAAD SPACE                                                -- PAGE NO.
6
    AT END   SUPPORT - Vu=   37.73 KIP  Vc=   21.34 KIP  Vs=   28.97 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     1
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   47. IN.
  ___    1J____________________  120.X  10.X  16_____________________   2J____
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |  8#4 C/C  7                                                 8#4 C/C  7     |
 | 2#10H 2.63   0.TO  120.                                                    |
 ||=========================================================================||
```

```
|                                                                            |
|____________________________________________________________________________|
 _________     _________     _________     _________     _________      _________
|         |   |         |   |         |   |         |   |         |    |         |
|         |   |         |   |         |   |         |   |         |    |         |
|         |   |         |   |         |   |         |   |         |    |         |
|         |   |         |   |         |   |         |   |         |    |         |
| 2#10    |   | 2#10    |   | 2#10    |   | 2#10    |   | 2#10    |    | 2#10    |
|   OO    |   |   OO    |   |   OO    |   |   OO    |   |   OO    |    |   OO    |
|         |   |         |   |         |   |         |   |         |    |         |
|_________|   |_________|   |_________|   |_________|   |_________|    |_________|
  ********************END OF BEAM DESIGN**************************
```

Related Links

- *D1.F.4.6 Beam Design per ACI 318-11 and Earlier* (on page 1224)

V. ACI 318-05 Rectangular Singly Reinforced Beam

Design of a singly reinforced concrete beam per the ACI 318-2005 code.

References

1. ACI 318-05
2. PCA Notes 2005: Example 7.1

Related Links

- *D1.F.4.6 Beam Design per ACI 318-11 and Earlier* (on page 1224)

Details

Determine the Required Area of Tension Reinforcement for given Concrete and Steel Properties with Given Factored Moment.

Select a rectangular beam size and required reinforcement to carry service load moments M_{DL} = *56 ft-kips* and M_{LL}= *35 ft-kips*. Select reinforcement to control flexural cracking.

A 10' simply supported beam is subjected to an ultimate uniform load of 9.736 k/ft.

f'_c = 4,000 psi
f_y = 60,000 psi
cover = 2 in

Validation

This is a complete design procedure for rectangular sections with tension reinforcement only, a minimum beam depth will be computed using the maximum reinforcement permitted for tension-controlled flexural members, ρ_t.

d_{cover} = dist. from bottom of beam up to center of bottom bar = C_c + 1/2(bar diameter) = 2.5 in = 63.5 mm

Determine the maximum tension-controlled reinforcement ratio fro material strengths from Table 6-1 of PCA Notes on ACI 318-05.

$$\rho_{t.} = 0.01806$$

Compute bd^2 required:

$$M_u = 1.2 \cdot M_{DL} + 1.6 \cdot M_{LL} = 123 \text{ ft} \cdot \text{kips}$$

$$R_n = \rho_t f_y \left(1 - 0.5\rho_t \frac{f_y}{0.85f'_c}\right) = 910.9 \text{ psi}$$

$$bd^2{}_{req'd} = \frac{M_u}{\phi R_n} = 1,803 \text{in}^3$$

Size members so that $bd^2_{(prov)} \geq bd^2_{(req'd)}$.

$$b = 10 \text{ in (column width)}$$

$$d = \sqrt{\frac{1,803}{10}} = 13.43 \text{in}$$

Total beam depth, $D > d + d_{cover} = 15.93\ in$; use a $D = 16\ in$; $d = 13.5\ in$.

$$\rho_{prov} = \frac{0.85f'_c}{f_y}\left(1 - \sqrt{1 - \frac{2R_{n,prov}}{085f'_c}}\right) = 0.018$$

$$A_s = \rho_{prov} \cdot b \cdot d_{prov} = 2.406 \text{ in}^2$$

Verify the design.

$$\underline{T = A_s f_y = 2.406 \times 60 = 144.36 \text{ kips}}$$

Design moment strength, $\phi M_n = D_m$

Strength reduction factor, $\phi = 0.9$

$$D_m = \phi\left[A_s f_y \left(d_{prov} - \frac{a}{2}\right)\right] = 123.2 \text{ ft} \cdot \text{kip}$$

$D_m > M_u$, hence design OK.

Compute minimum reinforcement

$$A_{s,\min} = \max \left| \begin{array}{l} \frac{200}{f_y} bd = 0.45 \text{ in}^2 \\ \frac{3\sqrt{f'_c}}{f_y} bd = 0.427 \text{ in}^2 \end{array} \right.$$

Since $A_s > A_{s,min}$, the design satisfies the minimum reinforcement criteria.

Compute the reinforcement provided

Use 2-#10 bars: $A_s = 2.53\ in^2$

Check Bar Spacing

End bar diameter, □ $= 1.27\ in$, two bars in a layer

$$f_s = \frac{2}{3} f_y = 40(10)^3 \text{ psi}$$

Maximum spacing allowed:

$$s_{max} = \min \left| \begin{array}{l} 15\left(\frac{40{,}000}{f_s}\right) - 2.5C_c \\ 12\left(\frac{40{,}000}{f_s}\right) \end{array} \right. = 10 \text{ in}$$

Spacing provided:

$$s_{prov} = \frac{b - (2C_c + 2 \times 0.5\varnothing)}{N - 1} = 4.73 \text{ in}$$

Results

Table 888: Comparison of results

Parameter	Hand Calculation	STAAD.Pro	Difference	Comments
Area of steel required (in^2)	2.406	2.44	negligible	
Maximum bar spacing (in)	10	10	none	
Bar spacing (in)	4.73	4.73	none	

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-05 Rectangular Singly Reinforced Beam.std` is typically installed with the program.

```
STAAD PLANE RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 2005
START JOB INFORMATION
ENGINEER DATE 14-May-2020
JOB NAME PCA Example 7.1
JOB CLIENT Bentley Systems Inc.
ENGINEER NAME TK
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 16 ZD 10
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
```

```
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -4.48
LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI GY -2.8
LOAD COMB 3 ACI 318R-02
1 1.2 2 1.6
PERFORM ANALYSIS
UNIT INCHES KIP
START CONCRETE DESIGN
CODE ACI 2005
TRACK 2 ALL
DESIGN BEAM 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
   =====================================================================
          BEAM  NO.     1 DESIGN RESULTS - FLEXURE PER CODE ACI 318-05
   LEN -  10.00FT.  FY - 60000.  FC - 4000.  SIZE - 10.00 X 16.00 INCHES
   LEVEL    HEIGHT     BAR INFO       FROM            TO          ANCHOR
          FT.    IN.              FT.     IN.     FT.     IN.   STA  END
   _____________________________________________________________________
     1    0 + 2-3/4    2-NUM.10     0 + 0-0/0     10 + 0-0/0    YES  YES
    |----------------------------------------------------------------|
    |   CRITICAL POS MOMENT=    123.20 KIP-FT  AT  5.00 FT, LOAD    3|
    |   REQD STEEL=  2.44 IN2, RHO=0.0183, RHOMX=0.0214 RHOMN=0.0033 |
    |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 2.54/ 4.73 INCH           |
    |   REQD. DEVELOPMENT LENGTH = 48.52 INCH                        |
    |----------------------------------------------------------------|
     Cracked Moment of Inertia Iz at above location =    1817.34 inch^4
   REQUIRED REINF. STEEL SUMMARY :
   -------------------------------
   SECTION     REINF STEEL(+VE/-VE)     MOMENTS(+VE/-VE)    LOAD(+VE/-VE)
   (FEET)         (SQ. INCH)               (KIP-FEET)
     0.00        0.000/    0.000          0.00/      0.00        3/    1
     0.83        0.650/    0.000         37.64/      0.00        3/    0
     1.67        1.231/    0.000         68.44/      0.00        3/    0
     2.50        1.723/    0.000         92.40/      0.00        3/    0
     3.33        2.101/    0.000        109.51/      0.00        3/    0
     4.17        2.341/    0.000        119.78/      0.00        3/    0
     5.00        2.423/    0.000        123.20/      0.00        3/    0
     5.83        2.340/    0.000        119.78/      0.00        3/    0
     6.67        2.101/    0.000        109.51/      0.00        3/    0
     7.50        1.723/    0.000         92.40/      0.00        3/    0
     8.33        1.231/    0.000         68.44/      0.00        3/    0
     9.17        0.650/    0.000         37.64/      0.00        3/    0
    10.00        0.000/    0.000          0.00/      0.00        3/    1
          B E A M  N O.     1 D E S I G N  R E S U L T S - SHEAR
    AT START SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     3
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
```

```
      RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318 2005        -- PAGE NO.
4
    AT END   SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     3
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
  ___   1J____________________  120.X  10.X  16_____________________   2J____
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |  8#4 C/C  7                                                  8#4 C/C  7    |
 | 2#10H 2.63   0.TO  120.                                                    |
 ||==========================================================================||
 |                                                                            |
 |____________________________________________________________________________|

  _________    _________    _________    _________    _________     _________
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |   | 2#10    |
 |   OO    |  |   OO    |  |   OO    |  |   OO    |  |   OO    |   |   OO    |
 |         |  |         |  |         |  |         |  |         |   |         |
 |_________|  |_________|  |_________|  |_________|  |_________|   |_________|
```

V. ACI 318-02 Square Column

Determine the reinforced steel quantity for a square column per ACI 318-02.

Reference

Notes on ACI 318-02 Building Code Requirements for Structural Concrete, Example 7.8, p7-46, Design of Square Column for Biaxial loading.

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

Problem

P = 1,200 kips

M_{uz} = 300 ft·kips

M_{uy} = 125 ft·kips

f'_c = 5,000 psi

f_y = 60,000 psi

Comparison

Table 889: Comparison of results

Result Type	ACI Notes	STAAD.Pro	Difference
Provided steel area per ACI 318-02(in^2)	5.93 (4- #11)	5.93 (4- #11)	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-02 Square Column.STD` is typically installed with the program.

```
STAAD SPACE SQUARE COLUMN DESIGN PER ACI 318-02
START JOB INFORMATION
ENGINEER DATE 21-Sep-18
END JOB INFORMATION
*
* REFERENCE : NOTES ON ACI 318-02 BUILDING CODE REQUIREMENTS
*             FOR STRUCUTURAL CONCRETE EXAMPLE 7.8 (PAGE # 7-46)
*
*  OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A
*              SQUARE COLUMN PER THE ACI 318-02 CODE
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 12 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 24 ZD 24
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT FEET KIP
LOAD 1
JOINT LOAD
2 FY -1200
2 MZ 300
2 MX 125
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 2002
UNIT INCHES KIP
FC 5 ALL
MINMAIN 11 ALL
MAXMAIN 11 ALL
TRACK 2 ALL
```

```
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
  ====================================================================
      COLUMN  NO.     1  DESIGN PER  ACI 318-02 - AXIAL + BENDING
  FY - 60000  FC - 5000 PSI,  SQRE SIZE - 24.00 X 24.00 INCHES, TIED
          ONLY MINIMUM STEEL IS REQUIRED.
          AREA OF STEEL REQUIRED =   5.760  SQ. IN.
  BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI-Z  PHI-Y
  -------------------------------------------------------------------
    4 - NUMBER 11           1.083        1      END     0.650  0.650
  (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
  TIE BAR NUMBER    4 SPACING  22.56 IN
      COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)
   --------------------------------------------------------
     P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
   2795.88   2236.70   1043.87    793.30    9.12
      M0       P-tens.   Des.Pn    Des.Mn     e/h
    294.69   -374.40   1846.15    461.54    0.02083
   --------------------------------------------------------
      COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)
   --------------------------------------------------------
     P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
   2795.88   2236.70   1043.87    793.30    9.12
      M0       P-tens.   Des.Pn    Des.Mn     e/h
    294.69   -374.40   1846.15    192.31    0.00868
   --------------------------------------------------------
                             Pn       Mn       Pn       Mn    (@ Z )
            |              2064.65   486.58  1032.32   793.01
         P0 |*             1892.60   580.71   860.27   780.26
            | *            1720.54   652.53   688.22   755.81
     Pn,max |__*           1548.49   709.10   516.16   677.87
            |   *          1376.43   749.65   344.11   565.14
     Pn     |    *         1204.38   776.44   172.05   435.29
    NOMINAL |     *          Pn       Mn       Pn       Mn    (@ Y )
      AXIAL |      *       2064.65   486.58  1032.32   793.01
COMPRESSION |       *      1892.60   580.71   860.27   780.26
         Pb |-------*Mb    1720.54   652.53   688.22   755.81
            |      *       1548.49   709.10   516.16   677.87
 ___________|____*_______  1376.43   749.65   344.11   565.14
            |  * M0   Mn,  1204.38   776.44   172.05   435.29
            | *   BENDING
     P-tens |*     MOMENT
            |
  ******************END OF COLUMN DESIGN RESULTS*******************
```

V. ACI 318-02 Rectangular Beam

Determine the reinforced steel quantity for a rectangular beam per the ACI 318 code.

Reference

Notes on ACI 318-02 Building Code Requirements for Structural Concrete, Example 7.1, p7-24, Design of Rectangular Beam with Tension Reinforcement Only

Notes on ACI 318-99 Building Code Requirements for Structural Concrete, Example 10.1, p10-10, Design of Rectangular Beam with Tension Reinforcement Only

Related Links

- *D1.F.4.6 Beam Design per ACI 318-11 and Earlier* (on page 1224)

Problem

Dead Load moment (service) = 56 fit-kips

Live Load moment (service) = 35 ft ·kips

f'_c = 4,000 psi

f_y = 60,000 psi

Comparison

Table 890: Comparison of results

Result Type		ACI Notes	STAAD.Pro	Difference
Required steel area (in^2)	per ACI 318-99	2.78	2.793	none
	per ACI 318-02	2.40	2.44	1.7%
Provided steel area (in^2)	per ACI 318-02	2.40 (2 - #8 & 1- #9)	(2- #10) 2.45	1.7%

STAAD.Pro reports that it is unable to find a suitable bar arrangement to satisfy the reinforcement requirement per ACI 318-99. However, this does not mean that it is impossible to come up with a bar arrangement. When STAAD looks for a bar arrangement, it uses only bars of the same size. It begins with the bar size corresponding to the parameter `MINMAIN`. If an arrangement is not possible with that bar, it tries with the next larger bar size. If all the permissible bar sizes are exhausted, the program reports that it could not come up with a bar arrangement. However, the user may be able to satisfy the requirement by mixing bars of various diameters. For example, 3 # 11 bars and 2 # 10 bars may satisfy the requirement. The program is not equipped with facilities to come up with such combinations of bar sizes.

In fact, in the reference mentioned for the 1999 edition of ACI 318, the bar arrangement chosen is 2 # 9 and 1 # 8. It proves the point that finding bars of the same diameter is not possible for the 1999 edition based solution.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-02 Rectangular Beam.STD` is typically installed with the program.

```
STAAD PLANE RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318
START JOB INFORMATION
ENGINEER DATE 21-Sep-18
END JOB INFORMATION
```

```
*
* REFERENCE : NOTES ON ACI 318-02 BUILDING CODE REQUIREMENTS
*             FOR STRUCUTURAL CONCRETE EXAMPLE 7.1 (PAGE # 7-24)
*                             AND
*             NOTES ON ACI 318-99 BUILDING CODE REQUIREMENTS
*             FOR STRUCUTURAL CONCRETE EXAMPLE 10.1 (PAGE # 10-10)
*
*  OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A
*              RECTANGULAR BEAM PER THE ACI 318 CODE
*
*
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 16 ZD 10
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 2 PINNED
UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
1 UNI GY -4.48
LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI GY -2.8
LOAD COMB 3 ACI 318R-99
1 1.4 2 1.7
LOAD COMB 4 ACI 318R-02
1 1.2 2 1.6
PERFORM ANALYSIS
*
** CONCRETE DESIGN AS PER ACI 318R-99
*
UNIT INCHES KIP
LOAD LIST 3
START CONCRETE DESIGN
CODE ACI 1999
FC 4 ALL
FYMAIN 60 ALL
TRACK 1 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
*
** CONCRETE DESIGN AS PER ACI 318R-02
*
LOAD LIST 4
START CONCRETE DESIGN
```

```
CODE ACI 2002
FC 4 ALL
FYMAIN 60 ALL
TRACK 1 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
      RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318             -- PAGE NO.
3
   ======================================================================
          BEAM  NO.     1 DESIGN RESULTS - FLEXURE PER CODE ACI 318-99
   LEN -  10.00FT.  FY - 60000.  FC - 4000.  SIZE - 10.00 X 16.00 INCHES
   LEVEL    HEIGHT     BAR INFO       FROM            TO          ANCHOR
          FT.    IN.              FT.     IN.     FT.     IN.   STA  END
   ______________________________________________________________________
   *** A SUITABLE BAR ARRANGEMENT COULD NOT BE DETERMINED.
   REQD. STEEL =  2.793 IN2,  MAX. STEEL PERMISSIBLE =  2.873 IN2
   MAX POS MOMENT =    137.90 KIP-FT,  LOADING    3
  ___    1J____________________  120.X  10.X  16_____________________   2J____
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |                                                                            |
 |____________________________________________________________________________|
  _________    _________    _________    _________    _________     _________
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |_________|  |_________|  |_________|  |_________|  |_________|   |_________|
   ********************END OF BEAM DESIGN**************************
    57. END CONCRETE DESIGN
    58. *
    59. ** CONCRETE DESIGN AS PER ACI 318R-02
    60. *
    61. LOAD LIST 4
    62. START CONCRETE DESIGN
 CONCRETE DESIGN
    63. CODE ACI 2002
    64. FC 4 ALL
    65. FYMAIN 60 ALL
    66. TRACK 1 ALL
    67. DESIGN BEAM ALL
      RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318             -- PAGE NO.
4
   ======================================================================
          BEAM  NO.     1 DESIGN RESULTS - FLEXURE PER CODE  ACI 318-02
   LEN -  10.00FT.  FY - 60000.  FC - 4000.  SIZE - 10.00 X 16.00 INCHES
```

```
   LEVEL    HEIGHT     BAR INFO       FROM           TO         ANCHOR
          FT.    IN.              FT.     IN.     FT.     IN.   STA  END
   _____________________________________________________________________
     1    0 + 2-3/4    2-NUM.10     0 + 0-0/0     10 + 0-0/0    YES  YES
   |----------------------------------------------------------------|
   |   CRITICAL POS MOMENT=    123.20 KIP-FT  AT   5.00 FT, LOAD    4|
   |   REQD STEEL=  2.44 IN2, RHO=0.0183, RHOMX=0.0214 RHOMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING= 10.00/ 2.54/ 4.73 INCH            |
   |   REQD. DEVELOPMENT LENGTH = 48.52 INCH                         |
   |----------------------------------------------------------------|
     Cracked Moment of Inertia Iz at above location =    1817.34 inch^4
          B E A M  N O.     1 D E S I G N  R E S U L T S - SHEAR
    AT START SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     4
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
    AT END   SUPPORT - Vu=   38.19 KIP  Vc=   21.42 KIP  Vs=   29.50 KIP
    Tu=    0.00 KIP-FT  Tc=   1.95 KIP-FT  Ts=   0.00 KIP-FT  LOAD     4
    NO STIRRUPS ARE REQUIRED FOR TORSION.
    REINFORCEMENT IS REQUIRED FOR SHEAR.
    PROVIDE NUM. 4 2-LEGGED STIRRUPS AT  6.7 IN. C/C FOR   46. IN.
      RECTANGULAR CONCRETE BEAM DESIGN PER ACI 318             -- PAGE NO.
5
  ___    1J____________________  120.X  10.X  16_____________________   2J____
 |                                                                          |
 |                                                                          |
 |                                                                          |
 |  8#4 C/C  7                                                8#4 C/C  7    |
 | 2#10H 2.63   0.TO  120.                                                  |
 ||========================================================================||
 |                                                                          |
 |__________________________________________________________________________|

  _________    _________    _________    _________    _________     _________
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 |         |  |         |  |         |  |         |  |         |   |         |
 | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |  | 2#10    |   | 2#10    |
 |   OO    |  |   OO    |  |   OO    |  |   OO    |  |   OO    |   |   OO    |
 |         |  |         |  |         |  |         |  |         |   |         |
 |_________|  |_________|  |_________|  |_________|  |_________|   |_________|
   *******************END OF BEAM DESIGN**************************
```

V. ACI 318-99 Square Column

Determine the reinforced steel quantity for the square column as per ACI 318-99.

Reference

Chu-Kia Wang and Charles G.Slamon, *Reinforced Concrete Design*, 5th Edition, Examples problem 13.21.1, p509 .

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

Problem

P_u = 144 kips

M_{uz} = 120 ft ·kips

M_{uy} = 54 ft ·kips

f'_c = 3,000 psi

f_y = 60,000 psi

Comparison

Table 891: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
Provided steel area per ACI 318-99(in^2)	8.0 (8- #9)	9.0 (4- #14)	none

Note: In the reference book, the axial capacity provided is 141 kips, which is less than the requirement of 144 kips. In the STAAD implementation of the code, any reinforcement arrangement which yields less capacity than the required value is considered un-acceptable. This is why STAAD reports a higher required steel than the reference book.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-99 Square Column.STD` is typically installed with the program.

```
STAAD SPACE SQUARE COLUMN DESIGN ACI 318-99
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* REFERENCE :  REINFORCED CONCRETE DESIGN 5TH EDITION BY CHU-KIA WANG
*             AND CHARLES G.SALMON  EX.13.21.1 PAGE 509.
*
*  OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A
*              SQUARE COLUMN PER THE ACI 318-99 CODE
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 12 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 16 ZD 16
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
```

```
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
UNIT FEET KIP
LOAD 1
JOINT LOAD
2 FY -144
2 MZ 120
2 MX 54
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 1999
UNIT INCHES KIP
FC 3 ALL
FYMAIN 40 ALL
FYSEC 40 ALL
MINMAIN 9 ALL
TRACK 2 ALL
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
      SQUARE COLUMN DESIGN ACI 318-99                          -- PAGE NO.
3
   ====================================================================
        COLUMN  NO.     1  DESIGN PER ACI 318-99 - AXIAL + BENDING
   FY - 40000  FC - 3000 PSI,  SQRE SIZE - 16.00 X 16.00 INCHES, TIED
           AREA OF STEEL REQUIRED =    8.348  SQ. IN.
   BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
   ----------------------------------------------------------
     4 - NUMBER 14           3.516        1      END     0.700
   (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
   TIE BAR NUMBER    4 SPACING  16.00 IN
       COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)
    --------------------------------------------------------
      P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
     989.85    791.88    312.89    272.99   10.47
       M0       P-tens.   Des.Pn    Des.Mn     e/h
     178.89   -360.00    205.71    171.43    0.06944
    --------------------------------------------------------
       COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)
    --------------------------------------------------------
      P0        Pn max    P-bal.    M-bal.   e-bal.(inch)
     989.85    791.88    312.89    272.99   10.47
       M0       P-tens.   Des.Pn    Des.Mn     e/h
     178.89   -360.00    205.71     77.14    0.03125
    --------------------------------------------------------
                           Pn       Mn       Pn       Mn    (@ Z )
            |             730.97   130.65   365.48   257.21
        P0  |*            670.05   156.16   304.57   272.89
            | *           609.14   178.25   243.66   267.81
      Pn,max|__*          548.22   199.53   182.74   255.16
            |   *         487.31   219.54   121.83   234.02
      Pn    |    *        426.40   238.66    60.91   206.96
     NOMINAL|     *        Pn       Mn       Pn       Mn    (@ Y )
```

```
       AXIAL|      *       730.97   130.65   365.48   257.21
 COMPRESSION|       *      670.05   156.16   304.57   272.89
          Pb|-------*Mb    609.14   178.25   243.66   267.81
            |      *       548.22   199.53   182.74   255.16
 ___________|____*_______  487.31   219.54   121.83   234.02
            |  * M0   Mn,  426.40   238.66    60.91   206.96
            | *   BENDING
      P-tens|*     MOMENT
            |
   ********************END OF COLUMN DESIGN RESULTS********************
```

V. ACI 318-99 Circular Column

Determine the reinforced steel quantity for a circular column per ACI 318-99.

Reference

Chu-Kia Wang and Charles G.Salmon, *Reinforced Concrete Design*, 5th Edition, Examples problem 13.15.1, p481 .

Related Links

- *D1.F.5.6 Column Design in Pre-2014 Codes* (on page 1239)

Problem

P_u = 1,050 kips

$M_{uz} = M_{uy}$ = 0 ft·kips

f'_c = 4,000 psi

f_y = 60,000 psi

Spiral lateral reinforcement used.

Comparison

Table 892: Comparison of results

Result Type	Reference	STAAD.Pro	Difference
Required steel area (in^2)	10.45	10.505	none
Provided steel area (in^2)	10.82 (7- #11)	10.82 (18- #7)	none

Note: STAAD reports a more economical bar arrangement than the reference book.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-99 Circular Column.STD` is typically installed with the program.

```
STAAD SPACE CIRCULAR CONCRETE COLUMN DESIGN PER ACI 318-99
START JOB INFORMATION
```

```
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
*
* REFERENCE : REINFORCED CONCRETE DESIGN, 5TH EDITION BY CHU-KIA WANG
*             AND CHARLES G.SALMON, EX.13.15.1 PAGE 481
*
*  OBJECTIVE : TO DETERMINE THE REINFORCED STEEL QUANTITY FOR A
*              CIRCULAR COLUMN PER THE ACI 318-99 CODE
*
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 12 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 PRIS YD 20
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 FIXED
LOAD 1
JOINT LOAD
2 FY -1060
PERFORM ANALYSIS
START CONCRETE DESIGN
CODE ACI 1999
REINF 1 ALL
DESIGN COLUMN ALL
END CONCRETE DESIGN
FINISH
```

STAAD Output

```
   ====================================================================
        COLUMN  NO.     1  DESIGN PER ACI 318-99 - AXIAL + BENDING
   FY - 60000  FC - 4000 PSI,  CIRC SIZE  20.00 INCHES DIAMETER  SPIR
           AREA OF STEEL REQUIRED =  10.505  SQ. IN.
   BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
   ----------------------------------------------------------
    18 - NUMBER  7           3.438        1      END     0.750
   (EQUALLY SPACED)
   TIE BAR NUMBER    4 SPACING   2.00 IN
   ********************END OF COLUMN DESIGN RESULTS********************
```

V. ACI 318-99 Beam and Column Reinforcement

Determine the area of reinforcing steel required for a beam and column in a moment frame.

Reference

CRSI Handbook and Notes on ACI-318-99.

Related Links

- *D1.F.4.6 Beam Design per ACI 318-11 and Earlier* (on page 1224)

Problem

A plane frame is created with such loading as to create 138 Kip-Ft moment on beam and 574 Kip of axial load coupled with above moment on column.

Size of the beam is 10" x 16"and size of the column 14" x 16".

$P = 521.32$ kip

$w = 5.268$ kip/ft

$L = 20$ ft, $H = 15$ ft

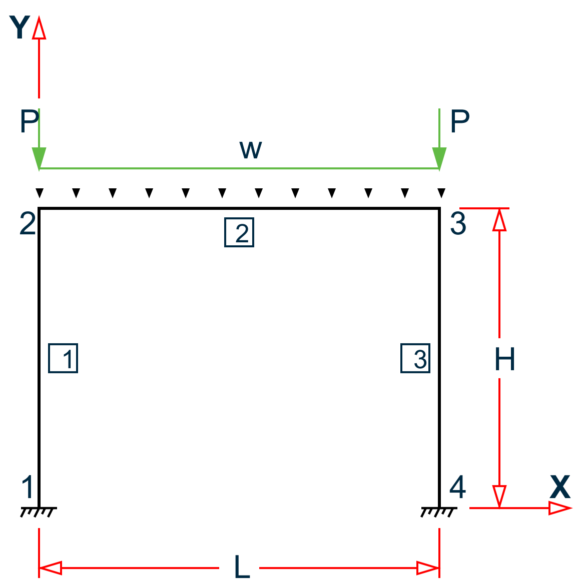


Figure 512: Concrete moment frame

Comparison

Table 893: Comparison of results

Result Type	Theory ACI Notes	Theory CRSI Handbook	STAAD.Pro	Difference
Area of Steel in beam	2.78 in^2	X	2.792 in^2	negligible
Area of Steel in column	X	4.01%	4.09% required(*) 4.232% provided	negligible

Note: (*) Area of steel required calculated as 9.164 in^2, which is 4.09% of a 14"x16" column.

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\10 Concrete Design\US\ACI 318-99 Beam and Column Reinforcement.STD` is typically installed with the program.

```
STAAD PLANE VERIFICATION FOR CONCRETE DESIGN
START JOB INFORMATION
ENGINEER DATE 23-Sep-18
END JOB INFORMATION
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 15 0; 3 20 15 0; 4 20 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
UNIT INCHES KIP
MEMBER PROPERTY AMERICAN
1 3 PRIS YD 16 ZD 14
2 PRIS YD 16 ZD 10
DEFINE MATERIAL START
ISOTROPIC MATERIAL1
E 3150
POISSON 0.17
END DEFINE MATERIAL
CONSTANTS
MATERIAL MATERIAL1 ALL
SUPPORTS
1 4 FIXED
UNIT FEET KIP
LOAD 1 DEAD + LIVE
JOINT LOAD
2 3 FY -521.32
MEMBER LOAD
2 UNI GY -5.268
PERFORM ANALYSIS
PRINT MEMBER FORCES
START CONCRETE DESIGN
CODE ACI 1999
UNIT INCHES KIP
TRACK 1 MEMB 2
FYMAIN 60 ALL
FC 4 ALL
CLB 1.4375 ALL
DESIGN BEAM 2
DESIGN COLUMN 1
END CONCRETE DESIGN
FINISH
```

STAAD Output

V.11 Timber Design

V. Canada

V. CSA 086 2001 - Glulam in Compression

Reference

Example 4, page 116, Canadian Wood Design Manual, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Length = 9,000 mm

Comparison

Table 894: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength (kN)	295	293.793	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086 2001 - Glulam in Compression.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 9 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-12C-E
E 9.7
POISSON 0.15
DENSITY 1.44676e-005
ALPHA 5.5e-006
```

```
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 0.748031 ZD 0.574147
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_SPRUCE-PINE-12C-E MEMB 1
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
JOINT LOAD
2 FY -214
PERFORM ANALYSIS
PARAMETER
CODE TIMBER CANADIAN
KY 0.5 ALL
KZ 0.5 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (S086)
                         ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
     1    175.00X228.00 CANADIAN GLULAM GRADE:GLT_SPRUCE-PINE-12C-
E
                         PASS      CL.5.5.10/6.5       0.728         1
                      214.00 C          0.00          0.00     0.0000
   |-----------------------------------------------------------------------|
   | LEZ =   4500.000 LEY =   4500.000 LUZ =   9000.000 LUY =   9000.000mm    |
   |                                                                       |
   | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
   | KSC = 1.000   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 1.000 |
   | KZV = 1.000   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.000   CHIX = 1.000 |
   | CV  = 1.000   KN    = 1.000                                             |
   |                                                                       |
   | ACTUAL LOADS : (KN-m)                                                  |
   |                     Pu  =      214.000                                 |
   |                     Tu  =        0.000                                 |
   |                     Muy =        0.000                                 |
   |                     Muz =        0.000                                 |
   |                     V   =        0.000                                 |
   |           SLENDERNESS_Y =       19.737                                 |
   |           SLENDERNESS_Z =       25.714                                 |
   | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                            |
   |                     PY  =      413.943                                 |
   |                     PZ  =      293.793                                 |
   |                     T   =        0.000                                 |
   |                     MY  =        0.000                                 |
   |                     MZ  =        0.000                                 |
   |                     V   =        0.000                                 |
```

```
|              SLENDERNESS =       50.000                                    |
|--------------------------------------------------------------------------|
```

V. CSA 086 2001 - Glulam in Bending

Reference

Example 2, page 59, Canadian Wood Design Manual, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Length = 7,500 mm, Beam Spacing = 5,000 mm, Standard load condition, Dry service condition, Untreated

Comparison

Table 895: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength in Bending (kN·m)	208	208.323	none
Design Strength in Shear (kN)	101	100.776	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086 2001 - Glulam in Bending.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 7.5 0 0
MEMBER INCIDENCES
1 1 2
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-12C-E
E 9.7
POISSON 0.15
DENSITY 1.44676E-005
ALPHA 5.5E-006
ISOTROPIC GLT_D.FIR-L-20F-E
E 12.4
```

```
POISSON 0.15
DENSITY 1.44676E-005
ALPHA 5.5E-006
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68E-005
ALPHA 5.5E-006
DAMP 0.05
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 2.11942 ZD 0.426508
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_D.FIR-L-20F-E MEMB 1
SUPPORTS
1 2 PINNED
UNIT METER KN
LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
MEMBER LOAD
1 UNI GY -27.1
PERFORM ANALYSIS
PARAMETER
CODE TIMBER CANADIAN
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (S086)
                         ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
     1    130.00X646.00 CANADIAN GLULAM GRADE:GLT_D.FIR-L-20F-
E
                         FAIL      CL.5.5.5/6.5.       1.008         1
                        0.00 T          0.00          0.00     7.5000
   |-----------------------------------------------------------------------|
   | LEZ =   7500.000 LEY =   7500.000 LUZ =   7500.000 LUY =   7500.000mm   |
   |                                                                       |
   | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
   | KSC = 1.000   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 1.000 |
   | KZV = 1.000   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.000   CHIX = 1.000 |
   | CV  = 1.000   KN    = 1.000                                             |
   |                                                                       |
   | ACTUAL LOADS : (KN-m)                                                 |
   |                     Pu  =        0.000                                |
   |                     Tu  =        0.000                                |
   |                     Muy =        0.000                                |
   |                     Muz =        0.000                                |
   |                     V   =     -101.625                                |
   |           SLENDERNESS_Y =       16.932                                |
   |           SLENDERNESS_Z =        1.529                                |
   | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                           |
```

```
|                       PY  =        0.000                                    |
|                       PZ  =        0.000                                    |
|                       T   =        0.000                                    |
|                       MY  =       41.923                                    |
|                       MZ  =      208.323                                    |
|                       V   =      100.776                                    |
|              SLENDERNESS =       50.000                                    |
|---------------------------------------------------------------------------|
```

V. CSA 086 2001 - Glulam in Tension

Reference

Example 3, page 158, Canadian Wood Design Manual, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Dry service condition, Untreated

Comparison

Table 896: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength in Tension (kN)	257	256.636	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086 2001 - Glulam in Tension.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 10-JUN-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 9 0;
MEMBER INCIDENCES
1 1 2;
UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT_SPRUCE-PINE-14T-E
E 10.7
POISSON 0.15
DENSITY 1.44676e-05
```

```
ALPHA 5.5e-06
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-05
ALPHA 5.5e-06
DAMP 0.05
END DEFINE MATERIAL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 PRIS YD 0.872702 ZD 0.262467
UNIT INCHES KIP
CONSTANTS
MATERIAL GLT_SPRUCE-PINE-14T-E ALL
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 LOADTYPE NONE  TITLE LOAD CASE 1
JOINT LOAD
2 FY 250
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE TIMBER CANADIAN
KY 0.5 ALL
KZ 0.5 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (S086)
                         ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
     1     80.00X266.00 CANADIAN GLULAM GRADE:GLT_SPRUCE-PINE-14T-
E
                         PASS      CL.5.5.10/6.5       0.974         1
                      250.00 T          0.00          0.00     0.0000
   |-----------------------------------------------------------------------|
   | LEZ =   4500.000 LEY =   4500.000 LUZ =   9000.000 LUY =   9000.000mm  |
   |                                                                       |
   | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
   | KSC = 1.000   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 1.000 |
   | KZV = 1.000   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.000   CHIX = 1.000 |
   | CV  = 1.000   KN    = 1.000                                           |
   |                                                                       |
   | ACTUAL LOADS : (KN-m)                                                 |
   |                     Pu  =        0.000                                |
   |                     Tu  =     -250.000                                |
   |                     Muy =        0.000                                |
   |                     Muz =        0.000                                |
   |                     V   =        0.000                                |
   | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                           |
   |                     PY  =        0.000                                |
   |                     PZ  =        0.000                                |
```

```
|                          T    =        256.636                             |
|                          MY   =          0.000                             |
|                          MZ   =          0.000                             |
|                          V    =          0.000                             |
|---------------------------------------------------------------------------|
```

V. CSA 086 2001 - Beam in Compression

Reference

Example 2, page 113, Canadian Wood Design Manual, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Unbraced Length = 5,000 mm

Comparison

Table 897: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength (kN)	130	129.223	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086 2001 - Beam in Compression.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 08-JUN-05
END JOB INFORMATION
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 0 16.4042 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC DFL_NO2_8X8_POST
E 1.368e+06
POISSON 0.15
DENSITY 25
ALPHA 5.5e-06
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL DFL_NO2_8X8_POST ALL
UNIT FEET POUND
```

```
MEMBER PROPERTY TIMBER CANADIAN
1 TABLE ST DFL_NO2_8X8_POST
SUPPORTS
1 PINNED
2 FIXED BUT FY
UNIT METER KN
LOAD 1 DEAD+LIVE LOAD
JOINT LOAD
2 FY -114
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE TIMBER CANADIAN
KSC 0.91 ALL
K_ZC 1.05 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - (S086)
                        ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY              MZ       LOCATION
 =======================================================================
     1 ST DFL_NO2_8X8_POST
                        PASS      CL.5.5.10/6.5.12       0.882         1
                     114.00 C          0.00           0.00     0.0000
  |-----------------------------------------------------------------------|
  | LEZ =   5000.000 LEY =   5000.000 LUZ =   5000.000 LUY =   5000.000mm |
  |                                                                       |
  | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
  | KSC = 0.910   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 1.000 |
  | KZV = 1.000   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.050   CHIX = 1.000 |
  | CV  = 1.000   KN    = 1.000                                           |
  |                                                                       |
  | ACTUAL LOADS : (KN-m)                                                 |
  |                       Pu  =      114.000                              |
  |                       Tu  =        0.000                              |
  |                       Muy =        0.000                              |
  |                       Muz =        0.000                              |
  |                       V   =        0.000                              |
  |             SLENDERNESS_Y =       26.178                              |
  |             SLENDERNESS_Z =       26.178                              |
  | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                           |
  |                       PY  =      129.223                              |
  |                       PZ  =      129.223                              |
  |                       T   =        0.000                              |
  |                       MY  =        0.000                              |
  |                       MZ  =        0.000                              |
  |                       V   =        0.000                              |
  |               SLENDERNESS =       50.000                              |
  |-----------------------------------------------------------------------|
```

V. CSA 086 2001 - Beam in Bending

Reference

Example 1, page 58, Canadian Wood Design Manual, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Length =6,000mm, Beam Spacing = 3,000mm, Standard load condition, Dry service condition, Untreated

Comparison

Table 898: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength in Bending (kN·m)	79.8	79.732	none
Design Strength in Shear (kN)	46.1	46.170	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Verification Models\11 Timber Design\Canada\CSA 086 2001 - Beam in Bending.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 08-JUN-05
END JOB INFORMATION
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 3 0 0;
MEMBER INCIDENCES
1 1 3; 2 3 2;
UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC DFL_NO1_10X16_BM
E 1.728e+06
POISSON 0.15
DENSITY 25
ALPHA 5.5e-06
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL DFL_NO1_10X16_BM ALL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
```

```
1 2 TABLE ST DFL_NO1_10X16_BM
SUPPORTS
1 2 FIXED
UNIT METER KN
LOAD 1 DEAD+LIVE LOAD
MEMBER LOAD
1 2 UNI GY -16.4
PERFORM ANALYSIS
PARAMETER 1
CODE TIMBER CANADIAN
KD 1 ALL
K_T 1 ALL
KSB 1 ALL
KZB 0.9 ALL
KZV 0.9 ALL
K_ZC 1.05 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                         STAAD.Pro CODE CHECKING - (S086)
                         ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
     1 ST DFL_NO1_10X16_BM
                         FAIL      CL.5.5.5/6.5.6        1.066         1
                        0.00 T          0.00          49.20     0.0000
   |-----------------------------------------------------------------------|
   | LEZ =   3000.000 LEY =   3000.000 LUZ =   3000.000 LUY =   3000.000mm |
   |                                                                       |
   | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
   | KSC = 1.000   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 0.900 |
   | KZV = 0.900   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.050   CHIX = 1.000 |
   | CV  = 1.000   KN    = 1.000                                          |
   |                                                                       |
   | ACTUAL LOADS : (KN-m)                                                 |
   |                     Pu  =        0.000                                |
   |                     Tu  =        0.000                                |
   |                     Muy =        0.000                                |
   |                     Muz =       49.200                                |
   |                     V   =       49.200                                |
   |           SLENDERNESS_Y =        4.511                                |
   |           SLENDERNESS_Z =        2.158                                |
   | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                           |
   |                     PY  =        0.000                                |
   |                     PZ  =        0.000                                |
   |                     T   =        0.000                                |
   |                     MY  =       79.800                                |
   |                     MZ  =       79.732                                |
   |                     V   =       46.170                                |
   |             SLENDERNESS =       50.000                                |
   |-----------------------------------------------------------------------|
      STAAD PLANE                                              -- PAGE NO.
4
```

```
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
    2 ST DFL_NO1_10X16_BM
                        FAIL      CL.5.5.5/6.5.6         1.066         1
                       0.00 T          0.00          49.20     3.0000
   |-----------------------------------------------------------------------------|
   | LEZ =   3000.000 LEY =   3000.000 LUZ =   3000.000 LUY =   3000.000mm       |
   |                                                                             |
   | KD  = 1.000   KH    = 1.000   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
   | KSC = 1.000   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 0.900 |
   | KZV = 0.900   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.050   CHIX = 1.000 |
   | CV  = 1.000   KN    = 1.000                                               |
   |                                                                             |
   | ACTUAL LOADS : (KN-m)                                                       |
   |                     Pu  =        0.000                                      |
   |                     Tu  =        0.000                                      |
   |                     Muy =        0.000                                      |
   |                     Muz =       49.200                                      |
   |                     V   =      -49.200                                      |
   |           SLENDERNESS_Y =        4.511                                      |
   |           SLENDERNESS_Z =        2.158                                      |
   | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                                 |
   |                     PY  =        0.000                                      |
   |                     PZ  =        0.000                                      |
   |                     T   =        0.000                                      |
   |                     MY  =       79.800                                      |
   |                     MZ  =       79.732                                      |
   |                     V   =       46.170                                      |
   |             SLENDERNESS =       50.000                                      |
   |-----------------------------------------------------------------------------|
```

V. CSA 086 2001 - Beam in Tension

Reference

Example 2, page 158, *Canadian Wood Design Manual*, 2001

Related Links

- *D4.D.5 Member Resistance* (on page 1442)

Given

Dry service condition, Untreated

Comparison

Table 899: Comparison of results

Criteria	Reference	STAAD.Pro	Difference
Design Strength in Tension (kN)	185	184.338	none

STAAD Input

The file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\ Verification Models\11 Timber Design\Canada\CSA 086 2001 - Beam in Tension.STD` is typically installed with the program.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 08-JUN-05
END JOB INFORMATION
UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 0 16.4042 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC DFL_NO1_6X8_BM
E 1.728e+06
POISSON 0.15
DENSITY 25
ALPHA 5.5e-06
END DEFINE MATERIAL
UNIT METER KN
CONSTANTS
MATERIAL DFL_NO1_6X8_BM ALL
UNIT FEET POUND
MEMBER PROPERTY TIMBER CANADIAN
1 TABLE ST DFL_NO1_6X8_BM
SUPPORTS
1 PINNED
UNIT METER KN
LOAD 1 DEAD+LIVE LOAD
JOINT LOAD
2 FY 144
PERFORM ANALYSIS PRINT STATICS CHECK
PARAMETER 1
CODE TIMBER CANADIAN
KH 1.1 ALL
KSC 0.91 ALL
K_ZC 1.05 ALL
CHECK CODE ALL
FINISH
```

STAAD Output

```
                        STAAD.Pro CODE CHECKING - (S086)
                        ***********************
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX           MY             MZ       LOCATION
=======================================================================
    1 ST DFL_NO1_6X8_BM
                        PASS      CL.5.5.10/6.5.12       0.781         1
                     144.00 T          0.00          0.00     0.0000
  |-----------------------------------------------------------------------|
  | LEZ =   5000.000 LEY =   5000.000 LUZ =   5000.000 LUY =   5000.000mm   |
  |                                                                       |
  | KD  = 1.000   KH    = 1.100   KT   = 1.000   KSB  = 1.000   KSV  = 1.000 |
  | KSC = 0.910   K_SCP = 1.000   KSE  = 1.000   KST  = 1.000   KZB  = 1.000 |
  | KZV = 1.000   KZT   = 1.000   KZCP = 1.000   K_ZC = 1.050   CHIX = 1.000 |
  | CV  = 1.000   KN    = 1.000                                            |
  |                                                                       |
  | ACTUAL LOADS : (KN-m)                                                 |
  |                     Pu  =        0.000                                |
  |                     Tu  =     -144.000                                |
  |                     Muy =        0.000                                |
  |                     Muz =        0.000                                |
  |                     V   =        0.000                                |
  | ALLOWABLE CAPACITIES OF THE SECTION: (KN-m)                           |
  |                     PY  =        0.000                                |
  |                     PZ  =        0.000                                |
  |                     T   =      184.338                                |
  |                     MY  =        0.000                                |
  |                     MZ  =        0.000                                |
  |                     V   =        0.000                                |
  |-----------------------------------------------------------------------|
```

V. Europe

V. EC5 - Timber Column

A Timber Column of length 1.0 meter, having c/s dimension of 73 mm × 198 mm, is subjected to an axial compressive force of 50.0 kN. Design the member for the ultimate limit state.

Details

Material properties:

Timber class: C24
Service classes: Class 2, moisture content ≤ 20%
Load duration classes: Medium-term

Cross section properties:

Length of the member is 1 m.

Rectangular cross section, b = 73 mm, h = 198 mm,
Effective cross sectional area A = 14,454 mm^2,
Radius of gyration of cross section about y-axis r_y = 21 mm,
Radius of gyration of cross section about z-axis r_z = 57 mm,
Section modulus of cross section about z-axis: $W_z = 4.770 \times 10^5$ mm^3
Section modulus of cross section about y-axis: $W_y = 1.759 \times 10^5$ mm^3

Validation

Characteristic material properties for timber:

Modification factor $Kmod = 0.80$ ...from table 3.1

Material factors $\gamma_m = 1.30$ ... from table 2.3

$$f_{c0k} = 21.00 \text{ N/mm}^2$$

$$F_{c0d} = (Kmod \cdot f_{c0k})/\gamma_m = (0.80 \cdot 21.00)/1.30 = 12.92 \text{ N/mm}^2 \text{ [Cl 2.4.1(1)P]}$$

Cross section loads:

$$F_x = 50.000 \text{ kN}$$

Compression parallel to the grain:

$$S_{c0d} = (1000xF_x)/A = (1000x50.000)/14454 = 3.46\text{N/mm}^2 < 12.92\text{N/mm}^2 \; (F_{c0d})$$

The ratio of actual compressive stress to allowable compressive strength:

$$S_{c0d} / F_{c0d} = 3.46 / 12.92 = 0.268 < 1.0 \qquad \text{[Cl. 6.1.4.(1)P]}$$

Check for Slenderness:

Slenderness ratios:

$$\lambda_z = (1000/57) = 17.54$$

$$\lambda_y = (1000/21) = 47.62$$

$$E_{0,mean} = 1.1031 \text{ kN/m}^2$$

As timber grade is C24 (i.e., Soft Wood)

$$E_{0,05} = 0.67 \cdot E_{0,mean} = 0.739 \text{ kN/m}^2$$

[Annex A,EN 338:2003]

$$\lambda_{rel,z} = \lambda_z/\pi \cdot (f_{c0k}/E_{0,05})^{1/2} = 17.54/\pi(21.00/0.739)^{1/2} = 0.298$$

$$\lambda_{rel,y} = \lambda_y/\pi \cdot (f_{c0k}/E_{0,05})^{1/2} = 47.62/\pi(21.00/0.739)^{1/2} = 0.809$$

Since, $\lambda_{rel,y}$ is greater than 0.3, following conditions should be satisfied:

$$S_{c0d}/(Kcz \cdot F_{c0d}) + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) \leq \text{RATIO}$$

$$S_{c0d}/(Kcy \cdot F_{c0d}) + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) \leq \text{RATIO}$$

Where:

$$Kz = 0.5 \cdot [1 + \beta_c \cdot (\lambda_{rel,z} - 0.3) + (\lambda_{rel,z})^2] = 0.50 \cdot [1 + 0.2(0.298 - 0.3) + (0.298)^2] = 0.541$$

$$Ky = 0.5 \cdot [1 + \beta_c \cdot (\lambda_{rel,y} - 0.3) + (\lambda_{rel,y})^2] = 0.50 \cdot [1 + 0.2(0.809 - 0.3) + (0.809)^2] = 0.878$$

$$Kcz = 1/\{K_z + [(K_z)^2 - (\lambda_{rel,z})^2]^{1/2}\} = 1/\{0.541 + [(0.541)^2 - (0.298)^2]^{1/2}\} = 1.008$$

$$Kcy = 1/\{K_y + [(K_{zy})^2 - (\lambda_{rel,y})^2]^{1/2}\} = 1/\{0.878 + [(0.878)^2 - (0.809)^2]^{1/2}\} = 0.820$$

For Rectangular cross section $Km = 0.70$. The member is subjected to Compression only, so actual bending stress is zero.

$$S_{c0d}/(Kcz \cdot F_{c0d}) + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) = 3.46/(1.008 \cdot 12.92) + 0.0 + 0.0 = 0.268 + 0.0 + 0.0 = 0.266$$

$$S_{c0d}/(Kcy \cdot F_{c0d}) + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) = 3.46 /(0.820 \cdot 12.92) + 0.0 + 0.0 = 0.326 + + 0.0 + 0.0 =$$

Hence the critical ratio is 0.326 < 1.0 and the section is safe.

Results

Table 900: EC 5: Part 1-1 Verification Example 1

Criteria	Reference	STAAD.Pro	Difference	Comments
Critical Ratio (Cl. 6.3.2)	0.326	0.327	negligible	

STAAD.Pro Input

The following file is included as
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\.`

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1.0 0 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC WOOD
E 1.10316e+007
POISSON 0.15
DENSITY 0.00231749
ALPHA 5.5e-006
END DEFINE MATERIAL
CONSTANTS
MATERIAL WOOD MEMB 1
MEMBER PROPERTY
1 PRIS YD 0.198 ZD 0.073
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
JOINT LOAD
2 FX -50
PERFORM ANALYSIS
PARAMETER
CODE TIMBER EC5
ALPHA 0 ALL
LDC 3 ALL
SCL 2 ALL
TSC 6 ALL
```

```
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                         STAAD.Pro CODE CHECKING - (EC5 )
                         ***********************
 ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
     1    PRIS ZD =      0.073 YD =      0.198
                         PASS      CL.6.3.2            0.327         1
                       50.00 C         0.00            0.00     0.0000
   |-------------------------------------------------------------------------|
   | AX =     0.01  IY =          0.00  IZ =           0.00                  |
   | LEZ =     1.00  LEY =     1.00                                          |
   |                                                                         |
   | ALLOWABLE STRESSES: (NEW MMS)                                           |
   |                       FBY   =       14.769 FBZ   =       14.769         |
   |                       FC    =       12.859                              |
   | ACTUAL STRESSES : (NEW MMS)                                             |
   |                       fby   =        0.000 fbz   =        0.000         |
   |                       fc    =        3.459                              |
   |-------------------------------------------------------------------------|
```

Related Links

- *D5.E.2 Analysis Methodology* (on page 1584)

V. EC5 - Timber Column with Bending

A timber column of length 1.0 meter, having c/s dimension of 73 mm × 198 mm, is subjected to an axial compressive force of 5.0 kN and moments of 2.0 kN·m and 1.0 kN·m about its major and minor axes respectively. Design the member for the ultimate limit state.

Details

Material properties:

Timber Strength Class: C24
Service classes: Class 2, moisture content <=20%
Load duration: Medium-term

Cross section properties:

Length of the member is 1 m.
Rectangular cross section, b = 73 mm, h = 198 mm,
Effective cross sectional area A = 14454 mm^2,
Radius of gyration of cross section about y-axis r_y = 21 mm,
Radius of gyration of cross section about z-axis r_z = 57 mm,
Section modulus of cross section about z-axis: $W_z = 4.770\text{x}10^5$ mm^3
Section modulus of cross section about y-axis: $W_y = 1.759\text{x}10^5$ mm^3

Validation

Characteristic material properties for timber:

Modification factor *Kmod = 0.80* ...from table 3.1

Material factors γ_m = *1.30* ... from table 2.3

f_{c0k}= 21.00 N/mm^2

$E_{0,05}$ = 7370 N/mm^2

F_{c0d}= (Kmod ·f_{c0k})/γ_m = (0.80 ·21.00)/1.30 = 12.92 N/mm^2 [Cl 2.4.1(1)P]

f_{myk} = 24.00 N/mm^2

F_{myd} = Kmod ·f_{myk}/γ_m = (0.80x24.00)/1.30 = 14.77N/mm^2

f_{mzk} = 24.00 N/mm^2

F_{mzd} = Kmod ·f_{mzk}/γ_m = (0.80x24.00)/1.30 = 14.77N/mm^2

Cross section loads:

F_x = 5.000 kN

M_z = 2.000 kN ·m

M_y = 1.000 kN ·m

Check for Slenderness:

Slenderness ratios:

λ_z = (1000/57) = 17.54

λ_y = (1000/21) = 47.62

$\lambda_{rel,z} = \lambda_z/\pi \cdot (f_{c0k}/E_{0,05})^{1/2}$ = 17.54/π(21.00/7370)$^{1/2}$ = 0.298

$\lambda_{rel,y} = \lambda_y/\pi \cdot (f_{c0k}/E_{0,05})^{1/2}$ = 47.62/π(21.00/7370)$^{1/2}$ = 0.809

Since, $\lambda_{rel,y}$ is greater than 0.3, following conditions should be satisfied [Cl 6.3.2.3]:

S_{c0d}/(Kcz ·F_{c0d}) + (S_{mzd}/F_{mzd}) + Km ·(S_{myd}/F_{myd}) ≤ `RATIO`

S_{c0d}/(Kcy ·F_{c0d}) + Km ·(S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) ≤ `RATIO`

Where:

Kz = 0.5 ·[1 + β_c ·($\lambda_{rel,z}$ - 0.3) + ($\lambda_{rel,z}$)^2] = 0.50 ·[1 + 0.2(0.298 - 0.3) + (0.298)^2] = 0.541

Ky = 0.5 ·[1 + β_c ·($\lambda_{rel,y}$ - 0.3) + ($\lambda_{rel,y}$)^2] = 0.50 ·[1 + 0.2(0.809 - 0.3) + (0.809)^2] = 0.878

Kcz = 1/{K_z + [(K_z)^2 - ($\lambda_{rel,z}$)^2]$^{1/2}$} = 1/{0.541 + [(0.541)^2 - (0.298)^2]$^{1/2}$}= 1.008

Kcy = 1/{K_y + [(K_{zy})^2 - ($\lambda_{rel,y}$)^2]$^{1/2}$} = 1/{0.878 + [(0.878)^2 - (0.809)^2]$^{1/2}$} = 0.820

For Rectangular cross section *Km = 0.70*.

S_{c0d} = (1000 ·F_x/A) = (1000 ·5.000)/14454 = 0.35 N/mm^2

S_{mzd} = (10^6 ·M_z)/W_z = (10^6 ·2.000)/(4.770x10^5) = 4.19 N/mm^2

S_{myd} = (10^6 ·M_y)/W_y = (10^6 ·1.000)/(1.759x10^5) = 5.69 N/mm^2

Combined stress ratio:

$S_{c0d}/(Kcz \cdot F_{c0d}) + (S_{mzd}/F_{mzd}) + Km \cdot (S_{myd}/F_{myd}) = 0.35/(1.008 \cdot 12.92) + 4.19/14.77 + 0.70(5.69/14.77) = 0.027 + 0.283 + 0.269 = 0.266$

$S_{c0d}/(Kcy \cdot F_{c0d}) + Km \cdot (S_{mzd}/F_{mzd}) + (S_{myd}/F_{myd}) = 0.35 /(0.820 \cdot 12.92) + 0.70(4.19/14.77) + 5.69/14.77 = 0.033 + 0.385 + 0.198 =$

Hence the critical ratio is 0.616 < 1.0 and the section is safe.

Results

Table 901: EC 5: Part 1-1 Verification Example 2

Criteria	Reference	STAAD.Pro	Difference	Comments
Critical Ratio (Cl. 6.3.2)	0.616	0.616	none	

STAAD.Pro Input

The following file is included as
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\.`

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 1 0;
MEMBER INCIDENCES
1 1 2;
DEFINE MATERIAL START
ISOTROPIC WOOD
E 1.10316e+007
POISSON 0.15
DENSITY 0.00231749
ALPHA 5.5e-006
END DEFINE MATERIAL
CONSTANTS
MATERIAL WOOD MEMB 1
MEMBER PROPERTY
1 PRIS YD 0.198 ZD 0.073
SUPPORTS
1 FIXED
LOAD 1 LOADTYPE NONE TITLE LOAD CASE 1
JOINT LOAD
2 FY -5.0 MX 1.0 MZ 2.0
PERFORM ANALYSIS
PARAMETER
CODE TIMBER EC5
ALPHA 0 ALL
LDC 3 ALL
SCL 2 ALL
TSC 6 ALL
TRACK 2 ALL
CHECK CODE ALL
FINISH
```

STAAD.Pro Output

```
                        STAAD.Pro CODE CHECKING - (EC5 )
                        ***********************
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
    1    PRIS ZD =      0.073 YD =      0.198
                        PASS      CL.6.3.2            0.616         1
                       5.00 C          1.00          -2.00     0.0000
  |-----------------------------------------------------------------------|
  | AX =     0.01  IY =          0.00  IZ =           0.00                |
  | LEZ =     1.00  LEY =     1.00                                        |
  |                                                                       |
  | ALLOWABLE STRESSES: (NEW MMS)                                         |
  |                     FBY  =       14.769 FBZ   =       14.769          |
  |                     FC   =       12.859                               |
  | ACTUAL STRESSES : (NEW MMS)                                           |
  |                     fby  =        5.686 fbz   =        4.193          |
  |                     fc   =        0.346                               |
  |-----------------------------------------------------------------------|
```

Related Links

- *D5.E.2 Analysis Methodology* (on page 1584)

C

Application Examples

This section contains various examples of analytical models, physical models, macro script files, and more which can be helpful in familiarizing yourself with aspects of STAAD.Pro features.

This UK and US sections include examples of various problems that can be solved using the STAAD engine. These examples represent various structural analyses and design problems commonly encountered by structural engineers.

The example models included here along with many other files are installed with the program by default. The default location for these files is `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\`.

Note: The option to include these files is selected during the product installation.

EX. Building Planner Example Models

The following models are included when STAAD.Pro is installed.

These files are located in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Building Planner\` folder (default location).

Table 902: Building Planner examples installed with the program

File name	Description
`Circular Building.std`	This model demonstrates how circular shaped non-orthogonal building can be created accurately in the Building Planner workflow. This is a complex model and with the use of plan in dxf format received from architect (included in this folder), the entire plan is created. With the use of tools like Copy, Paste, and Mirror, the symmetrical circular structure can be created by just using one half of the model created in dxf. Columns with different beta angle can be modeled effectively and accurately using the Building Planner.
`Typical Ground +3 Story Building.std`	This model is of a typical 4 story building having 3 different plans: Plinth Plan, Typical Plan, and Roof Plan. The Plinth plan consist of beam and column without slabs. The Roof Plan has wall loads only for parapet walls.

File name	Description
`Typical High-Rise Structure.std`	This model is of a typical high-rise residential structure (15 Levels). The model is orthogonal and has a E Shaped Lift Core in between. Two different plans (Plinth and Typical) are created and the same typical plan geometry with varying column sizes is used on all the levels.

Note: The file name of the STAAD input file is listed. Other files associated with this model are included with the same base file name in this folder using different file extensions.

EX. Chinese Design Examples

The following models are included when STAAD.Pro is installed.

These files are located in the
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\China\`
folder (default location).

Table 903: Chinese steel design examples installed with the program

File name	Description
`Chn-1 Plane Frame with Steel Design.STD`	A space steel structure is designed per GB 50017-2017 code and some typical design parameters is added at the same time. Moreover, seismic load, wind load and load combination are auto generated per GB 50009-2012 and GB50011-2010. There are two files in this example: a STAAD.Pro input file (`.STD`) which contains geometry, property, support, and load data and a `.GSP` file which contains member design parameters used in the Chinese Steel Design workflow.
`Chn-2 Analysis of a Frame Under Seismic Loads.STD`	A plane frame is analyzed for seismic loads. The seismic loads are generated using the procedures of the GB50011-2010 code.
`Chn-3 Mass Modelling with Floor Diaphragm.STD`	A steel frame structure is designed per GB 500017-2017 code. This model contains Floor Diaphragm, Mass type reference load and Wind load.

EX. CIS/2 Example Models

The following models are included when STAAD.Pro is installed.

These files are located in the
`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\CIS2\`
folder (default location).

Table 904: CIS/2 examples installed with the program

File name	Description
`fwp_analysis_model.stp`	Analysis model of a three-story steel building with sky light.
`fwp_geom.stp`	Geometry-only model of a three-story steel building with sky light.
`sds2_geom.stp`	A two by one bay braced frame structure with loads.

Related Links

- *I. To import a CIS/2 file* (on page 2078)

EX. Structure Wizard Macro Example Files

The following files are included when STAAD.Pro is installed.

These files are located in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Macro\` folder (default location).

Table 905: Macro examples installed with the program

File name	Description
`stadiumroof.vbs`	A Structure Wizard VBA macro that can be used to parametrically generate a stadium roof model.
`tower.vbs`	A Structure Wizard VBA macro that can be used to parametrically generate a simple tower model with X braced panels.

Note: These macros are loaded by default when **VBA-Macro Models** is selected as the **Model Type** in the **Structure Wizard** applicaiton.

Related Links

- *M. VBA Macro Models* (on page 711)

EX. OpenSTAAD Example Files

The following files are included when STAAD.Pro is installed.

These files are located in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\OpenSTAAD\` folder (default location).

Table 906: OpenSTAAD examples installed with the program

File name	Description
`boxgirder.vbs`	An OpenSTAAD macro used to parametrically generate a concrete box girder outline consisting of plate elements.
`concretebeam.vbs`	An OpenSTAAD macro used to perform the design of a reinforced concrete beam per ACI 318-99 for a selected member in an analyzed STAAD model.
`Rectangle-Beam.xls`	A Microsoft Office Excel spreadsheet which can be used to check the capacity of a rectangular concrete beam with the reinforcement already laid out.
`STAADAndWord.doc`	A Microsoft Office Word document that includes a macro to first load the current analytical model from STAAD.Pro and then load the support reaction values for a selected supported node and load case.
`STDandACAD.dwg`	A sample AutoCAD drawing file capable of writing the member section forces at any distance along a member in the current analytical model in STAAD.Pro.

Related Links

- *OS. To import an existing macro* (on page 7019)
- *OS. Microsoft Excel Macro Example* (on page 7010)
- *OS. Microsoft Word Macro Example* (on page 7014)

EX. Physical Model Examples

The following models are included when STAAD.Pro is installed.

These files are located in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\Physical Models\` folder (default location).

Table 907: Physical Model examples installed with the program

File name	Description
`Area Loads Example.std`	This physical model example showcases how Area loads is applied by creating a panel on the stories. This area load gets assigned to the adjoining members (on the panel's boundary edges) in terms of member distributed loads. Also, the model contains diaphragm defined for each of the floors. The structure is subjected to load categorized as Dead and Live Load groups which are then referenced in the load cases and consequently load combination.

File name	Description
`Cabled stayed Tower Example.std`	This example shows the modeling of a tower with cable stayed and fixed at their end. The self-weight of the cables is ignored by setting the 'Ignore Self Weight' property of the member. After analysis for this reason of ignoring the self-weight, it will show applied self -weight is less than the total weight of all. Also, Member Tension Only attribute is set for these cables. The columns are set with standard Pipe sections whereas the cables are set with circular sections. The structure is subjected Dead and wind load in the form of lateral loads in load groups and referenced in load cases and consequently load combination.
`Concrete Building Example.std`	This physical model example shows a concrete building with steel columns. The concrete slabs are created as single physical surface for each of the floors and for the lift well with openings for the doors and windows. The slabs are subjected to distributed load categorized in the dead and live load case directly as a load item. Also, linear load combination is defined.
`Steel Framed Warehouse Example.std`	This physical model example showcases a typical warehouse with standard steel sections of various shapes. Notice that the truss chords are defined as continuous members. The members are assigned with offsets in start and end. The structure is subjected to dead and live load on the frame structure in the form of self-weight and distributed loads. Also, linear load combination is defined.
`Storehouse with Walls Example.std`	This physical model example showcases a typical storehouse with standard steel sections of various shapes. The walls of the storehouse are created as physical surfaces. The structure is subjected to dead loads on the frame structure in the form of self- weight and point loads. Also, linear load combination is defined.

Note: The file name of the STAAD input file is listed. Other files associated with this model are included with the same base file name in this folder using different file extensions.

EX.Tutorials

The following models are included when STAAD.Pro is installed.

These files are located in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models \Tutorials\` folder (default location).

Table 908: Tutorial files installed with the program

File name	Description
`Tutorial 1 - Steel Portal Frame.std`	A three-member portal frame which utilized steel design to select members.
`Tutorial 2 - RC Frames Structure.std`	A 3D concrete frame that uses concrete design to select reinforcement.

File name	Description
`Tutorial 3 - Analysis of a Slab.std`	A finite element mesh analysis of a a concrete slab supported on two edges.

Related Links

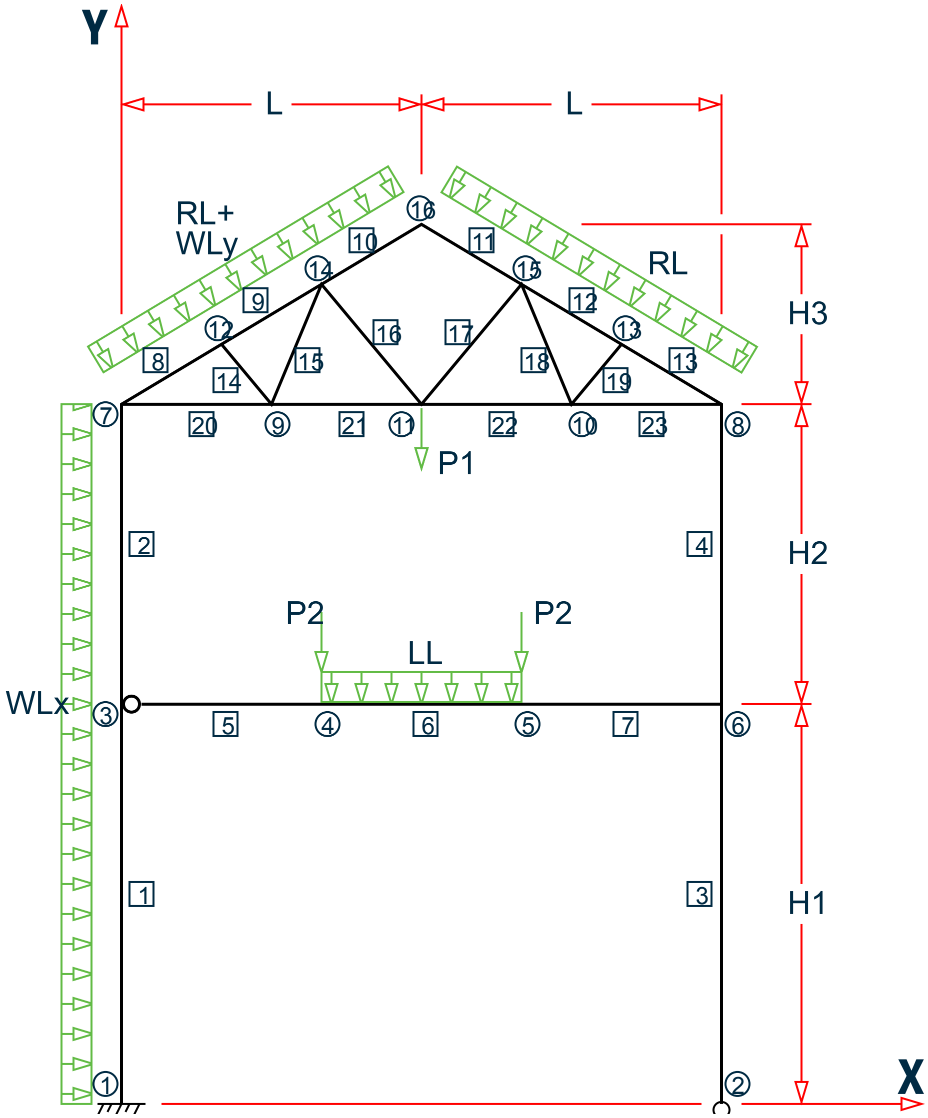
- *T.1 – Steel Portal Frame* (on page 387)
- *T.2 - RC Framed Structure* (on page 463)
- *T.3 - Analysis of a slab* (on page 549)

EX. American Design Examples

EX. US-1 Plane Frame with Steel Design

After one analysis, member selection is requested. Since member sizes change during the member selection, another analysis is done followed by final code checking to verify that the final sizes meet the requirements of the code based on the latest analysis results.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-1 Plane Frame with Steel Design.STD` when you install the program.



where:

L = 15 ft, H1 = 20 ft, H2 = 15 ft, and H3 = 9 ft

WLx = 0.6 k/ft, WLy = 1.0 k/ft, RL = 0.9 k/ft, LL = 1.2 k/ft, P1 = 35 kips, P2 = 15 kips

Members 1, 3, & 4 are a W14x90, Members 5, 6, & 7 are a W21x50, Member 2 is a W10x49, Members 8 through 13 are a W18x35. All other members comprising the truss are a L4x4x1/4

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE EXAMPLE PROBLEM NO. 1
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT FT KIP
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0.  0. ; 2  30  0 ; 3  0  20  0  6  30  20  0
7  0  35 ; 8  30  35 ; 9  7.5  35 ; 10  22.5  35.
11  15  35 ; 12  5.  38. ; 13  25  38
14  10  41 ; 15  20  41 ; 16  15  44
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTY AMERICAN
1  3  4 TABLE ST W14X90 ; 2  TA ST W10X49
5  6  7 TA ST W21X50 ; 8  TO  13  TA ST W18X35
14  TO  23 TA ST L40404
```

Member properties are from the British steel table. The term `ST` stands for standard single section.

```
MEMB TRUSS
14 TO 23
```

The above command defines that members 14 through 23 are of type truss. This means that these members can carry only axial tension/compression and no moments.

```
MEMB RELEASE
5 START MZ
```

Member 5 has local moment-z (`MZ`) released at the start joint. This means that the member cannot carry any moment-z (i.e., strong axis moment) at node 3.

```
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.000283
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

The `DEFINE MATERAL START` initiates input for a material definition, which can later be assigned to all members. STAAD.Pro has several built-in material definitions, including the `ISOTROPIC STEEL` material used here. The length unit is changed from `FEET` to `INCH` to facilitate the input in familiar units.

```
CONSTANTS
BETA 90.0 MEMB 3 4
MATERIAL STEEL ALL
UNIT FT
```

The CONSTANT command initiates input for material constants or material definition references. The BETA command specifies that members 3 and 4 are rotated by 90 degrees around their own longitudinal axis. See G. 4.3 Relationship Between Global and Local Coordinates (on page 2092) for the definition of the beta angle.

```
SUPPORT
1 FIXED ; 2 PINNED
```

A fixed support is located at joint 1 and a pinned support at joint 2.

```
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
```

The above PRINT commands are self-explanatory. The LIST option restricts the print output to the members listed.

```
LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title.

```
SELFWEIGHT Y -1.0
```

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```
JOINT LOAD
4  5  FY  -15. ; 11  FY  -35.
```

Load 1 contains joint loads also. FY indicates that the load is a force in the global Y direction.

```
MEMB LOAD
8  TO  13 UNI Y  -0.9 ; 6  UNI GY  -1.2
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The term UNI stands for uniformly distributed load. Loads are applied on members 6, and 8 to 13.

```
CALCULATE RAYLEIGH FREQUENCY
```

The above command at the end of load case 1, is an instruction to perform a natural frequency calculation based on the Rayleigh method using the data in the above load case.

```
LOADING  2 WIND FROM LEFT
MEMBER LOAD
1  2  UNI  GX  0.6 ; 8  TO  10  UNI Y  -1.
```

Load case 2 is initiated and contains several member loads.

```
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
```

The above command identifies a combination load (case no. 3) with a title. The second line provides the load cases and their respective factors used for the load combination.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
LOAD LIST 1 3
```

The above command activates load cases 1 and 3 only for the commands to follow. This also means that load case 2 will be made inactive.

```
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
```

The above `PRINT` commands are self-explanatory. Also note that all the forces and reactions will be printed for load cases 1 and 3 only.

```
PARAMETER
CODE AISC UNIFIED
NSF  0.85  ALL
BEAM 1.0 ALL
KY  1.2  MEMB  3  4
RATIO  0.9  ALL
PROFILE W14 MEMB  1  3  4
```

The `PARAMETER` command is used to specify steel design parameters such as net section factor (`NSF`), effective length factor for bending about the minor axis (`KY`), etc. Information on these parameters can be obtained from the manual where the implementation of the code is explained. The `BEAM` parameter is specified to perform design at every 1/12th point along the member length, which is also the default. The `RATIO` parameter specifies that the ratio of actual loading over section capacity should not exceed 0.9.

```
SELECT ALL
```

The above command instructs the program to select the most economic section for all the members based on the results of the analysis.

```
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
```

Although the program selects the most economical section for all members, it is not always practical to use many different sizes in one structure. Grouping is a procedure by which the cross section which has the largest value for the specified attribute, which in this case is the default and hence the AREA, from among the associated member list, is assigned to all members in the list. Hence, the cross sections for members 1, 3 and 4 are replaced with the one with the largest area from among the three.

```
PERFORM ANALYSIS
```

As a result of the selection and grouping, the member sizes are no longer the same as the ones used in the original analysis. Hence, it is necessary to reanalyze the structure using the new properties to get new values of forces in the members.

```
PARAMETER
BEAM 1.0 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
```

A new set of values are now provided for the above parameters. The actual load to member capacity `RATIO` has been redefined as 1.0. The `TRACK` parameter tells the program to print out the design results to the intermediate level of descriptively.

```
CHECK CODE ALL
```

With the above command, the latest member sizes with the latest analysis results are checked to verify that they satisfy the `CODE` specifications.

```
STEEL TAKE OFF
```

This command instructs the program to list the length and weight of all the different member sizes.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE EXAMPLE PROBLEM NO. 1
UNIT FT KIP
JOINT COORDINATES
1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
11 15 35 ; 12 5. 38. ; 13 25 38
14 10 41 ; 15 20 41 ; 16 15 44
MEMBER INCIDENCE
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
MEMBER PROPERTY AMERICAN
1 3 4 TA ST W14X90 ; 2 TA ST W10X49
5 6 7 TA ST W21X50 ; 8 TO 13 TA ST W18X35
14 TO 23 TA ST L40404
*MEMB TRUSS
*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT FT
SUPPORT
1 FIXED ; 2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
MEMB LOAD
8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
CALCULATE RAYLEIGH FREQUENCY
LOADING 2  WIND FROM LEFT
```

```
MEMBER LOAD
1 2 UNI GX 0.6 ; 8 TO 10 UNI Y -1.
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
PARAMETER
CODE AISC UNIFIED
NSF 0.85 ALL
BEAM 1.0 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
PROFILE W14 MEMB 1 3 4
SELECT ALL
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
PERFORM ANALYSIS
PARAMETER
BEAM 1.0 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
CHECK CODE ALL
STEEL TAKE OFF
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:22                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE PROBLEM NO. 1
INPUT FILE: US-1 Plane Frame with Steel Design.STD
     2. UNIT FT KIP
     3. JOINT COORDINATES
     4. 1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
     5. 7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
     6. 11 15 35 ; 12 5. 38. ; 13 25 38
     7. 14 10 41 ; 15 20 41 ; 16 15 44
     8. MEMBER INCIDENCE
     9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
    10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
    11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
    12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
    13. 18 10 15 ; 19 10 13 ; 20 7 9
```

```
  14. 21 9 11 ; 22 10 11 ; 23 8 10
  15. MEMBER PROPERTY AMERICAN
  16. 1 3 4 TA ST W14X90 ; 2 TA ST W10X49
  17. 5 6 7 TA ST W21X50 ; 8 TO 13 TA ST W18X35
  18. 14 TO 23 TA ST L40404
  19. *MEMB TRUSS
  20. *14 TO 23
  21. MEMB RELEASE
  22. 5 START MZ
  23. 14 TO 23 START MPY 0.99 MPZ 0.99
  24. 14 TO 23 END MPY 0.99 MPZ 0.99
  25. UNIT INCH
  26. DEFINE MATERIAL START
  27. ISOTROPIC STEEL
  28. E 29000
  29. POISSON 0.3
  30. DENSITY 283E-006
  31. ALPHA 6E-006
  32. DAMP 0.03
  33. TYPE STEEL
  34. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  35. END DEFINE MATERIAL
  36. CONSTANTS
  37. MATERIAL STEEL ALL
  38. BETA 90.0 MEMB 3 4
    EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    2
  39. UNIT FT
  40. SUPPORT
  41. 1 FIXED ; 2 PINNED
  42. PRINT MEMBER INFORMATION LIST 1 5 14
MEMBER   INFORMAT LIST     1
    EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    3
 MEMBER INFORMATION
 ------------------
 MEMBER    START    END         LENGTH     BETA
           JOINT   JOINT        (FEET)     (DEG)     RELEASES
    1        1       3          20.000     0.00
    5        3       4          10.000     0.00    000001000000
   14        9      12           3.905     0.00    000011000011
 ************ END OF DATA FROM INTERNAL STORAGE ************
  43. PRINT MEMBER PROPERTY LIST 1 2 5 8 14
MEMBER   PROPERTY LIST     1
    EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    4
 MEMBER PROPERTIES. UNIT - INCH
 -----------------
 MEMB   PROFILE              AX/          IZ/          IY/        IX/
                             AY           AZ           SZ         SY
   1  ST  W14X90           26.50       999.00       362.00        4.06
                            6.16        13.73       142.71       49.93
   2  ST  W10X49           14.40       272.00        93.40        1.39
                            3.40         7.47        54.40       18.68
   5  ST  W21X50           14.70       984.00        24.90        1.14
                            7.90         4.66        94.62        7.63
   8  ST  W18X35           10.30       510.00        15.30        0.51
                            5.31         3.40        57.63        5.10
  14  ST  L40404            1.93         1.18         4.90        0.04
                            0.67         0.67         0.76        1.73
 ************ END OF DATA FROM INTERNAL STORAGE ************
```

```
  44. LOADING 1 DEAD AND LIVE LOAD
  45. SELFWEIGHT Y -1.0
  46. JOINT LOAD
  47. 4 5 FY -15. ; 11 FY -35.
  48. MEMB LOAD
  49. 8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
  50. CALCULATE RAYLEIGH FREQUENCY
  51. LOADING 2  WIND FROM LEFT
  52. MEMBER LOAD
  53. 1 2 UNI GX 0.6 ; 8 TO 10 UNI Y -1.
  54. * 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
  55. LOAD COMB 3 75 PERCENT DL LL WL
  56. 1 0.75 2 0.75
  57. PERFORM ANALYSIS
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
         Using 64-bit analysis engine.
    EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    5
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      43
 TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
 **********************************************************
 *                                                        *
 * RAYLEIGH FREQUENCY FOR LOADING     1 =    3.13858 CPS  *
 * MAX DEFLECTION =  1.21777 INCH GLO X,  AT JOINT     7  *
 *                                                        *
 **********************************************************
  58. LOAD LIST 1 3
  59. PRINT MEMBER FORCES
MEMBER   FORCES
    EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    6
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1     54.05     -2.00     0.00      0.00      0.00     -61.74
               3    -52.25      2.00     0.00      0.00      0.00      21.74
         3     1     40.71     18.99     0.00      0.00      0.00     247.88
               3    -39.36     -9.99     0.00      0.00      0.00      41.97
    2    1     3     33.81     -5.48     0.00      0.00      0.00     -21.74
               7    -33.07      5.48     0.00      0.00      0.00     -60.47
         3     3     28.90     -0.16     0.00      0.00      0.00     -41.97
               7    -28.35      6.91     0.00      0.00      0.00     -11.09
    3    1     2     58.79      0.00    -2.00      0.00     -0.00       0.00
               6    -56.99      0.00     2.00      0.00     40.00       0.00
         3     2     55.17      0.00    -3.51      0.00      0.00       0.00
               6    -53.82      0.00     3.51      0.00     70.15       0.00
    4    1     6     31.94      0.00    -5.48      0.00     59.02       0.00
               8    -30.59      0.00     5.48      0.00     23.19       0.00
         3     6     31.66      0.00   -13.66      0.00    105.29       0.00
               8    -30.65      0.00    13.66      0.00     99.65       0.00
    5    1     3     -3.48     18.45     0.00      0.00      0.00       0.00
               4      3.48    -17.95     0.00      0.00      0.00     181.98
         3     3    -10.16     10.46     0.00      0.00      0.00       0.00
               4     10.16    -10.09     0.00      0.00      0.00     102.77
```

```
    6    1     4     -3.48       2.95      0.00       0.00       0.00    -181.98
               5      3.48       9.55      0.00       0.00       0.00     148.97
         3     4    -10.16      -1.16      0.00       0.00       0.00    -102.77
               5     10.16      10.53      0.00       0.00       0.00      44.29
    7    1     5     -3.48     -24.55      0.00       0.00       0.00    -148.97
               6      3.48      25.05      0.00       0.00       0.00     -99.03
         3     5    -10.16     -21.78      0.00       0.00       0.00     -44.29
               6     10.16      22.16      0.00       0.00       0.00    -175.43
    8    1     7     36.53      16.62      0.00       0.00       0.00      60.47
              12    -36.43     -11.20      0.00       0.00       0.00      20.64
         3     7     37.60      10.47      0.00       0.00       0.00      11.09
              12    -37.52      -2.03      0.00       0.00       0.00      25.36
    9    1    12     36.77       8.94      0.00       0.00       0.00     -20.63
              14    -36.66      -3.52      0.00       0.00       0.00      56.94
    EXAMPLE PROBLEM NO. 1                                      -- PAGE NO.    7
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
         3    12     36.66       7.58      0.00       0.00       0.00     -25.36
              14    -36.58       0.87      0.00       0.00       0.00      44.93
   10    1    14     41.85     -19.60      0.00       0.00       0.00     -56.95
              16    -41.74      25.02      0.00       0.00       0.00     -73.14
         3    14     34.32     -13.87      0.00       0.00       0.00     -44.93
              16    -34.24      22.31      0.00       0.00       0.00     -60.54
   11    1    15     41.83     -19.63      0.00       0.00       0.00     -57.16
              16    -41.72      25.06      0.00       0.00       0.00     -73.14
         3    15     35.88     -15.65      0.00       0.00       0.00     -42.57
              16    -35.80      19.72      0.00       0.00       0.00     -60.54
   12    1    13     40.08       7.85      0.00       0.00       0.00     -27.16
              15    -39.97      -2.43      0.00       0.00       0.00      57.15
         3    13     27.72       8.78      0.00       0.00       0.00      -3.23
              15    -27.64      -4.71      0.00       0.00       0.00      42.56
   13    1     8     40.49      11.35      0.00       0.00       0.00      23.19
              13    -40.39      -5.92      0.00       0.00       0.00      27.16
         3     8     26.74      19.68      0.00       0.00       0.00      99.65
              13    -26.66     -15.61      0.00       0.00       0.00       3.23
   14    1     9     -2.27       0.01      0.00       0.00       0.00       0.00
              12      2.29       0.01      0.00       0.00       0.00       0.00
         3     9      5.63       0.01      0.00       0.00       0.00       0.00
              12     -5.61       0.01      0.00       0.00       0.00       0.00
   15    1     9      1.82       0.01      0.00       0.00       0.00      -0.00
              14     -1.78       0.01      0.00       0.00       0.00       0.00
         3     9     -4.73       0.01      0.00       0.00       0.00      -0.00
              14      4.76       0.01      0.00       0.00       0.00       0.00
   16    1    11    -24.42       0.02      0.00       0.00       0.00      -0.00
              14     24.45       0.02      0.00       0.00       0.00      -0.00
         3    11    -10.26       0.01      0.00       0.00       0.00      -0.00
              14     10.29       0.01      0.00       0.00       0.00      -0.00
   17    1    11    -21.23       0.02      0.00       0.00       0.00      -0.00
              15     21.27       0.02      0.00       0.00       0.00      -0.00
         3    11    -23.97       0.01      0.00       0.00       0.00      -0.00
              15     24.00       0.01      0.00       0.00       0.00      -0.00
   18    1    10     -1.71       0.01      0.00       0.00       0.00      -0.00
              15      1.75       0.01      0.00       0.00       0.00       0.00
    EXAMPLE PROBLEM NO. 1                                      -- PAGE NO.    8
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
```

```
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          3    10      5.69      0.01     0.00      0.00      0.00      -0.00
               15     -5.66      0.01     0.00      0.00      0.00       0.00
    19    1    10      1.97      0.01     0.00      0.00      0.00       0.00
               13     -1.95      0.01     0.00      0.00      0.00       0.00
          3    10     -6.90      0.01     0.00      0.00      0.00       0.00
               13      6.91      0.01     0.00      0.00      0.00       0.00
    20    1     7    -17.29      0.02     0.00      0.00      0.00       0.00
                9     17.29      0.02     0.00      0.00      0.00       0.00
          3     7    -19.94      0.02     0.00      0.00      0.00      -0.00
                9     19.94      0.02     0.00      0.00      0.00       0.00
    21    1     9    -19.44      0.02     0.00      0.00      0.00       0.00
               11     19.44      0.02     0.00      0.00      0.00       0.00
          3     9    -14.52      0.02     0.00      0.00      0.00       0.00
               11     14.52      0.02     0.00      0.00      0.00       0.00
    22    1    10    -21.48      0.02     0.00      0.00      0.00       0.00
               11     21.48      0.02     0.00      0.00      0.00       0.00
          3    10     -5.75      0.02     0.00      0.00      0.00       0.00
               11      5.75      0.02     0.00      0.00      0.00       0.00
    23    1     8    -23.40      0.02     0.00      0.00      0.00       0.00
               10     23.40      0.02     0.00      0.00      0.00       0.00
          3     8      0.86      0.02     0.00      0.00      0.00       0.00
               10     -0.86      0.02     0.00      0.00      0.00      -0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   60. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    9
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      2.00     54.05      0.00      0.00      0.00    -61.74
          3    -18.99     40.71      0.00      0.00      0.00    247.88
     2    1     -2.00     58.79      0.00      0.00      0.00      0.00
          3     -3.51     55.17      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   61. PARAMETER
   62. CODE AISC UNIFIED
   63. NSF 0.85 ALL
   64. BEAM 1.0 ALL
   65. KY 1.2 MEMB 3 4
   66. RATIO 0.9 ALL
   67. PROFILE W14 MEMB 1 3 4
   68. SELECT ALL
PARAMETER 1
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   10
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
========================
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.
  AISC Spec.      STAAD.Pro      Description
 ------------    -----------     -------------
```

```
      X/U             Z          Axis typically parallel to the sections principal
major axis.
      Y/V             Y          Axis typically parallel to the sections principal
minor axis.
       Z              X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz             FX          Axial force.
       Vy             FY          Shear force along minor axis.
       Vx             FZ          Shear force along major axis.
       Tz             MX          Torsional moment.
       My             MY          Bending moment about minor axis.
       Mx             MZ          Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), while
       the results for shear are reported as being for shear forces �along� the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   11
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------|
|  Member No:        1       Profile:  ST  W14X74              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.827       Loadcase:        3    |
|  Location:      0.00       Ref:      Cl.F2.2                                |
|  Pz:       40.71     C     Vy:        18.99           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:      247.9        |
|------------------------------------------------------------------------------|
 - Member :     2
|------------------------------------------------------------------------------|
|  Member No:        2       Profile:  ST  W12X30              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.810       Loadcase:        1    |
|  Location:     15.00       Ref:      Eq.H1-3b                               |
|  Pz:       33.07     C     Vy:       -5.481           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:      60.47        |
|------------------------------------------------------------------------------|
 - Member :     3
|------------------------------------------------------------------------------|
|  Member No:        3       Profile:  ST  W14X61              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.888       Loadcase:        3    |
```

```
|  Location:     20.00        Ref:      Eq.H1-1b                                 |
|  Pz:       53.82     C     Vy:        0.000          Vx:     -3.507            |
|  Tz:       0.000           My:       -70.15          Mx:      0.000            |
|--------------------------------------------------------------------------------|
- Member :     4
|--------------------------------------------------------------------------------|
|  Member No:        4       Profile:  ST  W14X90              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.538       Loadcase:        3      |
|  Location:       0.00      Ref:      Eq.H1-1b                                  |
|  Pz:       31.66     C     Vy:        0.000          Vx:     -13.66            |
|  Tz:       0.000           My:        105.3          Mx:      0.000            |
|--------------------------------------------------------------------------------|
- Member :     5
|--------------------------------------------------------------------------------|
|  Member No:        5       Profile:  ST  W21X44              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.864       Loadcase:        1      |
|  Location:     10.00       Ref:      Eq.H1-1b                                  |
|  Pz:       3.481     T     Vy:        17.95          Vx:      0.000            |
|  Tz:       0.000           My:        0.000          Mx:     -182.0            |
|--------------------------------------------------------------------------------|
- Member :     6
|--------------------------------------------------------------------------------|
|  Member No:        6       Profile:  ST  W21X44              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.880       Loadcase:        1      |
|  Location:       2.50      Ref:      Eq.H1-1b                                  |
|  Pz:       3.481     T     Vy:       -.1761          Vx:      0.000            |
|  Tz:       0.000           My:        0.000          Mx:     -185.4            |
|--------------------------------------------------------------------------------|
- Member :     7
|--------------------------------------------------------------------------------|
|  Member No:        7       Profile:  ST  W21X44              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.834       Loadcase:        3      |
|  Location:     10.00       Ref:      Eq.H1-1b                                  |
|  Pz:       10.16     T     Vy:       -22.16          Vx:      0.000            |
|  Tz:       0.000           My:        0.000          Mx:      175.4            |
|--------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                        -- PAGE NO.   12
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     8
|--------------------------------------------------------------------------------|
|  Member No:        8       Profile:  ST  W14X22              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.842       Loadcase:        1      |
|  Location:       0.00      Ref:      Eq.H1-3b                                  |
|  Pz:       36.53     C     Vy:        16.62          Vx:      0.000            |
|  Tz:       0.000           My:        0.000          Mx:      60.47            |
|--------------------------------------------------------------------------------|
- Member :     9
|--------------------------------------------------------------------------------|
|  Member No:        9       Profile:  ST  W14X22              (AISC SECTIONS)|
|  Status:         PASS      Ratio:          0.784       Loadcase:        1      |
|  Location:       5.83      Ref:      Eq.H1-3b                                  |
|  Pz:       36.66     C     Vy:        3.515          Vx:      0.000            |
|  Tz:       0.000           My:        0.000          Mx:     -56.94            |
|--------------------------------------------------------------------------------|
- Member :    10
|--------------------------------------------------------------------------------|
```

```
|  Member No:       10       Profile:  ST  W12X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.815      Loadcase:        1     |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       41.74     C     Vy:       -25.02          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      73.14         |
|-----------------------------------------------------------------------------|
 - Member :    11
|-----------------------------------------------------------------------------|
|  Member No:       11       Profile:  ST  W12X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.815      Loadcase:        1     |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       41.72     C     Vy:       -25.06          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      73.14         |
|-----------------------------------------------------------------------------|
 - Member :    12
|-----------------------------------------------------------------------------|
|  Member No:       12       Profile:  ST  W14X22              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.813      Loadcase:        1     |
|  Location:      5.83       Ref:      Eq.H1-3b                               |
|  Pz:       39.97     C     Vy:        2.432          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:     -57.15         |
|-----------------------------------------------------------------------------|
 - Member :    13
|-----------------------------------------------------------------------------|
|  Member No:       13       Profile:  ST  W14X30              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.828      Loadcase:        3     |
|  Location:      0.00       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       26.74     C     Vy:        19.68          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      99.65         |
|-----------------------------------------------------------------------------|
 - Member :    14
|-----------------------------------------------------------------------------|
|  Member No:       14       Profile:  ST  L20202              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.768      Loadcase:        3     |
|  Location:      1.95       Ref:      Eq.H2-1                                |
|  Pz:       5.618     C     Vy:       -.2092E-03      Vx:     -.2092E-03     |
|  Tz:       0.000           My:       -.4549E-02      Mx:     0.4549E-02     |
|-----------------------------------------------------------------------------|
      EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   13
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :    15
|-----------------------------------------------------------------------------|
|  Member No:       15       Profile:  ST  L25253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.249      Loadcase:        1     |
|  Location:      3.25       Ref:      Eq.H2-1                                |
|  Pz:       1.804     C     Vy:       -.1251E-03      Vx:     -.1251E-03     |
|  Tz:       0.000           My:       -.9994E-02      Mx:     0.9994E-02     |
|-----------------------------------------------------------------------------|
 - Member :    16
|-----------------------------------------------------------------------------|
|  Member No:       16       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.823      Loadcase:        1     |
|  Location:      3.91       Ref:      Eq.H2-1                                |
|  Pz:       24.43     T     Vy:       0.2114E-03      Vx:     0.2114E-03     |
|  Tz:       0.000           My:       -.2237E-01      Mx:     0.2237E-01     |
|-----------------------------------------------------------------------------|
```

```
 - Member :    17
|--------------------------------------------------------------------------------|
|  Member No:       17       Profile:  ST  L20204              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.877       Loadcase:        3     |
|  Location:      3.91       Ref:      Eq.H2-1                                 |
|  Pz:       23.99     T     Vy:       0.1653E-03       Vx:     0.1653E-03     |
|  Tz:       0.000           My:       -.1677E-01       Mx:     0.1677E-01     |
|--------------------------------------------------------------------------------|
 - Member :    18
|--------------------------------------------------------------------------------|
|  Member No:       18       Profile:  ST  L25253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.743       Loadcase:        3     |
|  Location:      3.25       Ref:      Eq.H2-1                                 |
|  Pz:       5.680     C     Vy:       -.7942E-04       Vx:     -.7942E-04     |
|  Tz:       0.000           My:       -.7397E-02       Mx:     0.7397E-02     |
|--------------------------------------------------------------------------------|
 - Member :    19
|--------------------------------------------------------------------------------|
|  Member No:       19       Profile:  ST  L20202              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.804       Loadcase:        3     |
|  Location:      2.28       Ref:      Eq.H2-1                                 |
|  Pz:       6.907     T     Vy:       0.4779E-03       Vx:     0.4779E-03     |
|  Tz:       0.000           My:       -.4374E-02       Mx:     0.4374E-02     |
|--------------------------------------------------------------------------------|
 - Member :    20
|--------------------------------------------------------------------------------|
|  Member No:       20       Profile:  ST  L25253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.790       Loadcase:        3     |
|  Location:      3.75       Ref:      Eq.H2-1                                 |
|  Pz:       19.94     T     Vy:       -.2987E-04       Vx:     -.2987E-04     |
|  Tz:       0.000           My:       -.2461E-01       Mx:     0.2461E-01     |
|--------------------------------------------------------------------------------|
 - Member :    21
|--------------------------------------------------------------------------------|
|  Member No:       21       Profile:  ST  L25253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.785       Loadcase:        1     |
|  Location:      3.75       Ref:      Eq.H2-1                                 |
|  Pz:       19.44     T     Vy:       -.1450E-03       Vx:     -.1450E-03     |
|  Tz:       0.000           My:       -.3268E-01       Mx:     0.3268E-01     |
|--------------------------------------------------------------------------------|
      EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   14
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :    22
|--------------------------------------------------------------------------------|
|  Member No:       22       Profile:  ST  L25253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.861       Loadcase:        1     |
|  Location:      3.75       Ref:      Eq.H2-1                                 |
|  Pz:       21.48     T     Vy:       -.1182E-03       Vx:     -.1182E-03     |
|  Tz:       0.000           My:       -.3275E-01       Mx:     0.3275E-01     |
|--------------------------------------------------------------------------------|
 - Member :    23
|--------------------------------------------------------------------------------|
|  Member No:       23       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.805       Loadcase:        1     |
|  Location:      3.75       Ref:      Eq.H2-1                                 |
|  Pz:       23.40     T     Vy:       -.6927E-04       Vx:     -.6927E-04     |
```

```
|  Tz:       0.000           My:       -.3254E-01       Mx:     0.3254E-01    |
|-----------------------------------------------------------------------------|
    69. GROUP MEMB 1 3 4
   GROUPING BASED ON MEMBER      4  (ST  W14X90          ) LIST=      1....
    70. GROUP MEMB 5 6 7
   GROUPING BASED ON MEMBER      7  (ST  W21X44          ) LIST=      5....
    71. GROUP MEMB 8 TO 13
   GROUPING BASED ON MEMBER     13  (ST  W14X30          ) LIST=      8....
    72. GROUP MEMB 14 TO 23
   GROUPING BASED ON MEMBER     23  (ST  L30253          ) LIST=     14....
    73. PERFORM ANALYSIS
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
   **********************************************************
   *                                                        *
   * RAYLEIGH FREQUENCY FOR LOADING     1 =    2.92702 CPS  *
   * MAX DEFLECTION =  1.49837 INCH GLO X,  AT JOINT     7  *
   *                                                        *
   **********************************************************
    74. PARAMETER
    75. BEAM 1.0 ALL
    76. RATIO 1.0 ALL
    77. TRACK 1.0 ALL
    78. CHECK CODE ALL
 PARAMETER 2
      EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   15
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
      X/U             Z          Axis typically parallel to the sections principal
major axis.
      Y/V             Y          Axis typically parallel to the sections principal
minor axis.
       Z              X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz             FX         Axial force.
       Vy             FY         Shear force along minor axis.
       Vx             FZ         Shear force along major axis.
       Tz             MX         Torsional moment.
       My             MY         Bending moment about minor axis.
       Mx             MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
```

```
   3. Bending results are reported as being �about� the relevant axis (X/Y), while
      the results for shear are reported as being for shear forces �along� the axis.
      E.g : Mx indicates bending about the X axis, while Vx indicates shear along
      the X axis.
*** ABBREVIATIONS ***:
======================
   F-T-B = Flexural-Torsional Buckling
   L-T-B = Lateral-Torsional Buckling
   F-L-B = Flange Local Buckling
   W-L-B = Web Local Buckling
   L-L-B = Leg Local Buckling
   C-F-Y = Compression Flange Yielding
   T-F-Y = Tension Flange Yielding
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   16
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     1
|-----------------------------------------------------------------------------|
|  Member No:       1       Profile:  ST  W14X90              (AISC SECTIONS)|
|  Status:        PASS      Ratio:         0.622       Loadcase:        3    |
|  Location:      0.00      Ref:      Cl.F2.2                                |
|  Pz:       40.44     C    Vy:        19.06            Vx:      0.000       |
|  Tz:       0.000          My:        0.000            Mx:      252.5       |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     64.935                                    |
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.622(PASS)          |
|          Loc  :    0.00            Condition :    Cl.F2.2                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )                  |
| Ag   :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00             |
| Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00              |
| Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02              |
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01              |
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:        20.000                                         |
| Design Parameters                                             (Rolled)      |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: NonSlender      10.21       N/A      15.89     Table.4.1a.Case1    |
| Web   : NonSlender      25.55       N/A      42.29     Table.4.1a.Case5    |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: Compact         10.21      10.79     28.38     Table.4.1b.Case10   |
| Web   : Compact         25.55     106.72    161.78     Table.4.1b.Case15   |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   17
```

```
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                                  - Member :     1 Contd.
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      858.6       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      858.6       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    53.20      792.2       0.067     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    53.20      687.7       0.077     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  53.20      728.1       0.073     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      400.3       0.000     Cl.G1           1    0.00 |
| SHEAR Y          19.06      133.1       0.143     Cl.G1           3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X     252.5      423.9       0.596     Cl.F2.1         3    0.00 |
| FLEX. YLD. Y     0.000      204.1       0.000     Cl.F6.1         1    0.00 |
| L-T-B ABT X      252.5      406.2       0.622     Cl.F2.2         3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.651     Eq.H1-1b            3    0.00                  |
| CL.H13-IP          0.621     Eq.H1-3a(H1-1b)     3    0.00                  |
| CL.H13-OP          0.473     Eq.H1-3b            3    0.00                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   18
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     2
|-----------------------------------------------------------------------------|
|  Member No:        2       Profile:  ST  W12X30              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.928       Loadcase:        1    |
|  Location:     15.00       Ref:      Eq.H1-1a                               |
|  Pz:       32.83     C     Vy:       -6.421          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      67.78         |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :    118.446                                    |
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00         |
```

```
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1            Ratio     :        0.928(PASS)            |
|          Loc  :   15.00           Condition :    Eq.H1-1a                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:    15.00, PROPERTIES UNIT: IN  )                  |
| Ag  :   8.790E+00     Axx :   5.738E+00     Ayy :   3.198E+00              |
| Ixx :   2.380E+02     Iyy :   2.030E+01     J   :   4.570E-01              |
| Sxx+:   3.870E+01     Sxx-:   3.870E+01     Zxx :   4.310E+01              |
| Syy+:   6.227E+00     Syy-:   6.227E+00     Zyy :   9.560E+00              |
| Cw  :   7.147E+02     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:        15.000                                         |
| Design Parameters                                           (Rolled)        |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00        |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE              |
| Flange: NonSlender       7.41       N/A      15.89     Table.4.1a.Case1    |
| Web   : NonSlender      41.62       N/A      42.29     Table.4.1a.Case5    |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE              |
| Flange: Compact          7.41      10.79     28.38     Table.4.1b.Case10   |
| Web   : Compact         41.62     106.72    161.78     Table.4.1b.Case15   |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   19
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     2 Contd.
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      284.8      0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      284.8      0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    33.27      267.4      0.124     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    33.27      136.1      0.245     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  33.27      202.5      0.164     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      111.5      0.000     Cl.G1           1    0.00 |
| SHEAR Y          7.155      69.08      0.104     Cl.G1           3   15.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
```

```
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X     67.78      116.4       0.582     Cl.F2.1         1   15.00 |
| FLEX. YLD. Y     0.000      25.81       0.000     Cl.F6.1         1    0.00 |
| L-T-B ABT X      67.78      87.73       0.773     Cl.F2.2         1   15.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.928     Eq.H1-1a            1   15.00                  |
| CL.H13-IP          0.644     Eq.H1-3a(H1-1b)     1   15.00                  |
| CL.H13-OP          0.930     Eq.H1-3b            1   15.00                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   20
                               STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                             *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     3
|-----------------------------------------------------------------------------|
|  Member No:        3       Profile:  ST  W14X90              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.380       Loadcase:        3    |
|  Location:     20.00       Ref:      Eq.H1-1b                               |
|  Pz:       53.46     C     Vy:         0.000          Vx:     -3.441        |
|  Tz:       0.000           My:       -68.83           Mx:      0.000        |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     77.922                                    |
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00          |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.380(PASS)           |
|          Loc  :   20.00           Condition :    Eq.H1-1b                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:    20.00, PROPERTIES UNIT: IN  )                  |
| Ag  :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00              |
| Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00              |
| Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02              |
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01              |
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:        20.000                                         |
| Design Parameters                                               (Rolled)    |
| Kx:    1.00  Ky:    1.20  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE                 |
| Flange: NonSlender      10.21       N/A      15.89     Table.4.1a.Case1     |
| Web   : NonSlender      26.18       N/A      42.29     Table.4.1a.Case5     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE                 |
| Flange: Compact         10.21      10.79     28.38     Table.4.1b.Case13    |
| Web   : Compact         26.18     ******    ******     Table.4.1.           |
```

```
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   21
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     3 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      858.6       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      858.6       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    58.80      792.2       0.074     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    58.80      623.7       0.094     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  58.80      728.1       0.081     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          3.441      400.3       0.009     Cl.G1           3    0.00 |
| SHEAR Y          0.000      133.1       0.000     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X     0.000      423.9       0.000     Cl.F2.1         1    0.00 |
| FLEX. YLD. Y     68.83      204.1       0.337     Cl.F6.1         3   20.00 |
| L-T-B ABT X      0.000      406.2       0.000     Cl.F2.2         1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.380     Eq.H1-1b            3   20.00                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   22
                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
               *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     4
|-----------------------------------------------------------------------------|
|  Member No:        4       Profile:  ST  W14X90             (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.563       Loadcase:        3     |
|  Location:      0.00       Ref:       Eq.H1-1b                              |
|  Pz:       31.24     C     Vy:         0.000         Vx:     -13.91         |
|  Tz:       0.000           My:         110.4         Mx:      0.000         |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     58.442                                    |
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00          |
```

```
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.563(PASS)           |
|          Loc  :    0.00            Condition :    Eq.H1-1b                    |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )                  |
| Ag  :   2.650E+01     Axx :   2.059E+01     Ayy :   6.160E+00               |
| Ixx :   9.990E+02     Iyy :   3.620E+02     J   :   4.060E+00               |
| Sxx+:   1.427E+02     Sxx-:   1.427E+02     Zxx :   1.570E+02               |
| Syy+:   4.993E+01     Syy-:   4.993E+01     Zyy :   7.560E+01               |
| Cw  :   1.593E+04     x0  :   0.000E+00     y0  :   0.000E+00               |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:        15.000                                         |
| Design Parameters                                            (Rolled)       |
| Kx:    1.00  Ky:    1.20  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: NonSlender      10.21       N/A      15.89     Table.4.1a.Case1     |
| Web   : NonSlender      25.86       N/A      42.29     Table.4.1a.Case5     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: Compact         10.21      10.79     28.38     Table.4.1b.Case13    |
| Web   : Compact         25.86     ******    ******     Table.4.1.           |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   23
                          STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                        *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     4 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                     DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC  |
|                                                                             |
| TEN. YLD.        0.000      858.6       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      858.6       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                     DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC  |
|                                                                             |
| FLEX. BUCK. X    31.80      820.6       0.039     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    31.80      717.3       0.044     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  31.80      769.0       0.041     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                     DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC  |
|                                                                             |
| SHEAR X          13.91      400.3       0.035     Cl.G1           3    0.00 |
| SHEAR Y          0.000      133.1       0.000     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
```

```
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. YLD. X     0.000      423.9      0.000     Cl.F2.1         1    0.00 |
| FLEX. YLD. Y    -110.4      204.1      0.541     Cl.F6.1         3    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                   |
| CLAUSE H1        0.563     Eq.H1-1b            3    0.00                   |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   24
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     5
|-----------------------------------------------------------------------------|
|  Member No:        5       Profile:  ST  W21X44              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.850       Loadcase:        1    |
|  Location:     10.00       Ref:      Eq.H1-1b                               |
|  Pz:       4.327     T     Vy:        17.68           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:     -179.1        |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :     95.097                                    |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1             Ratio     :        0.850(PASS)          |
|          Loc  :   10.00            Condition :    Eq.H1-1b                  |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:    10.00, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.300E+01     Axx :   5.850E+00     Ayy :   7.245E+00              |
| Ixx :   8.430E+02     Iyy :   2.070E+01     J   :   7.700E-01              |
| Sxx+:   8.145E+01     Sxx-:   8.145E+01     Zxx :   9.540E+01              |
| Syy+:   6.369E+00     Syy-:   6.369E+00     Zyy :   1.020E+01              |
| Cw  :   2.112E+03     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:        10.000                                         |
| Design Parameters                                            (Rolled)      |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   120.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: NonSlender       7.22       N/A      15.89     Table.4.1a.Case1     |
| Web   : Slender         53.71       N/A      42.29     Table.4.1a.Case5     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   120.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: Compact          7.22      10.79     28.38     Table.4.1b.Case10    |
| Web   : Compact         53.71     106.72    161.78     Table.4.1b.Case15    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   25
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
```

```
                    ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                          - Member :     5 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        10.46      421.2       0.025     Cl.D2           3    0.00 |
| TEN. RUPT.       10.46      421.2       0.025     Cl.D2           3    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    0.000      383.2       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    0.000      261.6       0.000     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  0.000      313.1       0.000     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      113.7       0.000     Cl.G1           1    0.00 |
| SHEAR Y          18.13      156.5       0.116     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X    -179.1      257.6       0.695     Cl.F2.1         1   10.00 |
| FLEX. YLD. Y     0.000      27.52       0.000     Cl.F6.1         1    0.00 |
| L-T-B ABT X     -179.1      210.8       0.849     Cl.F2.2         1   10.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.850     Eq.H1-1b            1   10.00                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                 -- PAGE NO.   26
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     6
|-----------------------------------------------------------------------------|
|  Member No:        6       Profile:  ST  W21X44             (AISC SECTIONS)|
|  Status:       PASS        Ratio:        0.863       Loadcase:        1     |
|  Location:     2.50        Ref:      Eq.H1-1b                               |
|  Pz:      4.327     T     Vy:       -.4256           Vx:      0.000         |
|  Tz:      0.000           My:        0.000           Mx:     -181.9         |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :     95.097                                    |
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1             Ratio     :        0.863(PASS)          |
```

```
|          Loc  :    2.50           Condition :    Eq.H1-1b                      |
|-------------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     2.50, PROPERTIES UNIT: IN  )                    |
| Ag  :   1.300E+01     Axx :   5.850E+00     Ayy :   7.245E+00                 |
| Ixx :   8.430E+02     Iyy :   2.070E+01     J   :   7.700E-01                 |
| Sxx+:   8.145E+01     Sxx-:   8.145E+01     Zxx :   9.540E+01                 |
| Syy+:   6.369E+00     Syy-:   6.369E+00     Zyy :   1.020E+01                 |
| Cw  :   2.112E+03     x0  :   0.000E+00     y0  :   0.000E+00                 |
|-------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                           |
| Fyld:        5184.000             Fu:        8351.999                         |
|-------------------------------------------------------------------------------|
| Actual Member Length:        10.000                                           |
| Design Parameters                                              (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00            |
|-------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   120.00)                        |
|                            λ         λp        λr       CASE                 |
| Flange: NonSlender       7.22       N/A      15.89     Table.4.1a.Case1       |
| Web   : Slender         53.71       N/A      42.29     Table.4.1a.Case5       |
|                                                                               |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   120.00)                        |
|                            λ         λp        λr       CASE                 |
| Flange: Compact          7.22      10.79     28.38     Table.4.1b.Case10      |
| Web   : Compact         53.71     106.72    161.78     Table.4.1b.Case15      |
|-------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   27
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     6 Contd.
|-------------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL TENSION                             |
|-------------------------------------------------------------------------------|
|                  DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC    |
|                                                                               |
| TEN. YLD.        10.46      421.2       0.025     Cl.D2           3    0.00   |
| TEN. RUPT.       10.46      421.2       0.025     Cl.D2           3    0.00   |
|-------------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL COMPRESSION                         |
|-------------------------------------------------------------------------------|
|                  DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC    |
|                                                                               |
| FLEX. BUCK. X    0.000      383.2       0.000     Cl.E3           1    0.00   |
| FLEX. BUCK. Y    0.000      261.6       0.000     Cl.E3           1    0.00   |
| FLEX. TOR. BUCK  0.000      313.1       0.000     Cl.E4           1    0.00   |
|-------------------------------------------------------------------------------|
|                          CHECKS FOR SHEAR                                     |
|-------------------------------------------------------------------------------|
|                  DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC    |
|                                                                               |
| SHEAR X          0.000      113.7       0.000     Cl.G1           1    0.00   |
| SHEAR Y          10.64      156.5       0.068     Cl.G1           3   10.00   |
|-------------------------------------------------------------------------------|
|                          CHECKS FOR BENDING                                   |
|-------------------------------------------------------------------------------|
|                  DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC    |
|                                                                               |
```

```
| FLEX. YLD. X    -181.9      257.6       0.706     Cl.F2.1         1    2.50 |
| FLEX. YLD. Y     0.000      27.52       0.000     Cl.F6.1         1    0.00 |
| L-T-B ABT X     -181.9      210.8       0.863     Cl.F2.2         1    2.50 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.863     Eq.H1-1b            1    2.50                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                 -- PAGE NO.   28
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     7
|-----------------------------------------------------------------------------|
|  Member No:        7       Profile:  ST  W21X44             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.852       Loadcase:        3    |
|  Location:     10.00       Ref:      Eq.H1-1b                               |
|  Pz:       10.46     T     Vy:       -22.22           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:      179.2        |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :     95.097                                    |
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00          |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.852(PASS)           |
|          Loc  :   10.00           Condition :    Eq.H1-1b                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:    10.00, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.300E+01     Axx :   5.850E+00     Ayy :   7.245E+00              |
| Ixx :   8.430E+02     Iyy :   2.070E+01     J   :   7.700E-01              |
| Sxx+:   8.145E+01     Sxx-:   8.145E+01     Zxx :   9.540E+01              |
| Syy+:   6.369E+00     Syy-:   6.369E+00     Zyy :   1.020E+01              |
| Cw  :   2.112E+03     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:        10.000                                         |
| Design Parameters                                              (Rolled)     |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:   120.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: NonSlender       7.22       N/A      15.89     Table.4.1a.Case1     |
| Web   : Slender         53.71       N/A      42.29     Table.4.1a.Case5     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:   120.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: Compact          7.22      10.79     28.38     Table.4.1b.Case10    |
| Web   : Compact         53.71     106.72    161.78     Table.4.1b.Case15    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                 -- PAGE NO.   29
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
```

```
                                                   - Member :     7 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| TEN. YLD.        10.46      421.2       0.025     Cl.D2           3    0.00 |
| TEN. RUPT.       10.46      421.2       0.025     Cl.D2           3    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. BUCK. X    0.000      383.2       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    0.000      261.6       0.000     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  0.000      313.1       0.000     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| SHEAR X          0.000      113.7       0.000     Cl.G1           1    0.00 |
| SHEAR Y          25.20      156.5       0.161     Cl.G1           1   10.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. YLD. X     179.2      257.6       0.696     Cl.F2.1         3   10.00 |
| FLEX. YLD. Y     0.000      27.52       0.000     Cl.F6.1         1    0.00 |
| L-T-B ABT X      179.2      210.8       0.850     Cl.F2.2         3   10.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1        0.852     Eq.H1-1b            3   10.00                   |
|-----------------------------------------------------------------------------|
      EXAMPLE PROBLEM NO. 1                                -- PAGE NO.   30
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     8
|-----------------------------------------------------------------------------|
|  Member No:        8       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.593      Loadcase:        1     |
|  Location:      0.00       Ref:      Eq.H1-3a(H1-1b)                       |
|  Pz:       34.94     C     Vy:        17.30           Vx:      0.000       |
|  Tz:       0.000           My:        0.000           Mx:      67.77       |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     47.018                                    |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1            Ratio     :        0.593(PASS)           |
|          Loc  :    0.00           Condition :    Eq.H1-3a(H1-1b            |
|-----------------------------------------------------------------------------|
```

```
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )                    |
| Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00                 |
| Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01                 |
| Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01                 |
| Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00                 |
| Cw  :   8.800E+02     x0  :   0.000E+00     y0  :   0.000E+00                 |
|-------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                           |
| Fyld:        5184.000             Fu:        8351.999                         |
|-------------------------------------------------------------------------------|
| Actual Member Length:         5.831                                           |
| Design Parameters                                             (Rolled)        |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00            |
|-------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:     0.00)                        |
|                            λ         λp        λr       CASE                  |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1       |
| Web   : Slender         45.67       N/A      42.29     Table.4.1a.Case5       |
|                                                                               |
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:     0.00)                        |
|                            λ         λp        λr       CASE                  |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10      |
| Web   : Compact         45.67     106.72    161.78     Table.4.1b.Case15      |
|-------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   31
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :     8 Contd.
|-------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                              |
|-------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                               |
| TEN. YLD.        0.000      286.7       0.000     Cl.D2           1    0.00   |
| TEN. RUPT.       0.000      286.7       0.000     Cl.D2           1    0.00   |
|-------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                          |
|-------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                               |
| FLEX. BUCK. X    36.43      279.1       0.131     Cl.E3           3    0.00   |
| FLEX. BUCK. Y    36.43      254.0       0.143     Cl.E3           3    0.00   |
| FLEX. TOR. BUCK  36.43      261.2       0.139     Cl.E4           3    0.00   |
|-------------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                      |
|-------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                               |
| SHEAR X          0.000      100.7       0.000     Cl.G1           1    0.00   |
| SHEAR Y          17.30      80.48       0.215     Cl.G1           1    0.00   |
|-------------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                    |
|-------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                               |
| FLEX. YLD. X     67.77      127.7       0.531     Cl.F2.1         1    0.00   |
| FLEX. YLD. Y     0.000      24.27       0.000     Cl.F6.1         1    0.00   |
```

```
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.599     Eq.H1-1b            1    0.00                  |
| CL.H13-IP          0.593     Eq.H1-3a(H1-1b)     1    0.00                  |
| CL.H13-OP          0.479     Eq.H1-3b            1    0.00                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   32
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     9
|-----------------------------------------------------------------------------|
|  Member No:        9       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.494       Loadcase:        1    |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       35.18     C     Vy:        3.755           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:     -55.01        |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     47.018                                    |
| Allowable Slenderness Ratio :    200.000           LOC  :     0.00          |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1             Ratio     :        0.494(PASS)           |
|          Loc  :    5.83            Condition :    Eq.H1-3a(H1-1b            |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     5.83, PROPERTIES UNIT: IN  )                  |
| Ag  :  8.850E+00     Axx :  5.182E+00     Ayy :  3.726E+00                  |
| Ixx :  2.910E+02     Iyy :  1.960E+01     J   :  3.800E-01                  |
| Sxx+:  4.217E+01     Sxx-:  4.217E+01     Zxx :  4.730E+01                  |
| Syy+:  5.825E+00     Syy-:  5.825E+00     Zyy :  8.990E+00                  |
| Cw  :  8.800E+02     x0  :  0.000E+00     y0  :  0.000E+00                  |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         5.831                                         |
| Design Parameters                                          (Rolled)         |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:     0.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1     |
| Web   : Slender         45.67       N/A      42.29     Table.4.1a.Case5     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:     0.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10    |
| Web   : Compact         45.67     106.72    161.78     Table.4.1b.Case15    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   33
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                     - Member :     9 Contd.
```

```
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      286.7       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      286.7       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    35.61      279.1       0.128     Cl.E3           3    0.00 |
| FLEX. BUCK. Y    35.61      254.0       0.140     Cl.E3           3    0.00 |
| FLEX. TOR. BUCK  35.61      261.2       0.136     Cl.E4           3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      100.7       0.000     Cl.G1           1    0.00 |
| SHEAR Y          9.153      80.48       0.114     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X    -55.01      127.7       0.431     Cl.F2.1         1    5.83 |
| FLEX. YLD. Y     0.000      24.27       0.000     Cl.F6.1         1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.500     Eq.H1-1b            1    5.83                  |
| CL.H13-IP          0.494     Eq.H1-3a(H1-1b)     1    5.83                  |
| CL.H13-OP          0.384     Eq.H1-3b            1    5.83                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   34
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    10
|-----------------------------------------------------------------------------|
|  Member No:       10       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.631       Loadcase:        1     |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       40.64     C     Vy:       -24.36            Vx:      0.000       |
|  Tz:       0.000           My:         0.000           Mx:      71.29       |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     47.018                                    |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1            Ratio     :        0.631(PASS)            |
|          Loc  :    5.83           Condition :    Eq.H1-3a(H1-1b             |
|-----------------------------------------------------------------------------|
```

```
| SECTION PROPERTIES  (LOC:     5.83, PROPERTIES UNIT: IN  )                     |
| Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00                  |
| Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01                  |
| Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01                  |
| Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00                  |
| Cw  :   8.800E+02     x0  :   0.000E+00     y0  :   0.000E+00                  |
|--------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                            |
| Fyld:        5184.000             Fu:        8351.999                          |
|--------------------------------------------------------------------------------|
| Actual Member Length:         5.831                                            |
| Design Parameters                                                (Rolled)      |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00             |
|--------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                         |
|                            λ         λp        λr       CASE                   |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1        |
| Web   : Slender         46.07       N/A      42.29     Table.4.1a.Case5        |
|                                                                                |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                         |
|                            λ         λp        λr       CASE                   |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10       |
| Web   : Compact         46.07     106.72    161.78     Table.4.1b.Case15       |
|--------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   35
                                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                               *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                                - Member :    10 Contd.
|--------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                               |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| TEN. YLD.       0.000      286.7      0.000     Cl.D2           1    0.00     |
| TEN. RUPT.      0.000      286.7      0.000     Cl.D2           1    0.00     |
|--------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                           |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| FLEX. BUCK. X    40.73      278.4      0.146     Cl.E3           1    0.00     |
| FLEX. BUCK. Y    40.73      253.4      0.161     Cl.E3           1    0.00     |
| FLEX. TOR. BUCK  40.73      260.6      0.156     Cl.E4           1    0.00     |
|--------------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                       |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| SHEAR X          0.000      100.7      0.000     Cl.G1           1    0.00     |
| SHEAR Y          24.36      80.48      0.303     Cl.G1           1    5.83     |
|--------------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                     |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| FLEX. YLD. X     71.29      127.7      0.558     Cl.F2.1         1    5.83     |
| FLEX. YLD. Y     0.000      24.27      0.000     Cl.F6.1         1    0.00     |
```

```
|-----------------------------------------------------------------------------|
|                          CHECKS FOR AXIAL BEND INTERACTION                  |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.638     Eq.H1-1b            1    5.83                 |
| CL.H13-IP          0.631     Eq.H1-3a(H1-1b)     1    5.83                 |
| CL.H13-OP          0.539     Eq.H1-3b            1    5.83                 |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   36
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    11
|-----------------------------------------------------------------------------|
|  Member No:       11       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.631       Loadcase:        1    |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       40.62     C     Vy:       -24.40           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:      71.29        |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     47.018                                    |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1             Ratio     :        0.631(PASS)          |
|          Loc  :    5.83            Condition :    Eq.H1-3a(H1-1b            |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     5.83, PROPERTIES UNIT: IN  )                  |
| Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00              |
| Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01              |
| Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01              |
| Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00              |
| Cw  :   8.800E+02     x0  :   0.000E+00     y0  :   0.000E+00              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:         5.831                                         |
| Design Parameters                                            (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1    |
| Web   : Slender         46.07       N/A      42.29     Table.4.1a.Case5    |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                            λ         λp        λr       CASE               |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10   |
| Web   : Compact         46.07     106.72    161.78     Table.4.1b.Case15   |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   37
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                     - Member :    11 Contd.
```

```
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      286.7       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      286.7       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    40.71      278.4       0.146     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    40.71      253.4       0.161     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  40.71      260.6       0.156     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      100.7       0.000     Cl.G1           1    0.00 |
| SHEAR Y          24.40      80.48       0.303     Cl.G1           1    5.83 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X     71.29      127.7       0.558     Cl.F2.1         1    5.83 |
| FLEX. YLD. Y     0.000      24.27       0.000     Cl.F6.1         1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.638     Eq.H1-1b            1    5.83                  |
| CL.H13-IP          0.631     Eq.H1-3a(H1-1b)     1    5.83                  |
| CL.H13-OP          0.539     Eq.H1-3b            1    5.83                  |
|-----------------------------------------------------------------------------|
      EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   38
                        STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                      *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    12
|-----------------------------------------------------------------------------|
|  Member No:       12       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.501       Loadcase:        1     |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                        |
|  Pz:       38.28     C     Vy:        2.675          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:     -55.24         |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     47.018                                    |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1            Ratio     :        0.501(PASS)           |
|          Loc  :    5.83           Condition :    Eq.H1-3a(H1-1b            |
|-----------------------------------------------------------------------------|
```

```
| SECTION PROPERTIES  (LOC:     5.83, PROPERTIES UNIT: IN  )                   |
| Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00                |
| Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01                |
| Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01                |
| Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00                |
| Cw  :   8.800E+02     x0  :   0.000E+00     y0  :   0.000E+00                |
|------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                          |
| Fyld:        5184.000             Fu:        8351.999                        |
|------------------------------------------------------------------------------|
| Actual Member Length:         5.831                                          |
| Design Parameters                                             (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                       |
|                            λ         λp        λr       CASE                 |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1      |
| Web   : Slender         45.67       N/A      42.29     Table.4.1a.Case5      |
|                                                                              |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                       |
|                            λ         λp        λr       CASE                 |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10     |
| Web   : Compact         45.67     106.72    161.78     Table.4.1b.Case15     |
|------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   39
                         STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                       *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :    12 Contd.
|------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                              |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| TEN. YLD.       0.000      286.7       0.000     Cl.D2           1    0.00   |
| TEN. RUPT.      0.000      286.7       0.000     Cl.D2           1    0.00   |
|------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                          |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| FLEX. BUCK. X    38.37      279.1       0.137     Cl.E3           1    0.00   |
| FLEX. BUCK. Y    38.37      254.0       0.151     Cl.E3           1    0.00   |
| FLEX. TOR. BUCK  38.37      261.2       0.147     Cl.E4           1    0.00   |
|------------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                      |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| SHEAR X          0.000      100.7       0.000     Cl.G1           1    0.00   |
| SHEAR Y          8.835      80.48       0.110     Cl.G1           3    0.00   |
|------------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                    |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| FLEX. YLD. X    -55.24      127.7       0.433     Cl.F2.1         1    5.83   |
| FLEX. YLD. Y     0.000      24.27       0.000     Cl.F6.1         1    0.00   |
```

```
|--------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                      |
|--------------------------------------------------------------------------------|
|                                                                                |
|                    RATIO      CRITERIA          L/C    LOC                     |
| CLAUSE H1          0.508     Eq.H1-1b            1    5.83                    |
| CL.H13-IP          0.501     Eq.H1-3a(H1-1b)     1    5.83                    |
| CL.H13-OP          0.402     Eq.H1-3b            1    5.83                    |
|--------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   40
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    13
|--------------------------------------------------------------------------------|
|  Member No:       13       Profile:  ST  W14X30             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.817       Loadcase:        3     |
|  Location:      0.00       Ref:      Eq.H1-3a(H1-1b)                         |
|  Pz:       26.80     C     Vy:        19.16           Vx:      0.000         |
|  Tz:       0.000           My:        0.000           Mx:      98.18         |
|--------------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                        |
| Actual Slenderness Ratio    :     47.018                                       |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00            |
|--------------------------------------------------------------------------------|
| STRENGTH CHECKS                                                                |
| Critical L/C  :      3             Ratio     :        0.817(PASS)             |
|          Loc  :    0.00            Condition :    Eq.H1-3a(H1-1b              |
|--------------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     0.00, PROPERTIES UNIT: IN  )                     |
| Ag  :   8.850E+00     Axx :   5.182E+00     Ayy :   3.726E+00                 |
| Ixx :   2.910E+02     Iyy :   1.960E+01     J   :   3.800E-01                 |
| Sxx+:   4.217E+01     Sxx-:   4.217E+01     Zxx :   4.730E+01                 |
| Syy+:   5.825E+00     Syy-:   5.825E+00     Zyy :   8.990E+00                 |
| Cw  :   8.800E+02     x0  :   0.000E+00     y0  :   0.000E+00                 |
|--------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                            |
| Fyld:        5184.000              Fu:        8351.999                         |
|--------------------------------------------------------------------------------|
| Actual Member Length:         5.831                                            |
| Design Parameters                                             (Rolled)         |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00            |
|--------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                         |
|                            λ         λp        λr       CASE                  |
| Flange: NonSlender       8.74       N/A      15.89     Table.4.1a.Case1       |
| Web   : Slender         45.30       N/A      42.29     Table.4.1a.Case5       |
|                                                                                |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                         |
|                            λ         λp        λr       CASE                  |
| Flange: Compact          8.74      10.79     28.38     Table.4.1b.Case10      |
| Web   : Compact         45.30     106.72    161.78     Table.4.1b.Case15      |
|--------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   41
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :    13 Contd.
```

```
|-----------------------------------------------------------------------------|
|                       CHECKS FOR AXIAL TENSION                              |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        0.000      286.7       0.000     Cl.D2           1    0.00 |
| TEN. RUPT.       0.000      286.7       0.000     Cl.D2           1    0.00 |
|-----------------------------------------------------------------------------|
|                       CHECKS FOR AXIAL COMPRESSION                          |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    38.64      279.8       0.138     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    38.64      254.5       0.152     Cl.E3           1    0.00 |
| FLEX. TOR. BUCK  38.64      261.8       0.148     Cl.E4           1    0.00 |
|-----------------------------------------------------------------------------|
|                       CHECKS FOR SHEAR                                      |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X          0.000      100.7       0.000     Cl.G1           1    0.00 |
| SHEAR Y          19.16      80.48       0.238     Cl.G1           3    0.00 |
|-----------------------------------------------------------------------------|
|                       CHECKS FOR BENDING                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. X     98.18      127.7       0.769     Cl.F2.1         3    0.00 |
| FLEX. YLD. Y     0.000      24.27       0.000     Cl.F6.1         1    0.00 |
|-----------------------------------------------------------------------------|
|                       CHECKS FOR AXIAL BEND INTERACTION                     |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H1          0.821     Eq.H1-1b             3    0.00                 |
| CL.H13-IP          0.817     Eq.H1-3a(H1-1b)      3    0.00                 |
| CL.H13-OP          0.743     Eq.H1-3b             3    0.00                 |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   42
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    14
|-----------------------------------------------------------------------------|
|  Member No:       14       Profile:  ST  L30253             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.193       Loadcase:        3    |
|  Location:      1.95       Ref:      Eq.H2-1                                |
|  Pz:       4.890     C     Vy:       -.2419E-03       Vx:     -.3520E-03    |
|  Tz:       0.000           My:       -.3112E-02       Mx:     0.2139E-02    |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
| Actual Slenderness Ratio    :     61.594                                    |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3            Ratio     :        0.193(PASS)            |
|          Loc  :    1.95           Condition :    Eq.H2-1                    |
|-----------------------------------------------------------------------------|
```

```
| SECTION PROPERTIES  (LOC:     1.95, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01               |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02               |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01               |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00               |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01               |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:         3.905                                         |
| Design Parameters                                             (Rolled)      |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:     0.00)                      |
|                            λ         λp        λr       CASE              |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:     0.00)                      |
|                            λ         λp        λr       CASE              |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   43
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :    14 Contd.
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        2.784      32.40       0.086     Cl.D2            1    3.91 |
| TEN. RUPT.       2.784      32.40       0.086     Cl.D2            1    3.91 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    4.893      27.15       0.180     Cl.E3            3    0.00 |
| FLEX. BUCK. Y    4.893      25.55       0.192     Cl.E3            3    0.00 |
| FLEX. BUCK. U    4.893      27.15       0.180     Cl.E3            3    0.00 |
| FLEX. BUCK. V    4.893      25.55       0.192     Cl.E3            3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X         0.4050E-02  9.137       0.000     Cl.G1            1    0.00 |
| SHEAR Y         0.2783E-02  10.96       0.000     Cl.G1            1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. U    0.2891E-18  1.752       0.000     Cl.F10.1         1    1.30 |
```

```
| FLEX. YLD. V    0.4971E-02  2.270       0.002     Cl.F10.1        1    2.28 |
| L-T-B ABT U     -.2891E-18  1.602       0.000     Cl.F10.2        1    2.28 |
| F-L-B ABT U     -.2891E-18  1.709       0.000     Cl.F10.3        1    2.28 |
| F-L-B ABT V     -.5835E-03  2.213       0.000     Cl.F10.3        1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H2          0.193     Eq.H2-1             3    1.95                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                -- PAGE NO.   44
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    15
|-----------------------------------------------------------------------------|
|  Member No:       15       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.130       Loadcase:        3    |
|  Location:      3.79       Ref:      Eq.H2-1                                |
|  Pz:       4.106     T     Vy:       0.1702E-03       Vx:     0.2477E-03    |
|  Tz:       0.000           My:       -.5201E-02       Mx:     0.3574E-02    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :    102.522                                    |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.130(PASS)           |
|          Loc  :    3.79            Condition :    Eq.H2-1                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.79, PROPERTIES UNIT: IN  )                  |
| Ag   :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx  :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01      Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01      Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw   :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         6.500                                         |
| Design Parameters                                               (Rolled)    |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE               |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE               |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                -- PAGE NO.   45
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :    15 Contd.
|-------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                               |
|-------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC     |
|                                                                               |
| TEN. YLD.        4.112      32.40       0.127     Cl.D2            3    6.50  |
| TEN. RUPT.       4.112      32.40       0.127     Cl.D2            3    6.50  |
|-------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                           |
|-------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC     |
|                                                                               |
| FLEX. BUCK. X    2.276      22.47       0.101     Cl.E3            1    0.00  |
| FLEX. BUCK. Y    2.276      18.63       0.122     Cl.E3            1    0.00  |
| FLEX. BUCK. U    2.276      22.47       0.101     Cl.E3            1    0.00  |
| FLEX. BUCK. V    2.276      18.63       0.122     Cl.E3            1    0.00  |
|-------------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                       |
|-------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC     |
|                                                                               |
| SHEAR X         0.3690E-02  9.137       0.000     Cl.G1            1    0.00  |
| SHEAR Y         0.2536E-02  10.96       0.000     Cl.G1            1    0.00  |
|-------------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                     |
|-------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC     |
|                                                                               |
| FLEX. YLD. U    -.5782E-18  1.752       0.000     Cl.F10.1         1    3.25  |
| FLEX. YLD. V    0.7994E-02  2.270       0.004     Cl.F10.1         1    3.25  |
| L-T-B ABT U     -.5782E-18  1.469       0.000     Cl.F10.2         1    3.25  |
| F-L-B ABT U     -.5782E-18  1.709       0.000     Cl.F10.3         1    3.25  |
|-------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                      |
|-------------------------------------------------------------------------------|
|                                                                               |
|                   RATIO      CRITERIA          L/C    LOC                     |
| CLAUSE H2          0.130     Eq.H2-1             3    3.79                    |
|-------------------------------------------------------------------------------|
      EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   46
                                 STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                               ***************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :    16
|-------------------------------------------------------------------------------|
|  Member No:       16       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.758       Loadcase:        1    |
|  Location:      3.91       Ref:      Eq.H2-1                                 |
|  Pz:       24.33     T     Vy:       0.2228E-03       Vx:     0.3243E-03     |
|  Tz:       0.000           My:       -.1351E-01       Mx:     0.9286E-02     |
|-------------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                           |
| Actual Slenderness Ratio    :    123.188                                      |
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00           |
|-------------------------------------------------------------------------------|
| STRENGTH CHECKS                                                               |
```

```
| Critical L/C  :      1             Ratio     :        0.758(PASS)            |
|          Loc  :    3.91            Condition :    Eq.H2-1                    |
|------------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.91, PROPERTIES UNIT: IN  )                   |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01               |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02               |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01               |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00               |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01               |
|------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                          |
| Fyld:        5184.000              Fu:        8351.999                       |
|------------------------------------------------------------------------------|
| Actual Member Length:         7.810                                          |
| Design Parameters                                             (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00          |
|------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    93.72)                       |
|                            λ         λp        λr       CASE                 |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3      |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3      |
|                                                                              |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    93.72)                       |
|                            λ         λp        λr       CASE                 |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12     |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12     |
|------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   47
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :    16 Contd.
|------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                              |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| TEN. YLD.        24.34      32.40       0.751     Cl.D2           1    7.81  |
| TEN. RUPT.       24.34      32.40       0.751     Cl.D2           1    7.81  |
|------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                          |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| FLEX. BUCK. X    0.000      19.49       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. Y    0.000      14.57       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. U    0.000      19.49       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. V    0.000      14.57       0.000     Cl.E3           1    0.00  |
|------------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                      |
|------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC   |
|                                                                              |
| SHEAR X         0.7321E-02  9.137       0.001     Cl.G1           1    7.81  |
| SHEAR Y         0.5031E-02  10.96       0.000     Cl.G1           1    7.81  |
|------------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                    |
|------------------------------------------------------------------------------|
```

```
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                            |
| FLEX. YLD. U    0.5782E-18  1.752       0.000     Cl.F10.1        1    1.30 |
| FLEX. YLD. V    0.1640E-01  2.270       0.007     Cl.F10.1        1    3.91 |
| L-T-B ABT U     -.5782E-18  1.409       0.000     Cl.F10.2        3    1.30 |
| F-L-B ABT U     -.5782E-18  1.709       0.000     Cl.F10.3        3    1.30 |
| F-L-B ABT V     -.1718E-02  2.213       0.001     Cl.F10.3        1    7.81 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H2          0.758     Eq.H2-1             1    3.91                 |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   48
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    17
|-----------------------------------------------------------------------------|
|  Member No:       17       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.732       Loadcase:        3    |
|  Location:      3.91       Ref:      Eq.H2-1                                |
|  Pz:       23.54     T     Vy:       0.1760E-03       Vx:     0.2561E-03    |
|  Tz:       0.000           My:       -.1011E-01       Mx:     0.6946E-02    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :    123.188                                    |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3              Ratio     :        0.732(PASS)          |
|          Loc  :    3.91             Condition :    Eq.H2-1                  |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.91, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000              Fu:        8351.999                      |
|-----------------------------------------------------------------------------|
| Actual Member Length:         7.810                                         |
| Design Parameters                                              (Rolled)     |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    93.72)                      |
|                           λ         λp        λr       CASE                 |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    93.72)                      |
|                           λ         λp        λr       CASE                 |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|-----------------------------------------------------------------------------|
```

```
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   49
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :    17 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        23.55      32.40       0.727     Cl.D2           3    7.81 |
| TEN. RUPT.       23.55      32.40       0.727     Cl.D2           3    7.81 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    0.000      19.49       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    0.000      14.57       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. U    0.000      19.49       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. V    0.000      14.57       0.000     Cl.E3           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X         0.7336E-02  9.137       0.001     Cl.G1           1    7.81 |
| SHEAR Y         0.5041E-02  10.96       0.000     Cl.G1           1    7.81 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. U    0.1156E-17  1.752       0.000     Cl.F10.1        3    3.91 |
| FLEX. YLD. V    0.1638E-01  2.270       0.007     Cl.F10.1        1    3.91 |
| L-T-B ABT U     0.1156E-17  1.542       0.000     Cl.F10.2        3    3.91 |
| F-L-B ABT U     -.5782E-18  1.709       0.000     Cl.F10.3        1    1.30 |
| F-L-B ABT V     -.1801E-02  2.213       0.001     Cl.F10.3        1    7.81 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H2          0.732     Eq.H2-1              3    3.91                 |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   50
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    18
|-----------------------------------------------------------------------------|
|  Member No:       18       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.284       Loadcase:        3    |
|  Location:      3.25       Ref:      Eq.H2-1                                |
|  Pz:       5.245     C     Vy:       -.8356E-04       Vx:     -.1216E-03    |
|  Tz:       0.000           My:       -.4805E-02       Mx:     0.3302E-02    |
|-----------------------------------------------------------------------------|
| COMPRESSION SLENDERNESS                                                     |
```

```
| Actual Slenderness Ratio    :    102.522                                       |
| Allowable Slenderness Ratio :    200.000            LOC  :     0.00            |
|--------------------------------------------------------------------------------|
| STRENGTH CHECKS                                                                |
| Critical L/C  :      3             Ratio     :        0.284(PASS)              |
|          Loc  :    3.25            Condition :    Eq.H2-1                      |
|--------------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.25, PROPERTIES UNIT: IN  )                     |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01                  |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02                  |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01                  |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00                  |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01                  |
|--------------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                            |
| Fyld:        5184.000             Fu:        8351.999                          |
|--------------------------------------------------------------------------------|
| Actual Member Length:         6.500                                            |
| Design Parameters                                               (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00             |
|--------------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:     0.00)                         |
|                           λ         λp        λr       CASE                    |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3        |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3        |
|                                                                                |
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:     0.00)                         |
|                           λ         λp        λr       CASE                    |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12       |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12       |
|--------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   51
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :    18 Contd.
|--------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                                |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| TEN. YLD.        1.041      32.40       0.032     Cl.D2           1    6.50    |
| TEN. RUPT.       1.041      32.40       0.032     Cl.D2           1    6.50    |
|--------------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                            |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| FLEX. BUCK. X    5.252      22.47       0.234     Cl.E3           3    0.00    |
| FLEX. BUCK. Y    5.252      18.63       0.282     Cl.E3           3    0.00    |
| FLEX. BUCK. U    5.252      22.47       0.234     Cl.E3           3    0.00    |
| FLEX. BUCK. V    5.252      18.63       0.282     Cl.E3           3    0.00    |
|--------------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                        |
|--------------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C     LOC     |
|                                                                                |
| SHEAR X         0.3704E-02  9.137       0.000     Cl.G1           1    0.00    |
```

```
| SHEAR Y         0.2545E-02  10.96       0.000     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                              |
| FLEX. YLD. U    -.5782E-18  1.752       0.000     Cl.F10.1        1    1.63 |
| FLEX. YLD. V    0.8144E-02  2.270       0.004     Cl.F10.1        1    3.25 |
| L-T-B ABT U     -.5782E-18  1.469       0.000     Cl.F10.2        1    1.63 |
| F-L-B ABT U     -.5782E-18  1.709       0.000     Cl.F10.3        1    1.63 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                              |
|                   RATIO      CRITERIA          L/C    LOC                   |
| CLAUSE H2          0.284     Eq.H2-1             3    3.25                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   52
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    19
|-----------------------------------------------------------------------------|
|  Member No:       19       Profile:  ST  L30253              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.197       Loadcase:        3    |
|  Location:      2.28       Ref:      Eq.H2-1                                |
|  Pz:       6.345     T     Vy:       0.2359E-04       Vx:     0.3432E-04    |
|  Tz:       0.000           My:       -.2951E-02       Mx:     0.2028E-02    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :     61.594                                    |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3             Ratio     :        0.197(PASS)           |
|          Loc  :    2.28            Condition :    Eq.H2-1                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     2.28, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:       5184.000              Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         3.905                                         |
| Design Parameters                                              (Rolled)     |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE                 |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                              |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:     0.00)                      |
|                           λ         λp        λr       CASE                 |
```

```
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   53
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :    19 Contd.
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                            |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| TEN. YLD.        6.348      32.40       0.196     Cl.D2           3    3.91 |
| TEN. RUPT.       6.348      32.40       0.196     Cl.D2           3    3.91 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                        |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. BUCK. X    1.187      27.15       0.044     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    1.187      25.55       0.046     Cl.E3           1    0.00 |
| FLEX. BUCK. U    1.187      27.15       0.044     Cl.E3           1    0.00 |
| FLEX. BUCK. V    1.187      25.55       0.046     Cl.E3           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X         0.4009E-02  9.137       0.000     Cl.G1           1    0.00 |
| SHEAR Y         0.2755E-02  10.96       0.000     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. U    -.2891E-18  1.752       0.000     Cl.F10.1        3    1.30 |
| FLEX. YLD. V    0.4975E-02  2.270       0.002     Cl.F10.1        1    2.28 |
| L-T-B ABT U     -.2891E-18  1.602       0.000     Cl.F10.2        3    1.30 |
| F-L-B ABT U     -.2891E-18  1.709       0.000     Cl.F10.3        3    1.30 |
| F-L-B ABT V     -.5547E-03  2.213       0.000     Cl.F10.3        3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H2          0.197     Eq.H2-1             3    2.28                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   54
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    20
|-----------------------------------------------------------------------------|
|  Member No:       20       Profile:  ST  L30253               (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.571      Loadcase:        3     |
|  Location:      3.75       Ref:      Eq.H2-1                                |
|  Pz:       18.24     T     Vy:       -.3709E-04      Vx:     -.5397E-04     |
```

```
| Tz:       0.000           My:       -.1515E-01       Mx:     0.1041E-01    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :    118.295                                   |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00        |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      3              Ratio     :        0.571(PASS)          |
|          Loc  :    3.75             Condition :    Eq.H2-1                  |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                 |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         7.500                                         |
| Design Parameters                                            (Rolled)       |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    90.00)                      |
|                           λ         λp        λr       CASE               |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3    |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3    |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    90.00)                      |
|                           λ         λp        λr       CASE               |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12   |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12   |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   55
                     STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                   *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                         - Member :    20 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| TEN. YLD.       18.24      32.40       0.563     Cl.D2           3    0.00  |
| TEN. RUPT.      18.24      32.40       0.563     Cl.D2           3    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. BUCK. X   0.000      20.27       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. Y   0.000      15.51       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. U   0.000      20.27       0.000     Cl.E3           1    0.00  |
| FLEX. BUCK. V   0.000      15.51       0.000     Cl.E3           1    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
```

```
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X         0.1076E-01  9.137       0.001     Cl.G1           1    0.00 |
| SHEAR Y         0.7393E-02  10.96       0.001     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. U    -.1156E-17  1.752       0.000     Cl.F10.1        1    1.88 |
| FLEX. YLD. V    0.2301E-01  2.270       0.010     Cl.F10.1        1    3.75 |
| L-T-B ABT U     -.1156E-17  1.423       0.000     Cl.F10.2        1    1.88 |
| F-L-B ABT U     -.1156E-17  1.709       0.000     Cl.F10.3        1    1.88 |
| F-L-B ABT V     -.2066E-02  2.213       0.001     Cl.F10.3        1    0.00 |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                   |
| CLAUSE H2          0.571     Eq.H2-1             3    3.75                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   56
                    STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                  *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    21
|-----------------------------------------------------------------------------|
|  Member No:       21       Profile:  ST  L30253               (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.544       Loadcase:        1    |
|  Location:      3.75       Ref:      Eq.H2-1                               |
|  Pz:      17.29     T      Vy:       -.1627E-03       Vx:     -.2368E-03    |
|  Tz:      0.000            My:       -.2007E-01       Mx:     0.1379E-01    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :    118.295                                   |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00         |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1            Ratio     :        0.544(PASS)           |
|          Loc  :    3.75           Condition :    Eq.H2-1                    |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         7.500                                         |
| Design Parameters                                               (Rolled)   |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    90.00)                      |
|                              λ        λp        λr       CASE              |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
```

```
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                              |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    90.00)                       |
|                           λ         λp        λr       CASE                  |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   57
                  STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                ******************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                        - Member :    21 Contd.
|------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL TENSION                             |
|------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                              |
| TEN. YLD.        17.29      32.40       0.534     Cl.D2           1    0.00 |
| TEN. RUPT.       17.29      32.40       0.534     Cl.D2           1    0.00 |
|------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL COMPRESSION                         |
|------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                              |
| FLEX. BUCK. X    0.000      20.27       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    0.000      15.51       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. U    0.000      20.27       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. V    0.000      15.51       0.000     Cl.E3           1    0.00 |
|------------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                     |
|------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                              |
| SHEAR X         0.1073E-01  9.137       0.001     Cl.G1           1    0.00 |
| SHEAR Y         0.7375E-02  10.96       0.001     Cl.G1           1    0.00 |
|------------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                   |
|------------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                              |
| FLEX. YLD. U    -.1156E-17  1.752       0.000     Cl.F10.1        1    2.50 |
| FLEX. YLD. V    0.2435E-01  2.270       0.011     Cl.F10.1        1    3.75 |
| L-T-B ABT U     -.1156E-17  1.423       0.000     Cl.F10.2        1    2.50 |
| F-L-B ABT U     -.1156E-17  1.709       0.000     Cl.F10.3        1    2.50 |
| F-L-B ABT V     -.6014E-03  2.213       0.000     Cl.F10.3        1    0.00 |
|------------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                    |
|------------------------------------------------------------------------------|
|                                                                              |
|                    RATIO      CRITERIA          L/C    LOC                   |
| CLAUSE H2          0.544     Eq.H2-1             1    3.75                   |
|------------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   58
                  STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                ******************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    22
|------------------------------------------------------------------------------|
```

```
|  Member No:       22       Profile:  ST  L30253             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.604       Loadcase:        1   |
|  Location:      3.75       Ref:      Eq.H2-1                               |
|  Pz:       19.23     T     Vy:       -.1333E-03       Vx:     -.1939E-03   |
|  Tz:       0.000           My:       -.2019E-01       Mx:     0.1387E-01   |
|----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                        |
| Actual Slenderness Ratio    :    118.295                                   |
| Allowable Slenderness Ratio :    300.000            LOC  :     0.00        |
|----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                            |
| Critical L/C  :      1             Ratio     :        0.604(PASS)          |
|          Loc  :    3.75            Condition :    Eq.H2-1                  |
|----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                 |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01             |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02             |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01             |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00             |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01             |
|----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                        |
| Fyld:        5184.000             Fu:        8351.999                      |
|----------------------------------------------------------------------------|
| Actual Member Length:         7.500                                        |
| Design Parameters                                             (Rolled)     |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00        |
|----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      1 LOC:    90.00)                     |
|                           λ         λp        λr       CASE                |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3    |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3    |
|                                                                            |
| FLEXURE CLASSIFICATION     (L/C:      1 LOC:    90.00)                     |
|                           λ         λp        λr       CASE                |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12   |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12   |
|----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   59
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                       - Member :    22 Contd.
|----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                            |
|----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC |
|                                                                            |
| TEN. YLD.        19.23      32.40       0.594     Cl.D2           1    0.00 |
| TEN. RUPT.       19.23      32.40       0.594     Cl.D2           1    0.00 |
|----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                        |
|----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC |
|                                                                            |
| FLEX. BUCK. X    0.000      20.27       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. Y    0.000      15.51       0.000     Cl.E3           1    0.00 |
| FLEX. BUCK. U    0.000      20.27       0.000     Cl.E3           1    0.00 |
```

```
| FLEX. BUCK. V    0.000      15.51       0.000     Cl.E3           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR SHEAR                                    |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| SHEAR X         0.1069E-01  9.137       0.001     Cl.G1           1    0.00 |
| SHEAR Y         0.7346E-02  10.96       0.001     Cl.G1           1    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR BENDING                                  |
|-----------------------------------------------------------------------------|
|                    DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC  |
|                                                                             |
| FLEX. YLD. U    -.1156E-17  1.752       0.000     Cl.F10.1        1    1.25 |
| FLEX. YLD. V    0.2449E-01  2.270       0.011     Cl.F10.1        1    3.75 |
| L-T-B ABT U     -.1156E-17  1.423       0.000     Cl.F10.2        1    1.25 |
| F-L-B ABT U     -.1156E-17  1.709       0.000     Cl.F10.3        1    1.25 |
| F-L-B ABT V     -.1036E-02  2.213       0.000     Cl.F10.3        3    0.00 |
|-----------------------------------------------------------------------------|
|                         CHECKS FOR AXIAL BEND INTERACTION                   |
|-----------------------------------------------------------------------------|
|                                                                             |
|                    RATIO      CRITERIA          L/C    LOC                  |
| CLAUSE H2          0.604     Eq.H2-1             1    3.75                  |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   60
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    23
|-----------------------------------------------------------------------------|
|  Member No:       23       Profile:  ST  L30253             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.640       Loadcase:        1    |
|  Location:      3.75       Ref:      Eq.H2-1                                |
|  Pz:       20.38     T     Vy:       -.8185E-04       Vx:     -.1191E-03    |
|  Tz:       0.000           My:       -.1976E-01       Mx:     0.1358E-01    |
|-----------------------------------------------------------------------------|
| TENSION SLENDERNESS                                                         |
| Actual Slenderness Ratio    :    118.295                                    |
| Allowable Slenderness Ratio :    300.000           LOC  :     0.00          |
|-----------------------------------------------------------------------------|
| STRENGTH CHECKS                                                             |
| Critical L/C  :      1             Ratio     :        0.640(PASS)           |
|          Loc  :    3.75            Condition :    Eq.H2-1                   |
|-----------------------------------------------------------------------------|
| SECTION PROPERTIES  (LOC:     3.75, PROPERTIES UNIT: IN  )                  |
| Ag  :   1.000E+00     Axx :   4.700E-01     Ayy :   5.640E-01              |
| Ixx :   9.095E-01     Iyy :   5.788E-01     J   :   1.178E-02              |
| Sxx+:   5.634E-01     Sxx-:   5.604E-01     Zxx :   4.733E-01              |
| Syy+:   4.326E-01     Syy-:   5.093E-01     Zyy :   1.042E+00              |
| Cw  :   7.100E-03     x0  :   5.432E-01     y0  :   7.929E-01              |
|-----------------------------------------------------------------------------|
| MATERIAL PROPERTIES                                                         |
| Fyld:        5184.000             Fu:        8351.999                       |
|-----------------------------------------------------------------------------|
| Actual Member Length:         7.500                                         |
| Design Parameters                                          (Rolled)         |
| Kx:    1.00  Ky:    1.00  NSF:    0.85  SLF:    1.00  CSP:    1.00         |
```

```
|-----------------------------------------------------------------------------|
| COMPRESSION CLASSIFICATION (L/C:      3 LOC:    90.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: Slender         13.30       N/A      12.77     Table.4.1a.Case3     |
| Web   : Slender         15.96       N/A      12.77     Table.4.1a.Case3     |
|                                                                             |
| FLEXURE CLASSIFICATION     (L/C:      3 LOC:    90.00)                      |
|                            λ         λp        λr       CASE                |
| Flange: Compact         13.30      15.33     25.83     Table.4.1b.Case12    |
| Web   : NonCompact      15.96      15.33     25.83     Table.4.1b.Case12    |
|-----------------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   61
                      STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                    *****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
                                                      - Member :    23 Contd.
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL TENSION                             |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| TEN. YLD.        20.38      32.40      0.629     Cl.D2           1    0.00  |
| TEN. RUPT.       20.38      32.40      0.629     Cl.D2           1    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL COMPRESSION                         |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. BUCK. X   0.7803      20.27      0.038     Cl.E3           3    0.00  |
| FLEX. BUCK. Y   0.7803      15.51      0.050     Cl.E3           3    0.00  |
| FLEX. BUCK. U   0.7803      20.27      0.038     Cl.E3           3    0.00  |
| FLEX. BUCK. V   0.7803      15.51      0.050     Cl.E3           3    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR SHEAR                                     |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| SHEAR X         0.1061E-01  9.137      0.001     Cl.G1           1    0.00  |
| SHEAR Y         0.7295E-02  10.96      0.001     Cl.G1           1    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR BENDING                                   |
|-----------------------------------------------------------------------------|
|                   DEMAND    CAPACITY   RATIO     REFERENCE     L/C    LOC   |
|                                                                             |
| FLEX. YLD. U    0.2313E-17  1.752      0.000     Cl.F10.1        1    3.13  |
| FLEX. YLD. V    0.2397E-01  2.270      0.011     Cl.F10.1        1    3.75  |
| L-T-B ABT U     0.2313E-17  1.558      0.000     Cl.F10.2        1    3.13  |
| F-L-B ABT U     -.1156E-17  1.709      0.000     Cl.F10.3        1    5.00  |
| F-L-B ABT V     -.3818E-02  2.213      0.002     Cl.F10.3        3    0.00  |
|-----------------------------------------------------------------------------|
|                        CHECKS FOR AXIAL BEND INTERACTION                    |
|-----------------------------------------------------------------------------|
|                                                                             |
|                   RATIO      CRITERIA          L/C    LOC                   |
| CLAUSE H2         0.640     Eq.H2-1             1    3.75                   |
|-----------------------------------------------------------------------------|
   79. STEEL TAKE OFF
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   62
```

```
STEEL TAKE OFF
  STEEL TAKE-OFF
  --------------
     PROFILE                 LENGTH(FEET)      WEIGHT(KIP )
  ST  W14X90                      55.00            4.950
  ST  W12X30                      15.00            0.448
  ST  W21X44                      30.00            1.324
  ST  W14X30                      34.99            1.051
  ST  L30253                      66.43            0.226
                                          ----------------
                                   TOTAL =         7.999
  ************ END OF DATA FROM INTERNAL STORAGE ************
   80. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:26 ****
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   63
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-2 Area Load Generation on Floor Structure

A floor structure (bound by global X-Z axis) made up of steel beams is subjected to area load (i.e., load/area of floor). Load generation based on one-way distribution is illustrated in this example.

In the case of loads such as joint loads and member loads, the magnitude and direction of the load at the applicable joints and members is directly known from the input. However, the area load is a different sort of load where a load intensity on the given area has to be converted to joint and member loads. The calculations required to perform this conversion are done only during the analysis. Consequently, the loads generated from the `AREA LOAD` command can be viewed only after the analysis is completed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-2 Area Load Generation on Floor Structure.STD` when you install the program.

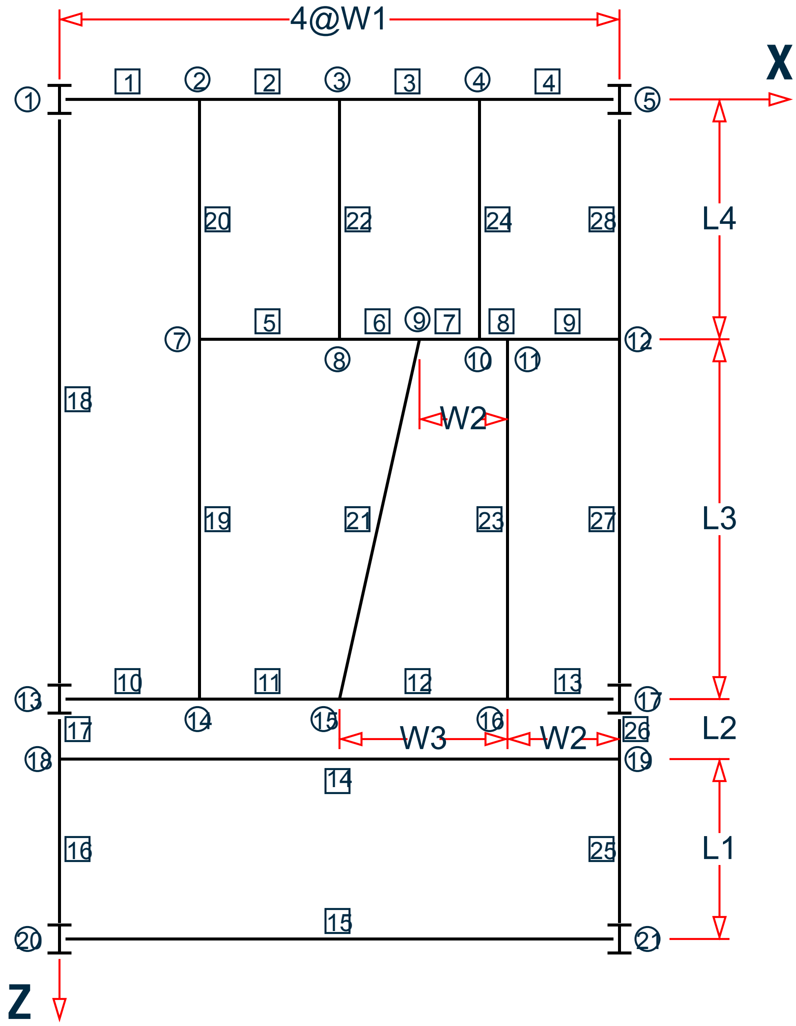


Figure 513: Example Problem No. 2

Where:

W1 = 5 ft, W2 = 3.5 ft, W3 = 5.5 ft, L1 = 7 ft, L2 = 15 ft, L3 = 10 ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD FLOOR A FLOOR FRAME DESIGN WITH AREA LOAD
```

Every input has to start with the term STAAD. The term FLOOR signifies that the structure is a floor structure and the structure is in the x – z plane.

```
UNIT FT KIP
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0. 0. 0.  5 20. 0. 0. ; 7 5. 0. 10.
8 10. 0. 10. ; 9 13. 0. 10. ; 10 15. 0. 10. ; 11 16.5 0. 10.
12 20. 0. 10. ; 13 0. 0. 25. ; 14 5. 0. 25. ; 15 11. 0. 25.
16 16.5 0. 25 ; 17 20. 0. 25. 18 0. 0. 28.
19 20. 0. 28. ; 20 0. 0. 35. ; 21 20. 0. 35.
```

Joint numbers followed by X, Y and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). Joints between 1 and 5 (i.e., 2, 3, 4) are generated in the first line of input, taking advantage of the equal spacing between the joints (refer to TR.11 Joint Coordinates Specification (on page 2217) for more information).

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12
```

Defines the members by the joints to which they are connected.

```
MEMB PROP AMERICAN
1 TO 28 TABLE ST W12X26
```

Member properties are specified from the AISC steel table. In this case, the W12X26 section is chosen. The term `ST` stands for standard single section.

```
* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
```

The first set of members (1 5 10 etc) have local moment-z (MZ) released at the start joint. This means that these members cannot carry any moment-z (i.e., strong axis moment) at the start joint. The second set of members have MZ released at the end joints.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FT
```

Define the material properties for steel. The units are changed to inches and then back to feet in order to facilitate the input of values in familiar units.

```
CONSTANT
MATERIAL STEEL ALL
```

The `CONSTANT` command instructs the program to use the defined steel material for all members.

```
SUPPORT
1 5 13 17 20 21 FIXED
```

A fixed support has been specified at the above joints.

```
LOADING  1  300  POUNDS PER SFT DL+LL
```

Load case 1 is initiated followed by a title.

```
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -0.30
```

All members within a range of -0.5 feet to 0.5 feet in the global Y direction (which is the entire floor of this model) are subjected to an floor load of 0.3 kips/sq.ft. The program converts area loads into individual member loads.

```
   PERFORM ANALYSIS PRINT LOAD DATA
```

This command instructs the program to proceed with the analysis. The `PRINT LOAD DATA` command is specified to obtain a listing of the member loads which were generated from the `FLOOR LOAD`.

```
   PARAMETERS
   CODE AISC UNIFIED
   BEAM 1 ALL
   DMAX 2.0 ALL
   DMIN 1.0 ALL
   UNT 1.0 ALL
   UNB 1.0 ALL
```

The `PARAMETER` command is used to specify steel design parameters (refer to D1.B.1.2 Design Parameters (on page 1134)). Design is to be performed per the specifications of the AISC ASD Code. The `BEAM` parameter is specified to perform design at every 1/12th point along the member length. `DMAX` and `DMIN` specify maximum and minimum depth limitations to be used during member selection. `UNT` and `UNB` are used to specify unsupported length for top and bottom flange to be used for calculation of allowable bending stress.

```
   SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
```

The above command instructs the program to select the most economical section from the British steel table for the members listed.

```
   FINISH
```

The `FINISH` command terminates the STAAD run.

Input File

```
STAAD FLOOR  A FLOOR FRAME DESIGN WITH AREA LOAD
UNIT FT KIP
JOINT COORDINATES
1 0. 0. 0.  5 20. 0. 0. ; 7 5. 0. 10.
8 10. 0. 10. ; 9 13. 0. 10. ; 10 15. 0. 10. ; 11 16.5 0. 10.
12 20. 0. 10. ; 13 0. 0. 25. ; 14 5. 0. 25. ; 15 11. 0. 25.
16 16.5 0. 25 ; 17 20. 0. 25. 18 0. 0. 28.
19 20. 0. 28. ; 20 0. 0. 35. ; 21 20. 0. 35.
MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12
MEMB PROP AMERICAN
1 TO 28 TABLE ST W12X26
* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
UNIT INCHES
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT FT
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 5 13 17 20 21 FIXED
LOADING 1 300 POUNDS PER SFT DL+LL
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -0.30
PERFORM ANALYSIS PRINT LOAD DATA
PARAMETERS
CODE AISC UNIFIED
BEAM 1.0 ALL
DMAX 2.0 ALL
DMIN 1.0 ALL
UNT 1.0 ALL
UNB 1.0 ALL
SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:54                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD FLOOR  A FLOOR FRAME DESIGN WITH AREA LOAD
INPUT FILE: US-2 Area Load Generation on Floor Structure.STD
     2. UNIT FT KIP
     3. JOINT COORDINATES
     4. 1 0. 0. 0.  5 20. 0. 0. ; 7 5. 0. 10.
     5. 8 10. 0. 10. ; 9 13. 0. 10. ; 10 15. 0. 10. ; 11 16.5 0. 10.
     6. 12 20. 0. 10. ; 13 0. 0. 25. ; 14 5. 0. 25. ; 15 11. 0. 25.
     7. 16 16.5 0. 25 ; 17 20. 0. 25. 18 0. 0. 28.
     8. 19 20. 0. 28. ; 20 0. 0. 35. ; 21 20. 0. 35.
     9. MEMBER INCIDENCES
    10. 1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
    11. 15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
    12. 19 7 14 ; 20 2 7 ; 21 9 15
    13. 22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
    14. 26 17 19 ; 27 12 17 ; 28 5 12
```

```
   15. MEMB PROP AMERICAN
   16. 1 TO 28 TABLE ST W12X26
   17. * MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
   18. MEMB RELEASE
   19. 1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
   20. 4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
   21. UNIT INCHES
   22. DEFINE MATERIAL START
   23. ISOTROPIC STEEL
   24. E 29000
   25. POISSON 0.3
   26. DENSITY 283E-006
   27. ALPHA 6E-006
   28. DAMP 0.03
   29. TYPE STEEL
   30. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
   31. END DEFINE MATERIAL
   32. UNIT FT
   33. CONSTANT
   34. MATERIAL STEEL ALL
   35. SUPPORT
   36. 1 5 13 17 20 21 FIXED
   37. LOADING 1 300 POUNDS PER SFT DL+LL
   38. FLOOR LOAD
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    2
   39. YRANGE -0.5 0.5 FLOAD -0.30
**NOTE** about Floor/OneWay Loads/Weights.
  Please note that depending on the shape of the floor you may
  have to break up the FLOOR/ONEWAY LOAD into multiple commands.
  For details please refer to Technical Reference Manual
  Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
   40. PERFORM ANALYSIS PRINT LOAD DATA
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         20  NUMBER OF MEMBERS      28
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      42
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    3
  LOADING     1  300 POUNDS PER SFT DL+LL
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     1                            -0.0146 GY   0.21
     1                            -0.0439 GY   0.49
     1                            -0.0732 GY   0.79
     1                            -0.1025 GY   1.10
     1                            -0.1318 GY   1.41
     1                            -0.1611 GY   1.72
     1                            -0.1904 GY   2.04
     1                            -0.2197 GY   2.35
     1                            -0.2197 GY   2.65
     1                            -0.1904 GY   2.96
     1                            -0.1611 GY   3.28
     1                            -0.1318 GY   3.59
```

```
      1                            -0.1025 GY   3.90
      1                            -0.0732 GY   4.21
      1                            -0.0439 GY   4.51
      1                            -0.0146 GY   4.79
     20    -0.7500 GY   2.50   10.00
     20                            -0.0146 GY   0.21
     20                            -0.0439 GY   0.49
     20                            -0.0732 GY   0.79
     20                            -0.1025 GY   1.10
     20                            -0.1318 GY   1.41
     20                            -0.1611 GY   1.72
     20                            -0.1904 GY   2.04
     20                            -0.2197 GY   2.35
     19    -0.7500 GY   0.00   12.50
     19                            -0.2197 GY  12.65
     19                            -0.1904 GY  12.96
     19                            -0.1611 GY  13.28
     19                            -0.1318 GY  13.59
     19                            -0.1025 GY  13.90
     19                            -0.0732 GY  14.21
     19                            -0.0439 GY  14.51
     19                            -0.0146 GY  14.79
     10                            -0.0146 GY   0.21
     10                            -0.0439 GY   0.49
     10                            -0.0732 GY   0.79
     10                            -0.1025 GY   1.10
     10                            -0.1318 GY   1.41
     10                            -0.1611 GY   1.72
     10                            -0.1904 GY   2.04
     10                            -0.2197 GY   2.35
     10                            -0.2197 GY   2.65
     10                            -0.1904 GY   2.96
     10                            -0.1611 GY   3.28
     10                            -0.1318 GY   3.59
      A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.    4
     10                            -0.1025 GY   3.90
     10                            -0.0732 GY   4.21
     10                            -0.0439 GY   4.51
     10                            -0.0146 GY   4.79
     18                            -0.0146 GY   0.21
     18                            -0.0439 GY   0.49
     18                            -0.0732 GY   0.79
     18                            -0.1025 GY   1.10
     18                            -0.1318 GY   1.41
     18                            -0.1611 GY   1.72
     18                            -0.1904 GY   2.04
     18                            -0.2197 GY   2.35
     18    -0.7500 GY   2.50   22.50
     18                            -0.2197 GY  22.65
     18                            -0.1904 GY  22.96
     18                            -0.1611 GY  23.28
     18                            -0.1318 GY  23.59
     18                            -0.1025 GY  23.90
     18                            -0.0732 GY  24.21
     18                            -0.0439 GY  24.51
     18                            -0.0146 GY  24.79
      2                            -0.0146 GY   0.21
      2                            -0.0439 GY   0.49
```

```
      2                            -0.0732 GY   0.79
      2                            -0.1025 GY   1.10
      2                            -0.1318 GY   1.41
      2                            -0.1611 GY   1.72
      2                            -0.1904 GY   2.04
      2                            -0.2197 GY   2.35
      2                            -0.2197 GY   2.65
      2                            -0.1904 GY   2.96
      2                            -0.1611 GY   3.28
      2                            -0.1318 GY   3.59
      2                            -0.1025 GY   3.90
      2                            -0.0732 GY   4.21
      2                            -0.0439 GY   4.51
      2                            -0.0146 GY   4.79
     22                            -0.0146 GY   0.21
     22                            -0.0439 GY   0.49
     22                            -0.0732 GY   0.79
     22                            -0.1025 GY   1.10
     22                            -0.1318 GY   1.41
     22                            -0.1611 GY   1.72
     22                            -0.1904 GY   2.04
     22                            -0.2197 GY   2.35
     22    -0.7500 GY   2.50    7.50
     22                            -0.2197 GY   7.65
     22                            -0.1904 GY   7.96
     22                            -0.1611 GY   8.28
     22                            -0.1318 GY   8.59
     22                            -0.1025 GY   8.90
     22                            -0.0732 GY   9.21
     22                            -0.0439 GY   9.51
     22                            -0.0146 GY   9.79
      5                            -0.0146 GY   0.21
      5                            -0.0439 GY   0.49
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    5
      5                            -0.0732 GY   0.79
      5                            -0.1025 GY   1.10
      5                            -0.1318 GY   1.41
      5                            -0.1611 GY   1.72
      5                            -0.1904 GY   2.04
      5                            -0.2197 GY   2.35
      5                            -0.2197 GY   2.65
      5                            -0.1904 GY   2.96
      5                            -0.1611 GY   3.28
      5                            -0.1318 GY   3.59
      5                            -0.1025 GY   3.90
      5                            -0.0732 GY   4.21
      5                            -0.0439 GY   4.51
      5                            -0.0146 GY   4.79
     20                            -0.0146 GY   0.21
     20                            -0.0439 GY   0.49
     20                            -0.0732 GY   0.79
     20                            -0.1025 GY   1.10
     20                            -0.1318 GY   1.41
     20                            -0.1611 GY   1.72
     20                            -0.1904 GY   2.04
     20                            -0.2197 GY   2.35
     20    -0.7500 GY   2.50    7.50
     20                            -0.2197 GY   7.65
```

```
     20                            -0.1904 GY   7.96
     20                            -0.1611 GY   8.28
     20                            -0.1318 GY   8.59
     20                            -0.1025 GY   8.90
     20                            -0.0732 GY   9.21
     20                            -0.0439 GY   9.51
     20                            -0.0146 GY   9.79
      3                            -0.0146 GY   0.21
      3                            -0.0439 GY   0.49
      3                            -0.0732 GY   0.79
      3                            -0.1025 GY   1.10
      3                            -0.1318 GY   1.41
      3                            -0.1611 GY   1.72
      3                            -0.1904 GY   2.04
      3                            -0.2197 GY   2.35
      3                            -0.2197 GY   2.65
      3                            -0.1904 GY   2.96
      3                            -0.1611 GY   3.28
      3                            -0.1318 GY   3.59
      3                            -0.1025 GY   3.90
      3                            -0.0732 GY   4.21
      3                            -0.0439 GY   4.51
      3                            -0.0146 GY   4.79
     24                            -0.0146 GY   0.21
     24                            -0.0439 GY   0.49
     24                            -0.0732 GY   0.79
     24                            -0.1025 GY   1.10
     24                            -0.1318 GY   1.41
     24                            -0.1611 GY   1.72
     24                            -0.1904 GY   2.04
     24                            -0.2197 GY   2.35
     24    -0.7500 GY   2.50    7.50
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    6
     24                            -0.2197 GY   7.65
     24                            -0.1904 GY   7.96
     24                            -0.1611 GY   8.28
     24                            -0.1318 GY   8.59
     24                            -0.1025 GY   8.90
     24                            -0.0732 GY   9.21
     24                            -0.0439 GY   9.51
     24                            -0.0146 GY   9.79
      7                            -0.1406 GY   0.12
      7                            -0.1219 GY   0.37
      7                            -0.1031 GY   0.62
      7                            -0.0844 GY   0.87
      7                            -0.0656 GY   1.12
      7                            -0.0469 GY   1.37
      7                            -0.0281 GY   1.61
      7                            -0.0094 GY   1.83
      6                            -0.0146 GY   0.21
      6                            -0.0439 GY   0.49
      6                            -0.0732 GY   0.79
      6                            -0.1025 GY   1.10
      6                            -0.1318 GY   1.41
      6                            -0.1611 GY   1.72
      6                            -0.1904 GY   2.04
      6                            -0.2197 GY   2.35
      6                            -0.0463 GY   2.53
```

```
      6                           -0.0451 GY   2.59
      6                           -0.0439 GY   2.66
      6                           -0.0428 GY   2.72
      6                           -0.0416 GY   2.78
      6                           -0.0404 GY   2.84
      6                           -0.0393 GY   2.91
      6                           -0.0381 GY   2.97
     22                           -0.0146 GY   0.21
     22                           -0.0439 GY   0.49
     22                           -0.0732 GY   0.79
     22                           -0.1025 GY   1.10
     22                           -0.1318 GY   1.41
     22                           -0.1611 GY   1.72
     22                           -0.1904 GY   2.04
     22                           -0.2197 GY   2.35
     22    -0.7500 GY   2.50    7.50
     22                           -0.2197 GY   7.65
     22                           -0.1904 GY   7.96
     22                           -0.1611 GY   8.28
     22                           -0.1318 GY   8.59
     22                           -0.1025 GY   8.90
     22                           -0.0732 GY   9.21
     22                           -0.0439 GY   9.51
     22                           -0.0146 GY   9.79
      4                           -0.0146 GY   0.21
      4                           -0.0439 GY   0.49
      4                           -0.0732 GY   0.79
      4                           -0.1025 GY   1.10
      4                           -0.1318 GY   1.41
      4                           -0.1611 GY   1.72
      4                           -0.1904 GY   2.04
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    7
      4                           -0.2197 GY   2.35
      4                           -0.2197 GY   2.65
      4                           -0.1904 GY   2.96
      4                           -0.1611 GY   3.28
      4                           -0.1318 GY   3.59
      4                           -0.1025 GY   3.90
      4                           -0.0732 GY   4.21
      4                           -0.0439 GY   4.51
      4                           -0.0146 GY   4.79
     28                           -0.0146 GY   0.21
     28                           -0.0439 GY   0.49
     28                           -0.0732 GY   0.79
     28                           -0.1025 GY   1.10
     28                           -0.1318 GY   1.41
     28                           -0.1611 GY   1.72
     28                           -0.1904 GY   2.04
     28                           -0.2197 GY   2.35
     28    -0.7500 GY   2.50    7.50
     28                           -0.2197 GY   7.65
     28                           -0.1904 GY   7.96
     28                           -0.1611 GY   8.28
     28                           -0.1318 GY   8.59
     28                           -0.1025 GY   8.90
     28                           -0.0732 GY   9.21
     28                           -0.0439 GY   9.51
     28                           -0.0146 GY   9.79
```

```
    9                            -0.0586 GY   0.06
    9                            -0.0633 GY   0.19
    9                            -0.0680 GY   0.31
    9                            -0.0727 GY   0.44
    9                            -0.0773 GY   0.56
    9                            -0.0820 GY   0.69
    9                            -0.0867 GY   0.81
    9                            -0.0914 GY   0.94
    9                            -0.2197 GY   1.15
    9                            -0.1904 GY   1.46
    9                            -0.1611 GY   1.78
    9                            -0.1318 GY   2.09
    9                            -0.1025 GY   2.40
    9                            -0.0732 GY   2.71
    9                            -0.0439 GY   3.01
    9                            -0.0146 GY   3.29
    8                            -0.0053 GY   0.12
    8                            -0.0158 GY   0.29
    8                            -0.0264 GY   0.47
    8                            -0.0369 GY   0.66
    8                            -0.0475 GY   0.85
    8                            -0.0580 GY   1.03
    8                            -0.0686 GY   1.22
    8                            -0.0791 GY   1.41
   24                            -0.0146 GY   0.21
   24                            -0.0439 GY   0.49
   24                            -0.0732 GY   0.79
   24                            -0.1025 GY   1.10
   24                            -0.1318 GY   1.41
   24                            -0.1611 GY   1.72
    A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    8
   24                            -0.1904 GY   2.04
   24                            -0.2197 GY   2.35
   24    -0.7500 GY   2.50    7.50
   24                            -0.2197 GY   7.65
   24                            -0.1904 GY   7.96
   24                            -0.1611 GY   8.28
   24                            -0.1318 GY   8.59
   24                            -0.1025 GY   8.90
   24                            -0.0732 GY   9.21
   24                            -0.0439 GY   9.51
   24                            -0.0146 GY   9.79
   21                            -0.0628 GY   0.64
   21                            -0.1884 GY   1.49
   21                            -0.3140 GY   2.43
   21                            -0.4396 GY   3.38
   21                            -0.5652 GY   4.33
   21                            -0.6909 GY   5.29
   21                            -0.8165 GY   6.25
   21                            -0.9421 GY   7.20
   21                            -0.9162 GY   8.13
   21                            -0.7940 GY   9.06
   21                            -0.6719 GY   9.99
   21                            -0.5497 GY  10.92
   21                            -0.4276 GY  11.85
   21                            -0.3054 GY  12.77
   21                            -0.1832 GY  13.68
   21                            -0.0611 GY  14.51
```

```
     11                            -0.0649 GY   0.29
     11                            -0.1947 GY   0.69
     11                            -0.3245 GY   1.12
     11                            -0.4542 GY   1.55
     11                            -0.5840 GY   1.99
     11                            -0.7138 GY   2.43
     11                            -0.8436 GY   2.87
     11                            -0.9734 GY   3.31
     11                            -0.6840 GY   3.68
     11                            -0.5928 GY   3.98
     11                            -0.5016 GY   4.29
     11                            -0.4104 GY   4.60
     11                            -0.3192 GY   4.91
     11                            -0.2280 GY   5.22
     11                            -0.1368 GY   5.52
     11                            -0.0456 GY   5.79
     19                            -0.0590 GY   0.60
     19                            -0.1770 GY   1.39
     19                            -0.2950 GY   2.26
     19                            -0.4129 GY   3.15
     19                            -0.5309 GY   4.03
     19                            -0.6489 GY   4.92
     19                            -0.7669 GY   5.82
     19                            -0.8849 GY   6.71
     19                            -0.9734 GY   7.62
     19                            -0.8436 GY   8.60
     19                            -0.7138 GY   9.58
     19                            -0.5840 GY  10.56
     19                            -0.4542 GY  11.54
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    9
     19                            -0.3245 GY  12.51
     19                            -0.1947 GY  13.47
     19                            -0.0649 GY  14.35
      6                            -0.5049 GY   0.18
      6                            -0.4376 GY   0.56
      6                            -0.3703 GY   0.93
      6                            -0.3029 GY   1.31
      6                            -0.2356 GY   1.68
      6                            -0.1683 GY   2.05
      6                            -0.1010 GY   2.42
      6                            -0.0337 GY   2.75
      5                            -0.0590 GY   0.29
      5                            -0.1770 GY   0.69
      5                            -0.2950 GY   1.12
      5                            -0.4129 GY   1.55
      5                            -0.5309 GY   1.99
      5                            -0.6489 GY   2.43
      5                            -0.7669 GY   2.87
      5                            -0.8849 GY   3.31
      5                            -0.3873 GY   3.62
      5                            -0.3710 GY   3.80
      5                            -0.3547 GY   3.98
      5                            -0.3384 GY   4.17
      5                            -0.3221 GY   4.35
      5                            -0.3058 GY   4.54
      5                            -0.2895 GY   4.72
      5                            -0.2732 GY   4.91
     23                            -0.0432 GY   0.67
```

```
      23                            -0.1295 GY   1.57
      23                            -0.2159 GY   2.55
      23                            -0.3023 GY   3.55
      23                            -0.3886 GY   4.55
      23                            -0.4750 GY   5.55
      23                            -0.5613 GY   6.56
      23                            -0.6477 GY   7.56
      23                            -0.5584 GY   8.48
      23                            -0.4839 GY   9.35
      23                            -0.4095 GY  10.21
      23                            -0.3350 GY  11.08
      23                            -0.2606 GY  11.94
      23                            -0.1861 GY  12.80
      23                            -0.1117 GY  13.65
      23                            -0.0372 GY  14.42
      12                            -0.0523 GY   0.27
      12                            -0.1569 GY   0.62
      12                            -0.2615 GY   1.02
      12                            -0.3661 GY   1.42
      12                            -0.4706 GY   1.81
      12                            -0.5752 GY   2.21
      12                            -0.6798 GY   2.62
      12                            -0.7844 GY   3.02
      12                            -0.5584 GY   3.35
      12                            -0.4839 GY   3.64
      12                            -0.4095 GY   3.92
      12                            -0.3350 GY   4.21
      12                            -0.2606 GY   4.49
       A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   10
      12                            -0.1861 GY   4.78
      12                            -0.1117 GY   5.06
      12                            -0.0372 GY   5.31
      21                            -0.0416 GY   0.65
      21                            -0.1247 GY   1.52
      21                            -0.2079 GY   2.48
      21                            -0.2910 GY   3.45
      21                            -0.3742 GY   4.42
      21                            -0.4573 GY   5.39
      21                            -0.5405 GY   6.37
      21                            -0.6236 GY   7.35
      21                            -0.5824 GY   8.27
      21                            -0.5048 GY   9.18
      21                            -0.4271 GY  10.09
      21                            -0.3495 GY  11.00
      21                            -0.2718 GY  11.91
      21                            -0.1941 GY  12.82
      21                            -0.1165 GY  13.71
      21                            -0.0388 GY  14.52
       8                            -0.2786 GY   0.09
       8                            -0.2415 GY   0.28
       8                            -0.2043 GY   0.47
       8                            -0.1672 GY   0.65
       8                            -0.1300 GY   0.84
       8                            -0.0929 GY   1.02
       8                            -0.0557 GY   1.21
       8                            -0.0186 GY   1.38
       7                            -0.0229 GY   0.10
       7                            -0.0687 GY   0.24
```

```
      7                           -0.1145 GY   0.38
      7                           -0.1603 GY   0.53
      7                           -0.2061 GY   0.69
      7                           -0.2519 GY   0.84
      7                           -0.2977 GY   0.99
      7                           -0.3435 GY   1.14
      7                           -0.2326 GY   1.26
      7                           -0.2224 GY   1.36
      7                           -0.2122 GY   1.46
      7                           -0.2020 GY   1.56
      7                           -0.1917 GY   1.66
      7                           -0.1815 GY   1.75
      7                           -0.1713 GY   1.85
      7                           -0.1610 GY   1.95
      9                           -0.0072 GY   0.15
      9                           -0.0215 GY   0.34
      9                           -0.0359 GY   0.55
      9                           -0.0502 GY   0.77
      9                           -0.0646 GY   0.99
      9                           -0.0790 GY   1.21
      9                           -0.0933 GY   1.42
      9                           -0.1077 GY   1.64
      9                           -0.1077 GY   1.86
      9                           -0.0933 GY   2.08
      9                           -0.0790 GY   2.29
      9                           -0.0646 GY   2.51
      9                           -0.0502 GY   2.73
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   11
      9                           -0.0359 GY   2.95
      9                           -0.0215 GY   3.16
      9                           -0.0072 GY   3.35
     27                           -0.0072 GY   0.15
     27                           -0.0215 GY   0.34
     27                           -0.0359 GY   0.55
     27                           -0.0502 GY   0.77
     27                           -0.0646 GY   0.99
     27                           -0.0790 GY   1.21
     27                           -0.0933 GY   1.42
     27                           -0.1077 GY   1.64
     27    -0.5250 GY   1.75   13.25
     27                           -0.1077 GY  13.36
     27                           -0.0933 GY  13.58
     27                           -0.0790 GY  13.79
     27                           -0.0646 GY  14.01
     27                           -0.0502 GY  14.23
     27                           -0.0359 GY  14.45
     27                           -0.0215 GY  14.66
     27                           -0.0072 GY  14.85
     13                           -0.0072 GY   0.15
     13                           -0.0215 GY   0.34
     13                           -0.0359 GY   0.55
     13                           -0.0502 GY   0.77
     13                           -0.0646 GY   0.99
     13                           -0.0790 GY   1.21
     13                           -0.0933 GY   1.42
     13                           -0.1077 GY   1.64
     13                           -0.1077 GY   1.86
     13                           -0.0933 GY   2.08
```

```
     13                            -0.0790 GY   2.29
     13                            -0.0646 GY   2.51
     13                            -0.0502 GY   2.73
     13                            -0.0359 GY   2.95
     13                            -0.0215 GY   3.16
     13                            -0.0072 GY   3.35
     23                            -0.0072 GY   0.15
     23                            -0.0215 GY   0.34
     23                            -0.0359 GY   0.55
     23                            -0.0502 GY   0.77
     23                            -0.0646 GY   0.99
     23                            -0.0790 GY   1.21
     23                            -0.0933 GY   1.42
     23                            -0.1077 GY   1.64
     23    -0.5250 GY   1.75   13.25
     23                            -0.1077 GY  13.36
     23                            -0.0933 GY  13.58
     23                            -0.0790 GY  13.79
     23                            -0.0646 GY  14.01
     23                            -0.0502 GY  14.23
     23                            -0.0359 GY  14.45
     23                            -0.0215 GY  14.66
     23                            -0.0072 GY  14.85
     26                            -0.0053 GY   0.12
     26                            -0.0158 GY   0.29
     26                            -0.0264 GY   0.47
      A FLOOR FRAME DESIGN WITH AREA LOAD                         -- PAGE NO.   12
     26                            -0.0369 GY   0.66
     26                            -0.0475 GY   0.85
     26                            -0.0580 GY   1.03
     26                            -0.0686 GY   1.22
     26                            -0.0791 GY   1.41
     26                            -0.0791 GY   1.59
     26                            -0.0686 GY   1.78
     26                            -0.0580 GY   1.97
     26                            -0.0475 GY   2.15
     26                            -0.0369 GY   2.34
     26                            -0.0264 GY   2.53
     26                            -0.0158 GY   2.71
     26                            -0.0053 GY   2.88
     14                            -0.0053 GY   0.12
     14                            -0.0158 GY   0.29
     14                            -0.0264 GY   0.47
     14                            -0.0369 GY   0.66
     14                            -0.0475 GY   0.85
     14                            -0.0580 GY   1.03
     14                            -0.0686 GY   1.22
     14                            -0.0791 GY   1.41
     14    -0.4500 GY   1.50   18.50
     14                            -0.0791 GY  18.59
     14                            -0.0686 GY  18.78
     14                            -0.0580 GY  18.97
     14                            -0.0475 GY  19.15
     14                            -0.0369 GY  19.34
     14                            -0.0264 GY  19.52
     14                            -0.0158 GY  19.71
     14                            -0.0053 GY  19.88
     17                            -0.0053 GY   0.12
```

```
     17                            -0.0158 GY   0.29
     17                            -0.0264 GY   0.47
     17                            -0.0369 GY   0.66
     17                            -0.0475 GY   0.85
     17                            -0.0580 GY   1.03
     17                            -0.0686 GY   1.22
     17                            -0.0791 GY   1.41
     17                            -0.0791 GY   1.59
     17                            -0.0686 GY   1.78
     17                            -0.0580 GY   1.97
     17                            -0.0475 GY   2.15
     17                            -0.0369 GY   2.34
     17                            -0.0264 GY   2.53
     17                            -0.0158 GY   2.71
     17                            -0.0053 GY   2.88
     13    -0.4500 GY   0.00    2.00
     13                            -0.0791 GY   2.09
     13                            -0.0686 GY   2.28
     13                            -0.0580 GY   2.47
     13                            -0.0475 GY   2.65
     13                            -0.0369 GY   2.84
     13                            -0.0264 GY   3.03
     13                            -0.0158 GY   3.21
     13                            -0.0053 GY   3.38
     10    -0.4500 GY   1.50    5.00
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   13
     10                            -0.0053 GY   0.12
     10                            -0.0158 GY   0.29
     10                            -0.0264 GY   0.47
     10                            -0.0369 GY   0.66
     10                            -0.0475 GY   0.85
     10                            -0.0580 GY   1.03
     10                            -0.0686 GY   1.22
     10                            -0.0791 GY   1.41
     11    -0.4500 GY   0.00    6.00
     12    -0.4500 GY   0.00    5.50
     14                            -0.0287 GY   0.29
     14                            -0.0861 GY   0.68
     14                            -0.1436 GY   1.11
     14                            -0.2010 GY   1.54
     14                            -0.2584 GY   1.98
     14                            -0.3158 GY   2.41
     14                            -0.3732 GY   2.85
     14                            -0.4307 GY   3.29
     14    -1.0500 GY   3.50   16.50
     14                            -0.4307 GY  16.71
     14                            -0.3732 GY  17.15
     14                            -0.3158 GY  17.59
     14                            -0.2584 GY  18.02
     14                            -0.2010 GY  18.46
     14                            -0.1436 GY  18.89
     14                            -0.0861 GY  19.32
     14                            -0.0287 GY  19.71
     25                            -0.0287 GY   0.29
     25                            -0.0861 GY   0.68
     25                            -0.1436 GY   1.11
     25                            -0.2010 GY   1.54
     25                            -0.2584 GY   1.98
```

```
     25                            -0.3158 GY   2.41
     25                            -0.3732 GY   2.85
     25                            -0.4307 GY   3.29
     25                            -0.4307 GY   3.71
     25                            -0.3732 GY   4.15
     25                            -0.3158 GY   4.59
     25                            -0.2584 GY   5.02
     25                            -0.2010 GY   5.46
     25                            -0.1436 GY   5.89
     25                            -0.0861 GY   6.32
     25                            -0.0287 GY   6.71
     15                            -0.0287 GY   0.29
     15                            -0.0861 GY   0.68
     15                            -0.1436 GY   1.11
     15                            -0.2010 GY   1.54
     15                            -0.2584 GY   1.98
     15                            -0.3158 GY   2.41
     15                            -0.3732 GY   2.85
     15                            -0.4307 GY   3.29
     15    -1.0500 GY   3.50   16.50
     15                            -0.4307 GY  16.71
     15                            -0.3732 GY  17.15
     15                            -0.3158 GY  17.59
     15                            -0.2584 GY  18.02
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   14
     15                            -0.2010 GY  18.46
     15                            -0.1436 GY  18.89
     15                            -0.0861 GY  19.32
     15                            -0.0287 GY  19.71
     16                            -0.0287 GY   0.29
     16                            -0.0861 GY   0.68
     16                            -0.1436 GY   1.11
     16                            -0.2010 GY   1.54
     16                            -0.2584 GY   1.98
     16                            -0.3158 GY   2.41
     16                            -0.3732 GY   2.85
     16                            -0.4307 GY   3.29
     16                            -0.4307 GY   3.71
     16                            -0.3732 GY   4.15
     16                            -0.3158 GY   4.59
     16                            -0.2584 GY   5.02
     16                            -0.2010 GY   5.46
     16                            -0.1436 GY   5.89
     16                            -0.0861 GY   6.32
     16                            -0.0287 GY   6.71
   ************ END OF DATA FROM INTERNAL STORAGE ************
    41. PARAMETERS
    42. CODE AISC UNIFIED
    43. BEAM 1.0 ALL
    44. DMAX 2.0 ALL
    45. DMIN 1.0 ALL
    46. UNT 1.0 ALL
    47. UNB 1.0 ALL
    48. SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
 PARAMETER 1
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   15
                     STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                   *******************************************************
```

```
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections principal
major axis.
       Y               Y          Axis typically parallel to the sections principal
minor axis.
       Z               X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), while
       the results for shear are reported as being for shear forces �along� the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   16
                     STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                   ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     2
|-----------------------------------------------------------------------------|
|  Member No:        2       Profile:  ST  W16X31             (AISC SECTIONS)|
|  Status:       PASS       Ratio:         0.958       Loadcase:        1     |
|  Location:     0.00       Ref:      Cl.F2.1                                 |
|  Pz:      0.000     T     Vy:       -.8750           Vx:      0.000         |
|  Tz:      0.000           My:        0.000           Mx:     -139.7         |
|-----------------------------------------------------------------------------|
 - Member :     6
|-----------------------------------------------------------------------------|
```

```
|  Member No:        6       Profile:  ST  W16X26             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.862      Loadcase:        1    |
|  Location:      3.00       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:        3.529          Vx:      0.000        |
|  Tz:      0.6495E-02       My:        0.000          Mx:     -102.8        |
|----------------------------------------------------------------------------|
- Member :    11
|----------------------------------------------------------------------------|
|  Member No:       11       Profile:  ST  W18X35             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.960      Loadcase:        1    |
|  Location:      3.50       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:       0.4731          Vx:      0.000        |
|  Tz:      0.7231E-03       My:        0.000          Mx:     -172.4        |
|----------------------------------------------------------------------------|
- Member :    14
|----------------------------------------------------------------------------|
|  Member No:       14       Profile:  ST  W12X22             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.919      Loadcase:        1    |
|  Location:     10.00       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:        0.000          Vx:      0.000        |
|  Tz:      0.2891E-18       My:        0.000          Mx:     -72.69        |
|----------------------------------------------------------------------------|
- Member :    15
|----------------------------------------------------------------------------|
|  Member No:       15       Profile:  ST  W12X19             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.755      Loadcase:        1    |
|  Location:     10.00       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:        0.000          Vx:      0.000        |
|  Tz:       0.000           My:        0.000          Mx:     -50.36        |
|----------------------------------------------------------------------------|
- Member :    16
|----------------------------------------------------------------------------|
|  Member No:       16       Profile:  ST  W12X16             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.580      Loadcase:        1    |
|  Location:      0.00       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:       -2.663          Vx:      0.000        |
|  Tz:       0.000           My:        0.000          Mx:     -31.50        |
|----------------------------------------------------------------------------|
- Member :    18
|----------------------------------------------------------------------------|
|  Member No:       18       Profile:  ST  W14X22             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.645      Loadcase:        1    |
|  Location:     12.50       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:        0.000          Vx:      0.000        |
|  Tz:       0.000           My:        0.000          Mx:     -57.81        |
|----------------------------------------------------------------------------|
     A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.   17
                      STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                    ********************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :    19
|----------------------------------------------------------------------------|
|  Member No:       19       Profile:  ST  W21X44             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.861      Loadcase:        1    |
|  Location:      0.00       Ref:      Cl.F2.1                               |
|  Pz:       0.000     T     Vy:       -5.183          Vx:      0.000        |
|  Tz:      0.4163E-02       My:        0.000          Mx:     -221.8        |
|----------------------------------------------------------------------------|
```

```
 - Member :    21
|-----------------------------------------------------------------------------|
|  Member No:       21       Profile:  ST  W12X16             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.608       Loadcase:        1    |
|  Location:      7.57       Ref:      Cl.F2.1                                |
|  Pz:       0.000     T     Vy:       -.1907           Vx:      0.000        |
|  Tz:      0.1216E-01       My:        0.000           Mx:     -32.99        |
|-----------------------------------------------------------------------------|
 - Member :    23
|-----------------------------------------------------------------------------|
|  Member No:       23       Profile:  ST  W12X16             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.505       Loadcase:        1    |
|  Location:      7.50       Ref:      Cl.F2.1                                |
|  Pz:       0.000     T     Vy:       0.3936           Vx:      0.000        |
|  Tz:      -.8275E-03       My:        0.000           Mx:     -27.39        |
|-----------------------------------------------------------------------------|
 - Member :    24
|-----------------------------------------------------------------------------|
|  Member No:       24       Profile:  ST  W12X16             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.317       Loadcase:        1    |
|  Location:      5.00       Ref:      Cl.F2.1                                |
|  Pz:       0.000     T     Vy:       0.9537E-06       Vx:      0.000        |
|  Tz:      -.1067E-01       My:        0.000           Mx:     -17.19        |
|-----------------------------------------------------------------------------|
 - Member :    27
|-----------------------------------------------------------------------------|
|  Member No:       27       Profile:  ST  W18X35             (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.952       Loadcase:        1    |
|  Location:      0.00       Ref:      Cl.F2.1                                |
|  Pz:       0.000     T     Vy:       -7.917           Vx:      0.000        |
|  Tz:       0.000           My:        0.000           Mx:     -170.9        |
|-----------------------------------------------------------------------------|
    49. FINISH
  ****************************************************************************
  **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
              IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
              ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
              TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
              FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
              FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
              COMMANDS.
  ****************************************************************************
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:56 ****
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   18
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-3 Soil Springs for Portal Frame

A portal frame type steel structure is sitting on concrete footings. The soil is to be considered as an elastic foundation. The value of soil subgrade reaction is known from which spring constants are calculated by multiplying the subgrade reaction by the tributary area of each modeled spring.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-3 Soil Springs for Portal Frame.STD` when you install the program.

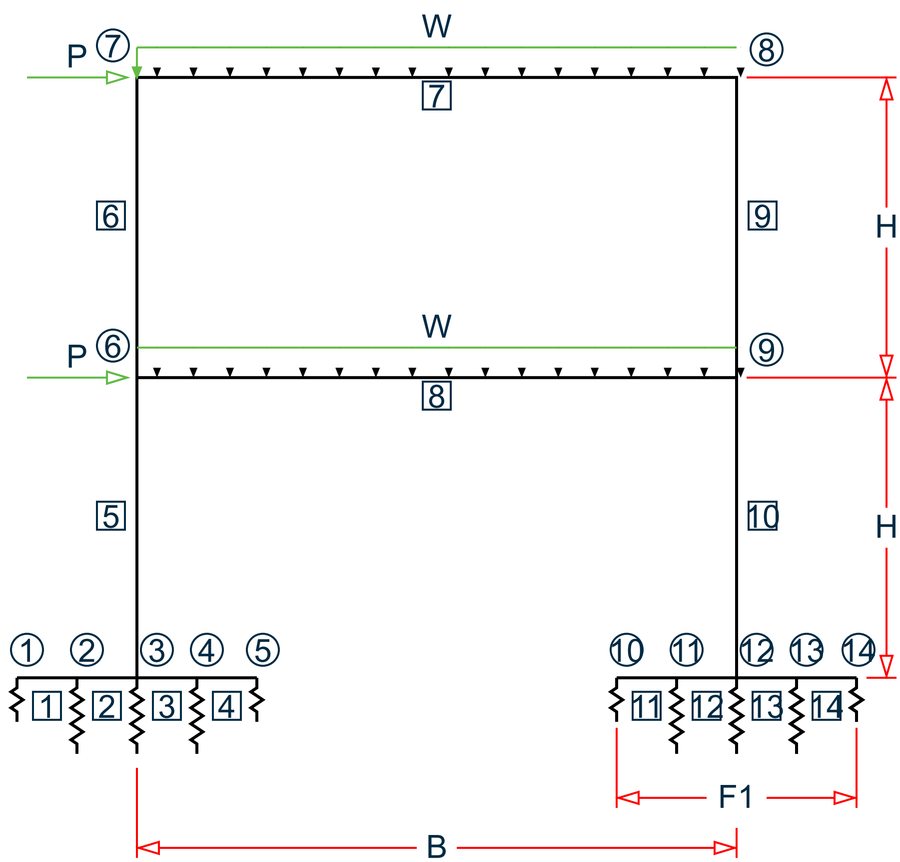


Figure 514: Example Problem No. 3

Where:

B = 20 ft, H = 10 ft, F1 = 4 ft, F2 = 8 ft
P = 5 kips, w = 3 kips/ft
Soil Subgrade Reaction = 250 kips/ft^3

Table 909: Spring constant calculation

Springs of Joints	Spring Constant
1, 5, 10, & 14 (Edges)	8 x 1 x 250 = 2,000 kips/ft
2, 3, 4, 11, 12, & 13 (Interior)	8 x 2 x 250 = 4,000 kips/ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE PORTAL ON FOOTING FOUNDATION
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT FT KIPS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1   0.0  0.0  0.0  5  8.0  0.0 0.0
6   4.0 10.0 0.0 ; 7  4.0 20.0 0.0
8  24.0 20.0 0.0 ; 9  24.0 10.0 0.0
10 20.0 0.0 0.0 14 28.0 0.0 0.0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ; 10 9 12
11 10 11 14
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTIES AMERICAN
1 4 11 14 PRIS YD 1.0 ZD 8.0
2 3 12 13 PRIS YD 2.0 ZD 8.0
5 6 9 10 TA ST W10X33
7 8 TA ST W12X26
```

The first two lines define member properties as prismatic (PRIS) followed by depth (YD) and width (ZD) values. The program will calculate the properties necessary to perform the analysis. See G.6.1 Prismatic Properties (on page 2114) for additional information. Member properties for the remaining members are chosen from the British steel table. The term ST stands for standard single section.

```
* E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
```

```
    POISSON 0.3
    DENSITY 0.283e-003
    ALPHA 6e-006
    DAMP 0.03
    TYPE STEEL
    STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    ISOTROPIC CONCRETE
    E 3000
    POISSON 0.17
    DENSITY 8.68e-005
    ALPHA 5e-006
    DAMP 0.05
    G 1346.15
    TYPE CONCRETE
    STRENGTH FCU 4
    END DEFINE MATERIAL
    CONSTANTS
    MATERIAL STEEL MEMB 5 TO 10
    MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
```

The `CONSTANT` command initiates input for material constants like modulus of elasticity, Density, and Poisson's ratio. The length unit is changed from `FT` to `INCH` to facilitate the input in familiar units.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
    UNIT FT
    SUPPORTS
    2 TO 4  11  TO  13  FIXED BUT MZ KFY 4000.
    1  5  10  14  FIXED BUT MZ KFY 2000.
```

The supports for the structure are specified above. The first set of joints are supports restrained in all directions except global moment-z (`MZ`). Also, a spring having a spring constant of 4,000 kip/ft is provided in the global Y direction at these nodes. The second set is similar to the former except for a different value of the spring constant.

```
 LOADING 1 DEAD AND WIND LOAD COMBINED
```

Load case 1 is initiated followed by a title.

```
 SELF Y -1.0
```

The selfweight of the structure is specified as acting in the global Y direction with a -1.0 factor.

Since global Y is vertically upwards, the -1.0 factor indicates that this load will act downwards.

```
    JOINT LOAD
    6 7  FX  5.0
```

Load 1 contains joint loads also. `FX` indicates that the load is a force in the global X direction.

```
    MEMBER LOAD
    7 8 UNI GY  -3.0
```

Load 1 contains member loads also. `GY` indicates that the load acts in the global Y direction. The term `UNI` stands for uniformly distributed load, and is applied on members 7 and 8, acting downwards.

```
    PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
    PRINT ANALYSIS RESULTS
```

The above PRINT command instructs the program to print analysis results which include joint displacements, member forces and support reactions.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE PORTAL ON FOOTING FOUNDATION
UNIT FT KIPS
JOINT COORDINATES
1   0.0  0.0  0.0  5  8.0  0.0 0.0
6   4.0 10.0 0.0 ; 7  4.0 20.0 0.0
8  24.0 20.0 0.0 ; 9  24.0 10.0 0.0
10 20.0 0.0 0.0 14 28.0 0.0 0.0
MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ;10 9 12
11 10 11 14
MEMBER PROPERTIES AMERICAN
1 4 11 14 PRIS YD 1.0 ZD 8.0
2 3 12 13 PRIS YD 2.0 ZD 8.0
5 6 9 10 TA ST W10X33
7 8 TA ST W12X26
* E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 0.283e-003
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3000
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 5 TO 10
MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
UNIT FT
SUPPORTS
2 TO 4  11 TO 13 FIXED BUT MZ KFY 4000.
1 5 10 14 FIXED BUT MZ KFY 2000.
LOADING 1 DEAD AND WIND LOAD COMBINED
SELF Y -1.0
JOINT LOAD
6 7 FX 5.0
```

```
MEMBER LOAD
7 8 UNI GY -3.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:28                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE PORTAL ON FOOTING FOUNDATION
INPUT FILE: US-3 Soil Springs for Portal Frame.STD
     2. UNIT FT KIPS
     3. JOINT COORDINATES
     4. 1   0.0  0.0  0.0  5  8.0  0.0 0.0
     5. 6   4.0 10.0 0.0 ; 7  4.0 20.0 0.0
     6. 8  24.0 20.0 0.0 ; 9  24.0 10.0 0.0
     7. 10 20.0 0.0 0.0 14 28.0 0.0 0.0
     8. MEMBER INCIDENCES
     9. 1 1 2 4
    10. 5 3 6 ; 6 6 7
    11. 7 7 8 ; 8 6 9
    12. 9 8 9 ;10 9 12
    13. 11 10 11 14
    14. MEMBER PROPERTIES AMERICAN
    15. 1 4 11 14 PRIS YD 1.0 ZD 8.0
    16. 2 3 12 13 PRIS YD 2.0 ZD 8.0
    17. 5 6 9 10 TA ST W10X33
    18. 7 8 TA ST W12X26
    19. * E FOR STEEL IS 29,000 AND FOR CONCRETE 3000
    20. UNIT INCHES
    21. DEFINE MATERIAL START
    22. ISOTROPIC STEEL
    23. E 29000
    24. POISSON 0.3
    25. DENSITY 0.283E-003
    26. ALPHA 6E-006
    27. DAMP 0.03
    28. TYPE STEEL
    29. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    30. ISOTROPIC CONCRETE
    31. E 3000
    32. POISSON 0.17
    33. DENSITY 8.68E-005
    34. ALPHA 5E-006
    35. DAMP 0.05
    36. G 1346.15
    37. TYPE CONCRETE
```

```
   38. STRENGTH FCU 4
     PORTAL ON FOOTING FOUNDATION                             -- PAGE NO.    2
   39. END DEFINE MATERIAL
   40. CONSTANTS
   41. MATERIAL STEEL MEMB 5 TO 10
   42. MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
   43. UNIT FT
   44. SUPPORTS
   45. 2 TO 4  11 TO 13 FIXED BUT MZ KFY 4000.
   46. 1 5 10 14 FIXED BUT MZ KFY 2000.
   47. LOADING 1 DEAD AND WIND LOAD COMBINED
   48. SELF Y -1.0
   49. JOINT LOAD
   50. 6 7 FX 5.0
   51. MEMBER LOAD
   52. 7 8 UNI GY -3.0
   53. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         14  NUMBER OF MEMBERS      14
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     10
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      32
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   54. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     PORTAL ON FOOTING FOUNDATION                             -- PAGE NO.    3
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000  -0.04256   0.00000   0.00000   0.00000  -0.00028
     2    1    0.00000  -0.04892   0.00000   0.00000   0.00000  -0.00023
     3    1    0.00000  -0.05439   0.00000   0.00000   0.00000  -0.00020
     4    1    0.00000  -0.05854   0.00000   0.00000   0.00000  -0.00017
     5    1    0.00000  -0.06134   0.00000   0.00000   0.00000  -0.00010
     6    1    0.32187  -0.07856   0.00000   0.00000   0.00000  -0.00483
     7    1    0.64023  -0.09085   0.00000   0.00000   0.00000  -0.00669
     8    1    0.62253  -0.10247   0.00000   0.00000   0.00000   0.00393
     9    1    0.32940  -0.08883   0.00000   0.00000   0.00000   0.00014
    10    1    0.00000  -0.03597   0.00000   0.00000   0.00000  -0.00055
    11    1    0.00000  -0.04884   0.00000   0.00000   0.00000  -0.00051
    12    1    0.00000  -0.06099   0.00000   0.00000   0.00000  -0.00049
    13    1    0.00000  -0.07163   0.00000   0.00000   0.00000  -0.00043
    14    1    0.00000  -0.08045   0.00000   0.00000   0.00000  -0.00035
     PORTAL ON FOOTING FOUNDATION                             -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KIPS FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     2    1      0.00     16.31      0.00      0.00      0.00      0.00
     3    1     -0.60     18.13      0.00      0.00      0.00      0.00
     4    1      0.00     19.51      0.00      0.00      0.00      0.00
    11    1      0.00     16.28      0.00      0.00      0.00      0.00
    12    1     -9.40     20.33      0.00      0.00      0.00      0.00
    13    1      0.00     23.88      0.00      0.00      0.00      0.00
     1    1      0.00      7.09      0.00      0.00      0.00      0.00
     5    1      0.00     10.22      0.00      0.00      0.00      0.00
```

```
   10    1      0.00       5.99      0.00      0.00      0.00      0.00
   14    1      0.00      13.41      0.00      0.00      0.00      0.00
    PORTAL ON FOOTING FOUNDATION                              -- PAGE NO.    5
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIPS FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1     1     1      0.00      7.09     0.00      0.00      0.00       0.00
                2      0.00     -4.69     0.00      0.00      0.00      11.79
    2     1     2      0.00     21.00     0.00      0.00      0.00     -11.79
                3      0.00    -16.20     0.00      0.00      0.00      48.99
    3     1     3      0.00    -22.54     0.00      0.00      0.00     -67.92
                4      0.00     27.34     0.00      0.00      0.00      18.05
    4     1     4      0.00     -7.82     0.00      0.00      0.00     -18.05
                5      0.00     10.22     0.00      0.00      0.00       0.00
    5     1     3     56.87      0.60     0.00      0.00      0.00      18.93
                6    -56.54     -0.60     0.00      0.00      0.00     -12.95
    6     1     6     29.01    -11.36     0.00      0.00      0.00     -50.40
                7    -28.68     11.36     0.00      0.00      0.00     -63.23
    7     1     7     16.36     28.68     0.00      0.00      0.00      63.23
                8    -16.36     31.84     0.00      0.00      0.00     -94.86
    8     1     6     -6.96     27.53     0.00      0.00      0.00      63.35
                9      6.96     32.99     0.00      0.00      0.00    -117.94
    9     1     8     31.84     16.36     0.00      0.00      0.00      94.86
                9    -32.17    -16.36     0.00      0.00      0.00      68.77
   10     1     9     65.16      9.40     0.00      0.00      0.00      49.17
               12    -65.49     -9.40     0.00      0.00      0.00      44.85
   11     1    10      0.00      5.99     0.00      0.00      0.00       0.00
               11      0.00     -3.59     0.00      0.00      0.00       9.59
   12     1    11      0.00     19.87     0.00      0.00      0.00      -9.59
               12      0.00    -15.07     0.00      0.00      0.00      44.54
   13     1    12      0.00    -30.08     0.00      0.00      0.00     -89.39
               13      0.00     34.88     0.00      0.00      0.00      24.42
   14     1    13      0.00    -11.01     0.00      0.00      0.00     -24.42
               14      0.00     13.41     0.00      0.00      0.00       0.00
    PORTAL ON FOOTING FOUNDATION                              -- PAGE NO.    6
 ************** END OF LATEST ANALYSIS RESULT **************
  55. FINISH
          *********** END OF THE STAAD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:46:29 ****
      ************************************************************
      *   For technical assistance on STAAD.Pro, please visit    *
      *   http://www.bentley.com/en/support/                     *
      *                                                          *
      *   Details about additional assistance from               *
      *   Bentley and Partners can be found at program menu      *
      *   Help->Technical Support                                *
      *                                                          *
      *           Copyright (c) Bentley Systems, Inc.            *
      *                http://www.bentley.com                    *
      ************************************************************
```

Related Links

- *M. To assign a spring support* (on page 821)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)

EX. US-4 Inactive Members in a Braced Frame

This example is a typical case of a load-dependent structure where the structural condition changes for different load cases. In this example, different bracing members are made inactive for different load cases. This is done to prevent these members from carrying any compressive forces.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-4 Inactive Members in a Braced Frame.STD` when you install the program.

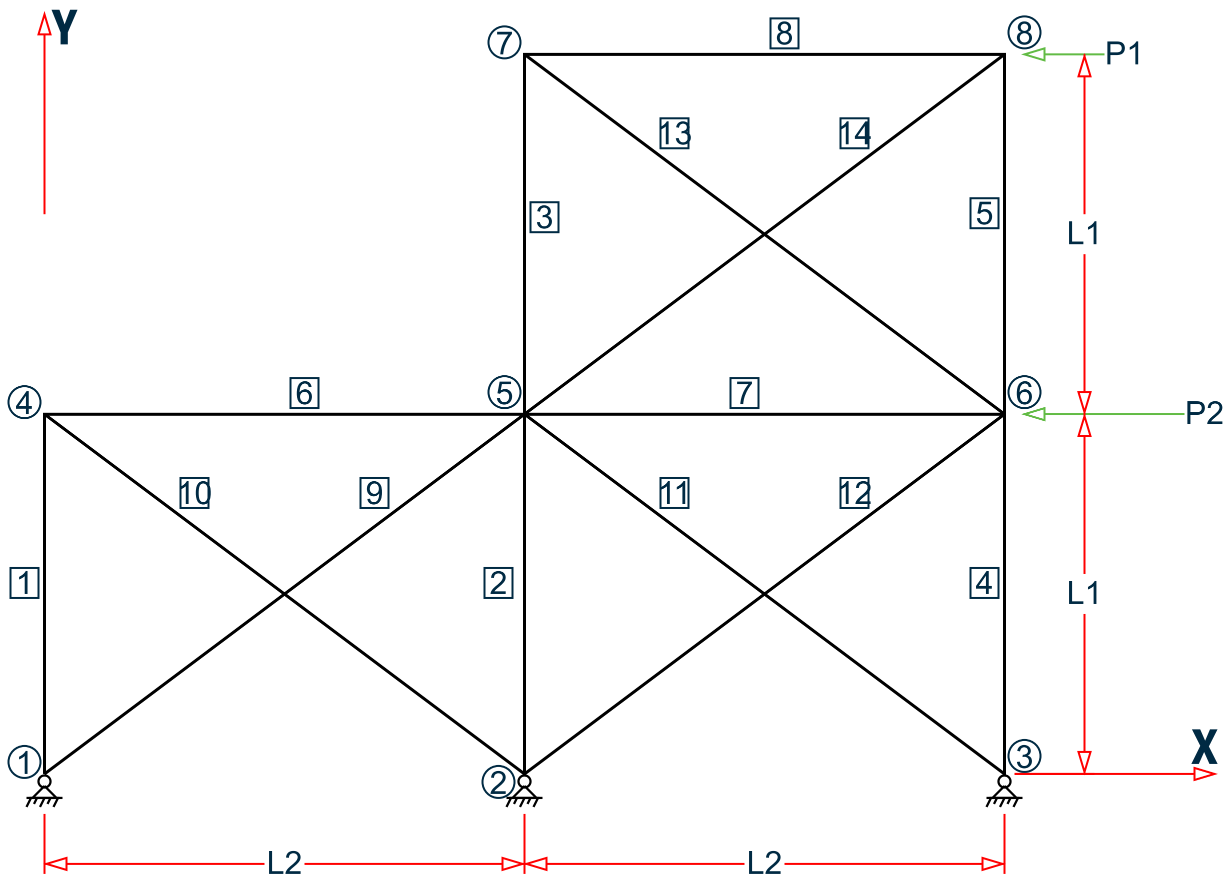


Figure 515: Example Problem No. 4

Where:

L1 = 180 in., L2 = 240 in.
P1 = 15 kips, P2 = 30 kips

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE
* A PLANE FRAME STRUCTURE WITH TENSION BRACING
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT INCH KIP
```

Defines the input units for the data that follows.

```
SET NL 3
```

This structure has to be analyzed for three primary load cases. Consequently, the modeling of our problem requires us to define three sets of data, with each set containing a load case and an associated analysis command. Also, the members which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have two commands, one called SET NL and the other called CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

```
JOINT COORDINATES
1  0  0  0  3  480.  0  0
4  0  180.  0  6  480.  180.  0
7  240.  360.  0  ;  8  480.  360.  0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ;  9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ; 14 5 8
```

Defines the members by the joints to which they are connected.

```
MEMBER TRUSS
9 TO 14
```

The preceding command defines that members 9 through 14 are of type truss. This means these members can only carry axial tension/compression and no moments.

```
MEMBER PROP AMERICAN
1 TO 5 TABLE ST W12X26
6  7  8  TA ST W18X35
9 TO 14 TA LD L50505
```

Properties for all members are assigned from the American (AISC) steel table. The word ST stands for standard single section. The word LD stands for long leg back-to-back double angle. Since the spacing between the two angles of the double angle is not provided, it is assumed to be 0.0.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

Define the material properties for steel. The units are changed to inches and then back to feet in order to facilite inputing the values in familiar units.

```
    CONSTANTS
   MATERIAL STEEL ALL
```

The `CONSTANT` command initiates input for material defintions.

```
    SUPPORT
    1 2 3 PINNED
```

Pinned supports are specified at Joints 1, 2 and 3. The word `PINNED` signifies that no moments will be carried by these supports.

```
INACTIVE MEMBERS 9 TO 14
```

The preceding command makes the listed members inactive. The stiffness contribution of these members will not be considered in the analysis till they are made active again.

```
    UNIT FT
    LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title. The length UNIT is changed from INCH to FT for input values which follow.

```
    MEMBER LOAD
    6 8 UNI GY -1.0
    7 UNI GY -1.5
```

Load 1 contains member loads. `GY` indicates that the load acts in the global Y direction. The word `UNI` stands for uniformly distributed load. The load is applied on members 6, 7, and 8.

```
    PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. It is worth noting that members 9 to 14 will not be used in this analysis since they were declared inactive earlier. In other words, for dead and live load, the bracing members are not used to carry any load.

```
    CHANGES
```

The members inactivated earlier are restored using the `CHANGE` command.

```
    INACTIVE MEMBERS 10 11 13
```

A new set of members are made inactive. The stiffness contribution from these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
    LOADING 2 WIND FROM LEFT
```

Load case 2 is initiated followed by a title.

```
    JOINT LOAD
    4 FX 30 ; 7 FX 15
```

Load 2 contains joint loads. `FX` indicates that the load is a force in the global X direction. Nodes 4 and 7 are subjected to the loads.

```
    PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis will be performed for load case 2 only.

```
    CHANGE
```

The above CHANGE command is an instruction to re-activate all inactive members.

```
INACTIVE MEMBERS 9 12 14
```

Members 9, 12 and 14 are made inactive. The stiffness contribution of these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 3 WIND FROM RIGHT
```

Load case 3 is initiated followed by a title.

```
JOINT LOAD
6 FX -30 ; 8 FX -15
```

Load 3 contains joint loads at nodes 6 and 8. FX indicates that the load is a force in the global X direction. The negative numbers

(-30 and -15) indicate that the load is acting along the negative global X direction.

```
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
```

Load combination case 4 involves the algebraic summation of the results of load cases 1 and 2 after multiplying each by a factor of 0.75. For load combinations, the program simply gathers the results of the component primary cases, factors them appropriately, and combines them algebraically. Thus, an analysis in the real sense of the term (multiplying the inverted stiffness matrix by the load vector) is not carried out for load combination cases. Load combination case 5 combines the results of load cases 1 and 3.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. Only primary load case 3 will be considered for this analysis. (As explained earlier, a combination case is not truly analyzed for, but handled using other means.)

```
CHANGE
```

The above `CHANGE` command will re-activate all inactive members.

```
LOAD LIST ALL
```

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the "active" load cases. The `LOAD LIST ALL` command enables all the load cases in the structure to be made active for further processing.

```
PRINT MEMBER FORCES
```

The preceding `PRINT` command is an instruction to produce a report, in the output file, of the member end forces.

```
LOAD LIST 1 4 5
```

A `LOAD LIST` command is a means of instructing the program to use only the listed load cases for further processing.

```
PARAMETER
CODE AISC UNIFIED
BEAM 1.0 ALL
UNT 6.0 ALL
UNB 6.0 ALL
KY 0.5 ALL
```

The `PARAMETER` command is used to specify the steel design parameters (information on these parameters can be obtained from the manual where the implementation of the code is explained). Design will be done according to the specifications of the AISC ASD Code. The `BEAM` parameter is specified to perform design at every 1/12th point along the member length. `UNT` and `UNB` represent the unsupported length of the flanges to be used for calculation of allowable bending stress. `KY 0.5 ALL` sets the effective length factor for column buckling about the local Y-axis to be 0.5 for ALL members.

```
    CHECK CODE ALL
```

The above command instructs the program to perform a check to determine how you defined member sizes along with the latest analysis results meet the code requirements.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE A PLANE FRAME STRUCTURE WITH TENSION BRACING
UNIT INCH KIP
SET NL 3
JOINT COORDINATES
1 0 0 0 3 480. 0 0
4 0 180. 0 6 480. 180. 0
7 240. 360. 0 ; 8 480. 360. 0
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ;14 5 8
MEMBER TRUSS
9 TO 14
MEMBER PROP AMERICAN
1 TO 5 TABLE ST W12X26
6 7 8 TA ST W18X35
9 TO 14 TA LD L50505
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 2 3 PINNED
INACTIVE MEMBERS 9 TO 14
UNIT FT
LOADING 1 DEAD AND LIVE LOAD
MEMBER LOAD
6 8 UNI GY -1.0
7 UNI GY -1.5
PERFORM ANALYSIS
CHANGES
INACTIVE MEMBERS 10 11 13
```

```
LOADING 2 WIND FROM LEFT
JOINT LOAD
4 FX 30 ; 7 FX 15
PERFORM ANALYSIS
CHANGE
INACTIVE MEMBERS 9 12 14
LOADING 3 WIND FROM RIGHT
JOINT LOAD
6 FX -30 ; 8 FX -15
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 4 5
PARAMETER
CODE AISC UNIFIED
BEAM 1.0 ALL
UNT 6.0 ALL
UNB 6.0 ALL
KY 0.5 ALL
CHECK CODE ALL
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:31                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE A PLANE FRAME STRUCTURE WITH TENSION BRACING
INPUT FILE: US-4 Inactive Members in a Braced Frame.STD
     2. UNIT INCH KIP
     3. SET NL 3
     4. JOINT COORDINATES
     5. 1 0 0 0 3 480. 0 0
     6. 4 0 180. 0 6 480. 180. 0
     7. 7 240. 360. 0 ; 8 480. 360. 0
     8. MEMBER INCIDENCE
     9. 1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
    10. 8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
    11. 13 6 7 ;14 5 8
    12. MEMBER TRUSS
    13. 9 TO 14
    14. MEMBER PROP AMERICAN
    15. 1 TO 5 TABLE ST W12X26
    16. 6 7 8 TA ST W18X35
```

```
  17. 9 TO 14 TA LD L50505
  18. DEFINE MATERIAL START
  19. ISOTROPIC STEEL
  20. E 29000
  21. POISSON 0.3
  22. DENSITY 283E-006
  23. ALPHA 6E-006
  24. DAMP 0.03
  25. TYPE STEEL
  26. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  27. END DEFINE MATERIAL
  28. CONSTANT
  29. MATERIAL STEEL ALL
  30. SUPPORT
  31. 1 2 3 PINNED
  32. INACTIVE MEMBERS 9 TO 14
  33. UNIT FT
  34. LOADING 1 DEAD AND LIVE LOAD
  35. MEMBER LOAD
  36. 6 8 UNI GY -1.0
  37. 7 UNI GY -1.5
  38. PERFORM ANALYSIS
    A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    2
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS          8  NUMBER OF MEMBERS      14
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
 TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  39. CHANGES
  40. INACTIVE MEMBERS 10 11 13
  41. LOADING 2 WIND FROM LEFT
  42. JOINT LOAD
  43. 4 FX 30 ; 7 FX 15
  44. PERFORM ANALYSIS
  45. CHANGE
  46. INACTIVE MEMBERS 9 12 14
  47. LOADING 3 WIND FROM RIGHT
  48. JOINT LOAD
  49. 6 FX -30 ; 8 FX -15
  50. LOAD COMBINATION 4
  51. 1 0.75 2 0.75
  52. LOAD COMBINATION 5
  53. 1 0.75 3 0.75
  54. PERFORM ANALYSIS
  55. CHANGE
  56. LOAD LIST ALL
  57. PRINT MEMBER FORCES
MEMBER   FORCES
    A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    3
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1      8.26     -0.67     0.00      0.00      0.00      -0.00
```

```
                4     -8.26      0.67     0.00      0.00      0.00    -10.06
          2     1     -0.31      0.22     0.00      0.00      0.00      0.00
                4      0.31     -0.22     0.00      0.00      0.00      3.25
          3     1     15.81     -0.19     0.00      0.00      0.00     -0.00
                4    -15.81      0.19     0.00      0.00      0.00     -2.88
          4     1      5.97     -0.34     0.00      0.00      0.00     -0.00
                4     -5.97      0.34     0.00      0.00      0.00     -5.11
          5     1     18.06     -0.65     0.00      0.00      0.00     -0.00
                4    -18.06      0.65     0.00      0.00      0.00     -9.71
     2    1     2     38.47     -0.05     0.00      0.00      0.00      0.00
                5    -38.47      0.05     0.00      0.00      0.00     -0.77
          2     2      9.09      0.16     0.00      0.00      0.00      0.00
                5     -9.09     -0.16     0.00      0.00      0.00      2.43
          3     2     28.81     -0.15     0.00      0.00      0.00     -0.00
                5    -28.81      0.15     0.00      0.00      0.00     -2.21
          4     2     35.67      0.08     0.00      0.00      0.00      0.00
                5    -35.67     -0.08     0.00      0.00      0.00      1.24
          5     2     50.46     -0.15     0.00      0.00      0.00     -0.00
                5    -50.46      0.15     0.00      0.00      0.00     -2.24
     3    1     5     10.14     -2.20     0.00      0.00      0.00    -13.11
                7    -10.14      2.20     0.00      0.00      0.00    -19.84
          2     5     -0.29      0.42     0.00      0.00      0.00      3.34
                7      0.29     -0.42     0.00      0.00      0.00      2.94
          3     5     10.88     -0.63     0.00      0.00      0.00     -5.23
                7    -10.88      0.63     0.00      0.00      0.00     -4.20
          4     5      7.39     -1.33     0.00      0.00      0.00     -7.32
                7     -7.39      1.33     0.00      0.00      0.00    -12.68
          5     5     15.76     -2.12     0.00      0.00      0.00    -13.76
                7    -15.76      2.12     0.00      0.00      0.00    -18.03
     4    1     3     23.26      0.72     0.00      0.00      0.00      0.00
                6    -23.26     -0.72     0.00      0.00      0.00     10.83
          2     3     24.63      0.07     0.00      0.00      0.00      0.00
                6    -24.63     -0.07     0.00      0.00      0.00      1.08
          3     3    -11.25     -0.16     0.00      0.00      0.00     -0.00
                6     11.25      0.16     0.00      0.00      0.00     -2.41
          4     3     35.92      0.60     0.00      0.00      0.00      0.00
                6    -35.92     -0.60     0.00      0.00      0.00      8.94
          5     3      9.01      0.42     0.00      0.00      0.00     -0.00
                6     -9.01     -0.42     0.00      0.00      0.00      6.31
     A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    4
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     5    1     6      9.86      2.20     0.00      0.00      0.00     15.84
                8     -9.86     -2.20     0.00      0.00      0.00     17.11
          2     6     10.95      0.37     0.00      0.00      0.00      2.68
                8    -10.95     -0.37     0.00      0.00      0.00      2.82
          3     6     -0.35     -0.33     0.00      0.00      0.00     -2.10
                8      0.35      0.33     0.00      0.00      0.00     -2.78
          4     6     15.61      1.92     0.00      0.00      0.00     13.89
                8    -15.61     -1.92     0.00      0.00      0.00     14.95
          5     6      7.14      1.40     0.00      0.00      0.00     10.30
                8     -7.14     -1.40     0.00      0.00      0.00     10.75
     6    1     4      0.67      8.26     0.00      0.00      0.00     10.06
                5     -0.67     11.74     0.00      0.00      0.00    -44.76
          2     4     29.78     -0.31     0.00      0.00      0.00     -3.25
                5    -29.78      0.31     0.00      0.00      0.00     -2.92
```

```
          3     4     20.77       0.38      0.00       0.00       0.00        2.88
                5    -20.77      -0.38      0.00       0.00       0.00        4.69
          4     4     22.84       5.97      0.00       0.00       0.00        5.11
                5    -22.84       9.03      0.00       0.00       0.00      -35.76
          5     4     16.08       6.48      0.00       0.00       0.00        9.71
                5    -16.08       8.52      0.00       0.00       0.00      -30.05
     7    1     5     -1.47      16.60      0.00       0.00       0.00       58.64
                6      1.47      13.40      0.00       0.00       0.00      -26.67
          2     5     17.51      -0.33      0.00       0.00       0.00       -2.85
                6    -17.51       0.33      0.00       0.00       0.00       -3.76
          3     5     44.21       0.36      0.00       0.00       0.00        2.75
                6    -44.21      -0.36      0.00       0.00       0.00        4.52
          4     5     12.03      12.20      0.00       0.00       0.00       41.84
                6    -12.03      10.30      0.00       0.00       0.00      -22.82
          5     5     32.05      12.72      0.00       0.00       0.00       46.04
                6    -32.05       9.78      0.00       0.00       0.00      -16.61
     8    1     7      2.20      10.14      0.00       0.00       0.00       19.84
                8     -2.20       9.86      0.00       0.00       0.00      -17.11
          2     7     14.58      -0.29      0.00       0.00       0.00       -2.94
                8    -14.58       0.29      0.00       0.00       0.00       -2.82
          3     7     14.67       0.35      0.00       0.00       0.00        4.20
                8    -14.67      -0.35      0.00       0.00       0.00        2.78
          4     7     12.58       7.39      0.00       0.00       0.00       12.68
                8    -12.58       7.61      0.00       0.00       0.00      -14.95
          5     7     12.65       7.86      0.00       0.00       0.00       18.03
                8    -12.65       7.14      0.00       0.00       0.00      -10.75
     9    1     1      0.00       0.00      0.00       0.00       0.00        0.00
                5      0.00       0.00      0.00       0.00       0.00        0.00
     A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2     1    -33.43       0.00      0.00       0.00       0.00        0.00
                5     33.43       0.00      0.00       0.00       0.00        0.00
          3     1      0.00       0.00      0.00       0.00       0.00        0.00
                5      0.00       0.00      0.00       0.00       0.00        0.00
          4     1    -25.07       0.00      0.00       0.00       0.00        0.00
                5     25.07       0.00      0.00       0.00       0.00        0.00
          5     1      0.00       0.00      0.00       0.00       0.00        0.00
                5      0.00       0.00      0.00       0.00       0.00        0.00
    10    1     2      0.00       0.00      0.00       0.00       0.00        0.00
                4      0.00       0.00      0.00       0.00       0.00        0.00
          2     2      0.00       0.00      0.00       0.00       0.00        0.00
                4      0.00       0.00      0.00       0.00       0.00        0.00
          3     2    -25.73       0.00      0.00       0.00       0.00        0.00
                4     25.73       0.00      0.00       0.00       0.00        0.00
          4     2      0.00       0.00      0.00       0.00       0.00        0.00
                4      0.00       0.00      0.00       0.00       0.00        0.00
          5     2    -19.29       0.00      0.00       0.00       0.00        0.00
                4     19.29       0.00      0.00       0.00       0.00        0.00
    11    1     3      0.00       0.00      0.00       0.00       0.00        0.00
                5      0.00       0.00      0.00       0.00       0.00        0.00
          2     3      0.00       0.00      0.00       0.00       0.00        0.00
                5      0.00       0.00      0.00       0.00       0.00        0.00
          3     3    -29.90       0.00      0.00       0.00       0.00        0.00
                5     29.90       0.00      0.00       0.00       0.00        0.00
          4     3      0.00       0.00      0.00       0.00       0.00        0.00
```

```
              5      0.00      0.00     0.00      0.00      0.00       0.00
        5     3    -22.42      0.00     0.00      0.00      0.00       0.00
              5     22.42      0.00     0.00      0.00      0.00       0.00
   12   1     2      0.00      0.00     0.00      0.00      0.00       0.00
              6      0.00      0.00     0.00      0.00      0.00       0.00
        2     2    -22.26      0.00     0.00      0.00      0.00       0.00
              6     22.26      0.00     0.00      0.00      0.00       0.00
        3     2      0.00      0.00     0.00      0.00      0.00       0.00
              6      0.00      0.00     0.00      0.00      0.00       0.00
        4     2    -16.69      0.00     0.00      0.00      0.00       0.00
              6     16.69      0.00     0.00      0.00      0.00       0.00
        5     2      0.00      0.00     0.00      0.00      0.00       0.00
              6      0.00      0.00     0.00      0.00      0.00       0.00
   13   1     6      0.00      0.00     0.00      0.00      0.00       0.00
              7      0.00      0.00     0.00      0.00      0.00       0.00
        2     6      0.00      0.00     0.00      0.00      0.00       0.00
              7      0.00      0.00     0.00      0.00      0.00       0.00
    A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    6
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
        3     6    -17.56      0.00     0.00      0.00      0.00       0.00
              7     17.56      0.00     0.00      0.00      0.00       0.00
        4     6      0.00      0.00     0.00      0.00      0.00       0.00
              7      0.00      0.00     0.00      0.00      0.00       0.00
        5     6    -13.17      0.00     0.00      0.00      0.00       0.00
              7     13.17      0.00     0.00      0.00      0.00       0.00
   14   1     5      0.00      0.00     0.00      0.00      0.00       0.00
              8      0.00      0.00     0.00      0.00      0.00       0.00
        2     5    -17.77      0.00     0.00      0.00      0.00       0.00
              8     17.77      0.00     0.00      0.00      0.00       0.00
        3     5      0.00      0.00     0.00      0.00      0.00       0.00
              8      0.00      0.00     0.00      0.00      0.00       0.00
        4     5    -13.33      0.00     0.00      0.00      0.00       0.00
              8     13.33      0.00     0.00      0.00      0.00       0.00
        5     5      0.00      0.00     0.00      0.00      0.00       0.00
              8      0.00      0.00     0.00      0.00      0.00       0.00
 ************** END OF LATEST ANALYSIS RESULT **************
  58. LOAD LIST 1 4 5
  59. PARAMETER
  60. CODE AISC UNIFIED
  61. BEAM 1.0 ALL
  62. UNT 6.0 ALL
  63. UNB 6.0 ALL
  64. KY 0.5 ALL
  65. CHECK CODE ALL
PARAMETER 1
    A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    7
                  STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
========================
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
```

```
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X              Z          Axis typically parallel to the sections principal
major axis.
       Y              Y          Axis typically parallel to the sections principal
minor axis.
       Z              X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz             FX         Axial force.
       Vy             FY         Shear force along minor axis.
       Vx             FZ         Shear force along major axis.
       Tz             MX         Torsional moment.
       My             MY         Bending moment about minor axis.
       Mx             MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), while
       the results for shear are reported as being for shear forces �along� the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      A PLANE FRAME STRUCTURE WITH TENSION BRACING             -- PAGE NO.    8
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     *****************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|-----------------------------------------------------------------------------|
|  Member No:        1       Profile:  ST  W12X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.138       Loadcase:        5     |
|  Location:     15.00       Ref:      Eq.H1-3b                               |
|  Pz:       18.06     C     Vy:       -.6472          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      9.708         |
|-----------------------------------------------------------------------------|
 - Member :     2
|-----------------------------------------------------------------------------|
|  Member No:        2       Profile:  ST  W12X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:        0.319       Loadcase:        5     |
|  Location:     15.00       Ref:      Eq.H1-1a                               |
|  Pz:       50.46     C     Vy:       -.1491          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      2.237         |
|-----------------------------------------------------------------------------|
 - Member :     3
```

```
|-----------------------------------------------------------------------------|
|  Member No:        3       Profile:  ST  W12X26            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.220      Loadcase:        1   |
|  Location:     15.00       Ref:      Eq.H1-3a(H1-1b)                      |
|  Pz:       10.14     C     Vy:       -2.197            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:      19.84     |
|-----------------------------------------------------------------------------|
- Member :     4
|-----------------------------------------------------------------------------|
|  Member No:        4       Profile:  ST  W12X26            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.256      Loadcase:        4   |
|  Location:     15.00       Ref:      Eq.H1-3b                             |
|  Pz:       35.92     C     Vy:       0.5957            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:     -8.935     |
|-----------------------------------------------------------------------------|
- Member :     5
|-----------------------------------------------------------------------------|
|  Member No:        5       Profile:  ST  W12X26            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.192      Loadcase:        1   |
|  Location:     15.00       Ref:      Eq.H1-3a(H1-1b)                      |
|  Pz:       9.864     C     Vy:        2.197            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:     -17.11     |
|-----------------------------------------------------------------------------|
- Member :     6
|-----------------------------------------------------------------------------|
|  Member No:        6       Profile:  ST  W18X35            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.259      Loadcase:        1   |
|  Location:     20.00       Ref:      Cl.F2.2                              |
|  Pz:      0.6707     C     Vy:       -11.74            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:      44.76     |
|-----------------------------------------------------------------------------|
- Member :     7
|-----------------------------------------------------------------------------|
|  Member No:        7       Profile:  ST  W18X35            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.340      Loadcase:        1   |
|  Location:      0.00       Ref:      Eq.H1-1b                             |
|  Pz:       1.474     T     Vy:        16.60            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:      58.64     |
|-----------------------------------------------------------------------------|
      A PLANE FRAME STRUCTURE WITH TENSION BRACING            -- PAGE NO.    9
                       STAAD.PRO CODE CHECKING - AISC 360-16 LRFD (V1.2)
                     ****************************************************
ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
- Member :     8
|-----------------------------------------------------------------------------|
|  Member No:        8       Profile:  ST  W18X35            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.182      Loadcase:        1   |
|  Location:     10.00       Ref:      Cl.F2.2                              |
|  Pz:       2.197     C     Vy:       0.1365            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:     -31.52     |
|-----------------------------------------------------------------------------|
- Member :     9
|-----------------------------------------------------------------------------|
|  Member No:        9       Profile:  LD  L50505            (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.126      Loadcase:        4   |
|  Location:      0.00       Ref:      Cl.D2                                |
|  Pz:       25.07     T     Vy:        0.000            Vx:      0.000     |
|  Tz:       0.000           My:        0.000            Mx:      0.000     |
```

```
|-----------------------------------------------------------------------------|
 - Member :    10
|-----------------------------------------------------------------------------|
|  Member No:       10       Profile:  LD  L50505              (AISC SECTIONS)|
|  Status:         PASS      Ratio:         0.097       Loadcase:        5    |
|  Location:       0.00      Ref:      Cl.D2                                  |
|  Pz:       19.29     T     Vy:        0.000           Vx:       0.000       |
|  Tz:       0.000           My:        0.000           Mx:       0.000       |
|-----------------------------------------------------------------------------|
 - Member :    11
|-----------------------------------------------------------------------------|
|  Member No:       11       Profile:  LD  L50505              (AISC SECTIONS)|
|  Status:         PASS      Ratio:         0.113       Loadcase:        5    |
|  Location:       0.00      Ref:      Cl.D2                                  |
|  Pz:       22.42     T     Vy:        0.000           Vx:       0.000       |
|  Tz:       0.000           My:        0.000           Mx:       0.000       |
|-----------------------------------------------------------------------------|
 - Member :    12
|-----------------------------------------------------------------------------|
|  Member No:       12       Profile:  LD  L50505              (AISC SECTIONS)|
|  Status:         PASS      Ratio:         0.084       Loadcase:        4    |
|  Location:       0.00      Ref:      Cl.D2                                  |
|  Pz:       16.69     T     Vy:        0.000           Vx:       0.000       |
|  Tz:       0.000           My:        0.000           Mx:       0.000       |
|-----------------------------------------------------------------------------|
 - Member :    13
|-----------------------------------------------------------------------------|
|  Member No:       13       Profile:  LD  L50505              (AISC SECTIONS)|
|  Status:         PASS      Ratio:         0.066       Loadcase:        5    |
|  Location:       0.00      Ref:      Cl.D2                                  |
|  Pz:       13.17     T     Vy:        0.000           Vx:       0.000       |
|  Tz:       0.000           My:        0.000           Mx:       0.000       |
|-----------------------------------------------------------------------------|
 - Member :    14
|-----------------------------------------------------------------------------|
|  Member No:       14       Profile:  LD  L50505              (AISC SECTIONS)|
|  Status:         PASS      Ratio:         0.067       Loadcase:        4    |
|  Location:       0.00      Ref:      Cl.D2                                  |
|  Pz:       13.33     T     Vy:        0.000           Vx:       0.000       |
|  Tz:       0.000           My:        0.000           Mx:       0.000       |
|-----------------------------------------------------------------------------|
   66. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46:32 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *Member Specification dialog* (on page 2786)

- *TR.18 Inactive/Delete Specification* (on page 2240)

EX. US-5 Support Settlement on a Portal Frame

This example demonstrates the application of support displacement load (also known as a sinking support) on a space frame structure.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-5 Support Settlement on a Portal Frame.STD` when you install the program.

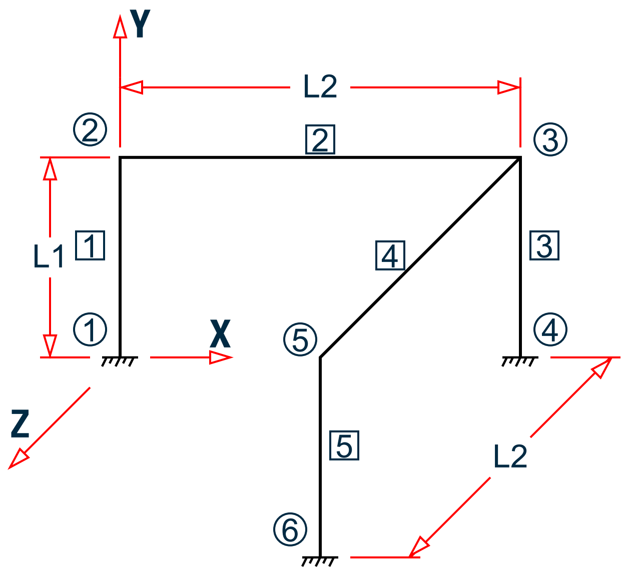


Figure 516: Example Problem No. 5

Where:

L1 = 10 ft
L2 = 20 ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
```

Every input has to start with the term `STAAD`. The word `SPACE` signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNITS KIP FEET
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0.0  0.0  0.0  ;  2  0.0  10.0  0.0
3  20.0  10.0  0.0  ;  4  20.0  0.0  0.0
5  20.  10.  20.  ;  6  20.  0.  20.
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6
```

Defines the members by the joints to which they are connected.

```
UNIT INCH
MEMB PROP
1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.
```

Member properties have been defined above using the `PRISMATIC` attribute. Values of AX (area), IZ (moment of inertia about major axis), IY (moment of inertia about minor axis) and IX (torsional constant) are provided in `INCH` unit.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

Material are defined using the `DEFINE MATERIAL` command and then assigned using the `CONSTANT` command.

```
SUPPORT
1 4 6 FIXED
```

Joints 1, 4 and 6 are fixed supports.

```
LOADING 1 SINKING SUPPORT
```

Load case 1 is initiated followed by a title.

```
SUPPORT DISPLACEMENT LOAD
4 FY -0.50
```

Load 1 is a support displacement load which is also commonly known as a sinking support. `FY` signifies that the support settlement is in the global Y direction and the value of this settlement is 0.5 inch downward.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT ANALYSIS RESULTS
```

The above `PRINT` command instructs the program to print joint displacements, support reactions and member forces.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
UNITS KIP FEET
```

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 10.0 0.0
3 20.0 10.0 0.0 ; 4 20.0 0.0 0.0
5 20. 10. 20. ; 6 20. 0. 20.
MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6
UNIT INCH
MEMB PROP
1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 4 6 FIXED
LOADING 1 SINKING SUPPORT
SUPPORT DISPLACEMENT LOAD
4 FY -0.50
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                                    PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:34                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
INPUT FILE: US-5 Support Settlement on a Portal Frame.STD
     2. UNITS KIP FEET
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 ; 2 0.0 10.0 0.0
     5. 3 20.0 10.0 0.0 ; 4 20.0 0.0 0.0
     6. 5 20. 10. 20. ; 6 20. 0. 20.
     7. MEMBER INCIDENCE
     8. 1 1 2 3
     9. 4 3 5 ; 5 5 6
    10. UNIT INCH
    11. MEMB PROP
    12. 1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.
```

```
   13. DEFINE MATERIAL START
   14. ISOTROPIC STEEL
   15. E 29000.
   16. POISSON 0.3
   17. DENSITY 283E-006
   18. ALPHA 6E-006
   19. DAMP 0.03
   20. TYPE STEEL
   21. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
   22. END DEFINE MATERIAL
   23. CONSTANT
   24. MATERIAL STEEL ALL
   25. SUPPORT
   26. 1 4 6 FIXED
   27. LOADING 1 SINKING SUPPORT
   28. SUPPORT DISPLACEMENT LOAD
   29. 4 FY -0.50
   30. PERFORM ANALYSIS
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    2
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   31. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    3
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     2    1    0.09125  -0.00040  -0.01078  -0.00014   0.00050  -0.00154
     3    1    0.09118  -0.49919  -0.09118  -0.00154   0.00000  -0.00154
     4    1    0.00000  -0.50000   0.00000   0.00000   0.00000   0.00000
     5    1    0.01078  -0.00040  -0.09125  -0.00154  -0.00050  -0.00014
     6    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      0.08      0.97      0.15     19.22     -0.46    107.07
     4    1      0.07     -1.95     -0.07    107.18      0.00    107.18
     6    1     -0.15      0.97     -0.08    107.07      0.46     19.22
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KIP  INCH     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      0.97     -0.08     0.15     -0.46    -19.22     107.07
                2     -0.97      0.08    -0.15      0.46      0.65    -116.64
     2    1     2      0.08      0.97     0.15      0.65     -0.46     116.64
                3     -0.08     -0.97    -0.15     -0.65    -36.66     116.82
     3    1     3     -1.95     -0.07     0.07     -0.00   -116.17    -116.17
                4      1.95      0.07    -0.07      0.00    107.18     107.18
```

```
     4    1     3      0.08     -0.97    -0.15     -0.65     36.66    -116.82
                5     -0.08      0.97     0.15      0.65      0.46    -116.64
     5    1     5      0.97      0.15     0.08      0.46   -116.64      -0.65
                6     -0.97     -0.15    -0.08     -0.46    107.07      19.22
  ************** END OF LATEST ANALYSIS RESULT **************
   32. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:46:35 ****
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    6
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-6 Prestress and Poststress Loading

This is an example of prestress loading in a plane frame structure.

It covers two situations:

1. From the member on which it is applied, the prestressing effect is transmitted to the rest of the structure through the connecting members (known in the program as `PRESTRESS` load).
2. The prestressing effect is experienced by the member(s) alone and not transmitted to the rest of the structure (known in the program as `POSTSTRESS` load).

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-6 Prestress and Poststress Loading.STD` when you install the program.

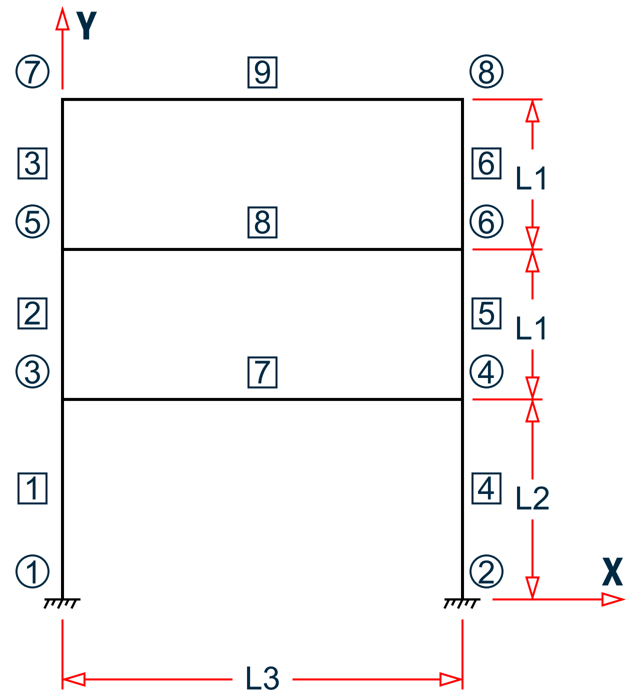


Figure 517: Example Problem No. 6

Where:

L1 = 15 ft
L2 = 20 ft
L3 = 40 ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE FRAME WITH PRESTRESSING LOAD
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT KIP FT
```

Defines the input units for the data that follows.

```
JOINT COORD
1  0.  0.  ;  2  40.  0.  ;  3  0.  20.  ;  4  40.  20.
5  0.  35.  ;  6  40.  35.  ;  7  0.  50.  ;  8  40.  50.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
```

Defines the members by the joints to which they are connected.

```
SUPPORT
1 2 FIXED
```

The supports at joints 1 and 2 are defined to be fixed supports.

```
MEMB PROP
1 TO 9 PRI AX 2.2 IZ 1.0
```

Member properties are provided using the PRI (prismatic) attribute. Values of area (AX) and moment of inertia about the major axis (IZ) are provided.

```
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properities and the `CONSTANT` command is used to assign the material to all members. Length unit is changed from FT to INCH to facilitate the input.

```
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7 8 FORCE 300. ES 3. EM -12. EE 3.
```

Load case 1 is initiated along with an accompanying title. Load 1 contains `PRESTRESS` load. Members 7 and 8 have a cable force of 300 kips. The location of the cable at the start (`ES`) and end (`EE`) is 3 inches above the center of gravity while at the middle (`EM`) it is 12 inches below the c.g. The assumptions and facts associated with this type of loading are explained in G.15.5 Prestress and Poststress Member Load (on page 2131).

```
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7 8 FORCE 300. ES 3. EM -12. EE 3.
```

Load case 2 is initiated along with an accompanying title. Load 2 is a `POSTSTRESS` load. Members 7 and 8 have cable force of 300 kips. The location of the cable is the same as in load case 1. For a difference between PRESTRESS loading and POSTSTRESS loading, as well as additional information about both types of loads, please refer to G.15.5 Prestress and Poststress Member Load (on page 2131).

```
PERFORM ANALYSIS
```

This command instructs the program to perform the analysis.

```
UNIT FT
PRINT ANALYSIS RESULT
```

The preceding command is an instruction to write joint displacements, support reactions and member forces in the output file. The preceding line causes the results to be written in the length unit of feet.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE  FRAME WITH PRESTRESSING LOAD
UNIT KIP FT
JOINT COORD
1 0. 0. ; 2 40. 0. ; 3 0. 20. ; 4 40. 20.
5 0. 35. ; 6 40. 35. ; 7 0. 50. ; 8 40. 50.
MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
SUPPORT
1 2 FIXED
MEMB PROP
1 TO 9 PRI AX 2.2 IZ 1.0
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7 8 FORCE 300. ES 3. EM -12. EE 3.
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7 8 FORCE 300. ES 3. EM -12. EE 3.
PERFORM ANALYSIS
UNIT FT
PRINT ANALYSIS RESULT
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:37                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE  FRAME WITH PRESTRESSING LOAD
INPUT FILE: US-6 Prestress and Poststress Loading.STD
     2. UNIT KIP FT
     3. JOINT COORD
```

```
    4. 1 0. 0. ; 2 40. 0. ; 3 0. 20. ; 4 40. 20.
    5. 5 0. 35. ; 6 40. 35. ; 7 0. 50. ; 8 40. 50.
    6. MEMBER INCIDENCE
    7. 1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
    8. 6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
    9. SUPPORT
   10. 1 2 FIXED
   11. MEMB PROP
   12. 1 TO 9 PRI AX 2.2 IZ 1.0
   13. UNIT INCH
   14. DEFINE MATERIAL START
   15. ISOTROPIC CONCRETE
   16. E 3000.
   17. POISSON 0.17
   18. DENSITY 8.7E-005
   19. ALPHA 5E-006
   20. DAMP 0.05
   21. G 1346.15
   22. TYPE CONCRETE
   23. STRENGTH FCU 4
   24. END DEFINE MATERIAL
   25. CONSTANTS
   26. MATERIAL CONCRETE ALL
   27. LOADING 1 PRESTRESSING LOAD
   28. MEMBER PRESTRESS
   29. 7 8 FORCE 300. ES 3. EM -12. EE 3.
   30. LOADING 2 POSTSTRESSING LOAD
   31. MEMBER POSTSTRESS
   32. 7 8 FORCE 300. ES 3. EM -12. EE 3.
   33. PERFORM ANALYSIS
     FRAME WITH PRESTRESSING LOAD                            -- PAGE NO.    2
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          8  NUMBER OF MEMBERS       9
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      18
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   34. UNIT FT
   35. PRINT ANALYSIS RESULT
 ANALYSIS RESULT
     FRAME WITH PRESTRESSING LOAD                            -- PAGE NO.    3
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
         2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
    2    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
         2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
    3    1    0.07698   0.00000   0.00000   0.00000   0.00000   0.00039
         2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
    4    1   -0.07698   0.00000   0.00000   0.00000   0.00000  -0.00039
         2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
    5    1    0.07224   0.00000   0.00000   0.00000   0.00000   0.00087
         2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
    6    1   -0.07224   0.00000   0.00000   0.00000   0.00000  -0.00087
```

```
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     7    1   -0.00059   0.00000   0.00000   0.00000   0.00000   0.00015
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     8    1    0.00059   0.00000   0.00000   0.00000   0.00000  -0.00015
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    4
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -6.71      0.00      0.00      0.00      0.00     58.62
          2      0.00      0.00      0.00      0.00      0.00      0.00
     2    1      6.71      0.00      0.00      0.00      0.00    -58.62
          2      0.00      0.00      0.00      0.00      0.00      0.00
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  FEET     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      0.00      6.71     0.00      0.00      0.00      58.62
                3     -0.00     -6.71     0.00      0.00      0.00      75.67
          2     1      0.00      0.00     0.00      0.00      0.00       0.00
                3      0.00      0.00     0.00      0.00      0.00       0.00
     2    1     3      0.00     13.92     0.00      0.00      0.00      90.81
                5     -0.00    -13.92     0.00      0.00      0.00     117.98
          2     3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
     3    1     5      0.00      2.34     0.00      0.00      0.00      38.31
                7     -0.00     -2.34     0.00      0.00      0.00      -3.16
          2     5      0.00      0.00     0.00      0.00      0.00       0.00
                7      0.00      0.00     0.00      0.00      0.00       0.00
     4    1     2     -0.00     -6.71     0.00      0.00      0.00     -58.62
                4      0.00      6.71     0.00      0.00      0.00     -75.67
          2     2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
     5    1     4     -0.00    -13.92     0.00      0.00      0.00     -90.81
                6      0.00     13.92     0.00      0.00      0.00    -117.98
          2     4      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
     6    1     6     -0.00     -2.34     0.00      0.00      0.00     -38.31
                8      0.00      2.34     0.00      0.00      0.00       3.16
          2     6      0.00      0.00     0.00      0.00      0.00       0.00
                8      0.00      0.00     0.00      0.00      0.00       0.00
     7    1     3    304.85    -37.50     0.00      0.00      0.00    -241.48
                4   -304.85    -37.50     0.00      0.00      0.00     241.48
          2     3    297.65    -37.50     0.00      0.00      0.00     -75.00
                4   -297.65    -37.50     0.00      0.00      0.00      75.00
     8    1     5    286.07    -37.50     0.00      0.00      0.00    -231.29
                6   -286.07    -37.50     0.00      0.00      0.00     231.29
          2     5    297.65    -37.50     0.00      0.00      0.00     -75.00
                6   -297.65    -37.50     0.00      0.00      0.00      75.00
     9    1     7     -2.34      0.00     0.00      0.00      0.00       3.16
                8      2.34     -0.00     0.00      0.00      0.00      -3.16
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    6
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KIP  FEET     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2     7      0.00      0.00     0.00      0.00      0.00       0.00
```

```
                 8      0.00      0.00     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    36. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:46:37 ****
      FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    7
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *TR.32.5 Prestress Load Specification* (on page 2489)
- *Member Load tab* (on page 2842)
- *M. To add a prestress or post-tension load to members* (on page 836)

EX. US-7 Modeling Offset Connections in a Frame

This example illustrates modeling of structures with offset connections. Offset connections arise when the center lines of the connected members do not intersect at the connection point. The connection eccentricity behaves as a rigid link and is modeled through specification of `MEMBER OFFSETS`.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-7 Modeling Offset Connections in a Frame.STD` when you install the program.

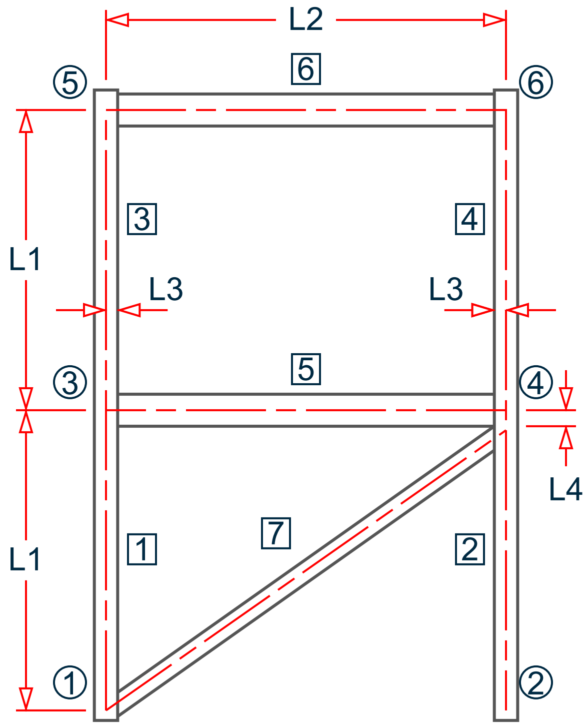


Figure 518: Example Problem No. 7

Where:

L1 = 15 ft
L2 = 20 ft
L3 = 7 in
L4 = 6 in

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE TEST FOR MEMBER OFFSETS
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT FT KIP
```

Defines the input units for the data that follows.

```
JOINT COORD
1  0.  0.  ;  2  20.  0.  ;  3  0.  15.
4  20.  15.  ;  5  0.  30.  ;  6  20.  30.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
SUPPORT
1 2 PINNED
```

Pinned supports are specified at joints 1 and 2. The word PINNED signifies that no moments will be carried by these supports.

```
MEMB INCI
1 1 3 2 ; 3 3 5 4
5 3 4 ; 6 5 6 ; 7 1 4
```

Defines the members by the joints to which they are connected.

```
MEMB PROP AMERICAN
1 TO 4 TABLE ST W14X90
5 6 TA ST W12X26
7 TA LD L80608
```

Member properties are assigned from the American (AISC) steel table for all members. The word ST stands for standard single section. LD stands for long leg back-to-back double angle.

Note: This problem uses a steel size from an older AISC catalog. In order to use this shape for analysis and design, you must select **AISCSections** or **aiscsections_all_editions** database for use as the **Amercian** section profile table.

```
UNIT INCH
MEMB OFFSET
5 6 START 7.0 0.0 0.0
5 6 END -7.0 0.0 0.0
7 END -7.0 -6.0 0.0
```

The preceding specification states that an OFFSET is located at the START/END joint of the members. The X, Y, and Z global coordinates of the offset distance from the corresponding incident joint are also provided. These attributes are applied to members 5, 6, and 7.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
LOADING 1 WIND LOAD
```

Load case 1 is initiated followed by a title.

```
JOINT LOAD
3 FX 50. ; 5 FX 25.0
```

Load 1 contains joint loads at nodes 3 and 5. FX indicates that the load is a force in the global X direction.

```
PERFORM ANALYSIS
```

The above command is an instruction to perform the analysis.

```
    UNIT FT
    PRINT FORCES
    PRINT REACTIONS
```

The above PRINT commands are instructions for writing the member forces and support reactions to the output file. The preceding line causes the results to be written in the length unit of feet.

```
    FINISH
```

This command terminates a STAAD run.

Input File

```
STAAD PLANE TEST FOR MEMBER OFFSETS
UNIT FT KIP
JOINT COORD
1 0. 0. ; 2 20. 0. ; 3 0. 15.
4 20. 15. ; 5 0. 30. ; 6 20. 30.
MEMB INCI
1 1 3 2; 3 3 5 4
5 3 4 ; 6 5 6 ; 7 1 4
MEMB PROP AMERICAN
1 TO 4 TABLE ST W14X90
5 6 TA ST W12X26
7 TA LD L80608
UNIT INCH
MEMB OFFSET
5 6 START 7.0 0.0 0.0
5 6 END  -7.0 0.0 0.0
7 END -7.0 -6.0 0.0
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 2 PINNED
LOADING 1 WIND LOAD
JOINT LOAD
3 FX 50. ; 5 FX 25.0
PERFORM ANALYSIS
UNIT FT
PRINT FORCES
PRINT REACTIONS
FINISH
```

STAAD Output File

```
                                                            PAGE NO.    1
             ****************************************************
```

```
             *                                                   *
             *           STAAD.Pro CONNECT Edition               *
             *           Version  22.10.00.***                   *
             *           Proprietary Program of                  *
             *           Bentley Systems, Inc.                   *
             *           Date=    MAR 24, 2022                   *
             *           Time=     9:46:40                       *
             *                                                   *
             *  Licensed to: Bentley Systems Inc                 *
             *****************************************************
     1. STAAD PLANE TEST FOR MEMBER OFFSETS
INPUT FILE: US-7 Modeling Offset Connections in a Frame.STD
     2. UNIT FT KIP
     3. JOINT COORD
     4. 1 0. 0. ; 2 20. 0. ; 3 0. 15.
     5. 4 20. 15. ; 5 0. 30. ; 6 20. 30.
     6. MEMB INCI
     7. 1 1 3 2; 3 3 5 4
     8. 5 3 4 ; 6 5 6 ; 7 1 4
     9. MEMB PROP AMERICAN
    10. 1 TO 4 TABLE ST W14X90
    11. 5 6 TA ST W12X26
    12. 7 TA LD L80608
    13. UNIT INCH
    14. MEMB OFFSET
    15. 5 6 START 7.0 0.0 0.0
    16. 5 6 END  -7.0 0.0 0.0
    17. 7 END -7.0 -6.0 0.0
    18. DEFINE MATERIAL START
    19. ISOTROPIC STEEL
    20. E 29000
    21. POISSON 0.3
    22. DENSITY 283E-006
    23. ALPHA 6E-006
    24. DAMP 0.03
    25. TYPE STEEL
    26. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    27. END DEFINE MATERIAL
    28. CONSTANT
    29. MATERIAL STEEL ALL
    30. SUPPORT
    31. 1 2 PINNED
    32. LOADING 1 WIND LOAD
    33. JOINT LOAD
    34. 3 FX 50. ; 5 FX 25.0
    35. PERFORM ANALYSIS
      TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS       7
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    36. UNIT FT
    37. PRINT FORCES
```

```
 FORCES
    TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    3
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1    -10.69     -4.61     0.00      0.00      0.00       3.56
                3     10.69      4.61     0.00      0.00      0.00     -72.68
     2    1     2     75.00     -5.73     0.00      0.00      0.00       0.00
                4    -75.00      5.73     0.00      0.00      0.00     -85.95
     3    1     3     -6.73     11.92     0.00      0.00      0.00     112.47
                5      6.73    -11.92     0.00      0.00      0.00      66.37
     4    1     4      6.73     13.08     0.00      0.00      0.00     127.91
                6     -6.73    -13.08     0.00      0.00      0.00      68.25
     5    1     3     66.53     -3.96     0.00      0.00      0.00     -37.49
                4    -66.53      3.96     0.00      0.00      0.00     -37.16
     6    1     5     13.08     -6.73     0.00      0.00      0.00     -62.44
                6    -13.08      6.73     0.00      0.00      0.00     -64.32
     7    1     1   -106.85     -0.46     0.00      0.00      0.00      -3.56
                4    106.85      0.46     0.00      0.00      0.00      -7.64
  ************** END OF LATEST ANALYSIS RESULT **************
   38. PRINT REACTIONS
 REACTION
    TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    1    1    -80.73    -75.00      0.00      0.00      0.00      0.00
    2    1      5.73     75.00      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   39. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:46:40 ****
    TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    5
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-8 Concrete Design for a Space Frame

In this example, concrete design is performed on some members of a space frame structure. Design calculations consist of computation of reinforcement for beams and columns. Secondary moments on the columns are obtained through the means of a P-Delta analysis.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-8 Concrete Design for a Space Frame.STD` when you install the program.

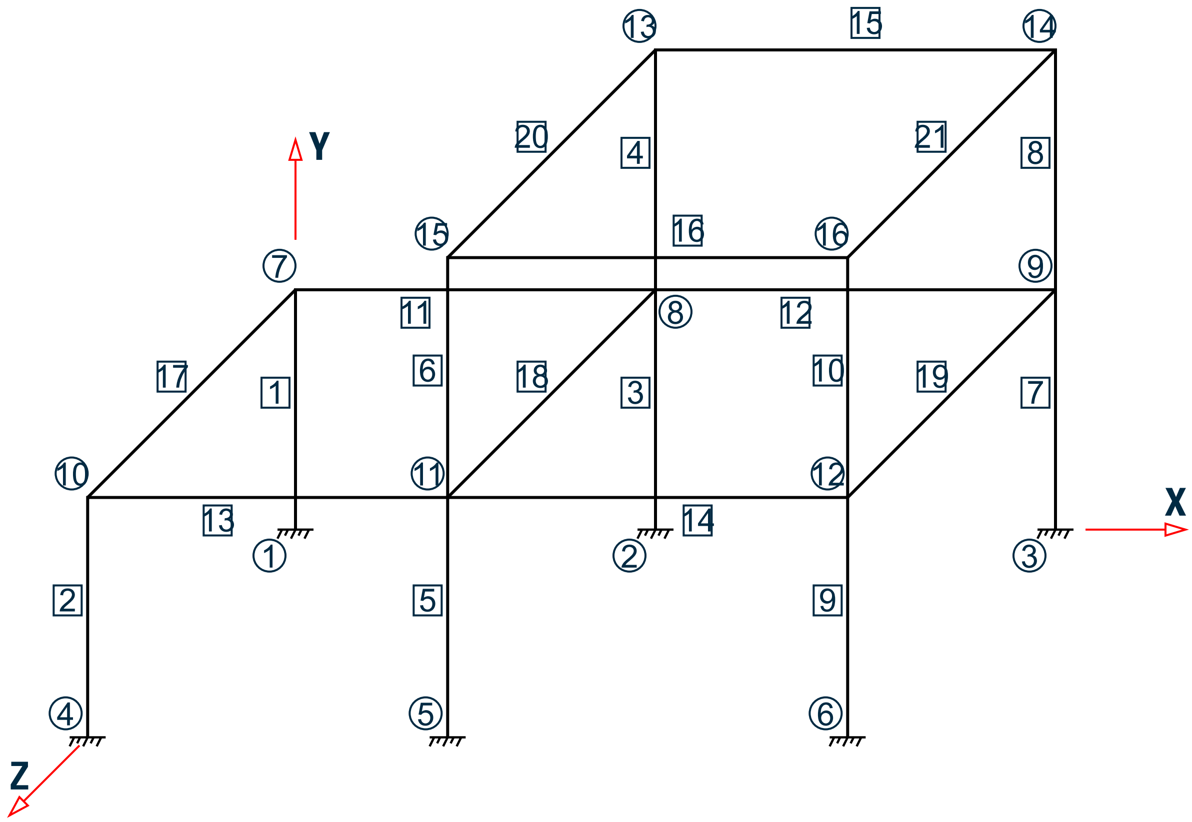


Figure 519: Example Problem No. 8

The above example represents a space frame, and the members are made of concrete. The input in the next page will show the dimensions of the members.

Two load cases, namely one for dead plus live load and another with dead, live and wind load, are considered in the design.

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE FRAME WITH CONCRETE DESIGN
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNIT KIP FT
```

Defines the input units for the data that follows.

```
JOINT COORDINATE
1 0 0 0 ; 2 18 0 0 ; 3 38 0. 0
4 0 0 24 ; 5 18 0 24 ; 6 38 0 24
7 0 12 0 ; 8 18 12 0 ; 9 38 12 0
10 0 12 24 ; 11 18 12 24 ; 12 38 12 24
13 18 24 0 ; 14 38 24 0 ; 15 18 24 24
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14  ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16
```

Defines the members by the joints to which they are connected.

```
UNIT INCH
MEMB PROP
1  2  PRISMATIC YD  12.0  IZ  509. IY  509. IX  1018.
3  TO  10 PR YD  12.0  ZD  12.0  IZ  864. IY  864. IX  1279.
11  TO  21 PR YD  21.0  ZD  16.0  IZ  5788.  IY  2953.  IX  6497.
```

All member properties are provided using the `PRISMATIC` option. `YD` and `ZD` stand for depth and width. If `ZD` is not provided, a circular shape with diameter = `YD` is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. For this particular example, moments of inertia (`IZ`, `IY`) and torsional constant (`IX`) are provided, so these will not be recalculated. The `IX`, `IY`, and `IZ` values provided in this example are only half the values of a full section to account for the fact that the full moments of inertia will not be effective due to cracking of concrete. Clause 10.11.1 of ACI 318-99 offers some guidelines on the amount of reduction to be applied on the gross section moment of inertia for beams, columns, walls and slabs to account for cracking.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FT
```

The `DEFINE MATERIAL` command is used to define material properties for concrete. The `CONSTANT` command is used to assign this to all members.

```
SUPPORT
1 TO 6 FIXED
```

Joints 1 to 6 are fixed supports.

```
LOAD 1 (1.4DL + 1.7LL)
```

Load case 1 is initiated followed by a title.

```
SELF Y -1.4
```

The selfweight of the structure is applied in the global Y direction with a -1.4 factor. Since global Y is vertically upward, the negative factor indicates that this load will act downwards.

```

MEMB LOAD
```

```
11  TO  16  UNI  Y  -2.8
11  TO  16  UNI  Y  -5.1
```

Load 1 contains member loads also. Y indicates that the load is in the local Y direction. The word UNI stands for uniformly distributed load.

```
LOAD 2 .75 (1.4DL + 1.7LL + 1.7WL)
```

Load case 2 is initiated followed by a title.

```
REPEAT LOAD
1 0.75
```

The preceding command will gather the load data values from load case 1, multiply them with a factor of 0.75 and utilize the resulting values in load 2.

```
JOINT LOAD
15  16  FZ  8.5
11  FZ  20.0
12  FZ  16.0
10  FZ  8.5
```

Load 2 contains some additional joint loads also. FZ indicates that the load is a force in the global Z direction.

```
PDELTA ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis type is P-DELTA indicating that second-order effects are to be calculated.

```
PRINT FORCES LIST 2 5 9 14 16
```

Member end forces are printed using the above PRINT commands. The LIST option restricts the print output to the members listed.

```
START CONCRETE DESIGN
```

The above command initiates a concrete design.

```
CODE ACI
TRACK 1.0 MEMB 14
TRACK 2.0 MEMB 16
MAXMAIN 11 ALL
```

The values for the concrete design parameters are defined in the above commands. Design is performed per the ACI 318 Code. The TRACK value dictates the extent of design related information that should appear in the output. MAXMAIN indicates that the maximum size of main reinforcement is the #11 bar. These parameters are described in the manual where American concrete design related information is available.

```
DESIGN BEAM 14 16
```

The above command instructs the program to design beams 14 and 16 for flexure, shear, and torsion.

```
DESIGN COLUMN 2 5
```

The above command instructs the program to design columns 2 and 5 for axial load and biaxial bending.

```
END CONCRETE DESIGN
```

This will end the concrete design.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE FRAME WITH CONCRETE DESIGN
UNIT KIP FT
JOINT COORDINATE
1 0 0 0 ; 2 18 0 0 ; 3 38 0. 0
4 0 0 24 ; 5 18 0 24 ; 6 38 0 24
7 0 12 0 ; 8 18 12 0 ; 9 38 12 0
10 0 12 24 ; 11 18 12 24 ; 12 38 12 24
13 18 24 0 ; 14 38 24 0 ; 15 18 24 24
16 38 24 24
MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14 ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16
UNIT INCH
MEMB PROP
1 2 PRISMATIC YD 12.0 IZ 509. IY 509. IX 1018.
3 TO 10 PR YD 12.0 ZD 12.0 IZ 864. IY 864. IX 1279.
11 TO 21 PR YD 21.0 ZD 16.0 IZ 5788. IY 2953. IX 6497.
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FT
SUPPORT
1 TO 6 FIXED
LOAD 1 (1.4DL + 1.7LL)
SELF Y -1.4
MEMB LOAD
11 TO 16 UNI Y -2.8
11 TO 16 UNI Y -5.1
LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
REPEAT LOAD
1 0.75
JOINT LOAD
15 16 FZ 8.5
11 FZ 20.0
12 FZ 16.0
10 FZ 8.5
PDELTA ANALYSIS
PRINT FORCES LIST 2 5 9 14 16
START CONCRETE DESIGN
CODE ACI
TRACK 1.0 MEMB 14
```

```
TRACK 2.0 MEMB 16
MAXMAIN 11 ALL
DESIGN BEAM 14 16
DESIGN COLUMN 2 5
END CONCRETE DESIGN
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:42                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FRAME WITH CONCRETE DESIGN
INPUT FILE: US-8 Concrete Design for a Space Frame.STD
     2. UNIT KIP FT
     3. JOINT COORDINATE
     4. 1 0 0 0 ; 2 18 0 0 ; 3 38 0. 0
     5. 4 0 0 24 ; 5 18 0 24 ; 6 38 0 24
     6. 7 0 12 0 ; 8 18 12 0 ; 9 38 12 0
     7. 10 0 12 24 ; 11 18 12 24 ; 12 38 12 24
     8. 13 18 24 0 ; 14 38 24 0 ; 15 18 24 24
     9. 16 38 24 24
    10. MEMBER INCIDENCE
    11. 1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
    12. 5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
    13. 9 6 12 ; 10 12 16 ; 11 7 8 12
    14. 13 10 11 14 ; 15 13 14 ; 16 15 16
    15. 17 7 10 ; 18 8 11 ; 19 9 12
    16. 20 13 15 ; 21 14 16
    17. UNIT INCH
    18. MEMB PROP
    19. 1 2 PRISMATIC YD 12.0 IZ 509. IY 509. IX 1018.
    20. 3 TO 10 PR YD 12.0 ZD 12.0 IZ 864. IY 864. IX 1279.
    21. 11 TO 21 PR YD 21.0 ZD 16.0 IZ 5788. IY 2953. IX 6497.
    22. DEFINE MATERIAL START
    23. ISOTROPIC CONCRETE
    24. E 3150
    25. POISSON 0.17
    26. DENSITY 8.7E-005
    27. ALPHA 5E-006
    28. DAMP 0.05
    29. G 1346.15
    30. TYPE CONCRETE
    31. STRENGTH FCU 4
    32. END DEFINE MATERIAL
    33. CONSTANTS
    34. MATERIAL CONCRETE ALL
    35. UNIT FT
    36. SUPPORT
```

```
   37. 1 TO 6 FIXED
   38. LOAD 1 (1.4DL + 1.7LL)
     FRAME WITH CONCRETE DESIGN                              -- PAGE NO.    2
   39. SELF Y -1.4
   40. MEMB LOAD
   41. 11 TO 16 UNI Y -2.8
   42. 11 TO 16 UNI Y -5.1
   43. LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
   44. REPEAT LOAD
   45. 1 0.75
   46. JOINT LOAD
   47. 15 16 FZ 8.5
   48. 11 FZ 20.0
   49. 12 FZ 16.0
   50. 10 FZ 8.5
   51. PDELTA ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          16  NUMBER OF MEMBERS      21
    NUMBER OF PLATES           0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES         0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      60
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  ++ Adjusting Displacements.
   52. PRINT FORCES LIST 2 5 9 14 16
 FORCES   LIST     2        5
     FRAME WITH CONCRETE DESIGN                              -- PAGE NO.    3
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     2    1     4     68.01     -4.02    -0.69      0.00      2.85     -17.70
               10    -66.03      4.02     0.69     -0.00      5.48     -30.94
          2     4     54.80     -3.35    -6.58      1.03     41.93     -15.37
               10    -53.31      3.35     6.58     -1.03     41.50     -25.37
     5    1     5    289.52     -0.45    -0.72      0.00      3.08      -4.21
               11   -286.99      0.45     0.72     -0.00      5.54      -3.04
          2     5    227.65     -0.88   -12.16      0.90     88.74      -6.81
               11   -225.76      0.88    12.16     -0.90     82.17      -5.84
     9    1     6    170.70      4.47    -0.67     -0.00      2.79      16.47
               12   -168.18     -4.47     0.67      0.00      5.27      36.05
          2     6    139.15      2.84   -13.36      0.17     92.01       8.79
               12   -137.26     -2.84    13.36     -0.17     84.24      24.01
    14    1    11     -9.12     97.25     0.00     -0.23     -0.01     371.86
               12      9.12     70.57    -0.00      0.23     -0.00    -104.95
          2    11     -7.75     73.11     0.64     -0.87     -8.77     280.25
               12      7.75     52.76    -0.64      0.87     -3.99     -76.74
    16    1    15     13.58     84.52    -0.00      0.03     -0.00     104.14
               16    -13.58     83.30     0.00     -0.03      0.00     -91.95
          2    15     10.19     63.36     0.07     -0.08     -0.79      78.03
               16    -10.19     62.50    -0.07      0.08     -0.59     -69.48
  ************** END OF LATEST ANALYSIS RESULT **************
   53. START CONCRETE DESIGN
CONCRETE DESIGN
   54. CODE ACI
   55. TRACK 1.0 MEMB 14
```

```
    56. TRACK 2.0 MEMB 16
    57. MAXMAIN 11 ALL
    58. DESIGN BEAM 14 16
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    4
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    5
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************
Units: KIP   , FEET   (Unless Noted Otherwise)
 Member :     14
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length:   20.000 |
| Critical Ratio :   0.886        Criteria: Flexure                         |
| Critical Clause:   9.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
-------------------------------------------------------
| Shape: Rectangular | Width:    1.33 | Depth:    1.75 |
-------------------------------------------------------
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |      Location       | Distance  |  Anchor  |
|        Position | Nums | Size | Start  |    End     | From Face | Start End |
---------------------------------------------------------------------------
|          Bottom |    3 | # 6  |    0.00|     20.00 |      0.16 |  Yes  Yes |
|          Bottom |    3 | # 6  |    4.52|     18.65 |      0.16 |   No   No |
|             Top |    3 | #10  |    0.00|     20.00 |      0.18 |  Yes  Yes |
|             Top |    2 | #10  |    0.00|      9.55 |      0.18 |  Yes   No |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                                 |         Asv         |     Rebar Specification      |
| Zone | Dir.|  From    |    To    |  Reqd.   |  Prov.   | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
|    1 |  Y  |    0.00 |     4.82 |     0.00 |     0.00 |   22 |  # 4 |   0.23  |   2  |
|    1 |  Z  |    0.00 |     4.82 |     0.00 |     0.00 |   22 |  # 4 |   0.23  |   2  |
|    2 |  Y  |    4.82 |     5.00 |     0.00 |     0.00 |    2 |  # 4 |   0.18  |   2  |
|    2 |  Z  |    4.82 |     5.00 |     0.00 |     0.00 |    2 |  # 4 |   0.18  |   2  |
|    3 |  Y  |    5.00 |    10.00 |     0.00 |     0.00 |   23 |  # 4 |   0.23  |   2  |
|    3 |  Z  |    5.00 |    10.00 |     0.00 |     0.00 |   23 |  # 4 |   0.23  |   2  |
|    4 |  Y  |   10.00 |    15.00 |     0.00 |     0.00 |    9 |  # 4 |   0.63  |   2  |
|    4 |  Z  |   10.00 |    15.00 |     0.00 |     0.00 |    9 |  # 4 |   0.63  |   2  |
|    5 |  Y  |   15.00 |    20.00 |     0.00 |     0.00 |   15 |  # 4 |   0.36  |   2  |
|    5 |  Z  |   15.00 |    20.00 |     0.00 |     0.00 |   15 |  # 4 |   0.36  |   2  |
------------------------------------------------------------------------------------
------------------------ Member:    14 Design Ends ------------------------
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    6
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************
Units: KIP   , FEET   (Unless Noted Otherwise)
 Member :     16
DESIGN SUMMARY
-----------------------------------------------------------------------------
| Status         :   Pass         Type    : Beam           Length:   20.000 |
| Critical Ratio :   0.943        Criteria: Flexure                         |
| Critical Clause:   9.5.2                                                  |
-----------------------------------------------------------------------------
CROSS SECTION
```

```
---------------------------------------------------------
| Shape: Rectangular | Width:    1.33 | Depth:    1.75 |
---------------------------------------------------------
DESIGN INPUTS
---------------------------------------------------------------------------
| Concrete | Fc          576.000 |                       | Ec     0.519E+06 |
| Steel    | Fy(main)  8639.999 | Fy(trans)  8639.999 | Es     0.418E+07 |
| Cover    | Top          0.125 | Bottom        0.125 | Sides      0.125 |
---------------------------------------------------------------------------
Design Messages
---------------------------------------------------------------------------
   WARNINGS: DESIGN FOR MEMBER    16
   ---------------------------------
    1) Review shear min density/max spacing Cl 9.7.6.2.2
CRITICAL STRENGTH RESULTS
-------------------------------------------------------------------------
| Category |    Demand     | Min Capacity  | Max Capacity  |  Ratio  |
-------------------------------------------------------------------------
| Axial    |        -13.584|      -699.120 |       192.780 |  0.019 |
| Flexure  |       -104.139|      -110.383 |       199.179 |  0.943 |
| Shear Y  |         84.523|      -127.636 |       127.636 |  0.662 |
| Shear Z  |         -0.073|       -24.061 |        24.061 |  0.003 |
| Torsion  |          0.000|         0.000 |         0.000 |  0.000 |
-------------------------------------------------------------------------
LONGITUDINAL BAR DETAILS AT CROSS SECTIONS
-----------------------------------------------------------------------------------
| Distance | Position |  Ast-reqd   |  Ast-prov   | No(s)bars| Size | No of Layers |
-----------------------------------------------------------------------------------
|    0.000 |    Top   |     0.008 |       0.008 |    6     | # 4 |       1        |
|          |  Bottom  |     0.010 |       0.016 |    3     | # 8 |       1        |
|    5.000 |    Top   |     0.003 |       0.008 |    6     | # 4 |       1        |
|          |  Bottom  |     0.018 |       0.033 |    6     | # 8 |       1        |
|   10.000 |    Top   |     0.003 |       0.008 |    6     | # 4 |       1        |
|          |  Bottom  |     0.030 |       0.033 |    6     | # 8 |       1        |
|   15.000 |    Top   |     0.003 |       0.008 |    6     | # 4 |       1        |
|          |  Bottom  |     0.019 |       0.033 |    6     | # 8 |       1        |
|   20.000 |    Top   |     0.007 |       0.008 |    6     | # 4 |       1        |
|          |  Bottom  |     0.010 |       0.016 |    3     | # 8 |       1        |
-----------------------------------------------------------------------------------
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    7
LONGITUDINAL BAR LAYOUT
---------------------------------------------------------------------------
|                 |    Bars     |    Location      | Distance  |  Anchor   |
|        Position | Nums | Size | Start  |   End   | From Face | Start End |
---------------------------------------------------------------------------
|          Bottom |    3 | # 8 |    0.00|   20.00 |    0.17   |  Yes  Yes |
|          Bottom |    3 | # 8 |    1.36|   18.77 |    0.17   |   No   No |
|             Top |    6 | # 4 |    0.00|   20.00 |    0.15   |  Yes  Yes |
---------------------------------------------------------------------------
TRANSVERSE BAR LAYOUT
------------------------------------------------------------------------------------
|                               |        Asv        |     Rebar Specification      |
| Zone | Dir.|   From    |    To    |  Reqd.  |  Prov.  | Nums | Size | Spacing | Legs |
------------------------------------------------------------------------------------
|    1 |  Y  |    0.00 |   10.00 |    0.00 |    0.00 |   40 | # 4 |   0.26   |   2  |
|    1 |  Z  |    0.00 |   10.00 |    0.00 |    0.00 |   40 | # 4 |   0.26   |   2  |
|    2 |  Y  |   10.00 |   15.00 |    0.00 |    0.00 |   13 | # 4 |   0.42   |   2  |
|    2 |  Z  |   10.00 |   15.00 |    0.00 |    0.00 |   13 | # 4 |   0.42   |   2  |
```

```
|    3 |  Y   |   15.00 |   20.00 |     0.00 |     0.00 |    20 |  # 4 |   0.26   |    2 |
|    3 |  Z   |   15.00 |   20.00 |     0.00 |     0.00 |    20 |  # 4 |   0.26   |    2 |
-----------------------------------------------------------------------------------------
------------------------ Member:    16 Design Ends ------------------------
    59. DESIGN COLUMN 2 5
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    8
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    9
                       STAAD.PRO CONCRETE DESIGN - (ACI-318-14)   v2.0
                       ***********************************************
Units: KIP   , FEET   (Unless Noted Otherwise)
  |---------------------------------------------------------------------|
  |   Member :      2                                                     |
  |  Type   : Column           Shape  : Circular                         |
  |---------------------------------------------------------------------|
  |  AREA OF STEEL REQUIRED                                             |
  |---------------------------------------------------------------------|
  |  Section -        |    1    |    2    |    3    |    4    |    5    |
  |---------------------------------------------------------------------|
  |  Location         |    0.00 |    3.00 |    6.00 |    9.00 |   12.00 |
  |  As(Longitudinal)|    0.02 |    0.01 |    0.01 |    0.01 |    0.02 |
  |  As/sv(Trans Y)  |    0.00 |    0.00 |    0.00 |    0.00 |    0.00 |
  |  As/sv(Trans Z)  |    0.00 |    0.00 |    0.00 |    0.00 |    0.00 |
  |---------------------------------------------------------------------|
  |---------------------------------------------------------------------|
  |   Member :      5                                                     |
  |  Type   : Column           Shape  : Rectangular                      |
  |---------------------------------------------------------------------|
  |  AREA OF STEEL REQUIRED                                             |
  |---------------------------------------------------------------------|
  |  Section -        |    1    |    2    |    3    |    4    |    5    |
  |---------------------------------------------------------------------|
  |  Location         |    0.00 |    3.00 |    6.00 |    9.00 |   12.00 |
  |  As(Longitudinal)|    0.06 |    0.02 |    0.01 |    0.01 |    0.05 |
  |  As/sv(Trans Y)  |    0.00 |    0.00 |    0.00 |    0.00 |    0.00 |
  |  As/sv(Trans Z)  |    0.00 |    0.00 |    0.00 |    0.00 |    0.00 |
  |---------------------------------------------------------------------|
    60. END CONCRETE DESIGN
    61. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:46:45 ****
      FRAME WITH CONCRETE DESIGN                               -- PAGE NO.   10
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. US-9 Modeling Slabs and Shear Walls Using Finite Elements

The space frame structure in this example consists of frame members and finite elements (plates). The finite element part is used to model floor slabs and a shear wall. Concrete design of an element is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-9 Modeling Slabs and Shear Walls Using Finite Elements.STD` when you install the program.

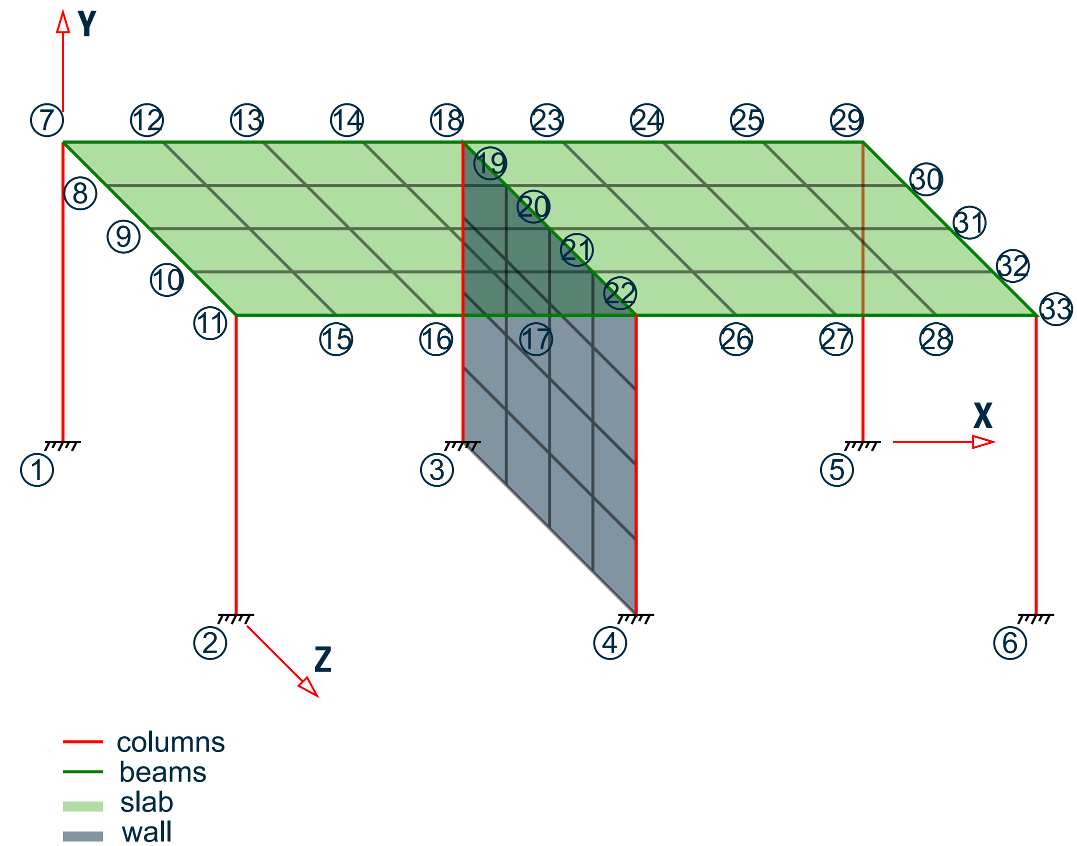


Figure 520: Example Problem No. 9

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND FINITE ELEMENTS
```

Every STAAD input file has to begin with the word `STAAD`. The word `SPACE` signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line forms the title to identify this project.

```
UNIT FEET KIP
```

The units for the data that follows are specified above.

```
JOINT COORD
1 0 0 0 ; 2 0 0 20
REP ALL 2 20 0 0
7 0 15 0 11 0 15 20
12 5 15 0 14 15 15 0
15 5 15 20 17 15 15 20
18 20 15 0 22 20 15 20
23 25 15 0 25 35 15 0
26 25 15 20 28 35 15 20
29 40 15 0 33 40 15 20
```

```
34 20 3.75 0 36 20 11.25 0
37 20 3.75 20 39 20 11.25 20
```

The joint numbers and their coordinates are defined through the above set of commands. The automatic generation facility has been used several times in the above lines. See TR.11 Joint Coordinates Specification (on page 2217) where the joint coordinate generation facilities are described.

```
MEMBER INCI
*COLUMNS
1 1 7 ; 2 2 11
3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
11 5 29 ; 12 6 33
*BEAMS IN Z DIRECTION AT X=0
13 7 8 16
*BEAMS IN Z DIRECTION AT X=20
17 18 19 20
*BEAMS IN Z DIRECTION AT X=40
21 29 30 24
*BEAMS IN X DIRECTION AT Z = 0
25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
*BEAMS IN X DIRECTION AT Z = 20
33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
```

The member incidences are defined through the above set of commands. For some members, the member number followed by the start and end joint numbers are defined. In other cases, STAAD's automatic generation facilities are utilized. Refer to TR.12 Member Incidences Specification (on page 2221) for additional details.

```
DEFINE MESH
A JOINT 7
B JOINT 11
C JOINT 22
D JOINT 18
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
```

The above lines define the nodes of super-elements. Super-elements are plate/shell surfaces from which a number of individual plate/shell elements can be generated. In this case, the points describe the outer corners of a slab and that of a shear wall. Our goal is to define the slab and the wall as several plate/shell elements.

```
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
```

The above lines form the instructions to generate individual 4-noded elements from the super-element profiles. For example, the command `MESH ABCD 4 4` means that STAAD.Pro has to generate 16 elements from the surface formed by the points A, B, C and D with 4 elements along the edges AB & CD and 4 elements along the edges BC & DA.

```
MEMB PROP
1 TO 40 PRIS YD 1 ZD 1
```

Members 1 to 40 are defined as a rectangular prismatic section with 1 ft depth and 1 ft width.

```
ELEM PROP
41 TO 88 TH 0.5
```

Elements 41 to 88 are defined to be 0.5 ft thick.

```
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 6 FIXED
```

Joints 1 to 6 are defined as fixed supported.

```
UNIT FEET
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -1.0
```

Load 1 consists of a pressure load of 1 Kip/sq.ft. intensity on elements 41 to 72. The negative sign (and the default value for the axis) indicates that the load acts opposite to the positive direction of the element local z-axis.

```
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -20.
22 FZ -100.
```

Load 2 consists of joint loads in the Z direction at joints 11, 22, and 33.

```
LOAD COMB 3
1 0.9 2 1.3
```

Load 3 is a combination of 0.9 times load case 1 and 1.3 times load case 2.

```
PERFORM ANALYSIS
```

The command to perform an elastic analysis is specified above.

```
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
```

Support reactions, members forces and element stresses are printed for load cases 1 and 3.

```
START CONCRETE DESIGN
CODE ACI
DESIGN ELEMENT 47
END CONCRETE DESIGN
```

The above set of command form the instructions to STAAD to perform a concrete design on element 47. Design is done according to the ACI 318 code. Note that design will consist only of flexural reinforcement calculations in the longitudinal and transverse directions of the elements for the moments MX and MY.

```
    FINI
```

The STAAD run is terminated.

Input File

```
STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
* FINITE ELEMENTS
UNIT FEET KIP
JOINT COORD
1 0 0 0 ; 2 0 0 20
REP ALL 2 20 0 0
7 0 15 0 11 0 15 20
12 5 15 0 14 15 15 0
15 5 15 20 17 15 15 20
18 20 15 0 22 20 15 20
23 25 15 0 25 35 15 0
26 25 15 20 28 35 15 20
29 40 15 0 33 40 15 20
34 20 3.75 0 36 20 11.25 0
37 20 3.75 20 39 20 11.25 20
MEMBER INCI
*COLUMNS
1 1 7 ; 2 2 11
3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
11 5 29 ; 12 6 33
*BEAMS IN Z DIRECTION AT X=0
13 7 8 16
*BEAMS IN Z DIRECTION AT X=20
17 18 19 20
*BEAMS IN Z DIRECTION AT X=40
21 29 30 24
*BEAMS IN X DIRECTION AT Z = 0
25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
*BEAMS IN X DIRECTION AT Z = 20
33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
DEFINE MESH
A JOINT 7
B JOINT 11
C JOINT 22
D JOINT 18
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
MEMB PROP
```

```
1 TO 40 PRIS YD 1 ZD 1
ELEM PROP
41 TO 88 TH 0.5
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 6 FIXED
UNIT FEET
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -1.0
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -20.
22 FZ -100.
LOAD COMB 3
1 0.9 2 1.3
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
START CONCRETE DESIGN
CODE ACI
DESIGN ELEMENT 47
END CONCRETE DESIGN
FINI
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:47                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: US-9 Modeling Slabs and Shear Walls Using Finite Elements.STD
     2. * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
     3. * FINITE ELEMENTS
```

```
     4. UNIT FEET KIP
     5. JOINT COORD
     6. 1 0 0 0 ; 2 0 0 20
     7. REP ALL 2 20 0 0
     8. 7 0 15 0 11 0 15 20
     9. 12 5 15 0 14 15 15 0
    10. 15 5 15 20 17 15 15 20
    11. 18 20 15 0 22 20 15 20
    12. 23 25 15 0 25 35 15 0
    13. 26 25 15 20 28 35 15 20
    14. 29 40 15 0 33 40 15 20
    15. 34 20 3.75 0 36 20 11.25 0
    16. 37 20 3.75 20 39 20 11.25 20
    17. MEMBER INCI
    18. *COLUMNS
    19. 1 1 7 ; 2 2 11
    20. 3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
    21. 7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
    22. 11 5 29 ; 12 6 33
    23. *BEAMS IN Z DIRECTION AT X=0
    24. 13 7 8 16
    25. *BEAMS IN Z DIRECTION AT X=20
    26. 17 18 19 20
    27. *BEAMS IN Z DIRECTION AT X=40
    28. 21 29 30 24
    29. *BEAMS IN X DIRECTION AT Z = 0
    30. 25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
    31. 29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
    32. *BEAMS IN X DIRECTION AT Z = 20
    33. 33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
    34. 37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
    35. DEFINE MESH
    36. A JOINT 7
    37. B JOINT 11
    38. C JOINT 22
      STAAD SPACE                                              -- PAGE NO.    2
    * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
    39. D JOINT 18
    40. E JOINT 33
    41. F JOINT 29
    42. G JOINT 3
    43. H JOINT 4
    44. GENERATE ELEMENT
    45. MESH ABCD 4 4
    46. MESH DCEF 4 4
    47. MESH DCHG 4 4
    48. MEMB PROP
    49. 1 TO 40 PRIS YD 1 ZD 1
    50. ELEM PROP
    51. 41 TO 88 TH 0.5
    52. UNIT INCH
    53. DEFINE MATERIAL START
    54. ISOTROPIC CONCRETE
    55. E 3000
    56. POISSON 0.17
    57. DENSITY 8.7E-005
    58. ALPHA 5E-006
    59. DAMP 0.05
```

```
   60. G 1346.15
   61. TYPE CONCRETE
   62. STRENGTH FCU 4
   63. END DEFINE MATERIAL
   64. CONSTANTS
   65. MATERIAL CONCRETE ALL
   66. SUPPORT
   67. 1 TO 6 FIXED
   68. UNIT FEET
   69. LOAD 1 DEAD LOAD FROM FLOOR
   70. ELEMENT LOAD
   71. 41 TO 72 PRESSURE -1.0
   72. LOAD 2 WIND LOAD
   73. JOINT LOAD
   74. 11 33 FZ -20.
   75. 22 FZ -100.
   76. LOAD COMB 3
   77. 1 0.9 2 1.3
   78. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
     NUMBER OF JOINTS         69  NUMBER OF MEMBERS      40
     NUMBER OF PLATES         48  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
      STAAD SPACE                                           -- PAGE NO.    3
    * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     378
   TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
   *** NOTE: CAPACITY FOR MAXIMUM #    254 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
    79. LOAD LIST 1 3
    80. PRINT SUPP REAC
  SUPP     REAC
      STAAD SPACE                                           -- PAGE NO.    4
    * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      9.11     82.56     11.46     56.95     -0.01    -45.28
           3      8.23     74.62     10.65     54.04      0.03    -40.81
      2    1      9.11     82.56    -11.46    -56.95      0.01    -45.28
           3      8.19     73.98     -9.97    -48.33      0.19    -40.77
      3    1      0.00    234.89     75.78    -39.25      0.00      0.00
           3      0.00    345.09    160.48    -19.17      0.00      0.00
      4    1      0.00    234.89    -75.78     39.25      0.00      0.00
           3      0.00     77.69     20.17     50.98      0.00      0.00
      5    1     -9.11     82.56     11.46     56.95      0.01     45.28
           3     -8.23     74.62     10.65     54.04     -0.03     40.81
      6    1     -9.11     82.56    -11.46    -56.95     -0.01     45.28
           3     -8.19     73.98     -9.97    -48.33     -0.19     40.77
   ************** END OF LATEST ANALYSIS RESULT **************
    81. PRINT MEMBER FORCES LIST 27
  MEMBER   FORCES   LIST     27
      STAAD SPACE                                           -- PAGE NO.    5
    * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
```

```
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    27    1    13      0.73    -13.21    -0.07     25.88      0.18     -80.00
               14     -0.73     13.21     0.07    -25.88      0.18      13.96
          3    13      5.42    -11.85    -0.21     23.44      0.51     -72.04
               14     -5.42     11.85     0.21    -23.44      0.53      12.79
  ************** END OF LATEST ANALYSIS RESULT **************
   82. PRINT ELEMENT STRESSES LIST 47
 ELEMENT  STRESSES LIST     47
    STAAD SPACE                                            -- PAGE NO.    6
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
  ----------------
          STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
    47      1         1.71        0.50      -11.52      -14.78        1.47
                    330.22      326.90       -1.21       -1.70        0.52
                    370.35      366.37
      TOP : SMAX=    -263.74 SMIN=    -370.35 TMAX=      53.30 ANGLE= 21.2
      BOTT: SMAX=     366.37 SMIN=     261.89 TMAX=      52.24 ANGLE=-69.1
            3         1.53        0.45      -10.40      -13.30        1.29
                    299.11      292.24       -1.83       -2.46        4.00
                    336.09      326.02
      TOP : SMAX=    -236.95 SMIN=    -336.09 TMAX=      49.57 ANGLE= 22.4
      BOTT: SMAX=     326.02 SMIN=     238.44 TMAX=      43.79 ANGLE=-71.0
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         3.663715E+02 -3.703516E+02  5.330365E+01  3.302169E+02  3.703516E+02
PLATE NO.      47            47            47            47            47
CASE  NO.       1             1             1             1             1
  ********************END OF ELEMENT FORCES********************
   83. START CONCRETE DESIGN
    STAAD SPACE                                            -- PAGE NO.    7
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
CONCRETE DESIGN
   84. CODE ACI
   85. DESIGN ELEMENT 47
    STAAD SPACE                                            -- PAGE NO.    8
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
   **WARNING** - ELEMENT DESIGN to ACI 318-14 is not available in this version
of STAAD.Pro
   86. END CONCRETE DESIGN
   87. FINI
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46:48 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
```

```
*                   http://www.bentley.com                       *
******************************************************************
```

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)

EX. US-10 Finite Element Model for a Rectangular Tank

A tank structure is modeled with four-noded plate elements. Water pressure from inside is used as loading for the tank. Reinforcement calculations have been done for some elements.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-10 Finite Element Model for a Rectangular Tank.STD` when you install the program.

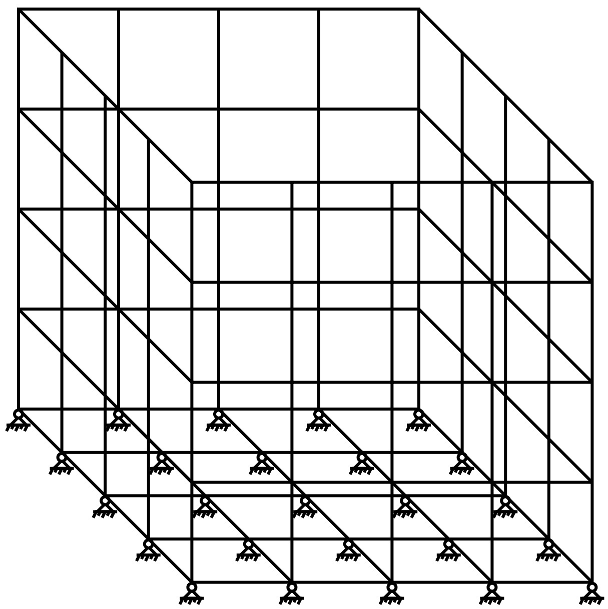


Figure 521: Tank Model

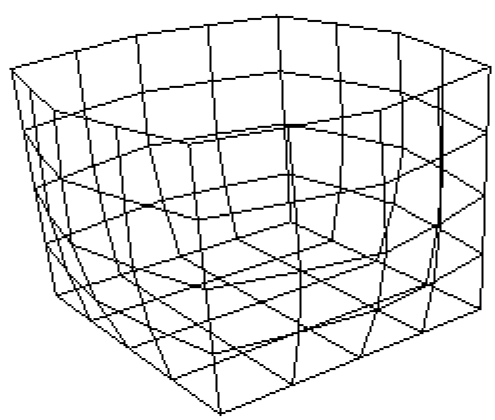


Figure 522: Deflected Shape

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK
* STRUCTURE
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D) structure.

```
UNITS FEET KIPS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0.  0.  0.  5  0.  20.  0.
REPEAT 4  5.  0.  0.
REPEAT 4  0.  0.  5.
REPEAT 4  -5.  0.  0.
REPEAT 3  0.  0.  -5.
81  5.  0.  5.  83  5.  0.  15.
REPEAT 2  5.  0.  0.
```

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command generates joint coordinates by repeating the pattern of the previous line of joint coordinates. The number following the REPEAT command is the number of repetitions to be carried out. This is followed by X, Y and Z coordinate increments. See TR.11 Joint Coordinates Specification (on page 2217) .

```
ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
78 87 26 31 88
79 88 31 36 89
80 89 36 41 46
```

Element connectivities are input as above by providing the element number followed by joint numbers defining the element. The REPEAT command generates element incidences by repeating the pattern of the previous line of element nodes. The number following the REPEAT command is the number of repetitions to be carried out and that is followed by element and joint number increments. See TR.12 Member Incidences Specification (on page 2221) .

```
UNIT INCHES
ELEMENT PROPERTIES
1 TO 80 TH 8.0
```

Element properties are provided by specifying that the elements are 8.0 inches THick.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
```

```
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED
```

Pinned supports are specified at the joints listed above. No moments will be carried by these supports. The expression `1 TO 76 BY 5` means 1, 6, 11, etc. up to 76.

```
UNIT FT
LOAD 1
ELEMENT LOAD
4 TO 64 BY 4 PR 1.
3 TO 63 BY 4 PR 2.
2 TO 62 BY 4 PR 3.
1 TO 61 BY 4 PR 4.
```

Load case 1 is initiated. It consists of element loads in the form of uniform pressure(indicated by `PR`) acting along the local z-axis.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
UNIT INCHES
PRINT JOINT DISPLACEMENTS LIST  5  25  45  65
```

The joint displacement values for the listed nodes will be reported in the output file as a result of the above command.

```
PRINT ELEM FORCE LIST 13 16
PRINT ELEM STRESS LIST 9 12
```

Two types of results are requested for elements. The first one requests the nodal point forces in the global axes directions to be reported for elements 13 and 16. The second one requests element centroid stresses in the element local axes directions to be reported for elements 9 and 12. These results will appear in a tabular form in the output file.

```
START CONCRETE DESIGN
```

The above command initiates concrete design.

```
CODE ACI
DESIGN SLAB  9  12
```

Slabs (i.e., elements) 9 and 12 will be designed and the reinforcement requirements obtained. In STAAD, elements are typically designed for the moments MX and MY at the centroid of the element.

```
END CONCRETE DESIGN
```

Terminates the concrete design operation.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
UNITS FEET KIPS
JOINT COORDINATES
1 0. 0. 0. 5 0. 20. 0.
REPEAT 4 5. 0. 0.
REPEAT 4 0. 0. 5.
REPEAT 4 -5. 0. 0.
REPEAT 3 0. 0. -5.
81 5. 0. 5. 83 5. 0. 15.
REPEAT 2 5. 0. 0.
ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
78 87 26 31 88
79 88 31 36 89
80 89 36 41 46
UNIT INCHES
ELEMENT PROPERTIES
1 TO 80 TH 8.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3000.
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED
UNIT FT
LOAD 1
ELEMENT LOAD
```

```
4 TO 64 BY 4 PR 1.
3 TO 63 BY 4 PR 2.
2 TO 62 BY 4 PR 3.
1 TO 61 BY 4 PR 4.
PERFORM ANALYSIS
UNIT INCHES
PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
PRINT ELEM FORCE LIST 13 16
PRINT ELEM STRESS LIST 9 12
START CONCRETE DESIGN
CODE ACI
DESIGN SLAB 9 12
END CONCRETE DESIGN
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:28                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
INPUT FILE: US-10 Finite Element Model for a Rectangular Tank.STD
     2. UNITS FEET KIPS
     3. JOINT COORDINATES
     4. 1 0. 0. 0. 5 0. 20. 0.
     5. REPEAT 4 5. 0. 0.
     6. REPEAT 4 0. 0. 5.
     7. REPEAT 4 -5. 0. 0.
     8. REPEAT 3 0. 0. -5.
     9. 81 5. 0. 5. 83 5. 0. 15.
    10. REPEAT 2 5. 0. 0.
    11. ELEMENT INCIDENCES
    12. 1 1 2 7 6 TO 4 1 1
    13. REPEAT 14 4 5
    14. 61 76 77 2 1 TO 64 1 1
    15. 65 1 6 81 76
    16. 66 76 81 82 71
    17. 67 71 82 83 66
    18. 68 66 83 56 61
    19. 69 6 11 84 81
    20. 70 81 84 85 82
    21. 71 82 85 86 83
    22. 72 83 86 51 56
    23. 73 11 16 87 84
    24. 74 84 87 88 85
    25. 75 85 88 89 86
    26. 76 86 89 46 51
    27. 77 16 21 26 87
    28. 78 87 26 31 88
```

```
   29. 79 88 31 36 89
   30. 80 89 36 41 46
   31. UNIT INCHES
   32. ELEMENT PROPERTIES
   33. 1 TO 80 TH 8.0
   34. DEFINE MATERIAL START
   35. ISOTROPIC CONCRETE
   36. E 3000.
   37. POISSON 0.17
   38. DENSITY 8.7E-005
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    2
   39. ALPHA 5E-006
   40. DAMP 0.05
   41. G 1346.15
   42. TYPE CONCRETE
   43. STRENGTH FCU 4
   44. END DEFINE MATERIAL
   45. CONSTANTS
   46. MATERIAL CONCRETE ALL
   47. SUPPORT
   48. 1 TO 76 BY 5 81 TO 89 PINNED
   49. UNIT FT
   50. LOAD 1
   51. ELEMENT LOAD
   52. 4 TO 64 BY 4 PR 1.
   53. 3 TO 63 BY 4 PR 2.
   54. 2 TO 62 BY 4 PR 3.
   55. 1 TO 61 BY 4 PR 4.
   56. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         89  NUMBER OF MEMBERS       0
    NUMBER OF PLATES         80  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     25
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     459
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   57. UNIT INCHES
   58. PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
 JOINT    DISPLACE LIST     5
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    3
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     5    1   -0.00403   0.00027  -0.00403   0.00021   0.00000  -0.00021
    25    1    0.00403   0.00027  -0.00403   0.00021   0.00000   0.00021
    45    1    0.00403   0.00027   0.00403  -0.00021   0.00000   0.00021
    65    1   -0.00403   0.00027   0.00403  -0.00021   0.00000  -0.00021
  ************** END OF LATEST ANALYSIS RESULT **************
   59. PRINT ELEM FORCE LIST 13 16
 ELEM     FORCE    LIST     13
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    4
  ELEMENT FORCES    FORCE,LENGTH UNITS= KIPS INCH
  --------------
    GLOBAL CORNER FORCES
JOINT     FX          FY          FZ          MX          MY          MZ
```

```
              ELE.NO.     13 FOR LOAD CASE     1
   16 -2.3570E+01 -1.8550E+01  2.1089E+01 -1.3649E+02  2.4437E+02  1.3427E+02
   17 -7.0147E+00 -1.5328E+01  8.0498E+00 -8.5004E+01 -4.0990E+01 -2.1558E+02
   22  4.0892E+01  5.9984E+00  4.3154E+01 -1.4891E+02  5.4757E+02  2.6486E+02
   21 -1.0308E+01  2.7879E+01  2.7707E+01  2.9817E+02  5.0071E+02 -1.8356E+02
              ELE.NO.     16 FOR LOAD CASE     1
   19 -3.7438E+01  3.4537E+00 -2.2002E+01 -2.4192E+02  1.9474E+02  2.4055E+02
   20 -3.6565E+01  6.0104E-01 -2.0799E+01  2.5567E+02  2.1788E+02 -4.0380E+02
   25  3.2510E+01  1.7764E-15  3.2510E+01 -4.9347E+02  1.4400E+03  4.9347E+02
   24  4.1493E+01 -4.0547E+00  3.5291E+01  5.2703E+02  1.4655E+03 -3.3022E+02
   60. PRINT ELEM STRESS LIST 9 12
 ELEM     STRESS   LIST     9
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    5
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KIPS INCH
  ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                     VONT       VONB         SX           SY           SXY
                     TRESCAT    TRESCAB
     9      1         0.15        0.01       -5.43         5.56         5.60
                      1.35        1.20        0.02         0.08         0.03
                      1.55        1.39
      TOP : SMAX=       0.83 SMIN=      -0.72 TMAX=       0.78 ANGLE= 67.3
      BOTT: SMAX=       0.74 SMIN=      -0.65 TMAX=       0.69 ANGLE=-22.8
    12      1        -0.02       -0.04       -0.11        20.01        -0.21
                      2.05        1.72       -0.00         0.16        -0.01
                      2.05        1.72
      TOP : SMAX=       2.04 SMIN=      -0.01 TMAX=       1.03 ANGLE=-89.3
      BOTT: SMAX=       0.01 SMIN=      -1.71 TMAX=       0.86 ANGLE=  0.4
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         2.040916E+00 -1.712074E+00  1.026498E+00  2.046982E+00  2.052995E+00
PLATE NO.      12            12            12            12            12
CASE  NO.       1             1             1             1             1
  ********************END OF ELEMENT FORCES********************
   61. START CONCRETE DESIGN
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    6
CONCRETE DESIGN
   62. CODE ACI
   63. DESIGN SLAB 9 12
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    7
   **WARNING** - ELEMENT DESIGN to ACI 318-14 is not available in this version
of STAAD.Pro
   64. END CONCRETE DESIGN
   65. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:28 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
```

```
          *                  http://www.bentley.com                      *
          ********************************************************************
```

Related Links

- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.11 Joint Coordinates Specification* (on page 2217)

EX. US-11 Response Spectrum Analysis of a Frame

Dynamic analysis (Response Spectrum) is performed for a steel structure. Results of a static and dynamic analysis are combined. The combined results are then used for steel design.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-11 Response Spectrum Analysis of a Frame.STD` when you install the program.

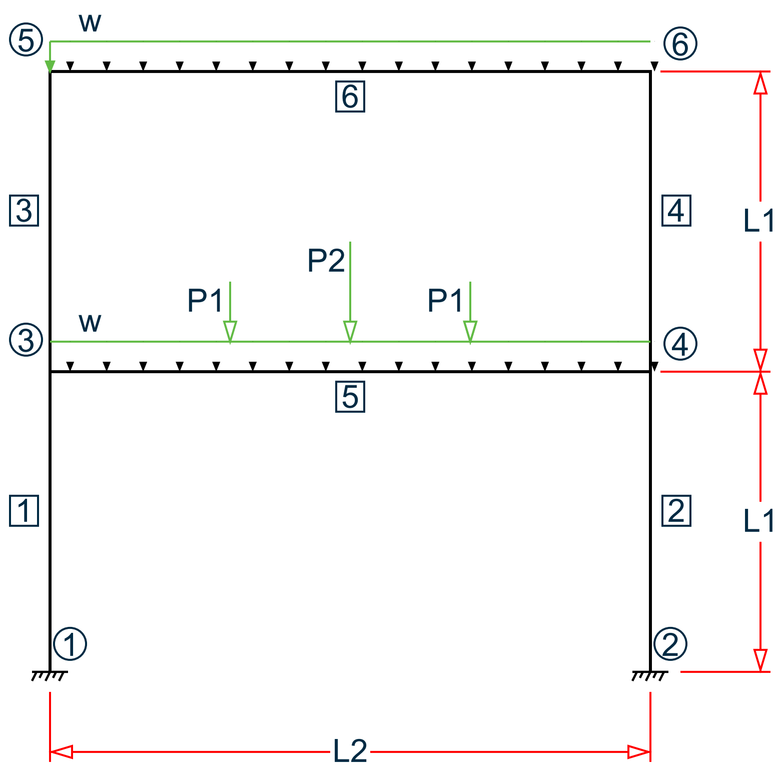


Figure 523: Example Problem No. 11

Where:

L1 = 10 ft, L2 = 20 ft
w = 1.5 k/ft
P1 = 5.0 kips, P2 = 7.5 kips

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT FEET KIPS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0  0  0  ;  2  20  0  0
3  0  10  0  ;  4  20  10  0
5  0  20  0  ;  6  20  20  0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are all the same, in this case, zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTIES AMERICAN
1 TO 4 TA ST W10X33
5 TA ST W12X40
6 TA ST W8X40
```

Properties for all members are assigned from the American (AISC) steel table. The word ST stands for standard single section.

```
SUPPORTS
1 2 FIXED
```

Fixed supports are specified at joints 1 and 2.

```
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

Material constants such as E (modulus of elasticity), Poisson's ratio and density (DEN) are specified above. Length unit is changed from FT to INCH to facilitate the input.

```
CUT OFF MODE SHAPE 2
```

The number of mode shapes to be considered in dynamic analysis is set to 2. Without the above command, this will be set to the default. See TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) .

```
* LOAD 1 WILL BE STATIC LOAD
UNIT FEET
LOAD 1 DEAD AND LIVE LOADS
```

Load case 1 is initiated followed by a title. Prior to this, the length unit is changed to METER for specifying distributed member loads. A line starting with an asterisk (*) mark indicates a comment line.

```
SELFWEIGHT Y -1.0
```

The above command indicates that the selfweight of the structure acting in the global Y direction is part of this load case. The factor of -1.0 is meant to indicate that the load acts opposite to the positive direction of global Y, hence downwards.

```
MEMBER LOADS
5  CON  GY  -5.0  6.0
5  CON  GY  -7.5  10.0
5  CON  GY  -5.0  14.0
5  6  UNI  Y  -1.5
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The word UNI stands for uniformly distributed load while CON stands for concentrated load. GY is followed by the value of the load and the distance at which it is applied.

```
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
```

The two lines which begin with the asterisk are comment lines which tell us the purpose of the next load case. Load case 2 is then initiated along with an optional title. This will be a dynamic load case. Permanent masses will be provided in the form of loads. These masses (in terms of loads) will be considered for the eigensolution. Internally, the program converts these loads to masses, hence it is best to specify them as absolute values (without a negative sign). Also, the direction (X, Y, Z etc.) of the loads will correspond to the dynamic degrees of freedom in which the masses are capable of vibrating. In a PLANE frame, only X and Y directions need to be considered. In a SPACE frame, masses (loads) should be provided in all three (X, Y and Z) directions if they are active along all three. The user has the freedom to restrict one or more directions.

```
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
```

The above commands indicate that the selfweight of the structure acting in the global X and Y directions with a factor of 1.0 is taken into consideration for the mass matrix.

```
MEMBER LOADS
5  CON  GX  5.0  6.0
5  CON  GY  5.0  6.0
5  CON  GX  7.5  10.0
5  CON  GY  7.5  10.0
5  CON  GX  5.0  14.0
5  CON  GY  5.0  14.0
```

The mass matrix will also consist of terms derived from the above member loads. GX and GY indicate that the load, and hence the resulting mass, is capable of vibration along the global X and Y directions. The word CON stands for concentrated load. Concentrated forces of 5, 7.5, and 5 kips are located at 6ft, 10ft and 14ft from the start of member 5.

```
SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
ZIP 92806 SITE CLASS D TL 12.0
```

The `SPECTRUM` command specifies a 2012 International Buidling Code / ASCE 7-10 seismic response spectrum load. The modal responses will be combined using the CQC method. Here, the spectrum effect is in the global X direction with a factor of 1.0. IBC 2012 response spectra are always given in terms of acceleration (`ACC`). A damping ratio of 0.05 (5%) is used. The second line then gives the location and site class values (using the US ZIP postal code and IBC/ASCE site class) along with the long period transition time.

```
    LOAD LIST  1  3  4
    PARAMETER
    CODE AISC UNIFIED
    SELECT ALL
```

A steel design in the form of a member selection is performed based on the rules of the American Code. Only the member forces resulting from load cases 1, 3 and 4 will be considered for these calculations.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
UNIT FEET KIPS
JOINT COORDINATES
1 0 0 0 ; 2 20 0 0
3 0 10 0 ; 4 20 10 0
5 0 20 0 ; 6 20 20 0
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
MEMBER PROPERTIES AMERICAN
1 TO 4 TA ST W10X33
5 TA ST W12X40
6 TA ST W8X40
SUPPORTS
1 2 FIXED
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
CUT OFF MODE SHAPE 2
*LOAD 1 WILL BE STATIC LOAD
UNIT FEET
LOAD 1 DEAD AND LIVE LOADS
SELFWEIGHT Y -1.0
MEMBER LOADS
5 CON GY -5.0  6.0
5 CON GY -7.5 10.0
5 CON GY -5.0 14.0
5 6 UNI Y -1.5
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
```

```
SELFWEIGHT Y 1.0
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
ZIP 92806 SITE CLASS D TL 12.0
LOAD COMBINATION 3
1 0.75 2 0.75
LOAD COMBINATION 4
1 0.75 2 -0.75
PERFORM ANALYSIS PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
LOAD LIST 1 3 4
PARAMETER
CODE AISC UNIFIED
SELECT ALL
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:30                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE RESPONSE SPECTRUM ANALYSIS
INPUT FILE: US-11 Response Spectrum Analysis of a Frame.STD
     2. UNIT FEET KIPS
     3. JOINT COORDINATES
     4. 1 0 0 0 ; 2 20 0 0
     5. 3 0 10 0 ; 4 20 10 0
     6. 5 0 20 0 ; 6 20 20 0
     7. MEMBER INCIDENCES
     8. 1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
     9. 5 3 4 ; 6 5 6
    10. MEMBER PROPERTIES AMERICAN
    11. 1 TO 4 TA ST W10X33
    12. 5 TA ST W12X40
    13. 6 TA ST W8X40
    14. SUPPORTS
    15. 1 2 FIXED
    16. UNIT INCH
    17. DEFINE MATERIAL START
    18. ISOTROPIC STEEL
    19. E 29000.
    20. POISSON 0.3
    21. DENSITY 283E-006
```

```
   22. ALPHA 6E-006
   23. DAMP 0.03
   24. TYPE STEEL
   25. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
   26. END DEFINE MATERIAL
   27. CONSTANT
   28. MATERIAL STEEL ALL
   29. CUT OFF MODE SHAPE 2
   30. *LOAD 1 WILL BE STATIC LOAD
   31. UNIT FEET
   32. LOAD 1 DEAD AND LIVE LOADS
   33. SELFWEIGHT Y -1.0
   34. MEMBER LOADS
   35. 5 CON GY -5.0  6.0
   36. 5 CON GY -7.5 10.0
   37. 5 CON GY -5.0 14.0
   38. 5 6 UNI Y -1.5
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    2
   39. * NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
   40. * WITH MASSES PROVIDED IN TERMS OF LOAD.
   41. LOAD 2 SEISMIC LOADING
   42. SELFWEIGHT X 1.0
   43. SELFWEIGHT Y 1.0
   44. MEMBER LOADS
   45. 5 CON GX 5.0 6.0
   46. 5 CON GY 5.0 6.0
   47. 5 CON GX 7.5 10.0
   48. 5 CON GY 7.5 10.0
   49. 5 CON GX 5.0 14.0
   50. 5 CON GY 5.0 14.0
   51. SPECTRUM CQC IBC 2012 X 1.0 ACC DAMP 0.05
   52. ZIP 92806 SITE CLASS D TL 12.0
   53. LOAD COMBINATION 3
   54. 1 0.75 2 0.75
   55. LOAD COMBINATION 4
   56. 1 0.75 2 -0.75
   57. PERFORM ANALYSIS PRINT MODE SHAPES
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          6  NUMBER OF MEMBERS       6
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      12
  TOTAL LOAD COMBINATION  CASES =     2  SO FAR.
  ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED             =     2
NUMBER OF EXISTING MASSES IN THE MODEL =     8
NUMBER OF MODES THAT WILL BE USED     =     2
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    3
              CALCULATED FREQUENCIES FOR LOAD CASE       2
      MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                      4.493                  0.22258
```

```
        2                      16.308                  0.06132
      RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    4
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     1    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     1    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     1    0.67185   0.00308   0.00000  0.000E+00  0.000E+00 -3.367E-03
     4     1    0.67185  -0.00308   0.00000  0.000E+00  0.000E+00 -3.367E-03
     5     1    1.00000   0.00360   0.00000  0.000E+00  0.000E+00 -1.467E-03
     6     1    1.00000  -0.00360   0.00000  0.000E+00  0.000E+00 -1.467E-03
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     2   -0.08832   0.00516   0.00000  0.000E+00  0.000E+00 -2.620E-03
     4     2   -0.08832  -0.00516   0.00000  0.000E+00  0.000E+00 -2.620E-03
     5     2    1.00000   0.00780   0.00000  0.000E+00  0.000E+00 -7.148E-03
     6     2    1.00000  -0.00780   0.00000  0.000E+00  0.000E+00 -7.148E-03
  RESPONSE SPECTRUM LOAD     2
     RESPONSE LOAD CASE      2
            MODAL WEIGHT (MODAL MASS TIMES g) IN KIPS         GENERALIZED
      MODE           X            Y             Z             WEIGHT
         1       1.984110E+01  2.545506E-33  0.000000E+00    9.680166E+00
         2       2.373892E-01  2.473935E-28  0.000000E+00    1.272852E+00
     CQC           MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   2.007858E+01  2.007858E+01  0.000000E+00 KIPS
     MISSING WEIGHT X Y Z  -9.338434E-05 -2.007858E+01  0.000000E+00 KIPS
       MODAL WEIGHT X Y Z   2.007849E+01  2.473961E-28  0.000000E+00 KIPS
      RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    5
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
              1                      1.03090       0.05000
              2                      0.74526       0.05000
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KIPS LENGTH IN FEET
  -----------------------------------------------------------
                                                            MOMENTS ARE ABOUT THE
ORIGIN
   MODE     PERIOD        FX           FY           FZ          MX          MY
MZ
      1      0.223       20.45       -0.00         0.00        0.00        0.00
-222.06
      2      0.061        0.18       -0.00         0.00        0.00        0.00
2.19
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT         BASE SHEAR IN KIPS
           --------------------------------------         ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   98.82  0.00  0.00   98.817    0.000    0.000     20.45     0.00     0.00
   2    1.18  0.00  0.00   99.999    0.000    0.000      0.18     0.00     0.00
                                                    ---------------------------
                                  TOTAL SRSS  SHEAR     20.46     0.00     0.00
                                  TOTAL 10PCT SHEAR     20.46     0.00     0.00
                                  TOTAL ABS   SHEAR     20.63     0.00     0.00
                                  TOTAL CQC   SHEAR     20.46     0.00     0.00
    58. PRINT ANALYSIS RESULTS
```

```
 ANALYSIS RESULTS
     RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    6
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          4    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     2    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
          4    0.00000   0.00000   0.00000   0.00000   0.00000   0.00000
     3    1   -0.00151  -0.01706   0.00000   0.00000   0.00000  -0.00248
          2    0.48043   0.00220   0.00000   0.00000   0.00000   0.00241
          3    0.35919  -0.01115   0.00000   0.00000   0.00000  -0.00005
          4   -0.36146  -0.01445   0.00000   0.00000   0.00000  -0.00366
     4    1    0.00151  -0.01706   0.00000   0.00000   0.00000   0.00248
          2    0.48043   0.00220   0.00000   0.00000   0.00000   0.00241
          3    0.36146  -0.01115   0.00000   0.00000   0.00000   0.00366
          4   -0.35919  -0.01445   0.00000   0.00000   0.00000   0.00005
     5    1    0.00313  -0.02369   0.00000   0.00000   0.00000  -0.00244
          2    0.71519   0.00258   0.00000   0.00000   0.00000   0.00105
          3    0.53874  -0.01584   0.00000   0.00000   0.00000  -0.00104
          4   -0.53404  -0.01970   0.00000   0.00000   0.00000  -0.00262
     6    1   -0.00313  -0.02369   0.00000   0.00000   0.00000   0.00244
          2    0.71519   0.00258   0.00000   0.00000   0.00000   0.00105
          3    0.53404  -0.01584   0.00000   0.00000   0.00000   0.00262
          4   -0.53874  -0.01970   0.00000   0.00000   0.00000   0.00104
     RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    7
  SUPPORT REACTIONS -UNIT KIPS FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      4.57     40.20      0.00      0.00      0.00    -14.33
          2     10.23      5.16      0.00      0.00      0.00     59.43
          3     11.10     34.02      0.00      0.00      0.00     33.83
          4     -4.24     26.28      0.00      0.00      0.00    -55.32
     2    1     -4.57     40.20      0.00      0.00      0.00     14.33
          2     10.23      5.16      0.00      0.00      0.00     59.43
          3      4.24     34.02      0.00      0.00      0.00     55.32
          4    -11.10     26.28      0.00      0.00      0.00    -33.83
     RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    8
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIPS FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1     40.20     -4.57     0.00      0.00      0.00     -14.33
                3    -39.87      4.57     0.00      0.00      0.00     -31.39
          2     1      5.16     10.23     0.00      0.00      0.00      59.43
                3      5.16     10.23     0.00      0.00      0.00      42.85
          3     1     34.02      4.24     0.00      0.00      0.00      33.83
                3    -33.78     -4.24     0.00      0.00      0.00     -55.68
          4     1     26.28    -11.10     0.00      0.00      0.00     -55.32
                3    -26.03     11.10     0.00      0.00      0.00       8.59
     2    1     2     40.20      4.57     0.00      0.00      0.00      14.33
                4    -39.87     -4.57     0.00      0.00      0.00      31.39
          2     2      5.16     10.23     0.00      0.00      0.00      59.43
                4      5.16     10.23     0.00      0.00      0.00      42.85
```

```
          3     2     34.02     11.10      0.00      0.00      0.00     55.32
                4    -33.78    -11.10      0.00      0.00      0.00     -8.59
          4     2     26.28     -4.24      0.00      0.00      0.00    -33.83
                4    -26.03      4.24      0.00      0.00      0.00     55.68
     3    1     3     15.73     -8.85      0.00      0.00      0.00    -44.36
                5    -15.40      8.85      0.00      0.00      0.00    -44.14
          2     3      0.89      0.85      0.00      0.00      0.00      1.21
                5      0.89      0.85      0.00      0.00      0.00      8.86
          3     3     12.46     -6.00      0.00      0.00      0.00    -32.36
                5    -12.21      6.00      0.00      0.00      0.00    -39.74
          4     3     11.13     -7.27      0.00      0.00      0.00    -34.18
                5    -10.88      7.27      0.00      0.00      0.00    -26.46
     4    1     4     15.73      8.85      0.00      0.00      0.00     44.36
                6    -15.40     -8.85      0.00      0.00      0.00     44.14
          2     4      0.89      0.85      0.00      0.00      0.00      1.21
                6      0.89      0.85      0.00      0.00      0.00      8.86
          3     4     12.46      7.27      0.00      0.00      0.00     34.18
                6    -12.21     -7.27      0.00      0.00      0.00     26.46
          4     4     11.13      6.00      0.00      0.00      0.00     32.36
                6    -10.88     -6.00      0.00      0.00      0.00     39.74
     5    1     3     -4.28     24.15      0.00      0.00      0.00     75.75
                4      4.28     24.15      0.00      0.00      0.00    -75.75
          2     3      0.00      4.23      0.00      0.00      0.00     42.32
                4      0.00      4.23      0.00      0.00      0.00     42.32
          3     3     -3.21     21.28      0.00      0.00      0.00     88.55
                4      3.21     14.94      0.00      0.00      0.00    -88.55
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    9
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIPS FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          4     3     -3.21     14.94      0.00      0.00      0.00     25.08
                4      3.21     21.28      0.00      0.00      0.00    -25.08
     6    1     5      8.85     15.40      0.00      0.00      0.00     44.14
                6     -8.85     15.40      0.00      0.00      0.00    -44.14
          2     5      0.00      0.89      0.00      0.00      0.00      8.86
                6      0.00      0.89      0.00      0.00      0.00      8.86
          3     5      6.64     12.21      0.00      0.00      0.00     39.74
                6     -6.64     10.88      0.00      0.00      0.00    -39.74
          4     5      6.64     10.88      0.00      0.00      0.00     26.46
                6     -6.64     12.21      0.00      0.00      0.00    -26.46
  ************** END OF LATEST ANALYSIS RESULT **************
   59. LOAD LIST 1 3 4
   60. PARAMETER
   61. CODE AISC UNIFIED
   62. SELECT ALL
PARAMETER 1
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.   10
                    STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                  ********************************************************
ALL UNITS ARE - KIPS FEET (UNLESS OTHERWISE Noted).
***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
*** AXIS CONVENTION ***:
========================
The capacity results and intermediate results in the report follow the notations
and axes labels as defined in the AISC 360-16 code.
The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
design results are reported in AISC 360-16 code axis convention.
```

```
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       X               Z          Axis typically parallel to the sections principal
major axis.
       Y               Y          Axis typically parallel to the sections principal
minor axis.
       Z               X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
 =======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), while
       the results for shear are reported as being for shear forces �along� the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear along
       the X axis.
 *** ABBREVIATIONS ***:
 ======================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.   11
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIPS FEET (UNLESS OTHERWISE Noted).
 - Member :     1
|------------------------------------------------------------------------------|
|  Member No:        1       Profile:  ST  W10X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.851       Loadcase:        3     |
|  Location:     10.00       Ref:      Eq.H1-3b                                |
|  Pz:       33.78     C     Vy:        4.242           Vx:      0.000         |
|  Tz:       0.000           My:        0.000           Mx:      55.68         |
|------------------------------------------------------------------------------|
 - Member :     2
|------------------------------------------------------------------------------|
|  Member No:        2       Profile:  ST  W10X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.846       Loadcase:        3     |
|  Location:      0.00       Ref:      Eq.H1-3b                                |
|  Pz:       34.02     C     Vy:        11.10           Vx:      0.000         |
|  Tz:       0.000           My:        0.000           Mx:      55.32         |
|------------------------------------------------------------------------------|
 - Member :     3
|------------------------------------------------------------------------------|
```

```
|  Member No:        3       Profile:  ST  W12X22              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.951      Loadcase:        1     |
|  Location:      0.00       Ref:      Eq.H1-1a                               |
|  Pz:       15.73     C     Vy:       -8.850          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:     -44.36         |
|-----------------------------------------------------------------------------|
- Member :     4
|-----------------------------------------------------------------------------|
|  Member No:        4       Profile:  ST  W12X22              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.951      Loadcase:        1     |
|  Location:      0.00       Ref:      Eq.H1-1a                               |
|  Pz:       15.73     C     Vy:        8.850          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      44.36         |
|-----------------------------------------------------------------------------|
- Member :     5
|-----------------------------------------------------------------------------|
|  Member No:        5       Profile:  ST  W12X35              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.980      Loadcase:        3     |
|  Location:      0.00       Ref:      Cl.F2.2                                |
|  Pz:       3.209     T     Vy:        21.28          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      88.55         |
|-----------------------------------------------------------------------------|
- Member :     6
|-----------------------------------------------------------------------------|
|  Member No:        6       Profile:  ST  W12X26              (AISC SECTIONS)|
|  Status:        PASS       Ratio:         0.821      Loadcase:        1     |
|  Location:      0.00       Ref:      Eq.H1-3b                               |
|  Pz:       8.850     C     Vy:        15.40          Vx:      0.000         |
|  Tz:       0.000           My:        0.000          Mx:      44.14         |
|-----------------------------------------------------------------------------|
   63. FINISH
  ******************************************************************************
  **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
              IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
              ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
              TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
              FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
              FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
              COMMANDS.
  ******************************************************************************
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:32 ****
     RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.   12
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-12 Moving Load Generation on a Bridge Deck

This example demonstrates generation of load cases for the type of loading known as a moving load. This type of loading occurs classically when the load-causing units move on the structure, as in the case of trucks on a bridge deck. The mobile loads are discretized into several individual immobile load cases at discrete positions. During this process, enormous number of load cases may be created resulting in plenty of output to be sorted. To avoid looking into a lot of output, the maximum force envelope is requested for a few specific members.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-12 Moving Load Generation on a Bridge Deck.STD` when you install the program.

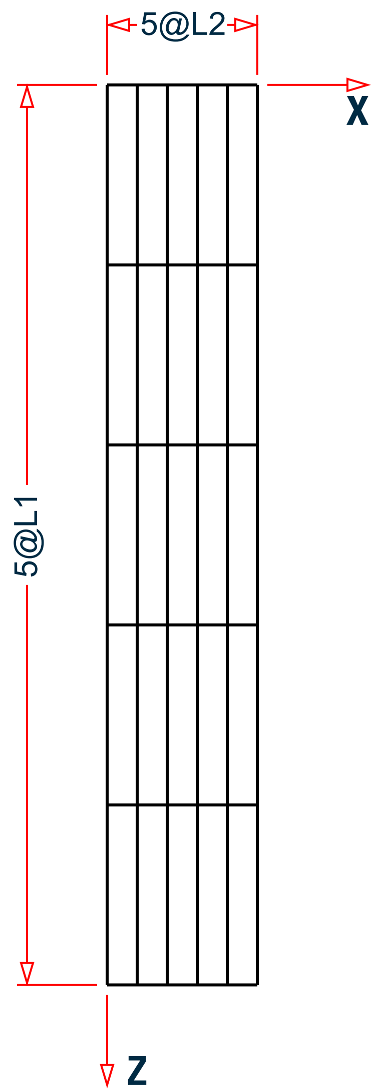


Figure 524: Example Problem No. 12

Where:

L1 = 30 ft
L2 = 5 ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD FLOOR A SIMPLE BRIDGE DECK
```

Every input has to start with the term `STAAD`. The word `FLOOR` signifies that the structure is a floor structure and the geometry is defined through X and Z axis.

```
UNITS FEET KIPS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0  0  0  6  25  0  0
R  5  0  0  30
```

Joint number followed by X, Y, and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). The first line generates joints 1 through 6. With the repeat (`R`) command, the coordinates of the next 30 joints are generated by repeating the pattern of the coordinates of the first 6 joints 5 times with X, Y and Z increments of 0,0, and 9 respectively.

```
MEMBER INCIDENCES
1 1 7 6
7 1 2 11
R A 4 11 6
56 31 32 60
```

Defines the members by the joints to which they are connected. The fourth number indicates the final member number up to which they will be generated. Repeat all (abbreviated as `R A`) will create members by repeating the member incidence pattern of the previous 11 members. The number of repetitions to be carried out is provided after the R A command and the member increment and joint increment are defined as 11 and 6 respectively. The fifth line of input defines the member incidences for members 56 to 60.

```
MEMBER PROPERTIES AMERICAN
1 TO 60 TA ST W12X26
```

Member properties are assigned from the American AISC table for all members. The word ST stands for standard single section.

```
SUPPORTS
1 TO 6 31 TO 36 PINNED
```

Pinned supports are specified at the above joints. A pinned support is one which can resist only translational forces.

```
UNITS INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The input units are changed from FT to INCH. The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
UNIT FEET KIP
DEFINE MOVING LOAD
TYPE 1 LOAD  20.  20.  10.  DISTANCE  10.  5.  WIDTH  10.0
```

The characteristics of the vehicle are defined above in METER and KNS units. The above lines represent the first out of two sets of data required in moving load generation. The type number (1) is a label for identification of the load-causing unit, such as a truck. Three axles (20 20 10) are specified with the LOAD command. The spacing between the axles in the direction of movement (longitudinal direction) is specified after the DISTANCE command. WIDTH is the spacing in the transverse direction, that is, it is the distance between the 2 prongs of an axle of the truck.

```
LOAD 1
```

Load case 1 is initiated.

```
SELF Y -1.0
```

Selfweight of the structure acting in the negative (due to the factor -1.0) global Y direction is the only component of load case 1.

```
LOAD GENERATION  10
TYPE  1  7.5  0.  0.  ZI  10.
```

This constitutes the second of the two sets of data required for moving load generation. 10 load cases are generated using the Type 1 vehicle whose characteristics were described earlier. For the first of these load cases, the X, Y and Z location of the reference load (see TR.31.1 Definition of Moving Load System (on page 2346)) have been specified after the command TYPE 1. The Z Increment of 10ft denotes that the vehicle moves along the Z direction and the individual positions which are 10ft apart will be used to generate the remaining 9 load cases.

When defining a moving load in STAAD.Pro, the reference wheel is on the last axle. The first load case which is generated will be the one for which the first axle is just about to enter the bridge. The last load case should be the one for which the last axle is just about to exit the bridge. Thus, the total distance travelled by the reference load will be the length of the vehicle (distance from first axle to last axle) plus the span of the bridge. In this problem, that comes to

$$(10+5) + 150 = 165 \text{ feet.}$$

This example uses 10 ft increments and generates 10 load cases.

However, if you want the vehicle to move forward in 15 feet increments (each 15 foot increment will create a discrete position of the truck on the bridge), it would required (165/15)+1 = 12 cases to be generated.

```
PERFORM ANALYSIS PRINT LOAD
```

The above command instructs the program to proceed with the analysis and print the values and positions of all the generated load cases.

```
PRINT MAXFORCE ENVELOP LIST 3 41 42
```

A maximum force envelope consisting of the highest forces for each degree of freedom on the listed members will be written into the output file.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD FLOOR A SIMPLE BRIDGE DECK
UNITS FEET KIPS
JOINT COORDINATES
1 0 0 0 6 25 0 0
R 5 0 0 30
MEMBER INCIDENCES
1 1 7 6
7 1 2 11
R A 4 11 6
56 31 32 60
MEMBER PROPERTIES AMERICAN
1 TO 60 TA ST W12X26
SUPPORTS
1 TO 6 31 TO 36 PINNED
UNITS INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
UNIT FEET KIP
DEFINE MOVING LOAD
TYPE 1 LOAD 20. 20. 10. DISTANCE 10. 5. WIDTH 10.
LOAD 1
SELF Y -1.0
LOAD GENERATION 10
TYPE 1 7.5 0. 0. ZI 10.
PERFORM ANALYSIS PRINT LOAD
PRINT MAXFORCE ENVELOP LIST 3 41 42
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:34                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD FLOOR A SIMPLE BRIDGE DECK
INPUT FILE: US-12 Moving Load Generation on a Bridge Deck.STD
     2. UNITS FEET KIPS
```

```
   3. JOINT COORDINATES
   4. 1 0 0 0 6 25 0 0
   5. R 5 0 0 30
   6. MEMBER INCIDENCES
   7. 1 1 7 6
   8. 7 1 2 11
   9. R A 4 11 6
  10. 56 31 32 60
  11. MEMBER PROPERTIES AMERICAN
  12. 1 TO 60 TA ST W12X26
  13. SUPPORTS
  14. 1 TO 6 31 TO 36 PINNED
  15. UNITS INCH
  16. DEFINE MATERIAL START
  17. ISOTROPIC STEEL
  18. E 29000
  19. POISSON 0.3
  20. DENSITY 283E-006
  21. ALPHA 6E-006
  22. DAMP 0.03
  23. TYPE STEEL
  24. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  25. END DEFINE MATERIAL
  26. CONSTANT
  27. MATERIAL STEEL ALL
  28. UNIT FEET KIP
  29. DEFINE MOVING LOAD
  30. TYPE 1 LOAD 20. 20. 10. DISTANCE 10. 5. WIDTH 10.
  31. LOAD 1
  32. SELF Y -1.0
  33. LOAD GENERATION 10
  34. TYPE 1 7.5 0. 0. ZI 10.
  35. PERFORM ANALYSIS PRINT LOAD
    A SIMPLE BRIDGE DECK                                    -- PAGE NO.    2
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS         36  NUMBER OF MEMBERS      60
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     12
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =    11, TOTAL DEGREES OF FREEDOM =      96
 TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    A SIMPLE BRIDGE DECK                                    -- PAGE NO.    3
 LOADING     1
 -----------
     SELFWEIGHT  Y   -1.000
    ACTUAL WEIGHT OF THE STRUCTURE =      27.278 KIP
 LOADING     2
 -----------
 MEMBER LOAD - UNIT KIP  FEET
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    8                           -20.0000 GY   2.50
   10                           -20.0000 GY   2.50
    3                           -10.0000 GY  10.00
    2                           -10.0000 GY  10.00
    5                           -10.0000 GY  10.00
    4                           -10.0000 GY  10.00
```

```
     3                           -5.0000 GY  15.00
     2                           -5.0000 GY  15.00
     5                           -5.0000 GY  15.00
     4                           -5.0000 GY  15.00
  LOADING     3
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     3                          -10.0000 GY  10.00
     2                          -10.0000 GY  10.00
     5                          -10.0000 GY  10.00
     4                          -10.0000 GY  10.00
     3                          -10.0000 GY  20.00
     2                          -10.0000 GY  20.00
     5                          -10.0000 GY  20.00
     4                          -10.0000 GY  20.00
     3                           -5.0000 GY  25.00
     2                           -5.0000 GY  25.00
     5                           -5.0000 GY  25.00
     4                           -5.0000 GY  25.00
  LOADING     4
  -----------
     A SIMPLE BRIDGE DECK                                  -- PAGE NO.    4
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     3                          -10.0000 GY  20.00
     2                          -10.0000 GY  20.00
     5                          -10.0000 GY  20.00
     4                          -10.0000 GY  20.00
    19                          -20.0000 GY   2.50
    21                          -20.0000 GY   2.50
    14                           -5.0000 GY   5.00
    13                           -5.0000 GY   5.00
    16                           -5.0000 GY   5.00
    15                           -5.0000 GY   5.00
  LOADING     5
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    19                          -20.0000 GY   2.50
    21                          -20.0000 GY   2.50
    14                          -10.0000 GY  10.00
    13                          -10.0000 GY  10.00
    16                          -10.0000 GY  10.00
    15                          -10.0000 GY  10.00
    14                           -5.0000 GY  15.00
    13                           -5.0000 GY  15.00
    16                           -5.0000 GY  15.00
    15                           -5.0000 GY  15.00
  LOADING     6
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    14                          -10.0000 GY  10.00
    13                          -10.0000 GY  10.00
    16                          -10.0000 GY  10.00
    15                          -10.0000 GY  10.00
    14                          -10.0000 GY  20.00
```

```
    13                           -10.0000 GY  20.00
    16                           -10.0000 GY  20.00
    15                           -10.0000 GY  20.00
    14                            -5.0000 GY  25.00
    13                            -5.0000 GY  25.00
     A SIMPLE BRIDGE DECK                                        -- PAGE NO.    5
    16                            -5.0000 GY  25.00
    15                            -5.0000 GY  25.00
  LOADING     7
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    14                           -10.0000 GY  20.00
    13                           -10.0000 GY  20.00
    16                           -10.0000 GY  20.00
    15                           -10.0000 GY  20.00
    30                           -20.0000 GY   2.50
    32                           -20.0000 GY   2.50
    25                            -5.0000 GY   5.00
    24                            -5.0000 GY   5.00
    27                            -5.0000 GY   5.00
    26                            -5.0000 GY   5.00
  LOADING     8
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    30                           -20.0000 GY   2.50
    32                           -20.0000 GY   2.50
    25                           -10.0000 GY  10.00
    24                           -10.0000 GY  10.00
    27                           -10.0000 GY  10.00
    26                           -10.0000 GY  10.00
    25                            -5.0000 GY  15.00
    24                            -5.0000 GY  15.00
    27                            -5.0000 GY  15.00
    26                            -5.0000 GY  15.00
  LOADING     9
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    25                           -10.0000 GY  10.00
    24                           -10.0000 GY  10.00
    27                           -10.0000 GY  10.00
    26                           -10.0000 GY  10.00
     A SIMPLE BRIDGE DECK                                        -- PAGE NO.    6
    25                           -10.0000 GY  20.00
    24                           -10.0000 GY  20.00
    27                           -10.0000 GY  20.00
    26                           -10.0000 GY  20.00
    25                            -5.0000 GY  25.00
    24                            -5.0000 GY  25.00
    27                            -5.0000 GY  25.00
    26                            -5.0000 GY  25.00
  LOADING    10
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    25                           -10.0000 GY  20.00
```

```
    24                          -10.0000 GY  20.00
    27                          -10.0000 GY  20.00
    26                          -10.0000 GY  20.00
    41                          -20.0000 GY   2.50
    43                          -20.0000 GY   2.50
    36                           -5.0000 GY   5.00
    35                           -5.0000 GY   5.00
    38                           -5.0000 GY   5.00
    37                           -5.0000 GY   5.00
  LOADING    11
  -----------
  MEMBER LOAD - UNIT KIP  FEET
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    41                          -20.0000 GY   2.50
    43                          -20.0000 GY   2.50
    36                          -10.0000 GY  10.00
    35                          -10.0000 GY  10.00
    38                          -10.0000 GY  10.00
    37                          -10.0000 GY  10.00
    36                           -5.0000 GY  15.00
    35                           -5.0000 GY  15.00
    38                           -5.0000 GY  15.00
    37                           -5.0000 GY  15.00
  ************ END OF DATA FROM INTERNAL STORAGE ************
   36. PRINT MAXFORCE ENVELOP LIST 3 41 42
 MAXFORCE ENVELOP  LIST     3
     A SIMPLE BRIDGE DECK                                       -- PAGE NO.    7
        MEMBER FORCE ENVELOPE
        ---------------------
        ALL UNITS ARE KIP  FEET
  MAX AND MIN FORCE VALUES AMONGST ALL SECTION LOCATIONS
 MEMB          FY/    DIST  LD        MZ/    DIST  LD
               FZ     DIST  LD        MY     DIST  LD        FX     DIST  LD
    3 MAX     18.03   0.00    3       0.02   0.00    4
               0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
      MIN     -6.97  27.50    3    -373.90  30.00    5
               0.00  30.00   11       0.00  30.00   11      0.00    30.00   11
   41 MAX     16.33   0.00   10       6.80   5.00    5
               0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
      MIN     -4.08   5.00   11    -109.08   2.50   10
               0.00   5.00   11       0.00   5.00   11      0.00     5.00   11
   42 MAX      0.06   0.00    1       6.80   0.00    5
               0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
      MIN     -0.06   5.00    1     -99.89   5.00   10
               0.00   5.00   11       0.00   5.00   11      0.00     5.00   11
  ********** END OF FORCE ENVELOPE FROM INTERNAL STORAGE **********
   37. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:35 ****
     A SIMPLE BRIDGE DECK                                       -- PAGE NO.    8
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
```

```
         *           Copyright (c) Bentley Systems, Inc.          *
         *               http://www.bentley.com                   *
         **********************************************************
```

Related Links

- *(Moving) Load Generation Type dialog* (on page 2832)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)
- *M. To add vehicles to the load generation* (on page 874)
- *TR.31.1 Definition of Moving Load System* (on page 2346)
- *V. Moving Load Generator* (on page 3597)

EX. US-13 Section Displacements for a Frame

Calculation of displacements at intermediate points of members of a plane frame is demonstrated in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-13 Section Displacements for a Frame.STD` when you install the program.

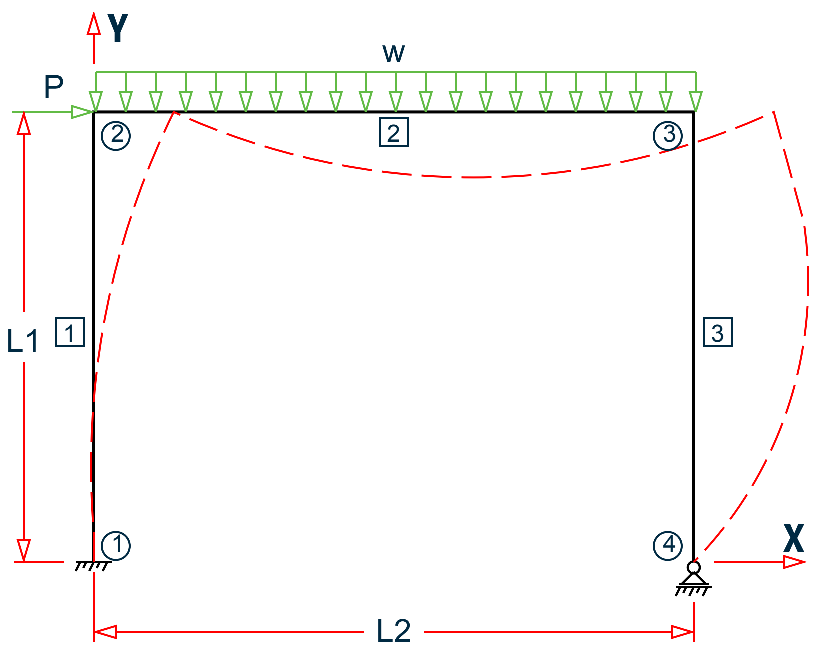


Figure 525: Example Problem No. 13

Where:

L1 = 15 ft, L2 =20 ft
P = 5.0 k
w = 3.0 k/ft

The dashed line represents the deflected shape of the structure. The shape is generated on the basis of displacements at the ends plus several intermediate points of the members.

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE TEST FOR SECTION DISPLACEMENT
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT KIP FEET
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1  0.  0.  ;  2  0.  15.  ;  3  20.  15.  ;  4  20.  0.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTY AMERICAN
1  3  TABLE  ST  W8X18
2  TABLE  ST  W12X26
```

Properties for all members are assigned from the American AISC steel table. The word `ST` stands for standard single section.

```
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.0
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The input units are changed from FT to INCH. The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 FIXED ; 4 PINNED
```

A fixed support is specified at Joint 1 and a pinned support at Joint 4.

```
UNIT FT
LOADING 1 DEAD + LIVE + WIND
```

Load case 1 is initiated followed by an optional title.

```
JOINT LOAD
2 FX 5.
```

Load 1 contains a joint load of 5 kips at node 2. FX indicates that the load is a force in the global X direction.

```
    MEMBER LOAD
    2 UNI GY -3.0
```

Load 1 contains a member load also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load.

```
    PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
    PRINT MEMBER FORCES
```

The above PRINT command is self-explanatory.

```
    *
    * FOLLOWING PRINT COMMAND WILL PRINT
    * DISPLACEMENTS OF THE MEMBERS
    * CONSIDERING EVERY TWELFTH INTERMEDIATE
    * POINT (THAT IS TOTAL OF 13 POINTS). THESE
    * DISPLACEMENTS ARE MEASURED IN GLOBAL X
    * Y Z COORDINATE SYSTEM AND THE VALUES
    * ARE FROM ORIGINAL COORDINATES (UNDEFLECTED
    * POSITION) OF CORRESPONDING TWELFTH
    * POINTS.
    *
    * MAX LOCAL DISPLACEMENT IS ALSO PRINTED.
    * THE LOCATION OF MAXIMUM INTERMEDIATE
    * DISPLACEMENT IS DETERMINED. THIS VALUE IS
    * MEASURED FROM ABOVE LOCATION TO THE
    * STRAIGHT LINE JOINING START AND END
    * JOINTS OF THE DEFLECTED MEMBER.
    *
    PRINT SECTION DISPLACEMENT
```

Above PRINT command is explained in the comment lines above.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE TEST FOR SECTION DISPLACEMENT
UNIT KIP FEET
JOINT COORDINATES
1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY AMERICAN
1 3 TABLE ST W8X18
2 TABLE ST W12X26
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.0
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT FT
LOADING 1 DEAD + LIVE + WIND
JOINT LOAD
2 FX 5.
MEMBER LOAD
2 UNI GY -3.0
PERFORM ANALYSIS
PRINT MEMBER FORCES
*
* FOLLOWING PRINT COMMAMND WILL PRINT DISPLACEMENTS
* OF THE MEMBERS CONSIDERING EVERY TWELVETH INTERMEDIATE
* POINTS (THAT IS TOTAL 13 POINTS). THESE DISPLACEMENTS
* ARE MEASURED IN GLOBAL X Y Z COORDINATE SYSTEM AND
* THE VALUES ARE FROM ORIGINAL COORDINATES (THAT IS
* UNDEFLECTED) OF CORRESPONDING TWELVETH POINTS.
*
* MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE LOCATION
* OF THE MAXIMUM INTERMEDIATE DISPLACEMENT IS DETERMINED.
* THIS VALUE IS MEASURED FROM ABOVE LOCATION TO THE STRAIGHT
* LINE JOINING START AND END JOINTS OF THE DEFLECTED MEMBER.
*
PRINT SECTION DISPLACEMENT
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:36                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE TEST FOR SECTION DISPLACEMENT
INPUT FILE: US-13 Section Displacements for a Frame.STD
     2. UNIT KIP FEET
     3. JOINT COORDINATES
     4. 1 0. 0. ; 2 0. 15. ; 3 20. 15. ; 4 20. 0.
     5. MEMBER INCIDENCE
     6. 1 1 2 ; 2 2 3 ; 3 3 4
     7. MEMBER PROPERTY AMERICAN
     8. 1 3 TABLE ST W8X18
     9. 2 TABLE ST W12X26
    10. UNIT INCHES
    11. DEFINE MATERIAL START
    12. ISOTROPIC STEEL
```

```
  13. E 29000.0
  14. POISSON 0.3
  15. DENSITY 283E-006
  16. ALPHA 6E-006
  17. DAMP 0.03
  18. TYPE STEEL
  19. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
  20. END DEFINE MATERIAL
  21. CONSTANT
  22. MATERIAL STEEL ALL
  23. SUPPORT
  24. 1 FIXED ; 4 PINNED
  25. UNIT FT
  26. LOADING 1 DEAD + LIVE + WIND
  27. JOINT LOAD
  28. 2 FX 5.
  29. MEMBER LOAD
  30. 2 UNI GY -3.0
  31. PERFORM ANALYSIS
    TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    2
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       7
 TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  32. PRINT MEMBER FORCES
MEMBER   FORCES
    TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    3
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1     27.37      0.97     0.00      0.00      0.00      22.49
               2    -27.37     -0.97     0.00      0.00      0.00      -7.88
    2    1     2      4.03     27.37     0.00      0.00      0.00       7.88
               3     -4.03     32.63     0.00      0.00      0.00     -60.39
    3    1     3     32.63      4.03     0.00      0.00      0.00      60.39
               4    -32.63     -4.03     0.00      0.00      0.00       0.00
 ************** END OF LATEST ANALYSIS RESULT **************
  33. *
  34. * FOLLOWING PRINT COMMAMND WILL PRINT DISPLACEMENTS
  35. * OF THE MEMBERS CONSIDERING EVERY TWELVETH INTERMEDIATE
  36. * POINTS (THAT IS TOTAL 13 POINTS). THESE DISPLACEMENTS
  37. * ARE MEASURED IN GLOBAL X Y Z COORDINATE SYSTEM AND
  38. * THE VALUES ARE FROM ORIGINAL COORDINATES (THAT IS
  39. * UNDEFLECTED) OF CORRESPONDING TWELVETH POINTS.
  40. *
  41. * MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE LOCATION
  42. * OF THE MAXIMUM INTERMEDIATE DISPLACEMENT IS DETERMINED.
  43. * THIS VALUE IS MEASURED FROM ABOVE LOCATION TO THE STRAIGHT
  44. * LINE JOINING START AND END JOINTS OF THE DEFLECTED MEMBER.
  45. *
  46. PRINT SECTION DISPLACEMENT
SECTION  DISPLACE
```

```
      TEST FOR SECTION DISPLACEMENT                             -- PAGE NO.    4
     MEMBER SECTION DISPLACEMENTS
     ----------------------------
     UNITS ARE - INCH
     MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
       1     1       0.0000    0.0000    0.0000    0.0173   -0.0027    0.0000
                     0.0666   -0.0054    0.0000    0.1461   -0.0081    0.0000
                     0.2539   -0.0108    0.0000    0.3882   -0.0135    0.0000
                     0.5472   -0.0162    0.0000    0.7290   -0.0188    0.0000
                     0.9318   -0.0215    0.0000    1.1538   -0.0242    0.0000
                     1.3931   -0.0269    0.0000    1.6480   -0.0296    0.0000
                     1.9165   -0.0323    0.0000
     MAX LOCAL  DISP =    0.41112   AT      90.00  LOAD    1   L/DISP=    437
       2     1       1.9165   -0.0323    0.0000    1.9162   -0.3903    0.0000
                     1.9158   -0.7221    0.0000    1.9154   -1.0010    0.0000
                     1.9151   -1.2067    0.0000    1.9147   -1.3260    0.0000
                     1.9144   -1.3523    0.0000    1.9140   -1.2856    0.0000
                     1.9136   -1.1331    0.0000    1.9133   -0.9082    0.0000
                     1.9129   -0.6316    0.0000    1.9125   -0.3303    0.0000
                     1.9122   -0.0385    0.0000
     MAX LOCAL  DISP =    1.31688   AT     120.00  LOAD    1   L/DISP=    182
       3     1       1.9122   -0.0385    0.0000    2.0720   -0.0353    0.0000
                     2.1486   -0.0321    0.0000    2.1495   -0.0289    0.0000
                     2.0822   -0.0257    0.0000    1.9544   -0.0225    0.0000
                     1.7736   -0.0192    0.0000    1.5474   -0.0160    0.0000
                     1.2833   -0.0128    0.0000    0.9890   -0.0096    0.0000
                     0.6719   -0.0064    0.0000    0.3398   -0.0032    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    0.83895   AT      75.00  LOAD    1   L/DISP=    214
         ************ END OF SECT DISPL RESULTS ***********
    47. FINISH
      TEST FOR SECTION DISPLACEMENT                             -- PAGE NO.    5
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:45:37 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. US-14 P-Delta Analysis of a Frame Under Seismic Loads

A space frame is analyzed for seismic loads. The seismic loads are generated using the procedures of the building code. A P-Delta analysis is performed to obtain the secondary effects of the lateral and vertical loads acting simultaneously.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-14 P-Delta Analysis of a Frame Under Seismic Loads.STD` when you install the program.

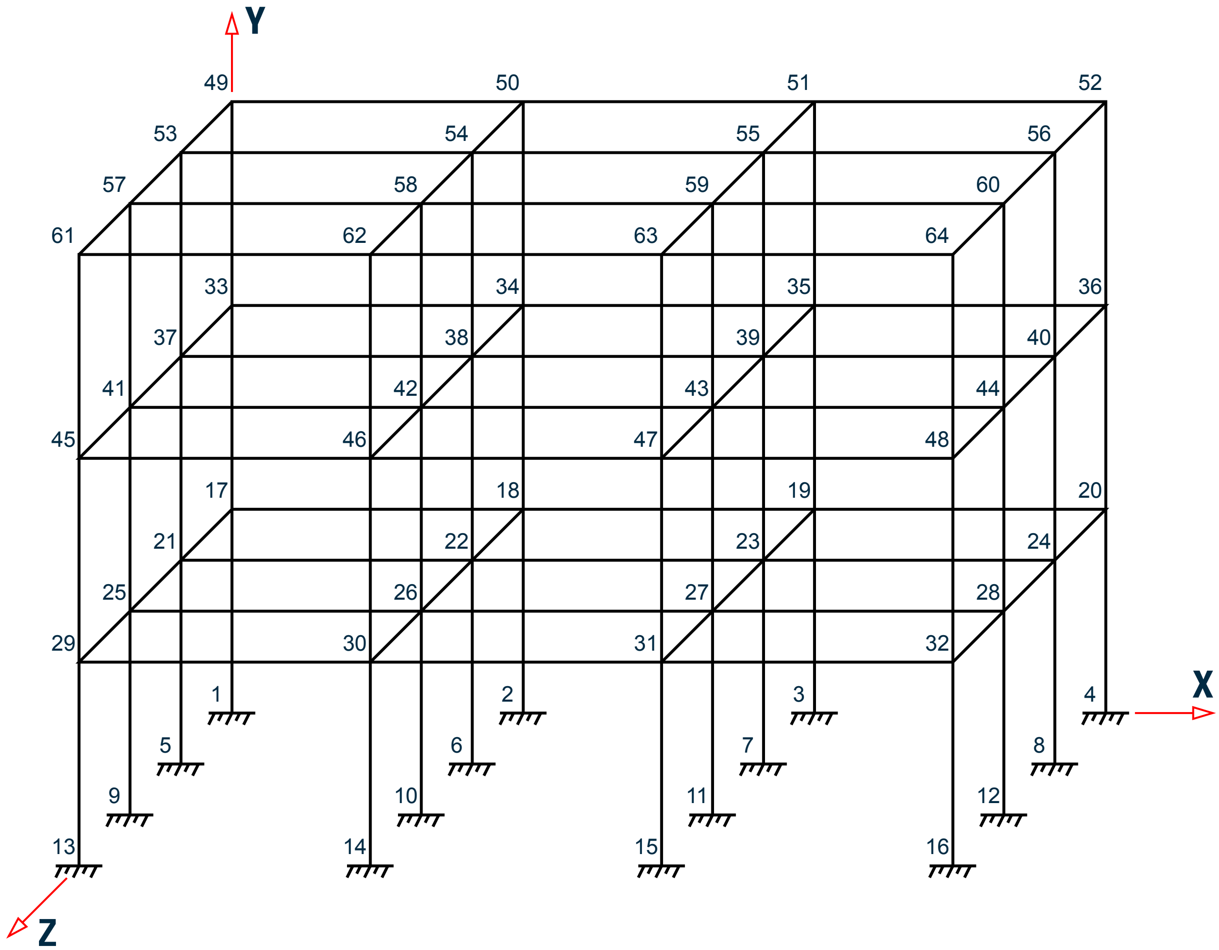


Figure 526: Example Problem No. 14

```
STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame.

```
UNIT FEET KIP
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 4 30 0 0
REPEAT 3 0 0 10
REPEAT ALL 3 0 10 0
```

The X, Y and Z coordinates of the joints are specified here. First, coordinates of joints 1 through 4 are generated by taking advantage of the fact that they are equally spaced. Then, this pattern is REPEATed 3 times with a Z increment of 3.5 m for each repetition to generate joints 5 to 16. The REPEAT ALL command will then repeat 3 times, the pattern of joints 1 to 16 to generate joints 17 to 64.

```
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
107 25 26 109
```

```
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
```

Defines the members by the joints to which they are connected. Following the specification of incidences for members 101 to 112, the REPEAT ALL command is used to repeat the pattern and generate incidences for members 113 through 136. A similar logic is used in specification of incidences of members 201 through 212 and generation of incidences for members 213 to 236. Finally, members incidences of columns 301 to 348 are specified.

```
UNIT INCH
MEMBER PROPERTIES AMERICAN
101 TO 136 201 TO 236 PRIS YD 15 ZD 15
301 TO 348 TA ST W18X35
```

The beam members have prismatic member property specification (YD & ZD) while the columns (members 301 to 348) have their properties called from the built-in American (AISC) steel table.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

Tip: You may see these values with the help of the command `PRINT MATERIAL PROPERTIES` following the preceding commands.

```
SUPPORT
1 TO 16 FIXED
```

Indicates the joints where the supports are located as well as the type of support restraints.

```
UNIT FEET
DEFINE IBC 2012 LOAD
ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
```

There are two stages in a static seismic load. The first stage is to define the code-specified load parameters along witht he vertical loads (weights) from whic the base shear will be calculated. The vertical loads may be specified in the form of selfweight, joint weights and/or member weights. Member weights are not shown in this example. It is important to note that these vertical loads are used purely in the determination of the horizontal base shear only. In other words, the structure is not analyzed for these vertical loads.

```
LOAD 1
IBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
```

This is the second stage in which the static seimsic load is applied with the help of a load case number, corresponding direction (X in the above case) and a factor by which the generated horizontal loads should be multiplied. Along with the seismic lateral load, deadweight is also added to the same load case. Since we will be doing second-order (PDELTA) analysis, it is important that we include horizontal and vertical loads in the same load case.

```
LOAD 2
IBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
```

In load case 2, the static seimsic load is being applied in the Z direction. Vertical loads are part of this case, also.

```
PDELTA ANALYSIS PRINT LOAD DATA
```

We are requesting a second-order analysis by specifying the command `PDELTA ANALYSIS`. `PRINT LOAD DATA` is used to obtain a report of all the applied and generated loadings.

```
PRINT SUPPORT REACTIONS
FINISH
```

The above commands are self-explanatory.

Input File

```
STAAD SPACE EXAMPLE PROBLEM FOR IBC LOAD
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0 4 30 0 0
REPEAT 3 0 0 10
REPEAT ALL 3 0 10 0
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
```

```
107 25 26 109
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
UNIT INCH
MEMBER PROPERTIES AMERICAN
101 TO 136 201 TO 236 PRIS YD 15 ZD 15
301 TO 348 TA ST W18X35
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
SUPPORT
1 TO 16 FIXED
UNIT FEET
DEFINE IBC 2012 LOAD
ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
IBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
LOAD 2
IBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
49 TO 64 FY -1.25
PDELTA ANALYSIS PRINT LOAD DATA
```

```
PRINT SUPPORT REACTIONS
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:39                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE PROBLEM FOR IBC LOAD
INPUT FILE: US-14 P-Delta Analysis of a Frame Under Seismic Loads.STD
     2. UNIT FEET KIP
     3. JOINT COORDINATES
     4. 1 0 0 0 4 30 0 0
     5. REPEAT 3 0 0 10
     6. REPEAT ALL 3 0 10 0
     7. MEMBER INCIDENCES
     8. * BEAMS IN X DIRECTION
     9. 101 17 18 103
    10. 104 21 22 106
    11. 107 25 26 109
    12. 110 29 30 112
    13. REPEAT ALL 2 12 16
    14. * BEAMS IN Z DIRECTION
    15. 201 17 21 204
    16. 205 21 25 208
    17. 209 25 29 212
    18. REPEAT ALL 2 12 16
    19. * COLUMNS
    20. 301 1 17 348
    21. UNIT INCH
    22. MEMBER PROPERTIES AMERICAN
    23. 101 TO 136 201 TO 236 PRIS YD 15 ZD 15
    24. 301 TO 348 TA ST W18X35
    25. DEFINE MATERIAL START
    26. ISOTROPIC STEEL
    27. E 29000
    28. POISSON 0.3
    29. DENSITY 283E-006
    30. ALPHA 6E-006
    31. DAMP 0.03
    32. TYPE STEEL
    33. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    34. ISOTROPIC CONCRETE
    35. E 3150
    36. POISSON 0.17
    37. DENSITY 8.7E-005
    38. ALPHA 5E-006
      EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    2
    39. DAMP 0.05
```

```
 40. G 1346.15
 41. TYPE CONCRETE
 42. STRENGTH FCU 4
 43. END DEFINE MATERIAL
 44. CONSTANT
 45. MATERIAL STEEL MEMB 301 TO 348
 46. MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
 47. SUPPORT
 48. 1 TO 16 FIXED
 49. UNIT FEET
 50. DEFINE IBC 2012 LOAD
 51. ZIP 92806 RX 9 RZ 9 I 1.0 TL 12.0 SCLASS 4 CT 0.032
*****************************************************************************
* EQUIV. SEISMIC LOADS AS PER IBC 2012                                      *
* PARAMETERS CONSIDERED FOR SUBSEQUENT LOAD GENERATION                      *
*  SS =  1.546 S1 =  0.587 FA =  1.000 FV =  1.500                          *
*  SDS =  1.031 SD1 =  0.587                                                *
*****************************************************************************
 52. SELFWEIGHT
 53. JOINT WEIGHT
 54. 17 TO 48 WEIGHT 2.5
 55. 49 TO 64 WEIGHT 1.25
 56. LOAD 1
 57. IBC LOAD X 0.75
 58. SELFWEIGHT Y -1.0
 59. JOINT LOADS
 60. 17 TO 48 FY -2.5
 61. 49 TO 64 FY -1.25
 62. LOAD 2
 63. IBC LOAD Z 0.75
 64. SELFWEIGHT Y -1.0
 65. JOINT LOADS
 66. 17 TO 48 FY -2.5
 67. 49 TO 64 FY -1.25
 68. PDELTA ANALYSIS PRINT LOAD DATA
   EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    3
         P R O B L E M   S T A T I S T I C S
         -----------------------------------
  NUMBER OF JOINTS         64  NUMBER OF MEMBERS     120
  NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
  NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
        Using 64-bit analysis engine.
        SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     288
TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    4
LOADING     1
-----------
    SELFWEIGHT  Y   -1.000
   ACTUAL WEIGHT OF THE STRUCTURE =     185.918 KIP
JOINT LOAD - UNIT KIP  FEET
JOINT   FORCE-X   FORCE-Y    FORCE-Z     MOM-X     MOM-Y     MOM-Z
   17      0.00     -2.50       0.00      0.00      0.00      0.00
   18      0.00     -2.50       0.00      0.00      0.00      0.00
   19      0.00     -2.50       0.00      0.00      0.00      0.00
   20      0.00     -2.50       0.00      0.00      0.00      0.00
   21      0.00     -2.50       0.00      0.00      0.00      0.00
   22      0.00     -2.50       0.00      0.00      0.00      0.00
```

```
    23      0.00     -2.50         0.00       0.00       0.00       0.00
    24      0.00     -2.50         0.00       0.00       0.00       0.00
    25      0.00     -2.50         0.00       0.00       0.00       0.00
    26      0.00     -2.50         0.00       0.00       0.00       0.00
    27      0.00     -2.50         0.00       0.00       0.00       0.00
    28      0.00     -2.50         0.00       0.00       0.00       0.00
    29      0.00     -2.50         0.00       0.00       0.00       0.00
    30      0.00     -2.50         0.00       0.00       0.00       0.00
    31      0.00     -2.50         0.00       0.00       0.00       0.00
    32      0.00     -2.50         0.00       0.00       0.00       0.00
    33      0.00     -2.50         0.00       0.00       0.00       0.00
    34      0.00     -2.50         0.00       0.00       0.00       0.00
    35      0.00     -2.50         0.00       0.00       0.00       0.00
    36      0.00     -2.50         0.00       0.00       0.00       0.00
    37      0.00     -2.50         0.00       0.00       0.00       0.00
    38      0.00     -2.50         0.00       0.00       0.00       0.00
    39      0.00     -2.50         0.00       0.00       0.00       0.00
    40      0.00     -2.50         0.00       0.00       0.00       0.00
    41      0.00     -2.50         0.00       0.00       0.00       0.00
    42      0.00     -2.50         0.00       0.00       0.00       0.00
    43      0.00     -2.50         0.00       0.00       0.00       0.00
    44      0.00     -2.50         0.00       0.00       0.00       0.00
    45      0.00     -2.50         0.00       0.00       0.00       0.00
    46      0.00     -2.50         0.00       0.00       0.00       0.00
    47      0.00     -2.50         0.00       0.00       0.00       0.00
    48      0.00     -2.50         0.00       0.00       0.00       0.00
    49      0.00     -1.25         0.00       0.00       0.00       0.00
    50      0.00     -1.25         0.00       0.00       0.00       0.00
    51      0.00     -1.25         0.00       0.00       0.00       0.00
    52      0.00     -1.25         0.00       0.00       0.00       0.00
    53      0.00     -1.25         0.00       0.00       0.00       0.00
    54      0.00     -1.25         0.00       0.00       0.00       0.00
    55      0.00     -1.25         0.00       0.00       0.00       0.00
    56      0.00     -1.25         0.00       0.00       0.00       0.00
    57      0.00     -1.25         0.00       0.00       0.00       0.00
    58      0.00     -1.25         0.00       0.00       0.00       0.00
    EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    5
    59      0.00     -1.25         0.00       0.00       0.00       0.00
    60      0.00     -1.25         0.00       0.00       0.00       0.00
    61      0.00     -1.25         0.00       0.00       0.00       0.00
    62      0.00     -1.25         0.00       0.00       0.00       0.00
    63      0.00     -1.25         0.00       0.00       0.00       0.00
    64      0.00     -1.25         0.00       0.00       0.00       0.00
 LOADING     2
 -----------
     SELFWEIGHT  Y   -1.000
    ACTUAL WEIGHT OF THE STRUCTURE =     185.918 KIP
 JOINT LOAD - UNIT KIP  FEET
 JOINT   FORCE-X   FORCE-Y    FORCE-Z     MOM-X     MOM-Y     MOM-Z
    17      0.00     -2.50         0.00       0.00       0.00       0.00
    18      0.00     -2.50         0.00       0.00       0.00       0.00
    19      0.00     -2.50         0.00       0.00       0.00       0.00
    20      0.00     -2.50         0.00       0.00       0.00       0.00
    21      0.00     -2.50         0.00       0.00       0.00       0.00
    22      0.00     -2.50         0.00       0.00       0.00       0.00
    23      0.00     -2.50         0.00       0.00       0.00       0.00
    24      0.00     -2.50         0.00       0.00       0.00       0.00
    25      0.00     -2.50         0.00       0.00       0.00       0.00
```

```
      26      0.00     -2.50        0.00      0.00      0.00      0.00
      27      0.00     -2.50        0.00      0.00      0.00      0.00
      28      0.00     -2.50        0.00      0.00      0.00      0.00
      29      0.00     -2.50        0.00      0.00      0.00      0.00
      30      0.00     -2.50        0.00      0.00      0.00      0.00
      31      0.00     -2.50        0.00      0.00      0.00      0.00
      32      0.00     -2.50        0.00      0.00      0.00      0.00
      33      0.00     -2.50        0.00      0.00      0.00      0.00
      34      0.00     -2.50        0.00      0.00      0.00      0.00
      35      0.00     -2.50        0.00      0.00      0.00      0.00
      36      0.00     -2.50        0.00      0.00      0.00      0.00
      37      0.00     -2.50        0.00      0.00      0.00      0.00
      38      0.00     -2.50        0.00      0.00      0.00      0.00
      39      0.00     -2.50        0.00      0.00      0.00      0.00
      40      0.00     -2.50        0.00      0.00      0.00      0.00
      41      0.00     -2.50        0.00      0.00      0.00      0.00
      42      0.00     -2.50        0.00      0.00      0.00      0.00
      43      0.00     -2.50        0.00      0.00      0.00      0.00
      44      0.00     -2.50        0.00      0.00      0.00      0.00
      45      0.00     -2.50        0.00      0.00      0.00      0.00
      46      0.00     -2.50        0.00      0.00      0.00      0.00
      47      0.00     -2.50        0.00      0.00      0.00      0.00
      48      0.00     -2.50        0.00      0.00      0.00      0.00
      49      0.00     -1.25        0.00      0.00      0.00      0.00
      50      0.00     -1.25        0.00      0.00      0.00      0.00
      51      0.00     -1.25        0.00      0.00      0.00      0.00
      52      0.00     -1.25        0.00      0.00      0.00      0.00
      EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    6
      53      0.00     -1.25        0.00      0.00      0.00      0.00
      54      0.00     -1.25        0.00      0.00      0.00      0.00
      55      0.00     -1.25        0.00      0.00      0.00      0.00
      56      0.00     -1.25        0.00      0.00      0.00      0.00
      57      0.00     -1.25        0.00      0.00      0.00      0.00
      58      0.00     -1.25        0.00      0.00      0.00      0.00
      59      0.00     -1.25        0.00      0.00      0.00      0.00
      60      0.00     -1.25        0.00      0.00      0.00      0.00
      61      0.00     -1.25        0.00      0.00      0.00      0.00
      62      0.00     -1.25        0.00      0.00      0.00      0.00
      63      0.00     -1.25        0.00      0.00      0.00      0.00
      64      0.00     -1.25        0.00      0.00      0.00      0.00
  **WARNING: IF THIS UBC/IBC ANALYSIS HAS TENSION/COMPRESSION
  OR REPEAT LOAD OR RE-ANALYSIS OR SELECT OPTIMIZE, THEN EACH
  UBC/IBC CASE SHOULD BE FOLLOWED BY PERFORM ANALYSIS & CHANGE.
   ***********************************************************
   *  IBC 2012 SEISMIC LOAD ALONG X  :                       *
   *     CT =  0.032 Cu =  1.400 x =  0.8000                 *
   *  TIME PERIODS :                                         *
   *     Ta =  0.486 T =  0.252 Tuser = 0.000                *
   *  TIME PERIOD USED (T) =  0.252                          *
   *  Cs LIMITS : LOWER =  0.045 UPPER =  0.259              *
   *  LOAD FACTOR          =  0.750                          *
   *  DESIGN BASE SHEAR    =  0.750 X  0.115 X       285.92  *
   *                       =        24.56  KIP               *
   ***********************************************************
   ***********************************************************
   *  IBC 2012 SEISMIC LOAD ALONG Z  :                       *
   *     CT =  0.032 Cu =  1.400 x =  0.8000                 *
   *  TIME PERIODS :                                         *
```

```
*     Ta =  0.486 T =  0.989 Tuser = 0.000               *
*  TIME PERIOD USED (T) =  0.681                         *
*  Cs LIMITS : LOWER =  0.045 UPPER =  0.096             *
*  LOAD FACTOR         =  0.750                          *
*  DESIGN BASE SHEAR    =  0.750 X  0.096 X       285.92   *
*                      =        20.56  KIP               *
************************************************************
 JOINT           LATERAL          TORSIONAL           LOAD -    1
                 LOAD (KIP )      MOMENT (KIP -FEET) FACTOR -   0.750
 -----           -------          ---------
   17     FX        0.235    MY        0.000
   18     FX        0.288    MY        0.000
   19     FX        0.288    MY        0.000
   EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    7
   20     FX        0.235    MY        0.000
   21     FX        0.288    MY        0.000
   22     FX        0.341    MY        0.000
   23     FX        0.341    MY        0.000
   24     FX        0.288    MY        0.000
   25     FX        0.288    MY        0.000
   26     FX        0.341    MY        0.000
   27     FX        0.341    MY        0.000
   28     FX        0.288    MY        0.000
   29     FX        0.235    MY        0.000
   30     FX        0.288    MY        0.000
   31     FX        0.288    MY        0.000
   32     FX        0.235    MY        0.000
              -----------       -----------
       TOTAL =      4.609              0.000 AT LEVEL    10.000 FEET
   33     FX        0.470    MY        0.000
   34     FX        0.576    MY        0.000
   35     FX        0.576    MY        0.000
   36     FX        0.470    MY        0.000
   37     FX        0.576    MY        0.000
   38     FX        0.682    MY        0.000
   39     FX        0.682    MY        0.000
   40     FX        0.576    MY        0.000
   41     FX        0.576    MY        0.000
   42     FX        0.682    MY        0.000
   43     FX        0.682    MY        0.000
   44     FX        0.576    MY        0.000
   45     FX        0.470    MY        0.000
   46     FX        0.576    MY        0.000
   47     FX        0.576    MY        0.000
   48     FX        0.470    MY        0.000
              -----------       -----------
       TOTAL =      9.218              0.000 AT LEVEL    20.000 FEET
   49     FX        0.512    MY        0.000
   50     FX        0.671    MY        0.000
   51     FX        0.671    MY        0.000
   52     FX        0.512    MY        0.000
   53     FX        0.671    MY        0.000
   54     FX        0.830    MY        0.000
   55     FX        0.830    MY        0.000
   56     FX        0.671    MY        0.000
   57     FX        0.671    MY        0.000
   58     FX        0.830    MY        0.000
   59     FX        0.830    MY        0.000
```

```
   60     FX        0.671    MY        0.000
   61     FX        0.512    MY        0.000
   62     FX        0.671    MY        0.000
   63     FX        0.671    MY        0.000
   64     FX        0.512    MY        0.000
              -----------       -----------
       TOTAL =     10.736            0.000 AT LEVEL    30.000 FEET
   EXAMPLE PROBLEM FOR IBC LOAD                              -- PAGE NO.    8
 JOINT            LATERAL           TORSIONAL            LOAD -    2
                  LOAD (KIP )       MOMENT (KIP -FEET) FACTOR -   0.750
 -----            -------           ---------
   17     FZ        0.184    MY        0.000
   18     FZ        0.225    MY        0.000
   19     FZ        0.225    MY        0.000
   20     FZ        0.184    MY        0.000
   21     FZ        0.225    MY        0.000
   22     FZ        0.267    MY        0.000
   23     FZ        0.267    MY        0.000
   24     FZ        0.225    MY        0.000
   25     FZ        0.225    MY        0.000
   26     FZ        0.267    MY        0.000
   27     FZ        0.267    MY        0.000
   28     FZ        0.225    MY        0.000
   29     FZ        0.184    MY        0.000
   30     FZ        0.225    MY        0.000
   31     FZ        0.225    MY        0.000
   32     FZ        0.184    MY        0.000
              -----------       -----------
       TOTAL =      3.605            0.000 AT LEVEL    10.000 FEET
   33     FZ        0.391    MY        0.000
   34     FZ        0.480    MY        0.000
   35     FZ        0.480    MY        0.000
   36     FZ        0.391    MY        0.000
   37     FZ        0.480    MY        0.000
   38     FZ        0.568    MY        0.000
   39     FZ        0.568    MY        0.000
   40     FZ        0.480    MY        0.000
   41     FZ        0.480    MY        0.000
   42     FZ        0.568    MY        0.000
   43     FZ        0.568    MY        0.000
   44     FZ        0.480    MY        0.000
   45     FZ        0.391    MY        0.000
   46     FZ        0.480    MY        0.000
   47     FZ        0.480    MY        0.000
   48     FZ        0.391    MY        0.000
              -----------       -----------
       TOTAL =      7.677            0.000 AT LEVEL    20.000 FEET
   49     FZ        0.442    MY        0.000
   50     FZ        0.580    MY        0.000
   51     FZ        0.580    MY        0.000
   52     FZ        0.442    MY        0.000
   53     FZ        0.580    MY        0.000
   54     FZ        0.717    MY        0.000
   55     FZ        0.717    MY        0.000
   56     FZ        0.580    MY        0.000
   57     FZ        0.580    MY        0.000
   58     FZ        0.717    MY        0.000
   59     FZ        0.717    MY        0.000
```

```
     60     FZ        0.580    MY        0.000
     61     FZ        0.442    MY        0.000
     62     FZ        0.580    MY        0.000
     EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.    9
     63     FZ        0.580    MY        0.000
     64     FZ        0.442    MY        0.000
              -----------        -----------
         TOTAL =      9.274              0.000 AT LEVEL    30.000 FEET
  ++ Adjusting Displacements.
  ************ END OF DATA FROM INTERNAL STORAGE ************
   69. PRINT SUPPORT REACTIONS
 SUPPORT  REACTION
     EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.   10
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -1.23     10.47      0.01      0.05     -0.00      8.48
          2      0.11     10.25     -1.28     -6.71      0.00     -0.33
     2    1     -1.70     17.50      0.01      0.05     -0.00      9.87
          2      0.01     13.61     -1.27     -6.70      0.00     -0.02
     3    1     -1.71     17.01      0.01      0.05     -0.00      9.92
          2     -0.01     13.61     -1.27     -6.70     -0.00      0.02
     4    1     -1.45     17.30      0.01      0.05     -0.00      9.13
          2     -0.11     10.25     -1.28     -6.71     -0.00      0.33
     5    1     -1.26     15.02     -0.00     -0.01     -0.00      8.62
          2      0.11     20.00     -1.30     -6.87      0.00     -0.33
     6    1     -1.72     22.10     -0.00     -0.01     -0.00     10.03
          2      0.01     23.36     -1.28     -6.86      0.00     -0.02
     7    1     -1.74     21.61     -0.00     -0.01     -0.00     10.08
          2     -0.01     23.36     -1.28     -6.86     -0.00      0.02
     8    1     -1.47     21.95     -0.00     -0.01     -0.00      9.28
          2     -0.11     20.00     -1.30     -6.87     -0.00      0.33
     9    1     -1.26     15.02      0.00      0.01      0.00      8.62
          2      0.11     16.98     -1.31     -6.87      0.00     -0.33
    10    1     -1.72     22.10      0.00      0.01      0.00     10.03
          2      0.01     20.35     -1.29     -6.86      0.00     -0.02
    11    1     -1.74     21.61      0.00      0.01      0.00     10.08
          2     -0.01     20.35     -1.29     -6.86     -0.00      0.02
    12    1     -1.47     21.95      0.00      0.01      0.00      9.28
          2     -0.11     16.98     -1.31     -6.87     -0.00      0.33
    13    1     -1.23     10.47     -0.01     -0.05      0.00      8.48
          2      0.11     17.52     -1.28     -6.77      0.00     -0.33
    14    1     -1.70     17.50     -0.01     -0.05      0.00      9.87
          2      0.01     20.89     -1.27     -6.77      0.00     -0.02
    15    1     -1.71     17.01     -0.01     -0.05      0.00      9.92
          2     -0.01     20.89     -1.27     -6.77     -0.00      0.02
    16    1     -1.45     17.30     -0.01     -0.05      0.00      9.13
          2     -0.11     17.52     -1.28     -6.77     -0.00      0.33
  ************** END OF LATEST ANALYSIS RESULT **************
   70. FINISH
     EXAMPLE PROBLEM FOR IBC LOAD                             -- PAGE NO.   11
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:40 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
```

```
          *   Bentley and Partners can be found at program menu      *
          *   Help->Technical Support                                *
          *                                                          *
          *           Copyright (c) Bentley Systems, Inc.            *
          *                http://www.bentley.com                    *
          ************************************************************
```

EX. US-15 Wind and Floor Load Generation on a Space Frame

A space frame is analyzed for loads generated using the built-in wind and floor load generation facilities.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-15 Wind and Floor Load Generation on a Space Frame.STD` when you install the program.

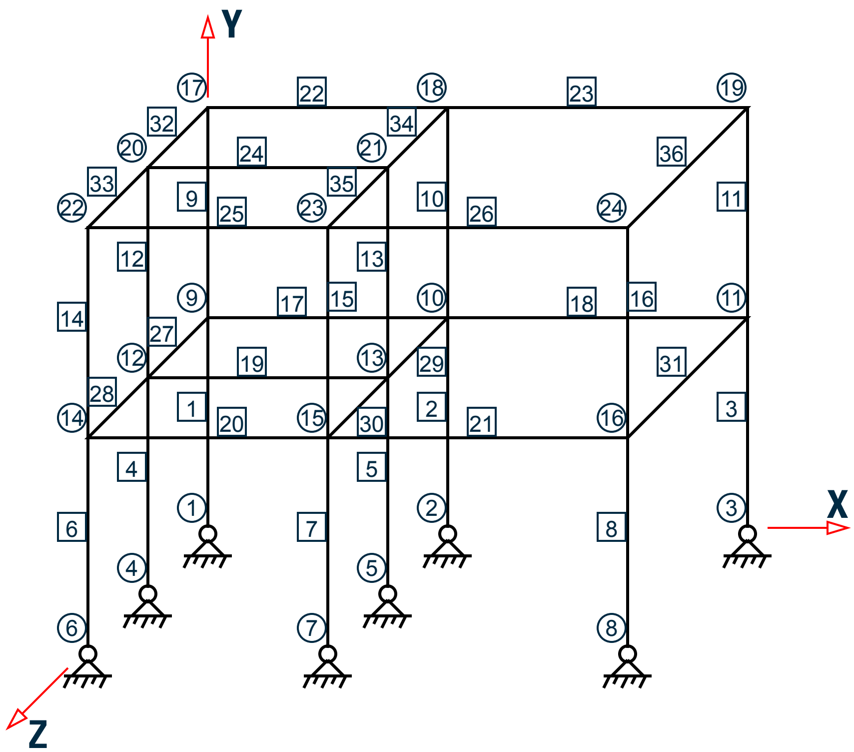


Figure 527: Example Problem No. 15

```
    STAAD SPACE - WIND AND FLOOR LOAD GENERATION
```

This is a space frame analysis problem. Every STAAD input has to start with the command `STAAD`. The `SPACE` specification is used to denote a space (3D) frame.

```
    UNIT FEET KIP
```

The `UNIT` specification is used to specify the length and/or force units to be used.

```
    JOINT COORDINATES
    1 0 0 0
    2 10 0 0
    3 21 0 0
```

```
4 0 0 10
5 10 0 10
6 0 0 20
7 10 0 20
8 21 0 20
REPEAT ALL 2 0 12 0
```

The `JOINT COORDINATE` specification is used to specify the X, Y, and Z coordinates of the joints. Note that the `REPEAT ALL` command has been used to generate joints for the two upper stories each with a Y increment of 12 ft.

```
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
```

The `MEMBER INCIDENCE` specification is used for specifying member connectivities.

```
UNIT INCH
MEMBER PROPERTIES AMERICAN
1 TO 16 TA ST W21X50
17 TO 26 TA ST W18X35
27 TO 36 TA ST W14X90
```

Properties for all members are specified from the built-in American (AISC) steel table. Three different sections have been used.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 8 FIXED BUT MX MZ
```

The supports of the structure are defined through the `SUPPORT` specification. Here all the supports are `FIXED` with releases specified in the `MX` (rotation about global X-axis) and `MZ` (rotation about global Z-axis) directions.

```
UNIT FEET
DEFINE WIND LOAD
TYPE 1
```

```
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOINT 17 20 22
```

When a structure has to be analyzed for wind loading, the engineer is confronted with the task of first converting an abstract quantity like wind velocity or wind pressure into concentrated loads at joints, distributed loads on members, or pressure loads on plates. The large number of calculations involved in this conversion can be avoided by making use of STAAD's wind load generation utility. This utility takes wind pressure at various heights as the input, and converts them to values that can then be used as concentrated forces known as joint loads in specific load cases. The input specification is done in two stages. The first stage is initiated above through the `DEFINE WIND LOAD` command. The basic parameters of the WIND loading are specified here. All values need to be provided in the current UNIT system. Each wind category is identified with a `TYPE` number (an identification mark) which is used later to specify load cases.

In this example, two different wind intensities (0.1 Kips/sq. ft and 0.15 Kips/sq. ft) are specified for two different height zones (0 to 12 ft. and 12 to 24 ft.). The EXPOSURE specification is used to mitigate or magnify the effect at specific nodes due to special considerations like openings in the structure. In this case, two different exposure factors are specified. The first EXPOSURE specification specifies the exposure factor as 0.9 for all joints within the height range (defined as global Y-range) of 11 ft. - 13 ft. The second EXPOSURE specification specifies the exposure factor as 0.85 for joints 17, 20 and 22. In the EXPOSURE factor specification, the joints may be specified directly or through a vertical range specification.

```
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
```

This is the second stage of input specification for the wind load generation. The term `WIND LOAD` and the direction term that follows are used to specify the wind loading in a particular lateral direction. In this case, WIND loading `TYPE 1`, defined previously, is being applied in the global X-direction with a positive multiplication factor of 1.2 .

```
LOAD 2 FLOOR LOAD @ Y = 12 FT AND 24 FT
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.45 XRANGE 0.0 10.0 ZRANGE 0.0 20.0
YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0
YRANGE 23.9 24.1 FLOAD -0.25
```

In load case 2 in this problem, a floor load generation is performed. In a floor load generation, a pressure load (force per unit area) is converted by the program into specific points forces and distributed forces on the members located in that region. The `YRANGE`, `XRANGE`, and `ZRANGE` specifications are used to define the area of the structure on which the pressure is acting. The `FLOAD` specification is used to specify the value of that pressure. All values need to be provided in the current UNIT system. For example, in the first line in the above FLOOR LOAD specification, the region is defined as being located within the bounds YRANGE of 11.9-12.1 ft, XRANGE of 0.0-10.0 ft and ZRANGE of 0.0-20.0 ft. The -0.45 signifies that the pressure is 0.45 Kip/sq. ft in the negative global Y direction.

The program will identify the members lying within the specified region and derive member loads on these members based on two-way load distribution.

```
PERFORM ANALYSIS PRINT LOAD DATA
```

We can view the values and position of the generated loads with the help of the `PRINT LOAD DATA` command used above along with the `PERFORM ANALYSIS` command.

```
PRINT SUPPORT REACTION
FINISH
```

Above commands are self-explanatory.

Input File

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0
2 10 0 0
3 21 0 0
4 0 0 10
5 10 0 10
6 0 0 20
7 10 0 20
8 21 0 20
REPEAT ALL 2 0 12 0
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
UNIT INCH
MEMBER PROPERTIES AMERICAN
1 TO 16 TA ST W21X50
17 TO 26 TA ST W18X35
27 TO 36 TA ST W14X90
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 TO 8 FIXED BUT MX MZ
UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOINT 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
LOAD 2 FLOOR LOAD @ Y = 12ft AND 24ft
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD -0.45 XRANGE  0.0 10.0 ZRANGE 0.0 20.0
```

```
YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0
YRANGE 23.9 24.1 FLOAD -0.25
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:42                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: US-15 Wind and Floor Load Generation on a Space Frame.STD
     2. UNIT FEET KIP
     3. JOINT COORDINATES
     4. 1 0 0 0
     5. 2 10 0 0
     6. 3 21 0 0
     7. 4 0 0 10
     8. 5 10 0 10
     9. 6 0 0 20
    10. 7 10 0 20
    11. 8 21 0 20
    12. REPEAT ALL 2 0 12 0
    13. MEMBER INCIDENCES
    14. * COLUMNS
    15. 1 1 9 16
    16. * BEAMS IN THE X DIRECTION
    17. 17 9 10 18
    18. 19 12 13
    19. 20 14 15 21
    20. 22 17 18 23
    21. 24 20 21
    22. 25 22 23 26
    23. * BEAMS IN THE Z DIRECTION
    24. 27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
    25. 32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
    26. UNIT INCH
    27. MEMBER PROPERTIES AMERICAN
    28. 1 TO 16 TA ST W21X50
    29. 17 TO 26 TA ST W18X35
    30. 27 TO 36 TA ST W14X90
    31. DEFINE MATERIAL START
    32. ISOTROPIC STEEL
    33. E 29000
    34. POISSON 0.3
    35. DENSITY 283E-006
    36. ALPHA 6E-006
    37. DAMP 0.03
```

```
    38. TYPE STEEL
      STAAD SPACE                                                -- PAGE NO.    2
    39. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    40. END DEFINE MATERIAL
    41. CONSTANT
    42. MATERIAL STEEL ALL
    43. SUPPORT
    44. 1 TO 8 FIXED BUT MX MZ
    45. UNIT FEET
    46. DEFINE WIND LOAD
   *** NOTE: If any floor diaphragm is present in the model Wind Load definition
             should be defined after Floor Diaphragm definition. Otherwise wind
             load generation may be unsuccessful during analysis.
    47. TYPE 1
    48. INTENSITY 0.1 0.15 HEIGHT 12 24
    49. EXPOSURE 0.90 YRANGE 11 13
    50. EXPOSURE 0.85 JOINT 17 20 22
    51. LOAD 1 WIND LOAD IN X-DIRECTION
    52. WIND LOAD X 1.2 TYPE 1
    53. LOAD 2 FLOOR LOAD @ Y = 12FT AND 24FT
    54. FLOOR LOAD
    55. YRANGE 11.9 12.1 FLOAD -0.45 XRANGE  0.0 10.0 ZRANGE 0.0 20.0
 **NOTE** about Floor/OneWay Loads/Weights.
   Please note that depending on the shape of the floor you may
   have to break up the FLOOR/ONEWAY LOAD into multiple commands.
   For details please refer to Technical Reference Manual
   Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
    56. YRANGE 11.9 12.1 FLOAD -0.25 XRANGE 10.0 21.0 ZRANGE 0.0 20.0
    57. YRANGE 23.9 24.1 FLOAD -0.25
    58. PERFORM ANALYSIS PRINT LOAD DATA
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         24  NUMBER OF MEMBERS      36
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      8
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     112
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
      STAAD SPACE                                                -- PAGE NO.    3
      STAAD SPACE                                                -- PAGE NO.    4
   LOADING     1  WIND LOAD IN X-DIRECTION
   -----------
   JOINT LOAD - UNIT KIP  FEET
   JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
       1      3.60      0.00        0.00      0.00      0.00      0.00
       4      7.20      0.00        0.00      0.00      0.00      0.00
       6      3.60      0.00        0.00      0.00      0.00      0.00
       9      8.10      0.00        0.00      0.00      0.00      0.00
      12     16.20      0.00        0.00      0.00      0.00      0.00
      14      8.10      0.00        0.00      0.00      0.00      0.00
      17      4.59      0.00        0.00      0.00      0.00      0.00
      20      9.18      0.00        0.00      0.00      0.00      0.00
      22      4.59      0.00        0.00      0.00      0.00      0.00
   LOADING     2  FLOOR LOAD @ Y = 12FT AND 24FT
   -----------
   MEMBER LOAD - UNIT KIP  FEET
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
```

```
     17                            -0.0879 GY   0.42
     17                            -0.2637 GY   0.97
     17                            -0.4395 GY   1.58
     17                            -0.6152 GY   2.20
     17                            -0.7910 GY   2.82
     17                            -0.9668 GY   3.45
     17                            -1.1426 GY   4.07
     17                            -1.3184 GY   4.69
     17                            -1.3184 GY   5.31
     17                            -1.1426 GY   5.93
     17                            -0.9668 GY   6.55
     17                            -0.7910 GY   7.18
     17                            -0.6152 GY   7.80
     17                            -0.4395 GY   8.42
     17                            -0.2637 GY   9.03
     17                            -0.0879 GY   9.58
     29                            -0.0879 GY   0.42
     29                            -0.2637 GY   0.97
     29                            -0.4395 GY   1.58
     29                            -0.6152 GY   2.20
     29                            -0.7910 GY   2.82
     29                            -0.9668 GY   3.45
     29                            -1.1426 GY   4.07
     29                            -1.3184 GY   4.69
     29                            -1.3184 GY   5.31
     29                            -1.1426 GY   5.93
     29                            -0.9668 GY   6.55
      STAAD SPACE                                                  -- PAGE NO.    5
     29                            -0.7910 GY   7.18
     29                            -0.6152 GY   7.80
     29                            -0.4395 GY   8.42
     29                            -0.2637 GY   9.03
     29                            -0.0879 GY   9.58
     19                            -0.0879 GY   0.42
     19                            -0.2637 GY   0.97
     19                            -0.4395 GY   1.58
     19                            -0.6152 GY   2.20
     19                            -0.7910 GY   2.82
     19                            -0.9668 GY   3.45
     19                            -1.1426 GY   4.07
     19                            -1.3184 GY   4.69
     19                            -1.3184 GY   5.31
     19                            -1.1426 GY   5.93
     19                            -0.9668 GY   6.55
     19                            -0.7910 GY   7.18
     19                            -0.6152 GY   7.80
     19                            -0.4395 GY   8.42
     19                            -0.2637 GY   9.03
     19                            -0.0879 GY   9.58
     27                            -0.0879 GY   0.42
     27                            -0.2637 GY   0.97
     27                            -0.4395 GY   1.58
     27                            -0.6152 GY   2.20
     27                            -0.7910 GY   2.82
     27                            -0.9668 GY   3.45
     27                            -1.1426 GY   4.07
     27                            -1.3184 GY   4.69
     27                            -1.3184 GY   5.31
```

```
     27                            -1.1426 GY   5.93
     27                            -0.9668 GY   6.55
     27                            -0.7910 GY   7.18
     27                            -0.6152 GY   7.80
     27                            -0.4395 GY   8.42
     27                            -0.2637 GY   9.03
     27                            -0.0879 GY   9.58
     19                            -0.0879 GY   0.42
     19                            -0.2637 GY   0.97
     19                            -0.4395 GY   1.58
     19                            -0.6152 GY   2.20
     19                            -0.7910 GY   2.82
     19                            -0.9668 GY   3.45
     19                            -1.1426 GY   4.07
     19                            -1.3184 GY   4.69
     19                            -1.3184 GY   5.31
     19                            -1.1426 GY   5.93
     19                            -0.9668 GY   6.55
     19                            -0.7910 GY   7.18
     19                            -0.6152 GY   7.80
     19                            -0.4395 GY   8.42
     19                            -0.2637 GY   9.03
     19                            -0.0879 GY   9.58
     30                            -0.0879 GY   0.42
     30                            -0.2637 GY   0.97
     30                            -0.4395 GY   1.58
      STAAD SPACE                                                 -- PAGE NO.    6
     30                            -0.6152 GY   2.20
     30                            -0.7910 GY   2.82
     30                            -0.9668 GY   3.45
     30                            -1.1426 GY   4.07
     30                            -1.3184 GY   4.69
     30                            -1.3184 GY   5.31
     30                            -1.1426 GY   5.93
     30                            -0.9668 GY   6.55
     30                            -0.7910 GY   7.18
     30                            -0.6152 GY   7.80
     30                            -0.4395 GY   8.42
     30                            -0.2637 GY   9.03
     30                            -0.0879 GY   9.58
     20                            -0.0879 GY   0.42
     20                            -0.2637 GY   0.97
     20                            -0.4395 GY   1.58
     20                            -0.6152 GY   2.20
     20                            -0.7910 GY   2.82
     20                            -0.9668 GY   3.45
     20                            -1.1426 GY   4.07
     20                            -1.3184 GY   4.69
     20                            -1.3184 GY   5.31
     20                            -1.1426 GY   5.93
     20                            -0.9668 GY   6.55
     20                            -0.7910 GY   7.18
     20                            -0.6152 GY   7.80
     20                            -0.4395 GY   8.42
     20                            -0.2637 GY   9.03
     20                            -0.0879 GY   9.58
     28                            -0.0879 GY   0.42
     28                            -0.2637 GY   0.97
```

```
     28                            -0.4395 GY   1.58
     28                            -0.6152 GY   2.20
     28                            -0.7910 GY   2.82
     28                            -0.9668 GY   3.45
     28                            -1.1426 GY   4.07
     28                            -1.3184 GY   4.69
     28                            -1.3184 GY   5.31
     28                            -1.1426 GY   5.93
     28                            -0.9668 GY   6.55
     28                            -0.7910 GY   7.18
     28                            -0.6152 GY   7.80
     28                            -0.4395 GY   8.42
     28                            -0.2637 GY   9.03
     28                            -0.0879 GY   9.58
   MEMBER LOAD - UNIT KIP  FEET
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     18                            -0.0591 GY   0.46
     18                            -0.1772 GY   1.07
     18                            -0.2954 GY   1.74
     18                            -0.4136 GY   2.42
     18                            -0.5317 GY   3.11
      STAAD SPACE                                           -- PAGE NO.    7
     18                            -0.6499 GY   3.79
     18                            -0.7681 GY   4.48
     18                            -0.8862 GY   5.16
     18                            -0.8862 GY   5.84
     18                            -0.7681 GY   6.52
     18                            -0.6499 GY   7.21
     18                            -0.5317 GY   7.89
     18                            -0.4136 GY   8.58
     18                            -0.2954 GY   9.26
     18                            -0.1772 GY   9.93
     18                            -0.0591 GY  10.54
     31                            -0.0591 GY   0.46
     31                            -0.1772 GY   1.07
     31                            -0.2954 GY   1.74
     31                            -0.4136 GY   2.42
     31                            -0.5317 GY   3.11
     31                            -0.6499 GY   3.79
     31                            -0.7681 GY   4.48
     31                            -0.8862 GY   5.16
     31    -1.3750 GY   5.50   14.50
     31                            -0.8862 GY  14.84
     31                            -0.7681 GY  15.52
     31                            -0.6499 GY  16.21
     31                            -0.5317 GY  16.89
     31                            -0.4136 GY  17.58
     31                            -0.2954 GY  18.26
     31                            -0.1772 GY  18.93
     31                            -0.0591 GY  19.54
     21                            -0.0591 GY   0.46
     21                            -0.1772 GY   1.07
     21                            -0.2954 GY   1.74
     21                            -0.4136 GY   2.42
     21                            -0.5317 GY   3.11
     21                            -0.6499 GY   3.79
     21                            -0.7681 GY   4.48
     21                            -0.8862 GY   5.16
```

```
    21                            -0.8862 GY   5.84
    21                            -0.7681 GY   6.52
    21                            -0.6499 GY   7.21
    21                            -0.5317 GY   7.89
    21                            -0.4136 GY   8.58
    21                            -0.2954 GY   9.26
    21                            -0.1772 GY   9.93
    21                            -0.0591 GY  10.54
    30                            -0.8862 GY   4.84
    30                            -0.7681 GY   5.52
    30                            -0.6499 GY   6.21
    30                            -0.5317 GY   6.89
    30                            -0.4136 GY   7.58
    30                            -0.2954 GY   8.26
    30                            -0.1772 GY   8.93
    30                            -0.0591 GY   9.54
    30    -1.3750 GY   0.00    4.50
    29                            -0.0591 GY   0.46
    29                            -0.1772 GY   1.07
    29                            -0.2954 GY   1.74
     STAAD SPACE                                                -- PAGE NO.    8
    29                            -0.4136 GY   2.42
    29                            -0.5317 GY   3.11
    29                            -0.6499 GY   3.79
    29                            -0.7681 GY   4.48
    29                            -0.8862 GY   5.16
    29    -1.3750 GY   5.50   10.00
   MEMBER LOAD - UNIT KIP  FEET
   MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    22                            -0.0488 GY   0.42
    22                            -0.1465 GY   0.97
    22                            -0.2441 GY   1.58
    22                            -0.3418 GY   2.20
    22                            -0.4395 GY   2.82
    22                            -0.5371 GY   3.45
    22                            -0.6348 GY   4.07
    22                            -0.7324 GY   4.69
    22                            -0.7324 GY   5.31
    22                            -0.6348 GY   5.93
    22                            -0.5371 GY   6.55
    22                            -0.4395 GY   7.18
    22                            -0.3418 GY   7.80
    22                            -0.2441 GY   8.42
    22                            -0.1465 GY   9.03
    22                            -0.0488 GY   9.58
    34                            -0.0488 GY   0.42
    34                            -0.1465 GY   0.97
    34                            -0.2441 GY   1.58
    34                            -0.3418 GY   2.20
    34                            -0.4395 GY   2.82
    34                            -0.5371 GY   3.45
    34                            -0.6348 GY   4.07
    34                            -0.7324 GY   4.69
    34                            -0.7324 GY   5.31
    34                            -0.6348 GY   5.93
    34                            -0.5371 GY   6.55
    34                            -0.4395 GY   7.18
    34                            -0.3418 GY   7.80
```

```
      34                            -0.2441 GY   8.42
      34                            -0.1465 GY   9.03
      34                            -0.0488 GY   9.58
      24                            -0.0488 GY   0.42
      24                            -0.1465 GY   0.97
      24                            -0.2441 GY   1.58
      24                            -0.3418 GY   2.20
      24                            -0.4395 GY   2.82
      24                            -0.5371 GY   3.45
      24                            -0.6348 GY   4.07
      24                            -0.7324 GY   4.69
      24                            -0.7324 GY   5.31
      24                            -0.6348 GY   5.93
      24                            -0.5371 GY   6.55
      24                            -0.4395 GY   7.18
       STAAD SPACE                                                 -- PAGE NO.    9
      24                            -0.3418 GY   7.80
      24                            -0.2441 GY   8.42
      24                            -0.1465 GY   9.03
      24                            -0.0488 GY   9.58
      32                            -0.0488 GY   0.42
      32                            -0.1465 GY   0.97
      32                            -0.2441 GY   1.58
      32                            -0.3418 GY   2.20
      32                            -0.4395 GY   2.82
      32                            -0.5371 GY   3.45
      32                            -0.6348 GY   4.07
      32                            -0.7324 GY   4.69
      32                            -0.7324 GY   5.31
      32                            -0.6348 GY   5.93
      32                            -0.5371 GY   6.55
      32                            -0.4395 GY   7.18
      32                            -0.3418 GY   7.80
      32                            -0.2441 GY   8.42
      32                            -0.1465 GY   9.03
      32                            -0.0488 GY   9.58
      23                            -0.0591 GY   0.46
      23                            -0.1772 GY   1.07
      23                            -0.2954 GY   1.74
      23                            -0.4136 GY   2.42
      23                            -0.5317 GY   3.11
      23                            -0.6499 GY   3.79
      23                            -0.7681 GY   4.48
      23                            -0.8862 GY   5.16
      23                            -0.8862 GY   5.84
      23                            -0.7681 GY   6.52
      23                            -0.6499 GY   7.21
      23                            -0.5317 GY   7.89
      23                            -0.4136 GY   8.58
      23                            -0.2954 GY   9.26
      23                            -0.1772 GY   9.93
      23                            -0.0591 GY  10.54
      36                            -0.0591 GY   0.46
      36                            -0.1772 GY   1.07
      36                            -0.2954 GY   1.74
      36                            -0.4136 GY   2.42
      36                            -0.5317 GY   3.11
      36                            -0.6499 GY   3.79
```

```
     36                            -0.7681 GY   4.48
     36                            -0.8862 GY   5.16
     36    -1.3750 GY   5.50   14.50
     36                            -0.8862 GY  14.84
     36                            -0.7681 GY  15.52
     36                            -0.6499 GY  16.21
     36                            -0.5317 GY  16.89
     36                            -0.4136 GY  17.58
     36                            -0.2954 GY  18.26
     36                            -0.1772 GY  18.93
     36                            -0.0591 GY  19.54
     26                            -0.0591 GY   0.46
     26                            -0.1772 GY   1.07
     26                            -0.2954 GY   1.74
      STAAD SPACE                                              -- PAGE NO.   10
     26                            -0.4136 GY   2.42
     26                            -0.5317 GY   3.11
     26                            -0.6499 GY   3.79
     26                            -0.7681 GY   4.48
     26                            -0.8862 GY   5.16
     26                            -0.8862 GY   5.84
     26                            -0.7681 GY   6.52
     26                            -0.6499 GY   7.21
     26                            -0.5317 GY   7.89
     26                            -0.4136 GY   8.58
     26                            -0.2954 GY   9.26
     26                            -0.1772 GY   9.93
     26                            -0.0591 GY  10.54
     35                            -0.8862 GY   4.84
     35                            -0.7681 GY   5.52
     35                            -0.6499 GY   6.21
     35                            -0.5317 GY   6.89
     35                            -0.4136 GY   7.58
     35                            -0.2954 GY   8.26
     35                            -0.1772 GY   8.93
     35                            -0.0591 GY   9.54
     35    -1.3750 GY   0.00    4.50
     34                            -0.0591 GY   0.46
     34                            -0.1772 GY   1.07
     34                            -0.2954 GY   1.74
     34                            -0.4136 GY   2.42
     34                            -0.5317 GY   3.11
     34                            -0.6499 GY   3.79
     34                            -0.7681 GY   4.48
     34                            -0.8862 GY   5.16
     34    -1.3750 GY   5.50   10.00
     24                            -0.0488 GY   0.42
     24                            -0.1465 GY   0.97
     24                            -0.2441 GY   1.58
     24                            -0.3418 GY   2.20
     24                            -0.4395 GY   2.82
     24                            -0.5371 GY   3.45
     24                            -0.6348 GY   4.07
     24                            -0.7324 GY   4.69
     24                            -0.7324 GY   5.31
     24                            -0.6348 GY   5.93
     24                            -0.5371 GY   6.55
     24                            -0.4395 GY   7.18
```

```
    24                            -0.3418 GY   7.80
    24                            -0.2441 GY   8.42
    24                            -0.1465 GY   9.03
    24                            -0.0488 GY   9.58
    35                            -0.0488 GY   0.42
    35                            -0.1465 GY   0.97
    35                            -0.2441 GY   1.58
    35                            -0.3418 GY   2.20
    35                            -0.4395 GY   2.82
    35                            -0.5371 GY   3.45
    35                            -0.6348 GY   4.07
    35                            -0.7324 GY   4.69
    35                            -0.7324 GY   5.31
     STAAD SPACE                                              -- PAGE NO.   11
    35                            -0.6348 GY   5.93
    35                            -0.5371 GY   6.55
    35                            -0.4395 GY   7.18
    35                            -0.3418 GY   7.80
    35                            -0.2441 GY   8.42
    35                            -0.1465 GY   9.03
    35                            -0.0488 GY   9.58
    25                            -0.0488 GY   0.42
    25                            -0.1465 GY   0.97
    25                            -0.2441 GY   1.58
    25                            -0.3418 GY   2.20
    25                            -0.4395 GY   2.82
    25                            -0.5371 GY   3.45
    25                            -0.6348 GY   4.07
    25                            -0.7324 GY   4.69
    25                            -0.7324 GY   5.31
    25                            -0.6348 GY   5.93
    25                            -0.5371 GY   6.55
    25                            -0.4395 GY   7.18
    25                            -0.3418 GY   7.80
    25                            -0.2441 GY   8.42
    25                            -0.1465 GY   9.03
    25                            -0.0488 GY   9.58
    33                            -0.0488 GY   0.42
    33                            -0.1465 GY   0.97
    33                            -0.2441 GY   1.58
    33                            -0.3418 GY   2.20
    33                            -0.4395 GY   2.82
    33                            -0.5371 GY   3.45
    33                            -0.6348 GY   4.07
    33                            -0.7324 GY   4.69
    33                            -0.7324 GY   5.31
    33                            -0.6348 GY   5.93
    33                            -0.5371 GY   6.55
    33                            -0.4395 GY   7.18
    33                            -0.3418 GY   7.80
    33                            -0.2441 GY   8.42
    33                            -0.1465 GY   9.03
    33                            -0.0488 GY   9.58
 ************ END OF DATA FROM INTERNAL STORAGE ************
  59. PRINT SUPPORT REACTION
SUPPORT  REACTION
     STAAD SPACE                                              -- PAGE NO.   12
 SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
```

```
 -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -9.35    -16.17     -0.00      0.00     -0.01      0.00
          2      0.30     15.66      0.03      0.00      0.00      0.00
     2    1     -7.26      2.45      0.01      0.00     -0.00      0.00
          2     -0.15     29.65      0.04      0.00      0.00      0.00
     3    1     -5.57     14.22      0.00      0.00      0.00      0.00
          2     -0.25     27.41      0.17      0.00     -0.00      0.00
     4    1    -14.03    -19.32      0.00      0.00      0.00      0.00
          2      0.74     38.48     -0.00      0.00      0.00      0.00
     5    1     -6.76     18.33      0.00      0.00      0.00      0.00
          2     -0.57     66.05      0.00      0.00      0.00      0.00
     6    1     -9.35    -16.17      0.00      0.00      0.01      0.00
          2      0.30     15.66     -0.03      0.00     -0.00      0.00
     7    1     -7.26      2.45     -0.01      0.00      0.00      0.00
          2     -0.15     29.65     -0.04      0.00     -0.00      0.00
     8    1     -5.57     14.22     -0.00      0.00     -0.00      0.00
          2     -0.25     27.41     -0.17      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   60. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:43 ****
     STAAD SPACE                                              -- PAGE NO.   13
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-16 Time History Analysis for Forcing Function and Ground Motion

Dynamic Analysis (Time History) is performed for a 3 span beam with concentrated and distributed masses. The structure is subjected to “forcing function” and “ground motion” loading. The maxima of joint displacements, member end forces and support reactions are determined.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-16 Time History Analysis for Forcing Function and Ground Motion.STD` when you install the program.

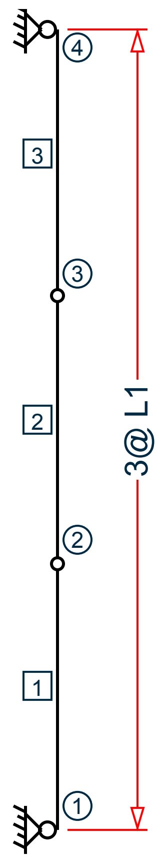


Figure 528: Example Problem No. 16

Where:

L1 = 3.5 ft

```
STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
```

Every input file has to start with the word `STAAD`. The term `PLANE` signifies that the structure is a plane frame.

```
UNITS FEET KIP
```

Specifies the units to be used.

```
JOINT COORDINATES
1  0.0   0.0   0.0
2  0.0   3.5   0.0
3  0.0   7.0   0.0
4  0.0  10.5   0.0
```

Joint number followed by the X, Y and Z coordinates are specified above.

```
MEMBER INCIDENCES
1 1 2 3
```

Incidences of members 1 to 3 are specified above.

```
UNIT INCH
MEMBER PROPERTIES
1 2 3 PRIS AX 3.0 IZ 240.0
```

All the members have PRISMATIC property specification. Since this is a plane frame, Area of cross section (AX) and Moment of Inertia (IZ) about the Z axis are adequate for the analysis.

```
SUPPORTS
1 4 PINNED
```

Pinned supports are located at nodes 1 and 4.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 14000
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
TYPE 2 ACCELERATION
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.075
```

There are two stages in the command specification required for a time history analysis. The first stage is defined above. First the characteristics of the time varying load are provided. The loading type may be a forcing function (vibrating machinery) or ground motion (earthquake). The former is input in the form of time-force pairs while the latter is in the form of time-acceleration pairs. Following this data, all possible arrival times for these loads on the structure as well as the modal damping ratio are specified. In this example, the damping ratio is the same (7.5%) for all modes.

```
UNIT FEET
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 0.5
```

Load case 1 above is a static load. A uniformly distributed force of 0.5 kip/ft acts along the global X direction on all 3 members.

```
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 2.5
TIME LOAD
```

```
    2 3 FX 1 1
    GROUND MOTION X 2 1
```

This is the second stage in the command specification for time history analysis. This involves the application of the time varying load on the structure. The masses that constitute the mass matrix of the structure are specified through the selfweight and joint load commands. The program will extract the lumped masses from these weights. Following that, both the `TIME LOAD` and `GROUND MOTION` are applied simultaneously.

Note: This example is only for illustration purposes and that it may be unlikely that a `TIME LOAD` and `GROUND MOTION` both act on the structure at the same time.

The Time load command is used to apply the Type 1 force, acting in the global X direction, at arrival time number 1, at nodes 2 and 3. The Ground motion, namely, the Type 2 time history loading, is also in the global X direction at arrival time 1.

```
    PERFORM ANALYSIS
```

The above command initiates the analysis process.

```
    UNIT INCH
    PRINT JOINT DISPLACEMENTS
```

During the analysis, the program calculates joint displacements for every time step. The absolute maximum value of the displacement for every joint is then extracted from this joint displacement history. So, the value printed using the above command is the absolute maximum value for each of the six degrees of freedom at each node.

```
    UNIT FEET
    PRINT MEMBER FORCES
    PRINT SUPPORT REACTION
```

The member forces and support reactions too are calculated for every time step. For each degree of freedom, the maximum value of the member force and support reaction is extracted from these histories and reported in the output file using the above command.

```
    FINISH
```

Input File

```
STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
UNITS FEET KIP
JOINT COORDINATES
1  0.0   0.0  0.0
2  0.0   3.5  0.0
3  0.0   7.0  0.0
4  0.0  10.5  0.0
MEMBER INCIDENCES
1 1 2 3
UNIT INCH
MEMBER PROPERTIES
1 2 3 PRIS AX 3.0 IZ 240.0
SUPPORTS
1 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 14000
POISSON 0.17
DENSITY 8.7e-005
```

```
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
TYPE 2 ACCELERATION
0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
ARRIVAL TIMES
0.0
DAMPING 0.075
UNIT FEET
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 0.5
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 2.5
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1
PERFORM ANALYSIS
UNIT INCH
PRINT JOINT DISPLACEMENTS
UNIT FEET
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:45                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
INPUT FILE: US-16 Time History Analysis for Forcing Function and Ground Motion.STD
     2. UNITS FEET KIP
     3. JOINT COORDINATES
     4. 1  0.0   0.0  0.0
     5. 2  0.0   3.5  0.0
     6. 3  0.0   7.0  0.0
     7. 4  0.0  10.5  0.0
```

```
     8. MEMBER INCIDENCES
     9. 1 1 2 3
    10. UNIT INCH
    11. MEMBER PROPERTIES
    12. 1 2 3 PRIS AX 3.0 IZ 240.0
    13. SUPPORTS
    14. 1 4 PINNED
    15. DEFINE MATERIAL START
    16. ISOTROPIC CONCRETE
    17. E 14000
    18. POISSON 0.17
    19. DENSITY 8.7E-005
    20. ALPHA 5E-006
    21. DAMP 0.05
    22. G 1346.15
    23. TYPE CONCRETE
    24. STRENGTH FCU 4
    25. END DEFINE MATERIAL
    26. CONSTANTS
    27. MATERIAL CONCRETE ALL
    28. DEFINE TIME HISTORY
    29. TYPE 1 FORCE
    30. 0.0 -0.0001 0.5 0.0449 1.0 0.2244 1.5 0.2244 2.0 0.6731 2.5 -0.6731
    31. TYPE 2 ACCELERATION
    32. 0.0 0.001 0.5 -7.721 1.0 -38.61 1.5 -38.61 2.0 -115.82 2.5 115.82
    33. ARRIVAL TIMES
    34. 0.0
    35. DAMPING 0.075
    36. UNIT FEET
    37. LOAD 1 STATIC LOAD
    38. MEMBER LOAD
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    2
    39. 1 2 3 UNI GX 0.5
    40. LOAD 2 TIME HISTORY LOAD
    41. SELFWEIGHT X 1.0
    42. SELFWEIGHT Y 1.0
    43. JOINT LOAD
    44. 2 3 FX 2.5
    45. TIME LOAD
    46. 2 3 FX 1 1
    47. GROUND MOTION X 2 1
    48. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       8
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
            THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
 MORE MODES WERE REQUESTED THAN THERE ARE FREE MASSES.
 NUMBER OF MODES REQUESTED              =     6
 NUMBER OF EXISTING MASSES IN THE MODEL =     4
 NUMBER OF MODES THAT WILL BE USED      =     4
   ***  EIGENSOLUTION : ADVANCED METHOD ***
```

```
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    3
               CALCULATED FREQUENCIES FOR LOAD CASE       2
       MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                      14.559                 0.06869
         2                      56.387                 0.01773
         3                     944.536                 0.00106
         4                    1635.985                 0.00061
            MODAL WEIGHT (MODAL MASS TIMES g) IN KIP          GENERALIZED
      MODE           X             Y             Z              WEIGHT
         1      5.021924E+00  0.000000E+00  0.000000E+00    5.021924E+00
         2      2.258180E-27  0.000000E+00  0.000000E+00    5.021924E+00
         3      0.000000E+00  2.192400E-02  0.000000E+00    2.192400E-02
         4      0.000000E+00  3.824466E-33  0.000000E+00    2.192400E-02
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1   100.00   0.00   0.00  100.000    0.000    0.000
             2     0.00   0.00   0.00  100.000    0.000    0.000
             3     0.00 100.00   0.00  100.000  100.000    0.000
             4     0.00   0.00   0.00  100.000  100.000    0.000
    A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
    MODE       DAMPING
       1      0.07500000
       2      0.07500000
       3      0.07500000
       4      0.07500000
   TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
   NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =    2
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    4
   WARNING-NUMBER OF MODES LIMITED TO A FREQUENCY OF   360.0 DUE TO THE DT VALUE
ENTERED.
   TIME DURATION OF TIME HISTORY ANALYSIS =      2.500 SECONDS
   NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =     1800
    49. UNIT INCH
  BASE SHEAR UNITS ARE -- KIP  FEET
  MAXIMUM BASE SHEAR  X=  -2.864840E+00  Y=   0.000000E+00  Z=   0.000000E+00
      AT TIMES               2.006944           0.000000           0.000000
    50. PRINT JOINT DISPLACEMENTS
  JOINT    DISPLACE
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    5
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00103
          2    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00075
     2    1    0.03537   0.00000   0.00000   0.00000   0.00000  -0.00050
          2    0.02632   0.00000   0.00000   0.00000   0.00000  -0.00038
     3    1    0.03537   0.00000   0.00000   0.00000   0.00000   0.00050
          2    0.02632   0.00000   0.00000   0.00000   0.00000   0.00038
     4    1    0.00000   0.00000   0.00000   0.00000   0.00000   0.00103
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00075
   ************** END OF LATEST ANALYSIS RESULT **************
    51. UNIT FEET
    52. PRINT MEMBER FORCES
  MEMBER   FORCES
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    6
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
```

```
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      0.00      2.62     0.00      0.00      0.00      -0.00
                2      0.00     -0.88     0.00      0.00      0.00       6.12
          2     1      0.00      1.43     0.00      0.00      0.00       0.00
                2      0.00     -1.43     0.00      0.00      0.00       5.01
     2    1     2      0.00      0.88     0.00      0.00      0.00      -6.12
                3      0.00      0.88     0.00      0.00      0.00       6.12
          2     2      0.00      0.00     0.00      0.00      0.00      -5.01
                3      0.00      0.00     0.00      0.00      0.00       5.01
     3    1     3      0.00     -0.88     0.00      0.00      0.00      -6.12
                4      0.00      2.62     0.00      0.00      0.00       0.00
          2     3      0.00     -1.43     0.00      0.00      0.00      -5.01
                4      0.00      1.43     0.00      0.00      0.00       0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   53. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    7
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    1    1     -2.62      0.00      0.00      0.00      0.00      0.00
         2     -1.43      0.00      0.00      0.00      0.00      0.00
    4    1     -2.62      0.00      0.00      0.00      0.00      0.00
         2     -1.43      0.00      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   54. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:45:45 ****
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    8
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-17 User-Provided Tables

The usage of user-provided steel tables is illustrated in this example for the analysis and design of a plane frame. User-provided tables allow you to specify property data for sections not found in the built-in steel section tables.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-17 User-Provided Tables.STD` when you install the program.

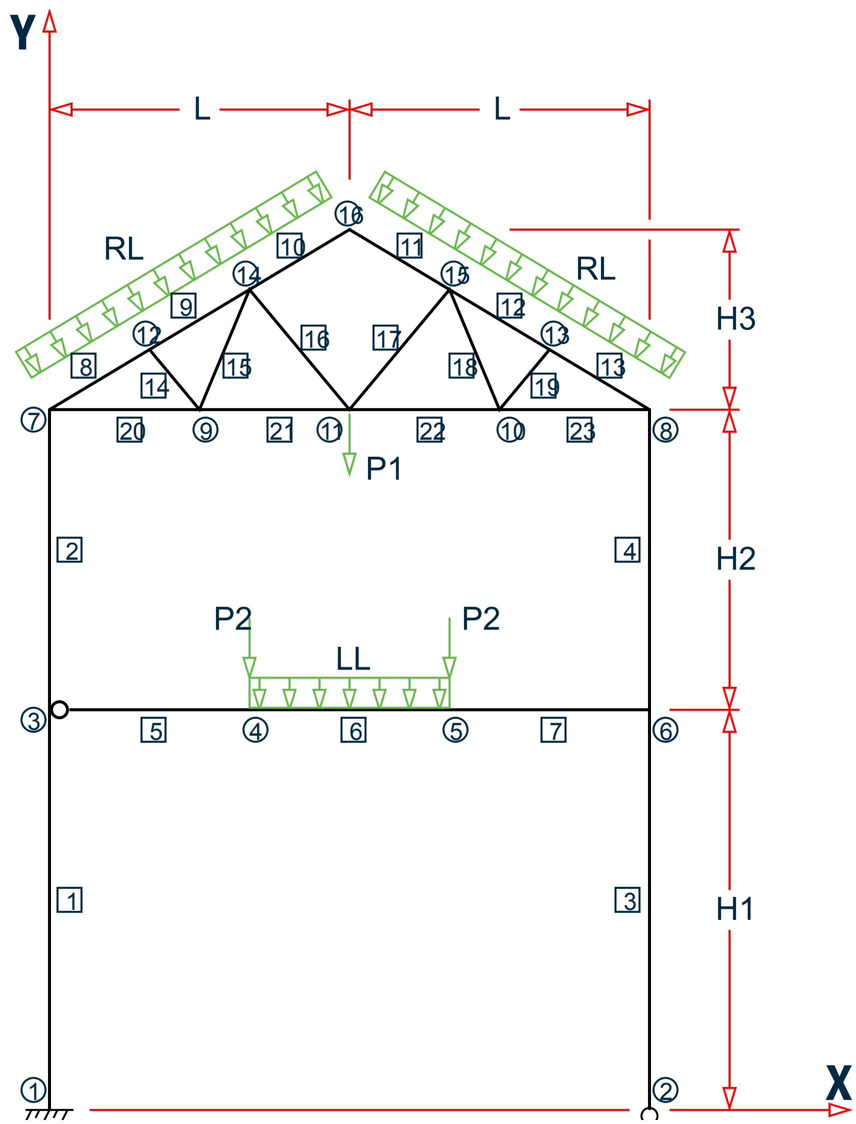


Figure 529: Example Problem No. 17

Where:

L1 = 15 ft, L2 =20 ft
P = 5.0 k
w = 3.0 k/ft

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE EXAMPLE FOR USER TABLE
```

Every input file has to start with the command `STAAD`. The `PLANE` command is used to designate the structure as a plane frame.

```
UNIT FT KIP
```

The `UNIT` command sets the length and force units to be used.

```
JOINT COORDINATES
1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
11 15 35 ; 12 5. 38. ; 13 25 38 ; 14 10 41 ; 15 20 41
16 15 44
```

The above set of data is used to provide joint coordinates for the various joints of the structure. The Cartesian system is being used here. The data consists of the joint number followed by global X and Y coordinates. Note that for a space frame, the Z coordinate(s) need to be provided also.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
```

The above data set contains the member incidence information or the joint connectivity data for each member. This completes the geometry of the structure.

```
START USER TABLE
```

This command is utilized to set up a user-provided steel table. All user-provided steel tables must start with this command.

```
TABLE 1
```

Each table needs an unique numerical identification. The above command starts setting up Table no. 1. Up to twenty tables may be specified per run.

```
UNIT INCH
WIDE FLANGE
```

This command is used to specify the section-type as WIDE FLANGE in this table. Note that several section-types such as WIDE FLANGE, CHANNEL, ANGLE, TEE etc. are available for specification (see TR.19 User Steel Table Specification (on page 2241)).

```
WFL14X30
8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
WFL21X62
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
WFL14X109
32. 14.32 .525 14.605 .86 1240 447 7.12 7.52 16.
```

The above data set is used to specify the properties of three wide flange sections. The data for each section consists of two parts. In the first line, the section-name is provided. You are allowed to provide any section name within twelve characters. The second line contains the section properties required for the particular section-type. Each section-type requires a certain number of data (area of cross-section, depth, moment of inertias etc.) provided in a certain order. For example, in this case, for wide flanges, ten different properties are required.

```
TABLE 2
ANGLES
LANG25255
2.5 2.5 .3125 .489 0 0
LANG40404
4 4 .25 .795 0 0
```

The above command and data lines set up another user provided table consisting of angle sections.

```
END
```

This command signifies the end of the user provided table data set. All user provided table related input must be terminated with this command.

```
MEMBER PROPERTIES
1 3 4 UPT 1 WFL14X109
2 UPT 1 WFL14X30 ; 5 6 7 UPT 1 WFL21X62
8 TO 13 UPT 1 WFL14X30
14 TO 23 UPT 2 LANG40404
```

In the above command lines, the member properties are being assigned from the user provided tables created earlier. The word UPT signifies that the properties are from the user-provided table. This is followed by the table number and then the section name as specified in the use- provided table. The numbers 1 or 2 following the word UPT indicate the table from which section names are fetched.

```
MEMBER TRUSS
14 TO 23
```

The above command is used to designate members 14 to 23 as truss members.

```
MEMBER RELEASE
5 START MZ
```

The MEMBER RELEASE command is used to release the MZ moment at the start joint of member no. 5.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
UNIT FT
```

The length unit is reset to feet using this command.

```
UNIT FT
```

The length unit is reset to feet using this command.

```
SUPPORT
1 FIXED ; 2 PINNED
```

The above command set is used to designate supports. Here, joint 1 is designated as a fixed support and joint 2 is designated as a pinned support.

```
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
```

```
    MEMB LOAD
    8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
```

The above command set is used to specify the loadings on the structure. In this case, dead and live loads are provided through load case 1. It consists of selfweight, concentrated loads at joints 4, 5 and 11, and distributed loads on some members.

```
    PERFORM ANALYSIS
```

This command instructs the program to execute the analysis at this point.

```
    PARAMETER
    CODE AISC UNIFIED
    BEAM 1.0 ALL
    NSF 0.85 ALL
    KY 1.2 MEMB 3 4
```

The above commands are used to specify parameters for steel design.

```
    SELECT MEMBER 3 6 9 19
```

This command will perform selection of members per the AISC ASD steel design code. For each member, the member selection will be performed from the table that was originally used for the specification of the member property. In this case, the selection will be from the respective user tables from which the properties were initially assigned. It may be noted that properties may be provided (and selection may be performed) from built-in steel tables and user provided tables in the same data file.

```
    FINISH
```

This command terminates a STAAD run.

Input File

```
STAAD PLANE EXAMPLE FOR USER TABLE
UNIT FT KIP
JOINT COORDINATES
1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
11 15 35 ; 12 5. 38. ; 13 25 38 ; 14 10 41 ; 15 20 41
16 15 44
MEMBER INCIDENCES
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
UNIT INCH
START USER TABLE
TABLE 1
WIDE FLANGE
WFL14X30
8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
WFL21X62
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
WFL14X109
32. 14.32 .525 14.605 .86 1240 447 7.12 7.52 16.
TABLE 2
ANGLES
LANG25255
```

```
2.5 2.5 .3125 .489 0 0
LANG40404
4 4 .25 .795 0 0
END
MEMBER PROPERTIES
1 3 4 UPT 1 WFL14X109
2 UPT 1 WFL14X30 ; 5 6 7 UPT 1 WFL21X62
8 TO 13 UPT 1 WFL14X30
14 TO 23 UPT 2 LANG40404
*MEMBER TRUSS
*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT FT
SUPPORT
1 FIXED ; 2 PINNED
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -15. ; 11 FY -35.
MEMB LOAD
8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
PERFORM ANALYSIS
PARAMETER
CODE AISC UNIFIED
BEAM 1.0 ALL
NSF 0.85 ALL
KY 1.2 MEMB 3 4
SELECT MEMB 3 6 9 19
FINISH
```

STAAD Output File

```
                                                                    PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:47                      *
```

```
             *                                                   *
             *  Licensed to: Bentley Systems Inc                 *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR USER TABLE
INPUT FILE: US-17 User-Provided Tables.STD
     2. UNIT FT KIP
     3. JOINT COORDINATES
     4. 1 0. 0. ; 2 30 0 ; 3 0 20 0 6 30 20 0
     5. 7 0 35 ; 8 30 35 ; 9 7.5 35 ; 10 22.5 35.
     6. 11 15 35 ; 12 5. 38. ; 13 25 38 ; 14 10 41 ; 15 20 41
     7. 16 15 44
     8. MEMBER INCIDENCES
     9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
    10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
    11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
    12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
    13. 18 10 15 ; 19 10 13 ; 20 7 9
    14. 21 9 11 ; 22 10 11 ; 23 8 10
    15. UNIT INCH
    16. START USER TABLE
    17. TABLE 1
    18. WIDE FLANGE
    19. WFL14X30
    20. 8.85 13.84 .27 6.73 .385 291. 19.6 .38 4.0 4.1
    21. WFL21X62
    22. 18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
    23. WFL14X109
    24. 32. 14.32 .525 14.605 .86 1240 447 7.12 7.52 16.
    25. TABLE 2
    26. ANGLES
    27. LANG25255
    28. 2.5 2.5 .3125 .489 0 0
    29. LANG40404
    30. 4 4 .25 .795 0 0
    31. END
    32. MEMBER PROPERTIES
    33. 1 3 4 UPT 1 WFL14X109
    34. 2 UPT 1 WFL14X30 ; 5 6 7 UPT 1 WFL21X62
    35. 8 TO 13 UPT 1 WFL14X30
    36. 14 TO 23 UPT 2 LANG40404
    37. *MEMBER TRUSS
    38. *14 TO 23
      EXAMPLE FOR USER TABLE                                  -- PAGE NO.    2
    39. MEMB RELEASE
    40. 5 START MZ
    41. 14 TO 23 START MPY 0.99 MPZ 0.99
    42. 14 TO 23 END MPY 0.99 MPZ 0.99
    43. UNIT INCH
    44. DEFINE MATERIAL START
    45. ISOTROPIC STEEL
    46. E 29000
    47. POISSON 0.3
    48. DENSITY 283E-006
    49. ALPHA 6E-006
    50. DAMP 0.03
    51. TYPE STEEL
    52. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    53. END DEFINE MATERIAL
```

```
    54. CONSTANT
    55. MATERIAL STEEL ALL
    56. BETA 90.0 MEMB 3 4
    57. UNIT FT
    58. SUPPORT
    59. 1 FIXED ; 2 PINNED
    60. LOADING 1 DEAD AND LIVE LOAD
    61. SELFWEIGHT Y -1.0
    62. JOINT LOAD
    63. 4 5 FY -15. ; 11 FY -35.
    64. MEMB LOAD
    65. 8 TO 13 UNI Y -0.9 ; 6 UNI GY -1.2
    66. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      43
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    67. PARAMETER
    68. CODE AISC UNIFIED
    69. BEAM 1.0 ALL
    70. NSF 0.85 ALL
    71. KY 1.2 MEMB 3 4
    72. SELECT MEMB 3 6 9 19
      EXAMPLE FOR USER TABLE                                  -- PAGE NO.    3
 PARAMETER 1
      EXAMPLE FOR USER TABLE                                  -- PAGE NO.    4
                     STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                   ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 ***NOTE : AISC 360-16 Design Statement for STAAD.Pro.
 *** AXIS CONVENTION ***:
 ========================
 The capacity results and intermediate results in the report follow the notations
 and axes labels as defined in the AISC 360-16 code.
 The analysis results are reported in STAAD.Pro axis convention and the AISC 360:16
 design results are reported in AISC 360-16 code axis convention.
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
      X/U              Z          Axis typically parallel to the sections principal
major axis.
      Y/V              Y          Axis typically parallel to the sections principal
minor axis.
       Z               X          Longitudinal axis perpendicular to the cross section.
 SECTION FORCES AXIS MAPPING: -
   AISC Spec.      STAAD.Pro      Description
  ------------    -----------     -------------
       Pz              FX         Axial force.
       Vy              FY         Shear force along minor axis.
       Vx              FZ         Shear force along major axis.
       Tz              MX         Torsional moment.
       My              MY         Bending moment about minor axis.
       Mx              MZ         Bending moment about major axis.
 *** DESIGN MESSAGES ***:
```

```
 ======================
    1. Section classification reported is for the cross section and loadcase that
       produced the worst case design ratio for flexure/compression Capacity
results.
    2. Results for any Capacity/Check that is not relevant for a section/loadcase
       based on the code clause in AISC 360-16 will not be shown in the report.
    3. Bending results are reported as being �about� the relevant axis (X/Y), while
       the results for shear are reported as being for shear forces �along� the axis.
       E.g : Mx indicates bending about the X axis, while Vx indicates shear along
       the X axis.
 *** ABBREVIATIONS ***:
 =====================
    F-T-B = Flexural-Torsional Buckling
    L-T-B = Lateral-Torsional Buckling
    F-L-B = Flange Local Buckling
    W-L-B = Web Local Buckling
    L-L-B = Leg Local Buckling
    C-F-Y = Compression Flange Yielding
    T-F-Y = Tension Flange Yielding
      EXAMPLE FOR USER TABLE                                -- PAGE NO.    5
                       STAAD.PRO MEMBER SELECTION - AISC 360-16 LRFD (V1.2)
                     ********************************************************
 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE Noted).
 - Member :     3
|-------------------------------------------------------------------------------|
|  Member No:        3       Profile:  ST  WFL21X62           (UPT)             |
|  Status:        PASS       Ratio:         0.908       Loadcase:        1      |
|  Location:     20.00       Ref:      Eq.H1-1a                                 |
|  Pz:       57.32     C     Vy:        0.000           Vx:     -1.784          |
|  Tz:       0.000           My:       -35.68           Mx:      0.000          |
|-------------------------------------------------------------------------------|
 - Member :     6
|-------------------------------------------------------------------------------|
|  Member No:        6       Profile:  ST  WFL21X62           (UPT)             |
|  Status:        PASS       Ratio:         0.534       Loadcase:        1      |
|  Location:      2.50       Ref:      Eq.H1-1b                                 |
|  Pz:       3.813     T     Vy:       0.1573           Vx:      0.000          |
|  Tz:       0.000           My:        0.000           Mx:     -190.6          |
|-------------------------------------------------------------------------------|
 - Member :     9
|-------------------------------------------------------------------------------|
|  Member No:        9       Profile:  ST  WFL14X30           (UPT)             |
|  Status:        PASS       Ratio:         0.504       Loadcase:        1      |
|  Location:      5.83       Ref:      Eq.H1-3a(H1-1b)                          |
|  Pz:       37.86     C     Vy:        4.144           Vx:      0.000          |
|  Tz:       0.000           My:        0.000           Mx:     -54.45          |
|-------------------------------------------------------------------------------|
 - Member :    19
|-------------------------------------------------------------------------------|
|  Member No:       19       Profile:  ST  LANG25255          (UPT)             |
|  Status:        PASS       Ratio:         0.397       Loadcase:        1      |
|  Location:      1.95       Ref:      Eq.H2-1                                  |
|  Pz:       3.970     C     Vy:       -.4173E-03       Vx:     -.4173E-03      |
|  Tz:       0.000           My:       -.6247E-02       Mx:     0.6247E-02      |
|-------------------------------------------------------------------------------|
    73. FINISH
   ****************************************************************************
   **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
```

```
               IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
               ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
               TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
               FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
               FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
               COMMANDS.
     ****************************************************************************
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:45:48 ****
        EXAMPLE FOR USER TABLE                                   -- PAGE NO.    6
           ************************************************************
           *   For technical assistance on STAAD.Pro, please visit    *
           *   http://www.bentley.com/en/support/                     *
           *                                                          *
           *   Details about additional assistance from               *
           *   Bentley and Partners can be found at program menu      *
           *   Help->Technical Support                                *
           *                                                          *
           *           Copyright (c) Bentley Systems, Inc.            *
           *                http://www.bentley.com                    *
           ************************************************************
```

EX. US-18 Stress Calculation for Plate Elements

This is an example which demonstrates the calculation of principal stresses on a finite element.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-18 Stress Calculation for Plate Elements.STD` when you install the program.

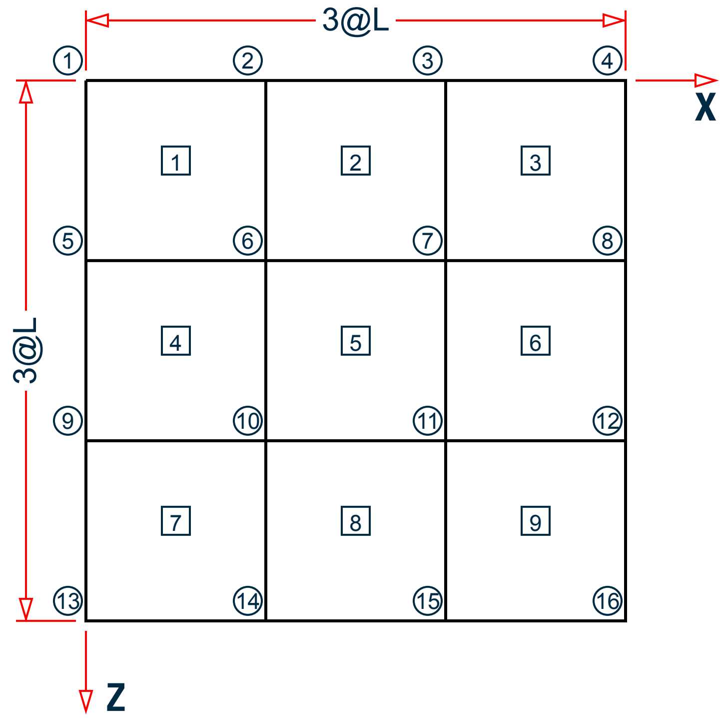


Figure 530: Example Problem No. 18

Where:

L = 1 ft

Fixed Supports at Joints 1, 2, 3, 4, 5, 9, 13

Load intensity = 1 psi (pound/in^2) in negative global Y direction

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D structure).

```
UNIT KIP FEET
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 4 3 0 0
REPEAT 3 0 0 1
```

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command is used to generate coordinates of joints 5 to 16 based on the pattern of joints 1 to 4.

```
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
```

Element connectivities of elements 1 to 3 are defined first, based on which, the connectivities of elements 4 to 9 are generated.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 9 THICK 1.0
```

Elements 1 to 9 have a thickness of 1.0 in.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 4 5 9 13 FIXED
```

Fixed support conditions exist at the above mentioned joints.

```
UNIT POUND
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -1.0
```

A uniform pressure of 1 pound/sq. in is applied on all the elements. In the absence of an explicit direction specification, the load is assumed to act along the local Z axis. The negative value indicates that the load acts opposite to the positive direction of the local Z.

```
PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```
PRINT SUPPORT REACTION
```

The above command is self-explanatory.

```
PRINT ELEMENT STRESSES LIST 4
```

Element stresses at the centroid of the element are printed using the above command. The output includes membrane stresses, shear stresses, bending moments per unit width and principal stresses.

```
FINISH
```

The STAAD run is terminated.

Calculation of principal stresses for element 4

Calculations are presented for the top surface only.

$$SX = 0.0 \text{ lb/in}^2$$

SY = 0.0 lb/in^2

SXY = 0.0 lb/in^2

MX = 16.90 lb·in/in

MY = 85.81 lb·in/in

MXY = 36.43 lb·in/in

S = $1/6t^2 = 1/6 \cdot 1^2 = 0.1667$ in^3 (Section Modulus)

σ_x = SX + MX/S = 0.0 + 16.90/0.1667 = 101.38 psi

σ_y = SY + MY/S = 0.0 + 85.81/0.1667 = 514.75 psi

τ_{xy} = SXY + MXY/S = 0.0 + 36.43/0.1667 = 218.54 psi

$$T_{Max} = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2} = \sqrt{\frac{(101.38 - 514.75)^2}{4} + 218.54^2} = 300.80 psi$$

$S_{Max} = (\sigma_x + \sigma_y)/2 + T_{Max} = (101.38 + 514.75)/2 + 300.80 = 608.87$ psi

$S_{Min} = (\sigma_x + \sigma_y)/2 - T_{Max} = (101.38 + 514.75)/2 - 300.80 = 7.27$ psi

$$\text{Angle} = \frac{1}{2}\tan^{-1}\left(\frac{2\tau_{xy}}{\sigma_x - \sigma_y}\right) = \frac{1}{2}\tan^{-1}\left\{\frac{2(218.54)}{101.38 - 514.75}\right\} = -23.30\ °$$

$$\text{VON} = 0.707\sqrt{(S_{Max} - S_{Min})^2 + S_{Max}^2 + S_{Min}^2} = 0.707\sqrt{(608.87 - 7.27)^2 + 608.87^2 + 7.27^2} = 605.18 psi$$

Input File

```
STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
UNIT KIP FEET
JOINT COORDINATES
1 0 0 0 4 3 0 0
REPEAT 3 0 0 1
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
UNIT INCH
ELEMENT PROPERTIES
1 TO 9 THICK 1.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.7e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 4 5 9 13 FIXED
UNIT POUND
```

```
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -1.0
PERFORM ANALYSIS
PRINT SUPPORT REACTION
PRINT ELEMENT STRESSES LIST 4
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:50                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE SAMPLE CALCULATION FOR
INPUT FILE: US-18 Stress Calculation for Plate Elements.STD
     2. * ELEMENT STRESSES
     3. UNIT KIP FEET
     4. JOINT COORDINATES
     5. 1 0 0 0 4 3 0 0
     6. REPEAT 3 0 0 1
     7. ELEMENT INCIDENCE
     8. 1 1 5 6 2 TO 3
     9. REPEAT 2 3 4
    10. UNIT INCH
    11. ELEMENT PROPERTIES
    12. 1 TO 9 THICK 1.0
    13. DEFINE MATERIAL START
    14. ISOTROPIC CONCRETE
    15. E 3150
    16. POISSON 0.17
    17. DENSITY 8.7E-005
    18. ALPHA 5E-006
    19. DAMP 0.05
    20. G 1346.15
    21. TYPE CONCRETE
    22. STRENGTH FCU 4
    23. END DEFINE MATERIAL
    24. CONSTANTS
    25. MATERIAL CONCRETE ALL
    26. SUPPORT
    27. 1 TO 4 5 9 13 FIXED
    28. UNIT POUND
    29. LOAD 1
    30. ELEMENT LOAD
    31. 1 TO 9 PRESSURE -1.0
    32. PERFORM ANALYSIS
      SAMPLE CALCULATION FOR                                   -- PAGE NO.    2
    * ELEMENT STRESSES
            P R O B L E M   S T A T I S T I C S
```

```
          -----------------------------------
    NUMBER OF JOINTS         16  NUMBER OF MEMBERS       0
    NUMBER OF PLATES          9  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      7
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      54
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   33. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    3
   * ELEMENT STRESSES
  SUPPORT REACTIONS -UNIT POUN INCH    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      0.00     -9.76      0.00    -12.51      0.00     12.51
     2    1      0.00     70.14      0.00   -853.81      0.00    -16.81
     3    1      0.00    301.41      0.00  -2821.43      0.00     95.50
     4    1      0.00    281.33      0.00  -2127.22      0.00   -769.10
     5    1      0.00     70.14      0.00     16.81      0.00    853.81
     9    1      0.00    301.41      0.00    -95.50      0.00   2821.43
    13    1      0.00    281.33      0.00    769.10      0.00   2127.22
  ************** END OF LATEST ANALYSIS RESULT **************
   34. PRINT ELEMENT STRESSES LIST 4
 ELEMENT  STRESSES LIST     4
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    4
   * ELEMENT STRESSES
  ELEMENT STRESSES    FORCE,LENGTH UNITS= POUN INCH
  ----------------
          STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
     4      1        10.43       -9.94       16.90       85.81       36.43
                    605.39      605.39        0.00        0.00        0.00
                    608.99      608.99
      TOP : SMAX=     608.99 SMIN=       7.25 TMAX=     300.87 ANGLE= 66.7
      BOTT: SMAX=      -7.25 SMIN=    -608.99 TMAX=     300.87 ANGLE=-23.3
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         6.089873E+02 -6.089873E+02  3.008683E+02  6.053945E+02  6.089873E+02
PLATE NO.       4             4             4             4             4
CASE  NO.       1             1             1             1             1
  ********************END OF ELEMENT FORCES********************
   35. FINISH
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    5
   * ELEMENT STRESSES
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:50 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
```

```
        *   Help->Technical Support                               *
        *                                                         *
        *           Copyright (c) Bentley Systems, Inc.           *
        *                http://www.bentley.com                   *
        ***********************************************************
```

EX. US-19 Inclined Supports

This example demonstrates the usage of inclined supports. The word `INCLINED` refers to the fact that the restraints at a joint where such a support is specified are along a user-specified axis system instead of along the default directions of the global axis system. STAAD.Pro offers a few different methods for assigning inclined supports, and we examine those in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-19 Inclined Supports.STD` when you install the program.

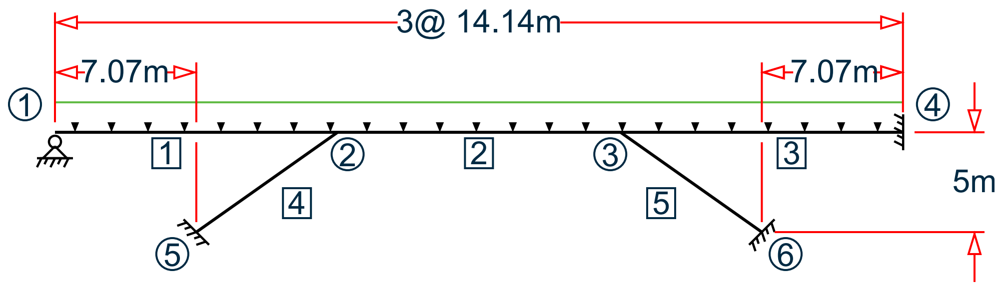


Figure 531: Example Problem No. 19

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
INPUT WIDTH 79
```

Every input has to start with the term `STAAD`. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y, and Z coordinates.

```
UNIT METER KN
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
```

Defines the members by the joints to which they are connected.

```
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
```

Properties for all members of the model are provided using the PRISMATIC option (abbreviated to PRIS here). YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. The values are provided in MMS unit.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.72
POISSON 0.17
DENSITY 2.35615e-008
ALPHA 5e-006
DAMP 0.05
G 9.281
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

Material constants like E (modulus of elasticity) and Poisson's ratio are specified following the command CONSTANTS.

```
UNIT METER KN
SUPPORTS
5 INCLINED REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INCLINED REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INCLINED 1 0 1 FIXED BUT FX MX MY MZ
```

We assign supports (restraints) at 4 nodes - 5, 6, 1 and 4. For 3 of those, namely, 5, 6 and 4, the node number is followed by the keyword INCLINED, signifying that an INCLINED support is defined there. For the remaining one - node 1 - that keyword is missing. Hence, the support at node 1 is a global direction support.

The most important aspect of inclined supports is their axis system. Each node where an inclined support is defined has its own distinct local X, local Y and local Z axes. In order to define the axis system, we first have to define a datum point. The support node and the datum point together help define the axis system.

Three different methods are shown in the above 3 instances for defining the datum point.

- At node 5, notice the keyword REF followed by the numbers (10,5,10). This means that the datum point associated with node 5 is one which has the global coordinates of (10m, 5m, 10m). Coincidentally, this happens to be node 2.
- At node 6, the keyword REFJT is used followed by the number 3. This means that the datum point for support node 6 is the joint number 3 of the model. The coordinates of the datum point are hence those of node 3, namely, (20m, 5m and 20m).
- At node 4, the word INCLINED is merely followed by 3 numbers (1,0,1). In the absence of the words REF and REFJT, the program sets the datum point to be the following. It takes the coordinates of node 4, which are (30m,5m,30m) and adds to them, the 3 numbers which comes after the word INCLINED. Thus, the datum point becomes (31m, 5m and 31m).

Once the datum point is established, the local axis system is defined as follows. Local X is a straight line (vector) pointing from the support node towards the datum point. Local Z is the vector obtained by the cross product of local X and the global Y axis (unless the SET Z UP command is used in which case one would use global Z instead of global Y and that would yield local Y). Local Y is the vector resulting from the cross product of local Z and local X. The right hand rule must be used when performing these cross products.

Notice the unique nature of these datum points. The one for node 5 tells us that a line connecting nodes 5 to 2 is the local X axis, and is hence along the axis of member 4. By defining a KFX spring at that one, we are saying that the lower end of member 4 can move along its axis like the piston of a car engine. Think of a pile bored into rock with a certain amount of freedom to expand and contract axially.

The same is true for the support at the bottom of member 5. The local X axis of that support is along the axis of member 5. That also happens to be the case for the supported end of member 3. The line going from node 4 to the datum point (31,5,31) happens to be coincident with the axis of the member, or the traffic direction. The expression FIXED BUT FX MX MY MZ for that support indicates that it is free to translate along local X, suggesting that it is an expansion joint - free to expand or contract along the axis of member 3.

Since MX, MY, and MZ are all released at these supports, no moment will be resisted by these supports.

```
    LOAD 1 DEAD LOAD
    SELFWEIGHT Y -1.2
    LOAD 2 LIVE LOAD
    MEMBER LOAD
    1 TO 3 UNI GY -6
    LOAD COMB 3
    1 1.0 2 1.0
    PERFORM ANALYSIS PRINT STATICS CHECK
```

Three load cases followed by the instruction for the type of analysis are specified. The `PRINT STATICS CHECK` option will instruct the program to produce a report consisting of total applied load versus total reactions from the supports for each primary load case.

```
    PRINT SUPPORT REACTION
```

By default, support reactions are printed in the global axis directions. The above command is an instruction for such a report.

```
    SET INCLINED REACTION
    PRINT SUPPORT REACTION
```

Just earlier, we saw how to obtain support reactions in the global axis system. What if we need them in the inclined axis system? The “SET INCLINED REACTION” is a switch for that purpose. It tells the program that reactions should be reported in the inclined axis system instead of the global axis system. This has to be followed by the PRINT SUPPORT REACTIONS command.

```
    PRINT MEMBER FORCES
    PRINT JOINT DISP
    FINISH
```

Member forces are reported in the local axis system of the members. Joint displacements at all joints are reported in the global axis system. Following this, the STAAD run is terminated.

Input File

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
DEFINE MATERIAL START
```

```
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER KN
SUPPORTS
5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INC 1 0 1 FIXED BUT FX MX MY MZ
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT SUPPORT REACTION
SET INCLINED REACTION
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:52                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: US-19 Inclined Supports.STD
     2. INPUT WIDTH 79
     3. UNIT METER KN
     4. JOINT COORDINATES
     5. 1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25
     6. MEMBER INCIDENCES
     7. 1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3
     8. UNIT MMS KN
     9. MEMBER PROPERTY AMERICAN
    10. 4 5 PRIS YD 800
```

```
   11. 1 TO 3 PRIS YD 750 ZD 500
   12. DEFINE MATERIAL START
   13. ISOTROPIC CONCRETE
   14. E 21.0
   15. POISSON 0.17
   16. DENSITY 2.36158E-008
   17. ALPHA 5E-006
   18. DAMP 0.05
   19. G 9.25
   20. TYPE CONCRETE
   21. STRENGTH FCU 0.0275
   22. END DEFINE MATERIAL
   23. CONSTANTS
   24. MATERIAL CONCRETE ALL
   25. UNIT METER KN
   26. SUPPORTS
   27. 5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
   28. 6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
   29. 1 PINNED
   30. 4 INC 1 0 1 FIXED BUT FX MX MY MZ
   31. LOAD 1 DEAD LOAD
   32. SELFWEIGHT Y -1.2
   33. LOAD 2 LIVE LOAD
   34. MEMBER LOAD
   35. 1 TO 3 UNI GY -6
   36. LOAD COMB 3
   37. 1 1.0 2 1.0
   38. PERFORM ANALYSIS PRINT STATICS CHECK
     STAAD SPACE                                              -- PAGE NO.    2
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      27
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
     STAAD SPACE                                              -- PAGE NO.    3
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
         DEAD LOAD
          CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.150000000E+02
                       Y =  0.411580006E+01
                       Z =  0.150000000E+02
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KN   METE ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =        -697.60
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       10463.94  MY=           0.00  MZ=      -10463.94
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KN   METE ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          -0.00
      SUMMATION FORCE-Y =         697.60
      SUMMATION FORCE-Z =          -0.00
```

```
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      -10463.94  MY=          -0.00  MZ=       10463.94
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      1)
           MAXIMUMS    AT NODE
     X = -8.02030E-01       5
     Y = -2.50768E+00       3
     Z = -8.02030E-01       5
     RX= -2.71938E-03       4
     RY=  8.02194E-18       1
     RZ=  2.71938E-03       4
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
         LIVE LOAD
          CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                        X =  0.149999999E+02
                        Y =  0.500000017E+01
                        Z =  0.149999999E+02
     STAAD SPACE                                              -- PAGE NO.    4
TOTAL APPLIED LOAD     2
  ***TOTAL APPLIED LOAD ( KN   METE ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =        -254.56
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=        3818.38  MY=           0.00  MZ=       -3818.38
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KN   METE ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =          -0.00
      SUMMATION FORCE-Y =         254.56
      SUMMATION FORCE-Z =          -0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       -3818.38  MY=          -0.00  MZ=        3818.38
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      2)
           MAXIMUMS    AT NODE
     X = -2.97766E-01       5
     Y = -9.34280E-01       3
     Z = -2.97766E-01       5
     RX= -1.21481E-03       4
     RY= -3.94177E-18       4
     RZ=  1.21481E-03       4
  ************ END OF DATA FROM INTERNAL STORAGE ************
   39. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     STAAD SPACE                                              -- PAGE NO.    5
  SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     5    1    216.27    289.28    216.27      0.00      0.00      0.00
          2     86.61     94.78     86.61      0.00      0.00      0.00
          3    302.88    384.06    302.88      0.00      0.00      0.00
     6    1   -213.07    287.50   -213.07      0.00      0.00      0.00
          2    -85.33     94.06    -85.33      0.00      0.00      0.00
          3   -298.40    381.56   -298.40      0.00      0.00      0.00
     1    1     -3.20     60.34     -3.20      0.00      0.00      0.00
          2     -1.28     32.84     -1.28      0.00      0.00      0.00
          3     -4.48     93.18     -4.48      0.00      0.00      0.00
     4    1      0.00     60.47      0.00      0.00      0.00      0.00
          2      0.00     32.89      0.00      0.00      0.00      0.00
```

```
          3      0.00     93.36      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    40. SET INCLINED REACTION
    41. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      STAAD SPACE                                              -- PAGE NO.    6
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     5    1    416.75     59.61      0.00      0.00      0.00      0.00
          2    154.72      6.67      0.00      0.00      0.00      0.00
          3    571.47     66.28      0.00      0.00      0.00      0.00
     6    1    412.02     60.77      0.00      0.00      0.00      0.00
          2    152.83      7.13      0.00      0.00      0.00      0.00
          3    564.85     67.90      0.00      0.00      0.00      0.00
     1    1     -3.20     60.34     -3.20      0.00      0.00      0.00
          2     -1.28     32.84     -1.28      0.00      0.00      0.00
          3     -4.48     93.18     -4.48      0.00      0.00      0.00
     4    1      0.00     60.47      0.00      0.00      0.00      0.00
          2      0.00     32.89      0.00      0.00      0.00      0.00
          3      0.00     93.36      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    42. PRINT MEMBER FORCES
  MEMBER   FORCES
      STAAD SPACE                                              -- PAGE NO.    7
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      -4.52     60.34     0.00      0.00     -0.00      -0.00
                2       4.52     89.95    -0.00      0.00     -0.00    -209.31
          2     1      -1.81     32.84     0.00     -0.00     -0.00       0.00
                2       1.81     52.02    -0.00      0.00     -0.00    -135.62
          3     1      -6.33     93.18     0.00     -0.00     -0.00      -0.00
                2       6.33    141.96    -0.00      0.00     -0.00    -344.93
     2    1     2     301.33     75.98    -0.00      0.00      0.00     129.19
                3    -301.33     74.31     0.00     -0.00     -0.00    -117.44
          2     2     120.67     42.76    -0.00      0.00      0.00      77.86
                3    -120.67     42.09     0.00     -0.00     -0.00     -73.16
          3     2     422.00    118.73    -0.00      0.00      0.00     207.05
                3    -422.00    116.41     0.00     -0.00     -0.00    -190.59
     3    1     3       0.00     89.82    -0.00      0.00      0.00     207.54
                4       0.00     60.47     0.00      0.00      0.00       0.00
          2     3       0.00     51.97    -0.00     -0.00      0.00     134.91
                4      -0.00     32.89     0.00      0.00      0.00       0.00
          3     3       0.00    141.79    -0.00     -0.00      0.00     342.45
                4      -0.00     93.36     0.00      0.00      0.00       0.00
     4    1     5     416.75     59.61     0.00     -0.00      0.00      -0.00
                2    -345.52     41.11    -0.00      0.00     -0.00      80.12
          2     5     154.72      6.67     0.00      0.00      0.00       0.00
                2    -154.72     -6.67    -0.00      0.00      0.00      57.76
          3     5     571.47     66.28     0.00     -0.00      0.00       0.00
                2    -500.25     34.44    -0.00      0.00     -0.00     137.88
     5    1     6     412.02     60.77     0.00     -0.00     -0.00      -0.00
                3    -340.80     39.96    -0.00      0.00     -0.00      90.10
          2     6     152.83      7.13     0.00     -0.00     -0.00       0.00
                3    -152.83     -7.13    -0.00      0.00     -0.00      61.75
          3     6     564.85     67.90     0.00     -0.00     -0.00       0.00
```

```
                 3   -493.63     32.83    -0.00      0.00     -0.00     151.86
   ************** END OF LATEST ANALYSIS RESULT **************
    43. PRINT JOINT DISP
     STAAD SPACE                                              -- PAGE NO.    8
  JOINT    DISP
     STAAD SPACE                                              -- PAGE NO.    9
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0027    0.0000   -0.0027
           2     0.0000    0.0000    0.0000    0.0012    0.0000   -0.0012
           3     0.0000    0.0000    0.0000    0.0039    0.0000   -0.0039
      2    1     0.0006   -2.4614    0.0006    0.0007    0.0000   -0.0007
           2     0.0002   -0.9157    0.0002    0.0003    0.0000   -0.0003
           3     0.0008   -3.3771    0.0008    0.0011    0.0000   -0.0011
      3    1    -0.0377   -2.5077   -0.0377   -0.0007    0.0000    0.0007
           2    -0.0151   -0.9343   -0.0151   -0.0003    0.0000    0.0003
           3    -0.0528   -3.4420   -0.0528   -0.0011    0.0000    0.0011
      4    1    -0.0377    0.0000   -0.0377   -0.0027    0.0000    0.0027
           2    -0.0151    0.0000   -0.0151   -0.0012    0.0000    0.0012
           3    -0.0528    0.0000   -0.0528   -0.0039    0.0000    0.0039
      5    1    -0.8020   -0.8020   -0.8020    0.0024    0.0000   -0.0024
           2    -0.2978   -0.2978   -0.2978    0.0007    0.0000   -0.0007
           3    -1.0998   -1.0998   -1.0998    0.0031    0.0000   -0.0031
      6    1     0.7929   -0.7929    0.7929   -0.0024    0.0000    0.0024
           2     0.2941   -0.2941    0.2941   -0.0008    0.0000    0.0008
           3     1.0871   -1.0871    1.0871   -0.0032    0.0000    0.0032
   ************** END OF LATEST ANALYSIS RESULT **************
    44. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:52 ****
     STAAD SPACE                                              -- PAGE NO.   10
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *M. To assign an inclined support* (on page 823)
- *TR.27.2 Inclined Support Specification* (on page 2318)
- *Create Support dialog* (on page 2815)

EX. US-20 Generating a Structure in Cylindrical Coordinates

This example generates the geometry of a cylindrical tank structure using the cylindrical coordinate system. The tank lies on its side in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-20 Generating a Structure in Cylindrical Coordinates.STD` when you install the program.

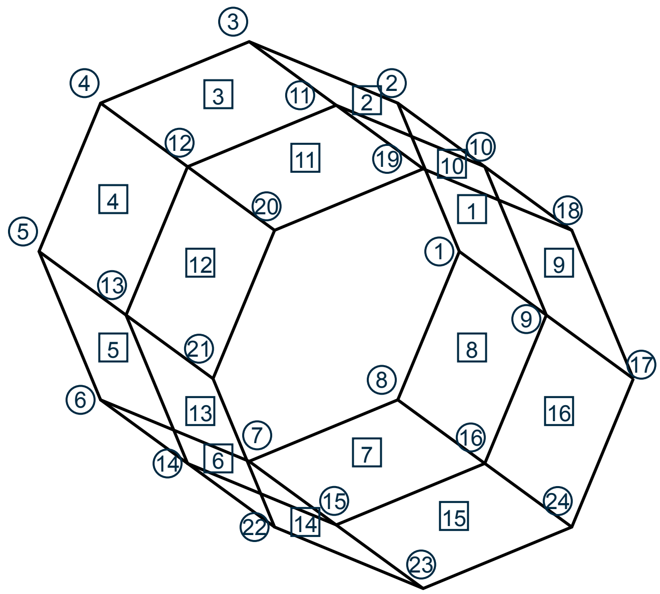


Figure 532: Example Problem No. 20

In this example, a cylindrical tank is modeled using finite elements. The radial direction is in the XY plane and longitudinal direction is along the Z-axis. Hence, the coordinates in the XY plane are generated using the cylindrical coordinate system.

```
STAAD SPACE
```

```
UNIT KIP FEET
```

The type of structure (space frame) and length and force units for data to follow are specified.

```
JOINT COORD CYLINDRICAL
```

The above command instructs the program that the coordinate data that follows is in the cylindrical coordinate system (r,theta,z).

```
1 10 0 0 8 10 315 0
```

Joint 1 has an 'r' of 10 feet, theta of 0 degrees and Z of 0 ft. Joint 8 has an 'r' of 10 feet, theta of 315 degrees and Z of 0 ft. The 315 degrees angle is measured counter-clockwise from the +ve direction of the X-axis. Joints 2 to 7 are generated by equal increments of the coordinate values between joints 1 and 8.

```
REPEAT 2 0 0 8.5
```

The REPEAT command is used to generate joints 9 through 24 by repeating twice, the pattern of joints 1 to 8 at Z-increments of 8.5 ft for each `REPEAT`.

```
PRINT JOINT COORD
```

The above command is used to produce a report consisting of the coordinates of all the joints in the Cartesian coordinate system. Note that even though the input data was in the cylindrical coordinate system, the output is in the Cartesian coordinate system.

```
ELEMENT INCIDENCES
1 1 2 10 9 TO 7 1 1
8 8 1 9 16
REPEAT ALL 1 8 8
```

The above 4 lines identify the element incidences of all 16 elements. Incidences of element 1 is defined as 1 2 10 9. Incidences of element 2 is generated by incrementing the joint numbers of element 1 by 1, incidences of element 3 is generated by incrementing the incidences of element 2 by 1 and so on up to element 7. Incidences of element 8 has been defined above as 8 1 9 16. The `REPEAT ALL` command states that the pattern of all the elements defined by the previous 2 lines, namely elements 1 to 8, must be repeated once with an element number increment of 8 and a joint number increment of 8 to generate elements 9 through 16.

```
PRINT ELEMENT INFO
```

The above command is self-explanatory.

```
FINISH
```

Input File

```
STAAD SPACE
UNIT KIP FEET
JOINT COORD CYLINDRICAL
1 10 0 0 8 10 315 0
REPEAT 2 0 0 8.5
PRINT JOINT COORD
ELEMENT INCIDENCES
1 1 2 10 9 TO 7 1 1
8 8 1 9 16
REPEAT ALL 1 8 8
PRINT ELEMENT INFO
FINISH
```

STAAD Output File

```
                                                              PAGE NO.    1
             ****************************************************
             *                                                  *
             *          STAAD.Pro CONNECT Edition               *
             *          Version  22.10.00.***                   *
             *          Proprietary Program of                  *
             *          Bentley Systems, Inc.                   *
             *          Date=    MAR 24, 2022                   *
             *          Time=     9:45:58                       *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: US-20 Generating a Structure in Cylindrical Coordinates.STD
     2. UNIT KIP FEET
     3. JOINT COORD CYLINDRICAL
     4. 1 10 0 0 8 10 315 0
     5. REPEAT 2 0 0 8.5
     6. PRINT JOINT COORD
  JOINT    COORD
```

```
     STAAD SPACE                                            -- PAGE NO.    2
   JOINT COORDINATES
   -----------------
   COORDINATES ARE FEET UNIT
   JOINT            X             Y             Z
      1          10.000         0.000         0.000
      2           7.071         7.071         0.000
      3           0.000        10.000         0.000
      4          -7.071         7.071         0.000
      5         -10.000         0.000         0.000
      6          -7.071        -7.071         0.000
      7          -0.000       -10.000         0.000
      8           7.071        -7.071         0.000
      9          10.000         0.000         8.500
     10           7.071         7.071         8.500
     11           0.000        10.000         8.500
     12          -7.071         7.071         8.500
     13         -10.000         0.000         8.500
     14          -7.071        -7.071         8.500
     15          -0.000       -10.000         8.500
     16           7.071        -7.071         8.500
     17          10.000         0.000        17.000
     18           7.071         7.071        17.000
     19           0.000        10.000        17.000
     20          -7.071         7.071        17.000
     21         -10.000         0.000        17.000
     22          -7.071        -7.071        17.000
     23          -0.000       -10.000        17.000
     24           7.071        -7.071        17.000
   ************ END OF DATA FROM INTERNAL STORAGE ************
     7. ELEMENT INCIDENCES
     8. 1 1 2 10 9 TO 7 1 1
     9. 8 8 1 9 16
    10. REPEAT ALL 1 8 8
    11. PRINT ELEMENT INFO
  ELEMENT  INFO
     STAAD SPACE                                            -- PAGE NO.    3
   ELEMENT INFORMATION
   -------------------
  ELEMENT          INCIDENCES          THICK   POISS    E1/E2        G1/G2        AREA
    NO.                                (FEET)
      1       1      2     10      9     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                      0.000000E+00 0.000000E+00
      2       2      3     11     10     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                      0.000000E+00 0.000000E+00
      3       3      4     12     11     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                      0.000000E+00 0.000000E+00
      4       4      5     13     12     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                      0.000000E+00 0.000000E+00
      5       5      6     14     13     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                      0.000000E+00 0.000000E+00
      6       6      7     15     14     0.000  0.000 0.000000E+00 0.000000E+00
6.505619E+01
```

```
                                                          0.000000E+00 0.000000E+00
      7       7       8      16      15     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
      8       8       1       9      16     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
      9       9      10      18      17     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     10      10      11      19      18     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     11      11      12      20      19     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     12      12      13      21      20     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     13      13      14      22      21     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     14      14      15      23      22     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     15      15      16      24      23     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
     16      16       9      17      24     0.000   0.000 0.000000E+00 0.000000E+00
6.505619E+01
                                                          0.000000E+00 0.000000E+00
   ******************END OF ELEMENT INFO******************
    12. FINISH
      STAAD SPACE                                              -- PAGE NO.    4
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:45:58 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. US-21 Analysis of a Structure with Tension-Only Members

This example illustrates the modeling of tension-only members using the `MEMBER TENSION` command.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-21 Analysis of a Structure with Tension-Only Members.STD` when you install the program.

It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of "active" members (and hence the stiffness matrix) is load case dependent.

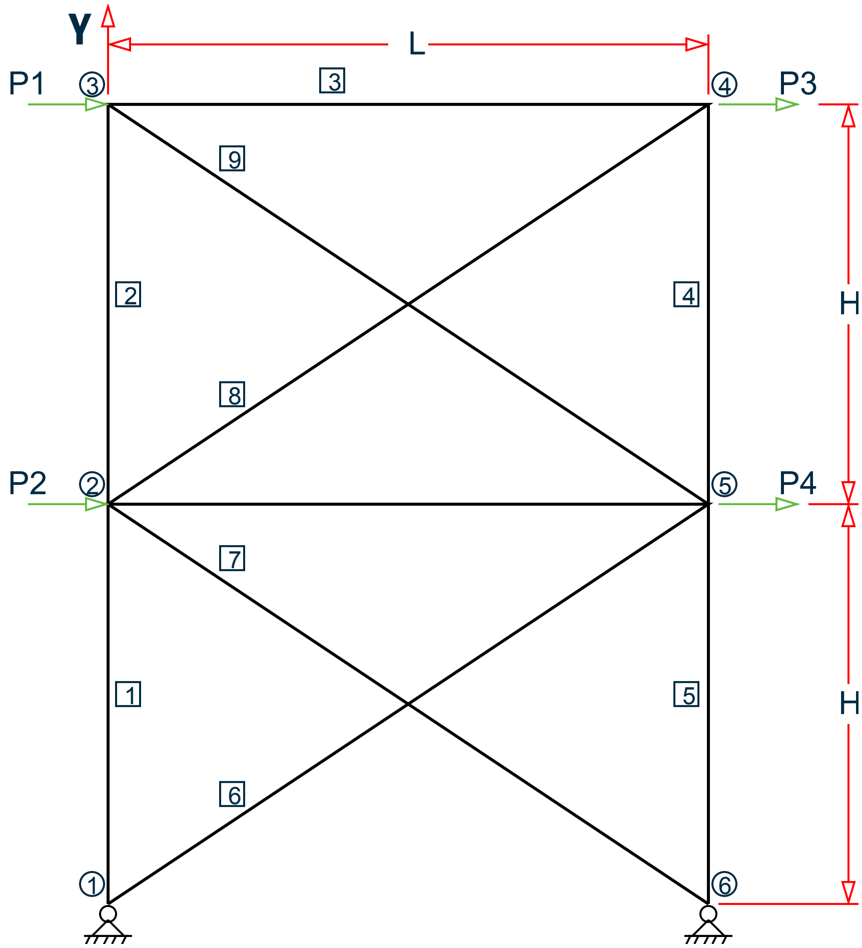


Figure 533: Example Problem No. 21

where:

L = 15 ft, H = 10 ft
Load case 1: P1 = 10 kips & P2 = 15 kips
Load case 2: P3 = -10 kips & P4 = -15 kips

```
STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
```

The input data is initiated with the word `STAAD`. This structure is a `PLANE` frame.

```
UNIT FEET KIP
```

Units for the commands to follow are defined above.

```
JOINT COORDINATES
1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
```

Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

Incidences of members 1 to 10 are defined.

```
MEMBER TENSION
6 TO 9
```

Members 6 to 9 are defined as tension-only members using the `MEMBER TENSION` command. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a compressive force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26
```

All members have been assigned a `WIDE FLANGE` section from the built in American table.

```
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The length units have been changed from feet to inch to facilitate the input of these values.

```
SUPPORT
1 6 PINNED
```

The supports are defined above.

```
MEMBER TENSION
6 TO 9
```

One or more among the members 6 to 9 may have been inactivated in the analysis.

```
LOAD 1
JOINT LOAD
2 FX 15
3 FX 10
```

Load 1 is defined above and consists of joint loads at joints 2 and 3.

```
LOAD 2
JOINT LOAD
4 FX -10
5 FX -15
```

Load case 2 is described above.

```
    LOAD 3
    REPEAT LOAD
    1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously.

```
    PRINT ANALYSIS RESULTS
    FINI
```

The analysis results are printed and the run terminated.

Input File

```
STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
UNIT FEET KIP
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER TENSION
6 TO 9
MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000.0
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 PINNED
6 PINNED
LOAD 1
JOINT LOAD
2 FX 15
3 FX 10
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
6 TO 9
LOAD 2
JOINT LOAD
4 FX -10
5 FX -15
PERFORM ANALYSIS
CHANGE
```

```
MEMBER TENSION
6 TO 9
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ANALYSIS RESULTS
FINI
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46: 0                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
INPUT FILE: US-21 Analysis of a Structure with Tension-Only Members.STD
     2. UNIT FEET KIP
     3. SET NL 3
     4. JOINT COORDINATES
     5. 1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
     6. MEMBER INCIDENCES
     7. 1 1 2 5
     8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
     9. MEMBER TENSION
    10. 6 TO 9
    11. MEMBER PROPERTY AMERICAN
    12. 1 TO 10 TA ST W12X26
    13. UNIT INCH
    14. DEFINE MATERIAL START
    15. ISOTROPIC STEEL
    16. E 29000.0
    17. POISSON 0.3
    18. DENSITY 283E-006
    19. ALPHA 6E-006
    20. DAMP 0.03
    21. TYPE STEEL
    22. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    23. END DEFINE MATERIAL
    24. CONSTANT
    25. MATERIAL STEEL ALL
    26. SUPPORT
    27. 1 PINNED
    28. 6 PINNED
    29. LOAD 1
    30. JOINT LOAD
    31. 2 FX 15
    32. 3 FX 10
```

```
    33. PERFORM ANALYSIS
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    1
    34. CHANGE
    35. MEMBER TENSION
    36. 6 TO 9
    37. LOAD 2
    38. JOINT LOAD
    39. 4 FX -10
    40. 5 FX -15
    41. PERFORM ANALYSIS
*** LOAD CASE     2 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    2
    42. CHANGE
    43. MEMBER TENSION
    44. 6 TO 9
    45. LOAD 3
    46. REPEAT LOAD
    47. 1 1.0 2 1.0
    48. PERFORM ANALYSIS
*** LOAD CASE     3 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    3
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    3
    49. CHANGE
    50. LOAD LIST ALL
    51. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    4
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00062
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00039
           3    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00007
      2    1    0.06285   0.00373   0.00000   0.00000   0.00000  -0.00030
           2   -0.04313  -0.01262   0.00000   0.00000   0.00000   0.00028
           3    0.00605   0.00000   0.00000   0.00000   0.00000  -0.00001
      3    1    0.09724   0.00387   0.00000   0.00000   0.00000  -0.00018
           2   -0.08929  -0.01613   0.00000   0.00000   0.00000   0.00029
           3    0.00408   0.00000   0.00000   0.00000   0.00000   0.00002
      4    1    0.08929  -0.01613   0.00000   0.00000   0.00000  -0.00029
           2   -0.09724   0.00387   0.00000   0.00000   0.00000   0.00018
           3   -0.00408   0.00000   0.00000   0.00000   0.00000  -0.00002
      5    1    0.04313  -0.01262   0.00000   0.00000   0.00000  -0.00028
           2   -0.06285   0.00373   0.00000   0.00000   0.00000   0.00030
           3   -0.00605   0.00000   0.00000   0.00000   0.00000   0.00001
      6    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00039
           2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00062
```

```
          3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00007
     EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    5
  SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1    -24.91    -23.33      0.00      0.00      0.00      0.00
          2      0.09     23.33      0.00      0.00      0.00      0.00
          3     -0.05      0.00      0.00      0.00      0.00      0.00
     6    1     -0.09     23.33      0.00      0.00      0.00      0.00
          2     24.91    -23.33      0.00      0.00      0.00      0.00
          3      0.05      0.00      0.00      0.00      0.00      0.00
     EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    6
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  INCH     (LOCAL )
 MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      -6.90      0.26     0.00      0.00      0.00       0.00
                2       6.90     -0.26     0.00      0.00      0.00      31.66
          2     1      23.33     -0.09     0.00      0.00      0.00      -0.00
                2     -23.33      0.09     0.00      0.00      0.00     -10.81
          3     1       0.00      0.05     0.00      0.00      0.00       0.00
                2      -0.00     -0.05     0.00      0.00      0.00       5.46
     2    1     2      -0.24      0.20     0.00      0.00      0.00       6.07
                3       0.24     -0.20     0.00      0.00      0.00      18.42
          2     2       6.49     -0.43     0.00      0.00      0.00     -25.74
                3      -6.49      0.43     0.00      0.00      0.00     -25.59
          3     2       0.00     -0.05     0.00      0.00      0.00      -4.66
                3      -0.00      0.05     0.00      0.00      0.00      -1.39
     3    1     3       9.80     -0.24     0.00      0.00      0.00     -18.42
                4      -9.80      0.24     0.00      0.00      0.00     -25.59
          2     3       9.80      0.24     0.00      0.00      0.00      25.59
                4      -9.80     -0.24     0.00      0.00      0.00      18.42
          3     3      10.05      0.00     0.00      0.00      0.00       1.39
                4     -10.05     -0.00     0.00      0.00      0.00      -1.39
     4    1     4       6.49      0.43     0.00      0.00      0.00      25.59
                5      -6.49     -0.43     0.00      0.00      0.00      25.74
          2     4      -0.24     -0.20     0.00      0.00      0.00     -18.42
                5       0.24      0.20     0.00      0.00      0.00      -6.07
          3     4      -0.00      0.05     0.00      0.00      0.00       1.39
                5       0.00     -0.05     0.00      0.00      0.00       4.66
     5    1     5      23.33      0.09     0.00      0.00      0.00      10.81
                6     -23.33     -0.09     0.00      0.00      0.00       0.00
          2     5      -6.90     -0.26     0.00      0.00      0.00     -31.66
                6       6.90      0.26     0.00      0.00      0.00       0.00
          3     5      -0.00     -0.05     0.00      0.00      0.00      -5.46
                6       0.00      0.05     0.00      0.00      0.00       0.00
     6    1     1     -29.62      0.00     0.00      0.00      0.00       0.00
                5      29.62      0.00     0.00      0.00      0.00       0.00
          2     1       0.00      0.00     0.00      0.00      0.00       0.00
                5       0.00      0.00     0.00      0.00      0.00       0.00
          3     1       0.00      0.00     0.00      0.00      0.00       0.00
                5       0.00      0.00     0.00      0.00      0.00       0.00
     7    1     2       0.00      0.00     0.00      0.00      0.00       0.00
                6       0.00      0.00     0.00      0.00      0.00       0.00
     EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    7
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  INCH     (LOCAL )
```

```
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    2    -29.62      0.00     0.00      0.00      0.00       0.00
                6     29.62      0.00     0.00      0.00      0.00       0.00
           3    2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
      8    1    2    -11.26      0.00     0.00      0.00      0.00       0.00
                4     11.26      0.00     0.00      0.00      0.00       0.00
           2    2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
           3    2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
      9    1    3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
           2    3    -11.26      0.00     0.00      0.00      0.00       0.00
                5     11.26      0.00     0.00      0.00      0.00       0.00
           3    3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
     10    1    2     24.31     -0.41     0.00      0.00      0.00     -37.72
                5    -24.31      0.41     0.00      0.00      0.00     -36.55
           2    2     24.31      0.41     0.00      0.00      0.00      36.55
                5    -24.31     -0.41     0.00      0.00      0.00      37.72
           3    2     14.90      0.00     0.00      0.00      0.00      -0.79
                5    -14.90     -0.00     0.00      0.00      0.00       0.79
  ************** END OF LATEST ANALYSIS RESULT **************
   52. FINI
     EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    8
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46: 0 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. US-22 Time History Analysis for Sinusoidal Loading

A space frame structure is subjected to a sinusoidal (dynamic) loading. The commands necessary to describe the sine function are demonstrated in this example. Time History analysis is performed on this model.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-22 Time History Analysis for Sinusoidal Loading.STD` when you install the program.

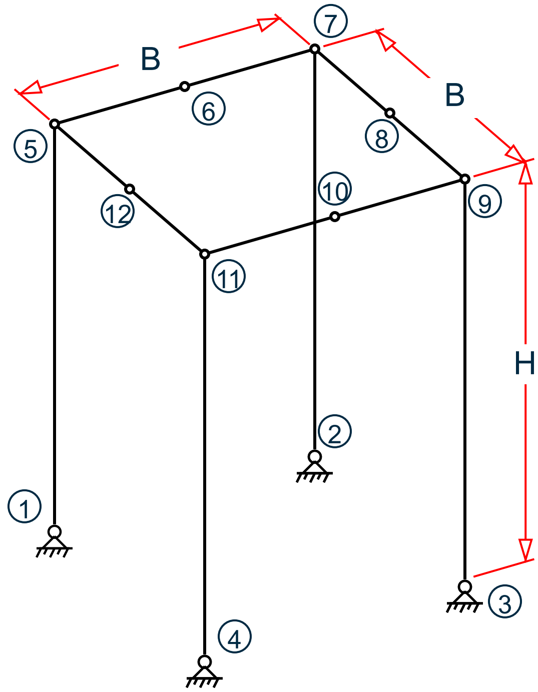


Figure 534: Example Problem No. 22

```
STAAD SPACE
*EXAMPLE FOR HARMONIC LOADING GENERATOR
```

Every STAAD input file has to begin with the word STAAD.

The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

```
UNIT KIP FEET
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0 ; 2 15 0 0 ; 3 15 0 15 ; 4 0 0 15
5 0 20 0 ; 6 7.5 20 0 ; 7 15 20 0 ; 8 15 20 7.5
9 15 20 15 ; 10 7.5 20 15 ; 11 0 20 15
12 0 20 7.5
```

The joint number followed by the X, Y and Z coordinates are specified above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
```

The members are defined by the joints to which they are connected.

```
UNIT INCH
MEMBER PROPERTIES
1 TO 12 PRIS YD 12 ZD 12
```

Members 1 to 12 are defined as PRISmatic sections with width and depth values of 12 inches. The UNIT command is specified to change the units for input from FEET to INCHes.

```
SUPPORTS
1 TO 4 PINNED
```

Joints 1 to 4 are declared to be pinned-supported.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
DEFINE TIME HISTORY
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
```

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the sinusoidal loading are provided.

Each data set is individually identified by the number that follows the `TYPE` command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word `FORCE` that follows the `TYPE 1` command signifies that this data set is for a forcing function. (If you want to specify an earthquake motion, an `ACCELERATION` may be specified.)

The command `FUNCTION SINE` indicates that instead of providing the data set as discrete TIME-FORCE pairs, a sinusoidal function, which describes the variation of force with time, is provided.

The parameters of the sine function, such as `FREQUENCY`, `AMPLITUDE`, and number of `CYCLES` of application are then defined. STAAD internally generates discrete TIME-FORCE pairs of data from the sine function in steps of time defined by the default value (refer to TR.31.4 Definition of Time History Load (on page 2441) for more information). The arrival time value indicates the relative value of time at which the force begins to act upon the structure. The modal damping ratio for all the modes is set to 0.075.

```
LOAD 1 STATIC LOAD CASE
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -1.0
```

The above data describe a static load case.

```
LOAD 2 DYNAMIC LOAD CASE
SELFWEIGHT X 1.0
```

```
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
8 12 FX 4.0
8 12 FY 4.0
8 12 FZ 4.0
TIME LOAD
8 12 FX 1 1
```

This is the second stage of command specification for time history analysis. The two sets of data specified here are:

a. the weights for generation of the mass matrix
b. the application of the time varying loads on the structure.

The weights (from which the masses for the mass matrix are obtained) are specified in the form of selfweight and joint loads.

Following that, the sinusoidal force is applied using the `TIME LOAD` command. The forcing function described by the TYPE 1 load is applied on joints 8 and 12 and it starts to act starting at a time defined by the 1st arrival time number.

```
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI
```

The above commands are self explanatory. The `FINISH` command terminates the STAAD run.

Input File

```
STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
UNIT KIP FEET
JOINT COORDINATES
1 0 0 0 ; 2 15 0 0 ; 3 15 0 15 ; 4 0 0 15
5 0 20 0 ; 6 7.5 20 0 ; 7 15 20 0 ; 8 15 20 7.5
9 15 20 15 ; 10 7.5 20 15 ; 11 0 20 15
12 0 20 7.5
MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
UNIT INCH
MEMBER PROPERTIES
1 TO 12 PRIS YD 12 ZD 12
SUPPORTS
1 TO 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
```

```
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
LOAD 1 STATIC LOAD CASE
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -1.0
LOAD 2 DYNAMIC LOAD CASE
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
8 12 FX 4.0
8 12 FY 4.0
8 12 FZ 4.0
TIME LOAD
8 12 FX 1 1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46: 2                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
INPUT FILE: US-22 Time History Analysis for Sinusoidal Loading.STD
     2. UNIT KIP FEET
     3. JOINT COORDINATES
     4. 1 0 0 0 ; 2 15 0 0 ; 3 15 0 15 ; 4 0 0 15
     5. 5 0 20 0 ; 6 7.5 20 0 ; 7 15 20 0 ; 8 15 20 7.5
     6. 9 15 20 15 ; 10 7.5 20 15 ; 11 0 20 15
     7. 12 0 20 7.5
     8. MEMBER INCIDENCES
     9. 1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
    10. 7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
    11. UNIT INCH
    12. MEMBER PROPERTIES
    13. 1 TO 12 PRIS YD 12 ZD 12
    14. SUPPORTS
    15. 1 TO 4 PINNED
    16. DEFINE MATERIAL START
    17. ISOTROPIC CONCRETE
    18. E 3150
```

```
    19. POISSON 0.17
    20. DENSITY 8.68E-005
    21. ALPHA 5E-006
    22. DAMP 0.05
    23. G 1346.15
    24. TYPE CONCRETE
    25. STRENGTH FCU 4
    26. END DEFINE MATERIAL
    27. CONSTANTS
    28. MATERIAL CONCRETE ALL
    29. DEFINE TIME HISTORY
    30. TYPE 1 FORCE
    31. * FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
    32. FUNCTION SINE
    33. AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100
      EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    2
   FOR SEQUENTIAL HARMONIC FORCING CURVE NUMBER=            1
   NUMBER OF POINTS IN DIGITIZED HARMONIC FUNCTION=       1201
   NUMBER OF POINTS PER QUARTER CYCLE OF HARMONIC FUNCTION=  3
   FORCE STEP DELTA TIME PER POINT                  1.38889E-03
   ENDING TIME FOR THIS DIGITIZED HARMONIC FUNCTION  1.66667E+00
    34. *
    35. ARRIVAL TIMES
    36. 0.0
    37. DAMPING 0.075
    38. LOAD 1 STATIC LOAD CASE
    39. MEMBER LOAD
    40. 5 6 7 8 9 10 11 12 UNI GY -1.0
    41. LOAD 2 DYNAMIC LOAD CASE
    42. SELFWEIGHT X 1.0
    43. SELFWEIGHT Y 1.0
    44. SELFWEIGHT Z 1.0
    45. JOINT LOAD
    46. 8 12 FX 4.0
    47. 8 12 FY 4.0
    48. 8 12 FZ 4.0
    49. TIME LOAD
    50. 8 12 FX 1 1
    51. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         12  NUMBER OF MEMBERS      12
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      60
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
            THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
      EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    3
 EIGEN METHOD   : SUBSPACE
 -------------------------
 NUMBER OF MODES REQUESTED             =     6
 NUMBER OF EXISTING MASSES IN THE MODEL =    24
 NUMBER OF MODES THAT WILL BE USED      =     6
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    4
```

```
              CALCULATED FREQUENCIES FOR LOAD CASE       2
      MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
        1                      1.202                  0.83191
        2                      1.204                  0.83057
        3                      1.451                  0.68908
        4                      7.559                  0.13229
        5                     11.073                  0.09031
        6                     11.670                  0.08569
           MODAL WEIGHT (MODAL MASS TIMES g) IN KIP          GENERALIZED
     MODE           X             Y             Z             WEIGHT
        1      2.299868E+01  5.213994E-29  2.154110E-25    2.280506E+01
        2      2.148403E-25  9.984362E-31  2.299902E+01    2.293083E+01
        3      2.180358E-21  1.626557E-31  2.281548E-21    3.300665E+01
        4      1.122613E-24  1.446952E-29  2.732241E-24    2.026121E+01
        5      4.006222E-23  5.345261E-06  4.306780E-20    1.127253E+01
        6      1.341889E-20  1.076830E+01  1.183963E-23    1.025517E+01
MASS PARTICIPATION FACTORS
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
          MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
            1   100.00   0.00   0.00   99.998    0.000    0.000
            2     0.00   0.00 100.00   99.998    0.000  100.000
            3     0.00   0.00   0.00   99.998    0.000  100.000
            4     0.00   0.00   0.00   99.998    0.000  100.000
            5     0.00   0.00   0.00   99.998    0.000  100.000
            6     0.00  46.82   0.00   99.998   46.821  100.000
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    5
   A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
   MODE       DAMPING
      1      0.07500000
      2      0.07500000
      3      0.07500000
      4      0.07500000
      5      0.07500000
      6      0.07500000
  TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
  NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =    6
  TIME DURATION OF TIME HISTORY ANALYSIS =      1.667 SECONDS
  NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =     1200
   52. PRINT ANALYSIS RESULTS
 BASE SHEAR UNITS ARE -- KIP  INCH
 MAXIMUM BASE SHEAR  X=  -2.228746E-01  Y=   1.490116E-08  Z=   2.273737E-13
     AT TIMES               0.194444          1.519444           0.072222
 ANALYSIS RESULTS
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    6
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   0.00000   0.00000  -0.01045   0.00000   0.01045
          2    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00037
     2    1    0.00000   0.00000   0.00000  -0.01045   0.00000  -0.01045
          2    0.00000   0.00000   0.00000  -0.00000   0.00000  -0.00037
     3    1    0.00000   0.00000   0.00000   0.01045   0.00000  -0.01045
          2    0.00000   0.00000   0.00000   0.00000  -0.00000  -0.00037
     4    1    0.00000   0.00000   0.00000   0.01045   0.00000   0.01045
          2    0.00000   0.00000   0.00000  -0.00000  -0.00000  -0.00037
     5    1    0.00118  -0.09524   0.00118   0.02103   0.00000  -0.02103
          2    0.06536   0.00008   0.00000  -0.00000   0.00000  -0.00008
```

```
     6    1    0.00000  -1.56841   0.00118   0.02103   0.00000   0.00000
          2    0.06537   0.00000   0.00000   0.00000  -0.00000   0.00004
     7    1   -0.00118  -0.09524   0.00118   0.02103   0.00000   0.02103
          2    0.06536  -0.00008  -0.00000   0.00000   0.00000  -0.00008
     8    1   -0.00118  -1.56841   0.00000   0.00000   0.00000   0.02103
          2    0.06587  -0.00008   0.00000   0.00000   0.00000  -0.00008
     9    1   -0.00118  -0.09524  -0.00118  -0.02103   0.00000   0.02103
          2    0.06536  -0.00008   0.00000  -0.00000  -0.00000  -0.00008
    10    1    0.00000  -1.56841  -0.00118  -0.02103   0.00000   0.00000
          2    0.06537   0.00000   0.00000   0.00000   0.00000   0.00004
    11    1    0.00118  -0.09524  -0.00118  -0.02103   0.00000  -0.02103
          2    0.06536   0.00008  -0.00000   0.00000  -0.00000  -0.00008
    12    1    0.00118  -1.56841   0.00000   0.00000   0.00000  -0.02103
          2    0.06587   0.00008   0.00000   0.00000   0.00000  -0.00008
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    7
  SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      5.95    180.00      5.95      0.00      0.00      0.00
          2     -0.06     -0.15     -0.00      0.00      0.00      0.00
     2    1     -5.95    180.00      5.95      0.00      0.00      0.00
          2     -0.06      0.15      0.00      0.00      0.00      0.00
     3    1     -5.95    180.00     -5.95      0.00      0.00      0.00
          2     -0.06      0.15     -0.00      0.00      0.00      0.00
     4    1      5.95    180.00     -5.95      0.00      0.00      0.00
          2     -0.06     -0.15      0.00      0.00      0.00      0.00
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    8
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KIP  INCH     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1    180.00     -5.95     5.95      0.00      0.00      -0.00
                5   -180.00      5.95    -5.95      0.00  -1428.10   -1428.10
          2     1     -0.15      0.06    -0.00      0.00      0.00       0.00
                5      0.15     -0.06     0.00      0.00      0.00      13.37
     2    1     2    180.00      5.95     5.95      0.00      0.00       0.00
                7   -180.00     -5.95    -5.95      0.00  -1428.10    1428.10
          2     2      0.15      0.06     0.00      0.00      0.00       0.00
                7     -0.15     -0.06    -0.00      0.00     -0.00      13.37
     3    1     3    180.00      5.95    -5.95      0.00     -0.00       0.00
                9   -180.00     -5.95     5.95      0.00   1428.10    1428.10
          2     3      0.15      0.06    -0.00      0.00      0.00       0.00
                9     -0.15     -0.06     0.00      0.00      0.00      13.37
     4    1     4    180.00     -5.95    -5.95      0.00      0.00       0.00
               11   -180.00      5.95     5.95      0.00   1428.10   -1428.10
          2     4     -0.15      0.06     0.00      0.00      0.00       0.00
               11      0.15     -0.06    -0.00      0.00     -0.00      13.37
     5    1     5      5.95     90.00    -0.00     -0.00      0.00    1428.10
                6     -5.95     -0.00     0.00      0.00     -0.00    2621.90
          2     5     -0.01     -0.15    -0.01     -0.00      0.84     -13.37
                6      0.01      0.15     0.01      0.00      0.00      -0.00
     6    1     6      5.95     -0.00    -0.00      0.00      0.00   -2621.90
                7     -5.95     90.00     0.00     -0.00      0.00   -1428.10
          2     6      0.01     -0.15    -0.01     -0.00      0.00       0.00
                7     -0.01      0.15     0.01      0.00      0.84     -13.37
     7    1     7      5.95     90.00     0.00     -0.00     -0.00    1428.10
                8     -5.95     -0.00    -0.00      0.00      0.00    2621.90
          2     7     -0.01      0.00     0.02      0.00     -0.84       0.00
```

```
              8      0.01     -0.00    -0.02     -0.00     -1.41       0.00
    8    1    8      5.95      0.00     0.00      0.00     -0.00   -2621.90
              9     -5.95     90.00    -0.00     -0.00     -0.00   -1428.10
         2    8     -0.01     -0.00    -0.02      0.00      1.41      -0.00
              9      0.01      0.00     0.02     -0.00      0.84      -0.00
    9    1    9      5.95     90.00    -0.00     -0.00      0.00    1428.10
             10     -5.95     -0.00     0.00      0.00      0.00    2621.90
    EXAMPLE FOR HARMONIC LOADING GENERATOR                 -- PAGE NO.    9
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KIP  INCH     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
         2    9      0.01      0.15     0.01      0.00     -0.84      13.37
             10     -0.01     -0.15    -0.01     -0.00      0.00      -0.00
   10    1   10      5.95      0.00    -0.00      0.00      0.00   -2621.90
             11     -5.95     90.00     0.00     -0.00      0.00   -1428.10
         2   10     -0.01      0.15     0.01      0.00      0.00       0.00
             11      0.01     -0.15    -0.01     -0.00     -0.84      13.37
   11    1   11      5.95     90.00     0.00     -0.00     -0.00    1428.10
             12     -5.95      0.00    -0.00      0.00      0.00    2621.90
         2   11      0.01     -0.00    -0.02      0.00      0.84      -0.00
             12     -0.01      0.00     0.02     -0.00      1.41      -0.00
   12    1   12      5.95      0.00     0.00      0.00     -0.00   -2621.90
              5     -5.95     90.00    -0.00     -0.00     -0.00   -1428.10
         2   12      0.01      0.00     0.02      0.00     -1.41       0.00
              5     -0.01     -0.00    -0.02     -0.00     -0.84       0.00
 ************** END OF LATEST ANALYSIS RESULT **************
  53. FINI
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:46: 3 ****
    EXAMPLE FOR HARMONIC LOADING GENERATOR                 -- PAGE NO.   10
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

EX. US-23 Spring Support Generation for a Slab on Grade

This example illustrates the usage of commands necessary to automatically generate spring supports for a slab on grade. The slab is subjected to pressure loading and analysis of the structure is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-23 Spring Support Generation for a Slab on Grade.STD` when you install the program.

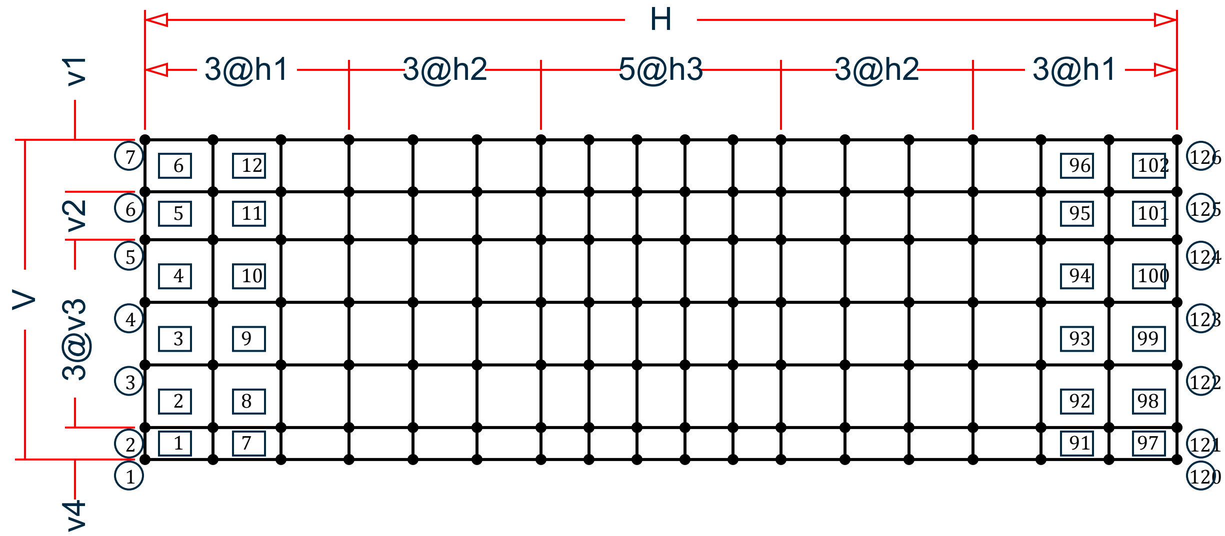


Figure 535: Example Problem No. 23

where:

H = 129', h1 = 8'-6", h2 = 8', h3 = 6'
V = 40', v1 = 6'-6", v2 = 6', v3 = 7'-10", v4 = 4'

```
STAAD SPACE SLAB ON GRADE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The remainder of the words form a title to identify this project.

```
UNIT FEET KIP
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
```

For joints 1 through 7, the joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these seven joints by 8.5 feet, three times REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and that data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first six elements is then used to generate the incidences of 96 (= 16 x 6) more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 5.5
```

The thickness of elements 1 to 102 is specified as 5.5 inches following the command ELEMENT PROPERTIES.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2916.7
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUB 10.0
```

The above command is used to instruct STAAD to generate supports with springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade modulus of the soil is specified as 10 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the FY degree of freedom at the joint. See TR. 27.3 Automatic Spring Support Generator for Foundations (on page 2319) .

```
PRINT SUPP INFO
```

This command will enable us to obtain the details of the support conditions which were generated using the earlier commands.

```
LOAD 1 WEIGHT OF MAT & EARTH
ELEMENT LOAD
1 TO 102 PR GY -1.55
```

The above data describe a static load case. A pressure load of 1.55 kip/sq.ft acting in the negative global Y direction is applied on all the 102 elements.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
```

```
    48 55 76 83 FY -528.3
    92 93 FY -205.0
    99 100 FY -410.0
    103 FY -487.0
    104 FY -974.0
    113 114 FY -109.0
    120 121 FY -217.0
    124 FY -273.3
    125 FY -546.6
```

Load case 2 consists of several joint loads acting in the negative global Y direction.

```
    LOADING COMBINATION 101 TOTAL LOAD
    1 1. 2 1.
```

A load combination case, identified with load case number 101, is specified above. It instructs STAAD to factor loads 1 and 2 by a value of 1.0 and then algebraically add the results.

```
    PERFORM ANALYSIS
```

The analysis is initiated using the above command.

```
    LOAD LIST 101
    PRINT JOINT DISPLACEMENTS LIST 33 56
    PRINT ELEMENT STRESSES LIST 34 67
```

Joint displacements for joints 33 and 56, and element stresses for elements 34 and 67, for load case 101, is obtained with the help of the above commands.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE SLAB ON GRADE
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 5.5
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2916.7
POISSON 0.12
DENSITY 8.68e-005
```

```
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT FEET
SUPPORTS
1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 10.0
PRINT SUPP INFO
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.55
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2   FY -217.
8 9   FY -109.
5     FY -308.7
6     FY -617.4
22 23 FY -410.
29 30 FY -205.
26    FY -542.7
27    FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82    FY -264.2
48 55 76 83 FY -528.3
92 93       FY -205.0
99 100      FY -410.0
103         FY -487.0
104         FY -974.0
113 114     FY -109.0
120 121     FY -217.0
124         FY -273.3
125         FY -546.6
LOADING COMBINATION 101 TOTAL LOAD
1 1. 2 1.
PERFORM ANALYSIS
LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
            ****************************************************
            *                                                  *
            *           STAAD.Pro CONNECT Edition              *
            *           Version  22.10.00.***                  *
            *           Proprietary Program of                 *
            *           Bentley Systems, Inc.                  *
            *           Date=    MAR 24, 2022                  *
            *           Time=     9:46: 5                      *
            *                                                  *
            *  Licensed to: Bentley Systems Inc                *
            ****************************************************
```

```
     1. STAAD SPACE SLAB ON GRADE
INPUT FILE: US-23 Spring Support Generation for a Slab on Grade.STD
     2. UNIT FEET KIP
     3. JOINT COORDINATES
     4. 1 0.0 0.0 40.0
     5. 2 0.0 0.0 36.0
     6. 3 0.0 0.0 28.167
     7. 4 0.0 0.0 20.333
     8. 5 0.0 0.0 12.5
     9. 6 0.0 0.0  6.5
    10. 7 0.0 0.0  0.0
    11. REPEAT ALL 3 8.5 0.0 0.0
    12. REPEAT 3 8.0 0.0 0.0
    13. REPEAT 5 6.0 0.0 0.0
    14. REPEAT 3 8.0 0.0 0.0
    15. REPEAT 3 8.5 0.0 0.0
    16. ELEMENT INCIDENCES
    17. 1 1 8 9 2 TO 6
    18. REPEAT 16 6 7
    19. UNIT INCH
    20. ELEMENT PROPERTIES
    21. 1 TO 102 TH 5.5
    22. DEFINE MATERIAL START
    23. ISOTROPIC CONCRETE
    24. E 2916.7
    25. POISSON 0.12
    26. DENSITY 8.68E-005
    27. ALPHA 5E-006
    28. DAMP 0.05
    29. G 1346.15
    30. TYPE CONCRETE
    31. STRENGTH FCU 4
    32. END DEFINE MATERIAL
    33. CONSTANTS
    34. MATERIAL CONCRETE ALL
    35. UNIT FEET
    36. SUPPORTS
    37. 1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 10.0
    38. PRINT SUPP INFO
      SLAB ON GRADE                                            -- PAGE NO.    2
  SUPP     INFO
   SUPPORT INFORMATION (1=FIXED, 0=RELEASED)
   -------------------
   UNITS FOR SPRING CONSTANTS ARE KIP  FEET DEGREES
   JOINT   FORCE-X/   FORCE-Y/   FORCE-Z/   MOM-X/   MOM-Y/   MOM-Z/
             KFX        KFY        KFZ       KMX      KMY      KMZ
      1       1          0          1         0        1        0
                 0.0       85.0        0.0      0.0      0.0      0.0
      2       1          0          1         0        1        0
                 0.0      251.5        0.0      0.0      0.0      0.0
      3       1          0          1         0        1        0
                 0.0      332.9        0.0      0.0      0.0      0.0
      4       1          0          1         0        1        0
                 0.0      332.9        0.0      0.0      0.0      0.0
      5       1          0          1         0        1        0
                 0.0      294.0        0.0      0.0      0.0      0.0
      6       1          0          1         0        1        0
                 0.0      265.6        0.0      0.0      0.0      0.0
```

```
  7       1          0          1         0        1        0
              0.0       138.1        0.0      0.0      0.0      0.0
  8       1          0          1         0        1        0
              0.0       170.0        0.0      0.0      0.0      0.0
  9       1          0          1         0        1        0
              0.0       502.9        0.0      0.0      0.0      0.0
 10       1          0          1         0        1        0
              0.0       665.8        0.0      0.0      0.0      0.0
 11       1          0          1         0        1        0
              0.0       665.8        0.0      0.0      0.0      0.0
 12       1          0          1         0        1        0
              0.0       587.9        0.0      0.0      0.0      0.0
 13       1          0          1         0        1        0
              0.0       531.2        0.0      0.0      0.0      0.0
 14       1          0          1         0        1        0
              0.0       276.2        0.0      0.0      0.0      0.0
 15       1          0          1         0        1        0
              0.0       170.0        0.0      0.0      0.0      0.0
 16       1          0          1         0        1        0
              0.0       502.9        0.0      0.0      0.0      0.0
 17       1          0          1         0        1        0
              0.0       665.8        0.0      0.0      0.0      0.0
 18       1          0          1         0        1        0
              0.0       665.8        0.0      0.0      0.0      0.0
 19       1          0          1         0        1        0
              0.0       587.9        0.0      0.0      0.0      0.0
 20       1          0          1         0        1        0
              0.0       531.2        0.0      0.0      0.0      0.0
 21       1          0          1         0        1        0
              0.0       276.2        0.0      0.0      0.0      0.0
 22       1          0          1         0        1        0
              0.0       165.0        0.0      0.0      0.0      0.0
  SLAB ON GRADE                                                -- PAGE NO.    3
 23       1          0          1         0        1        0
              0.0       488.1        0.0      0.0      0.0      0.0
 24       1          0          1         0        1        0
              0.0       646.3        0.0      0.0      0.0      0.0
 25       1          0          1         0        1        0
              0.0       646.3        0.0      0.0      0.0      0.0
 26       1          0          1         0        1        0
              0.0       570.6        0.0      0.0      0.0      0.0
 27       1          0          1         0        1        0
              0.0       515.6        0.0      0.0      0.0      0.0
 28       1          0          1         0        1        0
              0.0       268.1        0.0      0.0      0.0      0.0
 29       1          0          1         0        1        0
              0.0       160.0        0.0      0.0      0.0      0.0
 30       1          0          1         0        1        0
              0.0       473.3        0.0      0.0      0.0      0.0
 31       1          0          1         0        1        0
              0.0       626.7        0.0      0.0      0.0      0.0
 32       1          0          1         0        1        0
              0.0       626.7        0.0      0.0      0.0      0.0
 33       1          0          1         0        1        0
              0.0       553.3        0.0      0.0      0.0      0.0
 34       1          0          1         0        1        0
              0.0       500.0        0.0      0.0      0.0      0.0
 35       1          0          1         0        1        0
```

```
                 0.0      260.0        0.0      0.0      0.0      0.0
     36       1         0          1         0        1        0
                 0.0      160.0        0.0      0.0      0.0      0.0
     37       1         0          1         0        1        0
                 0.0      473.3        0.0      0.0      0.0      0.0
     38       1         0          1         0        1        0
                 0.0      626.7        0.0      0.0      0.0      0.0
     39       1         0          1         0        1        0
                 0.0      626.7        0.0      0.0      0.0      0.0
     40       1         0          1         0        1        0
                 0.0      553.3        0.0      0.0      0.0      0.0
     41       1         0          1         0        1        0
                 0.0      500.0        0.0      0.0      0.0      0.0
     42       1         0          1         0        1        0
                 0.0      260.0        0.0      0.0      0.0      0.0
     43       1         0          1         0        1        0
                 0.0      140.0        0.0      0.0      0.0      0.0
     44       1         0          1         0        1        0
                 0.0      414.2        0.0      0.0      0.0      0.0
     45       1         0          1         0        1        0
                 0.0      548.3        0.0      0.0      0.0      0.0
     46       1         0          1         0        1        0
                 0.0      548.3        0.0      0.0      0.0      0.0
     47       1         0          1         0        1        0
                 0.0      484.2        0.0      0.0      0.0      0.0
     48       1         0          1         0        1        0
                 0.0      437.5        0.0      0.0      0.0      0.0
     49       1         0          1         0        1        0
                 0.0      227.5        0.0      0.0      0.0      0.0
     50       1         0          1         0        1        0
                 0.0      120.0        0.0      0.0      0.0      0.0
      SLAB ON GRADE                                            -- PAGE NO.    4
     51       1         0          1         0        1        0
                 0.0      355.0        0.0      0.0      0.0      0.0
     52       1         0          1         0        1        0
                 0.0      470.0        0.0      0.0      0.0      0.0
     53       1         0          1         0        1        0
                 0.0      470.0        0.0      0.0      0.0      0.0
     54       1         0          1         0        1        0
                 0.0      415.0        0.0      0.0      0.0      0.0
     55       1         0          1         0        1        0
                 0.0      375.0        0.0      0.0      0.0      0.0
     56       1         0          1         0        1        0
                 0.0      195.0        0.0      0.0      0.0      0.0
     57       1         0          1         0        1        0
                 0.0      120.0        0.0      0.0      0.0      0.0
     58       1         0          1         0        1        0
                 0.0      355.0        0.0      0.0      0.0      0.0
     59       1         0          1         0        1        0
                 0.0      470.0        0.0      0.0      0.0      0.0
     60       1         0          1         0        1        0
                 0.0      470.0        0.0      0.0      0.0      0.0
     61       1         0          1         0        1        0
                 0.0      415.0        0.0      0.0      0.0      0.0
     62       1         0          1         0        1        0
                 0.0      375.0        0.0      0.0      0.0      0.0
     63       1         0          1         0        1        0
                 0.0      195.0        0.0      0.0      0.0      0.0
```

```
     64       1          0          1         0        1        0
                 0.0       120.0        0.0      0.0      0.0      0.0
     65       1          0          1         0        1        0
                 0.0       355.0        0.0      0.0      0.0      0.0
     66       1          0          1         0        1        0
                 0.0       470.0        0.0      0.0      0.0      0.0
     67       1          0          1         0        1        0
                 0.0       470.0        0.0      0.0      0.0      0.0
     68       1          0          1         0        1        0
                 0.0       415.0        0.0      0.0      0.0      0.0
     69       1          0          1         0        1        0
                 0.0       375.0        0.0      0.0      0.0      0.0
     70       1          0          1         0        1        0
                 0.0       195.0        0.0      0.0      0.0      0.0
     71       1          0          1         0        1        0
                 0.0       120.0        0.0      0.0      0.0      0.0
     72       1          0          1         0        1        0
                 0.0       355.0        0.0      0.0      0.0      0.0
     73       1          0          1         0        1        0
                 0.0       470.0        0.0      0.0      0.0      0.0
     74       1          0          1         0        1        0
                 0.0       470.0        0.0      0.0      0.0      0.0
     75       1          0          1         0        1        0
                 0.0       415.0        0.0      0.0      0.0      0.0
     76       1          0          1         0        1        0
                 0.0       375.0        0.0      0.0      0.0      0.0
     77       1          0          1         0        1        0
                 0.0       195.0        0.0      0.0      0.0      0.0
     78       1          0          1         0        1        0
                 0.0       140.0        0.0      0.0      0.0      0.0
      SLAB ON GRADE                                                  -- PAGE NO.    5
     79       1          0          1         0        1        0
                 0.0       414.2        0.0      0.0      0.0      0.0
     80       1          0          1         0        1        0
                 0.0       548.3        0.0      0.0      0.0      0.0
     81       1          0          1         0        1        0
                 0.0       548.3        0.0      0.0      0.0      0.0
     82       1          0          1         0        1        0
                 0.0       484.2        0.0      0.0      0.0      0.0
     83       1          0          1         0        1        0
                 0.0       437.5        0.0      0.0      0.0      0.0
     84       1          0          1         0        1        0
                 0.0       227.5        0.0      0.0      0.0      0.0
     85       1          0          1         0        1        0
                 0.0       160.0        0.0      0.0      0.0      0.0
     86       1          0          1         0        1        0
                 0.0       473.3        0.0      0.0      0.0      0.0
     87       1          0          1         0        1        0
                 0.0       626.7        0.0      0.0      0.0      0.0
     88       1          0          1         0        1        0
                 0.0       626.7        0.0      0.0      0.0      0.0
     89       1          0          1         0        1        0
                 0.0       553.3        0.0      0.0      0.0      0.0
     90       1          0          1         0        1        0
                 0.0       500.0        0.0      0.0      0.0      0.0
     91       1          0          1         0        1        0
                 0.0       260.0        0.0      0.0      0.0      0.0
     92       1          0          1         0        1        0
```

```
                 0.0     160.0        0.0      0.0      0.0      0.0
   93        1          0          1         0        1        0
                 0.0     473.3        0.0      0.0      0.0      0.0
   94        1          0          1         0        1        0
                 0.0     626.7        0.0      0.0      0.0      0.0
   95        1          0          1         0        1        0
                 0.0     626.7        0.0      0.0      0.0      0.0
   96        1          0          1         0        1        0
                 0.0     553.3        0.0      0.0      0.0      0.0
   97        1          0          1         0        1        0
                 0.0     500.0        0.0      0.0      0.0      0.0
   98        1          0          1         0        1        0
                 0.0     260.0        0.0      0.0      0.0      0.0
   99        1          0          1         0        1        0
                 0.0     165.0        0.0      0.0      0.0      0.0
  100        1          0          1         0        1        0
                 0.0     488.1        0.0      0.0      0.0      0.0
  101        1          0          1         0        1        0
                 0.0     646.3        0.0      0.0      0.0      0.0
  102        1          0          1         0        1        0
                 0.0     646.3        0.0      0.0      0.0      0.0
  103        1          0          1         0        1        0
                 0.0     570.6        0.0      0.0      0.0      0.0
  104        1          0          1         0        1        0
                 0.0     515.6        0.0      0.0      0.0      0.0
  105        1          0          1         0        1        0
                 0.0     268.1        0.0      0.0      0.0      0.0
  106        1          0          1         0        1        0
                 0.0     170.0        0.0      0.0      0.0      0.0
   SLAB ON GRADE                                           -- PAGE NO.    6
  107        1          0          1         0        1        0
                 0.0     502.9        0.0      0.0      0.0      0.0
  108        1          0          1         0        1        0
                 0.0     665.8        0.0      0.0      0.0      0.0
  109        1          0          1         0        1        0
                 0.0     665.8        0.0      0.0      0.0      0.0
  110        1          0          1         0        1        0
                 0.0     587.9        0.0      0.0      0.0      0.0
  111        1          0          1         0        1        0
                 0.0     531.2        0.0      0.0      0.0      0.0
  112        1          0          1         0        1        0
                 0.0     276.2        0.0      0.0      0.0      0.0
  113        1          0          1         0        1        0
                 0.0     170.0        0.0      0.0      0.0      0.0
  114        1          0          1         0        1        0
                 0.0     502.9        0.0      0.0      0.0      0.0
  115        1          0          1         0        1        0
                 0.0     665.8        0.0      0.0      0.0      0.0
  116        1          0          1         0        1        0
                 0.0     665.8        0.0      0.0      0.0      0.0
  117        1          0          1         0        1        0
                 0.0     587.9        0.0      0.0      0.0      0.0
  118        1          0          1         0        1        0
                 0.0     531.2        0.0      0.0      0.0      0.0
  119        1          0          1         0        1        0
                 0.0     276.2        0.0      0.0      0.0      0.0
  120        1          0          1         0        1        0
                 0.0      85.0        0.0      0.0      0.0      0.0
```

```
    121       1          0          1         0        1        0
                 0.0      251.5       0.0      0.0      0.0      0.0
    122       1          0          1         0        1        0
                 0.0      332.9       0.0      0.0      0.0      0.0
    123       1          0          1         0        1        0
                 0.0      332.9       0.0      0.0      0.0      0.0
    124       1          0          1         0        1        0
                 0.0      294.0       0.0      0.0      0.0      0.0
    125       1          0          1         0        1        0
                 0.0      265.6       0.0      0.0      0.0      0.0
    126       1          0          1         0        1        0
                 0.0      138.1       0.0      0.0      0.0      0.0
   ************ END OF DATA FROM INTERNAL STORAGE ************
    39. LOAD 1 'WEIGHT OF MAT & EARTH'
    40. ELEMENT LOAD
    41. 1 TO 102 PR GY -1.55
    42. LOAD 2 'COLUMN LOAD-DL+LL'
    43. JOINT LOADS
    44. 1 2   FY -217.
    45. 8 9   FY -109.
    46. 5     FY -308.7
    47. 6     FY -617.4
      SLAB ON GRADE                                          -- PAGE NO.    7
    48. 22 23 FY -410.
    49. 29 30 FY -205.
    50. 26    FY -542.7
    51. 27    FY -1085.4
    52. 43 44 50 51 71 72 78 79 FY -307.5
    53. 47 54 82    FY -264.2
    54. 48 55 76 83 FY -528.3
    55. 92 93       FY -205.0
    56. 99 100      FY -410.0
    57. 103         FY -487.0
    58. 104         FY -974.0
    59. 113 114     FY -109.0
    60. 120 121     FY -217.0
    61. 124         FY -273.3
    62. 125         FY -546.6
    63. LOADING COMBINATION 101 TOTAL LOAD
    64. 1 1. 2 1.
    65. PERFORM ANALYSIS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
     NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     378
   TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
   *** NOTE: CAPACITY FOR MAXIMUM #    254 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
    66. LOAD LIST 101
    67. PRINT JOINT DISPLACEMENTS LIST 33 56
  JOINT    DISPLACE LIST     33
      SLAB ON GRADE                                          -- PAGE NO.    8
   JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
```

```
    33  101    0.00000  -4.73840   0.00000  -0.03240   0.00000   0.06201
    56  101    0.00000  -5.63562   0.00000   0.07103   0.00000   0.03210
  ************** END OF LATEST ANALYSIS RESULT **************
   68. PRINT ELEMENT STRESSES LIST 34 67
 ELEMENT  STRESSES LIST     34
     SLAB ON GRADE                                                    -- PAGE NO.    9
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
  ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
    34    101        -6.17       -8.28        0.84        4.63       10.89
                    552.32      552.32        0.00        0.00        0.00
                    631.36      631.36
      TOP : SMAX=     393.78 SMIN=    -237.58 TMAX=     315.68 ANGLE= 49.9
      BOTT: SMAX=     237.58 SMIN=    -393.78 TMAX=     315.68 ANGLE=-40.1
    67    101        72.18        7.72       15.02        9.75       22.08
                   1155.35     1155.35        0.00        0.00        0.00
                   1269.99     1269.99
      TOP : SMAX=     988.79 SMIN=    -281.20 TMAX=     635.00 ANGLE= 41.6
      BOTT: SMAX=     281.20 SMIN=    -988.79 TMAX=     635.00 ANGLE=-48.4
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         9.887888E+02 -9.887888E+02  6.349952E+02  1.155347E+03  1.269990E+03
PLATE NO.      67            67            67            67            67
CASE  NO.     101           101           101           101           101
  ********************END OF ELEMENT FORCES********************
   69. FINISH
     SLAB ON GRADE                                                    -- PAGE NO.   10
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46: 6 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *M. To assign a foundation support* (on page 824)
- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2319)
- *Create Support dialog* (on page 2815)

EX. US-24 Analysis of a Concrete Block Using Solid Elements

This is an example of the analysis of a structure modeled using solid finite elements. This example also illustrates the method for applying an enforced displacement on the structure.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-24 Analysis of a Concrete Block Using Solid Elements.STD` when you install the program.

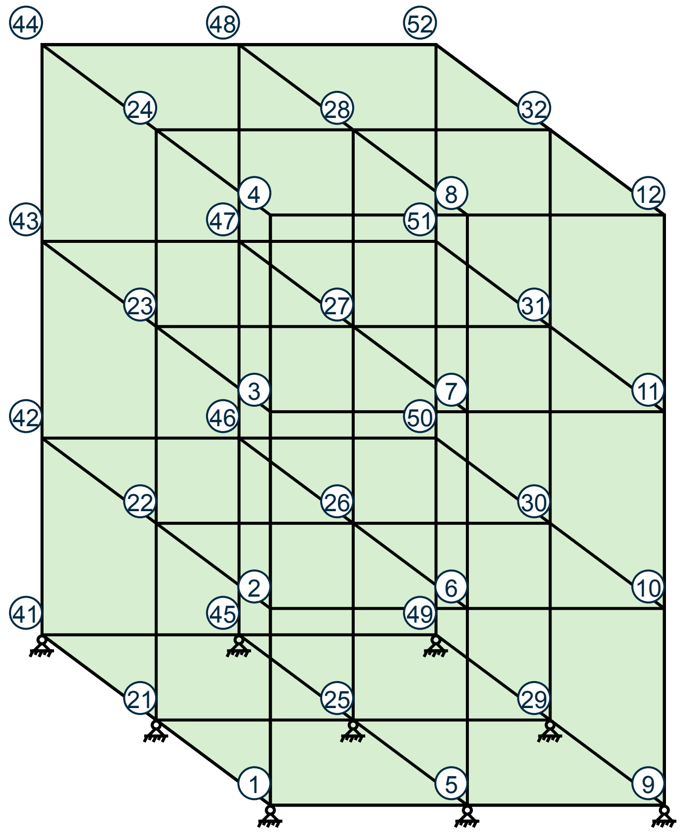


Figure 536: Example Problem No. 24

```
STAAD SPACE
*EXAMPLE PROBLEM USING SOLID ELEMENTS
```

Every STAAD input file has to begin with the word `STAAD`. The word `SPACE` signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

```
UNIT KNS MET
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
  1  0.0  0.0  2.0   4  0.0  3.0  2.0
  5  1.0  0.0  2.0   8  1.0  3.0  2.0
  9  2.0  0.0  2.0  12  2.0  3.0  2.0
 21  0.0  0.0  1.0  24  0.0  3.0  1.0
 25  1.0  0.0  1.0  28  1.0  3.0  1.0
 29  2.0  0.0  1.0  32  2.0  3.0  1.0
 41  0.0  0.0  0.0  44  0.0  3.0  0.0
 45  1.0  0.0  0.0  48  1.0  3.0  0.0
 49  2.0  0.0  0.0  52  2.0  3.0  0.0
```

The joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of some of those nodes are generated utilizing the fact that they are equally spaced between the extremities.

```
ELEMENT INCIDENCES SOLID
  1    1   5   6   2  21  25  26  22   TO   3
  4   21  25  26  22  41  45  46  42   TO   6  1 1
  7    5   9  10   6  25  29  30  26   TO   9  1 1
 10   25  29  30  26  45  49  50  46   TO  12  1 1
```

The incidences of solid elements are defined above. The word `SOLID` is used to signify that these are 8-node solid elements as opposed to 3-noded or 4-noded plate elements. Each line contains the data for generating 3 elements. For example, element number 1 is first defined by all of its 8 nodes. Then, increments of 1 to the joint number and 1 to the element number (the defaults) are used for generating incidences for elements 2 and 3. Similarly, incidences of elements 4, 7 and 10 are defined while those of 5, 6, 8, 9, 11 and 12 are generated.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METERL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
PRINT ELEMENT INFO SOLID LIST 1 TO 5
```

This command will enable us to obtain, in a tabular form, the details of the incidences and material property values of elements 1 to 5.

```
SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED
```

The above lines contain the data for supports for the model. The ENFORCED support condition is used to declare a point at which an enforced displacement load is applied later (see load case 3).

```
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
```

The above data describe a static load case. It consists of selfweight loading and a joint load, both in the negative global Y direction.

```
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0
```

Load case 2 consists of several joint loads acting in the positive global X direction.

```
LOAD 3
SUPPORT DISPLACEMENT
9 FX 0.0011
```

Load case 3 consists of an enforced displacement along the global X direction at node 9. The displacement in the other enforced support degrees of freedom will default to zero.

```
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
```

In Load case 4, a pressure load of 500 pounds/sq.ft is applied on Face # 4 of solid elements 3, 6, 9 and 12. Face 4 is defined as shown in the following table :

Face Number	Surface Joints			
	f_1	f_2	f_3	f_4
1 front	Jt 1	Jt 4	Jt 3	Jt 2
2 bottom	Jt 1	Jt 2	Jt 6	Jt 5
3 left	Jt 1	Jt 5	Jt 8	Jt 4
4 top	Jt 4	Jt 8	Jt 7	Jt 3
5 right	Jt 2	Jt 3	Jt 7	Jt 6
6 back	Jt 5	Jt 6	Jt 7	Jt 8

The above table, and other details of this type of loading can be found in TR.32.3.2 Element Load Specification - Solids (on page 2472) .

```
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
```

Load case 5 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We want the program to analyze the structure for loads from cases 1 through 4 acting simultaneously. In other words, the above instruction is the same as the following:

```
LOAD 5
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
2 TO 4 22 TO 24 42 TO 44 FX 100.0
SUPPORT DISPLACEMENT
9 FX .0011
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
LOAD COMB 10
1 1.0 2 1.0
```

Load case 10 is a combination load case, which combines the effects of cases 1 & 2. While the syntax of this might look very similar to that of the `REPEAT LOAD` case shown in case 5, there is a fundamental difference. In a `REPEAT LOAD` case, the program computes the displacements by multiplying the inverted stiffness matrix by the load vector built for the `REPEAT LOAD` case. But in solving load combination cases, the program merely calculates the end results (displacements, forces, reactions) by gathering up the corresponding values from the individual components of the combination case, factoring them, and then algebraically summing them up. This difference in approach is quite important in that non-linear problems such as `PDELTA ANALYSIS`, `MEMBER TENSION`, and `MEMBER COMPRESSION` situations, changes in support conditions etc. should be handled using `REPEAT LOAD` cases, *not* load combination cases.

```
    PERFORM ANALYSIS PRINT STATICS CHECK
```

A static equilibrium report, consisting of total applied loading and total support reactions from each primary load case is requested along with the instructions to carry out a linear static analysis.

```
    PRINT JOINT DISPLACEMENTS LIST 8 9
```

Global displacements at nodes 8 and 9 are obtained using the above command.

```
    UNIT KNS METER
    PRINT SUPPORT REACTIONS
```

Reactions at the supports are obtained using the above command.

```
    UNIT NEWTON MMS
    PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
```

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
UNIT KNS MET
JOINT COORDINATES
  1  0.0  0.0  2.0   4  0.0  3.0  2.0
  5  1.0  0.0  2.0   8  1.0  3.0  2.0
  9  2.0  0.0  2.0  12  2.0  3.0  2.0
 21  0.0  0.0  1.0  24  0.0  3.0  1.0
 25  1.0  0.0  1.0  28  1.0  3.0  1.0
 29  2.0  0.0  1.0  32  2.0  3.0  1.0
 41  0.0  0.0  0.0  44  0.0  3.0  0.0
 45  1.0  0.0  0.0  48  1.0  3.0  0.0
 49  2.0  0.0  0.0  52  2.0  3.0  0.0
ELEMENT INCIDENCES SOLID
  1    1   5   6   2  21  25  26  22   TO   3
  4   21  25  26  22  41  45  46  42   TO   6  1 1
  7    5   9  10   6  25  29  30  26   TO   9  1 1
 10   25  29  30  26  45  49  50  46   TO  12  1 1
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
PRINT ELEMENT INFO SOLID LIST 1 TO 5
SUPPORTS
1  5   21  25  29  41  45  49  PINNED
9 ENFORCED BUT MX MY MZ
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24  42 TO 44 FX 100.0
LOAD 3
SUPPORT DISPLACEMENT
9 FX .0011
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
LOAD COMB 10
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT JOINT DISPLACEMENTS LIST 8 9
UNIT KNS METER
PRINT SUPPORT REACTIONS
UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46: 8                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
INPUT FILE: US-24 Analysis of a Concrete Block Using Solid Elements.STD
     2. UNIT KNS MET
     3. JOINT COORDINATES
     4. 1  0.0  0.0  2.0   4  0.0  3.0  2.0
     5. 5  1.0  0.0  2.0   8  1.0  3.0  2.0
     6. 9  2.0  0.0  2.0  12  2.0  3.0  2.0
```

```
     7. 21  0.0  0.0  1.0  24  0.0  3.0  1.0
     8. 25  1.0  0.0  1.0  28  1.0  3.0  1.0
     9. 29  2.0  0.0  1.0  32  2.0  3.0  1.0
    10. 41  0.0  0.0  0.0  44  0.0  3.0  0.0
    11. 45  1.0  0.0  0.0  48  1.0  3.0  0.0
    12. 49  2.0  0.0  0.0  52  2.0  3.0  0.0
    13. ELEMENT INCIDENCES SOLID
    14. 1    1   5   6   2  21  25  26  22   TO   3
    15. 4   21  25  26  22  41  45  46  42   TO   6  1 1
    16. 7    5   9  10   6  25  29  30  26   TO   9  1 1
    17. 10   25  29  30  26  45  49  50  46   TO  12  1 1
    18. UNIT MMS
    19. DEFINE MATERIAL START
    20. ISOTROPIC STEEL
    21. E 210
    22. POISSON 0.25
    23. DENSITY 7.5E-008
    24. ALPHA 6E-006
    25. DAMP 0.03
    26. TYPE STEEL
    27. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
    28. END DEFINE MATERIAL
    29. CONSTANTS
    30. MATERIAL STEEL ALL
    31. UNIT METER
    32. PRINT ELEMENT INFO SOLID LIST 1 TO 5
  ELEMENT  INFO     SOLID    LIST
      EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    2
 ELEMENT NODE-1  NODE-2  NODE-3  NODE-4  NODE-5  NODE-6  NODE-7  NODE-8
       1      1       5       6       2      21      25      26      22
       2      2       6       7       3      22      26      27      23
       3      3       7       8       4      23      27      28      24
       4     21      25      26      22      41      45      46      42
       5     22      26      27      23      42      46      47      43
      EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    3
   MATERIAL PROPERTIES.
   --------------------
   ALL UNITS ARE - KNS  METE
 ELEMENT   YOUNG'S MODULUS   MODULUS OF RIGIDITY    DENSITY       ALPHA
       1      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
       2      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
       3      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
       4      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
       5      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
    33. SUPPORTS
    34. 1  5   21  25  29  41  45  49  PINNED
    35. 9 ENFORCED BUT MX MY MZ
    36. LOAD 1
    37. SELF Y -1.0
    38. JOINT LOAD
    39. 28 FY -1000.0
    40. LOAD 2
    41. JOINT LOADS
    42. 2 TO 4 22 TO 24  42 TO 44 FX 100.0
    43. LOAD 3
    44. SUPPORT DISPLACEMENT
    45. 9 FX .0011
    46. UNIT POUND FEET
```

```
   47. LOAD 4
   48. ELEMENT LOAD SOLIDS
   49. 3 6 9 12 FACE 4 PRE GY -500.0
   50. UNIT KNS MMS
   51. LOAD 5
   52. REPEAT LOAD
   53. 1 1.0 2 1.0 3 1.0 4 1.0
   54. LOAD COMB 10
   55. 1 1.0 2 1.0
   56. PERFORM ANALYSIS PRINT STAT CHECK
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         36  NUMBER OF MEMBERS       0
    NUMBER OF PLATES          0  NUMBER OF SOLIDS       12
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      9
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    4
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     5, TOTAL DEGREES OF FREEDOM =      84
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    5
  *** NOTE: CAPACITY FOR MAXIMUM #    260 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                      X =  0.999999993E+03
                      Y =  0.228947364E+04
                      Z =  0.999999993E+03
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =      -1900.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     1900000.15  MY=          0.00  MZ=    -1900000.15
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =       1900.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=    -1900000.15  MY=         -0.00  MZ=     1900000.15
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      1)
           MAXIMUMS    AT NODE
     X = -1.21106E-04      23
     Y = -1.15439E-03      28
     Z =  1.21106E-04       7
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
          CENTER OF FORCE BASED ON X FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                      X =  0.000000000E+00
                      Y =  0.199999999E+04
                      Z =  0.999999993E+03
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    6
TOTAL APPLIED LOAD     2
```

```
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =        900.00
      SUMMATION FORCE-Y =          0.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=      900000.03  MZ=    -1800000.06
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =       -900.00
      SUMMATION FORCE-Y =         -0.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=     -900000.03  MZ=     1800000.06
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      2)
            MAXIMUMS    AT NODE
     X =  2.22892E-03       4
     Y =  7.83934E-04      44
     Z =  9.49033E-05      10
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     3
TOTAL APPLIED LOAD     3
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  0.0000000E+00
      SUMMATION FORCE-Y =  0.0000000E+00
      SUMMATION FORCE-Z =  0.0000000E+00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=  0.0000000E+00  MY=  0.0000000E+00  MZ=  0.0000000E+00
TOTAL REACTION LOAD    3
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  1.6182536E-11
      SUMMATION FORCE-Y = -5.0570426E-12
      SUMMATION FORCE-Z =  2.6296621E-11
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=  5.5900954E-08  MY=  3.9459497E-08  MZ= -3.2882914E-09
     EXAMPLE PROBLEM USING SOLID ELEMENTS                    -- PAGE NO.    7
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      3)
            MAXIMUMS    AT NODE
     X =  1.10000E-01       9
     Y = -1.21497E-02       6
     Z =  1.61372E-02      24
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     4
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.999999993E+03
                       Y =  0.299999998E+04
                       Z =  0.999999993E+03
TOTAL APPLIED LOAD     4
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     4 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =        -95.76
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       95760.52  MY=           0.00  MZ=      -95760.52
```

```
TOTAL REACTION LOAD    4
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     4 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =          95.76
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      -95760.52  MY=           0.00  MZ=       95760.52
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      4)
            MAXIMUMS    AT NODE
     X =  3.17652E-06      50
     Y = -3.35288E-05      28
     Z = -3.17652E-06      50
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     5
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    8
          CENTER OF FORCE BASED ON X FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.000000000E+00
                       Y =  0.199999999E+04
                       Z =  0.999999993E+03
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.999999993E+03
                       Y =  0.232356609E+04
                       Z =  0.999999993E+03
TOTAL APPLIED LOAD     5
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     5 )
      SUMMATION FORCE-X =         900.00
      SUMMATION FORCE-Y =       -1995.76
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     1995760.67  MY=      900000.03  MZ=    -3795760.73
TOTAL REACTION LOAD    5
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     5 )
      SUMMATION FORCE-X =        -900.00
      SUMMATION FORCE-Y =        1995.76
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=    -1995760.67  MY=     -900000.03  MZ=     3795760.73
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      5)
            MAXIMUMS    AT NODE
     X =  1.10000E-01       9
     Y = -1.23568E-02       6
     Z =  1.61372E-02      24
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
  ************ END OF DATA FROM INTERNAL STORAGE ************
   57. PRINT JOINT DISPLACEMENTS LIST 8 9
 JOINT    DISPLACE LIST     8
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    9
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     8    1     0.0000   -0.0002   -0.0001    0.0000    0.0000    0.0000
          2     0.0020    0.0000   -0.0000    0.0000    0.0000    0.0000
```

```
          3     0.0193   -0.0049    0.0089    0.0000    0.0000    0.0000
          4     0.0000   -0.0000    0.0000    0.0000    0.0000    0.0000
          5     0.0213   -0.0052    0.0088    0.0000    0.0000    0.0000
         10     0.0020   -0.0002   -0.0001    0.0000    0.0000    0.0000
     9    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.1100    0.0000   -0.0000    0.0000    0.0000    0.0000
          4     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          5     0.1100    0.0000   -0.0000    0.0000    0.0000    0.0000
         10     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
  ************** END OF LATEST ANALYSIS RESULT **************
   58. UNIT KNS METER
   59. PRINT SUPPORT REACTIONS
 SUPPORT  REACTION
    EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   10
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     27.47    128.97    -27.47      0.00      0.00      0.00
          2    -72.24   -232.67     42.18      0.00      0.00      0.00
          3  -2022.70   -302.04  -1192.39      0.00      0.00      0.00
          4      1.52      6.63     -1.52      0.00      0.00      0.00
          5  -2065.94   -399.11  -1179.21      0.00      0.00      0.00
         10    -44.76   -103.70     14.70      0.00      0.00      0.00
     5    1      0.00    236.52    -54.44      0.00      0.00      0.00
          2    -62.32     11.42     -0.05      0.00      0.00      0.00
          3 -16410.02   7434.80  -2287.95      0.00      0.00      0.00
          4      0.00     11.97     -2.98      0.00      0.00      0.00
          5 -16472.33   7694.71  -2345.41      0.00      0.00      0.00
         10    -62.32    247.94    -54.49      0.00      0.00      0.00
    21    1     54.44    236.52     -0.00      0.00      0.00      0.00
          2   -159.92   -450.84     -0.00      0.00      0.00      0.00
          3  -3341.67  -2923.60  -1877.00      0.00      0.00      0.00
          4      2.98     11.97     -0.00      0.00      0.00      0.00
          5  -3444.18  -3125.95  -1877.00      0.00      0.00      0.00
         10   -105.49   -214.32     -0.00      0.00      0.00      0.00
    25    1     -0.00    438.06     -0.00      0.00      0.00      0.00
          2   -138.00      9.51     -0.00      0.00      0.00      0.00
          3 -19197.98   5248.66 -10975.25      0.00      0.00      0.00
          4     -0.00     21.34     -0.00      0.00      0.00      0.00
          5 -19335.98   5717.57 -10975.25      0.00      0.00      0.00
         10   -138.00    447.56     -0.00      0.00      0.00      0.00
    29    1    -54.44    236.52      0.00      0.00      0.00      0.00
          2   -170.27    431.34      0.00      0.00      0.00      0.00
          3   3902.73    512.05   3842.64      0.00      0.00      0.00
          4     -2.98     11.97      0.00      0.00      0.00      0.00
          5   3675.05   1191.87   3842.64      0.00      0.00      0.00
         10   -224.70    667.85      0.00      0.00      0.00      0.00
    41    1     27.47    128.97     27.47      0.00      0.00      0.00
          2    -72.24   -232.67    -42.18      0.00      0.00      0.00
          3   -891.15  -2739.86  -1598.54      0.00      0.00      0.00
          4      1.52      6.63      1.52      0.00      0.00      0.00
          5   -934.39  -2836.93  -1611.72      0.00      0.00      0.00
         10    -44.76   -103.70    -14.70      0.00      0.00      0.00
    45    1     -0.00    236.52     54.44      0.00      0.00      0.00
          2    -62.32     11.42      0.05      0.00      0.00      0.00
          3   -430.44   -752.46   -237.57      0.00      0.00      0.00
          4     -0.00     11.97      2.98      0.00      0.00      0.00
```

```
         5   -492.75   -492.56   -180.10      0.00      0.00      0.00
        10    -62.32    247.94     54.49      0.00      0.00      0.00
    49   1    -27.47    128.97     27.47      0.00      0.00      0.00
         2    -81.35    226.24     45.03      0.00      0.00      0.00
         3   -778.26   2073.77   1192.39      0.00      0.00      0.00
         4     -1.52      6.63      1.52      0.00      0.00      0.00
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   11
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
         5   -888.61   2435.62   1266.41      0.00      0.00      0.00
        10   -108.83    355.21     72.50      0.00      0.00      0.00
     9   1    -27.47    128.97    -27.47      0.00      0.00      0.00
         2    -81.35    226.24    -45.03      0.00      0.00      0.00
         3  39169.49  -8551.31  13133.66      0.00      0.00      0.00
         4     -1.52      6.63     -1.52      0.00      0.00      0.00
         5  39059.14  -8189.46  13059.64      0.00      0.00      0.00
        10   -108.83    355.21    -72.50      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   60. UNIT NEWTON MMS
   61. PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
 ELEMENT  JOINT    STRESS   SOLID
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   12
ELEMENT STRESSES        UNITS= NEWTMMS
------------------------------------------------------------------------------
             NODE/          NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
------------------------------------------------------------------------------
    4    1     21     -0.144     -0.449     -0.155     -0.006     -0.011      0.000
    4    1     25     -0.132     -0.368     -0.132     -0.011     -0.011      0.005
    4    1     26     -0.009     -0.377     -0.009     -0.003     -0.003      0.005
    4    1     22     -0.012     -0.449     -0.005      0.002     -0.018      0.009
    4    1     41     -0.152     -0.484     -0.152     -0.015     -0.015     -0.005
    4    1     45     -0.155     -0.449     -0.144     -0.011     -0.006     -0.000
    4    1     46     -0.005     -0.449     -0.012     -0.018      0.002      0.009
    4    1     42      0.007     -0.475      0.007     -0.023     -0.023      0.014
    4    1 CENTER     -0.075     -0.437     -0.075     -0.011     -0.011      0.005
             S1=     -0.070   S2=     -0.080   S3=     -0.438   SE=      0.363
             DC=      0.707     -0.041      0.707     -0.707     -0.000      0.707
    4    2     21      0.176      1.021      0.284      0.217      0.014      0.005
    4    2     25      0.154     -0.006      0.022      0.251      0.014     -0.029
    4    2     26     -0.028      0.053     -0.015      0.253      0.016     -0.002
    4    2     22     -0.054      1.031      0.103      0.219      0.012     -0.036
    4    2     41      0.189      1.034      0.321      0.258      0.038      0.029
    4    2     45      0.162     -0.006      0.054      0.223     -0.010     -0.005
    4    2     46     -0.225     -0.016     -0.051      0.221     -0.008     -0.026
    4    2     42     -0.247      0.976      0.071      0.255      0.036     -0.060
    4    2 CENTER      0.016      0.511      0.099      0.237      0.014     -0.015
             S1=      0.606   S2=      0.101   S3=     -0.082   SE=      0.617
             DC=      0.372      0.928      0.014     -0.106      0.027      0.994
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   13
ELEMENT STRESSES        UNITS= NEWTMMS
------------------------------------------------------------------------------
             NODE/          NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
------------------------------------------------------------------------------
    4    3     21      0.900      5.181      1.884      4.989      5.058      0.396
    4    3     25     -0.893     -5.740     -1.294      6.615      1.429     -1.229
```

```
    4    3     26      5.251     -3.282      3.647      5.654      0.468      3.274
    4    3     22      5.379      5.974      1.830      4.029      6.019      1.649
    4    3     41      2.148      9.507      2.550      0.107      5.891      1.229
    4    3     45      2.276      4.348      1.292     -1.518      0.596     -0.396
    4    3     46     -1.334      3.555      2.982     -0.558     -0.364      2.442
    4    3     42     -3.127      7.049     -0.756      1.067      6.851      0.816
    4    3 CENTER      1.325      3.324      1.517      2.548      3.244      1.023
             S1=      7.030   S2=      0.411   S3=     -1.275   SE=      7.604
             DC=      0.425      0.744      0.516      0.809     -0.055     -0.586
    4    4     21     -0.008     -0.024     -0.008     -0.001     -0.001     -0.000
    4    4     25     -0.008     -0.022     -0.008     -0.001     -0.001      0.000
    4    4     26      0.001     -0.022      0.001     -0.001     -0.001      0.000
    4    4     22      0.001     -0.024      0.001     -0.001     -0.001      0.000
    4    4     41     -0.008     -0.026     -0.008     -0.001     -0.001     -0.000
    4    4     45     -0.008     -0.024     -0.008     -0.001     -0.001     -0.000
    4    4     46      0.001     -0.024      0.001     -0.001     -0.001      0.000
    4    4     42      0.001     -0.026      0.001     -0.001     -0.001      0.000
    4    4 CENTER     -0.004     -0.024     -0.004     -0.001     -0.001      0.000
             S1=     -0.003   S2=     -0.004   S3=     -0.024   SE=      0.021
             DC=      0.705     -0.070      0.705     -0.707      0.000      0.707
    4    5     21      0.925      5.729      2.005      5.199      5.061      0.402
    4    5     25     -0.878     -6.136     -1.412      6.854      1.431     -1.254
    4    5     26      5.215     -3.629      3.624      5.903      0.481      3.277
    4    5     22      5.313      6.532      1.928      4.248      6.011      1.622
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   14
ELEMENT STRESSES        UNITS= NEWTMMS
-----------------------------------------------------------------------------
             NODE/          NORMAL STRESSES               SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
-----------------------------------------------------------------------------
    4    5     41      2.176     10.031      2.710      0.349      5.913      1.254
    4    5     45      2.275      3.869      1.194     -1.307      0.579     -0.402
    4    5     46     -1.564      3.066      2.920     -0.356     -0.371      2.425
    4    5     42     -3.366      7.524     -0.677      1.299      6.864      0.770
    4    5 CENTER      1.262      3.373      1.537      2.774      3.246      1.012
             S1=      7.193   S2=      0.379   S3=     -1.400   SE=      7.856
             DC=      0.435      0.745      0.505     -0.764      0.008      0.645
    4   10     21      0.032      0.572      0.129      0.211      0.004      0.005
    4   10     25      0.022     -0.374     -0.110      0.240      0.004     -0.024
    4   10     26     -0.038     -0.325     -0.024      0.250      0.013      0.003
    4   10     22     -0.067      0.582      0.098      0.221     -0.006     -0.027
    4   10     41      0.036      0.550      0.168      0.242      0.023      0.024
    4   10     45      0.007     -0.455     -0.090      0.213     -0.016     -0.005
    4   10     46     -0.230     -0.465     -0.063      0.203     -0.006     -0.017
    4   10     42     -0.240      0.501      0.078      0.233      0.013     -0.046
    4   10 CENTER     -0.060      0.073      0.023      0.227      0.004     -0.011
             S1=      0.243   S2=      0.024   S3=     -0.230   SE=      0.410
             DC=      0.600      0.800     -0.017     -0.024      0.039      0.999
    6    1     23      0.329      0.394      0.413     -0.043     -0.127     -0.060
    6    1     27     -0.071     -1.739     -0.071     -0.099     -0.099     -0.005
    6    1     28     -0.676     -1.849     -0.676     -0.553     -0.553     -0.005
    6    1     24     -0.166      0.394      0.140     -0.498      0.328      0.051
    6    1     43     -0.097     -0.200     -0.097     -0.182     -0.182     -0.115
    6    1     47      0.413      0.394      0.329     -0.127     -0.043     -0.060
    6    1     48      0.140      0.394     -0.166      0.328     -0.498      0.051
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   15
ELEMENT STRESSES        UNITS= NEWTMMS
-----------------------------------------------------------------------------
```

```
             NODE/           NORMAL STRESSES               SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
--------------------------------------------------------------------------------
    6    1     44     -0.259     -0.089     -0.259      0.273      0.273      0.106
    6    1 CENTER     -0.049     -0.287     -0.049     -0.113     -0.113     -0.005
             S1=      0.027   S2=     -0.044   S3=     -0.368   SE=       0.365
             DC=       0.631     -0.451      0.631     -0.707     -0.000      0.707
    6    2     23     -0.032      0.112     -0.001      0.030     -0.002      0.016
    6    2     27     -0.001     -0.025     -0.046      0.073     -0.013     -0.027
    6    2     28     -0.096     -0.003     -0.065      0.083     -0.003     -0.035
    6    2     24     -0.085      0.177      0.109      0.040     -0.012     -0.078
    6    2     43     -0.152      0.158      0.052      0.136     -0.023     -0.005
    6    2     47     -0.140     -0.041     -0.013      0.092      0.008     -0.049
    6    2     48     -0.496     -0.105     -0.119      0.082      0.019     -0.014
    6    2     44     -0.464      0.136      0.076      0.125     -0.033     -0.057
    6    2 CENTER     -0.183      0.051     -0.001      0.083     -0.007     -0.031
             S1=      0.081   S2=     -0.001   S3=     -0.213   SE=       0.263
             DC=       0.314      0.928     -0.202     -0.060      0.232      0.971
    6    3     23     -2.744     -0.535     -0.041     -0.327     -0.468      0.408
    6    3     27     -3.140     -0.556     -1.018      0.642      0.296     -0.560
    6    3     28      1.815      0.568      0.607      0.402      0.056      0.214
    6    3     24      1.900      0.279      0.654     -0.567     -0.228     -0.755
    6    3     43      0.636     -0.478      0.687     -0.031     -0.313      0.563
    6    3     47      0.721      0.942      0.191     -0.999      0.141     -0.405
    6    3     48     -0.136      0.128     -0.121     -0.759     -0.099      0.058
    6    3     44     -0.531     -1.602     -0.555      0.210     -0.073     -0.910
    6    3 CENTER     -0.185     -0.157      0.050     -0.179     -0.086     -0.173
             S1=      0.143   S2=     -0.010   S3=     -0.424   SE=       0.508
             DC=      -0.484      0.038      0.874     -0.507      0.802     -0.316
     EXAMPLE PROBLEM USING SOLID ELEMENTS                      -- PAGE NO.   16
ELEMENT STRESSES        UNITS= NEWTMMS
--------------------------------------------------------------------------------
             NODE/           NORMAL STRESSES               SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
--------------------------------------------------------------------------------
    6    4     23      0.000     -0.024      0.000     -0.000     -0.000     -0.000
    6    4     27      0.000     -0.024      0.000     -0.000     -0.000     -0.000
    6    4     28     -0.000     -0.024     -0.000     -0.000     -0.000     -0.000
    6    4     24     -0.000     -0.024     -0.000     -0.000     -0.000      0.000
    6    4     43      0.000     -0.024      0.000     -0.000     -0.000     -0.000
    6    4     47      0.000     -0.024      0.000     -0.000     -0.000     -0.000
    6    4     48     -0.000     -0.024     -0.000     -0.000     -0.000      0.000
    6    4     44     -0.000     -0.024     -0.000     -0.000     -0.000      0.000
    6    4 CENTER      0.000     -0.024      0.000     -0.000     -0.000     -0.000
             S1=      0.000   S2=     -0.000   S3=     -0.024   SE=       0.024
             DC=      -0.707      0.000      0.707      0.707     -0.002      0.707
    6    5     23     -2.448     -0.052      0.370     -0.340     -0.596      0.364
    6    5     27     -3.211     -2.343     -1.135      0.616      0.185     -0.592
    6    5     28      1.043     -1.309     -0.134     -0.068     -0.500      0.174
    6    5     24      1.649      0.826      0.902     -1.025      0.089     -0.782
    6    5     43      0.387     -0.545      0.642     -0.077     -0.518      0.443
    6    5     47      0.994      1.271      0.506     -1.034      0.106     -0.514
    6    5     48     -0.492      0.393     -0.406     -0.349     -0.578      0.096
    6    5     44     -1.255     -1.580     -0.739      0.608      0.167     -0.861
    6    5 CENTER     -0.417     -0.417      0.001     -0.209     -0.206     -0.209
             S1=      0.117   S2=     -0.208   S3=     -0.741   SE=       0.750
             DC=      -0.265     -0.255      0.930      0.705     -0.710      0.006
    6   10     23      0.296      0.507      0.412     -0.013     -0.128     -0.044
```

```
    6   10     27     -0.071    -1.764    -0.117    -0.025    -0.112    -0.032
    6   10     28     -0.773    -1.853    -0.741    -0.470    -0.556    -0.039
    6   10     24     -0.251     0.572     0.249    -0.458     0.316    -0.027
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   17
ELEMENT STRESSES        UNITS= NEWTMMS
-----------------------------------------------------------------------------
             NODE/          NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ       SZX
-----------------------------------------------------------------------------
    6   10     43     -0.249    -0.043    -0.045    -0.046    -0.205    -0.121
    6   10     47      0.272     0.354     0.315    -0.034    -0.035    -0.109
    6   10     48     -0.356     0.289    -0.285     0.410    -0.479     0.037
    6   10     44     -0.724     0.047    -0.184     0.398     0.239     0.050
    6   10 CENTER     -0.232    -0.236    -0.050    -0.030    -0.120    -0.036
             S1=      0.011   S2=     -0.212   S3=     -0.316   SE=      0.289
             DC=     -0.080    -0.428     0.900     0.888    -0.441    -0.131
   62. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46: 8 ****
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   18
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *M. To assign an enforced support* (on page 820)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)

EX. US-25 Analysis of a Structure with Compression-Only Members

This example demonstrates the usage of compression-only members. Since the structural condition is load dependent, the `PERFORM ANALYSIS` command is specified once for each primary load case.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-25 Analysis of a Structure with Compression-Only Members.STD` when you install the program.

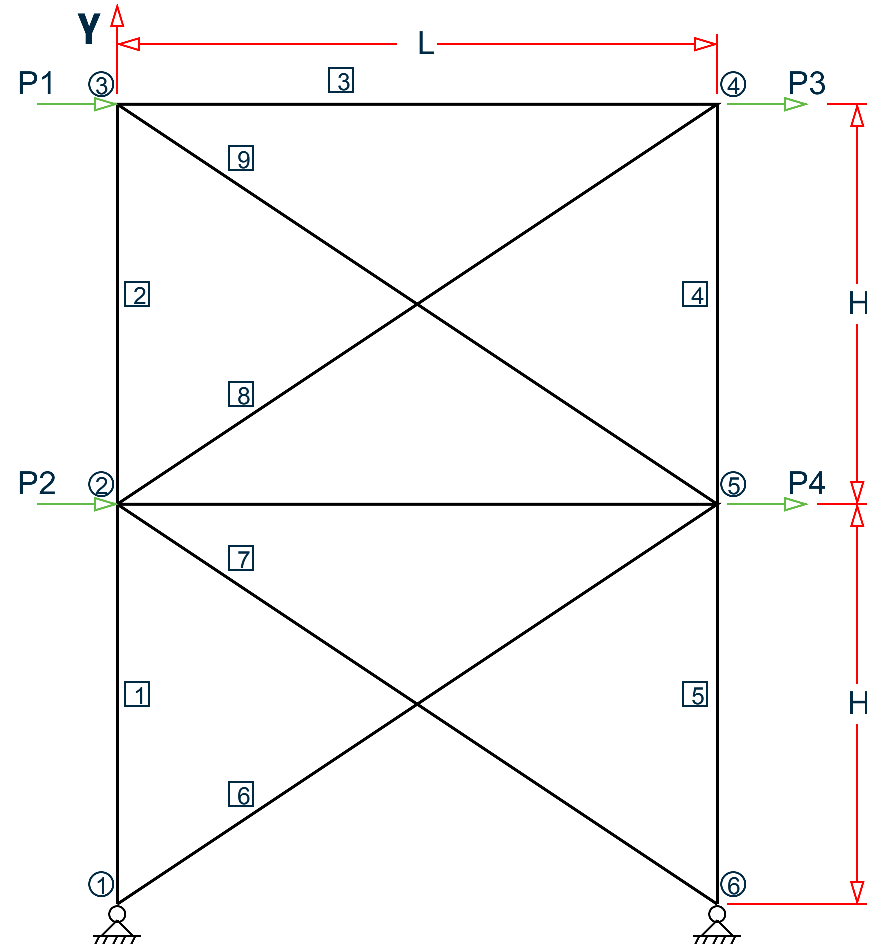


Figure 537: Example Problem No. 25

This example has been created to illustrate the command specification for a structure with certain members capable of carrying compressive force only. It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of “active” members (and hence the stiffness matrix) is load case dependent.

where:

L = 15 ft, H = 10 ft
Load case 1: P1 = 10 kips & P2 = 15 kips
Load case 2: P3 = -10 kips & P4 = -15 kips

```
STAAD PLANE
* EXAMPLE FOR COMPRESSION-ONLY MEMBERS
```

The input data is initiated with the word STAAD. This structure is a PLANE frame. The second line is an optional comment line.

```
UNIT FEET KIP
```

Units for the commands to follow are specified above.

```
JOINT COORDINATES
1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
```

Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
1 1 2 5 ; 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

The members 1 to 10 are defined along with the joints to which they are connected.

```
MEMBER COMPRESSION
6 TO 9
```

Members 6 to 9 are defined as COMPRESSION-only members. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a tensile force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26
```

Properties for members 1 to 10 are defined as the STandard W12X26 section from the American AISC steel table.

```
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 6 PINNED
```

Joints 1 and 6 are declared as pinned-supported.

```
LOAD 1
JOINT LOAD
2 FX 15
3 FX 10
```

Load 1 is defined above and consists of joint loads in the global X direction at joints 2 and 3.

```
PERFORM ANALYSIS
```

The above structure is analyzed for load case 1.

```
CHANGE
MEMBER COMPRESSION
6 TO 9
```

One or more among the members 6 to 9 may have been in-activated in the previous analysis. The `CHANGE` command restores the original structure to prepare it for the analysis for the next primary load case. The members with the compression-only attribute are specified again.

```
LOAD 2
JOINT LOAD
```

```
    4 FX -10
    5 FX -15
```

In load case 2, joint loads are applied in the negative global X direction at joints 4 and 5.

```
    PERFORM ANALYSIS
    CHANGE
```

The instruction to analyze the structure is specified again. Next, any compression-only members that were inactivated during the second analysis (due to the fact that they were subjected to tensile axial forces) are re-activated with the CHANGE command. Without the re-activation, these members cannot be accessed for further processing.

```
    LOAD 3
    REPEAT LOAD
    1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. In other words, the above instruction is the same as the following:

```
    LOAD 3
    JOINT LOAD
    2 FX 15
    3 FX 10
    4 FX -15
    5 FX –10
    PERFORM ANALYSIS
```

The analysis is carried out for load case 3.

```
    CHANGE
```

The members inactivated during the analysis of load case 3 are re-activated for further processing.

```
    LOAD LIST ALL
```

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the “active” load cases. The `LOAD LIST ALL` command enables all the load cases in the structure to be made active for further processing.

```
    PRINT ANALYSIS RESULTS
```

The program is instructed to write the joint displacements, support reactions and member forces to the output file.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD PLANE EXAMPLE FOR COMPRESSION-ONLY MEMBERS
UNIT FEET KIP
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER COMPRESSION
```

```
6 TO 9
MEMBER PROPERTY AMERICAN
1 TO 10 TA ST W12X26
UNIT INCH
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANT
MATERIAL STEEL ALL
SUPPORT
1 6 PINNED
LOAD 1
JOINT LOAD
2 FX 15
3 FX 10
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 2
JOINT LOAD
4 FX -10
5 FX -15
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:10                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR COMPRESSION-ONLY MEMBERS
```

```
INPUT FILE: US-25 Analysis of a Structure with Compression-Only Members.STD
     2. UNIT FEET KIP
     3. SET NL 3
     4. JOINT COORDINATES
     5. 1 0 0 ; 2 0 10 ; 3 0 20 ; 4 15 20 ; 5 15 10 ; 6 15 0
     6. MEMBER INCIDENCES
     7. 1 1 2 5
     8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
     9. MEMBER COMPRESSION
    10. 6 TO 9
    11. MEMBER PROPERTY AMERICAN
    12. 1 TO 10 TA ST W12X26
    13. UNIT INCH
    14. DEFINE MATERIAL START
    15. ISOTROPIC STEEL
    16. E 29000
    17. POISSON 0.3
    18. DENSITY 283E-006
    19. ALPHA 6E-006
    20. DAMP 0.03
    21. TYPE STEEL
    22. STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
    23. END DEFINE MATERIAL
    24. CONSTANT
    25. MATERIAL STEEL ALL
    26. SUPPORT
    27. 1 6 PINNED
    28. LOAD 1
    29. JOINT LOAD
    30. 2 FX 15
    31. 3 FX 10
    32. PERFORM ANALYSIS
      EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    1
    33. CHANGE
    34. MEMBER COMPRESSION
    35. 6 TO 9
    36. LOAD 2
    37. JOINT LOAD
    38. 4 FX -10
    39. 5 FX -15
    40. PERFORM ANALYSIS
*** LOAD CASE     2 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    2
    41. CHANGE
    42. MEMBER COMPRESSION
    43. 6 TO 9
    44. LOAD 3
```

```
   45. REPEAT LOAD
   46. 1 1.0 2 1.0
   47. PERFORM ANALYSIS
 **NOTE-Tension/Compression converged after  1 iterations, Case=    3
   48. CHANGE
     EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    3
   49. LOAD LIST ALL
   50. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    4
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00041
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00050
          3    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00004
     2    1    0.04314   0.01262   0.00000   0.00000   0.00000  -0.00025
          2   -0.05088  -0.00373   0.00000   0.00000   0.00000   0.00024
          3    0.00364   0.00080   0.00000   0.00000   0.00000  -0.00001
     3    1    0.07775   0.01618   0.00000   0.00000   0.00000  -0.00022
          2   -0.07766  -0.00381   0.00000   0.00000   0.00000   0.00015
          3    0.00255   0.00215   0.00000   0.00000   0.00000   0.00001
     4    1    0.07766  -0.00381   0.00000   0.00000   0.00000  -0.00015
          2   -0.07775   0.01618   0.00000   0.00000   0.00000   0.00022
          3   -0.00255   0.00215   0.00000   0.00000   0.00000  -0.00001
     5    1    0.05088  -0.00373   0.00000   0.00000   0.00000  -0.00024
          2   -0.04314   0.01262   0.00000   0.00000   0.00000   0.00025
          3   -0.00364   0.00080   0.00000   0.00000   0.00000   0.00001
     6    1    0.00000   0.00000   0.00000   0.00000   0.00000  -0.00050
          2    0.00000   0.00000   0.00000   0.00000   0.00000   0.00041
          3    0.00000   0.00000   0.00000   0.00000   0.00000   0.00004
     EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    5
  SUPPORT REACTIONS -UNIT KIP  INCH    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -0.13    -23.33      0.00      0.00      0.00      0.00
          2     24.87     23.33      0.00      0.00      0.00      0.00
          3      2.18      0.00      0.00      0.00      0.00      0.00
     6    1    -24.87     23.33      0.00      0.00      0.00      0.00
          2      0.13    -23.33      0.00      0.00      0.00      0.00
          3     -2.18      0.00      0.00      0.00      0.00      0.00
     EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    6
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KIP  INCH     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1    -23.33      0.13     0.00      0.00      0.00       0.00
                2     23.33     -0.13     0.00      0.00      0.00      15.87
          2     1      6.90     -0.22     0.00      0.00      0.00      -0.00
                2     -6.90      0.22     0.00      0.00      0.00     -25.90
          3     1     -1.47      0.03     0.00      0.00      0.00       0.00
                2      1.47     -0.03     0.00      0.00      0.00       3.24
     2    1     2     -6.58      0.24     0.00      0.00      0.00      12.72
                3      6.58     -0.24     0.00      0.00      0.00      15.70
          2     2      0.15     -0.11     0.00      0.00      0.00      -2.32
                3     -0.15      0.11     0.00      0.00      0.00     -11.30
          3     2     -2.50     -0.03     0.00      0.00      0.00      -2.74
                3      2.50      0.03     0.00      0.00      0.00      -0.79
```

```
   3    1     3      0.11     -0.15      0.00       0.00       0.00     -15.70
              4     -0.11      0.15      0.00       0.00       0.00     -11.30
        2     3      0.11      0.15      0.00       0.00       0.00      11.30
              4     -0.11     -0.15      0.00       0.00       0.00      15.70
        3     3      6.28      0.00      0.00       0.00       0.00       0.79
              4     -6.28     -0.00      0.00       0.00       0.00      -0.79
   4    1     4      0.15      0.11      0.00       0.00       0.00      11.30
              5     -0.15     -0.11      0.00       0.00       0.00       2.32
        2     4     -6.58     -0.24      0.00       0.00       0.00     -15.70
              5      6.58      0.24      0.00       0.00       0.00     -12.72
        3     4     -2.50      0.03      0.00       0.00       0.00       0.79
              5      2.50     -0.03      0.00       0.00       0.00       2.74
   5    1     5      6.90      0.22      0.00       0.00       0.00      25.90
              6     -6.90     -0.22      0.00       0.00       0.00       0.00
        2     5    -23.33     -0.13      0.00       0.00       0.00     -15.87
              6     23.33      0.13      0.00       0.00       0.00       0.00
        3     5     -1.47     -0.03      0.00       0.00       0.00      -3.24
              6      1.47      0.03      0.00       0.00       0.00       0.00
   6    1     1      0.00      0.00      0.00       0.00       0.00       0.00
              5      0.00      0.00      0.00       0.00       0.00       0.00
        2     1     29.63      0.00      0.00       0.00       0.00       0.00
              5    -29.63      0.00      0.00       0.00       0.00       0.00
        3     1      2.66      0.00      0.00       0.00       0.00       0.00
              5     -2.66      0.00      0.00       0.00       0.00       0.00
   7    1     2     29.63      0.00      0.00       0.00       0.00       0.00
              6    -29.63      0.00      0.00       0.00       0.00       0.00
   EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    7
MEMBER END FORCES    STRUCTURE TYPE = PLANE
-----------------
ALL UNITS ARE -- KIP  INCH     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
        2     2      0.00      0.00      0.00       0.00       0.00       0.00
              6      0.00      0.00      0.00       0.00       0.00       0.00
        3     2      2.66      0.00      0.00       0.00       0.00       0.00
              6     -2.66      0.00      0.00       0.00       0.00       0.00
   8    1     2      0.00      0.00      0.00       0.00       0.00       0.00
              4      0.00      0.00      0.00       0.00       0.00       0.00
        2     2     11.60      0.00      0.00       0.00       0.00       0.00
              4    -11.60      0.00      0.00       0.00       0.00       0.00
        3     2      4.51      0.00      0.00       0.00       0.00       0.00
              4     -4.51      0.00      0.00       0.00       0.00       0.00
   9    1     3     11.60      0.00      0.00       0.00       0.00       0.00
              5    -11.60      0.00      0.00       0.00       0.00       0.00
        2     3      0.00      0.00      0.00       0.00       0.00       0.00
              5      0.00      0.00      0.00       0.00       0.00       0.00
        3     3      4.51      0.00      0.00       0.00       0.00       0.00
              5     -4.51      0.00      0.00       0.00       0.00       0.00
  10    1     2     -9.55     -0.32      0.00       0.00       0.00     -28.60
              5      9.55      0.32      0.00       0.00       0.00     -28.21
        2     2     -9.55      0.32      0.00       0.00       0.00      28.21
              5      9.55     -0.32      0.00       0.00       0.00      28.60
        3     2      8.98     -0.00      0.00       0.00       0.00      -0.50
              5     -8.98      0.00      0.00       0.00       0.00       0.50
************** END OF LATEST ANALYSIS RESULT **************
 51. FINISH
   EXAMPLE FOR COMPRESSION-ONLY MEMBERS                     -- PAGE NO.    8
          *********** END OF THE STAAD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:46:11 ****
```

```
*********************************************************
*   For technical assistance on STAAD.Pro, please visit    *
*   http://www.bentley.com/en/support/                     *
*                                                          *
*   Details about additional assistance from               *
*   Bentley and Partners can be found at program menu      *
*   Help->Technical Support                                *
*                                                          *
*           Copyright (c) Bentley Systems, Inc.            *
*                http://www.bentley.com                    *
*********************************************************
```

EX. US-26 Modeling a Rigid Diaphragm Using Control-Dependent

The structure in this example is a building consisting of member columns as well as floors made up of beam members and plate elements. Using the control-dependent command, the floors are specified to be rigid diaphragms for in-plane actions but flexible for bending actions.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-26 Modeling a Rigid Diaphragm Using Control-Dependent.STD` when you install the program.

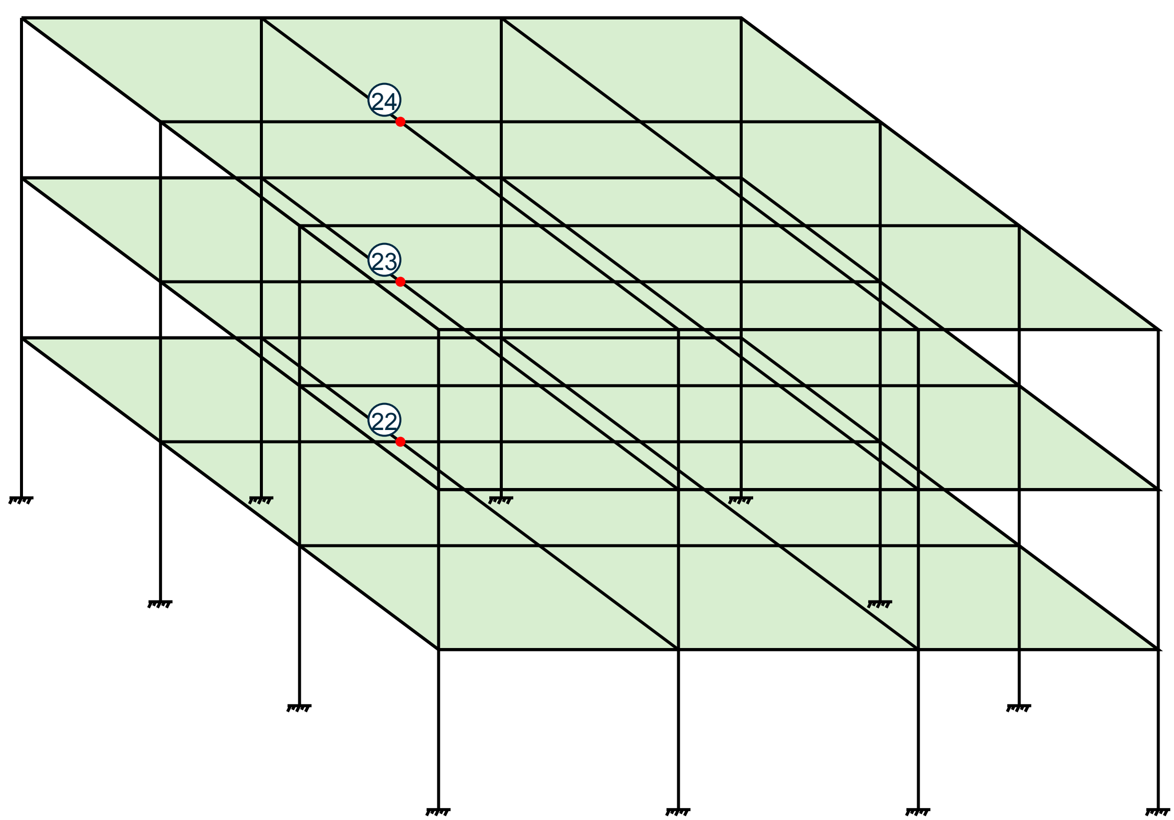


Figure 538: Example Problem No. 26

```
STAAD SPACE
*MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
```

Every STAAD input file has to begin with the word `STAAD`. The word `SPACE` signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line is an optional title to identify this project.

```
UNITS KIP FT
```

Specify units for the following data.

```
JOINT COORD
1 0 0 0 4 0 48 0
REPEAT 3 24 0 0
REPEAT ALL 3 0 0 24
DELETE JOINT 21 25 37 41
```

The joint numbers and coordinates are specified above. The unwanted joints, created during the generation process used above, are then deleted.

```
MEMBER INCI
1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
88 22 6 90 ; 91 26 10 93 ; 94 30 14 96
97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
115 58 42 117 ; 118 62 46 120
```

The `MEMBER INCIDENCE` specification is used for specifying MEMBER connectivities.

```
ELEMENT INCI
152 50 34 38 54 TO 154
155 54 38 42 58 TO 157
158 58 42 46 62 TO 160
161 34 18 22 38 TO 163
164 38 22 26 42 TO 166
167 42 26 30 46 TO 169
170 18 2 6 22 TO 172
173 22 6 10 26 TO 175
176 26 10 14 30 TO 178
```

The `ELEMENT INCIDENCE` specification is used for specifying plate element connectivities.

```
MEMBER PROPERTIES AMERICAN
1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
49 TO 120 TABLE ST W27X84
```

Tip: The elements are *not* what provides the diaphragm action for this model. Rather, the control-dependent node relationships defined later in the model dictate that interaction.

All members are `WIDE FLANGE` sections whose properties are obtained from the built in American steel table.

```
ELEMENT PROP
152 TO 178 THICK 0.75
```

The thickness of the plate elements is specified above.

```
UNIT INCHES
DEFINE MATERIAL START
```

```
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
UNIT FEET
CONSTANTS
MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
MATERIAL CONCRETE MEMB 152 TO 178
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The orientation of some of the members is set using the `BETA` angle command.

```
SUPPORTS
1 TO 17 BY 4 29 33 45 TO 61 BY 4 FIXED
```

The supports at the above mentioned joints are declared as fixed.

```
DEPENDENT DIA ZX CONTROL 22 JOINTS YR 15.0 17.0
DEPENDENT DIA ZX CONTROL 23 JOINTS YR 31.0 33.0
DEPENDENT DIA ZX CONTROL 24 JOINTS YR 47.0 49.0
```

The three floors of the structure are specified to act as rigid diaphragms in the `ZX` plane with the corresponding control joint specified. The associated dependent joints in a floor are specified by the `YRANGE` parameter. The floors may still resist out-of-plane bending actions flexibly.

```
LOADING 1 LATERAL LOADS
JOINT LOADS
2 3 4 14 15 16 50 51 52 62 63 64 FZ 10.0
6 7 8 10 11 12 18 19 20 30 31 32 FZ 20.0
34 35 36 46 47 48 54 55 56 58 59 60 FZ 20.0
22 23 24 26 27 28 38 39 40 42 43 44 FZ 40.0
```

The above data describe a static load case. It consists of joint loads in the global Z direction.

```
LOADING 2 TORSIONAL LOADS
JOINT LOADS
2 3 4 50 51 52 FZ 5.0
14 15 16 62 63 64 FZ 15.0
6 7 8 18 19 20 FZ 10.0
10 11 12 30 31 32 FZ 30.0
34 35 36 54 55 56 FZ 10.0
46 47 48 58 59 60 FZ 30.0
22 23 24 38 39 40 FZ 20.0
26 27 28 42 43 44 FZ 60.0
```

The above data describe a static load case. It consists of joint loads that create a torsional loading on the structure.

```
    LOADING 3 DEAD LOAD
    ELEMENT LOAD
    152 TO 178 PRESS GY -1.0
```

The above data describe a static load case. It consists of plate element pressure on a floor in the negative global Y direction.

```
    PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```
    PRINT JOINT DISP LIST 4 TO 60 BY 8
    PRINT MEMBER FORCES LIST 116 115
    PRINT SUPPORT REACTIONS LIST 9 57
```

Print displacements at selected joints, then print member forces for two members, then print support reactions at selected joints.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Dec-18
END JOB INFORMATION
*MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 16 0; 3 0 32 0; 4 0 48 0; 5 24 0 0; 6 24 16 0; 7 24 32 0;
8 24 48 0; 9 48 0 0; 10 48 16 0; 11 48 32 0; 12 48 48 0; 13 72 0 0;
14 72 16 0; 15 72 32 0; 16 72 48 0; 17 0 0 24; 18 0 16 24; 19 0 32 24;
20 0 48 24; 22 24 16 24; 23 24 32 24; 24 24 48 24; 26 48 16 24;
27 48 32 24; 28 48 48 24; 29 72 0 24; 30 72 16 24; 31 72 32 24;
32 72 48 24; 33 0 0 48; 34 0 16 48; 35 0 32 48; 36 0 48 48; 38 24 16 48;
39 24 32 48; 40 24 48 48; 42 48 16 48; 43 48 32 48; 44 48 48 48;
45 72 0 48; 46 72 16 48; 47 72 32 48; 48 72 48 48; 49 0 0 72;
50 0 16 72; 51 0 32 72; 52 0 48 72; 53 24 0 72; 54 24 16 72;
55 24 32 72; 56 24 48 72; 57 48 0 72; 58 48 16 72; 59 48 32 72;
60 48 48 72; 61 72 0 72; 62 72 16 72; 63 72 32 72; 64 72 48 72;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 9 10; 8 10 11; 9 11 12;
10 13 14; 11 14 15; 12 15 16; 13 17 18; 14 18 19; 15 19 20; 22 29 30;
23 30 31; 24 31 32; 25 33 34; 26 34 35; 27 35 36; 34 45 46; 35 46 47;
36 47 48; 37 49 50; 38 50 51; 39 51 52; 40 53 54; 41 54 55; 42 55 56;
43 57 58; 44 58 59; 45 59 60; 46 61 62; 47 62 63; 48 63 64; 49 2 6;
50 3 7; 51 4 8; 52 6 10; 53 7 11; 54 8 12; 55 10 14; 56 11 15; 57 12 16;
58 18 22; 59 19 23; 60 20 24; 61 22 26; 62 23 27; 63 24 28; 64 26 30;
65 27 31; 66 28 32; 67 34 38; 68 35 39; 69 36 40; 70 38 42; 71 39 43;
72 40 44; 73 42 46; 74 43 47; 75 44 48; 76 50 54; 77 51 55; 78 52 56;
79 54 58; 80 55 59; 81 56 60; 82 58 62; 83 59 63; 84 60 64; 85 18 2;
86 19 3; 87 20 4; 88 22 6; 89 23 7; 90 24 8; 91 26 10; 92 27 11;
93 28 12; 94 30 14; 95 31 15; 96 32 16; 97 34 18; 98 35 19; 99 36 20;
100 38 22; 101 39 23; 102 40 24; 103 42 26; 104 43 27; 105 44 28;
106 46 30; 107 47 31; 108 48 32; 109 50 34; 110 51 35; 111 52 36;
```

```
112 54 38; 113 55 39; 114 56 40; 115 58 42; 116 59 43; 117 60 44;
118 62 46; 119 63 47; 120 64 48;
ELEMENT INCIDENCES SHELL
152 50 34 38 54; 153 51 35 39 55; 154 52 36 40 56; 155 54 38 42 58;
156 55 39 43 59; 157 56 40 44 60; 158 58 42 46 62; 159 59 43 47 63;
160 60 44 48 64; 161 34 18 22 38; 162 35 19 23 39; 163 36 20 24 40;
164 38 22 26 42; 165 39 23 27 43; 166 40 24 28 44; 167 42 26 30 46;
168 43 27 31 47; 169 44 28 32 48; 170 18 2 6 22; 171 19 3 7 23;
172 20 4 8 24; 173 22 6 10 26; 174 23 7 11 27; 175 24 8 12 28;
176 26 10 14 30; 177 27 11 15 31; 178 28 12 16 32;
MEMBER PROPERTY AMERICAN
1 TO 15 22 TO 27 34 TO 48 TABLE ST W14X90
49 TO 120 TABLE ST W27X84
ELEMENT PROPERTY
152 TO 178 THICKNESS 0.75
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 13 TO 15 22 TO 27 34 TO 39
MATERIAL STEEL_50_KSI MEMB 1 TO 15 22 TO 27 34 TO 120
MATERIAL CONCRETE MEMB 152 TO 178
SUPPORTS
1 5 9 13 17 29 33 45 49 53 57 61 FIXED
DEPENDENT ZX CONTROL 22 JOINT YR 15 17
DEPENDENT ZX CONTROL 23 JOINT YR 31 33
DEPENDENT ZX CONTROL 24 JOINT YR 47 49
LOAD 1 LATERAL LOADS
JOINT LOAD
2 TO 4 14 TO 16 50 TO 52 62 TO 64 FZ 10
6 TO 8 10 TO 12 18 TO 20 30 TO 32 FZ 20
34 TO 36 46 TO 48 54 TO 56 58 TO 60 FZ 20
22 TO 24 26 TO 28 38 TO 40 42 TO 44 FZ 40
LOAD 2 TORSIONAL LOADS
JOINT LOAD
2 TO 4 50 TO 52 FZ 5
14 TO 16 62 TO 64 FZ 15
6 TO 8 18 TO 20 FZ 10
10 TO 12 30 TO 32 FZ 30
34 TO 36 54 TO 56 FZ 10
46 TO 48 58 TO 60 FZ 30
22 TO 24 38 TO 40 FZ 20
26 TO 28 42 TO 44 FZ 60
```

```
LOAD 3 DEAD LOAD
ELEMENT LOAD
152 TO 178 PR GY -1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 4 12 20 28 36 44 52 60
PRINT MEMBER FORCES LIST 115 116
PRINT SUPPORT REACTION LIST 9 57
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:13                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: US-26 Modeling a Rigid Diaphragm Using Control-Dependent.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 10-DEC-18
     4. END JOB INFORMATION
     5. *MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
     6. UNIT FEET KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 16 0; 3 0 32 0; 4 0 48 0; 5 24 0 0; 6 24 16 0; 7 24 32 0
     9. 8 24 48 0; 9 48 0 0; 10 48 16 0; 11 48 32 0; 12 48 48 0; 13 72 0 0
    10. 14 72 16 0; 15 72 32 0; 16 72 48 0; 17 0 0 24; 18 0 16 24; 19 0 32 24
    11. 20 0 48 24; 22 24 16 24; 23 24 32 24; 24 24 48 24; 26 48 16 24
    12. 27 48 32 24; 28 48 48 24; 29 72 0 24; 30 72 16 24; 31 72 32 24
    13. 32 72 48 24; 33 0 0 48; 34 0 16 48; 35 0 32 48; 36 0 48 48; 38 24 16 48
    14. 39 24 32 48; 40 24 48 48; 42 48 16 48; 43 48 32 48; 44 48 48 48
    15. 45 72 0 48; 46 72 16 48; 47 72 32 48; 48 72 48 48; 49 0 0 72
    16. 50 0 16 72; 51 0 32 72; 52 0 48 72; 53 24 0 72; 54 24 16 72
    17. 55 24 32 72; 56 24 48 72; 57 48 0 72; 58 48 16 72; 59 48 32 72
    18. 60 48 48 72; 61 72 0 72; 62 72 16 72; 63 72 32 72; 64 72 48 72
    19. MEMBER INCIDENCES
    20. 1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 9 10; 8 10 11; 9 11 12
    21. 10 13 14; 11 14 15; 12 15 16; 13 17 18; 14 18 19; 15 19 20; 22 29 30
    22. 23 30 31; 24 31 32; 25 33 34; 26 34 35; 27 35 36; 34 45 46; 35 46 47
    23. 36 47 48; 37 49 50; 38 50 51; 39 51 52; 40 53 54; 41 54 55; 42 55 56
    24. 43 57 58; 44 58 59; 45 59 60; 46 61 62; 47 62 63; 48 63 64; 49 2 6
    25. 50 3 7; 51 4 8; 52 6 10; 53 7 11; 54 8 12; 55 10 14; 56 11 15; 57 12 16
    26. 58 18 22; 59 19 23; 60 20 24; 61 22 26; 62 23 27; 63 24 28; 64 26 30
    27. 65 27 31; 66 28 32; 67 34 38; 68 35 39; 69 36 40; 70 38 42; 71 39 43
    28. 72 40 44; 73 42 46; 74 43 47; 75 44 48; 76 50 54; 77 51 55; 78 52 56
    29. 79 54 58; 80 55 59; 81 56 60; 82 58 62; 83 59 63; 84 60 64; 85 18 2
    30. 86 19 3; 87 20 4; 88 22 6; 89 23 7; 90 24 8; 91 26 10; 92 27 11
    31. 93 28 12; 94 30 14; 95 31 15; 96 32 16; 97 34 18; 98 35 19; 99 36 20
    32. 100 38 22; 101 39 23; 102 40 24; 103 42 26; 104 43 27; 105 44 28
    33. 106 46 30; 107 47 31; 108 48 32; 109 50 34; 110 51 35; 111 52 36
    34. 112 54 38; 113 55 39; 114 56 40; 115 58 42; 116 59 43; 117 60 44
```

```
    35. 118 62 46; 119 63 47; 120 64 48
    36. ELEMENT INCIDENCES SHELL
    37. 152 50 34 38 54; 153 51 35 39 55; 154 52 36 40 56; 155 54 38 42 58
    38. 156 55 39 43 59; 157 56 40 44 60; 158 58 42 46 62; 159 59 43 47 63
      STAAD SPACE                                              -- PAGE NO.    2
    39. 160 60 44 48 64; 161 34 18 22 38; 162 35 19 23 39; 163 36 20 24 40
    40. 164 38 22 26 42; 165 39 23 27 43; 166 40 24 28 44; 167 42 26 30 46
    41. 168 43 27 31 47; 169 44 28 32 48; 170 18 2 6 22; 171 19 3 7 23
    42. 172 20 4 8 24; 173 22 6 10 26; 174 23 7 11 27; 175 24 8 12 28
    43. 176 26 10 14 30; 177 27 11 15 31; 178 28 12 16 32
    44. MEMBER PROPERTY AMERICAN
    45. 1 TO 15 22 TO 27 34 TO 48 TABLE ST W14X90
    46. 49 TO 120 TABLE ST W27X84
    47. ELEMENT PROPERTY
    48. 152 TO 178 THICKNESS 0.75
    49. DEFINE MATERIAL START
    50. ISOTROPIC STEEL_50_KSI
    51. E 4.176E+06
    52. POISSON 0.3
    53. DENSITY 0.489024
    54. ALPHA 6.5E-06
    55. DAMP 0.03
    56. TYPE STEEL
    57. STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
    58. ISOTROPIC CONCRETE
    59. E 453600
    60. POISSON 0.17
    61. DENSITY 0.14999
    62. ALPHA 5.5E-06
    63. DAMP 0.05
    64. TYPE CONCRETE
    65. STRENGTH FCU 576
    66. END DEFINE MATERIAL
    67. CONSTANTS
    68. BETA 90 MEMB 13 TO 15 22 TO 27 34 TO 39
    69. MATERIAL STEEL_50_KSI MEMB 1 TO 15 22 TO 27 34 TO 120
    70. MATERIAL CONCRETE MEMB 152 TO 178
    71. SUPPORTS
    72. 1 5 9 13 17 29 33 45 49 53 57 61 FIXED
    73. DEPENDENT ZX CONTROL 22 JOINT YR 15 17
    74. DEPENDENT ZX CONTROL 23 JOINT YR 31 33
    75. DEPENDENT ZX CONTROL 24 JOINT YR 47 49
    76. LOAD 1 LATERAL LOADS
    77. JOINT LOAD
    78. 2 TO 4 14 TO 16 50 TO 52 62 TO 64 FZ 10
    79. 6 TO 8 10 TO 12 18 TO 20 30 TO 32 FZ 20
    80. 34 TO 36 46 TO 48 54 TO 56 58 TO 60 FZ 20
    81. 22 TO 24 26 TO 28 38 TO 40 42 TO 44 FZ 40
    82. LOAD 2 TORSIONAL LOADS
    83. JOINT LOAD
    84. 2 TO 4 50 TO 52 FZ 5
    85. 14 TO 16 62 TO 64 FZ 15
    86. 6 TO 8 18 TO 20 FZ 10
    87. 10 TO 12 30 TO 32 FZ 30
    88. 34 TO 36 54 TO 56 FZ 10
    89. 46 TO 48 58 TO 60 FZ 30
    90. 22 TO 24 38 TO 40 FZ 20
    91. 26 TO 28 42 TO 44 FZ 60
```

```
   92. LOAD 3 DEAD LOAD
   93. ELEMENT LOAD
   94. 152 TO 178 PR GY -1.
     STAAD SPACE                                              -- PAGE NO.    3
   95. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         60  NUMBER OF MEMBERS     108
    NUMBER OF PLATES         27  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     12
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =     153
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    256 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   96. PRINT JOINT DISPLACEMENTS LIST 4 12 20 28 36 44 52 60
 JOINT    DISPLACE LIST     4
     STAAD SPACE                                              -- PAGE NO.    4
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     4    1    0.23216   0.04609   8.13343   0.00108  -0.00056  -0.00008
          2    1.49692   0.04919   6.87507   0.00090  -0.00363  -0.00046
          3    0.02679  -0.19715  -0.32918   0.00792  -0.00041  -0.00625
    12    1    0.23216   0.02166   8.45820   0.00159  -0.00056   0.00014
          2    1.49692   0.02717   8.96788   0.00166  -0.00363  -0.00000
          3    0.02679  -0.86714  -0.09026   0.07454  -0.00041   0.00495
    20    1    0.06978  -0.00054   8.13343   0.00120  -0.00056  -0.00025
          2    0.45052   0.00140   6.87507   0.00103  -0.00363  -0.00031
          3   -0.09267  -0.88243  -0.32918   0.00452  -0.00041  -0.07454
    28    1    0.06978  -0.07793   8.45820  -0.00058  -0.00056   0.00024
          2    0.45052  -0.07823   8.96788  -0.00059  -0.00363   0.00028
          3   -0.09267 -21.50271  -0.09026   0.04716  -0.00041   0.04703
    36    1   -0.09260   0.02065   8.13343   0.00102  -0.00056   0.00030
          2   -0.59589   0.01536   6.87507   0.00088  -0.00363   0.00036
          3   -0.21213  -0.86781  -0.32918  -0.00503  -0.00041  -0.07452
    44    1   -0.09260   0.08469   8.45820  -0.00057  -0.00056  -0.00028
          2   -0.59589   0.08129   8.96788  -0.00059  -0.00363  -0.00031
          3   -0.21213 -21.51369  -0.09026  -0.04712  -0.00041   0.04704
    52    1   -0.25499  -0.06556   8.13343   0.00245  -0.00056  -0.00002
          2   -1.64229  -0.06312   6.87507   0.00207  -0.00363   0.00017
          3   -0.33159  -0.19363  -0.32918  -0.00649  -0.00041  -0.00791
    60    1   -0.25499  -0.02115   8.45820   0.00162  -0.00056  -0.00014
          2   -1.64229  -0.02678   8.96788   0.00167  -0.00363   0.00001
          3   -0.33159  -0.86677  -0.09026  -0.07468  -0.00041   0.00504
  ************** END OF LATEST ANALYSIS RESULT **************
   97. PRINT MEMBER FORCES LIST 115 116
 MEMBER   FORCES   LIST     115
     STAAD SPACE                                              -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KIP  FEET     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
   115    1    58      0.00      7.70     0.00     -0.01      0.00     322.16
               42      0.00     -7.70     0.00      0.01      0.00    -137.43
          2    58      0.00      8.32     0.00     -0.01     -0.00     336.90
               42      0.00     -8.32     0.00      0.01      0.00    -137.14
          3    58      0.00    125.39    -0.00      0.34      0.00    1173.78
```

```
                42      0.00   -125.39     0.00     -0.34      0.00    1835.57
    116    1    59      0.00      4.73     0.00     -0.00      0.00     198.85
                43      0.00     -4.73     0.00      0.00      0.00     -85.39
           2    59      0.00      5.10     0.00     -0.01     -0.00     208.24
                43      0.00     -5.10    -0.00      0.01     -0.00     -85.84
           3    59      0.00    129.34     0.00      0.32      0.00    1407.23
                43      0.00   -129.34    -0.00     -0.32      0.00    1696.94
   ************** END OF LATEST ANALYSIS RESULT **************
    98. PRINT SUPPORT REACTION LIST 9 57
  SUPPORT  REACTION LIST     9
      STAAD SPACE                                             -- PAGE NO.    6
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z    MOM-X     MOM-Y     MOM Z
      9    1     -6.98    -54.60    -54.87   -470.16      0.01     50.68
           2    -28.54    -69.18    -58.55   -500.67      0.03    231.30
           3    -14.10   1732.38     92.25    487.24      0.00     70.67
     57    1      7.65     53.36    -54.77   -469.60      0.01    -55.89
           2     31.74     68.15    -58.52   -500.51      0.03   -257.50
           3    -11.82   1731.54    -91.91   -483.94      0.00     51.07
   ************** END OF LATEST ANALYSIS RESULT **************
    99. FINISH
      STAAD SPACE                                             -- PAGE NO.    7
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:46:14 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *M. To display control nodes* (on page 652)

EX. US-27 Modeling Soil Springs for a Slab on Grade

This example illustrates the usage of commands necessary to apply the compression only attribute to spring supports for a slab on grade. The spring supports themselves are generated utilizing the built-in support generation facility. The slab is subjected to pressure and overturning loading. A tension/compression only analysis of the structure is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-27 Modeling Soil Springs for a Slab on Grade.STD` when you install the program.

The numbers shown in the diagram below are the element numbers.

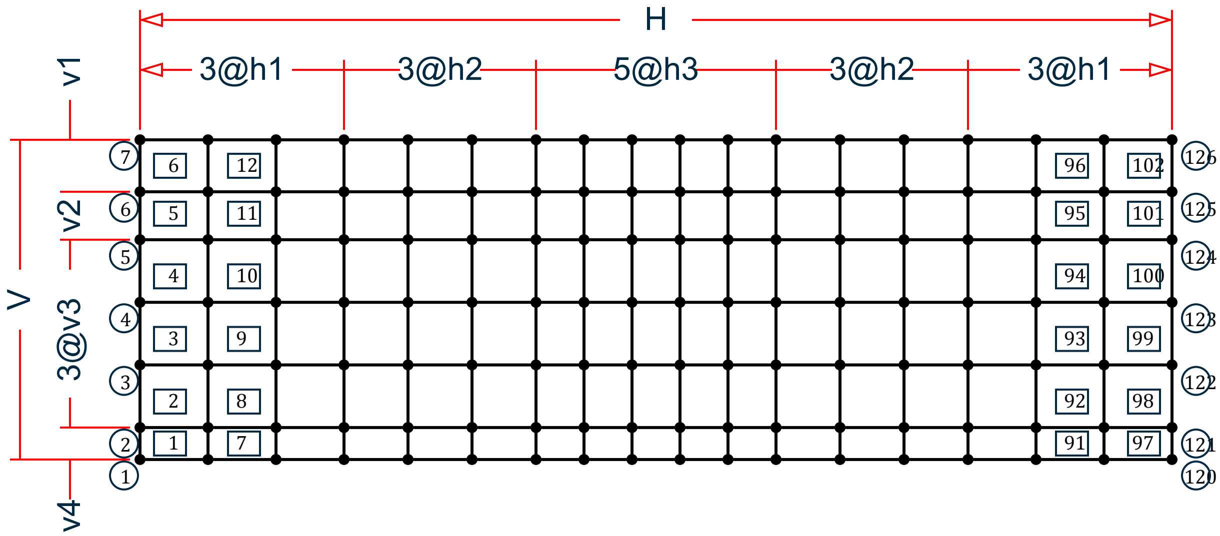


Figure 539: Example Problem No. 27

where:

H = 129', h1 = 8'-6", h2 = 8', h3 = 6'
V = 40', v1 = 6'-6", v2 = 6', v3 = 7'-10", v4 = 4'

```
STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. An optional title to identify this project is provided in the second line.

```
SET NL 3
```

This structure has to be analyzed for 3 primary load cases. Consequently, the modeling of our problem requires us to define 3 sets of data, with each set containing a load case and an associated analysis command. Also, the supports which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have 2 commands: SET NL and CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

```
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0 6.5
7 0.0 0.0 0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
```

For joints 1 through 7, the joint number followed by the X, Y and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these 7 joints by 8.5 feet, 3 times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and this data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first 6 elements is then used to generate the incidences of 96 more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
```

The thickness of elements 1 to 102 is specified as 8.0 inches following the command ELEMENT PROPERTIES.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The modulus of elasticity (E) and Poisson's Ratio are specified following the command CONSTANTS.

```
SPRING COMPRESSION
1 TO 126 KFY
```

The above two lines declare the spring supports at nodes 1 to 126 as having the compression-only attribute. The supports themselves are being generated later (see the ELASTIC MAT command which appears later).

```
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
```

The above command is used to instruct STAAD to generate supports with compression-only springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade reaction of the soil is specified as 12 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the "FY" degree of freedom at the joint. Units for length are changed to FEET to facilitate the input of subgrade reaction of soil. See TR.27.3 Automatic Spring Support Generator for Foundations (on page 2319) .

```
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
```

The above data describe a static load case. A pressure load of $1.50\ kip/ft$ acting in the negative global Y direction is applied on all the elements.

```
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
```

Tension/compression cases must each be followed by `PERFORM ANALYSIS` and `CHANGE` commands. The `CHANGE` command restores the original structure to prepare it for the analysis for the next primary load case.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2  FY -217.
8 9  FY -109.
5    FY -308.7
6    FY -617.4
22 23 FY -410.
29 30 FY -205.
26   FY -542.7
27   FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82   FY -264.2
48 55 76 83 FY -528.3
92 93      FY -205.0
99 100     FY -410.0
103        FY -487.0
104        FY -974.0
113 114    FY -109.0
120 121    FY -217.0
124        FY -273.3
FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
```

Load case 2 consists of several joint loads acting in the negative global Y direction. This is followed by another `ANALYSIS` command. The `CHANGE` command restores the original structure once again for the forthcoming load case.

```
LOAD 3 'COLUMN OVERTURNING LOAD'
ELEMENT LOAD
1 TO 102 PR GY -1.50
JOINT LOADS
1 2  FY -100.
8 9  FY -50.
5    FY -150.7
6    FY -310.4
22 23 FY -205.
29 30 FY -102.
26   FY -271.7
27    FY -542.4
43 44 50 51 71 72 78 79 FY -153.5
47 54 82   FY -132.2
48 55 76 83 FY -264.3
92 93      FY 102.0
99 100     FY 205.0
103        FY 243.0
104        FY 487.0
113 114    FY 54.0
120 121    FY 108.0
124        FY 136.3
125        FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
```

Load case 3 consists of several joint loads acting in the upward direction at one end and downward on the other end to apply an overturning moment that will lift off one end. The CHANGE command is not needed after the last analysis.

```
    LOAD LIST 3
    PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
    PRINT ELEMENT STRESSES LIST 34 67
    PRINT SUPPORT REACTIONS LIST 5 6 12 13
```

A list of joint displacements, element stresses for elements 34 and 67, and support reactions at a list of joints for load case 3, are obtained with the help of the above commands.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE
SET NL 3
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SPRING COMPRESSION
1 TO 126 KFY
UNIT FEET
SUPPORTS
1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 12.0
```

```
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2   FY -217.
8 9   FY -109.
5     FY -308.7
6     FY -617.4
22 23 FY -410.
29 30 FY -205.
26    FY -542.7
27    FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82    FY -264.2
48 55 76 83 FY -528.3
92 93       FY -205.0
99 100      FY -410.0
103         FY -487.0
104         FY -974.0
113 114     FY -109.0
120 121     FY -217.0
124         FY -273.3
125         FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 3 'COLUMN OVERTURNING LOAD'
ELEMENT LOAD
1 TO 102 PR GY -1.50
JOINT LOADS
1 2   FY -100.
8 9   FY -50.
5     FY -150.7
6     FY -310.4
22 23 FY -205.
29 30 FY -102.
26    FY -271.7
27    FY -542.4
43 44 50 51 71 72 78 79 FY -153.5
47 54 82    FY -132.2
48 55 76 83 FY -264.3
92 93       FY 102.0
99 100      FY 205.0
103         FY 243.0
104         FY 487.0
113 114     FY 54.0
120 121     FY 108.0
124         FY 136.3
125         FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
PRINT ELEMENT STRESSES LIST 34 67
PRINT SUPPORT REACTIONS LIST 5 6 12 13
FINISH
```

STAAD Output File

```
                                                                   PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:16                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE SLAB ON GRADE
INPUT FILE: US-27 Modeling Soil Springs for a Slab on Grade.STD
     2. * SPRING COMPRESSION EXAMPLE
     3. SET NL 3
     4. UNIT FEET KIP
     5. JOINT COORDINATES
     6. 1 0.0 0.0 40.0
     7. 2 0.0 0.0 36.0
     8. 3 0.0 0.0 28.167
     9. 4 0.0 0.0 20.333
    10. 5 0.0 0.0 12.5
    11. 6 0.0 0.0  6.5
    12. 7 0.0 0.0  0.0
    13. REPEAT ALL 3 8.5 0.0 0.0
    14. REPEAT 3 8.0 0.0 0.0
    15. REPEAT 5 6.0 0.0 0.0
    16. REPEAT 3 8.0 0.0 0.0
    17. REPEAT 3 8.5 0.0 0.0
    18. ELEMENT INCIDENCES
    19. 1 1 8 9 2 TO 6
    20. REPEAT 16 6 7
    21. UNIT INCH
    22. ELEMENT PROPERTIES
    23. 1 TO 102 TH 8.0
    24. DEFINE MATERIAL START
    25. ISOTROPIC CONCRETE
    26. E 4000.0
    27. POISSON 0.12
    28. DENSITY 8.68E-005
    29. ALPHA 5E-006
    30. DAMP 0.05
    31. G 1346.15
    32. TYPE CONCRETE
    33. STRENGTH FCU 4
    34. END DEFINE MATERIAL
    35. CONSTANTS
    36. MATERIAL CONCRETE ALL
    37. SPRING COMPRESSION
    38. 1 TO 126 KFY
      SLAB ON GRADE                                             -- PAGE NO.    2
    * SPRING COMPRESSION EXAMPLE
    39. UNIT FEET
    40. SUPPORTS
```

```
   41. 1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 12.0
   42. LOAD 1 'WEIGHT OF MAT & EARTH'
   43. ELEMENT LOAD
   44. 1 TO 102 PR GY -1.50
   45. PERFORM ANALYSIS PRINT STATICS CHECK
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
    NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     378
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    SLAB ON GRADE                                           -- PAGE NO.    3
   * SPRING COMPRESSION EXAMPLE
  *** NOTE: CAPACITY FOR MAXIMUM #    253 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
 **NOTE-Tension/Compression converged after  1 iterations, Case=    1
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
         'WEIGHT OF MAT & EARTH'
          CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                        X =  0.645000019E+02
                        Y =  0.000000000E+00
                        Z =  0.200000006E+02
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       -7740.00
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      154800.01  MY=           0.00  MZ=     -499230.03
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =        7740.00
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     -154800.01  MY=           0.00  MZ=      499230.03
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      1)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y = -1.50000E+00       1
     Z =  0.00000E+00       0
     RX=  9.79653E-10     121
     RY=  0.00000E+00       0
     RZ=  4.32018E-10      45
  ************ END OF DATA FROM INTERNAL STORAGE ************
   46. CHANGE
   47. LOAD 2 'COLUMN LOAD-DL+LL'
   48. JOINT LOADS
   49. 1 2   FY -217.
   50. 8 9   FY -109.
   51. 5     FY -308.7
   52. 6     FY -617.4
   53. 22 23 FY -410.
    SLAB ON GRADE                                           -- PAGE NO.    4
   * SPRING COMPRESSION EXAMPLE
```

```
   54. 29 30 FY -205.
   55. 26    FY -542.7
   56. 27    FY -1085.4
   57. 43 44 50 51 71 72 78 79 FY -307.5
   58. 47 54 82    FY -264.2
   59. 48 55 76 83 FY -528.3
   60. 92 93       FY -205.0
   61. 99 100      FY -410.0
   62. 103         FY -487.0
   63. 104         FY -974.0
   64. 113 114     FY -109.0
   65. 120 121     FY -217.0
   66. 124         FY -273.3
   67. 125         FY -546.6
   68. PERFORM ANALYSIS PRINT STATICS CHECK
     SLAB ON GRADE                                            -- PAGE NO.    5
   * SPRING COMPRESSION EXAMPLE
 **NOTE-Tension/Compression converged after  1 iterations, Case=    2
       STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
       'COLUMN LOAD-DL+LL'
        CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
      (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                     X =  0.633725605E+02
                     Y =  0.000000000E+00
                     Z =  0.215722031E+02
TOTAL APPLIED LOAD     2
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =      -13964.90
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      301253.66  MY=           0.00  MZ=     -884991.47
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       13964.90
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     -301253.66  MY=           0.00  MZ=      884991.47
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      2)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y = -1.12956E+01     120
     Z =  0.00000E+00       0
     RX=  8.11932E-02      99
     RY=  0.00000E+00       0
     RZ=  1.00569E-01       6
  ************ END OF DATA FROM INTERNAL STORAGE ************
   69. CHANGE
   70. LOAD 3 'COLUMN OVERTURNING LOAD'
   71. ELEMENT LOAD
   72. 1 TO 102 PR GY -1.50
   73. JOINT LOADS
   74. 1 2   FY -100.
   75. 8 9   FY -50.
   76. 5     FY -150.7
   77. 6     FY -310.4
     SLAB ON GRADE                                            -- PAGE NO.    6
```

```
    * SPRING COMPRESSION EXAMPLE
    78. 22 23 FY -205.
    79. 29 30 FY -102.
    80. 26    FY -271.7
    81. 27    FY -542.4
    82. 43 44 50 51 71 72 78 79 FY -153.5
    83. 47 54 82    FY -132.2
    84. 48 55 76 83 FY -264.3
    85. 92 93       FY 102.0
    86. 99 100      FY 205.0
    87. 103         FY 243.0
    88. 104         FY 487.0
    89. 113 114     FY 54.0
    90. 120 121     FY 108.0
    91. 124         FY 136.3
    92. 125         FY 273.6
    93. PERFORM ANALYSIS PRINT STATICS CHECK
      SLAB ON GRADE                                           -- PAGE NO.    7
    * SPRING COMPRESSION EXAMPLE
*** LOAD CASE     3 -- START ITERATION NO.   2
*** LOAD CASE     3 -- START ITERATION NO.   3
*** LOAD CASE     3 -- START ITERATION NO.   4
*** LOAD CASE     3 -- START ITERATION NO.   5
  **NOTE-Tension/Compression converged after  5 iterations, Case=    3
          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     3
          'COLUMN OVERTURNING LOAD'
           CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
         (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                        X =  0.454460478E+02
                        Y =  0.000000000E+00
                        Z =  0.202712741E+02
 TOTAL APPLIED LOAD     3
   ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     3 )
       SUMMATION FORCE-X =           0.00
       SUMMATION FORCE-Y =      -10533.10
       SUMMATION FORCE-Z =           0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=      213519.36  MY=           0.00  MZ=     -478687.78
 TOTAL REACTION LOAD    3
   ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     3 )
       SUMMATION FORCE-X =           0.00
       SUMMATION FORCE-Y =       10533.10
       SUMMATION FORCE-Z =           0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=     -213519.36  MY=           0.00  MZ=      478687.78
   MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      3)
             MAXIMUMS    AT NODE
      X =  0.00000E+00       0
      Y =  3.06386E+01     120
      Z =  0.00000E+00       0
      RX= -1.35247E-01     120
      RY=  0.00000E+00       0
      RZ=  1.12342E-01     125
   ************ END OF DATA FROM INTERNAL STORAGE ************
    94. LOAD LIST 3
    95. PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
  JOINT    DISPLACE LIST     113
      SLAB ON GRADE                                           -- PAGE NO.    8
```

```
   * SPRING COMPRESSION EXAMPLE
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
   113    3    0.00000  20.81368   0.00000  -0.10277   0.00000   0.07382
   114    3    0.00000  15.82565   0.00000  -0.10134   0.00000   0.06799
   120    3    0.00000  30.63862   0.00000  -0.13525   0.00000   0.10711
   121    3    0.00000  24.13106   0.00000  -0.12907   0.00000   0.09243
  ************** END OF LATEST ANALYSIS RESULT **************
   96. PRINT ELEMENT STRESSES LIST 34 67
 ELEMENT  STRESSES LIST     34
     SLAB ON GRADE                                            -- PAGE NO.    9
   * SPRING COMPRESSION EXAMPLE
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
  ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                     VONT       VONB         SX           SY           SXY
                     TRESCAT    TRESCAB
    34      3        -4.66       -6.79        2.34         7.95         5.74
                    164.71      164.71        0.00         0.00         0.00
                    172.44      172.44
      TOP : SMAX=     155.71 SMIN=     -16.73 TMAX=      86.22 ANGLE= 58.0
      BOTT: SMAX=      16.73 SMIN=    -155.71 TMAX=      86.22 ANGLE=-32.0
    67      3        32.82        6.55      -63.47         5.72        36.48
                   1238.45     1238.45        0.00         0.00         0.00
                   1357.37     1357.37
      TOP : SMAX=     288.91 SMIN=   -1068.46 TMAX=     678.69 ANGLE= 66.7
      BOTT: SMAX=    1068.46 SMIN=    -288.91 TMAX=     678.69 ANGLE=-23.3
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         1.068459E+03 -1.068459E+03  6.786865E+02  1.238454E+03  1.357373E+03
PLATE NO.      67            67            67            67            67
CASE  NO.       3             3             3             3             3
  ********************END OF ELEMENT FORCES********************
   97. PRINT SUPPORT REACTIONS LIST 5 6 12 13
 SUPPORT  REACTION LIST     5
     SLAB ON GRADE                                            -- PAGE NO.   10
   * SPRING COMPRESSION EXAMPLE
  SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     5    3      0.00    149.37      0.00      0.00      0.00      0.00
     6    3      0.00    170.57      0.00      0.00      0.00      0.00
    12    3      0.00    148.42      0.00      0.00      0.00      0.00
    13    3      0.00    152.40      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   98. FINISH
     SLAB ON GRADE                                            -- PAGE NO.   11
   * SPRING COMPRESSION EXAMPLE
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46:17 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
```

```
        *   Details about additional assistance from              *
        *   Bentley and Partners can be found at program menu     *
        *   Help->Technical Support                               *
        *                                                         *
        *           Copyright (c) Bentley Systems, Inc.           *
        *                http://www.bentley.com                   *
        ***********************************************************
```

EX. US-28 Calculation of Modes and Frequencies of a Bridge

This example demonstrates the input required for obtaining the modes and frequencies of the skewed bridge shown in the figure below. The structure consists of piers, pier-cap girders and a deck slab.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-28 Calculation of Modes and Frequencies of a Bridge.std` when you install the program.

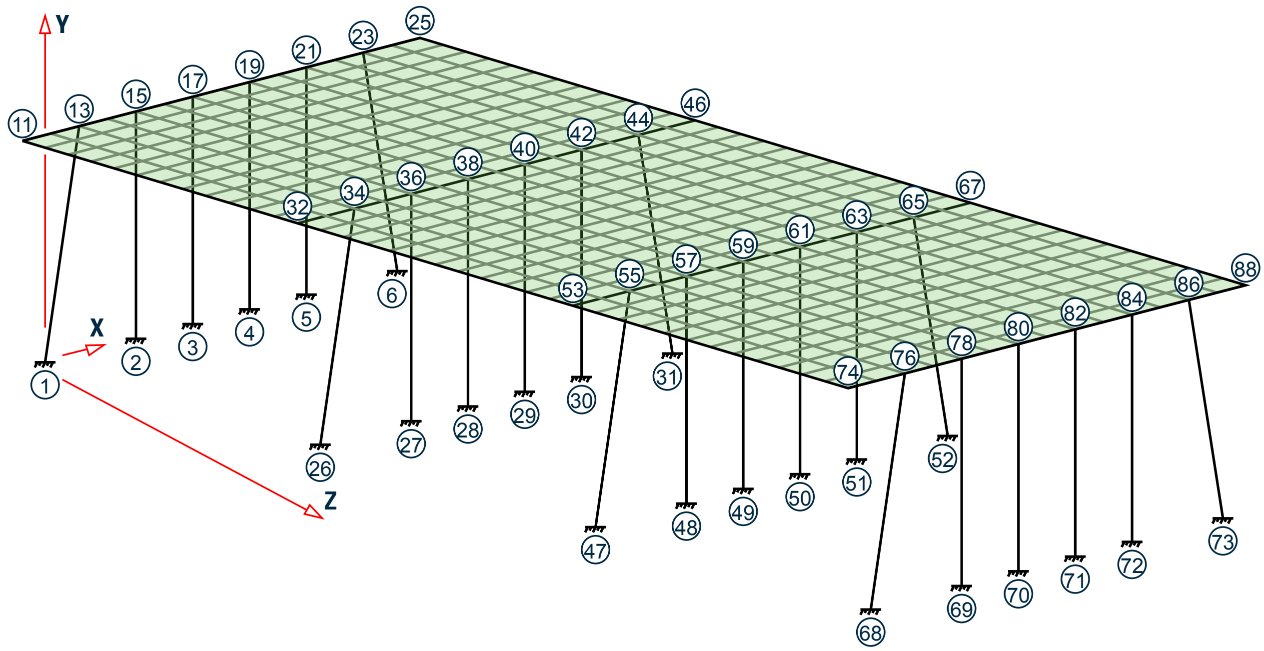


Figure 540: Example Problem No. 28

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words forms a title to identify this project.

```
IGNORE LIST
```

Further below in this file, we will call element lists in which some element numbers may not actually be present in the structure. We do so because it minimizes the effort involved in fetching the desired elements and reduces the size of the respective commands. To prevent the program from treating that condition (referring to elements which do not exist) as an error, the above command is required.

```
UNIT METER KN
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
```

For joints 1 through 6, the joint number followed by the X, Y and Z coordinates are specified first.

Next, using the coordinates of joints 11 and 25 as the basis, joints 12 through 24 are generated using linear interpolation.

Following this, using the data of these 21 joints (1 through 6 and 11 through 25), 63 new joints are generated. To achieve this, the X coordinate of these 21 joints is incremented by 4 meters and the Z coordinate is incremented by 14 meters, in 3 successive operations.

The `REPEAT ALL` command is used for the generation. Details of this command is available in TR.11 Joint Coordinates Specification (on page 2217) . The results of the generation may be visually verified using STAAD.Pro's graphical viewing facilities.

```
MEMBER INCI
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
```

The member connectivity data (joint numbers between which members are connected) is specified for the 24 columns for the structure. The above method, where the member number is followed by the 2 node numbers, is the explicit definition method. No generation is involved here.

```
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
```

The member connectivity data is specified for the pier cap beams for the structure. The above method is a combination of explicit definition and generation. For example, member 101 is defined as connected between 11 & 12. Then, by incrementing those nodes by 1 unit at a time (which is the default increment), the incidences of members 102 to 114 are generated. Similarly, we create members 202 to 215, 303 to 316, and, 404 to 417.

```
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
```

The next step is to generate the deck slab which will be modeled using plate elements. For this, we use a technique called mesh generation. Mesh generation is a process of generating several "child" elements from a "parent" or "super" element. The above set of commands defines the corner nodes of the super-element. Details of the above can be found in TR.14.2 Element Mesh Generation (on page 2229).

Note that instead of elaborately defining the coordinates of the corner nodes of the super-elements, we have taken advantage of the fact that the coordinates of these joints (A through H) have already been defined or generated earlier. Thus, A is the same as joint 11 while D is the same as joint 32. Alternatively, we could have defined the super-element nodes as A -1 10 0 ; B 16.5 10 0 ; C 20.5 10 14 ; D 3 10 14 ; etc.

```
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEGH 14 12
```

The above lines are the instructions for generating the "child" elements from the super-elements. For example, from the super-element bound by the corners A, B, C and D (which in turn are nodes 11, 25, 46 and 32), we generate a total of 14X12=168 elements, with 14 divisions along the edges AB and CD, and 12 along the edges BC and DA. These are the elements which make up the first span.

Similarly, 168 elements are created for the 2nd span, and another 168 for the 3rd span.

It may be noted here that we have taken great care to ensure that the resulting elements and the piercap beams form a perfect fit. In other words, there is no overlap between the two in a manner that nodes of the beams are at a different point in space than nodes of elements. At every node along their common boundary, plates and beams are properly connected. This is absolutely essential to ensure proper transfer of load and stiffness from beams to plates and vice versa. The tools in the user interface may be used to confirm that beam-plate connectivity is proper for this model.

```
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
```

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the piercap beams are being grouped to a name called GIRDERS, the pier columns are assigned the name PIERS, and so on. For the deck, a few selected elements are chosen into a few selective groups. The reason is that these elements happen to be right beneath wheels of vehicles whose weight will be used in the frequency calculation.

```
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
```

Member properties are assigned as prismatic rectangular sections for the girders, and prismatic circular sections for the columns.

```
ELEMENT PROPERTY
YRA 9 11 TH 0.375
```

The plate elements of the deck slab, which happen to be at a Y elevation of 10 metres (between a `YRANGE` of 9 metres and 11 metres) are assigned a thickness of 375 mms.

```
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
```

```
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The Modulus of elasticity (E) is set to 21000 N/sq.mm for all members. The keyword `CONSTANTS` has to precede this data. Built-in default value for Poisson's ratio for concrete is also assigned to ALL members and elements.

```
UNIT KNS METER
CONSTANTS
DENSITY 24 ALL
```

Following a change of units, density of concrete is specified.

```
SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
```

The base nodes of the piers are fully restrained (FIXED supports).

```
CUT OFF MODE SHAPE 65
```

Theoretically, a structure has as many modes of vibration as the number of degrees of freedom in the model. However, the limitations of the mathematical process used in extracting modes may limit the number of modes that can actually be extracted. In a large structure, the extraction process can also be very time consuming. Further, not all modes are of equal importance. (One measure of the importance of modes is the participation factor of that mode.) In many cases, the first few modes may be sufficient to obtain a significant portion of the total dynamic response.

Due to these reasons, in the absence of any explicit instruction, STAAD calculates only the first 6 modes. This is like saying that the command `CUT OFF MODE SHAPE 6` has been specified.

If the inspection of the first 6 modes reveals that the overall vibration pattern of the structure has not been obtained, one may ask STAAD to compute a larger (or smaller) number of modes with the help of this command. The number that follows this command is the number of modes being requested. In our example, we are asking for 65 modes by specifying `CUT OFF MODE SHAPE 65`.

```
UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
```

```
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700
```

The mathematical method that STAAD uses is called the eigen extraction method. Some information on this is available in G.17.3 Dynamic Analysis (on page 2154) . The method involves 2 matrices - the stiffness matrix, and the mass matrix.

The stiffness matrix, usually called the [K] matrix, is assembled using data such as member and element lengths, member and element properties, modulus of elasticity, Poisson's ratio, member and element releases, member offsets, support information, etc.

For assembling the mass matrix, called the [M] matrix, STAAD uses the load data specified in the load case in which the MODAL CAL REQ command is specified. So, some of the important aspects to bear in mind are :

1. The input you specify is weights, not masses. Internally, STAAD will convert weights to masses by dividing the input by "g", the acceleration due to gravity.
2. If the structure is declared as a PLANE frame, there are 2 possible directions of vibration - global X, and global Y. If the structure is declared as a SPACE frame, there are 3 possible directions - global X, global Y and global Z. However, this does not guarantee that STAAD will automatically consider the masses for vibration in all the available directions.

   You have control over and are responsible for specifying the directions in which the masses ought to vibrate. In other words, if a weight is not specified along a certain direction, the corresponding degrees of freedom (such as for example, global X at node 34 hypothetically) will not receive a contribution in the mass matrix. The mass matrix is assembled using only the masses from the weights and directions you specify.

   In our example, notice that we are specifying the selfweight along global X, Y and Z directions. Similarly, a 200 kg/sq.m pressure load is also specified along all 3 directions on the deck.

   But for the truck loads, we choose to apply it on just a few elements in the global Y and Z directions only. The reasoning is something like - for the X direction, the mass is not capable of vibrating because the tires allow the truck to roll along X. Remember, this is just a demonstration example, not necessarily what you may want to do.

   The point we want to illustrate is that if you want to restrict a certain weight to certain directions only, all you need to do is not provide the directions in which those weights cannot vibrate in.
3. As much as possible, provide absolute values for the weights. STAAD is programmed to algebraically add the weights at nodes. So, if some weights are specified as positive numbers and others as negative, the total weight at a given node is the algebraic summation of all the weights in the global directions at that node and the mass is then derived from this algebraic resultant.

```
MODAL CALCULATION REQUESTED
```

This is the command which tells the program that frequencies and modes should be calculated. It is specified inside a load case. In other words, this command accompanies the loads that are to be used in generating the mass matrix.

Frequencies and modes have to be calculated also when dynamic analysis such as response spectrum or time history analysis is carried out. But in such analyses, the `MODAL CALCULATION REQUESTED` command is not explicitly required. When STAAD encounters the commands for response spectrum (see example 11) and time history (see examples 16 and 22), it automatically will carry out a frequency extraction without the help of the MODAL .. command.

```
PERFORM ANALYSIS
```

This initiates the processes which are required to obtain the frequencies. Frequencies, periods and participation factors are automatically reported in the output file when the operation is completed.

```
    FINISH
```

This terminates the STAAD run.

Input File

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
IGNORE LIST
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
MEMBER INCI
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEGH 14 12
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
ELEMENT PROPERTY
YRA 9 11 TH 0.375
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
```

```
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
CUT OFF MODE SHAPE 65
UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output File

```
                                                                   PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:19                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
INPUT FILE: US-28 Calculation of Modes and Frequencies of a Bridge.STD
     2. IGNORE LIST
```

```
 3. UNIT METER KN
 4. JOINT COORDINATES
 5. 1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0
 6. 11 -1 10 0 25 16.5 10 0
 7. REPEAT ALL 3 4 0 14
 8. MEMBER INCI
 9. 1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
10. 26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
11. 47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
12. 68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
13. 101 11 12 114
14. 202 32 33 215
15. 303 53 54 316
16. 404 74 75 417
17. DEFINE MESH
18. A JOINT 11
19. B JOINT 25
20. C JOINT 46
21. D JOINT 32
22. E JOINT 67
23. F JOINT 53
24. G JOINT 88
25. H JOINT 74
26. GENERATE ELEMENT
27. MESH ABCD 14 12
28. MESH DCEF 14 12
29. MESH FEGH 14 12
30. START GROUP DEFINITION
31. MEMBER
32. _GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
33. _PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
34. ELEMENT
35. _P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
36. _P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
37. _P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
38. _P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
  FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    2
39. _P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
40. _P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
41. END GROUP DEFINITION
42. MEMBER PROPERTY
43. _GIRDERS PRIS YD 0.6 ZD 0.6
44. _PIERS PRIS YD 1.0
45. ELEMENT PROPERTY
46. YRA 9 11 TH 0.375
47. UNIT KNS MMS
48. DEFINE MATERIAL START
49. ISOTROPIC CONCRETE
50. E 21.0
51. POISSON 0.17
52. DENSITY 2.36158E-008
53. ALPHA 5E-006
54. DAMP 0.05
55. G 9.25
56. TYPE CONCRETE
57. STRENGTH FCU 0.0275
58. END DEFINE MATERIAL
59. CONSTANTS
```

```
   60. MATERIAL CONCRETE ALL
   61. SUPPORTS
   62. 1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
   63. CUT OFF MODE SHAPE 65
   64. UNIT KGS METER
   65. LOAD 1 FREQUENCY CALCULATION
   66. SELFWEIGHT X 1.0
   67. SELFWEIGHT Y 1.0
   68. SELFWEIGHT Z 1.0
   69. * PERMANENT WEIGHTS ON DECK
   70. ELEMENT LOAD
   71. YRA 9 11 PR GX 200
   72. YRA 9 11 PR GY 200
   73. YRA 9 11 PR GZ 200
   74. * VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
   75. ELEMENT LOAD
   76. _P1 PR GY 700
   77. _P2 PR GY 700
   78. _P3 PR GY 700
   79. _P4 PR GY 700
   80. _P5 PR GY 700
   81. _P6 PR GY 700
   82. _P1 PR GZ 700
   83. _P2 PR GZ 700
   84. _P3 PR GZ 700
   85. _P4 PR GZ 700
   86. _P5 PR GZ 700
   87. _P6 PR GZ 700
   88. MODAL CALCULATION REQUESTED
   89. PERFORM ANALYSIS
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    3
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS        579  NUMBER OF MEMBERS      80
    NUMBER OF PLATES        504  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     24
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =    3330
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED              =    65
NUMBER OF EXISTING MASSES IN THE MODEL =  1665
NUMBER OF MODES THAT WILL BE USED      =    65
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    4
              CALCULATED FREQUENCIES FOR LOAD CASE       1
      MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
```

```
       1                      1.648                  0.60681
       2                      2.622                  0.38132
       3                      2.906                  0.34409
       4                      3.783                  0.26436
       5                      4.108                  0.24345
       6                      4.423                  0.22608
       7                      4.561                  0.21927
       8                      4.725                  0.21162
       9                      5.080                  0.19684
      10                      7.277                  0.13742
      11                      7.328                  0.13647
      12                      7.454                  0.13416
      13                     10.418                  0.09599
      14                     10.818                  0.09244
      15                     11.260                  0.08881
      16                     11.377                  0.08790
      17                     11.672                  0.08567
      18                     11.945                  0.08372
      19                     12.028                  0.08314
      20                     12.209                  0.08191
      21                     12.619                  0.07925
      22                     13.823                  0.07234
      23                     14.807                  0.06754
      24                     14.920                  0.06702
      25                     15.294                  0.06539
      26                     17.489                  0.05718
      27                     17.664                  0.05661
      28                     17.937                  0.05575
      29                     19.923                  0.05019
      30                     20.116                  0.04971
      31                     20.724                  0.04825
      32                     20.817                  0.04804
      33                     21.024                  0.04756
      34                     21.340                  0.04686
      35                     21.633                  0.04623
      36                     22.002                  0.04545
      37                     22.290                  0.04486
      38                     23.393                  0.04275
      39                     23.738                  0.04213
      40                     24.235                  0.04126
      41                     24.881                  0.04019
      42                     25.690                  0.03893
      43                     26.275                  0.03806
      44                     26.700                  0.03745
      45                     27.124                  0.03687
      46                     27.610                  0.03622
      47                     28.063                  0.03563
      48                     29.272                  0.03416
      49                     29.866                  0.03348
      50                     30.118                  0.03320
      51                     31.309                  0.03194
    FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    5
             CALCULATED FREQUENCIES FOR LOAD CASE       1
     MODE             FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
      52                     31.832                  0.03142
      53                     32.014                  0.03124
      54                     32.312                  0.03095
      55                     32.906                  0.03039
```

```
    56                    33.204                0.03012
    57                    34.467                0.02901
    58                    35.270                0.02835
    59                    35.522                0.02815
    60                    35.766                0.02796
    61                    36.318                0.02753
    62                    36.919                0.02709
    63                    38.921                0.02569
    64                    39.189                0.02552
    65                    39.905                0.02506
      MODAL WEIGHT (MODAL MASS TIMES g) IN KGS          GENERALIZED
  MODE           X             Y             Z             WEIGHT
     1       1.291934E+02  1.123418E+00  1.185768E+06    1.205497E+06
     2       1.089784E+06  2.961298E+00  2.290601E+02    1.083499E+06
     3       2.111306E-01  2.767724E+03  2.799804E+00    5.416754E+05
     4       1.488365E+00  3.916563E+04  6.302102E+00    1.988939E+05
     5       2.650657E-01  4.869465E+02  5.916391E+02    1.348431E+05
     6       5.553985E+02  7.047738E+02  2.743600E+02    8.887365E+04
     7       1.872127E+01  3.523236E+05  3.255370E+00    7.402632E+04
     8       5.441681E+00  2.711803E+05  8.530684E+00    7.667142E+04
     9       5.838996E+03  1.713501E+03  2.233780E+03    6.923635E+04
    10       4.436158E+00  1.508718E+03  4.951104E-01    4.324923E+04
    11       4.522846E+00  6.614279E+02  1.224734E+00    4.305570E+04
    12       2.322126E-01  4.156161E+02  7.878947E-01    3.944772E+04
    13       3.968726E+01  2.882564E+00  6.818221E+03    1.532777E+05
    14       2.085091E-01  1.298117E+02  1.097372E+02    7.154577E+04
    15       2.752974E-02  4.303212E+03  1.232103E+02    4.541365E+04
    16       7.726809E+00  7.931786E+01  1.710543E+02    5.083780E+04
    17       9.187165E+00  2.295488E+00  5.600056E-01    5.231609E+04
    18       1.289274E+02  1.227798E+01  1.533801E+02    8.841806E+04
    19       3.797408E-01  3.151484E+01  4.203883E+00    4.601297E+04
    20       9.958856E+00  7.524730E+01  1.835670E+01    7.466321E+04
    21       8.006146E-01  2.974809E+00  4.620779E+00    7.378575E+04
    22       1.242891E+02  1.356224E-01  1.727536E+00    4.087678E+05
    23       6.938982E+00  4.623798E+01  8.257328E+01    6.864460E+04
    24       4.653175E-01  1.849773E+02  6.876842E+00    5.597259E+04
    25       8.352727E+00  1.093998E+01  2.047659E+02    4.884299E+04
    26       3.879229E-01  2.187085E+03  5.145218E-03    2.890765E+04
    27       8.086885E+00  3.229583E+02  9.429697E-01    3.202062E+04
  FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    6
    28       2.443751E+00  1.315502E+04  2.609740E-01    2.997395E+04
    29       6.642359E+02  6.512258E+03  5.785467E+01    3.489561E+04
    30       3.896515E+01  4.236802E+04  5.583570E+00    3.273818E+04
    31       2.124308E+02  2.362562E+02  6.791406E-01    2.856038E+04
    32       2.501307E+01  4.154716E+04  1.610248E-03    3.529900E+04
    33       9.944393E+00  2.188227E+03  7.002218E-01    3.488091E+04
    34       1.354775E+01  1.273812E+02  9.929674E+00    3.662762E+04
    35       2.684736E+00  5.035591E+03  2.202823E+00    4.255654E+04
    36       3.584198E-01  7.349881E+00  1.032835E+00    5.119778E+04
    37       1.674724E+00  2.647461E+04  8.112263E-01    4.340583E+04
    38       1.234580E+00  8.274586E+01  1.831619E-02    1.919433E+04
    39       1.177019E-02  3.952054E+02  4.526474E-01    1.820400E+04
    40       2.510682E-03  2.913923E+01  2.643008E-04    4.236454E+05
    41       1.752055E+00  1.750469E+03  1.098303E+01    4.948152E+04
    42       3.172853E-01  5.576266E+04  1.029036E-01    7.029313E+04
    43       4.088524E+00  7.230284E+02  1.647904E+00    7.840900E+04
    44       6.792844E-03  6.810530E+02  2.323051E-02    7.748353E+04
    45       1.286032E-02  2.969021E+02  6.276731E-04    1.698166E+05
```

```
        46       3.915110E-01  1.842050E+03  1.016859E-02    2.296592E+04
        47       1.222720E+01  2.250049E+00  1.763239E+00    1.408108E+04
        48       5.796924E-03  4.479735E+03  7.592468E-02    2.885324E+04
        49       3.062960E+00  1.622740E+01  2.516519E-01    4.734347E+04
        50       5.381654E-01  2.109875E-02  3.715996E+00    4.275013E+04
        51       6.988027E-03  1.414464E+02  1.037723E-01    2.721877E+04
        52       3.120314E-01  2.679203E+00  8.601323E-01    4.346363E+04
        53       7.712285E-01  1.227296E+03  2.159809E-01    3.644744E+04
        54       1.092693E+01  6.309261E+01  4.335824E+00    4.837668E+04
        55       4.389927E-01  2.654929E+02  5.449158E+00    2.579368E+04
        56       1.697410E-01  8.013763E+02  1.519167E-02    3.565397E+04
        57       5.812394E-01  1.003674E+02  1.297763E+00    3.819235E+04
        58       6.155775E+00  1.623267E+03  1.175780E+01    4.182908E+04
        59       3.892790E+00  1.515229E+01  1.764828E+01    3.953446E+04
        60       7.019893E+00  5.144657E+02  4.912786E+01    5.200588E+04
        61       5.075761E+01  1.707431E+03  3.625510E+01    1.190568E+04
        62       2.640553E+01  3.154571E+03  1.681321E+01    1.455738E+04
        63       1.733228E-01  5.382845E+02  4.096147E-01    7.325642E+04
        64       1.024425E-01  3.773902E+03  1.746056E-01    6.231664E+04
        65       7.508947E+00  2.059276E+03  1.501086E+01    3.298603E+04
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    7
 MASS PARTICIPATION FACTORS
                          MASS  PARTICIPATION FACTORS IN PERCENT
                          --------------------------------------
              MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z
                 1     0.01   0.00  99.04    0.012    0.000   99.042
                 2    99.14   0.00   0.02   99.152    0.000   99.061
                 3     0.00   0.23   0.00   99.152    0.232   99.061
                 4     0.00   3.27   0.00   99.152    3.503   99.062
                 5     0.00   0.04   0.05   99.152    3.544   99.111
                 6     0.05   0.06   0.02   99.202    3.602   99.134
                 7     0.00  29.43   0.00   99.204   33.030   99.134
                 8     0.00  22.65   0.00   99.205   55.681   99.135
                 9     0.53   0.14   0.19   99.736   55.824   99.322
                10     0.00   0.13   0.00   99.736   55.950   99.322
                11     0.00   0.06   0.00   99.737   56.005   99.322
                12     0.00   0.03   0.00   99.737   56.040   99.322
                13     0.00   0.00   0.57   99.740   56.040   99.891
                14     0.00   0.01   0.01   99.740   56.051   99.901
                15     0.00   0.36   0.01   99.740   56.411   99.911
                16     0.00   0.01   0.01   99.741   56.417   99.925
                17     0.00   0.00   0.00   99.742   56.417   99.925
                18     0.01   0.00   0.01   99.754   56.418   99.938
                19     0.00   0.00   0.00   99.754   56.421   99.938
                20     0.00   0.01   0.00   99.754   56.427   99.940
                21     0.00   0.00   0.00   99.755   56.428   99.940
                22     0.01   0.00   0.00   99.766   56.428   99.940
                23     0.00   0.00   0.01   99.766   56.431   99.947
                24     0.00   0.02   0.00   99.767   56.447   99.948
                25     0.00   0.00   0.02   99.767   56.448   99.965
                26     0.00   0.18   0.00   99.767   56.630   99.965
                27     0.00   0.03   0.00   99.768   56.657   99.965
                28     0.00   1.10   0.00   99.768   57.756   99.965
                29     0.06   0.54   0.00   99.829   58.300   99.970
                30     0.00   3.54   0.00   99.832   61.839   99.970
                31     0.02   0.02   0.00   99.852   61.859   99.971
                32     0.00   3.47   0.00   99.854   65.329   99.971
                33     0.00   0.18   0.00   99.855   65.512   99.971
```

```
          34     0.00   0.01   0.00   99.856   65.522   99.971
          35     0.00   0.42   0.00   99.856   65.943   99.972
          36     0.00   0.00   0.00   99.856   65.944   99.972
          37     0.00   2.21   0.00   99.856   68.155   99.972
          38     0.00   0.01   0.00   99.857   68.162   99.972
          39     0.00   0.03   0.00   99.857   68.195   99.972
          40     0.00   0.00   0.00   99.857   68.197   99.972
          41     0.00   0.15   0.00   99.857   68.343   99.973
          42     0.00   4.66   0.00   99.857   73.001   99.973
          43     0.00   0.06   0.00   99.857   73.061   99.973
          44     0.00   0.06   0.00   99.857   73.118   99.973
          45     0.00   0.02   0.00   99.857   73.143   99.973
          46     0.00   0.15   0.00   99.857   73.297   99.973
          47     0.00   0.00   0.00   99.858   73.297   99.973
          48     0.00   0.37   0.00   99.858   73.671   99.973
          49     0.00   0.00   0.00   99.859   73.673   99.973
          50     0.00   0.00   0.00   99.859   73.673   99.973
          51     0.00   0.01   0.00   99.859   73.685   99.973
          52     0.00   0.00   0.00   99.859   73.685   99.973
          53     0.00   0.10   0.00   99.859   73.787   99.973
          54     0.00   0.01   0.00   99.860   73.793   99.974
          55     0.00   0.02   0.00   99.860   73.815   99.974
          56     0.00   0.07   0.00   99.860   73.882   99.974
    FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    8
                   MASS  PARTICIPATION FACTORS IN PERCENT
                   --------------------------------------
         MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
          57     0.00   0.01   0.00   99.860   73.890   99.974
          58     0.00   0.14   0.00   99.860   74.026   99.975
          59     0.00   0.00   0.00   99.861   74.027   99.977
          60     0.00   0.04   0.00   99.861   74.070   99.981
          61     0.00   0.14   0.00   99.866   74.212   99.984
          62     0.00   0.26   0.00   99.868   74.476   99.985
          63     0.00   0.04   0.00   99.868   74.521   99.985
          64     0.00   0.32   0.00   99.868   74.836   99.985
          65     0.00   0.17   0.00   99.869   75.008   99.987
  90. FINISH
    FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    9
          *********** END OF THE STAAD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:46:22 ****
      ************************************************************
      *   For technical assistance on STAAD.Pro, please visit    *
      *   http://www.bentley.com/en/support/                     *
      *                                                          *
      *   Details about additional assistance from               *
      *   Bentley and Partners can be found at program menu      *
      *   Help->Technical Support                                *
      *                                                          *
      *           Copyright (c) Bentley Systems, Inc.            *
      *                http://www.bentley.com                    *
      ************************************************************
```

Understanding the output

After the analysis is complete, look at the output file. (This file can be viewed by selecting the **Analysis Output** tool in the **View** group on the **Utilities** ribbon tab).

i. Mode number and corresponding frequencies and periods

Since we asked for 65 modes, we obtain a report, a portion of which is as shown:

Table 910: Calculated Frequencies for Load Case 1

Mode	Frequency (Cycles/Sec)	Period (Sec)	Accuracy
1	1.636	0.61111	1.344E-16
2	2.602	0.38433	0.000E+00
3	2.882	0.34695	8.666E-16
4	3.754	0.26636	0.000E+00
5	4.076	0.24532	3.466E-16
6	4.373	0.22870	6.025E-16
7	4.519	0.22130	5.641E-16
8	4.683	0.21355	5.253E-16
9	5.028	0.19889	0.000E+00
10	7.189	0.13911	8.916E-16
11	7.238	0.13815	0.000E+00
12	7.363	0.13582	0.000E+00

ii. Participation factors in Percentage

Table 911: Mass Participation Factors in Percent

Mode	X	Y	Z	ΣX	ΣY	ΣZ
1	0.01	0.00	99.04	0.012	0.000	99.042
2	99.14	0.00	0.02	99.151	0.000	99.061
3	0.00	0.23	0.00	99.151	0.229	99.062
4	0.00	3.27	0.00	99.151	3.496	99.062
5	0.00	0.04	0.05	99.151	3.536	99.112
6	0.05	0.04	0.02	99.202	3.575	99.135
7	0.00	26.42	0.00	99.204	30.000	99.135
8	0.00	25.59	0.00	99.204	55.587	99.136

Mode	X	Y	Z	ΣX	ΣY	ΣZ
9	0.53	0.15	0.19	99.735	55.740	99.326
10	0.00	0.13	0.00	99.736	55.871	99.326
11	0.00	0.06	0.00	99.736	55.927	99.326
12	0.00	0.04	0.00	99.736	55.969	99.326

In the explanation earlier for the `CUT OFF MODE` command, we said that one measure of the importance of a mode is the participation factor of that mode. We can see from the above report that for vibration along Z direction, the first mode has a 99.04 percent participation. It is also apparent that the 7th mode is primarily a Y direction mode with a 26.42% participation along Y and 0 in X and Z.

The ΣX, ΣY and ΣZ columns show the cumulative value of the participation of all the modes up to and including a given mode (Corresponding to the SUMM-X, SUMM-Y, and SUMM-Z reported in the output, respectively). One can infer from those terms that if one is interested in 95% participation along X, the first 2 modes are sufficient.

But for the Y direction, even with 10 modes, we barely obtained 60%. The reason for this can be understood by an examination of the nature of the structure. The deck slab is capable of vibrating in several low energy and primarily vertical direction modes. The out-of-plane flexible nature of the slab enables it to vibrate in a manner resembling a series of wave like curves. Masses on either side of the equilibrium point have opposing eigenvector values leading to a lot of cancellation of the contribution from the respective masses. Localized modes, where small pockets in the structure undergo flutter due to their relative weak stiffness compared to the rest of the model, also result in small participation factors.

iii. After the analysis is completed, select Post-processing from the mode menu. This screen contains facilities for graphically examining the shape of the mode in static and animated views. The Dynamics page on the left side of the screen is available for viewing the shape of the mode statically. The Animation option of the Results menu can be used for animating the mode. The mode number can be selected from the **Loads and Results** tab of the **Diagrams** dialog box which opens when the Animation option is chosen. The size to which the mode is drawn is controlled using the **Scales** tab of the **Diagrams** dialog box.

Related Links

- *M. To calculate the structure frequency* (on page 843)
- *TR.34.2 Modal Calculation Command* (on page 2615)

EX. US-29 Time History Analysis of a Frame for Seismic Loads

Analysis and design of a structure for seismic loads is demonstrated in this example. In this model, static load cases are solved along with the seismic load case. For the seismic case, the maximum values of displacements, forces and reactions are obtained. The results of the dynamic case are combined with those of the static cases and steel design is performed on the combined cases.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\US-29 Time History Analysis of a Frame for Seismic Loads.std` when you install the program.

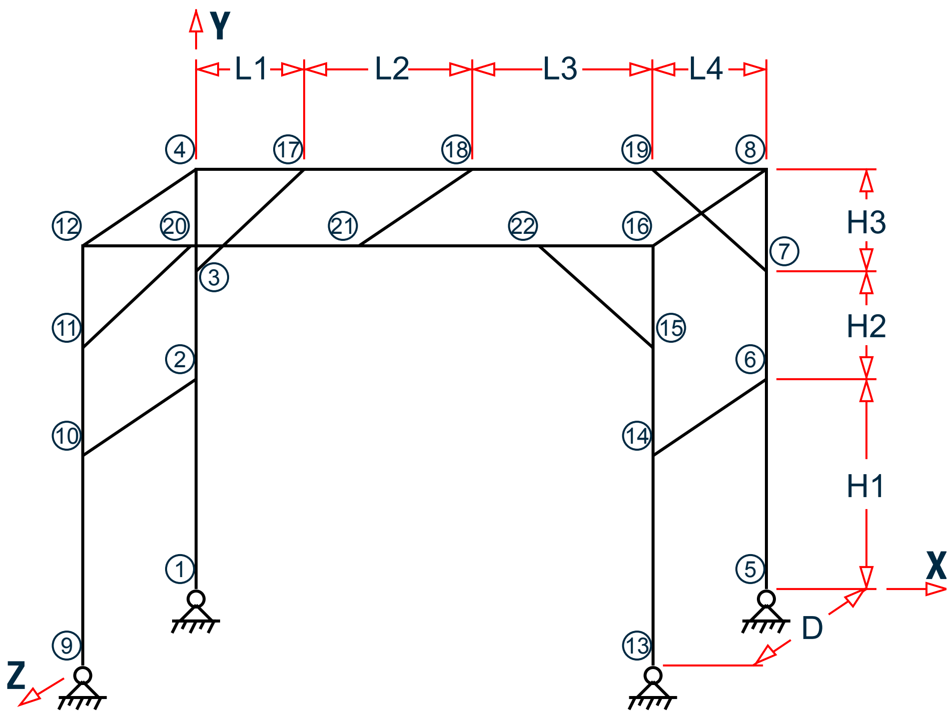


Figure 541: Example Problem No. 29

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words form a title to identify this project.

```
UNIT METER KNS
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
```

For joints 1 through 4, the joint number is followed by the X, Y and Z coordinates as specified above. The coordinates of these joints are used as a basis for generating 12 more joints by incrementing the X & Z coordinates by specific amounts. REPEAT ALL commands are used for the generation. Details of these commands are available in TR.11 Joint Coordinates Specification (on page 2217). Following this, another round of explicit definition (joints 17, 18 & 19) and generation (20, 21 & 22) is carried out. The results of the generation may be visually verified using graphical view features in STAAD.Pro.

```
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
```

```
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
```

A mixture of explicit definition and generation of member connectivity data (joint numbers between which members are connected) is used to generate 29 members for the structure.

```
START GROUP DEFINITION
MEMBER
_VERTICAL 1 TO 12
_XBEAM 13 TO 20
_ZBEAM 21 TO 24 29
_BRACE 25 TO 28
END GROUP DEFINITION
```

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the columns are being grouped to a name called VERTICAL, the beams running along the X direction are assigned the name XBEAM, and so on.

```
MEMBER PROPERTIES CANADIAN
_VERTICAL TA ST W310X97
_XBEAM TA ST W250X39
_ZBEAM TA ST C200X17
_BRACE TA ST L152X152X13
```

Member properties are assigned from the Canadian steel table. The members which receive these properties are those embedded within the respective group names. The benefit of using the group name is apparent here. Just from the looks of the command, we can understand that the diagonal braces are being assigned a single angle. The alternative, which would be

```
25 TO 28 TA ST L152X152X13
```

would have required us to go to the graphical tools to get a sense of what members 25 to 28 are.

```
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 200
POISSON 0.3
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The `BETA` angle for the channels along the left edge is set to 180 so their legs point toward the interior of the structure.

```
SUPPORTS
1 5 9 13 PINNED
```

The bottom ends of the columns of the platform are pinned supported.

```
CUT OFF MODE SHAPE 30
```

The above command is a critical command if you want to override the default number of modes computed and used in a dynamic analysis. The default, which is 6, may not always be sufficient to capture a significant portion of the structural response in a response spectrum or time history analysis, and hence the need to override the default. This command is explained in TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344).

```
UNIT METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
```

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the earthquake (ground acceleration) are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a ground acceleration. (If you want to specify a forcing function, the keyword FORCE must be used instead.)

Notice the expression `READ EQDATA.TXT`. It means that we have chosen to specify the time vs. ground acceleration data in the file called `EQDATA.TXT`. That file must reside in the same folder as the one in which the data file for this structure resides. As explained in the small examples shown in TR.31.4 Definition of Time History Load (on page 2441) , the `EQDATA.TXT` file is a simple text file containing several pairs of time-acceleration data. A sample portion of that file is as shown below.

```
0.0000   0.006300
0.0200   0.003640
0.0400   0.000990
0.0600   0.004280
0.0800   0.007580
0.1000   0.010870
```

While it may not be apparent from the above numbers, it may also be noted that the geological data for the site the building sits on indicate that the above acceleration values are a fraction of "g", the acceleration due to gravity. Thus, for example, at 0.02 seconds, the acceleration is 0.00364 multiplied by 9.806 m/sec^2 (or 0.00364 multiplied by 32.2 ft/sec^2). Consequently, the burden of informing the program that the values need to be multiplied by "g" is upon us, and we shall do so at a later step.

The arrival time value indicates the relative value of time at which the earthquake begins to act upon the structure. We have chosen 0.0, as there is no other dynamic load on the structure from the relative time standpoint. The modal damping ratio for all the modes is set to 0.05.

```
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
```

The above data describe a static load case. The selfweight of the structure is acting in the negative global Y direction.

```
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
```

Load case 2 is also a static load case. At the Y=7.0m elevation, our structure has a floor slab. But, it is a non-structural entity which, though capable of carrying the loads acting on itself, is not meant to be an integral part of the framing system. It merely transmits the load to the beam-column grid.

There are uniform area loads on the floor (think of the load as wooden pallets supporting boxes of paper). Since the slab is not part of the structural model, how do we tell the program to transmit the imposed load from the slab to the beams without manually converting them to distributed beam loads ourselves? That is where the floor load utility comes in handy. It is a facility where we specify the load as a pressure, and the program converts the pressure to individual beam loads. Thus, the input required is very simple - load intensity in the form of pressure, and the region of the structure in terms of X, Y and Z coordinates in space, of the area over which the pressure acts.

In the process of converting the pressure to beam loads, STAAD will consider the empty space between criss-crossing beams (in plan view) to be panels, similar to the squares of a chess board. The load on each panel is then transferred to beams surrounding the panel, using a triangular or trapezoidal load distribution method.

```
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
```

Load case 3 is the dynamic load case, the one which contains the second part of the instruction set for a dynamic analysis to be performed. The data here are

a. loads which will yield the mass values which will populate the mass matrix
b. the directions of the loads, which will yield the degree of freedom numbers of the mass matrix for being populated.

Thus, the selfweight, as well as the imposed loads on the non-structural slab are to be considered as participating in the vibration along all the global directions.

```
GROUND MOTION X 1 1 9.806
```

The above command too is part of load case 3. Here we say that the seismic force, whose characteristics are defined by the TYPE 1 time history input data, acting at arrival time 1, is to be applied along the X direction. We mentioned earlier that the acceleration input data was specified as a fraction of “g”. The number 9.806 indicates the value which the acceleration data, as read from EQDATA.TXT are to be factored by before they are used.

```
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1 1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1 1.0 2 1.0 3 -1.0
```

In a time history analysis, the member forces FX thru MZ each have a value for every time step. If there are a 1000 time steps, there will be 1000 values of FX, 1000 for FY etc. for that load case. Not all of them can be used in a further calculation like a steel or concrete design. However, the maximum from among those time steps is available. If we want to do a design, one way to make sure that the structure is not under-designed is to create 2 load combination cases involving the dynamic case, a positive combination, and a negative combination.

That is what is being done above. Load combination case no. 11 consists of the sum of the static load cases (1 & 2) with the positive direction of the dynamic load case (3). Load combination case no. 12 consists of the sum of the static load cases (1 & 2) with the negative direction of the dynamic load case (3). The user has discretion on what load factors to use with these combinations. We have chosen the factors to be 1.0.

```
    PERFORM ANALYSIS
```

The above is the instruction to perform the analysis related calculations. That means, computing nodal displacements, support reactions, etc.

```
    PRINT ANALYSIS RESULTS
```

The above command is an instruction to the program to produce a report of the joint displacements, support reactions and member end forces in the output file. As mentioned earlier, for the dynamic case, these will be just the maximum values, not the ones generated for every time step. If you want to see the results for each time step, you may do so by using STAAD's Post-processing facilities.

```
    LOAD LIST 11 12
    PARAMETER
    CODE CANADIAN
    CHECK CODE ALL
```

A steel design code check is done according to the Canadian code for load cases 11 and 12.

```
    FINISH
```

Input File

```
STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
START GROUP DEFINITION
MEMBER
_VERTICAL 1 TO 12
_XBEAM 13 TO 20
_ZBEAM 21 TO 24 29
_BRACE 25 TO 28
END GROUP DEFINITION
MEMBER PROPERTIES CANADIAN
_VERTICAL TA ST W310X97
_XBEAM TA ST W250X39
_ZBEAM TA ST C200X17
_BRACE TA ST L152X152X13
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 200
POISSON 0.3
```

```
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
SUPPORTS
1 5 9 13 PINNED
CUT OFF MODE SHAPE 30
UNIT KGS METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
GROUND MOTION X 1 1 9.806
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1 1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1 1.0 2 1.0 3 -1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 11 12
PARAMETER
CODE CANADIAN
CHECK CODE ALL
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:46:24                      *
             *                                                  *
```

```
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
INPUT FILE: US-29 Time History Analysis of a Frame for Seismic Loads.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
     5. REPEAT ALL 1 9.5 0 0
     6. REPEAT ALL 1 0 0 3
     7. 17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
     8. REPEAT ALL 1 0 0 3
     9. MEMBER INCIDENCES
    10. 1 1 2 3
    11. REPEAT 1 3 4
    12. 7 9 10 9
    13. 10 13 14 12
    14. 13 4 17; 14 17 18; 15 18 19; 16 19 8
    15. 17 12 20; 18 20 21; 19 21 22; 20 22 16
    16. 21 2 10; 22 4 12; 23 6 14
    17. 24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
    18. START GROUP DEFINITION
    19. MEMBER
    20. _VERTICAL 1 TO 12
    21. _XBEAM 13 TO 20
    22. _ZBEAM 21 TO 24 29
    23. _BRACE 25 TO 28
    24. END GROUP DEFINITION
    25. MEMBER PROPERTIES CANADIAN
    26. _VERTICAL TA ST W310X97
    27. _XBEAM TA ST W250X39
    28. _ZBEAM TA ST C200X17
    29. _BRACE TA ST L152X152X13
    30. UNIT KNS MMS
    31. DEFINE MATERIAL START
    32. ISOTROPIC STEEL
    33. E 200
    34. POISSON 0.3
    35. DENSITY 7.8E-008
    36. ALPHA 6E-006
    37. DAMP 0.03
    38. TYPE STEEL
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    2
    39. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
    40. END DEFINE MATERIAL
    41. CONSTANTS
    42. MATERIAL STEEL ALL
    43. BETA 180 MEMB 21 22
    44. SUPPORTS
    45. 1 5 9 13 PINNED
    46. CUT OFF MODE SHAPE 30
    47. UNIT KGS METER
    48. DEFINE TIME HISTORY
    49. TYPE 1 ACCELERATION
    50. READ EQDATA.TXT
    51. ARRIVAL TIME
    52. 0.0
    53. DAMPING 0.05
    54. LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
```

```
   55. SELFWEIGHT Y -1.0
   56. LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
   57. FLOOR LOAD
   58. YRA 6.9 7.1 FLOAD -500
**NOTE** about Floor/OneWay Loads/Weights.
  Please note that depending on the shape of the floor you may
  have to break up the FLOOR/ONEWAY LOAD into multiple commands.
  For details please refer to Technical Reference Manual
  Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
   59. LOAD 3 DYNAMIC LOAD
   60. * MASSES
   61. SELFWEIGHT X 1.0
   62. SELFWEIGHT Y 1.0
   63. SELFWEIGHT Z 1.0
   64. FLOOR LOAD
   65. YRANGE 6.9 7.1 FLOAD 500 GX
   66. YRANGE 6.9 7.1 FLOAD 500 GY
   67. YRANGE 6.9 7.1 FLOAD 500 GZ
   68. GROUND MOTION X 1 1 9.806
   69. LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
   70. 1 1.0 2 1.0 3 1.0
   71. LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
   72. 1 1.0 2 1.0 3 -1.0
   73. PERFORM ANALYSIS
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    3
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         22  NUMBER OF MEMBERS      29
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =     120
  TOTAL LOAD COMBINATION  CASES =     2  SO FAR.
  ***NOTE: MASSES DEFINED UNDER LOAD#       3 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED              =    30
NUMBER OF EXISTING MASSES IN THE MODEL =    54
NUMBER OF MODES THAT WILL BE USED      =    30
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    4
              CALCULATED FREQUENCIES FOR LOAD CASE       3
      MODE           FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                      0.693                 1.44295
        2                      1.215                 0.82296
        3                      1.365                 0.73265
        4                      1.561                 0.64059
        5                      2.077                 0.48142
        6                      3.044                 0.32846
        7                      4.217                 0.23712
        8                      4.273                 0.23404
        9                      5.538                 0.18058
       10                      5.543                 0.18039
       11                      5.728                 0.17457
       12                     12.732                 0.07854
       13                     12.741                 0.07849
```

```
       14                     15.103                  0.06621
       15                     15.160                  0.06596
       16                     16.354                  0.06115
       17                     16.361                  0.06112
       18                     45.292                  0.02208
       19                     45.315                  0.02207
       20                     48.670                  0.02055
       21                     48.698                  0.02053
       22                     51.820                  0.01930
       23                     51.966                  0.01924
       24                     54.942                  0.01820
       25                     55.877                  0.01790
       26                     55.892                  0.01789
       27                     65.315                  0.01531
       28                     65.477                  0.01527
       29                     87.675                  0.01141
       30                     87.698                  0.01140
          MODAL WEIGHT (MODAL MASS TIMES g) IN KGS          GENERALIZED
     MODE           X             Y             Z             WEIGHT
        1       8.428479E-17  3.973848E-17  1.501941E+04    8.845560E+03
        2       1.725302E+04  4.369967E-02  1.375553E-16    1.677680E+04
        3       1.921147E-11  1.030589E-14  8.419906E-01    1.266359E+04
        4       5.867172E-12  1.954666E-14  4.822830E+00    2.596549E+04
        5       3.639106E-16  1.767266E-19  2.366926E+03    6.440201E+03
        6       9.389162E-30  9.016470E-30  3.077720E-13    5.200768E+03
        7       7.448636E-29  8.510260E-28  1.081621E-12    4.804401E+03
        8       2.127649E-16  8.969083E-16  1.923308E+00    8.821168E+03
        9       6.201208E-03  9.100304E+03  1.700018E-10    6.018229E+03
       10       5.259372E-12  6.922816E-06  2.331645E-01    6.033932E+03
       11       1.221274E-16  1.806567E-11  4.085088E+01    8.734128E+03
       12       3.013838E+02  4.352953E+01  2.749213E-13    6.235822E+03
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    5
       13       9.397726E-08  1.312250E-08  8.607833E-04    6.238517E+03
       14       4.732753E-19  7.878291E-19  6.997098E+01    8.386677E+02
       15       1.000137E-15  1.586578E-14  9.354548E+01    8.424974E+02
       16       5.267608E+00  1.221315E+03  3.186588E-11    5.514818E+03
       17       2.898206E-09  6.827010E-07  5.604505E-02    5.516616E+03
       18       2.828040E-01  1.883040E+03  1.599798E-16    1.675764E+03
       19       5.315951E-12  3.899624E-08  7.850585E-06    1.675934E+03
       20       4.118067E+01  1.211323E+01  1.031537E-18    1.390516E+03
       21       3.229149E-10  9.662888E-11  1.550643E-07    1.389660E+03
       22       4.568585E-26  2.039624E-22  2.052465E-17    4.245108E+02
       23       9.871051E-26  1.307891E-22  3.766811E-17    4.220495E+02
       24       3.514037E-26  1.116731E-24  6.236122E-21    4.868892E+03
       25       8.054632E-03  3.932247E+02  3.819723E-16    5.355819E+03
       26       1.156172E-13  3.029329E-09  4.932334E-05    5.335906E+03
       27       3.125936E-20  5.889706E-16  2.044529E+00    5.603039E+02
       28       2.432179E-18  3.227255E-15  2.037832E+00    5.570734E+02
       29       7.267862E-01  1.713536E+03  7.387739E-16    2.807397E+03
       30       2.296865E-18  5.310772E-15  5.251737E-17    1.982699E+03
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    6
MASS PARTICIPATION FACTORS
                   MASS  PARTICIPATION FACTORS IN PERCENT
                   --------------------------------------
          MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z
            1     0.00   0.00  85.32    0.000    0.000   85.325
            2    98.01   0.00   0.00   98.014    0.000   85.325
            3     0.00   0.00   0.00   98.014    0.000   85.329
```

```
          4     0.00   0.00   0.03   98.014    0.000   85.357
          5     0.00   0.00  13.45   98.014    0.000   98.803
          6     0.00   0.00   0.00   98.014    0.000   98.803
          7     0.00   0.00   0.00   98.014    0.000   98.803
          8     0.00   0.00   0.01   98.014    0.000   98.814
          9     0.00  51.70   0.00   98.014   51.699   98.814
         10     0.00   0.00   0.00   98.014   51.699   98.815
         11     0.00   0.00   0.23   98.014   51.699   99.047
         12     1.71   0.25   0.00   99.726   51.946   99.047
         13     0.00   0.00   0.00   99.726   51.946   99.047
         14     0.00   0.00   0.40   99.726   51.946   99.445
         15     0.00   0.00   0.53   99.726   51.946   99.976
         16     0.03   6.94   0.00   99.756   58.884   99.976
         17     0.00   0.00   0.00   99.756   58.884   99.977
         18     0.00  10.70   0.00   99.757   69.582   99.977
         19     0.00   0.00   0.00   99.757   69.582   99.977
         20     0.23   0.07   0.00   99.991   69.650   99.977
         21     0.00   0.00   0.00   99.991   69.650   99.977
         22     0.00   0.00   0.00   99.991   69.650   99.977
         23     0.00   0.00   0.00   99.991   69.650   99.977
         24     0.00   0.00   0.00   99.991   69.650   99.977
         25     0.00   2.23   0.00   99.991   71.884   99.977
         26     0.00   0.00   0.00   99.991   71.884   99.977
         27     0.00   0.00   0.01   99.991   71.884   99.988
         28     0.00   0.00   0.01   99.991   71.884  100.000
         29     0.00   9.73   0.00   99.995   81.619  100.000
         30     0.00   0.00   0.00   99.995   81.619  100.000
 A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
 MODE       DAMPING
    1      0.05000000
    2      0.05000000
    3      0.05000000
    4      0.05000000
    5      0.05000000
    6      0.05000000
    7      0.05000000
    8      0.05000000
    9      0.05000000
   10      0.05000000
   11      0.05000000
   12      0.05000000
   13      0.05000000
   14      0.05000000
   15      0.05000000
   16      0.05000000
   DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    7
 MODE       DAMPING
   17      0.05000000
   18      0.05000000
   19      0.05000000
   20      0.05000000
   21      0.05000000
   22      0.05000000
   23      0.05000000
   24      0.05000000
   25      0.05000000
   26      0.05000000
   27      0.05000000
```

```
     28      0.05000000
     29      0.05000000
     30      0.05000000
   TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
   NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =   30
   TIME DURATION OF TIME HISTORY ANALYSIS =     31.160 SECONDS
   NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =    22435
    74. PRINT ANALYSIS RESULTS
  BASE SHEAR UNITS ARE -- KGS  METE
  MAXIMUM BASE SHEAR  X=  -9.498282E+03  Y=  -5.285202E+01  Z=   3.482067E-06
      AT TIMES               5.809722          2.445833           2.766667
  ANALYSIS RESULTS
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    8
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.0000    0.0000    0.0000   -0.0000   -0.0000    0.0001
           2    0.0000    0.0000    0.0000    0.0000   -0.0000    0.0015
           3    0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0173
          11    0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0157
          12    0.0000    0.0000    0.0000    0.0000   -0.0000    0.0190
      2    1   -0.0290   -0.0012   -0.0000    0.0000   -0.0000    0.0000
           2   -0.4146   -0.0050   -0.0004   -0.0001   -0.0000    0.0004
           3    5.7319    0.0046    0.0000    0.0000   -0.0000   -0.0142
          11    5.2883   -0.0015   -0.0005   -0.0001   -0.0000   -0.0138
          12   -6.1755   -0.0108   -0.0005   -0.0001   -0.0000    0.0145
      3    1   -0.0249   -0.0016   -0.0002   -0.0000   -0.0000   -0.0001
           2   -0.3581   -0.0075   -0.0122   -0.0000   -0.0001   -0.0012
           3    7.9529    0.0070    0.0000    0.0000   -0.0000   -0.0101
          11    7.5699   -0.0021   -0.0124   -0.0000   -0.0001   -0.0113
          12   -8.3358   -0.0161   -0.0124   -0.0000   -0.0001    0.0088
      4    1   -0.0030   -0.0016    0.0000    0.0000   -0.0000   -0.0001
           2   -0.0451   -0.0076    0.0004    0.0002   -0.0000   -0.0020
           3    9.3683    0.0023    0.0000    0.0000    0.0000   -0.0083
          11    9.3201   -0.0069    0.0004    0.0002   -0.0000   -0.0104
          12   -9.4164   -0.0115    0.0004    0.0002   -0.0000    0.0062
      5    1    0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0001
           2    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0014
           3    0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0174
          11    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0189
          12    0.0000    0.0000    0.0000    0.0000    0.0000    0.0159
      6    1    0.0261   -0.0012   -0.0000    0.0000    0.0000   -0.0000
           2    0.3723   -0.0050   -0.0004   -0.0001    0.0000   -0.0002
           3    5.7484   -0.0046   -0.0000   -0.0000   -0.0000   -0.0142
          11    6.1468   -0.0108   -0.0005   -0.0001    0.0000   -0.0144
          12   -5.3500   -0.0015   -0.0005   -0.0001    0.0000    0.0139
      7    1    0.0206   -0.0016   -0.0002   -0.0000    0.0000    0.0001
           2    0.2940   -0.0075   -0.0122   -0.0000    0.0001    0.0013
           3    7.9667   -0.0070   -0.0000   -0.0000   -0.0000   -0.0100
          11    8.2813   -0.0161   -0.0124   -0.0000    0.0001   -0.0086
          12   -7.6521   -0.0021   -0.0124   -0.0000    0.0001    0.0114
      8    1   -0.0027   -0.0016    0.0000    0.0000    0.0000    0.0001
           2   -0.0398   -0.0077    0.0004    0.0002    0.0000    0.0021
           3    9.3696   -0.0026   -0.0000   -0.0000    0.0000   -0.0082
          11    9.3271   -0.0119    0.0004    0.0002    0.0000   -0.0060
          12   -9.4121   -0.0068    0.0004    0.0002    0.0000    0.0105
      9    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
           2    0.0000    0.0000    0.0000   -0.0000    0.0000    0.0015
```

```
          3     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0173
         11     0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0157
         12     0.0000    0.0000    0.0000   -0.0000    0.0000    0.0190
    10    1    -0.0290   -0.0012    0.0000   -0.0000    0.0000    0.0000
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    9
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
          2    -0.4146   -0.0050    0.0004    0.0001    0.0000    0.0004
          3     5.7319    0.0046    0.0000    0.0000   -0.0000   -0.0142
         11     5.2883   -0.0015    0.0005    0.0001    0.0000   -0.0138
         12    -6.1755   -0.0108    0.0005    0.0001    0.0000    0.0145
    11    1    -0.0249   -0.0016    0.0002    0.0000    0.0000   -0.0001
          2    -0.3581   -0.0075    0.0122    0.0000    0.0001   -0.0012
          3     7.9529    0.0070    0.0000    0.0000   -0.0000   -0.0101
         11     7.5699   -0.0021    0.0124    0.0000    0.0001   -0.0113
         12    -8.3358   -0.0161    0.0124    0.0000    0.0001    0.0088
    12    1    -0.0030   -0.0016   -0.0000   -0.0000    0.0000   -0.0001
          2    -0.0451   -0.0076   -0.0004   -0.0002    0.0000   -0.0020
          3     9.3683    0.0023    0.0000    0.0000    0.0000   -0.0083
         11     9.3201   -0.0069   -0.0004   -0.0002    0.0000   -0.0104
         12    -9.4164   -0.0115   -0.0004   -0.0002    0.0000    0.0062
    13    1     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0001
          2     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0014
          3     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0174
         11     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0189
         12     0.0000    0.0000    0.0000   -0.0000   -0.0000    0.0159
    14    1     0.0261   -0.0012    0.0000   -0.0000   -0.0000   -0.0000
          2     0.3723   -0.0050    0.0004    0.0001   -0.0000   -0.0002
          3     5.7484   -0.0046   -0.0000   -0.0000   -0.0000   -0.0142
         11     6.1468   -0.0108    0.0005    0.0001   -0.0000   -0.0144
         12    -5.3500   -0.0015    0.0005    0.0001   -0.0000    0.0139
    15    1     0.0206   -0.0016    0.0002    0.0000   -0.0000    0.0001
          2     0.2940   -0.0075    0.0122    0.0000   -0.0001    0.0013
          3     7.9667   -0.0070   -0.0000   -0.0000   -0.0000   -0.0100
         11     8.2813   -0.0161    0.0124    0.0000   -0.0001   -0.0086
         12    -7.6521   -0.0021    0.0124    0.0000   -0.0001    0.0114
    16    1    -0.0027   -0.0016   -0.0000   -0.0000   -0.0000    0.0001
          2    -0.0398   -0.0077   -0.0004   -0.0002   -0.0000    0.0021
          3     9.3696   -0.0026   -0.0000   -0.0000    0.0000   -0.0082
         11     9.3271   -0.0119   -0.0004   -0.0002   -0.0000   -0.0060
         12    -9.4121   -0.0068   -0.0004   -0.0002   -0.0000    0.0105
    17    1    -0.0027   -0.0267    0.0000    0.0000    0.0000   -0.0002
          2    -0.0400   -0.3687    0.0008    0.0001    0.0000   -0.0025
          3     9.3513   -1.4019    0.0000   -0.0000    0.0000   -0.0033
         11     9.3086   -1.7973    0.0008    0.0001    0.0000   -0.0059
         12    -9.3941    1.0065    0.0008    0.0001    0.0000    0.0006
    18    1    -0.0029   -0.0644   -0.0000    0.0001    0.0000   -0.0000
          2    -0.0425   -0.9647   -0.0000    0.0038    0.0000   -0.0003
          3     9.3534   -0.0734    0.0000    0.0000    0.0000    0.0086
         11     9.3080   -1.1025   -0.0000    0.0039    0.0000    0.0083
         12    -9.3987   -0.9557   -0.0000    0.0039    0.0000   -0.0089
    19    1    -0.0031   -0.0296    0.0000    0.0000   -0.0000    0.0002
          2    -0.0451   -0.4115    0.0009    0.0001   -0.0000    0.0025
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   10
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
```

```
           3     9.3514    1.4648   -0.0000    0.0000    0.0000   -0.0031
          11     9.3032    1.0237    0.0009    0.0001   -0.0000   -0.0004
          12    -9.3995   -1.9060    0.0009    0.0001   -0.0000    0.0057
     20    1    -0.0027   -0.0267   -0.0000   -0.0000   -0.0000   -0.0002
           2    -0.0400   -0.3687   -0.0008   -0.0001   -0.0000   -0.0025
           3     9.3513   -1.4019    0.0000   -0.0000    0.0000   -0.0033
          11     9.3086   -1.7973   -0.0008   -0.0001   -0.0000   -0.0059
          12    -9.3941    1.0065   -0.0008   -0.0001   -0.0000    0.0006
     21    1    -0.0029   -0.0644    0.0000   -0.0001   -0.0000   -0.0000
           2    -0.0425   -0.9647    0.0000   -0.0038   -0.0000   -0.0003
           3     9.3534   -0.0734    0.0000    0.0000    0.0000    0.0086
          11     9.3080   -1.1025    0.0000   -0.0039   -0.0000    0.0083
          12    -9.3987   -0.9557    0.0000   -0.0039   -0.0000   -0.0089
     22    1    -0.0031   -0.0296   -0.0000   -0.0000    0.0000    0.0002
           2    -0.0451   -0.4115   -0.0009   -0.0001    0.0000    0.0025
           3     9.3514    1.4648   -0.0000    0.0000    0.0000   -0.0031
          11     9.3032    1.0237   -0.0009   -0.0001    0.0000   -0.0004
          12    -9.3995   -1.9060   -0.0009   -0.0001    0.0000    0.0057
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   11
  SUPPORT REACTIONS -UNIT KGS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    1    1      61.39   1009.30      0.98      0.00      0.00      0.00
         2     872.77   3562.50    -19.74      0.00      0.00      0.00
         3   -2356.64  -3311.14     -0.00      0.00      0.00      0.00
        11   -1422.49   1260.65    -18.76      0.00      0.00      0.00
        12    3290.79   7882.94    -18.76      0.00      0.00      0.00
    5    1     -61.39   1009.45      0.98      0.00      0.00      0.00
         2    -872.77   3562.50    -19.74      0.00      0.00      0.00
         3   -2392.50   3308.29      0.00      0.00      0.00      0.00
        11   -3326.65   7880.23    -18.76      0.00      0.00      0.00
        12    1458.35   1263.66    -18.76      0.00      0.00      0.00
    9    1      61.39   1009.30     -0.98      0.00      0.00      0.00
         2     872.77   3562.50     19.74      0.00      0.00      0.00
         3   -2356.64  -3311.14     -0.00      0.00      0.00      0.00
        11   -1422.49   1260.65     18.76      0.00      0.00      0.00
        12    3290.79   7882.94     18.76      0.00      0.00      0.00
   13    1     -61.39   1009.45     -0.98      0.00      0.00      0.00
         2    -872.77   3562.50     19.74      0.00      0.00      0.00
         3   -2392.50   3308.29      0.00      0.00      0.00      0.00
        11   -3326.65   7880.23     18.76      0.00      0.00      0.00
        12    1458.35   1263.66     18.76      0.00      0.00      0.00
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   12
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KGS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1   1009.30    -61.39     0.98      0.00      0.00       0.00
               2   -666.89     61.39    -0.98      0.00     -3.43    -214.85
         2     1   3562.50   -872.77   -19.74      0.00     -0.00      -0.00
               2  -3562.50    872.77    19.74      0.00     69.09   -3054.69
         3     1  -3311.14   2356.64    -0.00      0.00      0.00       0.00
               2   3311.14  -2356.64     0.00      0.00      0.00    8248.24
        11     1   1260.65   1422.49   -18.76      0.00     -0.00       0.00
               2   -918.24  -1422.49    18.76      0.00     65.66    4978.71
        12     1   7882.94  -3290.79   -18.76      0.00     -0.00      -0.00
               2  -7540.53   3290.79    18.76      0.00     65.66  -11517.78
    2    1     2    641.00    -61.39     5.97      0.01     -9.03     214.85
```

```
                 3   -464.90     61.39     -5.97     -0.01     -1.72    -325.34
           2     2   3562.50   -872.77    112.31      0.24    -79.35    3054.69
                 3  -3562.50    872.77   -112.31     -0.24   -122.82   -4625.67
           3     2  -3311.13   2253.63     -0.00     -0.00      0.00   -8248.24
                 3   3311.13  -2253.63      0.00      0.00      0.00   12304.78
          11     2    892.36   1319.48    118.29      0.24    -88.38   -4978.71
                 3   -716.27  -1319.48   -118.29     -0.24   -124.54    7353.77
          12     2   7514.63  -3187.79    118.29      0.24    -88.38   11517.78
                 3  -7338.53   3187.79   -118.29     -0.24   -124.54  -17255.79
      3    1     3    191.68    186.66      5.96     -0.01      1.70     310.00
                 4    -25.36   -186.66     -5.96      0.01    -11.84       7.31
           2     3     41.65   2832.69    111.63     -0.35    122.13    4570.39
                 4    -41.65  -2832.69   -111.63      0.35   -311.90     245.19
           3     3   6886.37  -8900.35     -0.00     -0.00     -0.00  -12336.68
                 4  -6886.37   8900.35      0.00      0.00      0.00   -2793.95
          11     3   7119.69  -5881.00    117.60     -0.36    123.83   -7456.29
                 4  -6953.38   5881.00   -117.60      0.36   -323.74   -2541.45
          12     3  -6653.05  11919.69    117.60     -0.36    123.83   17217.07
                 4   6819.36 -11919.69   -117.60      0.36   -323.74    3046.45
      4    1     5   1009.45     61.39      0.98      0.00      0.00      -0.00
                 6   -667.04    -61.39     -0.98      0.00     -3.43     214.85
           2     5   3562.50    872.77    -19.74      0.00      0.00      -0.00
                 6  -3562.50   -872.77     19.74      0.00     69.07    3054.69
           3     5   3308.29   2392.50      0.00      0.00      0.00       0.00
                 6  -3308.29  -2392.50     -0.00      0.00     -0.00    8373.75
          11     5   7880.23   3326.65    -18.76      0.00      0.00       0.00
                 6  -7537.82  -3326.65     18.76      0.00     65.65   11643.29
          12     5   1263.66  -1458.35    -18.76      0.00      0.00      -0.00
                 6   -921.25   1458.35     18.76      0.00     65.65   -5104.22
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                        -- PAGE NO.   13
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KGS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      5    1     6    641.15     61.39      5.97     -0.01     -9.03    -214.85
                 7   -465.05    -61.39     -5.97      0.01     -1.72     325.34
           2     6   3562.50    872.77    112.28     -0.22    -79.33   -3054.69
                 7  -3562.50   -872.77   -112.28      0.22   -122.76    4625.67
           3     6   3308.16   2288.65      0.00      0.00     -0.00   -8373.75
                 7  -3308.16  -2288.65     -0.00     -0.00     -0.00   12493.31
          11     6   7511.81   3222.80    118.25     -0.23    -88.36  -11643.29
                 7  -7335.71  -3222.80   -118.25      0.23   -124.49   17444.32
          12     6    895.48  -1354.49    118.25     -0.23    -88.36    5104.22
                 7   -719.39   1354.49   -118.25      0.23   -124.49   -7542.31
      6    1     7    206.02   -182.87      5.96      0.01      1.71    -308.77
                 8    -39.71    182.87     -5.96     -0.01    -11.84      -2.12
           2     7    257.08  -2789.79    111.63      0.33    122.14   -4566.17
                 8   -257.08   2789.79   -111.63     -0.33   -311.91    -176.47
           3     7  -6533.99  -9068.05      0.00     -0.00      0.00  -12526.20
                 8   6533.99   9068.05     -0.00      0.00     -0.00   -2889.65
          11     7  -6070.89 -12040.72    117.59      0.34    123.84  -17401.14
                 8   6237.20  12040.72   -117.59     -0.34   -323.75   -3068.23
          12     7   6997.09   6095.39    117.59      0.34    123.84    7651.26
                 8  -6830.78  -6095.39   -117.59     -0.34   -323.75    2711.06
      7    1     9   1009.30    -61.39     -0.98      0.00     -0.00       0.00
                10   -666.89     61.39      0.98      0.00      3.43    -214.85
           2     9   3562.50   -872.77     19.74      0.00      0.00       0.00
                10  -3562.50    872.77    -19.74      0.00    -69.09   -3054.69
```

```
          3     9  -3311.14   2356.64     -0.00       0.00       0.00      -0.00
               10   3311.14  -2356.64      0.00       0.00       0.00    8248.24
         11     9   1260.65   1422.49     18.76       0.00       0.00      -0.00
               10   -918.24  -1422.49    -18.76       0.00     -65.66    4978.71
         12     9   7882.94  -3290.79     18.76       0.00       0.00       0.00
               10  -7540.53   3290.79    -18.76       0.00     -65.66  -11517.78
      8   1    10    641.00    -61.39     -5.97      -0.01       9.03     214.85
               11   -464.90     61.39      5.97       0.01       1.72    -325.34
          2    10   3562.50   -872.77   -112.31      -0.24      79.35    3054.69
               11  -3562.50    872.77    112.31       0.24     122.82   -4625.67
          3    10  -3311.13   2253.63     -0.00      -0.00       0.00   -8248.24
               11   3311.13  -2253.63      0.00       0.00       0.00   12304.78
         11    10    892.36   1319.48   -118.29      -0.24      88.38   -4978.71
               11   -716.27  -1319.48    118.29       0.24     124.54    7353.77
         12    10   7514.63  -3187.79   -118.29      -0.24      88.38   11517.78
               11  -7338.53   3187.79    118.29       0.24     124.54  -17255.79
      9   1    11    191.68    186.66     -5.96       0.01      -1.70     310.00
               12    -25.36   -186.66      5.96      -0.01      11.84       7.31
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   14
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2    11     41.65   2832.69   -111.63       0.35    -122.13    4570.39
               12    -41.65  -2832.69    111.63      -0.35     311.90     245.19
          3    11   6886.37  -8900.35     -0.00      -0.00      -0.00  -12336.68
               12  -6886.37   8900.35      0.00       0.00       0.00   -2793.95
         11    11   7119.69  -5881.00   -117.60       0.36    -123.83   -7456.29
               12  -6953.38   5881.00    117.60      -0.36     323.74   -2541.45
         12    11  -6653.05  11919.69   -117.60       0.36    -123.83   17217.07
               12   6819.36 -11919.69    117.60      -0.36     323.74    3046.45
     10   1    13   1009.45     61.39     -0.98       0.00      -0.00      -0.00
               14   -667.04    -61.39      0.98       0.00       3.43     214.85
          2    13   3562.50    872.77     19.74       0.00       0.00       0.00
               14  -3562.50   -872.77    -19.74       0.00     -69.07    3054.69
          3    13   3308.29   2392.50      0.00       0.00       0.00      -0.00
               14  -3308.29  -2392.50     -0.00       0.00      -0.00    8373.75
         11    13   7880.23   3326.65     18.76       0.00       0.00      -0.00
               14  -7537.82  -3326.65    -18.76       0.00     -65.65   11643.29
         12    13   1263.66  -1458.35     18.76       0.00       0.00       0.00
               14   -921.25   1458.35    -18.76       0.00     -65.65   -5104.22
     11   1    14    641.15     61.39     -5.97       0.01       9.03    -214.85
               15   -465.05    -61.39      5.97      -0.01       1.72     325.34
          2    14   3562.50    872.77   -112.28       0.22      79.33   -3054.69
               15  -3562.50   -872.77    112.28      -0.22     122.76    4625.67
          3    14   3308.16   2288.65      0.00       0.00      -0.00   -8373.75
               15  -3308.16  -2288.65     -0.00      -0.00      -0.00   12493.31
         11    14   7511.81   3222.80   -118.25       0.23      88.36  -11643.29
               15  -7335.71  -3222.80    118.25      -0.23     124.49   17444.32
         12    14    895.48  -1354.49   -118.25       0.23      88.36    5104.22
               15   -719.39   1354.49    118.25      -0.23     124.49   -7542.31
     12   1    15    206.02   -182.87     -5.96      -0.01      -1.71    -308.77
               16    -39.71    182.87      5.96       0.01      11.84      -2.12
          2    15    257.08  -2789.79   -111.63      -0.33    -122.14   -4566.17
               16   -257.08   2789.79    111.63       0.33     311.91    -176.47
          3    15  -6533.99  -9068.05      0.00      -0.00       0.00  -12526.20
               16   6533.99   9068.05     -0.00       0.00      -0.00   -2889.65
         11    15  -6070.89 -12040.72   -117.59      -0.34    -123.84  -17401.14
```

```
                16   6237.20  12040.72   117.59      0.34    323.75   -3068.23
          12    15   6997.09   6095.39  -117.59     -0.34   -123.84    7651.26
                16  -6830.78  -6095.39   117.59      0.34    323.75    2711.06
    13     1     4   -186.66     -0.53    -0.01      0.00     -0.01      -7.31
                17    186.66     70.96     0.01     -0.00      0.03     -57.03
           2     4  -2832.69   -520.85    -0.69      0.09     -0.29    -245.19
                17   2832.69   1308.35     0.69     -0.09      1.54   -1176.10
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                          -- PAGE NO.   15
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           3     4   9442.80   6886.14    -0.00      0.00     -0.00    2793.95
                17  -9442.80  -6886.14     0.00     -0.00      0.00    9601.11
          11     4   6423.46   6364.76    -0.70      0.09     -0.30    2541.45
                17  -6423.46  -5506.82     0.70     -0.09      1.57    8367.98
          12     4 -12462.15  -7407.52    -0.70      0.09     -0.30   -3046.45
                17  12462.15   8265.46     0.70     -0.09      1.57  -10834.23
    14     1    17     61.39    129.40    -0.00     -0.03      0.01      67.81
                18    -61.39    -19.82     0.00      0.03     -0.01     141.09
           2    17    872.77   2212.50    -0.01     -1.76      0.27    1193.12
                18   -872.77   -675.00     0.01      1.76     -0.24    2343.13
           3    17   -725.43  -3432.33    -0.00     -0.00     -0.00  -10005.50
                18    725.43   3432.33     0.00      0.00     -0.00     395.71
          11    17    208.72  -1090.44    -0.01     -1.79      0.28   -8744.57
                18   -208.72   2737.51     0.01      1.79     -0.24    2879.93
          12    17   1659.58   5774.23    -0.01     -1.79      0.28   11266.44
                18  -1659.58  -4127.16     0.01      1.79     -0.24    2088.51
    15     1    18     61.39     -6.07     0.00      0.03      0.01    -141.09
                19    -61.39    123.46    -0.00     -0.03     -0.01     -53.20
           2    18    872.77   -450.00     0.01      1.64      0.23   -2343.13
                19   -872.77   2137.50    -0.01     -1.64     -0.26    -975.62
           3    18    667.43  -3441.37     0.00     -0.00      0.00    -395.71
                19   -667.43   3441.37    -0.00      0.00     -0.00   -9929.10
          11    18   1601.58  -3897.43     0.01      1.67      0.24   -2879.93
                19  -1601.58   5702.33    -0.01     -1.67     -0.27  -10957.92
          12    18    266.73   2985.30     0.01      1.67      0.24   -2088.51
                19   -266.73  -1180.40    -0.01     -1.67     -0.27    8900.27
    16     1    19   -182.87     60.53     0.01     -0.00     -0.03      42.27
                 8    182.87     13.82    -0.01      0.00      0.01       2.12
           2    19  -2789.79   1167.92     0.66     -0.08     -1.52     970.08
                 8   2789.79   -305.42    -0.66      0.08      0.27     176.47
           3    19  -9627.97   6534.44    -0.00      0.00      0.00    9525.79
                 8   9627.97  -6534.44     0.00     -0.00     -0.00    2889.65
          11    19 -12600.64   7762.89     0.67     -0.09     -1.55   10538.13
                 8  12600.64  -6826.04    -0.67      0.09      0.28    3068.23
          12    19   6655.31  -5305.98     0.67     -0.09     -1.55   -8513.44
                 8  -6655.31   6242.83    -0.67      0.09      0.28   -2711.06
    17     1    12   -186.66     -0.53     0.01     -0.00      0.01      -7.31
                20    186.66     70.96    -0.01      0.00     -0.03     -57.03
           2    12  -2832.69   -520.85     0.69     -0.09      0.29    -245.19
                20   2832.69   1308.35    -0.69      0.09     -1.54   -1176.10
           3    12   9442.80   6886.14    -0.00      0.00     -0.00    2793.95
                20  -9442.80  -6886.14     0.00     -0.00      0.00    9601.11
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                          -- PAGE NO.   16
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
```

```
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
        11    12   6423.46   6364.76     0.70     -0.09      0.30    2541.45
              20  -6423.46  -5506.82    -0.70      0.09     -1.57    8367.98
        12    12 -12462.15  -7407.52     0.70     -0.09      0.30   -3046.45
              20  12462.15   8265.46    -0.70      0.09     -1.57  -10834.23
   18    1    20     61.39    129.40     0.00      0.03     -0.01      67.81
              21    -61.39    -19.82    -0.00     -0.03      0.01     141.09
         2    20    872.77   2212.50     0.01      1.76     -0.27    1193.12
              21   -872.77   -675.00    -0.01     -1.76      0.24    2343.13
         3    20   -725.43  -3432.34    -0.00     -0.00     -0.00  -10005.50
              21    725.43   3432.34     0.00      0.00     -0.00     395.71
        11    20    208.72  -1090.44     0.01      1.79     -0.28   -8744.57
              21   -208.72   2737.51    -0.01     -1.79      0.24    2879.93
        12    20   1659.58   5774.23     0.01      1.79     -0.28   11266.44
              21  -1659.58  -4127.16    -0.01     -1.79      0.24    2088.51
   19    1    21     61.39     -6.07    -0.00     -0.03     -0.01    -141.09
              22    -61.39    123.46     0.00      0.03      0.01     -53.20
         2    21    872.77   -450.00    -0.01     -1.64     -0.23   -2343.13
              22   -872.77   2137.50     0.01      1.64      0.26    -975.62
         3    21    667.43  -3441.37     0.00     -0.00      0.00    -395.71
              22   -667.43   3441.37    -0.00      0.00     -0.00   -9929.10
        11    21   1601.58  -3897.43    -0.01     -1.67     -0.24   -2879.93
              22  -1601.58   5702.33     0.01      1.67      0.27  -10957.92
        12    21    266.73   2985.30    -0.01     -1.67     -0.24   -2088.51
              22   -266.73  -1180.40     0.01      1.67      0.27    8900.27
   20    1    22   -182.87     60.53    -0.01      0.00      0.03      42.27
              16    182.87     13.82     0.01     -0.00     -0.01       2.12
         2    22  -2789.79   1167.92    -0.66      0.08      1.52     970.08
              16   2789.79   -305.42     0.66     -0.08     -0.27     176.47
         3    22  -9627.98   6534.44    -0.00      0.00      0.00    9525.79
              16   9627.98  -6534.44     0.00     -0.00     -0.00    2889.65
        11    22 -12600.64   7762.89    -0.67      0.09      1.55   10538.13
              16  12600.64  -6826.04     0.67     -0.09     -0.28    3068.23
        12    22   6655.31  -5305.98    -0.67      0.09      1.55   -8513.44
              16  -6655.31   6242.84     0.67     -0.09     -0.28   -2711.06
   21    1     2     -5.00    -25.89     0.00     -0.00      0.01     -12.46
              10      5.00    -25.89    -0.00      0.00     -0.01      12.46
         2     2   -132.05     -0.00     0.00     -0.00      0.24     -10.26
              10    132.05      0.00    -0.00      0.00     -0.24      10.26
         3     2      0.00      0.00     0.00     -0.00     -0.00       0.00
              10      0.00     -0.00    -0.00      0.00     -0.00       0.00
        11     2   -137.05    -25.89     0.00     -0.00      0.24     -22.72
              10    137.05    -25.89    -0.00      0.00     -0.24      22.72
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   17
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
        12     2   -137.05    -25.89    -0.00      0.00      0.24     -22.72
              10    137.05    -25.89     0.00     -0.00     -0.24      22.72
   22    1     4      5.97    -25.89     0.00     -0.00      0.00     -11.85
              12     -5.97    -25.89    -0.00      0.00     -0.00      11.85
         2     4    112.33   -562.50     0.00     -0.00      0.06    -311.99
              12   -112.33   -562.50    -0.00      0.00     -0.06     311.99
         3     4      0.00      0.00     0.00     -0.00     -0.00       0.00
              12      0.00     -0.00    -0.00      0.00     -0.00       0.00
        11     4    118.30   -588.39     0.00     -0.00      0.06    -323.84
              12   -118.30   -588.39    -0.00      0.00     -0.06     323.84
```

```
        12     4    118.30   -588.39    -0.00      0.00      0.06    -323.84
              12   -118.30   -588.39     0.00     -0.00     -0.06     323.84
   23    1     6     -4.99     25.89    -0.00     -0.00      0.01      12.46
              14      4.99     25.89     0.00      0.00     -0.01     -12.46
         2     6   -132.01     -0.00    -0.00     -0.00      0.22      10.26
              14    132.01      0.00     0.00      0.00     -0.22     -10.26
         3     6      0.00      0.00     0.00     -0.00     -0.00       0.00
              14      0.00     -0.00    -0.00      0.00     -0.00       0.00
        11     6   -137.01     25.89     0.00     -0.00      0.23      22.71
              14    137.01     25.89    -0.00      0.00     -0.23     -22.71
        12     6   -137.01     25.89    -0.00      0.00      0.23      22.71
              14    137.01     25.89     0.00     -0.00     -0.23     -22.71
   24    1     8      5.97     25.89    -0.00     -0.00      0.00      11.85
              16     -5.97     25.89     0.00      0.00     -0.00     -11.85
         2     8    112.29    562.50    -0.00     -0.00      0.06     311.99
              16   -112.29    562.50     0.00      0.00     -0.06    -311.99
         3     8      0.00      0.00    -0.00     -0.00      0.00       0.00
              16      0.00     -0.00     0.00      0.00      0.00       0.00
        11     8    118.26    588.39    -0.00     -0.00      0.06     323.84
              16   -118.26    588.39     0.00      0.00     -0.06    -323.84
        12     8    118.26    588.39     0.00      0.00      0.06     323.84
              16   -118.26    588.39    -0.00     -0.00     -0.06    -323.84
   25    1     3    367.93     28.33     0.01     -0.00      0.02      15.34
              17   -317.90     24.65    -0.01      0.00     -0.05     -10.79
         2     3   5111.42     15.45     0.68     -0.10      0.90      55.28
              17  -5111.42    -15.45    -0.68      0.10     -2.59     -17.03
         3     3 -15037.77    176.23     0.00      0.00     -0.00      31.99
              17  15037.77   -176.23    -0.00     -0.00      0.00     404.40
        11     3  -9558.42    220.00     0.69     -0.10      0.93     102.61
              17   9608.45   -167.03    -0.69      0.10     -2.64     376.59
        12     3  20517.12   -132.45     0.69     -0.10      0.93      38.63
              17 -20467.09    185.42    -0.69      0.10     -2.64    -432.21
   DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   18
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
   26    1     7    354.75     30.17    -0.01      0.00     -0.02      16.58
              19   -304.72     25.75     0.01     -0.00      0.05     -10.94
         2     7   4933.53     21.16    -0.65      0.10     -0.83      59.49
              19  -4933.53    -21.16     0.65     -0.10      2.48      -5.54
         3     7  14950.07   -171.11    -0.00      0.00      0.00     -33.13
              19 -14950.07    171.11     0.00     -0.00      0.00    -403.31
        11     7  20238.35   -119.78    -0.66      0.10     -0.85      42.93
              19 -20188.32    175.70     0.66     -0.10      2.53    -419.79
        12     7  -9661.80    222.44    -0.66      0.10     -0.85     109.20
              19   9711.83   -166.53     0.66     -0.10      2.53     386.83
   27    1    11    367.93     28.33    -0.01      0.00     -0.02      15.34
              20   -317.90     24.65     0.01     -0.00      0.05     -10.79
         2    11   5111.42     15.45    -0.68      0.10     -0.90      55.28
              20  -5111.42    -15.45     0.68     -0.10      2.59     -17.03
         3    11 -15037.77    176.23     0.00      0.00     -0.00      31.99
              20  15037.77   -176.23    -0.00     -0.00      0.00     404.40
        11    11  -9558.42    220.00    -0.69      0.10     -0.93     102.61
              20   9608.45   -167.03     0.69     -0.10      2.64     376.59
        12    11  20517.12   -132.45    -0.69      0.10     -0.93      38.63
              20 -20467.09    185.42     0.69     -0.10      2.64    -432.21
   28    1    15    354.75     30.17     0.01     -0.00      0.02      16.58
```

```
               22   -304.72     25.75    -0.01      0.00     -0.05     -10.94
          2    15   4933.53     21.16     0.65     -0.10      0.83      59.49
               22  -4933.53    -21.16    -0.65      0.10     -2.48      -5.54
          3    15  14950.07   -171.11    -0.00      0.00      0.00     -33.13
               22 -14950.07    171.11     0.00     -0.00      0.00    -403.31
         11    15  20238.35   -119.78     0.66     -0.10      0.85      42.93
               22 -20188.32    175.70    -0.66      0.10     -2.53    -419.79
         12    15  -9661.80    222.44     0.66     -0.10      0.85     109.20
               22   9711.83   -166.53    -0.66      0.10     -2.53     386.83
     29   1    18     -0.00     25.89    -0.00      0.00      0.00       0.06
               21      0.00     25.89     0.00     -0.00     -0.00      -0.06
          2    18     -0.02   1125.00    -0.00      0.00      0.00       3.41
               21      0.02   1125.00     0.00     -0.00     -0.00      -3.41
          3    18      0.00      0.00     0.00      0.00     -0.00       0.00
               21      0.00     -0.00    -0.00     -0.00     -0.00       0.00
         11    18     -0.02   1150.89     0.00      0.00      0.00       3.47
               21      0.02   1150.89    -0.00     -0.00     -0.00      -3.47
         12    18     -0.02   1150.89    -0.00     -0.00      0.00       3.47
               21      0.02   1150.89     0.00      0.00     -0.00      -3.47
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   19
   ************** END OF LATEST ANALYSIS RESULT **************
    75. LOAD LIST 11 12
    76. PARAMETER
    77. CODE CANADIAN
    78. CHECK CODE ALL
 STEEL DESIGN
   WARNING: The current units are in KGS  METE. All the design results are in KN
MET  (unless otherwise noted).
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   20
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.321         LOADCASE   12      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      2      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   12      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      3      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.464         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      4      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.324         LOADCASE   11      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   21
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      5      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
```

```
|  STATUS:        PASS    CRITICAL RATIO:  0.471         LOADCASE   11       |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:      6      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   11       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:      7      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.321         LOADCASE   12       |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:      8      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   12       |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   22
                   STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                 ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|----------------------------------------------------------------------------|
|   MEMBER NO.:      9      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.464         LOADCASE   12       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     10      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.324         LOADCASE   11       |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     11      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.471         LOADCASE   11       |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     12      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   11       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1          |
|----------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   23
                   STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                 ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|----------------------------------------------------------------------------|
|   MEMBER NO.:     13      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.768         LOADCASE   12       |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     14      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.798         LOADCASE   12       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     15      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
```

```
|  STATUS:        PASS    CRITICAL RATIO:  0.776         LOADCASE   11       |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     16      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.747         LOADCASE   11       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   24
                        STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                      ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|----------------------------------------------------------------------------|
|   MEMBER NO.:     17      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.768         LOADCASE   12       |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     18      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.798         LOADCASE   12       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     19      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.776         LOADCASE   11       |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     20      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.747         LOADCASE   11       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8            |
|----------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   25
                        STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                      ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|----------------------------------------------------------------------------|
|   MEMBER NO.:     21      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.009         LOADCASE   12       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     22      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.118         LOADCASE   11       |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     23      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.009         LOADCASE   12       |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.9.1          |
|----------------------------------------------------------------------------|
|----------------------------------------------------------------------------|
|   MEMBER NO.:     24      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.118         LOADCASE   11       |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8.4          |
|----------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   26
                        STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
```

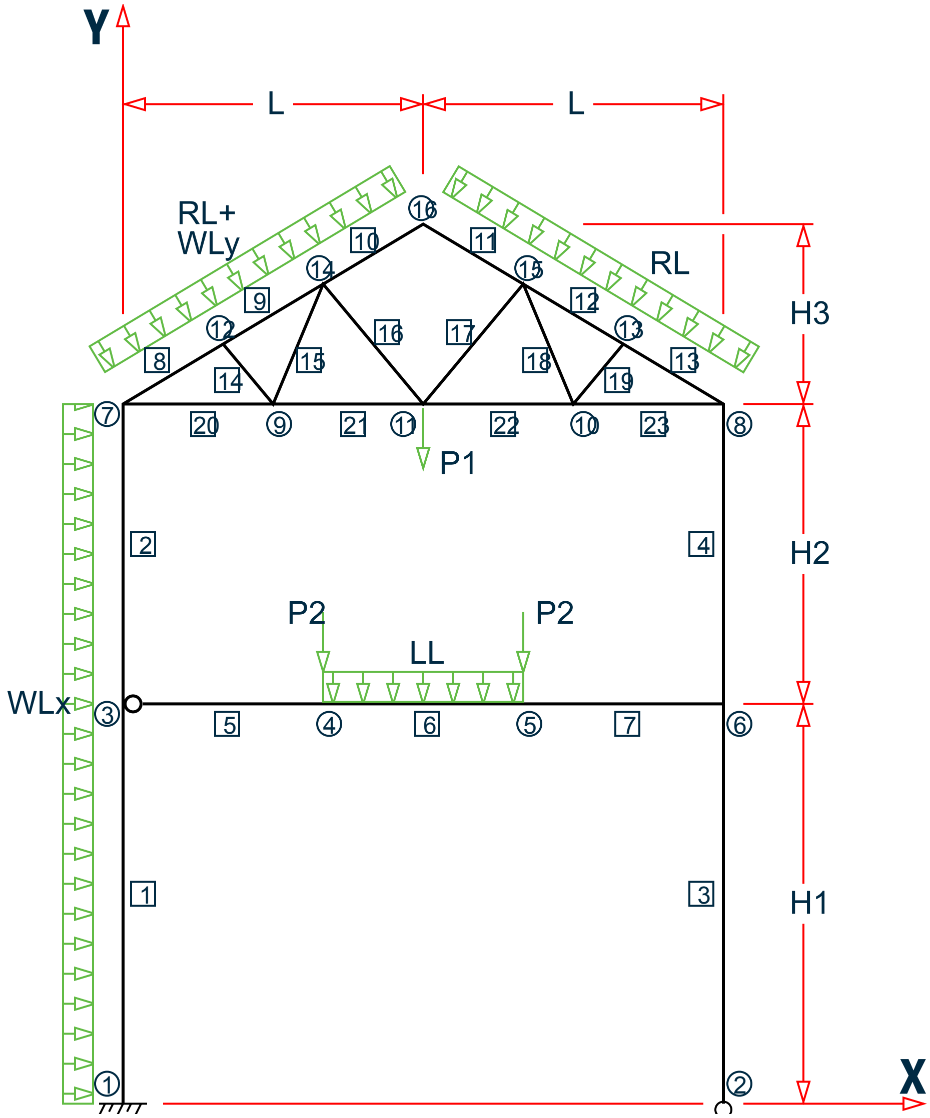
```
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|  MEMBER NO.:     25      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
| STATUS:        PASS    CRITICAL RATIO:  0.675         LOADCASE   12       |
| LOCATION:      2.48    CRITICAL CONDITION:            Cl. 13.8.4          |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|  MEMBER NO.:     26      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
| STATUS:        PASS    CRITICAL RATIO:  0.671         LOADCASE   11       |
| LOCATION:      2.55    CRITICAL CONDITION:            Cl. 13.8.4          |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|  MEMBER NO.:     27      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
| STATUS:        PASS    CRITICAL RATIO:  0.675         LOADCASE   12       |
| LOCATION:      2.48    CRITICAL CONDITION:            Cl. 13.8.4          |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|  MEMBER NO.:     28      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
| STATUS:        PASS    CRITICAL RATIO:  0.671         LOADCASE   11       |
| LOCATION:      2.55    CRITICAL CONDITION:            Cl. 13.8.4          |
|--------------------------------------------------------------------------|
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   27
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|  MEMBER NO.:     29      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
| STATUS:        PASS    CRITICAL RATIO:  0.444         LOADCASE   11       |
| LOCATION:      1.50    CRITICAL CONDITION:            Cl. 13.8.4          |
|--------------------------------------------------------------------------|
  ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL MEMBERS
   79. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:46:26 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. British Design Examples

EX. UK-1 Plane Frame with Steel Design

After one analysis, member selection is requested. Since member sizes change during the member selection, another analysis is done followed by final code checking to verify that the final sizes meet the requirements of the code based on the latest analysis results.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-1 Plane Frame with Steel Design.STD` when you install the program.



where:

L = 4.5 m, H1 = 6.0 m, H2 = 4.5 m, and H 3 = 2.7 m
WLx = 9.0 KN/m, WLy = 15 KN/m, RL = 13.5 KN/m, LL = 17.5 KN/m, P1 = 135 KN, P2 = 35 KN
Members 1, 3, & 4 are a UC356X368X129, Members 5, 6, & 7 are a UB533X210X82, Member 2 is a UC254X254X73, Members 8 through 13 are a UB457X152X52. All other members comprising the truss are a UA100X100X8

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE EXAMPLE PROBLEM NO. 1
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KN
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 ; 2 9 0 ; 3 0 6 ; 4 3 6
5 6 6 ; 6 9 6 ; 7 0 10.5
8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3 12.3 ; 15 6 12.3 ; 16 4.5 13.2
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTY BRITISH
1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
14 TO 23 TA ST UA100X100X8
```

Member properties are from the British steel table. The term ST stands for standard single section.

```
MEMB TRUSS
14 TO 23
```

The above command defines that members 14 through 23 are of type truss. This means that these members can carry only axial tension/compression and no moments.

```
MEMB RELEASE
5 START MZ
```

Member 5 has local moment-z (MZ) released at the start joint. This means that the member cannot carry any moment-z (i.e., strong axis moment) at node 3.

```
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

The `DEFINE MATERAL START` initiates input for a material definition, which can later be assigned to all members. STAAD.Pro has several built-in material definitions, including the `ISOTROPIC STEEL` material used here. The length unit is changed to `MMS` (millimeters) to facilitate the input in familiar units.

```
CONSTANTS
BETA 90.0 MEMB 3 4
MATERIAL STEEL ALL
UNIT METER
```

The `CONSTANT` command initiates input for material constants or material definition references. The `BETA` command specifies that members 3 and 4 are rotated by 90 degrees around their own longitudinal axis. See G. 4.3 Relationship Between Global and Local Coordinates (on page 2092) for the definition of the beta angle.

```
SUPPORT
1 FIXED ; 2 PINNED
```

A fixed support is located at joint 1 and a pinned support at joint 2.

```
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
```

The above `PRINT` commands are self-explanatory. The `LIST` option restricts the print output to the members listed.

```
LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title.

```
SELFWEIGHT Y -1.0
```

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

```
JOINT LOAD
4 5 FY -65. ; 11 FY -155.
```

Load 1 contains joint loads also. FY indicates that the load is a force in the global Y direction.

```
MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The term UNI stands for uniformly distributed load. Loads are applied on members 6, and 8 to 13.

```
CALCULATE RAYLEIGH FREQUENCY
```

The above command at the end of load case 1, is an instruction to perform a natural frequency calculation based on the Rayleigh method using the data in the above load case.

```
LOADING 2 WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 9.0 ; 8 TO 10 UNI Y -15.0
```

Load case 2 is initiated and contains several member loads.

```
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
```

The above command identifies a combination load (case no. 3) with a title. The second line provides the load cases and their respective factors used for the load combination.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
LOAD LIST 1 3
```

The above command activates load cases 1 and 3 only for the commands to follow. This also means that load case 2 will be made inactive.

```
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
```

The above `PRINT` commands are self-explanatory. Also note that all the forces and reactions will be printed for load cases 1 and 3 only.

```
PARAMETER
CODE EN 1993-1-1:2005
NA 1
NSF 0.85 ALL
BEAM 1 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
```

The `PARAMETER` command is used to specify steel design parameters such as net section factor (`NSF`), effective length factor for bending about the minor axis (`KY`), etc. Information on these parameters can be obtained from the manual where the implementation of the code is explained. The `BEAM` parameter is specified to perform design at every 1/12th point along the member length, which is also the default. The `RATIO` parameter specifies that the ratio of actual loading over section capacity should not exceed 0.9.

```
SELECT ALL
```

The above command instructs the program to select the most economic section for all the members based on the results of the analysis.

```
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
```

Although the program selects the most economical section for all members, it is not always practical to use many different sizes in one structure. Grouping is a procedure by which the cross section which has the largest value for the specified attribute, which in this case is the default and hence the AREA, from among the associated member list, is assigned to all members in the list. Hence, the cross sections for members 1, 3 and 4 are replaced with the one with the largest area from among the three.

```
PERFORM ANALYSIS
```

As a result of the selection and grouping, the member sizes are no longer the same as the ones used in the original analysis. Hence, it is necessary to reanalyze the structure using the new properties to get new values of forces in the members.

```
PARAMETER
BEAM 1.0 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
```

A new set of values are now provided for the above parameters. The actual load to member capacity RATIO has been redefined as 1.0. The TRACK parameter tells the program to print out the design results to the intermediate level of descriptively.

```
CHECK CODE ALL
```

With the above command, the latest member sizes with the latest analysis results are checked to verify that they satisfy the CODE specifications.

```
STEEL TAKE OFF
```

This command instructs the program to list the length and weight of all the different member sizes.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE EXAMPLE PROBLEM NO. 1
UNIT METER KN
JOINT COORDINATES
1 0. 0. ; 2 9. 0. ; 3 0. 6. ; 4 3. 6.
5 6. 6. ; 6 9. 6. ; 7 0 10.5
8 9. 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3. 12.3 ; 15 6. 12.3 ; 16 4.5 13.2
MEMBER INCIDENCE
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
MEMBER PROPERTY BRITISH
1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
14 TO 23 TA RA UA100X100X8
*MEMB TRUSS
*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT METER
SUPPORT
```

```
1 FIXED ; 2 PINNED
PRINT MEMBER INFORMATION LIST 1 5 14
PRINT MEMBER PROPERTY LIST 1 2 5 8 14
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -65. ; 11 FY -155.
MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
CALCULATE RAYLEIGH FREQUENCY
LOADING 2  WIND FROM LEFT
MEMBER LOAD
1 2 UNI GX 9.0 ; 8 TO 10 UNI Y -15.0
* 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
LOAD COMB 3 75 PERCENT DL LL WL
1 0.75 2 0.75
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
PARAMETER
CODE EN 1993-1-1:2005
NA 1
NSF 0.85 ALL
BEAM 3 ALL
KY 1.2 MEMB 3 4
RATIO 0.9 ALL
SELECT ALL
GROUP MEMB 1 3 4
GROUP MEMB 5 6 7
GROUP MEMB 8 TO 13
GROUP MEMB 14 TO 23
PERFORM ANALYSIS
PARAMETER
BEAM 3 ALL
RATIO 1.0 ALL
TRACK 1.0 ALL
CHECK CODE ALL
STEEL TAKE OFF
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44: 4                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE PROBLEM NO. 1
INPUT FILE: UK-1 Plane Frame with Steel Design.STD
     2. UNIT METER KN
```

```
    3. JOINT COORDINATES
    4. 1 0. 0. ; 2 9. 0. ; 3 0. 6. ; 4 3. 6.
    5. 5 6. 6. ; 6 9. 6. ; 7 0 10.5
    6. 8 9. 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
    7. 11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
    8. 14 3. 12.3 ; 15 6. 12.3 ; 16 4.5 13.2
    9. MEMBER INCIDENCE
   10. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
   11. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
   12. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
   13. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
   14. 18 10 15 ; 19 10 13 ; 20 7 9
   15. 21 9 11 ; 22 10 11 ; 23 8 10
   16. MEMBER PROPERTY BRITISH
   17. 1 3 4 TA ST UC356X368X129 ; 2 TA ST UC254X254X73
   18. 5 6 7 TA ST UB533X210X82 ; 8 TO 13 TA ST UB457X152X52
   19. 14 TO 23 TA RA UA100X100X8
   20. *MEMB TRUSS
   21. *14 TO 23
   22. MEMB RELEASE
   23. 5 START MZ
   24. 14 TO 23 START MPY 0.99 MPZ 0.99
   25. 14 TO 23 END MPY 0.99 MPZ 0.99
   26. UNIT KNS MMS
   27. DEFINE MATERIAL START
   28. ISOTROPIC STEEL
   29. E 210
   30. POISSON 0.3
   31. DENSITY 7.68191E-008
   32. ALPHA 6E-006
   33. DAMP 0.03
   34. TYPE STEEL
   35. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
   36. END DEFINE MATERIAL
   37. CONSTANTS
   38. MATERIAL STEEL ALL
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    2
   39. BETA 90.0 MEMB 3 4
   40. UNIT METER
   41. SUPPORT
   42. 1 FIXED ; 2 PINNED
   43. PRINT MEMBER INFORMATION LIST 1 5 14
 MEMBER   INFORMAT LIST     1
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    3
  MEMBER INFORMATION
  ------------------
  MEMBER    START    END         LENGTH     BETA
            JOINT   JOINT        (METE)     (DEG)     RELEASES
     1        1       3           6.000     0.00
     5        3       4           3.000     0.00    000001000000
    14        9      12           1.172     0.00    000011000011
  ************ END OF DATA FROM INTERNAL STORAGE ************
   44. PRINT MEMBER PROPERTY LIST 1 2 5 8 14
 MEMBER   PROPERTY LIST     1
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    4
  MEMBER PROPERTIES. UNIT - CM
  -----------------
  MEMB   PROFILE              AX/          IZ/          IY/        IX/
```

```
                               AY           AZ           SZ          SY
   1  ST  UC356X368X129
                             164.00     40200.00     14600.00      152.61
                              36.98       116.11      2260.97      792.19
   2  ST  UC254X254X73        93.10     11400.00      3910.00       57.62
                              21.85        65.07       897.28      307.15
   5  ST  UB533X210X82       105.00     47500.00      2010.00       51.52
                              50.72        49.61      1798.22      192.53
   8  ST  UB457X152X52        66.60     21400.00       645.00       21.37
                              34.18        29.90       951.53       84.65
  14  RA  UA100X100X8         15.50       236.82        59.54        3.31
                               5.33         5.33        33.49       15.20
 ************ END OF DATA FROM INTERNAL STORAGE ************
  45. LOADING 1 DEAD AND LIVE LOAD
  46. SELFWEIGHT Y -1.0
  47. JOINT LOAD
  48. 4 5 FY -65. ; 11 FY -155.
  49. MEMB LOAD
  50. 8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
  51. CALCULATE RAYLEIGH FREQUENCY
  52. LOADING 2  WIND FROM LEFT
  53. MEMBER LOAD
  54. 1 2 UNI GX 9.0 ; 8 TO 10 UNI Y -15.0
  55. * 1/3 RD INCREASE IS ACCOMPLISHED BY 75% LOAD
  56. LOAD COMB 3 75 PERCENT DL LL WL
  57. 1 0.75 2 0.75
  58. PERFORM ANALYSIS
         P R O B L E M   S T A T I S T I C S
         -----------------------------------
   NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
         Using 64-bit analysis engine.
    EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.    5
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      43
 TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
 **********************************************************
 *                                                        *
 * RAYLEIGH FREQUENCY FOR LOADING     1 =    3.48541 CPS  *
 * MAX DEFLECTION =  2.54900 CM   GLO X,  AT JOINT     7  *
 *                                                        *
 **********************************************************
  59. LOAD LIST 1 3
  60. PRINT MEMBER FORCES
MEMBER   FORCES
    EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.    6
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1    239.70     -7.78     0.00      0.00      0.00     -72.75
               3   -232.14      7.78     0.00      0.00      0.00      26.08
         3     1    179.63     85.56     0.00      0.00      0.00     334.46
               3   -173.96    -45.06     0.00      0.00      0.00      57.40
    2    1     3    150.33    -22.49     0.00      0.00      0.00     -26.08
               7   -147.12     22.49     0.00      0.00      0.00     -75.14
         3     3    128.83     -0.18     0.00      0.00      0.00     -57.40
```

```
                7   -126.42     30.56     0.00       0.00       0.00     -11.77
    3    1      2    258.31      0.00    -7.78       0.00       0.00       0.00
                6   -250.75      0.00     7.78       0.00      46.67       0.00
         3      2    244.50      0.00   -15.69       0.00       0.00       0.00
                6   -238.83      0.00    15.69       0.00      94.14       0.00
    4    1      6    142.80      0.00   -22.49       0.00      71.03       0.00
                8   -137.13      0.00    22.49       0.00      30.19       0.00
         3      6    141.64      0.00   -60.93       0.00     140.17       0.00
                8   -137.39      0.00    60.93       0.00     134.03       0.00
    5    1      3    -14.72     81.80     0.00       0.00       0.00       0.00
                4     14.72    -79.38     0.00       0.00       0.00     241.78
         3      3    -45.24     45.12     0.00       0.00       0.00       0.00
                4     45.24    -43.31     0.00       0.00       0.00     132.65
    6    1      4    -14.72     14.38     0.00       0.00       0.00    -241.78
                5     14.72     40.54     0.00       0.00       0.00     202.55
         3      4    -45.24     -5.44     0.00       0.00       0.00    -132.65
                5     45.24     46.63     0.00       0.00       0.00      54.55
    7    1      5    -14.72   -105.54     0.00       0.00       0.00    -202.55
                6     14.72    107.96     0.00       0.00       0.00    -117.70
         3      5    -45.24    -95.38     0.00       0.00       0.00     -54.55
                6     45.24     97.19     0.00       0.00       0.00    -234.31
    8    1      7    167.81     70.72     0.00       0.00       0.00      75.13
               12   -167.35    -46.34     0.00       0.00       0.00      27.25
         3      7    172.82     43.62     0.00       0.00       0.00      11.77
               12   -172.47     -5.65     0.00       0.00       0.00      31.33
    9    1     12    168.29     40.04     0.00       0.00       0.00     -27.24
               14   -167.83    -15.66     0.00       0.00       0.00      75.96
    EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.    7
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
         3     12    167.89     34.90     0.00       0.00       0.00     -31.32
               14   -167.55      3.06     0.00       0.00       0.00      59.17
   10    1     14    188.07    -88.15     0.00       0.00       0.00     -75.98
               16   -187.61    112.53     0.00       0.00       0.00     -99.54
         3     14    154.06    -61.65     0.00       0.00       0.00     -59.19
               16   -153.71     99.62     0.00       0.00       0.00     -81.86
   11    1     15    188.04    -88.20     0.00       0.00       0.00     -76.06
               16   -187.58    112.58     0.00       0.00       0.00     -99.54
         3     15    160.58    -70.47     0.00       0.00       0.00     -57.40
               16   -160.23     88.75     0.00       0.00       0.00     -81.86
   12    1     13    182.56     36.30     0.00       0.00       0.00     -33.87
               15   -182.10    -11.92     0.00       0.00       0.00      76.05
         3     13    123.15     38.74     0.00       0.00       0.00      -5.61
               15   -122.80    -20.45     0.00       0.00       0.00      57.39
   13    1      8    184.91     48.81     0.00       0.00       0.00      30.19
               13   -184.45    -24.43     0.00       0.00       0.00      33.87
         3      8    118.55     88.96     0.00       0.00       0.00     134.01
               13   -118.21    -70.68     0.00       0.00       0.00       5.62
   14    1      9     -6.25      0.05     0.00       0.00       0.00       0.00
               12      6.36      0.04     0.00       0.00       0.00       0.01
         3      9     29.69      0.04     0.00       0.00       0.00      -0.00
               12    -29.61      0.03     0.00       0.00       0.00       0.01
   15    1      9      4.86      0.05     0.00       0.00       0.00      -0.00
               14     -4.64      0.04     0.00       0.00       0.00       0.01
         3      9    -24.97      0.04     0.00       0.00       0.00      -0.00
               14     25.13      0.03     0.00       0.00       0.00       0.01
```

```
    16    1    11   -107.59      0.08     0.00      0.00      0.00      -0.01
               14    107.80      0.10     0.00      0.00      0.00      -0.01
          3    11    -44.00      0.06     0.00      0.00      0.00      -0.01
               14     44.16      0.07     0.00      0.00      0.00      -0.01
    17    1    11    -94.66      0.08     0.00      0.00      0.00      -0.01
               15     94.87      0.10     0.00      0.00      0.00      -0.01
          3    11   -107.68      0.06     0.00      0.00      0.00      -0.01
               15    107.84      0.07     0.00      0.00      0.00      -0.01
    18    1    10    -10.44      0.05     0.00      0.00      0.00      -0.00
               15     10.66      0.04     0.00      0.00      0.00       0.01
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    8
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          3    10     26.57      0.04     0.00      0.00      0.00      -0.00
               15    -26.41      0.03     0.00      0.00      0.00       0.01
    19    1    10     12.13      0.05     0.00      0.00      0.00       0.00
               13    -12.02      0.04     0.00      0.00      0.00       0.01
          3    10    -32.23      0.04     0.00      0.00      0.00       0.00
               13     32.31      0.03     0.00      0.00      0.00       0.01
    20    1     7    -85.02      0.14     0.00      0.00      0.00       0.01
                9     85.02      0.13     0.00      0.00      0.00       0.00
          3     7    -95.19      0.10     0.00      0.00      0.00      -0.00
                9     95.19      0.10     0.00      0.00      0.00       0.00
    21    1     9    -90.88      0.14     0.00      0.00      0.00       0.00
               11     90.88      0.13     0.00      0.00      0.00       0.01
          3     9    -66.58      0.10     0.00      0.00      0.00      -0.00
               11     66.58      0.10     0.00      0.00      0.00       0.01
    22    1    10    -99.16      0.14     0.00      0.00      0.00      -0.00
               11     99.16      0.13     0.00      0.00      0.00       0.01
          3    10    -25.81      0.10     0.00      0.00      0.00       0.00
               11     25.81      0.10     0.00      0.00      0.00       0.01
    23    1     8   -110.95      0.14     0.00      0.00      0.00       0.00
               10    110.95      0.13     0.00      0.00      0.00       0.00
          3     8      5.05      0.11     0.00      0.00      0.00       0.02
               10     -5.05      0.09     0.00      0.00      0.00      -0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   61. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.    9
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      7.78    239.70      0.00      0.00      0.00    -72.75
          3    -85.56    179.63      0.00      0.00      0.00    334.46
     2    1     -7.78    258.31      0.00      0.00      0.00      0.00
          3    -15.69    244.50      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   62. PARAMETER
   63. CODE EN 1993-1-1:2005
   64. NA 1
   65. NSF 0.85 ALL
   66. BEAM 3 ALL
   67. KY 1.2 MEMB 3 4
   68. RATIO 0.9 ALL
   69. SELECT ALL
STEEL DESIGN
```

```
                        STAAD.PRO MEMBER SELECTION - BS EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   10
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      1 ST  UC305X305X118(BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.868        3
                     179.63 C          0.00        334.46        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      2 ST  UC203X203X46 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.882        1
                     147.12 C          0.00        -75.14        4.50
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      3 ST  UC254X254X89 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.736        3
                     238.83 C         94.14         0.00         6.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      4 ST  UC305X305X97 (BRITISH SECTIONS)
                          PASS     EC-6.2.9.1         0.822        3
                     141.64 C       140.17          0.00         0.00
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   11
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      5 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.863        1
                      14.72 T          0.00        241.78        3.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      6 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.883        1
                      14.72 T          0.00       -247.42        0.75
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      7 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.836        3
                      45.24 T          0.00       -234.31        3.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
```

```
=======================================================================
      8 ST  UB356X127X33 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.844        1
                     167.81 C         0.00        75.13       0.00
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   12
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      9 ST  UB254X146X37 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.855        1
                     167.83 C         0.00        75.96       1.75
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     10 ST  UB406X140X39 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.803        1
                     187.61 C         0.00       -99.54       1.75
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     11 ST  UB406X140X39 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.803        1
                     187.58 C         0.00       -99.54       1.75
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     12 ST  UB254X146X37 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.870        1
                     182.10 C         0.00        76.05       1.75
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   13
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     13 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.851        3
                     118.55 C         0.00       134.01       0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     14 RA  UA50x50x4    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.780        3
                      29.68 C         0.00        -0.00       0.10
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     15 RA  UA35x35x4    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.865        1
                        4.75 C         0.00        -0.03       0.98
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
```

```
============================================================================
     16 RA  UA65x50x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.871        1
                   107.69 T         0.00         -0.05       1.17
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   14
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     17 RA  UA65x50x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.862        3
                   107.76 T         0.00         -0.04       1.17
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     18 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.783        3
                    26.57 C         0.00         -0.00       0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     19 RA  UA35x35x4    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662      0.832        1
                    12.07 C         0.00         -0.02       0.68
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     20 RA  UA60x40x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.897        3
                    95.19 T         0.00         -0.06       1.12
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   15
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     21 RA  UA60x40x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.875        1
                    90.88 T         0.00         -0.08       1.12
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     22 RA  UA65x50x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.826        1
                    99.16 T         0.00         -0.08       1.13
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
============================================================================
     23 RA  UA60x40x6    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3      0.888        1
                   110.95 T         0.00         -0.08       1.12
  ************** END OF TABULATED RESULT OF DESIGN **************
   70. GROUP MEMB 1 3 4
  GROUPING BASED ON MEMBER      1  (ST  UC305X305X118   ) LIST=      1....
```

```
    71. GROUP MEMB 5 6 7
   GROUPING BASED ON MEMBER      7  (ST  UB457X191X67    ) LIST=      5....
    72. GROUP MEMB 8 TO 13
   GROUPING BASED ON MEMBER     13  (ST  UB356X171X45    ) LIST=      8....
    73. GROUP MEMB 14 TO 23
      EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   16
   GROUPING BASED ON MEMBER     18  (RA  UA60x60x5       ) LIST=     14....
    74. PERFORM ANALYSIS
    ** ALL CASES BEING MADE ACTIVE BEFORE RE-ANALYSIS. **
   **********************************************************
   *                                                        *
   * RAYLEIGH FREQUENCY FOR LOADING     1 =    2.68348 CPS  *
   * MAX DEFLECTION =  4.57423 CM   GLO X,  AT JOINT     7  *
   *                                                        *
   **********************************************************
    75. PARAMETER
    76. BEAM 3 ALL
    77. RATIO 1.0 ALL
    78. TRACK 1.0 ALL
    79. CHECK CODE ALL
 STEEL DESIGN
                         STAAD.PRO CODE CHECKING - BS EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
 PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
      EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   17
 ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       1 ST  UC305X305X118(BRITISH SECTIONS)
                           PASS     EC-6.3.3-662       0.910         3
                      178.79 C         0.00        351.35        0.00
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     1 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  460.6 MCY=  210.3 PC=  2285.0 PT=  3525.0 MB=  410.5 PV=  582.9|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  3525.00    FX/PZ =   0.05       MRZ=  460.6      MRY=  210.3    |
 |----------------------------------------------------------------------|
 ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       2 ST  UC203X203X46 (BRITISH SECTIONS)
                           PASS     EC-6.3.3-662       0.802         1
                      144.94 C         0.00        -68.59        4.50
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     2 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  116.8 MCY=   54.3 PC=   800.0 PT=  1379.5 MB=   98.9 PV=  229.9|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  1379.45    FX/PZ =   0.11       MRZ=  116.8      MRY=   54.3    |
 |----------------------------------------------------------------------|
 ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
 MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                          FX            MY             MZ       LOCATION
 =======================================================================
       3 ST  UC305X305X118(BRITISH SECTIONS)
                           PASS     EC-6.2.9.1         0.438         3
```

```
                     235.25 C        92.19          0.00        6.00
      EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   18
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     3 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  460.6 MCY=  210.3 PC=  1930.8 PT=  3525.0 MB=  410.5 PV=  582.9|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  3525.00    FX/PZ =   0.07       MRZ=  460.6      MRY=  210.3    |
 |----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      4 ST  UC305X305X118(BRITISH SECTIONS)
                          PASS     EC-6.2.9.1         0.659         3
                     138.96 C       138.54          0.00        0.00
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     4 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  460.6 MCY=  210.3 PC=  2466.9 PT=  3525.0 MB=  438.0 PV=  582.9|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  3525.00    FX/PZ =   0.04       MRZ=  460.6      MRY=  210.3    |
 |----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      5 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.861         1
                      14.61 T         0.00        241.23        3.00
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     5 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  345.5 MCY=   55.7 PC=  1485.4 PT=  2009.3 MB=  280.1 PV=  555.4|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  2009.25    FX/PZ =   0.01       MRZ=  345.5      MRY=   55.7    |
 |----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      EXAMPLE PROBLEM NO. 1                                    -- PAGE NO.   19
      6 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.882         1
                      14.61 T         0.00       -246.94        0.75
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     6 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  345.5 MCY=   55.7 PC=  1485.4 PT=  2009.3 MB=  280.1 PV=  555.4|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000   |
 | PZ=  2009.25    FX/PZ =   0.01       MRZ=  345.5      MRY=   55.7    |
 |----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      7 ST  UB457X191X67 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.824         3
                      44.31 T         0.00       -230.73        3.00
 |----------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB     7 UNIT - kN,m  SECTION CLASS  1   |
 |MCZ=  345.5 MCY=   55.7 PC=  1485.4 PT=  2009.3 MB=  280.1 PV=  555.4|
```

```
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
| PZ=  2009.25    FX/PZ =   0.02       MRZ=  345.5      MRY=   55.7   |
|---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      8 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.512         1
                     165.72 C         0.00          68.59       0.00
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB     8 UNIT - kN,m  SECTION CLASS  1  |
|MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
| PZ=  1346.55    FX/PZ =   0.12       MRZ=  182.1      MRY=   34.5   |
|---------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   20
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      9 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.573         1
                     166.04 C         0.00          77.15       1.75
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB     9 UNIT - kN,m  SECTION CLASS  1  |
|MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
| PZ=  1346.55    FX/PZ =   0.12       MRZ=  182.1      MRY=   34.5   |
|---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     10 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.682         1
                     186.60 C         0.00         -97.18       1.75
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    10 UNIT - kN,m  SECTION CLASS  1  |
|MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
| PZ=  1346.55    FX/PZ =   0.14       MRZ=  182.1      MRY=   34.5   |
|---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     11 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.682         1
                     186.46 C         0.00         -97.18       1.75
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    11 UNIT - kN,m  SECTION CLASS  1  |
|MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
| PZ=  1346.55    FX/PZ =   0.14       MRZ=  182.1      MRY=   34.5   |
|---------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   21
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
```

```
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     12 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.584         1
                    175.61 C         0.00         77.56        1.75
 |---------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB    12 UNIT - kN,m  SECTION CLASS  1  |
 |MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
 | PZ=  1346.55    FX/PZ =   0.13       MRZ=  182.1      MRY=   34.5   |
 |---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     13 ST  UB356X171X45 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.831         3
                    121.12 C         0.00        130.01        0.00
 |---------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB    13 UNIT - kN,m  SECTION CLASS  1  |
 |MCZ=  182.1 MCY=   34.5 PC=  1193.5 PT=  1346.6 MB=  175.5 PV=  363.1|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.127    K =  1.000  |
 | PZ=  1346.55    FX/PZ =   0.09       MRZ=  182.1      MRY=   34.5   |
 |---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     14 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.285         3
                     19.99 C         0.00          0.00        0.00
 |---------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB    14 UNIT - kN,m  SECTION CLASS  4  |
 |MCZ=    1.5 MCY=    0.8 PC=    75.6 PT=   136.8 MB=    1.9 PV=   27.1|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
 | PZ=   136.77    FX/PZ =   0.15       MRZ=    1.5      MRY=    0.8   |
 |---------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   22
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     15 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.174         1
                       5.91 C        0.00         -0.00        0.00
 |---------------------------------------------------------------------|
 | CALCULATED CAPACITIES FOR MEMB    15 UNIT - kN,m  SECTION CLASS  4  |
 |MCZ=    1.5 MCY=    0.8 PC=    35.3 PT=   136.8 MB=    1.6 PV=   27.1|
 | BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
 | PZ=   136.77    FX/PZ =   0.04       MRZ=    1.5      MRY=    0.8   |
 |---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     16 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.797         1
```

```
                    105.96 T          0.00          -0.02        1.17
|----------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    16 UNIT - kN,m  SECTION CLASS  4   |
|MCZ=    1.5 MCY=    0.8 PC=    25.5 PT=   136.8 MB=    1.5 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.77       MRZ=    1.5      MRY=    0.8   |
|----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     17 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.781         3
                    104.19 T          0.00          -0.01        1.17
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   23
|----------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    17 UNIT - kN,m  SECTION CLASS  4   |
|MCZ=    1.5 MCY=    0.8 PC=    25.5 PT=   136.8 MB=    1.5 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.76       MRZ=    1.5      MRY=    0.8   |
|----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     18 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.670         3
                     22.72 C          0.00          -0.00        0.00
|----------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    18 UNIT - kN,m  SECTION CLASS  4   |
|MCZ=    1.5 MCY=    0.8 PC=    35.3 PT=   136.8 MB=    1.6 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.17       MRZ=    1.5      MRY=    0.8   |
|----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     19 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.206         3
                     27.43 T          0.00          -0.00        0.59
|----------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    19 UNIT - kN,m  SECTION CLASS  4   |
|MCZ=    1.5 MCY=    0.8 PC=    75.6 PT=   136.8 MB=    1.9 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.20       MRZ=    1.5      MRY=    0.8   |
|----------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   24
     20 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.639         1
                     83.85 T          0.00          -0.03        1.12
|----------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    20 UNIT - kN,m  SECTION CLASS  4   |
|MCZ=    1.5 MCY=    0.8 PC=    27.4 PT=   136.8 MB=    1.5 PV=   27.1|
```

```
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.61       MRZ=    1.5      MRY=    0.8   |
|---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     21 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.691         1
                      90.76 T          0.00         -0.03        1.12
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    21 UNIT - kN,m  SECTION CLASS  4  |
|MCZ=    1.5 MCY=    0.8 PC=    27.4 PT=   136.8 MB=    1.5 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.66       MRZ=    1.5      MRY=    0.8   |
|---------------------------------------------------------------------|
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     22 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.738         1
                      97.10 T          0.00         -0.03        1.13
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    22 UNIT - kN,m  SECTION CLASS  4  |
|MCZ=    1.5 MCY=    0.8 PC=    27.4 PT=   136.8 MB=    1.5 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.71       MRZ=    1.5      MRY=    0.8   |
|---------------------------------------------------------------------|
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   25
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     23 RA  UA60x60x5    (BRITISH SECTIONS)
                          PASS     EC-6.2.9.2/3       0.774         1
                     101.92 T          0.00         -0.03        1.12
|---------------------------------------------------------------------|
| CALCULATED CAPACITIES FOR MEMB    23 UNIT - kN,m  SECTION CLASS  4  |
|MCZ=    1.5 MCY=    0.8 PC=    27.4 PT=   136.8 MB=    1.5 PV=   27.1|
| BUCKLING  CO-EFFICIENTS C1  AND  K  :    C1 =  1.132    K =  1.000  |
| PZ=   136.77    FX/PZ =   0.75       MRZ=    1.5      MRY=    0.8   |
|---------------------------------------------------------------------|
  ************** END OF TABULATED RESULT OF DESIGN **************
   80. STEEL TAKE OFF
     EXAMPLE PROBLEM NO. 1                                   -- PAGE NO.   26
STEEL TAKE OFF
  STEEL TAKE-OFF
  --------------
     PROFILE                LENGTH(METE)      WEIGHT(KNS )
  ST  UC305X305X118              16.50           19.013
  ST  UC203X203X46                4.50            2.029
  ST  UB457X191X67                9.00            5.911
  ST  UB356X171X45               10.50            4.620
  RA  UA60x60x5                  19.93            0.891
                                            ----------------
                                   TOTAL =        32.464
  ************ END OF DATA FROM INTERNAL STORAGE ************
```

```
   81. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44: 5 ****
     EXAMPLE PROBLEM NO. 1                                  -- PAGE NO.   27
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-2 Area Load Generation on Floor Structure

A floor structure (bound by global X-Z axis) made up of steel beams is subjected to area load (i.e., load/area of floor). Load generation based on one-way distribution is illustrated in this example.

In the case of loads such as joint loads and member loads, the magnitude and direction of the load at the applicable joints and members is directly known from the input. However, the area load is a different sort of load where a load intensity on the given area has to be converted to joint and member loads. The calculations required to perform this conversion are done only during the analysis. Consequently, the loads generated from the `AREA LOAD` command can be viewed only after the analysis is completed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-2 Area Load Generation on Floor Structure.STD` when you install the program.

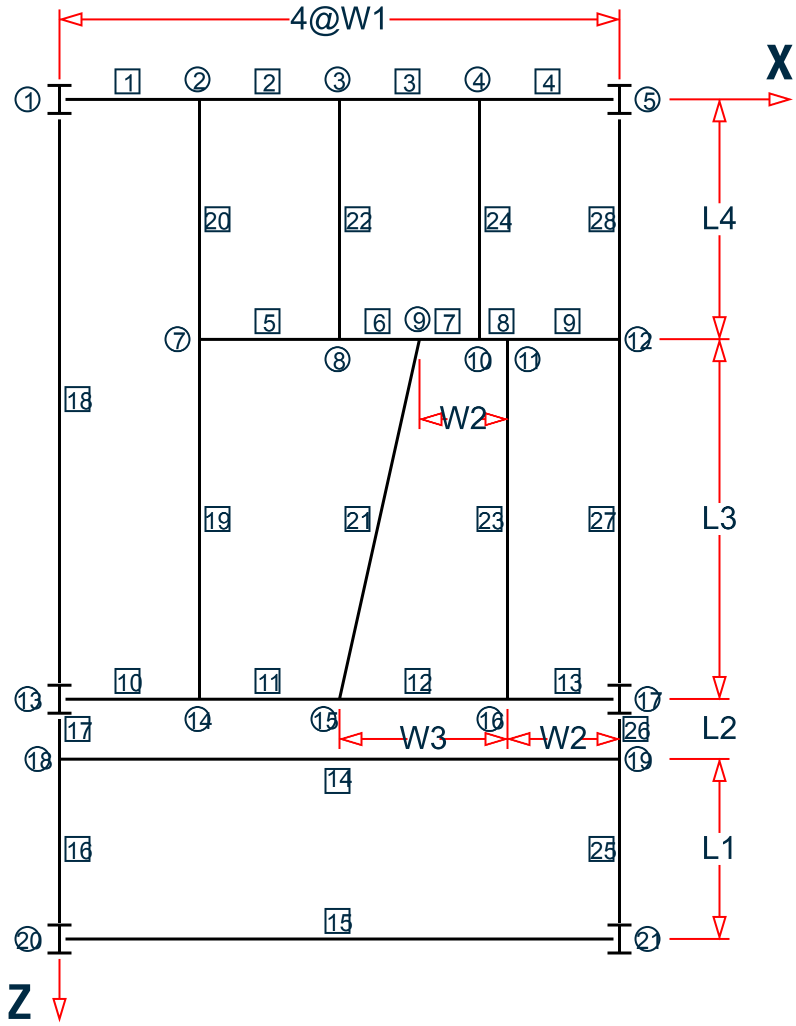


Figure 542: Example Problem No. 2

Where:

W1 = 1.5 m, W2 = 1.0 m, W3 = 1.5 m, L1 = 2.0 m, L2 = 4.5 m, L3 = 3.0 m

Actual input is shown in bold lettering followed by explanation.

```
STAAD FLOOR A FLOOR FRAME DESIGN WITH AREA LOAD
```

Every input has to start with the term STAAD. The term FLOOR signifies that the structure is a floor structure and the structure is in the x – z plane.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 5 6 0 0 ; 7 1.5 0 3
8 3 0 3 ; 9 4 0 3 ; 10 4.5 0 3 ; 11 5 0 3
12 6 0 3 ; 13 0 0 7.5 ; 14 1.5 0. 7.5
15 3.5 0 7.5
```

```
16 5 0 7.5 ; 17 6 0 7.5 ; 18 0 0 8.5
19 6 0 8.5 ; 20 0 0 10.5 ; 21 6 0 10.5
```

Joint numbers followed by X, Y and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). Joints between 1 and 5 (i.e., 2, 3, 4) are generated in the first line of input, taking advantage of the equal spacing between the joints (refer to TR.11 Joint Coordinates Specification (on page 2217) for more information).

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12
```

Defines the members by the joints to which they are connected.

```
MEMB PROP BRITISH
1 TO 28 TABLE ST UB305X165X40
```

Member properties are specified from the British steel table. The term `ST` stands for standard single section.

```
* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
```

The first set of members (1 5 10 etc) have local moment-z (MZ) released at the start joint. This means that these members cannot carry any moment-z (i.e., strong axis moment) at the start joint. The second set of members have MZ released at the end joints.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT METER
```

Define the material properties for steel. The units are changed to millimeters and then back to meters in order to facilite inputing the values in familiar units.

```
CONSTANT
MATERIAL STEEL ALL
```

The `CONSTANT` command instructs the program to use the defined steel material for all members.

```
SUPPORT
1 5 13 17 20 21 FIXED
```

A fixed support has been specified at the above joints.

```
   LOADING 1 14.5 KN/sq.m. DL+LL
```

Load case 1 is initiated followed by a title.

```
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -14.5
```

All members within a range of -0.5 meters to 0.5 meters in the global Y direction (which is the entire floor of this model) are subjected to an floor load of 14.5 KN/sq.m. The program converts area loads into individual member loads.

```
   PERFORM ANALYSIS PRINT LOAD DATA
```

This command instructs the program to proceed with the analysis. The `PRINT LOAD DATA` command is specified to obtain a listing of the member loads which were generated from the `FLOOR LOAD`.

```
   PARAMETERS
   CODE BRITISH
   BEAM 1 ALL
   DMAX 0.6 ALL
   DMIN 0.3 ALL
   UNL 0.3 ALL
```

The `PARAMETER` command is used to specify steel design parameters (refer to D3.B.6 Design Parameters (on page 1384)). Design is to be performed per the specifications of the BS 5950 2000 Code. The `BEAM` parameter is specified to perform design at every 1/12th point along the member length. `DMAX` and `DMIN` specify maximum and minimum depth limitations to be used during member selection. `UNL` is used to specify unsupported length of the compression flange to be used for calculation of allowable bending stress.

```
   SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
```

The above command instructs the program to select the most economical section from the British steel table for the members listed.

```
   FINISH
```

The `FINISH` command terminates the STAAD run.

Input File

```
STAAD FLOOR  A FLOOR FRAME DESIGN WITH AREA LOAD
UNIT METER KNS
JOINT COORDINATES
1 0. 0. 0. 5 6. 0. 0. ; 7 1.5 0. 3.
8 3. 0. 3. ; 9 4. 0. 3. ; 10 4.5 0. 3.
11 5. 0 3. ; 12 6. 0. 3. ; 13 0. 0. 7.5
14 1.5 0. 7.5 ; 15 3.5 0. 7.5
16 5. 0. 7.5 ; 17 6. 0. 7.5 18 0. 0. 8.5
19 6. 0. 8.5 ; 20 0. 0. 10.5 ; 21 6. 0. 10.5
MEMBER INCIDENCES
1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
19 7 14 ; 20 2 7 ; 21 9 15
22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
26 17 19 ; 27 12 17 ; 28 5 12
MEMB PROP BRITISH
1 TO 28 TABLE ST UB305X165X40
* MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
MEMB RELEASE
```

```
1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
UNIT MMS KN
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KN
SUPPORT
1 5 13 17 20 21 FIXED
LOADING 1 14.5 KN/sq.m. DL+LL
FLOOR LOAD
YRANGE -0.5 0.5 FLOAD -14.5
PERFORM ANALYSIS PRINT LOAD DATA
PARAMETERS
CODE EN 1993-1-1:2005
NA 1
BEAM 3 ALL
DMAX 0.6 ALL
DMIN 0.3 ALL
UNL 0.3 ALL
SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:33                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD FLOOR  A FLOOR FRAME DESIGN WITH AREA LOAD
INPUT FILE: UK-2 Area Load Generation on Floor Structure.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0. 0. 0. 5 6. 0. 0. ; 7 1.5 0. 3.
     5. 8 3. 0. 3. ; 9 4. 0. 3. ; 10 4.5 0. 3.
     6. 11 5. 0 3. ; 12 6. 0. 3. ; 13 0. 0. 7.5
     7. 14 1.5 0. 7.5 ; 15 3.5 0. 7.5
     8. 16 5. 0. 7.5 ; 17 6. 0. 7.5 18 0. 0. 8.5
     9. 19 6. 0. 8.5 ; 20 0. 0. 10.5 ; 21 6. 0. 10.5
    10. MEMBER INCIDENCES
```

```
   11. 1 1 2 4 ; 5 7 8 9 ; 10 13 14 13 ; 14 18 19
   12. 15 20 21 ; 16 18 20 ; 17 13 18 ; 18 1 13
   13. 19 7 14 ; 20 2 7 ; 21 9 15
   14. 22 3 8 ; 23 11 16 ; 24 4 10 ; 25 19 21
   15. 26 17 19 ; 27 12 17 ; 28 5 12
   16. MEMB PROP BRITISH
   17. 1 TO 28 TABLE ST UB305X165X40
   18. * MEMBERS WITH PINNED ENDS ARE RELEASED FOR MZ
   19. MEMB RELEASE
   20. 1 5 10 14 15 18 17 28 26 20 TO 24 START MZ
   21. 4 9 13 14 15 18 16 27 25 19 21 TO 24 END MZ
   22. UNIT MMS
   23. DEFINE MATERIAL START
   24. ISOTROPIC STEEL
   25. E 210
   26. POISSON 0.3
   27. DENSITY 7.68191E-008
   28. ALPHA 6E-006
   29. DAMP 0.03
   30. TYPE STEEL
   31. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
   32. END DEFINE MATERIAL
   33. UNIT MMS KN
   34. CONSTANTS
   35. MATERIAL STEEL ALL
   36. UNIT METER KN
   37. SUPPORT
   38. 1 5 13 17 20 21 FIXED
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    2
   39. LOADING 1 14.5 KN/SQ.M. DL+LL
   40. FLOOR LOAD
   41. YRANGE -0.5 0.5 FLOAD -14.5
**NOTE** about Floor/OneWay Loads/Weights.
  Please note that depending on the shape of the floor you may
  have to break up the FLOOR/ONEWAY LOAD into multiple commands.
  For details please refer to Technical Reference Manual
  Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
   42. PERFORM ANALYSIS PRINT LOAD DATA
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         20  NUMBER OF MEMBERS      28
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      42
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    3
  LOADING     1  14.5 KN/SQ.M. DL+LL
  -----------
  MEMBER LOAD - UNIT KN   METE
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
     1                            -0.0637 GY   0.06
     1                            -0.1912 GY   0.15
     1                            -0.3186 GY   0.24
     1                            -0.4460 GY   0.33
     1                            -0.5735 GY   0.42
     1                            -0.7009 GY   0.52
```

```
      1                            -0.8284 GY   0.61
      1                            -0.9558 GY   0.70
      1                            -0.9558 GY   0.80
      1                            -0.8284 GY   0.89
      1                            -0.7009 GY   0.98
      1                            -0.5735 GY   1.08
      1                            -0.4460 GY   1.17
      1                            -0.3186 GY   1.26
      1                            -0.1912 GY   1.35
      1                            -0.0637 GY   1.44
     20   -10.8750 GY   0.75    3.00
     20                            -0.0637 GY   0.06
     20                            -0.1912 GY   0.15
     20                            -0.3186 GY   0.24
     20                            -0.4460 GY   0.33
     20                            -0.5735 GY   0.42
     20                            -0.7009 GY   0.52
     20                            -0.8284 GY   0.61
     20                            -0.9558 GY   0.70
     19   -10.8750 GY   0.00    3.75
     19                            -0.9558 GY   3.80
     19                            -0.8284 GY   3.89
     19                            -0.7009 GY   3.98
     19                            -0.5735 GY   4.08
     19                            -0.4460 GY   4.17
     19                            -0.3186 GY   4.26
     19                            -0.1912 GY   4.35
     19                            -0.0637 GY   4.44
     10                            -0.0637 GY   0.06
     10                            -0.1912 GY   0.15
     10                            -0.3186 GY   0.24
     10                            -0.4460 GY   0.33
     10                            -0.5735 GY   0.42
     10                            -0.7009 GY   0.52
     10                            -0.8284 GY   0.61
     10                            -0.9558 GY   0.70
     10                            -0.9558 GY   0.80
     10                            -0.8284 GY   0.89
     10                            -0.7009 GY   0.98
     10                            -0.5735 GY   1.08
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    4
     10                            -0.4460 GY   1.17
     10                            -0.3186 GY   1.26
     10                            -0.1912 GY   1.35
     10                            -0.0637 GY   1.44
     18                            -0.0637 GY   0.06
     18                            -0.1912 GY   0.15
     18                            -0.3186 GY   0.24
     18                            -0.4460 GY   0.33
     18                            -0.5735 GY   0.42
     18                            -0.7009 GY   0.52
     18                            -0.8284 GY   0.61
     18                            -0.9558 GY   0.70
     18   -10.8750 GY   0.75    6.75
     18                            -0.9558 GY   6.80
     18                            -0.8284 GY   6.89
     18                            -0.7009 GY   6.98
     18                            -0.5735 GY   7.08
```

```
     18                            -0.4460 GY   7.17
     18                            -0.3186 GY   7.26
     18                            -0.1912 GY   7.35
     18                            -0.0637 GY   7.44
      2                            -0.0637 GY   0.06
      2                            -0.1912 GY   0.15
      2                            -0.3186 GY   0.24
      2                            -0.4460 GY   0.33
      2                            -0.5735 GY   0.42
      2                            -0.7009 GY   0.52
      2                            -0.8284 GY   0.61
      2                            -0.9558 GY   0.70
      2                            -0.9558 GY   0.80
      2                            -0.8284 GY   0.89
      2                            -0.7009 GY   0.98
      2                            -0.5735 GY   1.08
      2                            -0.4460 GY   1.17
      2                            -0.3186 GY   1.26
      2                            -0.1912 GY   1.35
      2                            -0.0637 GY   1.44
     22                            -0.0637 GY   0.06
     22                            -0.1912 GY   0.15
     22                            -0.3186 GY   0.24
     22                            -0.4460 GY   0.33
     22                            -0.5735 GY   0.42
     22                            -0.7009 GY   0.52
     22                            -0.8284 GY   0.61
     22                            -0.9558 GY   0.70
     22   -10.8750 GY   0.75    2.25
     22                            -0.9558 GY   2.30
     22                            -0.8284 GY   2.39
     22                            -0.7009 GY   2.48
     22                            -0.5735 GY   2.58
     22                            -0.4460 GY   2.67
     22                            -0.3186 GY   2.76
     22                            -0.1912 GY   2.85
     22                            -0.0637 GY   2.94
      5                            -0.0637 GY   0.06
      5                            -0.1912 GY   0.15
      A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.    5
      5                            -0.3186 GY   0.24
      5                            -0.4460 GY   0.33
      5                            -0.5735 GY   0.42
      5                            -0.7009 GY   0.52
      5                            -0.8284 GY   0.61
      5                            -0.9558 GY   0.70
      5                            -0.9558 GY   0.80
      5                            -0.8284 GY   0.89
      5                            -0.7009 GY   0.98
      5                            -0.5735 GY   1.08
      5                            -0.4460 GY   1.17
      5                            -0.3186 GY   1.26
      5                            -0.1912 GY   1.35
      5                            -0.0637 GY   1.44
     20                            -0.0637 GY   0.06
     20                            -0.1912 GY   0.15
     20                            -0.3186 GY   0.24
     20                            -0.4460 GY   0.33
```

```
     20                            -0.5735 GY   0.42
     20                            -0.7009 GY   0.52
     20                            -0.8284 GY   0.61
     20                            -0.9558 GY   0.70
     20   -10.8750 GY   0.75    2.25
     20                            -0.9558 GY   2.30
     20                            -0.8284 GY   2.39
     20                            -0.7009 GY   2.48
     20                            -0.5735 GY   2.58
     20                            -0.4460 GY   2.67
     20                            -0.3186 GY   2.76
     20                            -0.1912 GY   2.85
     20                            -0.0637 GY   2.94
      3                            -0.0637 GY   0.06
      3                            -0.1912 GY   0.15
      3                            -0.3186 GY   0.24
      3                            -0.4460 GY   0.33
      3                            -0.5735 GY   0.42
      3                            -0.7009 GY   0.52
      3                            -0.8284 GY   0.61
      3                            -0.9558 GY   0.70
      3                            -0.9558 GY   0.80
      3                            -0.8284 GY   0.89
      3                            -0.7009 GY   0.98
      3                            -0.5735 GY   1.08
      3                            -0.4460 GY   1.17
      3                            -0.3186 GY   1.26
      3                            -0.1912 GY   1.35
      3                            -0.0637 GY   1.44
     24                            -0.0637 GY   0.06
     24                            -0.1912 GY   0.15
     24                            -0.3186 GY   0.24
     24                            -0.4460 GY   0.33
     24                            -0.5735 GY   0.42
     24                            -0.7009 GY   0.52
     24                            -0.8284 GY   0.61
     24                            -0.9558 GY   0.70
     24   -10.8750 GY   0.75    2.25
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    6
     24                            -0.9558 GY   2.30
     24                            -0.8284 GY   2.39
     24                            -0.7009 GY   2.48
     24                            -0.5735 GY   2.58
     24                            -0.4460 GY   2.67
     24                            -0.3186 GY   2.76
     24                            -0.1912 GY   2.85
     24                            -0.0637 GY   2.94
      7                            -0.4248 GY   0.03
      7                            -0.3682 GY   0.09
      7                            -0.3115 GY   0.16
      7                            -0.2549 GY   0.22
      7                            -0.1982 GY   0.28
      7                            -0.1416 GY   0.34
      7                            -0.0850 GY   0.40
      7                            -0.0283 GY   0.46
      6                            -0.0637 GY   0.06
      6                            -0.1912 GY   0.15
      6                            -0.3186 GY   0.24
```

```
      6                            -0.4460 GY   0.33
      6                            -0.5735 GY   0.42
      6                            -0.7009 GY   0.52
      6                            -0.8284 GY   0.61
      6                            -0.9558 GY   0.70
      6                            -0.3328 GY   0.77
      6                            -0.3186 GY   0.80
      6                            -0.3044 GY   0.83
      6                            -0.2903 GY   0.86
      6                            -0.2761 GY   0.89
      6                            -0.2620 GY   0.92
      6                            -0.2478 GY   0.95
      6                            -0.2336 GY   0.98
     22                            -0.0637 GY   0.06
     22                            -0.1912 GY   0.15
     22                            -0.3186 GY   0.24
     22                            -0.4460 GY   0.33
     22                            -0.5735 GY   0.42
     22                            -0.7009 GY   0.52
     22                            -0.8284 GY   0.61
     22                            -0.9558 GY   0.70
     22   -10.8750 GY   0.75    2.25
     22                            -0.9558 GY   2.30
     22                            -0.8284 GY   2.39
     22                            -0.7009 GY   2.48
     22                            -0.5735 GY   2.58
     22                            -0.4460 GY   2.67
     22                            -0.3186 GY   2.76
     22                            -0.1912 GY   2.85
     22                            -0.0637 GY   2.94
      4                            -0.0637 GY   0.06
      4                            -0.1912 GY   0.15
      4                            -0.3186 GY   0.24
      4                            -0.4460 GY   0.33
      4                            -0.5735 GY   0.42
      4                            -0.7009 GY   0.52
      4                            -0.8284 GY   0.61
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    7
      4                            -0.9558 GY   0.70
      4                            -0.9558 GY   0.80
      4                            -0.8284 GY   0.89
      4                            -0.7009 GY   0.98
      4                            -0.5735 GY   1.08
      4                            -0.4460 GY   1.17
      4                            -0.3186 GY   1.26
      4                            -0.1912 GY   1.35
      4                            -0.0637 GY   1.44
     28                            -0.0637 GY   0.06
     28                            -0.1912 GY   0.15
     28                            -0.3186 GY   0.24
     28                            -0.4460 GY   0.33
     28                            -0.5735 GY   0.42
     28                            -0.7009 GY   0.52
     28                            -0.8284 GY   0.61
     28                            -0.9558 GY   0.70
     28   -10.8750 GY   0.75    2.25
     28                            -0.9558 GY   2.30
     28                            -0.8284 GY   2.39
```

```
     28                            -0.7009 GY   2.48
     28                            -0.5735 GY   2.58
     28                            -0.4460 GY   2.67
     28                            -0.3186 GY   2.76
     28                            -0.1912 GY   2.85
     28                            -0.0637 GY   2.94
      9                            -0.2336 GY   0.02
      9                            -0.2478 GY   0.05
      9                            -0.2620 GY   0.08
      9                            -0.2761 GY   0.11
      9                            -0.2903 GY   0.14
      9                            -0.3044 GY   0.17
      9                            -0.3186 GY   0.20
      9                            -0.3328 GY   0.23
      9                            -0.9558 GY   0.30
      9                            -0.8284 GY   0.39
      9                            -0.7009 GY   0.48
      9                            -0.5735 GY   0.58
      9                            -0.4460 GY   0.67
      9                            -0.3186 GY   0.76
      9                            -0.1912 GY   0.85
      9                            -0.0637 GY   0.94
      8                            -0.0283 GY   0.04
      8                            -0.0850 GY   0.10
      8                            -0.1416 GY   0.16
      8                            -0.1982 GY   0.22
      8                            -0.2549 GY   0.28
      8                            -0.3115 GY   0.34
      8                            -0.3682 GY   0.41
      8                            -0.4248 GY   0.47
     24                            -0.0637 GY   0.06
     24                            -0.1912 GY   0.15
     24                            -0.3186 GY   0.24
     24                            -0.4460 GY   0.33
     24                            -0.5735 GY   0.42
     24                            -0.7009 GY   0.52
      A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.    8
     24                            -0.8284 GY   0.61
     24                            -0.9558 GY   0.70
     24   -10.8750 GY   0.75    2.25
     24                            -0.9558 GY   2.30
     24                            -0.8284 GY   2.39
     24                            -0.7009 GY   2.48
     24                            -0.5735 GY   2.58
     24                            -0.4460 GY   2.67
     24                            -0.3186 GY   2.76
     24                            -0.1912 GY   2.85
     24                            -0.0637 GY   2.94
     21                            -0.2931 GY   0.19
     21                            -0.8794 GY   0.45
     21                            -1.4656 GY   0.73
     21                            -2.0519 GY   1.02
     21                            -2.6381 GY   1.30
     21                            -3.2244 GY   1.59
     21                            -3.8106 GY   1.88
     21                            -4.3969 GY   2.16
     21                            -4.2408 GY   2.44
     21                            -3.6754 GY   2.72
```

```
     21                            -3.1099 GY   3.00
     21                            -2.5445 GY   3.27
     21                            -1.9790 GY   3.55
     21                            -1.4136 GY   3.82
     21                            -0.8482 GY   4.10
     21                            -0.2827 GY   4.34
     11                            -0.2986 GY   0.09
     11                            -0.8958 GY   0.22
     11                            -1.4929 GY   0.36
     11                            -2.0901 GY   0.50
     11                            -2.6873 GY   0.64
     11                            -3.2845 GY   0.78
     11                            -3.8816 GY   0.92
     11                            -4.4788 GY   1.06
     11                            -3.4509 GY   1.18
     11                            -2.9908 GY   1.29
     11                            -2.5306 GY   1.40
     11                            -2.0705 GY   1.51
     11                            -1.6104 GY   1.62
     11                            -1.1503 GY   1.72
     11                            -0.6902 GY   1.83
     11                            -0.2301 GY   1.93
     19                            -0.2773 GY   0.18
     19                            -0.8318 GY   0.42
     19                            -1.3863 GY   0.69
     19                            -1.9408 GY   0.95
     19                            -2.4953 GY   1.22
     19                            -3.0499 GY   1.49
     19                            -3.6044 GY   1.76
     19                            -4.1589 GY   2.03
     19                            -4.4788 GY   2.31
     19                            -3.8816 GY   2.60
     19                            -3.2845 GY   2.89
     19                            -2.6873 GY   3.18
     19                            -2.0901 GY   3.47
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.    9
     19                            -1.4929 GY   3.76
     19                            -0.8958 GY   4.05
     19                            -0.2986 GY   4.31
      6                            -2.6866 GY   0.06
      6                            -2.3284 GY   0.19
      6                            -1.9702 GY   0.31
      6                            -1.6120 GY   0.44
      6                            -1.2537 GY   0.56
      6                            -0.8955 GY   0.68
      6                            -0.5373 GY   0.81
      6                            -0.1791 GY   0.92
      5                            -0.2773 GY   0.09
      5                            -0.8318 GY   0.22
      5                            -1.3863 GY   0.36
      5                            -1.9408 GY   0.50
      5                            -2.4953 GY   0.64
      5                            -3.0499 GY   0.78
      5                            -3.6044 GY   0.92
      5                            -4.1589 GY   1.06
      5                            -1.4299 GY   1.15
      5                            -1.3808 GY   1.20
      5                            -1.3316 GY   1.25
```

```
      5                          -1.2825 GY   1.29
      5                          -1.2334 GY   1.34
      5                          -1.1842 GY   1.38
      5                          -1.1351 GY   1.43
      5                          -1.0859 GY   1.48
     23                          -0.1722 GY   0.20
     23                          -0.5166 GY   0.47
     23                          -0.8609 GY   0.76
     23                          -1.2053 GY   1.06
     23                          -1.5497 GY   1.36
     23                          -1.8941 GY   1.65
     23                          -2.2384 GY   1.95
     23                          -2.5828 GY   2.25
     23                          -2.2600 GY   2.53
     23                          -1.9586 GY   2.79
     23                          -1.6573 GY   3.05
     23                          -1.3560 GY   3.31
     23                          -1.0546 GY   3.58
     23                          -0.7533 GY   3.84
     23                          -0.4520 GY   4.09
     23                          -0.1507 GY   4.33
     12                          -0.2062 GY   0.07
     12                          -0.6185 GY   0.17
     12                          -1.0309 GY   0.27
     12                          -1.4432 GY   0.38
     12                          -1.8555 GY   0.49
     12                          -2.2679 GY   0.60
     12                          -2.6802 GY   0.71
     12                          -3.0926 GY   0.81
     12                          -2.2600 GY   0.91
     12                          -1.9586 GY   0.98
     12                          -1.6573 GY   1.06
     12                          -1.3560 GY   1.14
     12                          -1.0546 GY   1.22
      A FLOOR FRAME DESIGN WITH AREA LOAD                    -- PAGE NO.   10
     12                          -0.7533 GY   1.30
     12                          -0.4520 GY   1.38
     12                          -0.1507 GY   1.45
     21                          -0.1672 GY   0.20
     21                          -0.5016 GY   0.46
     21                          -0.8360 GY   0.74
     21                          -1.1704 GY   1.03
     21                          -1.5048 GY   1.32
     21                          -1.8392 GY   1.62
     21                          -2.1736 GY   1.91
     21                          -2.5080 GY   2.20
     21                          -2.3348 GY   2.48
     21                          -2.0235 GY   2.75
     21                          -1.7122 GY   3.02
     21                          -1.4009 GY   3.29
     21                          -1.0896 GY   3.57
     21                          -0.7783 GY   3.84
     21                          -0.4670 GY   4.10
     21                          -0.1557 GY   4.35
      8                          -1.6098 GY   0.03
      8                          -1.3951 GY   0.09
      8                          -1.1805 GY   0.16
      8                          -0.9659 GY   0.22
```

```
      8                            -0.7512 GY   0.28
      8                            -0.5366 GY   0.34
      8                            -0.3220 GY   0.40
      8                            -0.1073 GY   0.46
      7                            -0.0997 GY   0.03
      7                            -0.2991 GY   0.07
      7                            -0.4984 GY   0.12
      7                            -0.6978 GY   0.16
      7                            -0.8972 GY   0.21
      7                            -1.0966 GY   0.25
      7                            -1.2959 GY   0.30
      7                            -1.4953 GY   0.34
      7                            -0.5724 GY   0.37
      7                            -0.5571 GY   0.39
      7                            -0.5418 GY   0.41
      7                            -0.5266 GY   0.42
      7                            -0.5113 GY   0.44
      7                            -0.4961 GY   0.46
      7                            -0.4808 GY   0.47
      7                            -0.4655 GY   0.49
      9                            -0.0283 GY   0.04
      9                            -0.0850 GY   0.10
      9                            -0.1416 GY   0.16
      9                            -0.1982 GY   0.22
      9                            -0.2549 GY   0.28
      9                            -0.3115 GY   0.34
      9                            -0.3682 GY   0.41
      9                            -0.4248 GY   0.47
      9                            -0.4248 GY   0.53
      9                            -0.3682 GY   0.59
      9                            -0.3115 GY   0.66
      9                            -0.2549 GY   0.72
      9                            -0.1982 GY   0.78
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   11
      9                            -0.1416 GY   0.84
      9                            -0.0850 GY   0.90
      9                            -0.0283 GY   0.96
     27                            -0.0283 GY   0.04
     27                            -0.0850 GY   0.10
     27                            -0.1416 GY   0.16
     27                            -0.1982 GY   0.22
     27                            -0.2549 GY   0.28
     27                            -0.3115 GY   0.34
     27                            -0.3682 GY   0.41
     27                            -0.4248 GY   0.47
     27    -7.2500 GY   0.50    4.00
     27                            -0.4248 GY   4.03
     27                            -0.3682 GY   4.09
     27                            -0.3115 GY   4.16
     27                            -0.2549 GY   4.22
     27                            -0.1982 GY   4.28
     27                            -0.1416 GY   4.34
     27                            -0.0850 GY   4.40
     27                            -0.0283 GY   4.46
     13                            -0.0283 GY   0.04
     13                            -0.0850 GY   0.10
     13                            -0.1416 GY   0.16
     13                            -0.1982 GY   0.22
```

```
     13                            -0.2549 GY   0.28
     13                            -0.3115 GY   0.34
     13                            -0.3682 GY   0.41
     13                            -0.4248 GY   0.47
     13                            -0.4248 GY   0.53
     13                            -0.3682 GY   0.59
     13                            -0.3115 GY   0.66
     13                            -0.2549 GY   0.72
     13                            -0.1982 GY   0.78
     13                            -0.1416 GY   0.84
     13                            -0.0850 GY   0.90
     13                            -0.0283 GY   0.96
     23                            -0.0283 GY   0.04
     23                            -0.0850 GY   0.10
     23                            -0.1416 GY   0.16
     23                            -0.1982 GY   0.22
     23                            -0.2549 GY   0.28
     23                            -0.3115 GY   0.34
     23                            -0.3682 GY   0.41
     23                            -0.4248 GY   0.47
     23    -7.2500 GY   0.50    4.00
     23                            -0.4248 GY   4.03
     23                            -0.3682 GY   4.09
     23                            -0.3115 GY   4.16
     23                            -0.2549 GY   4.22
     23                            -0.1982 GY   4.28
     23                            -0.1416 GY   4.34
     23                            -0.0850 GY   4.40
     23                            -0.0283 GY   4.46
     26                            -0.0283 GY   0.04
     26                            -0.0850 GY   0.10
     26                            -0.1416 GY   0.16
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   12
     26                            -0.1982 GY   0.22
     26                            -0.2549 GY   0.28
     26                            -0.3115 GY   0.34
     26                            -0.3682 GY   0.41
     26                            -0.4248 GY   0.47
     26                            -0.4248 GY   0.53
     26                            -0.3682 GY   0.59
     26                            -0.3115 GY   0.66
     26                            -0.2549 GY   0.72
     26                            -0.1982 GY   0.78
     26                            -0.1416 GY   0.84
     26                            -0.0850 GY   0.90
     26                            -0.0283 GY   0.96
     14                            -0.0283 GY   0.04
     14                            -0.0850 GY   0.10
     14                            -0.1416 GY   0.16
     14                            -0.1982 GY   0.22
     14                            -0.2549 GY   0.28
     14                            -0.3115 GY   0.34
     14                            -0.3682 GY   0.41
     14                            -0.4248 GY   0.47
     14    -7.2500 GY   0.50    5.50
     14                            -0.4248 GY   5.53
     14                            -0.3682 GY   5.59
     14                            -0.3115 GY   5.66
```

```
     14                            -0.2549 GY   5.72
     14                            -0.1982 GY   5.78
     14                            -0.1416 GY   5.84
     14                            -0.0850 GY   5.90
     14                            -0.0283 GY   5.96
     17                            -0.0283 GY   0.04
     17                            -0.0850 GY   0.10
     17                            -0.1416 GY   0.16
     17                            -0.1982 GY   0.22
     17                            -0.2549 GY   0.28
     17                            -0.3115 GY   0.34
     17                            -0.3682 GY   0.41
     17                            -0.4248 GY   0.47
     17                            -0.4248 GY   0.53
     17                            -0.3682 GY   0.59
     17                            -0.3115 GY   0.66
     17                            -0.2549 GY   0.72
     17                            -0.1982 GY   0.78
     17                            -0.1416 GY   0.84
     17                            -0.0850 GY   0.90
     17                            -0.0283 GY   0.96
     13    -7.2500 GY   0.00    0.50
     13                            -0.4248 GY   0.53
     13                            -0.3682 GY   0.59
     13                            -0.3115 GY   0.66
     13                            -0.2549 GY   0.72
     13                            -0.1982 GY   0.78
     13                            -0.1416 GY   0.84
     13                            -0.0850 GY   0.90
     13                            -0.0283 GY   0.96
     10    -7.2500 GY   0.50    1.50
      A FLOOR FRAME DESIGN WITH AREA LOAD                        -- PAGE NO.   13
     10                            -0.0283 GY   0.04
     10                            -0.0850 GY   0.10
     10                            -0.1416 GY   0.16
     10                            -0.1982 GY   0.22
     10                            -0.2549 GY   0.28
     10                            -0.3115 GY   0.34
     10                            -0.3682 GY   0.41
     10                            -0.4248 GY   0.47
     11    -7.2500 GY   0.00    2.00
     12    -7.2500 GY   0.00    1.50
     14                            -0.1133 GY   0.08
     14                            -0.3398 GY   0.19
     14                            -0.5664 GY   0.32
     14                            -0.7930 GY   0.44
     14                            -1.0195 GY   0.56
     14                            -1.2461 GY   0.69
     14                            -1.4727 GY   0.81
     14                            -1.6992 GY   0.94
     14   -14.5000 GY   1.00    5.00
     14                            -1.6992 GY   5.06
     14                            -1.4727 GY   5.19
     14                            -1.2461 GY   5.31
     14                            -1.0195 GY   5.44
     14                            -0.7930 GY   5.56
     14                            -0.5664 GY   5.68
     14                            -0.3398 GY   5.81
```

```
     14                            -0.1133 GY   5.92
     25                            -0.1133 GY   0.08
     25                            -0.3398 GY   0.19
     25                            -0.5664 GY   0.32
     25                            -0.7930 GY   0.44
     25                            -1.0195 GY   0.56
     25                            -1.2461 GY   0.69
     25                            -1.4727 GY   0.81
     25                            -1.6992 GY   0.94
     25                            -1.6992 GY   1.06
     25                            -1.4727 GY   1.19
     25                            -1.2461 GY   1.31
     25                            -1.0195 GY   1.44
     25                            -0.7930 GY   1.56
     25                            -0.5664 GY   1.68
     25                            -0.3398 GY   1.81
     25                            -0.1133 GY   1.92
     15                            -0.1133 GY   0.08
     15                            -0.3398 GY   0.19
     15                            -0.5664 GY   0.32
     15                            -0.7930 GY   0.44
     15                            -1.0195 GY   0.56
     15                            -1.2461 GY   0.69
     15                            -1.4727 GY   0.81
     15                            -1.6992 GY   0.94
     15   -14.5000 GY   1.00    5.00
     15                            -1.6992 GY   5.06
     15                            -1.4727 GY   5.19
     15                            -1.2461 GY   5.31
     15                            -1.0195 GY   5.44
      A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   14
     15                            -0.7930 GY   5.56
     15                            -0.5664 GY   5.68
     15                            -0.3398 GY   5.81
     15                            -0.1133 GY   5.92
     16                            -0.1133 GY   0.08
     16                            -0.3398 GY   0.19
     16                            -0.5664 GY   0.32
     16                            -0.7930 GY   0.44
     16                            -1.0195 GY   0.56
     16                            -1.2461 GY   0.69
     16                            -1.4727 GY   0.81
     16                            -1.6992 GY   0.94
     16                            -1.6992 GY   1.06
     16                            -1.4727 GY   1.19
     16                            -1.2461 GY   1.31
     16                            -1.0195 GY   1.44
     16                            -0.7930 GY   1.56
     16                            -0.5664 GY   1.68
     16                            -0.3398 GY   1.81
     16                            -0.1133 GY   1.92
  ************ END OF DATA FROM INTERNAL STORAGE ************
    43. PARAMETERS
    44. CODE EN 1993-1-1:2005
    45. NA 1
    46. BEAM 3 ALL
    47. DMAX 0.6 ALL
    48. DMIN 0.3 ALL
```

```
   49. UNL 0.3 ALL
   50. SELECT MEMB 2 6 11 14 15 16 18 19 21 23 24 27
STEEL DESIGN
                        STAAD.PRO MEMBER SELECTION - BS EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.   15
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
      2 ST  UB406X140X46 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.875        1
                       0.00           0.00       -182.50        0.00
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
      6 ST  UB406X140X39 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.782        1
                       0.00           0.00        133.08        1.00
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
     11 ST  UB457X152X52 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.878        1
                       0.00           0.00       -226.94        1.17
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
     14 ST  UB305X102X33 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.842        1
                       0.00           0.00        -95.16        3.00
     A FLOOR FRAME DESIGN WITH AREA LOAD                     -- PAGE NO.   16
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
     15 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.782        1
                       0.00           0.00        -62.83        3.00
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
     16 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.541        1
                       0.00           0.00        -43.50        0.00
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
     18 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.939        1
                       0.00           0.00        -75.45        3.75
```

```
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     19 ST  UB457X152X60 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.956        1
                       0.00           0.00        -289.84       0.00
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   17
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     21 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.540        1
                       0.00           0.00         -43.43       2.26
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     23 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.418        1
                       0.00           0.00         -33.61       2.25
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     24 ST  UB305X102X25 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.279        1
                       0.00           0.00         -22.43       1.50
ALL UNITS ARE - KN   METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
     27 ST  UB457X152X52 (BRITISH SECTIONS)
                          PASS     EC-6.2.5          0.861        1
                       0.00           0.00        -222.56       0.00
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   18
  ************** END OF TABULATED RESULT OF DESIGN **************
   51. FINISH
  ****************************************************************************
  **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
              IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
              ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
              TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
              FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
              FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
              COMMANDS.
  ****************************************************************************
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:34 ****
     A FLOOR FRAME DESIGN WITH AREA LOAD                      -- PAGE NO.   19
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
```

```
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-3 Soil Springs for Portal Frame

A portal frame type steel structure is sitting on concrete footings. The soil is to be considered as an elastic foundation. The value of soil subgrade reaction is known from which spring constants are calculated by multiplying the subgrade reaction by the tributary area of each modeled spring.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-3 Soil Springs for Portal Frame.STD` when you install the program.

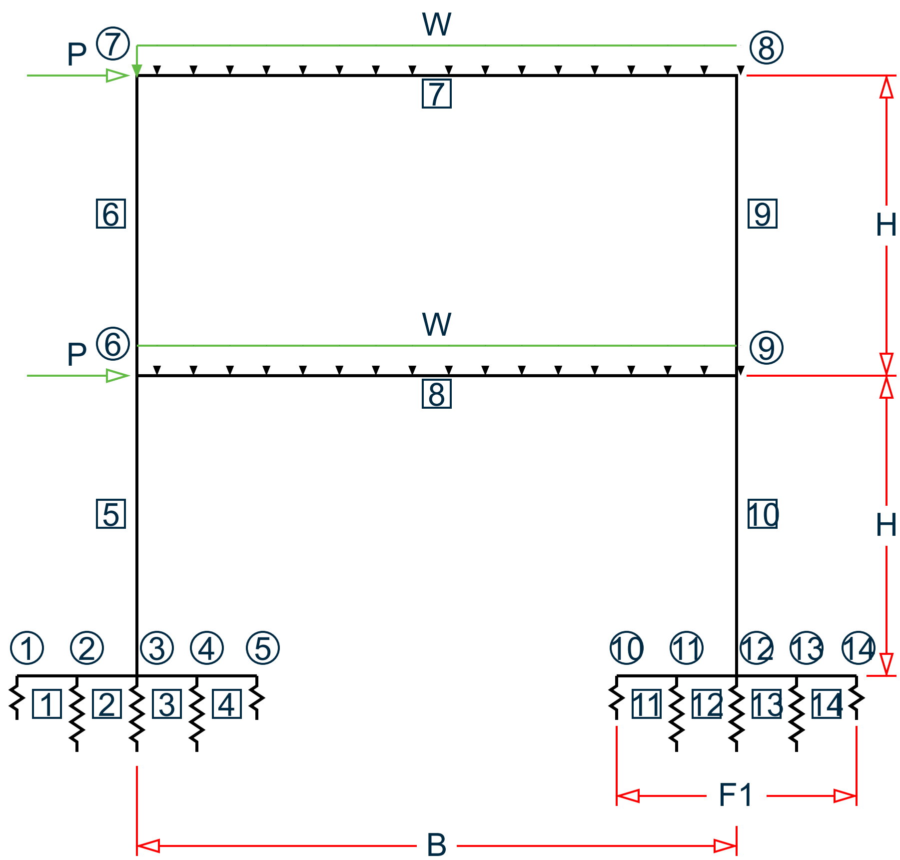


Figure 543: Example Problem No. 3

Where:

B = 6 m, H = 3 m, F1 = 1.2 m, F2 = 2.4 m
P = 20 KN, w = 4.5 KN/m
Soil Subgrade Reaction = 41,666.67 KN/m^3

Table 912: Spring constant calculation

Springs of Joints	Spring Constant
1, 5, 10, & 14 (Edges)	2.4 x 0.3 x 41,666.67 = 30,000 KN/m
2, 3, 4, 11, 12, & 13 (Interior)	2.4 x 0.6 x 41,666.67 = 60,000 KN/m

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE PORTAL ON FOOTING FOUNDATION
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0.0 0.0 0.0 5 2.4 0.0 0.0
6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
10 6.0 0.0 0.0 14 8.4 0.0 0.0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ; 10 9 12
11 10 11 14
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTIES BRITISH
1 4 11 14 PRIS YD 0.30 ZD 2.40
2 3 12 13 PRIS YD 0.60 ZD  2.40
5 6 9 10 TABLE ST JO254X203
7 8 TA ST UB305X165X40
```

The first two lines define member properties as prismatic (PRIS) followed by depth (YD) and width (ZD) values. The program will calculate the properties necessary to perform the analysis. See G.6.1 Prismatic Properties (on page 2114) for additional information. Member properties for the remaining members are chosen from the British steel table. The term ST stands for standard single section.

```
UNIT MMS
* E FOR STEEL IS 210 (KN/sq.mm.) AND FOR CONCRETE 21 (KN/sq.mm.)
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
```

```
POISSON 0.3
DENSITY 76.977e-009
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21
POISSON 0.17
DENSITY 23.534e-009
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
MATERIAL STEEL MEMB 5 TO 10
```

The `CONSTANT` command initiates input for material constants like modulus of elasticity, Density, and Poisson's ratio. The length unit is changed from `METER` to `MM` to facilitate the input in familiar units.

Note: Any line beginning with an asterisk (* character) is treated as a comment line.

```
SUPPORTS
2 TO 4 11 TO 13 FIXED BUT MZ KFY 60
1 5 10 14 FIXED BUT MZ KFY 30
```

The supports for the structure are specified above. The first set of joints are supports restrained in all directions except global moment-z (`MZ`). Also, a spring having a spring constant of 60 KN/mm is provided in the global Y direction at these nodes. The second set is similar to the former except for a different value of the spring constant.

```
UNIT METER
```

```
LOADING 1 DEAD AND WIND LOAD COMBINED
```

Load case 1 is initiated followed by a title.

```
SELF Y -1.0
```

The selfweight of the structure is specified as acting in the global Y direction with a -1.0 factor.

Since global Y is vertically upwards, the -1.0 factor indicates that this load will act downwards.

```
JOINT LOAD
6 7 FX 20.0
```

Load 1 contains joint loads also. `FX` indicates that the load is a force in the global X direction.

```
MEMBER LOAD
7 8 UNI GY -45.0
```

Load 1 contains member loads also. `GY` indicates that the load acts in the global Y direction. The term `UNI` stands for uniformly distributed load, and is applied on members 7 and 8, acting downwards.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT ANALYSIS RESULTS
```

The above PRINT command instructs the program to print analysis results which include joint displacements, member forces and support reactions.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE PORTAL ON FOOTING FOUNDATION
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 5 2.4 0.0 0.0
6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
10 6.0 0.0 0.0 14 8.4 0.0 0.0
MEMBER INCIDENCES
1 1 2 4
5 3 6 ; 6 6 7
7 7 8 ; 8 6 9
9 8 9 ; 10 9 12
11 10 11 14
MEMBER PROPERTIES BRITISH
1 4 11 14 PRIS YD 0.30 ZD 2.40
2 3 12 13 PRIS YD 0.60 ZD 2.40
5 6 9 10 TABLE ST JO254X203
7 8 TA ST UB305X165X40
UNIT MMS
* E FOR STEEL IS 210 (KN/sq.mm.) AND FOR CONCRETE 21 (KN/sq.mm.)
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 76.977e-009
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21
POISSON 0.17
DENSITY 23.534e-009
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
MATERIAL STEEL MEMB 5 TO 10
SUPPORTS
2 TO 4 11 TO 13 FIXED BUT MZ KFY 60
1 5 10 14 FIXED BUT MZ KFY 30
UNIT METER
LOADING 1 DEAD AND WIND LOAD COMBINED
SELF Y -1.0
JOINT LOAD
6 7 FX 20.0
```

```
MEMBER LOAD
7 8 UNI GY -45.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45: 4                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE PORTAL ON FOOTING FOUNDATION
INPUT FILE: UK-3 Soil Springs for Portal Frame.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 5 2.4 0.0 0.0
     5. 6 1.2 3.0 0.0 ; 7 1.2 6.0 0.0
     6. 8 7.2 6.0 0.0 ; 9 7.2 3.0 0.0
     7. 10 6.0 0.0 0.0 14 8.4 0.0 0.0
     8. MEMBER INCIDENCES
     9. 1 1 2 4
    10. 5 3 6 ; 6 6 7
    11. 7 7 8 ; 8 6 9
    12. 9 8 9 ; 10 9 12
    13. 11 10 11 14
    14. MEMBER PROPERTIES BRITISH
    15. 1 4 11 14 PRIS YD 0.30 ZD 2.40
    16. 2 3 12 13 PRIS YD 0.60 ZD 2.40
    17. 5 6 9 10 TABLE ST JO254X203
    18. 7 8 TA ST UB305X165X40
    19. UNIT MMS
    20. * E FOR STEEL IS 210 (KN/SQ.MM.) AND FOR CONCRETE 21 (KN/SQ.MM.)
    21. DEFINE MATERIAL START
    22. ISOTROPIC STEEL
    23. E 210
    24. POISSON 0.3
    25. DENSITY 76.977E-009
    26. ALPHA 6E-006
    27. DAMP 0.03
    28. TYPE STEEL
    29. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
    30. ISOTROPIC CONCRETE
    31. E 21
    32. POISSON 0.17
    33. DENSITY 23.534E-009
    34. ALPHA 5E-006
    35. DAMP 0.05
    36. G 9.25
    37. TYPE CONCRETE
```

```
   38. STRENGTH FCU 0.0275
     PORTAL ON FOOTING FOUNDATION                            -- PAGE NO.    2
   39. END DEFINE MATERIAL
   40. CONSTANTS
   41. MATERIAL CONCRETE MEMB 1 TO 4 11 TO 14
   42. MATERIAL STEEL MEMB 5 TO 10
   43. SUPPORTS
   44. 2 TO 4 11 TO 13 FIXED BUT MZ KFY 60
   45. 1 5 10 14 FIXED BUT MZ KFY 30
   46. UNIT METER
   47. LOADING 1 DEAD AND WIND LOAD COMBINED
   48. SELF Y -1.0
   49. JOINT LOAD
   50. 6 7 FX 20.0
   51. MEMBER LOAD
   52. 7 8 UNI GY -45.0
   53. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         14  NUMBER OF MEMBERS      14
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     10
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      32
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   54. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     PORTAL ON FOOTING FOUNDATION                            -- PAGE NO.    3
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1     0.0000   -0.1075    0.0000    0.0000    0.0000   -0.0003
     2    1     0.0000   -0.1232    0.0000    0.0000    0.0000   -0.0002
     3    1     0.0000   -0.1367    0.0000    0.0000    0.0000   -0.0002
     4    1     0.0000   -0.1466    0.0000    0.0000    0.0000   -0.0002
     5    1     0.0000   -0.1531    0.0000    0.0000    0.0000   -0.0001
     6    1     0.5036   -0.1720    0.0000    0.0000    0.0000   -0.0032
     7    1     1.0714   -0.1898    0.0000    0.0000    0.0000   -0.0045
     8    1     1.0286   -0.2097    0.0000    0.0000    0.0000    0.0022
     9    1     0.5240   -0.1901    0.0000    0.0000    0.0000   -0.0003
    10    1     0.0000   -0.0897    0.0000    0.0000    0.0000   -0.0005
    11    1     0.0000   -0.1210    0.0000    0.0000    0.0000   -0.0005
    12    1     0.0000   -0.1504    0.0000    0.0000    0.0000   -0.0005
    13    1     0.0000   -0.1759    0.0000    0.0000    0.0000   -0.0004
    14    1     0.0000   -0.1969    0.0000    0.0000    0.0000   -0.0003
     PORTAL ON FOOTING FOUNDATION                            -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     2    1      0.00     73.95      0.00      0.00      0.00      0.00
     3    1      0.09     82.00      0.00      0.00      0.00      0.00
     4    1      0.00     87.96      0.00      0.00      0.00      0.00
    11    1      0.00     72.58      0.00      0.00      0.00      0.00
    12    1    -40.09     90.24      0.00      0.00      0.00      0.00
    13    1      0.00    105.56      0.00      0.00      0.00      0.00
     1    1      0.00     32.25      0.00      0.00      0.00      0.00
     5    1      0.00     45.92      0.00      0.00      0.00      0.00
```

```
     10    1      0.00     26.90      0.00      0.00      0.00      0.00
     14    1      0.00     59.07      0.00      0.00      0.00      0.00
      PORTAL ON FOOTING FOUNDATION                             -- PAGE NO.    5
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00     32.25     0.00      0.00      0.00       0.00
                 2      0.00    -22.08     0.00      0.00      0.00      16.30
      2    1     2      0.00     96.03     0.00      0.00      0.00     -16.30
                 3      0.00    -75.70     0.00      0.00      0.00      67.82
      3    1     3      0.00   -103.39     0.00      0.00      0.00     -92.63
                 4      0.00    123.72     0.00      0.00      0.00      24.50
      4    1     4      0.00    -35.76     0.00      0.00      0.00     -24.50
                 5      0.00     45.92     0.00      0.00      0.00       0.00
      5    1     3    261.09     -0.09     0.00      0.00      0.00      24.82
                 6   -258.66      0.09     0.00      0.00      0.00     -25.09
      6    1     6    132.10    -56.74     0.00      0.00      0.00     -73.59
                 7   -129.67     56.74     0.00      0.00      0.00     -96.64
      7    1     7     76.74    129.67     0.00      0.00      0.00      96.64
                 8    -76.74    142.70     0.00      0.00      0.00    -135.72
      8    1     6    -36.65    126.57     0.00      0.00      0.00      98.68
                 9     36.65    145.80     0.00      0.00      0.00    -156.40
      9    1     8    142.70     76.74     0.00      0.00      0.00     135.72
                 9   -145.12    -76.74     0.00      0.00      0.00      94.51
     10    1     9    290.93     40.09     0.00      0.00      0.00      61.89
                12   -293.35    -40.09     0.00      0.00      0.00      58.38
     11    1    10      0.00     26.90     0.00      0.00      0.00      -0.00
                11      0.00    -16.74     0.00      0.00      0.00      13.09
     12    1    11      0.00     89.32     0.00      0.00      0.00     -13.09
                12      0.00    -68.99     0.00      0.00      0.00      60.58
     13    1    12      0.00   -134.13     0.00      0.00      0.00    -118.97
                13      0.00    154.46     0.00      0.00      0.00      32.39
     14    1    13      0.00    -48.90     0.00      0.00      0.00     -32.39
                14      0.00     59.07     0.00      0.00      0.00       0.00
      PORTAL ON FOOTING FOUNDATION                             -- PAGE NO.    6
   ************** END OF LATEST ANALYSIS RESULT **************
    55. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:45: 5 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *M. To assign a spring support* (on page 821)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)

EX. UK-4 Inactive Members in a Braced Frame

This example is a typical case of a load-dependent structure where the structural condition changes for different load cases. In this example, different bracing members are made inactive for different load cases. This is done to prevent these members from carrying any compressive forces.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-4 Inactive Members in a Braced Frame.STD` when you install the program.

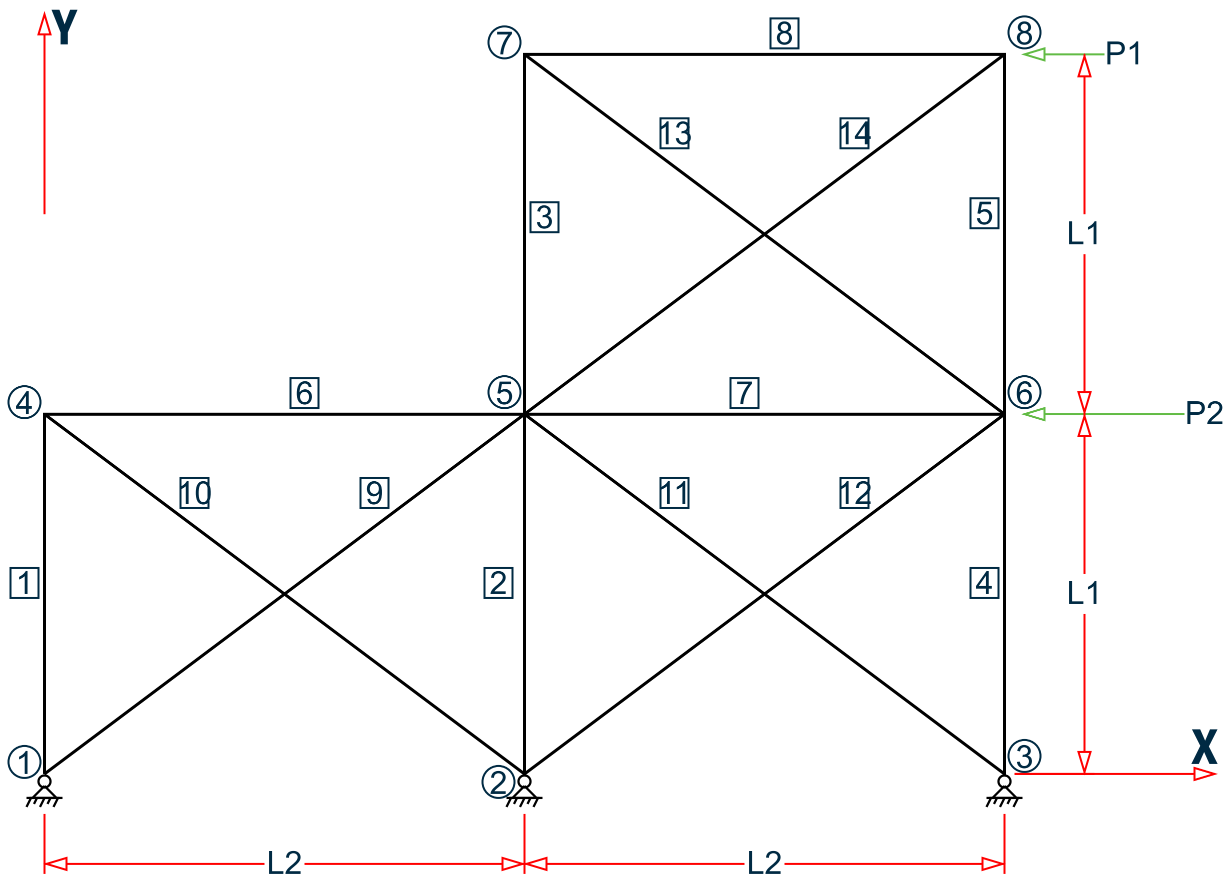


Figure 544: Example Problem No. 4

Where:

L1 = 4.5 m, L2 = 6.0m
P1 = 45 kN, P2 = 90 kN

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE
* A PLANE FRAME STRUCTURE WITH TENSION BRACING
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
SET NL 3
```

This structure has to be analyzed for three primary load cases. Consequently, the modeling of our problem requires us to define three sets of data, with each set containing a load case and an associated analysis command. Also, the members which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have two commands, one called `SET NL` and the other called `CHANGE`. The `SET NL` command is used above to indicate the total number of primary load cases that the file contains. The `CHANGE` command will come in later (after the `PERFORM ANALYSIS` command).

```
JOINT COORDINATES
1 0 0 0 3 12. 0. 0.
4 0 4.5 0 6 12. 4.5 0.
7 6. 9. 0. ; 8 12. 9. 0.
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are given as all zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ;  9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ; 14 5 8
```

Defines the members by the joints to which they are connected.

```
MEMBER TRUSS
9 TO 14
```

The preceding command defines that members 9 through 14 are of type truss. This means these members can only carry axial tension/compression and no moments.

```
MEMBER PROP BRITISH
1 TO 5 TABLE ST UB305X165X40
6 7 8 TA ST UB457X152X52
9 TO 14 TA LD UA150X150X10
```

Properties for all members are assigned from the British steel table. The word `ST` stands for standard single section. The word `LD` stands for long leg back-to-back double angle. Since the spacing between the two angles of the double angle is not provided, it is assumed to be 0.0.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
```

```
CONSTANTS
MATERIAL STEEL ALL
```

The DEFINE MATERIAL command is used to define a material. The CONSTANT command is used to assign this material all members. Length unit is changed from METER to MMS.

```
SUPPORT
1 2 3 PINNED
```

Pinned supports are specified at Joints 1, 2 and 3. The word PINNED signifies that no moments will be carried by these supports.

```
INACTIVE MEMBERS 9 TO 14
```

The preceding command makes the listed members inactive. The stiffness contribution of these members will not be considered in the analysis till they are made active again.

```
UNIT METER
LOADING 1 DEAD AND LIVE LOAD
```

Load case 1 is initiated followed by a title. The length UNIT is changed from MMS to METER for input values which follow.

```
MEMBER LOAD
6 8 UNI GY -4.5
7 UNI GY -6.75
```

Load 1 contains member loads. GY indicates that the load acts in the global Y direction. The word UNI stands for uniformly distributed load. The load is applied on members 6, 7, and 8.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. It is worth noting that members 9 to 14 will not be used in this analysis since they were declared inactive earlier. In other words, for dead and live load, the bracing members are not used to carry any load.

```
CHANGES
```

The members inactivated earlier are restored using the CHANGE command.

```
INACTIVE MEMBERS 10 11 13
```

A new set of members are made inactive. The stiffness contribution from these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 2 WIND FROM LEFT
```

Load case 2 is initiated followed by a title.

```
JOINT LOAD
4 FX 135 ; 7 FX 65
```

Load 2 contains joint loads. FX indicates that the load is a force in the global X direction. Nodes 4 and 7 are subjected to the loads.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis will be performed for load case 2 only.

```
CHANGE
```

The above CHANGE command is an instruction to re-activate all inactive members.

```
INACTIVE MEMBERS 9 12 14
```

Members 9, 12 and 14 are made inactive. The stiffness contribution of these members will not be used in the analysis till they are made active again. They have been inactivated to prevent them from being subject to compressive forces for the next load case.

```
LOADING 3 WIND FROM RIGHT
```

Load case 3 is initiated followed by a title.

```
JOINT LOAD
6 FX -135 ; 8 FX -65
```

Load 3 contains joint loads at nodes 6 and 8. FX indicates that the load is a force in the global X direction. The negative numbers

(-135 and -65) indicate that the load is acting along the negative global X direction.

```
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
```

Load combination case 4 involves the algebraic summation of the results of load cases 1 and 2 after multiplying each by a factor of 0.75. For load combinations, the program simply gathers the results of the component primary cases, factors them appropriately, and combines them algebraically. Thus, an analysis in the real sense of the term (multiplying the inverted stiffness matrix by the load vector) is not carried out for load combination cases. Load combination case 5 combines the results of load cases 1 and 3.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis. Only primary load case 3 will be considered for this analysis. (As explained earlier, a combination case is not truly analyzed for, but handled using other means.)

```
CHANGE
```

The above `CHANGE` command will re-activate all inactive members.

```
LOAD LIST ALL
```

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the "active" load cases. The `LOAD LIST ALL` command enables all the load cases in the structure to be made active for further processing.

```
PRINT MEMBER FORCES
```

The preceding `PRINT` command is an instruction to produce a report, in the output file, of the member end forces.

```
LOAD LIST 1 4 5
```

A `LOAD LIST` command is a means of instructing the program to use only the listed load cases for further processing.

```
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 1 ALL
UNL 1.8 ALL
KY 0.5 ALL
```

The `PARAMETER` command is used to specify the steel design parameters (information on these parameters can be obtained from the manual where the implementation of the code is explained). The `BEAM` parameter is specified to perform design at every 1/12th point along the member length. `UNL` represents the unsupported length to be used for calculation of allowable bending stress. `KY 0.5 ALL` sets the effective length factor for column buckling about the local Y-axis to be 0.5 for ALL members.

```
    CHECK CODE ALL
```

The above command instructs the program to perform a check to determine how you defined member sizes along with the latest analysis results meet the code requirements.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE A PLANE FRAME STRUCTURE WITH
* TENSION BRACING
UNIT METER KNS
SET NL 3
JOINT COORDINATES
1 0 0 0 3 12. 0. 0.
4 0 4.5 0 6 12. 4.5 0.
7 6. 9. 0. ; 8 12. 9. 0.
MEMBER INCIDENCE
1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
13 6 7 ; 14 5 8
MEMBER TRUSS
9 TO 14
MEMBER PROP BRITISH
1 TO 5 TABLE ST UB305X165X40
6 7 8 TA ST UB457X152X52
9 TO 14 TA LD UA150X150X10
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 2 3 PINNED
INACTIVE MEMBERS 9 TO 14
UNIT METER
LOADING 1 DEAD AND LIVE LOAD
MEMBER LOAD
6 8 UNI GY -4.5
7 UNI GY -6.75
PERFORM ANALYSIS
CHANGES
INACTIVE MEMBERS 10 11 13
```

```
LOADING 2 WIND FROM LEFT
JOINT LOAD
4 FX 135 ; 7 FX 65
PERFORM ANALYSIS
CHANGE
INACTIVE MEMBERS 9 12 14
LOADING 3 WIND FROM RIGHT
JOINT LOAD
6 FX -135 ; 8 FX -65
LOAD COMBINATION 4
1 0.75 2 0.75
LOAD COMBINATION 5
1 0.75 3 0.75
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 4 5
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 1 ALL
UNL 1.8 ALL
KY 0.5 ALL
CHECK CODE ALL
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45: 7                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE A PLANE FRAME STRUCTURE WITH
INPUT FILE: UK-4 Inactive Members in a Braced Frame.STD
     2. * TENSION BRACING
     3. UNIT METER KNS
     4. SET NL 3
     5. JOINT COORDINATES
     6. 1 0 0 0 3 12. 0. 0.
     7. 4 0 4.5 0 6 12. 4.5 0.
     8. 7 6. 9. 0. ; 8 12. 9. 0.
     9. MEMBER INCIDENCE
    10. 1 1 4 2 ; 3 5 7 ; 4 3 6 ; 5 6 8 ; 6 4 5 7
    11. 8 7 8 ; 9 1 5 ; 10 2 4 ; 11 3 5 ; 12 2 6
    12. 13 6 7 ; 14 5 8
    13. MEMBER TRUSS
    14. 9 TO 14
    15. MEMBER PROP BRITISH
    16. 1 TO 5 TABLE ST UB305X165X40
```

```
   17. 6 7 8 TA ST UB457X152X52
   18. 9 TO 14 TA LD UA150X150X10
   19. UNIT MMS
   20. DEFINE MATERIAL START
   21. ISOTROPIC STEEL
   22. E 210
   23. POISSON 0.3
   24. DENSITY 7.68191E-008
   25. ALPHA 6E-006
   26. DAMP 0.03
   27. TYPE STEEL
   28. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
   29. END DEFINE MATERIAL
   30. CONSTANTS
   31. MATERIAL STEEL ALL
   32. SUPPORT
   33. 1 2 3 PINNED
   34. INACTIVE MEMBERS 9 TO 14
   35. UNIT METER
   36. LOADING 1 DEAD AND LIVE LOAD
   37. MEMBER LOAD
   38. 6 8 UNI GY -4.5
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.    2
   * TENSION BRACING
   39. 7 UNI GY -6.75
   40. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          8  NUMBER OF MEMBERS      14
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   41. CHANGES
   42. INACTIVE MEMBERS 10 11 13
   43. LOADING 2 WIND FROM LEFT
   44. JOINT LOAD
   45. 4 FX 135 ; 7 FX 65
   46. PERFORM ANALYSIS
   47. CHANGE
   48. INACTIVE MEMBERS 9 12 14
   49. LOADING 3 WIND FROM RIGHT
   50. JOINT LOAD
   51. 6 FX -135 ; 8 FX -65
   52. LOAD COMBINATION 4
   53. 1 0.75 2 0.75
   54. LOAD COMBINATION 5
   55. 1 0.75 3 0.75
   56. PERFORM ANALYSIS
   57. CHANGE
   58. LOAD LIST ALL
   59. PRINT MEMBER FORCES
 MEMBER   FORCES
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.    3
   * TENSION BRACING
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
```

```
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1     11.15     -0.90     0.00      0.00      0.00       0.00
               4    -11.15      0.90     0.00      0.00      0.00      -4.06
         2     1     -1.07      0.78     0.00      0.00      0.00      -0.00
               4      1.07     -0.78     0.00      0.00      0.00       3.49
         3     1     68.16     -0.68     0.00      0.00      0.00       0.00
               4    -68.16      0.68     0.00      0.00      0.00      -3.05
         4     1      7.56     -0.09     0.00      0.00      0.00      -0.00
               4     -7.56      0.09     0.00      0.00      0.00      -0.43
         5     1     59.49     -1.18     0.00      0.00      0.00       0.00
               4    -59.49      1.18     0.00      0.00      0.00      -5.33
    2    1     2     51.94     -0.07     0.00      0.00      0.00       0.00
               5    -51.94      0.07     0.00      0.00      0.00      -0.32
         2     2     46.05      0.54     0.00      0.00      0.00      -0.00
               5    -46.05     -0.54     0.00      0.00      0.00       2.41
         3     2    129.15     -0.47     0.00      0.00      0.00      -0.00
               5   -129.15      0.47     0.00      0.00      0.00      -2.09
         4     2     73.50      0.35     0.00      0.00      0.00      -0.00
               5    -73.50     -0.35     0.00      0.00      0.00       1.57
         5     2    135.82     -0.40     0.00      0.00      0.00      -0.00
               5   -135.82      0.40     0.00      0.00      0.00      -1.81
    3    1     5     13.68     -2.95     0.00      0.00      0.00      -5.29
               7    -13.68      2.95     0.00      0.00      0.00      -8.01
         2     5     -0.99      1.45     0.00      0.00      0.00       3.48
               7      0.99     -1.45     0.00      0.00      0.00       3.03
         3     5     47.36     -2.44     0.00      0.00      0.00      -6.17
               7    -47.36      2.44     0.00      0.00      0.00      -4.82
         4     5      9.52     -1.13     0.00      0.00      0.00      -1.35
               7     -9.52      1.13     0.00      0.00      0.00      -3.73
         5     5     45.79     -4.05     0.00      0.00      0.00      -8.59
               7    -45.79      4.05     0.00      0.00      0.00      -9.62
    4    1     3     31.40      0.97     0.00      0.00      0.00       0.00
               6    -31.40     -0.97     0.00      0.00      0.00       4.38
         2     3    103.89      0.21     0.00      0.00      0.00      -0.00
               6   -103.89     -0.21     0.00      0.00      0.00       0.93
         3     3    -48.62     -0.59     0.00      0.00      0.00       0.00
               6     48.62      0.59     0.00      0.00      0.00      -2.65
         4     3    101.47      0.88     0.00      0.00      0.00      -0.00
               6   -101.47     -0.88     0.00      0.00      0.00       3.98
         5     3    -12.91      0.29     0.00      0.00      0.00       0.00
               6     12.91     -0.29     0.00      0.00      0.00       1.29
   A PLANE FRAME STRUCTURE WITH                               -- PAGE NO.    4
  * TENSION BRACING
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    5    1     6     13.32      2.95     0.00      0.00      0.00       6.39
               8    -13.32     -2.95     0.00      0.00      0.00       6.90
         2     6     47.69      1.28     0.00      0.00      0.00       2.86
               8    -47.69     -1.28     0.00      0.00      0.00       2.92
         3     6     -1.29     -1.13     0.00      0.00      0.00      -2.15
               8      1.29      1.13     0.00      0.00      0.00      -2.92
         4     6     45.76      3.18     0.00      0.00      0.00       6.94
               8    -45.76     -3.18     0.00      0.00      0.00       7.37
         5     6      9.02      1.37     0.00      0.00      0.00       3.18
```

```
                 8     -9.02      -1.37      0.00       0.00       0.00        2.99
     6     1     4      0.90      11.15      0.00       0.00       0.00        4.06
                 5     -0.90      15.85      0.00       0.00       0.00      -18.14
           2     4    134.22      -1.07      0.00       0.00       0.00       -3.49
                 5   -134.22       1.07      0.00       0.00       0.00       -2.96
           3     4     89.65       1.43      0.00       0.00       0.00        3.05
                 5    -89.65      -1.43      0.00       0.00       0.00        5.55
           4     4    101.34       7.56      0.00       0.00       0.00        0.43
                 5   -101.34      12.69      0.00       0.00       0.00      -15.82
           5     4     67.91       9.44      0.00       0.00       0.00        5.33
                 5    -67.91      10.81      0.00       0.00       0.00       -9.44
     7     1     5     -1.98      22.41      0.00       0.00       0.00       23.75
                 6      1.98      18.09      0.00       0.00       0.00      -10.77
           2     5     72.35      -1.12      0.00       0.00       0.00       -2.94
                 6    -72.35       1.12      0.00       0.00       0.00       -3.79
           3     5    196.97       1.25      0.00       0.00       0.00        2.71
                 6   -196.97      -1.25      0.00       0.00       0.00        4.81
           4     5     52.78      15.97      0.00       0.00       0.00       15.61
                 6    -52.78      14.41      0.00       0.00       0.00      -10.92
           5     5    146.24      17.75      0.00       0.00       0.00       19.84
                 6   -146.24      12.63      0.00       0.00       0.00       -4.47
     8     1     7      2.95      13.68      0.00       0.00       0.00        8.01
                 8     -2.95      13.32      0.00       0.00       0.00       -6.90
           2     7     63.55      -0.99      0.00       0.00       0.00       -3.03
                 8    -63.55       0.99      0.00       0.00       0.00       -2.92
           3     7     63.87       1.29      0.00       0.00       0.00        4.82
                 8    -63.87      -1.29      0.00       0.00       0.00        2.92
           4     7     49.88       9.52      0.00       0.00       0.00        3.73
                 8    -49.88      10.73      0.00       0.00       0.00       -7.37
           5     7     50.12      11.23      0.00       0.00       0.00        9.62
                 8    -50.12       9.02      0.00       0.00       0.00       -2.99
     9     1     1      0.00       0.00      0.00       0.00       0.00        0.00
                 5      0.00       0.00      0.00       0.00       0.00        0.00
     A PLANE FRAME STRUCTURE WITH                            -- PAGE NO.    5
   * TENSION BRACING
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2     1   -156.32       0.00      0.00       0.00       0.00        0.00
                 5    156.32       0.00      0.00       0.00       0.00        0.00
           3     1      0.00       0.00      0.00       0.00       0.00        0.00
                 5      0.00       0.00      0.00       0.00       0.00        0.00
           4     1   -117.24       0.00      0.00       0.00       0.00        0.00
                 5    117.24       0.00      0.00       0.00       0.00        0.00
           5     1      0.00       0.00      0.00       0.00       0.00        0.00
                 5      0.00       0.00      0.00       0.00       0.00        0.00
    10     1     2      0.00       0.00      0.00       0.00       0.00        0.00
                 4      0.00       0.00      0.00       0.00       0.00        0.00
           2     2      0.00       0.00      0.00       0.00       0.00        0.00
                 4      0.00       0.00      0.00       0.00       0.00        0.00
           3     2   -111.22       0.00      0.00       0.00       0.00        0.00
                 4    111.22       0.00      0.00       0.00       0.00        0.00
           4     2      0.00       0.00      0.00       0.00       0.00        0.00
                 4      0.00       0.00      0.00       0.00       0.00        0.00
           5     2    -83.41       0.00      0.00       0.00       0.00        0.00
                 4     83.41       0.00      0.00       0.00       0.00        0.00
    11     1     3      0.00       0.00      0.00       0.00       0.00        0.00
```

```
                5      0.00      0.00     0.00      0.00      0.00       0.00
          2     3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
          3     3   -136.62      0.00     0.00      0.00      0.00       0.00
                5    136.62      0.00     0.00      0.00      0.00       0.00
          4     3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
          5     3   -102.46      0.00     0.00      0.00      0.00       0.00
                5    102.46      0.00     0.00      0.00      0.00       0.00
    12    1     2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
          2     2    -91.79      0.00     0.00      0.00      0.00       0.00
                6     91.79      0.00     0.00      0.00      0.00       0.00
          3     2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
          4     2    -68.84      0.00     0.00      0.00      0.00       0.00
                6     68.84      0.00     0.00      0.00      0.00       0.00
          5     2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
    13    1     6      0.00      0.00     0.00      0.00      0.00       0.00
                7      0.00      0.00     0.00      0.00      0.00       0.00
          2     6      0.00      0.00     0.00      0.00      0.00       0.00
                7      0.00      0.00     0.00      0.00      0.00       0.00
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.    6
   * TENSION BRACING
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          3     6    -76.79      0.00     0.00      0.00      0.00       0.00
                7     76.79      0.00     0.00      0.00      0.00       0.00
          4     6      0.00      0.00     0.00      0.00      0.00       0.00
                7      0.00      0.00     0.00      0.00      0.00       0.00
          5     6    -57.59      0.00     0.00      0.00      0.00       0.00
                7     57.59      0.00     0.00      0.00      0.00       0.00
    14    1     5      0.00      0.00     0.00      0.00      0.00       0.00
                8      0.00      0.00     0.00      0.00      0.00       0.00
          2     5    -77.83      0.00     0.00      0.00      0.00       0.00
                8     77.83      0.00     0.00      0.00      0.00       0.00
          3     5      0.00      0.00     0.00      0.00      0.00       0.00
                8      0.00      0.00     0.00      0.00      0.00       0.00
          4     5    -58.38      0.00     0.00      0.00      0.00       0.00
                8     58.38      0.00     0.00      0.00      0.00       0.00
          5     5      0.00      0.00     0.00      0.00      0.00       0.00
                8      0.00      0.00     0.00      0.00      0.00       0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   60. LOAD LIST 1 4 5
   61. PARAMETER
   62. CODE EN 1993-1-1:2005
   63. NA 1
   64. BEAM 1 ALL
   65. UNL 1.8 ALL
   66. KY 0.5 ALL
   67. CHECK CODE ALL
STEEL DESIGN
                        STAAD.PRO CODE CHECKING - BS EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
```

```
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.    7
   * TENSION BRACING
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER            1
      1 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.097        5
                      59.49 C          0.00         -5.33        4.50
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER            2
      2 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.149        5
                     135.82 C          0.00         -1.81        4.50
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER            3
      3 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.112        5
                      45.79 C          0.00         -9.62        4.50
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.    8
   * TENSION BRACING
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER            4
      4 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.129        4
                     101.47 C          0.00          3.98        4.50
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER            5
      5 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.096        4
                      45.76 C          0.00          7.37        4.50
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
```

```
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           6
      6 ST  UB457X152X52 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.176        4
                     101.34 C         0.00         -15.82        6.00
     A PLANE FRAME STRUCTURE WITH                            -- PAGE NO.    9
   * TENSION BRACING
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ      LOCATION
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           7
      7 ST  UB457X152X52 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.241        5
                     146.24 C         0.00          19.84        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ      LOCATION
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           8
      8 ST  UB457X152X52 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.096        5
                      50.12 C         0.00           9.62        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ      LOCATION
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           9
      9 LD  UA150X150X10 (BRITISH SECTIONS)
                          PASS     EC-6.2.3 (T)       0.085        4
                     117.24 T         0.00           0.00        0.00
     A PLANE FRAME STRUCTURE WITH                            -- PAGE NO.   10
   * TENSION BRACING
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ      LOCATION
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER          10
     10 LD  UA150X150X10 (BRITISH SECTIONS)
                          PASS     EC-6.2.3 (T)       0.061        5
                      83.41 T         0.00           0.00        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ      LOCATION
======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER          11
```

```
     11 LD  UA150X150X10 (BRITISH SECTIONS)
                         PASS     EC-6.2.3 (T)       0.074        5
                    102.46 T         0.00          0.00        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           12
     12 LD  UA150X150X10 (BRITISH SECTIONS)
                         PASS     EC-6.2.3 (T)       0.050        4
                     68.84 T         0.00          0.00        0.00
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.   11
   * TENSION BRACING
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           13
     13 LD  UA150X150X10 (BRITISH SECTIONS)
                         PASS     EC-6.2.3 (T)       0.042        5
                     57.59 T         0.00          0.00        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY              MZ       LOCATION
=======================================================================
*** WARNING:BEAM PARAM CHANGED FROM DEFAULT OF 3
 DESIGN MIGHT NOT PROVIDE THE WORST CASE
 UTILIZATION FOR MEMBER           14
     14 LD  UA150X150X10 (BRITISH SECTIONS)
                         PASS     EC-6.2.3 (T)       0.042        4
                     58.38 T         0.00          0.00        0.00
  ************** END OF TABULATED RESULT OF DESIGN **************
   68. FINISH
     A PLANE FRAME STRUCTURE WITH                             -- PAGE NO.   12
   * TENSION BRACING
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45: 8 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *Member Specification dialog* (on page 2786)
- *TR.18 Inactive/Delete Specification* (on page 2240)

EX. UK-5 Support Settlement on a Portal Frame

This example demonstrates the application of support displacement load (also known as a sinking support) on a space frame structure.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-5 Support Settlement on a Portal Frame.STD` when you install the program.

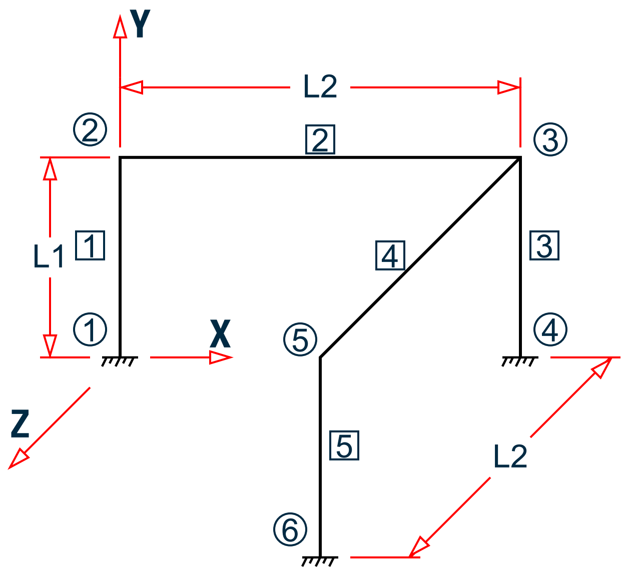


Figure 545: Example Problem No. 5

Where:

L1 = 3.0m
L2 = 6.0m

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
```

Every input has to start with the term `STAAD`. The word `SPACE` signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNITS METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.0 0.0
3 6.0 3.0 0.0 ; 4 6.0 0.0 0.0
5 6.0 3.0 6.0 ; 6 6.0 0.0. 6.0
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6
```

Defines the members by the joints to which they are connected.

```
UNIT MMS
MEMB PROP
1 TO 5 PRIS AX 6450 IZ 1.249E+08 IY 1.249E+08 IX 4.162E+06
```

Member properties have been defined above using the `PRISMATIC` attribute. Values of AX (area), IZ (moment of inertia about major axis), IY (moment of inertia about minor axis) and IX (torsional constant) are provided in `MMS` unit.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

Material are defined using the `DEFINE MATERIAL` command and then assigned using the `CONSTANT` command.

```
SUPPORT
1 4 6 FIXED
```

Joints 1, 4 and 6 are fixed supports.

```
LOADING 1 SINKING SUPPORT
```

Load case 1 is initiated followed by a title.

```
SUPPORT DISPLACEMENT LOAD
4 FY –15
```

Load 1 is a support displacement load which is also commonly known as a sinking support. `FY` signifies that the support settlement is in the global Y direction and the value of this settlement is 15mm downward.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT ANALYSIS RESULTS
```

The above `PRINT` command instructs the program to print joint displacements, support reactions and member forces.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
UNITS METER KNS
```

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.0 0.0
3 6.0 3.0 0.0 ; 4 6.0 0.0 0.0
5 6.0 3.0 6.0 ; 6 6.0 0.0 6.0
MEMBER INCIDENCE
1 1 2 3
4 3 5 ; 5 5 6
UNIT MMS
MEMB PROP
1 TO 5 PRIS AX 6450 IZ 1.249E+08 IY 1.249E+08 IX 4.162E+06
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.
POISSON 0.3
DENSITY 7.68191e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 4 6 FIXED
LOADING 1 SINKING SUPPORT
SUPPORT DISPLACEMENT LOAD
4 FY -15.
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                                    PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:10                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE TEST FOR SUPPORT DISPLACEMENT
INPUT FILE: UK-5 Support Settlement on a Portal Frame.STD
     2. UNITS METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 ; 2 0.0 3.0 0.0
     5. 3 6.0 3.0 0.0 ; 4 6.0 0.0 0.0
     6. 5 6.0 3.0 6.0 ; 6 6.0 0.0 6.0
     7. MEMBER INCIDENCE
     8. 1 1 2 3
     9. 4 3 5 ; 5 5 6
    10. UNIT MMS
    11. MEMB PROP
    12. 1 TO 5 PRIS AX 6450 IZ 1.249E+08 IY 1.249E+08 IX 4.162E+06
```

```
   13. DEFINE MATERIAL START
   14. ISOTROPIC STEEL
   15. E 210.
   16. POISSON 0.3
   17. DENSITY 7.68191E-008
   18. ALPHA 6E-006
   19. DAMP 0.03
   20. TYPE STEEL
   21. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
   22. END DEFINE MATERIAL
   23. CONSTANTS
   24. MATERIAL STEEL ALL
   25. SUPPORT
   26. 1 4 6 FIXED
   27. LOADING 1 SINKING SUPPORT
   28. SUPPORT DISPLACEMENT LOAD
   29. 4 FY -15.
   30. PERFORM ANALYSIS
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    2
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      3
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      18
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   31. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    3
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     2    1     0.2737   -0.0012   -0.0323   -0.0002    0.0006   -0.0019
     3    1     0.2735   -1.4975   -0.2735   -0.0018    0.0000   -0.0018
     4    1     0.0000   -1.5000    0.0000    0.0000    0.0000    0.0000
     5    1     0.0323   -0.0012   -0.2737   -0.0019   -0.0006   -0.0002
     6    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KNS  MMS     STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      0.46      5.63      0.90   2780.50    -67.13  15492.89
     4    1      0.43    -11.26     -0.43  15509.16      0.00  15509.16
     6    1     -0.90      5.63     -0.46  15492.89     67.13   2780.50
     TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KNS  MMS      (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      5.63     -0.46     0.90    -67.13  -2780.50   15492.89
                2     -5.63      0.46    -0.90     67.13     94.34  -16878.26
     2    1     2      0.46      5.63     0.90     94.34    -67.13   16878.26
                3     -0.46     -5.63    -0.90    -94.34  -5305.18   16904.29
     3    1     3    -11.26     -0.43     0.43     -0.00 -16809.94  -16809.94
                4     11.26      0.43    -0.43      0.00  15509.16   15509.16
```

```
     4    1     3      0.46     -5.63    -0.90    -94.34   5305.18  -16904.29
                5     -0.46      5.63     0.90     94.34     67.13  -16878.26
     5    1     5      5.63      0.90     0.46     67.13 -16878.26     -94.34
                6     -5.63     -0.90    -0.46    -67.13  15492.89    2780.50
 ************** END OF LATEST ANALYSIS RESULT **************
  32. FINISH
          *********** END OF THE STAAD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:45:10 ****
    TEST FOR SUPPORT DISPLACEMENT                            -- PAGE NO.    6
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

EX. UK-6 Prestress and Poststress Loading

This is an example of prestress loading in a plane frame structure.

It covers two situations:

1. From the member on which it is applied, the prestressing effect is transmitted to the rest of the structure through the connecting members (known in the program as PRESTRESS load).
2. The prestressing effect is experienced by the member(s) alone and not transmitted to the rest of the structure (known in the program as POSTSTRESS load).

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-6 Prestress and Poststress Loading.STD` when you install the program.

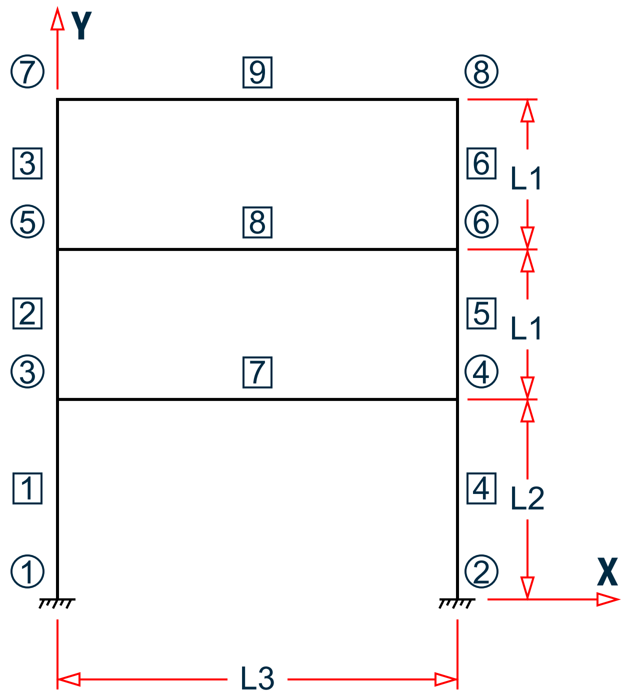


Figure 546: Example Problem No. 6

Where:

L1 = 4.5 m
L2 = 6 m
L3 = 12 m

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE FRAME WITH PRESTRESSING LOAD
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORD
1 0. 0. ; 2 12. 0. ; 3 0. 6. ; 4 12. 6.
5 0. 10.5 ; 6 12. 10.5 ; 7 0. 15. ; 8 12. 15.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
```

Defines the members by the joints to which they are connected.

```
SUPPORT
1 2 FIXED
```

The supports at joints 1 and 2 are defined to be fixed supports.

```
MEMB PROP
1 TO 9 PRI AX 0.2044 IZ 8.631E-03
```

Member properties are provided using the PRI (prismatic) attribute. Values of area (AX) and moment of inertia about the major axis (IZ) are provided.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

TThe `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. Length unit is changed from METER to MMS to facilitate the input.

```
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
```

Load case 1 is initiated followed by a title. Load 1 contains `PRESTRESS` load. Members 7 and 8 have a cable force of 1350 KNs. The location of the cable at the start (`ES`) and end (`EE`) is 75 MMs above the centre of gravity while at the middle (`EM`) it is 300 MMs below the c.g. The assumptions and facts associated with this type of loading are explained in G.15.5 Prestress and Poststress Member Load (on page 2131).

```
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
```

Load case 2 is initiated followed by a title. Load 2 is a `POSTSTRESS` load. Members 7 and 8 have a cable force of 1350 KNs. The location of the cable is the same as in load case 1. For a difference between PRESTRESS loading and POSTSTRESS loading, as well as additional information about both types of loads, please refer to G.15.5 Prestress and Poststress Member Load (on page 2131).

```
PERFORM ANALYSIS
```

This command instructs the program to perform the analysis.

```
UNIT METER
PRINT ANALYSIS RESULTS
```

The preceding command is an instruction to write joint displacements, support reactions and member forces in the output file. The preceding line causes the results to be written in the length unit of meters.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE FRAME WITH PRESTRESSING LOAD
UNIT METER KNS
JOINT COORD
1 0. 0. ; 2 12. 0. ; 3 0. 6. ; 4 12. 6.
5 0. 10.5 ; 6 12. 10.5 ; 7 0. 15. ; 8 12. 15.
MEMBER INCIDENCE
1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
SUPPORT
1 2 FIXED
MEMB PROP
1 TO 9 PRI AX 0.2044 IZ 8.631E-03
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
LOADING 1 PRESTRESSING LOAD
MEMBER PRESTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
LOADING 2 POSTSTRESSING LOAD
MEMBER POSTSTRESS
7 8 FORCE 1350. ES 75. EM -300. EE 75.
PERFORM ANALYSIS
UNIT METER
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:12                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE FRAME WITH PRESTRESSING LOAD
INPUT FILE: UK-6 Prestress and Poststress Loading.STD
     2. UNIT METER KNS
     3. JOINT COORD
```

```
    4. 1 0. 0. ; 2 12. 0. ; 3 0. 6. ; 4 12. 6.
    5. 5 0. 10.5 ; 6 12. 10.5 ; 7 0. 15. ; 8 12. 15.
    6. MEMBER INCIDENCE
    7. 1 1 3 ; 2 3 5 ; 3 5 7 ; 4 2 4 ; 5 4 6
    8. 6 6 8 ; 7 3 4 ; 8 5 6 ; 9 7 8
    9. SUPPORT
   10. 1 2 FIXED
   11. MEMB PROP
   12. 1 TO 9 PRI AX 0.2044 IZ 8.631E-03
   13. UNIT MMS
   14. DEFINE MATERIAL START
   15. ISOTROPIC CONCRETE
   16. E 21.0
   17. POISSON 0.17
   18. DENSITY 2.36158E-008
   19. ALPHA 5E-006
   20. DAMP 0.05
   21. G 9.25
   22. TYPE CONCRETE
   23. STRENGTH FCU 0.0275
   24. END DEFINE MATERIAL
   25. CONSTANTS
   26. MATERIAL CONCRETE ALL
   27. LOADING 1 PRESTRESSING LOAD
   28. MEMBER PRESTRESS
   29. 7 8 FORCE 1350. ES 75. EM -300. EE 75.
   30. LOADING 2 POSTSTRESSING LOAD
   31. MEMBER POSTSTRESS
   32. 7 8 FORCE 1350. ES 75. EM -300. EE 75.
   33. PERFORM ANALYSIS
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    2
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          8  NUMBER OF MEMBERS       9
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      18
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   34. UNIT METER
   35. PRINT ANALYSIS RESULTS
 ANALYSIS RESULTS
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    3
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     2    1    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     3    1    0.1917    0.0000    0.0000    0.0000    0.0000    0.0004
          2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     4    1   -0.1917    0.0000    0.0000    0.0000    0.0000   -0.0004
          2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     5    1    0.1799    0.0000    0.0000    0.0000    0.0000    0.0008
          2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     6    1   -0.1799    0.0000    0.0000    0.0000    0.0000   -0.0008
```

```
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     7    1    -0.0014    0.0000    0.0000    0.0000    0.0000    0.0002
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     8    1     0.0014    0.0000    0.0000    0.0000    0.0000   -0.0002
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    4
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1    -30.59      0.00      0.00      0.00      0.00     80.49
          2      0.00      0.00      0.00      0.00      0.00      0.00
     2    1     30.59      0.00      0.00      0.00      0.00    -80.49
          2      0.00      0.00      0.00      0.00      0.00      0.00
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    5
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1     -0.00     30.59     0.00      0.00      0.00      80.49
                3      0.00    -30.59     0.00      0.00      0.00     103.07
          2     1      0.00      0.00     0.00      0.00      0.00       0.00
                3      0.00      0.00     0.00      0.00      0.00       0.00
     2    1     3     -0.00     62.59     0.00      0.00      0.00     121.89
                5      0.00    -62.59     0.00      0.00      0.00     159.77
          2     3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
     3    1     5     -0.00     10.30     0.00      0.00      0.00      50.98
                7      0.00    -10.30     0.00      0.00      0.00      -4.64
          2     5      0.00      0.00     0.00      0.00      0.00       0.00
                7      0.00      0.00     0.00      0.00      0.00       0.00
     4    1     2      0.00    -30.59     0.00      0.00      0.00     -80.49
                4     -0.00     30.59     0.00      0.00      0.00    -103.07
          2     2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
     5    1     4      0.00    -62.59     0.00      0.00      0.00    -121.89
                6     -0.00     62.59     0.00      0.00      0.00    -159.77
          2     4      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
     6    1     6      0.00    -10.30     0.00      0.00      0.00     -50.98
                8     -0.00     10.30     0.00      0.00      0.00       4.64
          2     6      0.00      0.00     0.00      0.00      0.00       0.00
                8      0.00      0.00     0.00      0.00      0.00       0.00
     7    1     3   1371.41   -168.75     0.00      0.00      0.00    -326.21
                4  -1371.41   -168.75     0.00      0.00      0.00     326.21
          2     3   1339.41   -168.75     0.00      0.00      0.00    -101.25
                4  -1339.41   -168.75     0.00      0.00      0.00     101.25
     8    1     5   1287.12   -168.75     0.00      0.00      0.00    -312.00
                6  -1287.12   -168.75     0.00      0.00      0.00     312.00
          2     5   1339.41   -168.75     0.00      0.00      0.00    -101.25
                6  -1339.41   -168.75     0.00      0.00      0.00     101.25
     9    1     7    -10.30     -0.00     0.00      0.00      0.00       4.64
                8     10.30      0.00     0.00      0.00      0.00      -4.64
     FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    6
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2     7      0.00      0.00     0.00      0.00      0.00       0.00
```

```
                8      0.00      0.00     0.00      0.00      0.00       0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    36. FINISH
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:45:13 ****
      FRAME WITH PRESTRESSING LOAD                             -- PAGE NO.    7
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *TR.32.5 Prestress Load Specification* (on page 2489)
- *Member Load tab* (on page 2842)
- *M. To add a prestress or post-tension load to members* (on page 836)

EX. UK-7 Modeling Offset Connections in a Frame

This example illustrates modeling of structures with offset connections. Offset connections arise when the center lines of the connected members do not intersect at the connection point. The connection eccentricity behaves as a rigid link and is modeled through specification of `MEMBER OFFSETS`.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-7 Modeling Offset Connections in a Frame.STD` when you install the program.

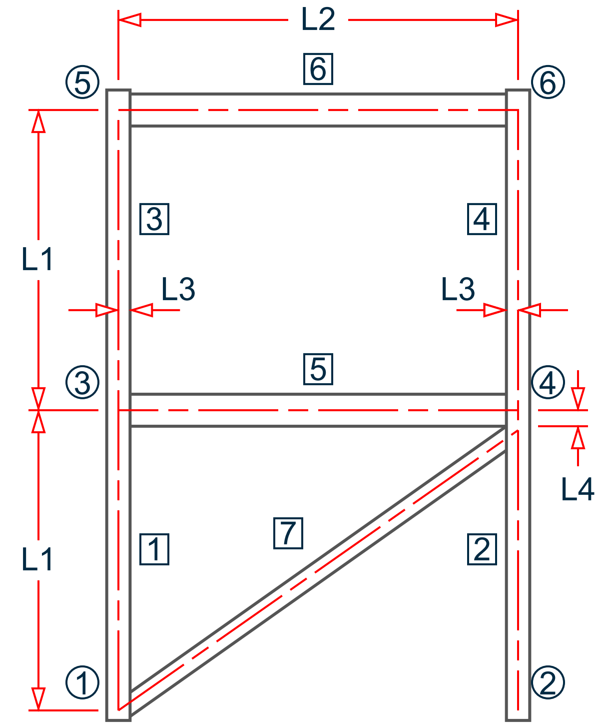


Figure 547: Example Problem No. 7

Where:

L1 = 4.5 m
L2 = 6 m
L3 = 178 mm
L4 = 152 mm

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE TEST FOR MEMBER OFFSETS
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORD
1 0. 0. ; 2 6. 0. ; 3 0. 4.5
4 6. 4.5 ; 5 0. 9. ; 6 6. 9.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
SUPPORT
1 2 PINNED
```

Pinned supports are specified at joints 1 and 2. The word PINNED signifies that no moments will be carried by these supports.

```
MEMB INCI
1 1 3 2 ; 3 3 5 4
5 3 4 ; 6 5 6 ; 7 1 4
```

Defines the members by the joints to which they are connected.

```
MEMB PROP BRITISH
1 TO 4 TABLE ST UC356X368X129
5 6 TA ST UB305X165X40
7 TA LD UA200X150X12
```

All member properties are from British steel table. The word ST stands for standard single section. LD stands for long leg back-to-back double angle.

```
UNIT MMS
MEMB OFFSET
5 6 START 178. 0.0 0.0
5 6 END -178. 0.0 0.0
7 END -178.0 -152.0 0.0
```

The preceding specification states that an OFFSET is located at the START/END joint of the members. The X, Y, and Z global coordinates of the offset distance from the corresponding incident joint are also provided. These attributes are applied to members 5, 6, and 7.

```
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
LOADING 1 WIND LOAD
```

Load case 1 is initiated followed by a title.

```
JOINT LOAD
3 FX 225.0 ; 5 FX 112.5
```

Load 1 contains joint loads at nodes 3 and 5. FX indicates that the load is a force in the global X direction.

```
PERFORM ANALYSIS
```

The above command is an instruction to perform the analysis.

```
UNIT METER
PRINT FORCES
PRINT REACTIONS
```

The above PRINT commands are instructions for writing the member forces and support reactions to the output file. The preceding line causes the results to be written in the length unit of meters

```
FINISH
```

This command terminates a STAAD run.

Input File

```
STAAD PLANE TEST FOR MEMBER OFFSETS
UNIT METER KNS
JOINT COORD
1 0. 0. ; 2 6. 0. ; 3 0. 4.5
4 6. 4.5 ; 5 0. 9. ; 6 6. 9.
SUPPORT
1 2 PINNED
MEMB INCI
1 1 3 2 ; 3 3 5 4
5 3 4 ; 6 5 6 ; 7 1 4
MEMB PROP BRITISH
1 TO 4 TABLE ST UC356X368X129
5 6 TA ST UB305X165X40
7 TA LD UA200X150X12
UNIT MMS
MEMB OFFSET
5 6 START 178. 0.0 0.0
5 6 END -178. 0.0 0.0
7 END -178.0 -152.0 0.0
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
LOADING 1 WIND LOAD
JOINT LOAD
3 FX 225.0 ; 5 FX 112.5
PERFORM ANALYSIS
UNIT METER
PRINT FORCES
PRINT REACTIONS
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:15                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
```

```
     1. STAAD PLANE TEST FOR MEMBER OFFSETS
INPUT FILE: UK-7 Modeling Offset Connections in a Frame.STD
     2. UNIT METER KNS
     3. JOINT COORD
     4. 1 0. 0. ; 2 6. 0. ; 3 0. 4.5
     5. 4 6. 4.5 ; 5 0. 9. ; 6 6. 9.
     6. SUPPORT
     7. 1 2 PINNED
     8. MEMB INCI
     9. 1 1 3 2 ; 3 3 5 4
    10. 5 3 4 ; 6 5 6 ; 7 1 4
    11. MEMB PROP BRITISH
    12. 1 TO 4 TABLE ST UC356X368X129
    13. 5 6 TA ST UB305X165X40
    14. 7 TA LD UA200X150X12
    15. UNIT MMS
    16. MEMB OFFSET
    17. 5 6 START 178. 0.0 0.0
    18. 5 6 END -178. 0.0 0.0
    19. 7 END -178.0 -152.0 0.0
    20. DEFINE MATERIAL START
    21. ISOTROPIC STEEL
    22. E 210
    23. POISSON 0.3
    24. DENSITY 7.6977E-008
    25. ALPHA 6E-006
    26. DAMP 0.03
    27. TYPE STEEL
    28. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
    29. END DEFINE MATERIAL
    30. CONSTANTS
    31. MATERIAL STEEL ALL
    32. LOADING 1 WIND LOAD
    33. JOINT LOAD
    34. 3 FX 225.0 ; 5 FX 112.5
    35. PERFORM ANALYSIS
      TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS       7
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    36. UNIT METER
    37. PRINT FORCES
  FORCES
      TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    3
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1    -49.14    -20.23     0.00      0.00      0.00       4.59
                 3     49.14     20.23     0.00      0.00      0.00     -95.65
      2    1     2    337.50    -25.11     0.00      0.00      0.00       0.00
                 4   -337.50     25.11     0.00      0.00      0.00    -112.99
```

```
     3    1     3    -30.77     53.79     0.00      0.00      0.00     150.98
                5     30.77    -53.79     0.00      0.00      0.00      91.06
     4    1     4     30.77     58.71     0.00      0.00      0.00     170.67
                6    -30.77    -58.71     0.00      0.00      0.00      93.55
     5    1     3    299.02    -18.38     0.00      0.00      0.00     -52.06
                4   -299.02     18.38     0.00      0.00      0.00     -51.65
     6    1     5     58.71    -30.77     0.00      0.00      0.00     -85.58
                6    -58.71     30.77     0.00      0.00      0.00     -88.07
     7    1     1   -479.29     -1.96     0.00      0.00      0.00      -4.59
                4    479.29      1.96     0.00      0.00      0.00      -9.62
   ************** END OF LATEST ANALYSIS RESULT **************
    38. PRINT REACTIONS
  REACTION
     TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    4
   SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1   -362.61   -337.50      0.00      0.00      0.00      0.00
     2    1     25.11    337.50      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    39. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45:15 ****
     TEST FOR MEMBER OFFSETS                                  -- PAGE NO.    5
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-8 Concrete Design for a Space Frame

In this example, concrete design is performed on some members of a space frame structure. Design calculations consist of computation of reinforcement for beams and columns. Secondary moments on the columns are obtained through the means of a P-Delta analysis.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-8 Concrete Design for a Space Frame.STD` when you install the program.

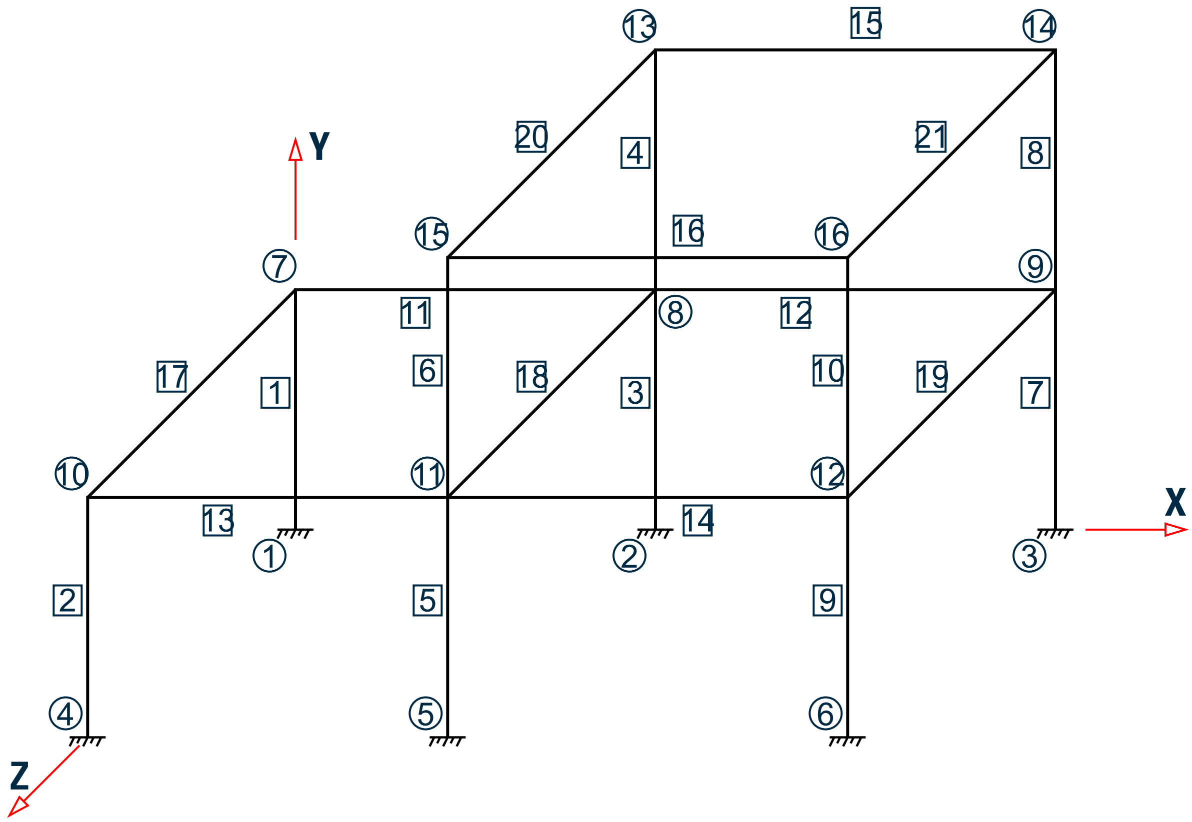


Figure 548: Example Problem No. 8

The above example represents a space frame, and the members are made of concrete. The input in the next page will show the dimensions of the members.

Two load cases, namely one for dead plus live load and another with dead, live and wind load, are considered in the design.

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE FRAME WITH CONCRETE DESIGN
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATE
1 0 0 0 ; 2 5.4 0 0 ; 3 11.4 0. 0
4 0 0 7.2 ; 5 5.4 0 7.2 ; 6 11.4 0 7.2
7 0 3.6 0 ; 8 5.4 3.6 0 ; 9 11.4 3.6 0
10 0 3.6 7.2 ; 11 5.4 3.6 7.2 ; 12 11.4 3.6 7.2
13 5.4 7.2 0 ; 14 11.4 7.2 0 ; 15 5.4 7.2 7.2
16 11.4 7.2 7.2
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14  ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16
```

Defines the members by the joints to which they are connected.

```
UNIT MMS
MEMB PROP
1 2 PRISMATIC YD 300.0 IZ 2.119E08 IY 2.119E08 -
IX 4.237E08
3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 –
IX 5.324E08
11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 –
IX 2.704E09
```

All member properties are provided using the `PRISMATIC` option. YD and ZD stand for depth and width. If ZD is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. For this particular example, moments of inertia (`IZ`, `IY`) and torsional constant (`IX`) are provided, so these will not be recalculated. The `IX`, `IY`, and `IZ` values provided in this example are only half the values of a full section to account for the fact that the full moments of inertia will not be effective due to cracking of concrete.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.356e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
```

The `DEFINE MATERIAL` command is used to define material properties for concrete. The `CONSTANT` command is used to assign this to all members.

```
SUPPORT
1 TO 6 FIXED
```

Joints 1 to 6 are fixed supports.

```
LOAD 1 (1.4DL + 1.7LL)
```

Load case 1 is initiated followed by a title.

```
SELF Y -1.4
```

The selfweight of the structure is applied in the global Y direction with a -1.4 factor. Since global Y is vertically upward, the negative factor indicates that this load will act downwards.

```
MEMB LOAD
11 TO 16 UNI Y -42.0
11 TO 16 UNI Y -76.5
```

Load 1 contains member loads also. Y indicates that the load is in the local Y direction. The word `UNI` stands for uniformly distributed load.

```
LOAD 2 .75 (1.4DL + 1.7LL + 1.7WL)
```

Load case 2 is initiated followed by a title.

```
REPEAT LOAD
1 0.75
```

The preceding command will gather the load data values from load case 1, multiply them with a factor of 0.75 and utilize the resulting values in load 2.

```
JOINT LOAD
15 16 FZ 40.0
11 FZ 90.0
12 FZ 70.0
10 FZ 40.0
```

Load 2 contains some additional joint loads also. `FZ` indicates that the load is a force in the global Z direction.

```
PDELTA ANALYSIS
```

This command instructs the program to proceed with the analysis. The analysis type is P-DELTA indicating that second-order effects are to be calculated.

```
PRINT FORCES LIST 2 5 9 14 16
```

Member end forces are printed using the above `PRINT` commands. The `LIST` option restricts the print output to the members listed.

```
START CONCRETE DESIGN
```

The above command initiates a concrete design.

```
CODE BRITISH
TRACK 1.0 MEMB 14
TRACK 2.0 MEMB 16
MAXMAIN 40 ALL
```

The values for the concrete design parameters are defined in the above commands. Design is performed per the BS 8110 Code. The TRACK value dictates the extent of design related information provided in the output. MAXMAIN indicates that the maximum size of main reinforcement is the 40 mm bar. These parameters are described in the manual where British concrete design related information is available.

```
DESIGN BEAM 14 16
```

The above command instructs the program to design beams 14 and 16 for flexure, shear, and torsion.

```
DESIGN COLUMN 2 5
```

The above command instructs the program to design columns 2 and 5 for axial load and biaxial bending.

```
END CONCRETE DESIGN
```

This will end the concrete design.

```
FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE FRAME WITH CONCRETE DESIGN
UNIT METER KNS
JOINT COORDINATE
1   0.0  0.0  0.0 ;  2   5.4  0.0  0.0
3  11.4  0.0  0.0 ;  4   0.0  0.0  7.2
5   5.4  0.0  7.2 ;  6  11.4  0.0  7.2
7   0.0  3.6  0.0 ;  8   5.4  3.6  0.0
9  11.4  3.6  0.0 ; 10   0.0  3.6  7.2
11  5.4  3.6  7.2 ; 12  11.4  3.6  7.2
13  5.4  7.2  0.0 ; 14  11.4  7.2  0.0
15  5.4  7.2  7.2 ; 16  11.4  7.2  7.2
MEMBER INCIDENCE
1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
9 6 12 ; 10 12 16 ; 11 7 8 12
13 10 11 14 ; 15 13 14 ; 16 15 16
17 7 10 ; 18 8 11 ; 19 9 12
20 13 15 ; 21 14 16
UNIT MMS
MEMB PROP
1 2 PRISMATIC YD 300.0 IZ 2.119E08 IY 2.119E08 -
IX 4.237E08
3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 -
IX 5.324E08
11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 -
IX 2.704E09
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.356e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
SUPPORT
1 TO 6 FIXED
LOAD 1 (1.4DL + 1.7LL)
SELF Y -1.4
MEMB LOAD
11 TO 16 UNI Y -42.0
11 TO 16 UNI Y -76.5
LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
REPEAT LOAD
1 0.75
JOINT LOAD
15 16 FZ 40.0
11 FZ 90.0
12 FZ 70.0
```

```
10 FZ 40.0
PDELTA ANALYSIS
PRINT FORCES LIST 2 5 9 14 16
START CONCRETE DESIGN
CODE BS8007
TRACK 1.0 MEMB 14
TRACK 2.0 MEMB 16
MAXMAIN 40 ALL
DESIGN BEAM 14 16
DESIGN COLUMN 2 5
END CONCRETE DESIGN
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:17                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FRAME WITH CONCRETE DESIGN
INPUT FILE: UK-8 Concrete Design for a Space Frame.STD
     2. UNIT METER KNS
     3. JOINT COORDINATE
     4. 1   0.0  0.0  0.0 ;  2   5.4  0.0  0.0
     5. 3  11.4  0.0  0.0 ;  4   0.0  0.0  7.2
     6. 5   5.4  0.0  7.2 ;  6  11.4  0.0  7.2
     7. 7   0.0  3.6  0.0 ;  8   5.4  3.6  0.0
     8. 9  11.4  3.6  0.0 ; 10   0.0  3.6  7.2
     9. 11  5.4  3.6  7.2 ; 12  11.4  3.6  7.2
    10. 13  5.4  7.2  0.0 ; 14  11.4  7.2  0.0
    11. 15  5.4  7.2  7.2 ; 16  11.4  7.2  7.2
    12. MEMBER INCIDENCE
    13. 1 1 7 ; 2 4 10 ; 3 2 8 ; 4 8 13
    14. 5 5 11 ; 6 11 15 ; 7 3 9 ; 8 9 14
    15. 9 6 12 ; 10 12 16 ; 11 7 8 12
    16. 13 10 11 14 ; 15 13 14 ; 16 15 16
    17. 17 7 10 ; 18 8 11 ; 19 9 12
    18. 20 13 15 ; 21 14 16
    19. UNIT MMS
    20. MEMB PROP
    21. 1 2 PRISMATIC YD 300.0 IZ 2.119E08 IY 2.119E08 -
    22. IX 4.237E08
    23. 3 TO 10 PR YD 300.0 ZD 300.0 IZ 3.596E08 IY 3.596E08 -
    24. IX 5.324E08
    25. 11 TO 21 PR YD 535.0 ZD 380 IZ 2.409E09 IY 1.229E09 -
    26. IX 2.704E09
    27. DEFINE MATERIAL START
    28. ISOTROPIC CONCRETE
    29. E 21.0
    30. POISSON 0.17
```

```
   31. DENSITY 2.356E-008
   32. ALPHA 5E-006
   33. DAMP 0.05
   34. G 9.25
   35. TYPE CONCRETE
   36. STRENGTH FCU 0.0275
   37. END DEFINE MATERIAL
   38. CONSTANTS
     FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    2
   39. MATERIAL CONCRETE ALL
   40. UNIT METER
   41. SUPPORT
   42. 1 TO 6 FIXED
   43. LOAD 1 (1.4DL + 1.7LL)
   44. SELF Y -1.4
   45. MEMB LOAD
   46. 11 TO 16 UNI Y -42.0
   47. 11 TO 16 UNI Y -76.5
   48. LOAD 2 .75(1.4DL + 1.7LL + 1.7WL)
   49. REPEAT LOAD
   50. 1 0.75
   51. JOINT LOAD
   52. 15 16 FZ 40.0
   53. 11 FZ 90.0
   54. 12 FZ 70.0
   55. 10 FZ 40.0
   56. PDELTA ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         16  NUMBER OF MEMBERS      21
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      60
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  ++ Adjusting Displacements.
   57. PRINT FORCES LIST 2 5 9 14 16
 FORCES   LIST     2       5
     FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    3
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     2    1     4    301.86    -18.04    -2.84      0.00      3.49     -23.83
               10   -293.46     18.04     2.84     -0.00      6.73     -41.68
          2     4    243.90    -14.96   -30.23      1.39     57.98     -20.50
               10   -237.60     14.96    30.23     -1.39     57.04     -33.98
     5    1     5   1290.04     -2.02    -2.94      0.00      3.79      -5.67
               11  -1279.35      2.02     2.94     -0.00      6.82      -4.10
          2     5   1015.94     -3.81   -55.16      1.18    121.20      -8.83
               11  -1007.92      3.81    55.16     -1.18    111.88      -7.55
     9    1     6    758.52     20.05    -2.75     -0.00      3.44      22.20
               12   -747.84    -20.05     2.75      0.00      6.48      48.55
          2     6    620.14     12.89   -60.00      0.16    124.29      12.18
               12   -612.12    -12.89    60.00     -0.16    113.38      32.62
    14    1    11    -40.76    435.24     0.00     -0.29     -0.01     498.88
               12     40.76    315.99    -0.00      0.29     -0.00    -141.08
```

```
         2    11    -34.70    327.08     3.02     -1.14    -12.30     375.74
              12     34.70    236.34    -3.02      1.14     -5.89    -103.49
   16    1    15     60.81    378.36    -0.00      0.04     -0.00     139.84
              16    -60.81    372.88     0.00     -0.04      0.00    -123.47
         2    15     45.61    283.60     0.40     -0.13     -1.29     104.67
              16    -45.61    279.83    -0.40      0.13     -1.01     -93.39
  ************** END OF LATEST ANALYSIS RESULT **************
   58. START CONCRETE DESIGN
CONCRETE DESIGN
   59. CODE BS8007
 PROGRAM CODE REVISION V1.0_8007_87/1
   60. TRACK 1.0 MEMB 14
   61. TRACK 2.0 MEMB 16
   62. MAXMAIN 40 ALL
   63. DESIGN BEAM 14 16
     FRAME WITH CONCRETE DESIGN                              -- PAGE NO.    4
  ====================================================================
         B E A M  N O.    14 D E S I G N  R E S U L T S -  FLEXURE
     LEN -  6000. mm   FY - 460.  FC - 30.  SIZE -  380. X  535. mm
  LEVEL    HEIGHT     BAR INFO       FROM           TO        ANCHOR
             mm                       mm            mm       STA  END
  -------------------------------------------------------------------
    1        35.      5- 20 MM       740.         6000.      NO   YES
  |-----------------------------------------------------------------|
  |   CRITICAL POS MOMENT=    257.51 KN-M   AT  3500. mm, LOAD  1   |
  |  REQD STEEL=  1480. mm2, ROW=0.0073, ROWMX=0.0400,ROWMN=0.0013 |
  |   MAX/MIN/ACTUAL BAR SPACING=  174./  45./  58. mm              |
  |-----------------------------------------------------------------|
    2       487.      4- 32 MM         0.         1760.      YES  NO
           COMP.      5- 10 MM   (REQD. STEEL=    375. SQ. MM)
  |-----------------------------------------------------------------|
  |   CRITICAL NEG MOMENT=    498.88 KN-M   AT     0. mm, LOAD  1   |
  |  REQD STEEL=  3192. mm2, ROW=0.0157, ROWMX=0.0400,ROWMN=0.0013 |
  |   MAX/MIN/ACTUAL BAR SPACING=  171./  35./  70. mm              |
  |-----------------------------------------------------------------|
    3       497.      7- 12 MM      5240.         6000.      NO   YES
  |-----------------------------------------------------------------|
  |   CRITICAL NEG MOMENT=    141.08 KN-M   AT  6000. mm, LOAD  1   |
  |  REQD STEEL=   762. mm2, ROW=0.0037, ROWMX=0.0400,ROWMN=0.0013 |
  |   MAX/MIN/ACTUAL BAR SPACING=  170./  37./  41. mm              |
  |-----------------------------------------------------------------|
       B E A M   N O.    14 D E S I G N  R E S U L T S -  SHEAR
                   PROVIDE SHEAR LINKS  AS FOLLOWS
  |-----------------------------------------------------------------|
  | FROM -  TO      | MAX. SHEAR | LOAD | LINKS | NO. | SPACING C/C |
  |-----------------|------------|------|-------|-----|-------------|
  | END 1  2001 mm |   435.2  kN |    1 | 8  mm | 28  |    74   mm   |
  | 2001   3502 mm |   184.8  kN |    1 | 8  mm |  6  |   250   mm   |
  |  3499  5000 mm |   190.8  kN |   1  | 8  mm |  6  |   250   mm   |
  |  5000   END 2  |   316.0  kN |    1 | 8  mm | 10  |   111   mm   |
  |-----------------------------------------------------------------|
     FRAME WITH CONCRETE DESIGN                              -- PAGE NO.    5
 ___  11J____________________ 6000.X 380.X 535_____________________  12J____
|                                                                  =========||
||====================                                  7No12 H*****5240.TO|6000 |
| 4No32 H*****|||0.TO|1760                                          | | | | | |
| 28*8 c/c 74|||||||||                                         10*8 c/c111| | |
|        |||||||||||||                                              | | | | | |
```

```
|        |5No20|H35.00|740.TO 6000                             | | | | | |
|        =================================================================||
|_________________________________________________________________________|

 ___________    ___________    ___________    ___________    ___________
|           |  |           |  |           |  |           |  |  ooooooo  |
|   OOOO    |  |   OOOO    |  |           |  |           |  | 7T12      |
| 4T32      |  | 4T32      |  |           |  |           |  |           |
|           |  |           |  |           |  |           |  |           |
|           |  |           |  |           |  |           |  |           |
|           |  | 5T20      |  | 5T20      |  | 5T20      |  | 5T20      |
|           |  |   ooooo   |  |   ooooo   |  |   ooooo   |  |   ooooo   |
|___________|  |___________|  |___________|  |___________|  |___________|
  ===================================================================
         B E A M   N O.    16 D E S I G N   R E S U L T S -  FLEXURE
     LEN -  6000. mm   FY - 460.  FC - 30.  SIZE -  380. X  535. mm
  LEVEL    HEIGHT     BAR INFO       FROM           TO         ANCHOR
             mm                       mm            mm       STA  END
  -------------------------------------------------------------------
    1        41.      4- 32 MM         0.         6000.      YES  YES
           COMP.      6- 8  MM   (REQD. STEEL=    264. SQ. MM)
    2       497.      7- 12 MM         0.          760.      YES  NO
    3       497.      6- 12 MM      5240.         6000.      NO   YES
  REQUIRED REINF. STEEL SUMMARY :
  -------------------------------
  SECTION     REINF STEEL(+VE/-VE)     MOMENTS(+VE/-VE)    LOAD(+VE/-VE)
  ( MM )         (SQ.  MM )                (KN-METER)
      0.           0.0/    754.7          0.00/    139.84        0/  1
    500.         264.3/      0.0         34.13/      0.00        1/  0
   1000.         971.9/      0.0        176.79/      0.00        1/  0
   1500.        1687.4/      0.0        288.09/      0.00        1/  0
   2000.        2278.7/      0.0        368.01/      0.00        1/  0
   2500.        2684.2/      0.0        416.52/      0.00        1/  0
   3000.        2829.9/    264.3        433.60/      0.00        1/  0
   3500.        2708.4/      0.0        419.25/      0.00        1/  0
   4000.        2322.3/      0.0        373.48/      0.00        1/  0
   4500.        1744.6/      0.0        296.29/      0.00        1/  0
   5000.        1037.9/      0.0        187.72/      0.00        1/  0
   5500.         264.3/      0.0         47.78/      0.00        1/  0
   6000.           0.0/    661.2          0.00/    123.47        0/  1
     FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    6
       B E A M   N O.    16 D E S I G N  R E S U L T S -  SHEAR
                   PROVIDE SHEAR LINKS  AS FOLLOWS
   |-----------------------------------------------------------------|
   | FROM -  TO      | MAX. SHEAR | LOAD | LINKS | NO. | SPACING C/C |
   |-----------------|------------|------|-------|-----|-------------|
   | END 1  2001 mm  |  378.4  kN |    1 | 8  mm | 28  |   74   mm   |
   | 2001   3002 mm  |  127.9  kN |    1 | 8  mm |  4  |  250   mm   |
   |  2999  4000 mm  |  122.5  kN |  1   | 8  mm |  4  |  250   mm   |
   |  4000   END 2   |  372.9  kN |    1 | 8  mm | 28  |   74   mm   |
   |-----------------------------------------------------------------|
 ___   15J_____________________ 6000.X 380.X 535_____________________  16J____
||========                                                        =========||
| 7No12|H*****   0.TO  760                        6No12 H*****5240.TO|6000 |
| ||||||||                                                        |||||||| |
| 28*8 c/c 74                                               28*8 c/c 74||| |
| ||||||||                                                        |||||||| |
| 4No32|H41.00   0.TO 6000                                        |||||||| |
||=========================================================================||
```

```
|_____________________________________________________________________________|
 ___________     ___________     ___________     ___________     ___________
| ooooooo   |   |           |   |           |   |           |   | oooooo    |
| 7T12      |   |           |   |           |   |           |   | 6T12      |
|           |   |           |   |           |   |           |   |           |
|           |   |           |   |           |   |           |   |           |
| 4T32      |   | 4T32      |   | 4T32      |   | 4T32      |   | 4T32      |
|   0000    |   |   0000    |   |   0000    |   |   0000    |   |   0000    |
|___________|   |___________|   |___________|   |___________|   |___________|
  ********************END OF BEAM DESIGN**************************
   64. DESIGN COLUMN 2 5
     FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    7
  ====================================================================
       C O L U M N   N O.      2  D E S I G N   R E S U L T S
     FY - 460.  FC - 30. N/MM2   CIRC SIZE   300. MM DIAMETER
        AREA OF STEEL REQUIRED =   1681.  SQ. MM.
      BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION
      ----------------------------------------------------
       15    12 MM             2.378        2    EACH END
         (ARRANGE  COLUMN REINFORCEMENTS SYMMETRICALLY)
  ====================================================================
       C O L U M N   N O.      5  D E S I G N   R E S U L T S
     FY - 460.  FC -30. N/MM2   SQRE SIZE -  300. X  300. MM,
        AREA OF STEEL REQUIRED =   2677.  SQ. MM.
      BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION
      ----------------------------------------------------
       16    16 MM             2.975        2    EACH END
         (ARRANGE  COLUMN REINFORCEMENTS SYMMETRICALLY)
  ********************END OF COLUMN DESIGN RESULTS********************
   65. END CONCRETE DESIGN
   66. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:45:17 ****
     FRAME WITH CONCRETE DESIGN                               -- PAGE NO.    8
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-9 Modeling Slabs and Shear Walls Using Finite Elements

The space frame structure in this example consists of frame members and finite elements (plates). The finite element part is used to model floor slabs and a shear wall. Concrete design of an element is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-9 Modeling Slabs and Shear Walls Using Finite Elements.STD` when you install the program.

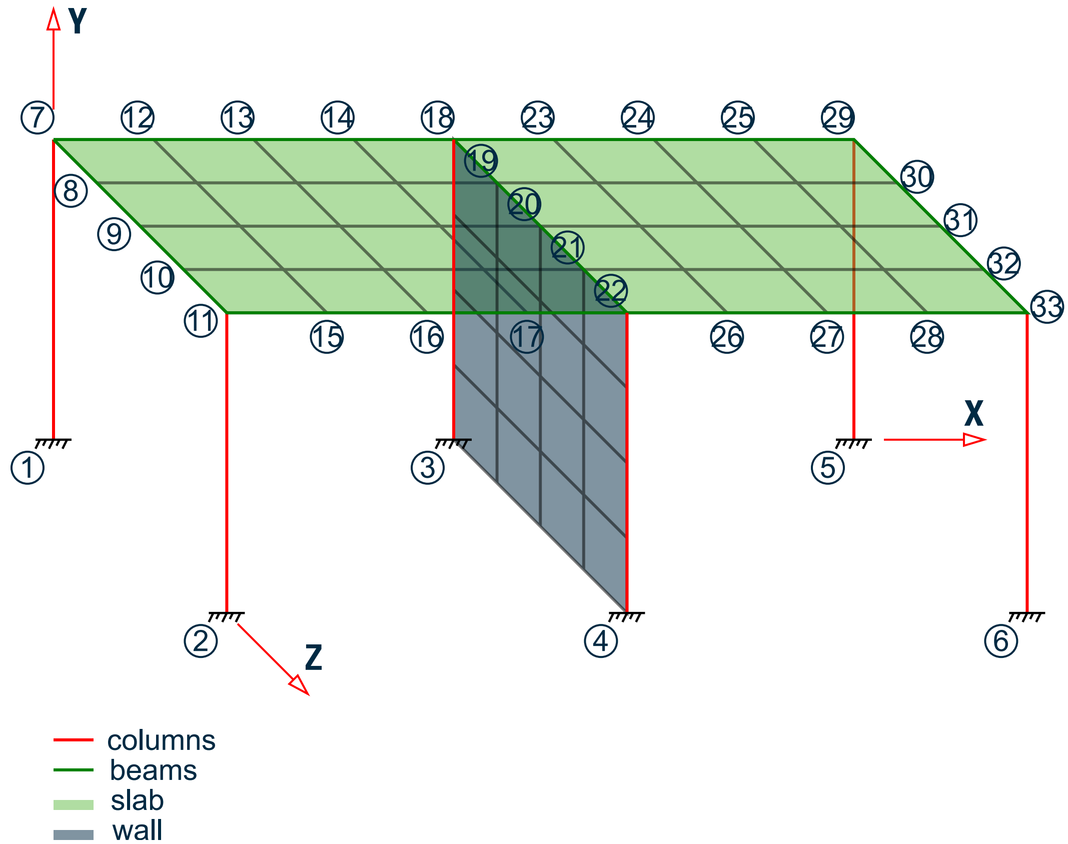


Figure 549: Example Problem No. 9

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND FINITE ELEMENTS
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line forms the title to identify this project.

```
UNIT METER NEWTON
```

The units for the data that follows are specified above.

```
JOINT COORD
1 0 0 0 ; 2 0 0 6
REP ALL 2 6 0 0
7 0 4.5 0 11 0 4.5 6
12 1.5 4.5 0 14 4.5 4.5 0
15 1.5 4.5 6 17 4.5 4.5 6
18 6 4.5 0 22 6 4.5 6
23 7.5 4.5 0 25 10.5 4.5 0
26 7.5 4.5 6 28 10.5 4.5 6
29 12 4.5 0 33 12 4.5 6
34 6 1.125 0 36 6 3.375 0
37 6 1.125 6 39 6 3.375 6
```

The joint numbers and their coordinates are defined through the above set of commands. The automatic generation facility has been used several times in the above lines. See TR.11 Joint Coordinates Specification (on page 2217) where the joint coordinate generation facilities are described.

```
MEMBER INCI
*COLUMNS
1 1 7 ; 2 2 11
3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
11 5 29 ; 12 6 33
*BEAMS IN Z DIRECTION AT X=0
13 7 8 16
*BEAMS IN Z DIRECTION AT X=6.0
17 18 19 20
*BEAMS IN Z DIRECTION AT X=12.0
21 29 30 24
*BEAMS IN X DIRECTION AT Z = 0
25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
*BEAMS IN X DIRECTION AT Z = 12.0
33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
```

The member incidences are defined through the above set of commands. For some members, the member number followed by the start and end joint numbers are defined. In other cases, STAAD's automatic generation facilities are utilized. Refer to TR.12 Member Incidences Specification (on page 2221) for additional details.

```
DEFINE MESH
A JOINT 7
B JOINT 11
C JOINT 22
D JOINT 18
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
```

The above lines define the nodes of super-elements. Super-elements are plate/shell surfaces from which a number of individual plate/shell elements can be generated. In this case, the points describe the outer corners of a slab and that of a shear wall. Our goal is to define the slab and the wall as several plate/shell elements.

```
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
```

The above lines form the instructions to generate individual 4-noded elements from the super-element profiles. For example, the command `MESH ABCD 4 4` means that STAAD.Pro has to generate 16 elements from the surface formed by the points A, B, C and D with 4 elements along the edges AB & CD and 4 elements along the edges BC & DA.

```
UNIT MMS
MEMB PROP
1 TO 40 PRIS YD 300 ZD 300
```

Members 1 to 40 are defined as a rectangular prismatic section with 300 mm depth and 300 mm width.

```
ELEM PROP
41 TO 88 TH 150
```

Elements 41 to 88 are defined to be 150 mm thick.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 6 FIXED
```

Joints 1 to 6 are defined as fixed supported.

```
UNIT KNS METER
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -10.0
```

Load 1 consists of a pressure load of 10 KNS/sq.m. The intensity on elements 41 to 72. The negative sign (and the default value for the axis) indicates that the load acts opposite to the positive direction of the element local z-axis.

```
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -90.
22 FZ -450.
```

Load 2 consists of joint loads in the Z direction at joints 11, 22, and 33.

```
LOAD COMB 3
1 0.9 2 1.3
```

Load 3 is a combination of 0.9 times load case 1 and 1.3 times load case 2.

```
PERFORM ANALYSIS
```

The command to perform an elastic analysis is specified above.

```
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
```

Support reactions, members forces and element stresses are printed for load cases 1 and 3.

```
START CONCRETE DESIGN
CODE BRITISH
DESIGN ELEMENT 47
END CONCRETE DESIGN
```

The above set of command form the instructions to STAAD to perform a concrete design on element 47. Design is done according to the British code. Note that design will consist only of flexural reinforcement calculations in the longitudinal and transverse directions of the elements for the moments MX and MY.

```
    FINI
```

The STAAD run is terminated.

Input File

```
STAAD SPACE
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
* FINITE ELEMENTS
UNIT METER NEWTON
JOINT COORD
1 0 0 0 ; 2 0 0 6.0
REP ALL 2 6.0 0 0
7 0 4.5 0 11 0 4.5 6.0
12 1.5 4.5 0 14 4.5 4.5 0
15 1.5 4.5 6.0 17 4.5 4.5 6.0
18 6.0 4.5 0 22 6.0 4.5 6.0
23 7.5 4.5 0 25 10.5 4.5 0
26 7.5 4.5 6.0 28 10.5 4.5 6.0
29 12. 4.5 0 33 12. 4.5 6.0
34 6.0 1.125 0 36 6.0 3.375 0
37 6.0 1.125 6.0 39 6.0 3.375 6.0
MEMBER INCI
*COLUMNS
1 1 7 ; 2 2 11
3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
11 5 29 ; 12 6 33
*BEAMS IN Z DIRECTION AT X=0
13 7 8 16
*BEAMS IN Z DIRECTION AT X=6.0
17 18 19 20
*BEAMS IN Z DIRECTION AT X=12.0
21 29 30 24
*BEAMS IN X DIRECTION AT Z = 0
25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
*BEAMS IN X DIRECTION AT Z = 12.0
33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
DEFINE MESH
A JOINT 7
B JOINT 11
C JOINT 22
D JOINT 18
E JOINT 33
F JOINT 29
G JOINT 3
H JOINT 4
GENERATE ELEMENT
MESH ABCD 4 4
MESH DCEF 4 4
MESH DCHG 4 4
UNIT MMS
```

```
MEMB PROP
1 TO 40 PRIS YD 300 ZD 300
ELEM PROP
41 TO 88 TH 150
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 6 FIXED
UNIT METER
LOAD 1 DEAD LOAD FROM FLOOR
ELEMENT LOAD
41 TO 72 PRESSURE -10.0
LOAD 2 WIND LOAD
JOINT LOAD
11 33 FZ -90.
22 FZ -450.
LOAD COMB 3
1 0.9 2 1.3
PERFORM ANALYSIS
LOAD LIST 1 3
PRINT SUPP REAC
PRINT MEMBER FORCES LIST 27
PRINT ELEMENT STRESSES LIST 47
START CONCRETE DESIGN
CODE BS8007
DESIGN ELEMENT 47
END CONCRETE DESIGN
FINI
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45:19                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE
INPUT FILE: UK-9 Modeling Slabs and Shear Walls Using Finite Elements.STD
     2. * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
```

```
 3. * FINITE ELEMENTS
 4. UNIT METER NEWTON
 5. JOINT COORD
 6. 1 0 0 0 ; 2 0 0 6.0
 7. REP ALL 2 6.0 0 0
 8. 7 0 4.5 0 11 0 4.5 6.0
 9. 12 1.5 4.5 0 14 4.5 4.5 0
10. 15 1.5 4.5 6.0 17 4.5 4.5 6.0
11. 18 6.0 4.5 0 22 6.0 4.5 6.0
12. 23 7.5 4.5 0 25 10.5 4.5 0
13. 26 7.5 4.5 6.0 28 10.5 4.5 6.0
14. 29 12. 4.5 0 33 12. 4.5 6.0
15. 34 6.0 1.125 0 36 6.0 3.375 0
16. 37 6.0 1.125 6.0 39 6.0 3.375 6.0
17. MEMBER INCI
18. *COLUMNS
19. 1 1 7 ; 2 2 11
20. 3 3 34 ; 4 34 35 ; 5 35 36 ; 6 36 18
21. 7 4 37 ; 8 37 38 ; 9 38 39 ; 10 39 22
22. 11 5 29 ; 12 6 33
23. *BEAMS IN Z DIRECTION AT X=0
24. 13 7 8 16
25. *BEAMS IN Z DIRECTION AT X=6.0
26. 17 18 19 20
27. *BEAMS IN Z DIRECTION AT X=12.0
28. 21 29 30 24
29. *BEAMS IN X DIRECTION AT Z = 0
30. 25 7 12 ; 26 12 13 ; 27 13 14 ; 28 14 18
31. 29 18 23 ; 30 23 24 ; 31 24 25 ; 32 25 29
32. *BEAMS IN X DIRECTION AT Z = 12.0
33. 33 11 15 ; 34 15 16 ; 35 16 17 ; 36 17 22
34. 37 22 26 ; 38 26 27 ; 39 27 28 ; 40 28 33
35. DEFINE MESH
36. A JOINT 7
37. B JOINT 11
38. C JOINT 22
  STAAD SPACE                                                -- PAGE NO.    2
* EXAMPLE PROBLEM WITH FRAME MEMBERS AND
39. D JOINT 18
40. E JOINT 33
41. F JOINT 29
42. G JOINT 3
43. H JOINT 4
44. GENERATE ELEMENT
45. MESH ABCD 4 4
46. MESH DCEF 4 4
47. MESH DCHG 4 4
48. UNIT MMS
49. MEMB PROP
50. 1 TO 40 PRIS YD 300 ZD 300
51. ELEM PROP
52. 41 TO 88 TH 150
53. UNIT KNS MMS
54. DEFINE MATERIAL START
55. ISOTROPIC CONCRETE
56. E 21.0
57. POISSON 0.17
58. DENSITY 2.36158E-008
```

```
   59. ALPHA 5E-006
   60. DAMP 0.05
   61. G 9.25
   62. TYPE CONCRETE
   63. STRENGTH FCU 0.0275
   64. END DEFINE MATERIAL
   65. CONSTANTS
   66. MATERIAL CONCRETE ALL
   67. SUPPORT
   68. 1 TO 6 FIXED
   69. UNIT METER
   70. LOAD 1 DEAD LOAD FROM FLOOR
   71. ELEMENT LOAD
   72. 41 TO 72 PRESSURE -10.0
   73. LOAD 2 WIND LOAD
   74. JOINT LOAD
   75. 11 33 FZ -90.
   76. 22 FZ -450.
   77. LOAD COMB 3
   78. 1 0.9 2 1.3
   79. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         69  NUMBER OF MEMBERS      40
    NUMBER OF PLATES         48  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      6
          Using 64-bit analysis engine.
     STAAD SPACE                                            -- PAGE NO.    3
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     378
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    254 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   80. LOAD LIST 1 3
   81. PRINT SUPP REAC
 SUPP     REAC
     STAAD SPACE                                            -- PAGE NO.    4
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      8.20     74.30     10.34     15.41     -0.00    -12.21
          3      7.48     68.32     10.82     17.66      0.05    -11.06
     2    1      8.20     74.30    -10.34    -15.41      0.00    -12.21
          3      7.30     65.43     -7.74     -9.89      0.24    -11.01
     3    1     -0.00    211.41     68.03    -10.57     -0.00      0.00
          3     -0.00    791.81    476.42     12.46     -0.00      0.00
     4    1     -0.00    211.41    -68.03     10.57      0.00      0.00
          3     -0.00   -411.31    336.42     30.82      0.00      0.00
     5    1     -8.20     74.30     10.34     15.41      0.00     12.21
          3     -7.48     68.32     10.82     17.66     -0.05     11.06
     6    1     -8.20     74.30    -10.34    -15.41     -0.00     12.21
          3     -7.30     65.43     -7.74     -9.89     -0.24     11.01
  ************** END OF LATEST ANALYSIS RESULT **************
   82. PRINT MEMBER FORCES LIST 27
 MEMBER   FORCES   LIST     27
     STAAD SPACE                                            -- PAGE NO.    5
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
```

```
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    27    1    13      0.66    -11.91    -0.06      6.91      0.05     -21.61
               14     -0.66     11.91     0.06     -6.91      0.05       3.75
          3    13     22.07    -10.55    -0.71      6.42      0.52     -19.50
               14    -22.07     10.55     0.71     -6.42      0.54       3.68
  ************** END OF LATEST ANALYSIS RESULT **************
   83. PRINT ELEMENT STRESSES LIST 47
 ELEMENT  STRESSES LIST     47
     STAAD SPACE                                          -- PAGE NO.    6
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KNS  METE
  ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX           MY           MXY
                     VONT       VONB         SX           SY           SXY
                     TRESCAT    TRESCAB
    47      1        17.12        4.80      -10.40      -13.32        1.30
                   3305.41     3272.53      -12.02      -16.95        5.17
                   3704.34     3664.81
      TOP : SMAX=   -2648.50 SMIN=   -3704.34 TMAX=     527.92 ANGLE= 21.0
      BOTT: SMAX=    3664.81 SMIN=    2630.10 TMAX=     517.36 ANGLE=-69.2
            3        14.94        4.42       -9.50      -11.97        1.01
                   3075.62     2867.90      -47.26      -61.01      180.98
                   3479.94     3143.47
      TOP : SMAX=   -2353.91 SMIN=   -3479.94 TMAX=     563.02 ANGLE= 26.7
      BOTT: SMAX=    3143.47 SMIN=    2473.83 TMAX=     334.82 ANGLE=-82.2
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         3.664812E+03 -3.704341E+03  5.630193E+02  3.305413E+03  3.704341E+03
PLATE NO.      47            47            47            47            47
CASE  NO.       1             1             3             1             1
  ********************END OF ELEMENT FORCES********************
   84. START CONCRETE DESIGN
     STAAD SPACE                                          -- PAGE NO.    7
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
CONCRETE DESIGN
   85. CODE BS8007
 PROGRAM CODE REVISION V1.0_8007_87/1
   86. DESIGN ELEMENT 47
     STAAD SPACE                                          -- PAGE NO.    8
   * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
 --------------------------------------------------------------------------
|  ELEMENT  DESIGN  TO  BS8007  AND  BS8110             ELEMENT NO.    47 |
|-------------------------------------------------------------------------|
|           A >                   e          f             g              |
|    l--------|---------k    Top___________________________________       |
|    |                  |      | o_______o_______o_______o_______o |      |
|    |         ! y      |      |        outer bars // to  x        | My   |
|    |         |        |      | o-------o-------o-------o-------o |      |
|    |        z+ --> x  |    Bot-----------------------------------       |
|    |                  |                 Section A-A                     |
|    |                  |                                                 |
|    i--------|---------j    Depth=150 mm  Width=1000 mm  Cover=20 mm     |
```

```
|           A >                                                            |
|--------------------------------------------------------------------------|
| Ultimate Limit State |  12 mm Bars     |  16 mm Bars      |  20 mm Bars  |
| Max.Momnt. kNm/m  Lo |C/C  AS R. AS P.|C/C  AS R.  AS P.|C/C  AS R. AS P.|
|---------------------|---|------|-----|---|------|------|---|------|-----|
| Mx Top =   0.0     0 |200|  194 | 565 |200|  194 | 1065 |200|  194 |1572 |
| Mx Bot = -10.4     1 |200|  218 | 565 |200|  222 | 1065 |200|  226 |1572 |
| My Top =   0.0     0 |200|  194 | 565 |200|  194 | 1065 |200|  194 |1572 |
| My Bot = -13.3     1 |200|  310 | 565 |200|  327 | 1065 |200|  347 |1572 |
 --------------------------------------------------------------------------
 --------------------------------------------------------------------------
|  SERVICEABILITY    LIMIT    STATE                         ELEMENT NO.    47 |
|--------------------------------------------------------------------------|
| Longitudinal  Moments  Mx    kNm/m  | Transverse    Moments    My   kNm/m |
| Flexural      Crack    Width    mm  | Flexural      Crack      Width    mm |
|-----------------------------------|-----------------------------------------|
| Top=  0.0  L.   0 Bot= -10.4 L.   1| Top=  0.0  L.   0 Bot= -13.3 L.   1 |
|-----------------------------------|-----------------------------------------|
| 12 | 16 | 20 |  @  | 12 | 16  |  20 | 12  | 16 | 20 |  @  | 12 | 16 | 20 |
|----|----|----|-----|----|----|-----|-----|----|----|-----|----|----|-----|
|0.00|0.00|0.00|  e  |0.05|0.03|0.02 | 0.00|0.00|0.00|  e  |0.17|0.12|0.14 |
|0.00|0.00|0.00|  f  |0.11|0.06|0.05 | 0.00|0.00|0.00|  f  |0.22|0.14|0.14 |
|0.00|0.00|0.00|  g  |0.06|0.04|0.03 | 0.00|0.00|0.00|  g  |0.12|0.08|0.08 |
|--------------------------------------------------------------------------|
|  Thermal   Crack   Width   Calculations  Based  On   Wmax=Smax*R*T1*Alfa |
|--------------------------------------------------------------------------|
| Surface Type : Suspended    Constuction type : 1     Temp. Range = 30 C |
|--------------------------------------------------------------------------|
| Surface  Zones & ROWcrit |  8 mm bars |10 mm Bars |12 mm Bars |16 mm Bars |
| Top   :  75   mm   262  mm2 |-----------|-----------|-----------|-----------|
| Bot.  :  75   mm   262  mm2 | Top | Bot.| Top | Bot.| Top | Bot.| Top | Bot.|
|-------------------------|-----|-----|-----|-----|-----|-----|-----|-----|
| Smax                   mm | 765 | 765 | 957 | 957 |1148 |1148 |1531 |1531 |
| Wmax                   mm |0.14 |0.14 |0.17 |0.17 |0.21 |0.21 |0.28 |0.28 |
| Sp. For  Wmax = 0.20   mm | 277 | 277 | 347 | 347 | 416 | 416 | 555 | 555 |
 --------------------------------------------------------------------------
 ***************************END OF ELEMENT DESIGN**************************
  87. END CONCRETE DESIGN
  88. FINI
    STAAD SPACE                                              -- PAGE NO.    9
  * EXAMPLE PROBLEM WITH FRAME MEMBERS AND
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=   9:45:20 ****
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

Related Links

- *TR.11 Joint Coordinates Specification* (on page 2217)

EX. UK-10 Finite Element Model for a Rectangular Tank

A tank structure is modeled with four-noded plate elements. Water pressure from inside is used as loading for the tank. Reinforcement calculations have been done for some elements.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-10 Finite Element Model for a Rectangular Tank.STD` when you install the program.

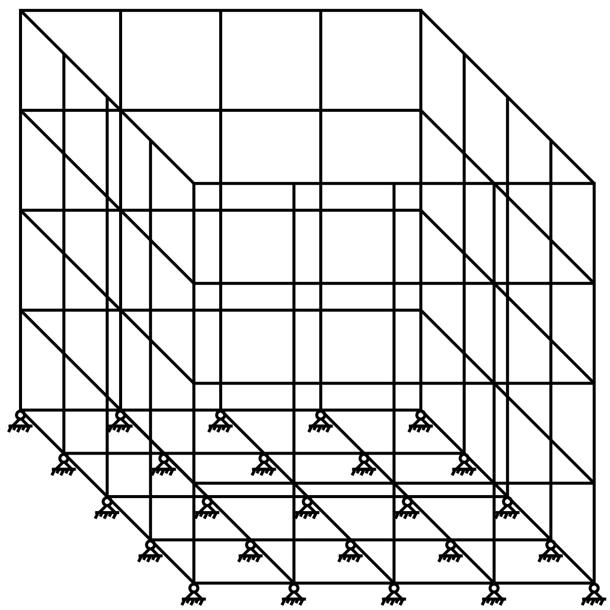


Figure 550: Tank Model

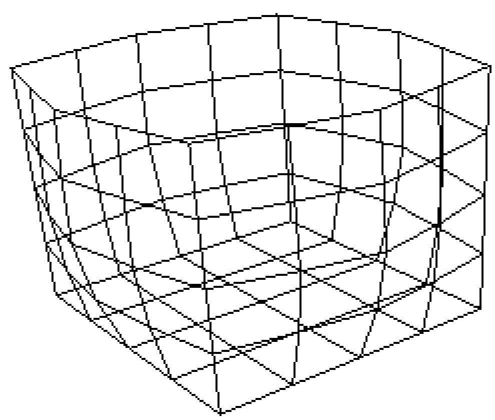


Figure 551: Deflected Shape

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK
* STRUCTURE
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D) structure.

```
UNITS METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0. 0. 0. 5 0. 6. 0.
REPEAT 4 1.5. 0. 0.
REPEAT 4 0. 0. 1.5.
REPEAT 4 -1.5. 0. 0.
REPEAT 3 0. 0. -1.5.
81 1.5. 0. 1.5. 83 1.5. 0. 4.5.
REPEAT 2 1.5. 0. 0.
```

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command generates joint coordinates by repeating the pattern of the previous line of joint coordinates. The number following the REPEAT command is the number of repetitions to be carried out. This is followed by X, Y and Z coordinate increments. See TR.11 Joint Coordinates Specification (on page 2217) .

```
ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
78 87 26 31 88
79 88 31 36 89
80 89 36 41 46
```

Element connectivities are input as above by providing the element number followed by joint numbers defining the element. The REPEAT command generates element incidences by repeating the pattern of the previous line of element nodes. The number following the REPEAT command is the number of repetitions to be carried out and that is followed by element and joint number increments. See TR.12 Member Incidences Specification (on page 2221) .

```
UNIT MMS
ELEMENT PROPERTIES
1 TO 80 TH 200.0
```

Element properties are provided by specifying that the elements are 200.0 mm THick.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.4e-008
ALPHA 5e-006
```

```
    DAMP 0.05
    G 9.25
    TYPE CONCRETE
    STRENGTH FCU 0.028
    END DEFINE MATERIAL
    CONSTANTS
    MATERIAL CONCRETE ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
    SUPPORT
    1 TO 76 BY 5 81 TO 89 PINNED
```

Pinned supports are specified at the joints listed above. No moments will be carried by these supports. The expression `1 TO 76 BY 5` means 1, 6, 11, etc. up to 76.

```
    UNIT METER
    LOAD 1
    ELEMENT LOAD
    4 TO 64 BY 4 PR 50.0
    3 TO 63 BY 4 PR 100.0
    2 TO 62 BY 4 PR 150.0
    1 TO 61 BY 4 PR 200.0
```

Load case 1 is initiated. It consists of element loads in the form of uniform pressure(indicated by `PR`) acting along the local z-axis.

```
    PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
    UNIT MMS
    PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
    PRINT ELEM FORCE LIST 9 TO 16
```

Joint displacements for a selected set of nodes and element corner forces for some elements are written in the output file as a result of the above commands. The forces printed are in the global directions at the nodes of the elements. The LIST option restricts the print output to that for the joints/elements listed.

```
    START CONCRETE DESIGN
```

The above command initiates concrete design.

```
    CODE BRITISH
    DESIGN SLAB 9 12
```

Slabs (i.e., elements) 9 and 12 will be designed and the reinforcement requirements obtained. In STAAD, elements are typically designed for the moments MX and MY at the centroid of the element.

```
    END CONCRETE DESIGN
```

Terminates the concrete design operation.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
UNIT METER KNS
JOINT COORDINATES
```

```
1 0.0 0.0 0.0 5 0.0 6.0 0.0
REPEAT 4  1.5  0.0  0.0
REPEAT 4  0.0  0.0  1.5
REPEAT 4 -1.5  0.0  0.0
REPEAT 3  0.0  0.0 -1.5
81 1.5 0.0 1.5 83 1.5 0.0 4.5
REPEAT 2  1.5  0.0  0.0
ELEMENT INCIDENCES
1 1 2 7 6 TO 4 1 1
REPEAT 14 4 5
61 76 77 2 1 TO 64 1 1
65 1 6 81 76
66 76 81 82 71
67 71 82 83 66
68 66 83 56 61
69 6 11 84 81
70 81 84 85 82
71 82 85 86 83
72 83 86 51 56
73 11 16 87 84
74 84 87 88 85
75 85 88 89 86
76 86 89 46 51
77 16 21 26 87
78 87 26 31 88
79 88 31 36 89
80 89 36 41 46
UNIT MMS
ELEMENT PROPERTIES
1 TO 80 TH 200.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.4e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.028
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORT
1 TO 76 BY 5 81 TO 89 PINNED
UNIT METER
LOAD 1
ELEMENT LOAD
4 TO 64 BY 4 PR 50.0
3 TO 63 BY 4 PR 100.0
2 TO 62 BY 4 PR 150.0
1 TO 61 BY 4 PR 200.0
PERFORM ANALYSIS
UNIT MMS
PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
PRINT ELEM FORCE LIST 9 TO 16
START CONCRETE DESIGN
CODE BS8007
```

```
DESIGN SLAB 9 12
END CONCRETE DESIGN
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44: 7                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FINITE ELEMENT MODEL OF TANK STRUCTURE
INPUT FILE: UK-10 Finite Element Model for a Rectangular Tank.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 5 0.0 6.0 0.0
     5. REPEAT 4  1.5  0.0  0.0
     6. REPEAT 4  0.0  0.0  1.5
     7. REPEAT 4 -1.5  0.0  0.0
     8. REPEAT 3  0.0  0.0 -1.5
     9. 81 1.5 0.0 1.5 83 1.5 0.0 4.5
    10. REPEAT 2  1.5  0.0  0.0
    11. ELEMENT INCIDENCES
    12. 1 1 2 7 6 TO 4 1 1
    13. REPEAT 14 4 5
    14. 61 76 77 2 1 TO 64 1 1
    15. 65 1 6 81 76
    16. 66 76 81 82 71
    17. 67 71 82 83 66
    18. 68 66 83 56 61
    19. 69 6 11 84 81
    20. 70 81 84 85 82
    21. 71 82 85 86 83
    22. 72 83 86 51 56
    23. 73 11 16 87 84
    24. 74 84 87 88 85
    25. 75 85 88 89 86
    26. 76 86 89 46 51
    27. 77 16 21 26 87
    28. 78 87 26 31 88
    29. 79 88 31 36 89
    30. 80 89 36 41 46
    31. UNIT MMS
    32. ELEMENT PROPERTIES
    33. 1 TO 80 TH 200.0
    34. DEFINE MATERIAL START
    35. ISOTROPIC CONCRETE
    36. E 21.0
    37. POISSON 0.17
    38. DENSITY 2.4E-008
      FINITE ELEMENT MODEL OF TANK STRUCTURE                    -- PAGE NO.    2
```

```
   39. ALPHA 5E-006
   40. DAMP 0.05
   41. G 9.25
   42. TYPE CONCRETE
   43. STRENGTH FCU 0.028
   44. END DEFINE MATERIAL
   45. CONSTANTS
   46. MATERIAL CONCRETE ALL
   47. SUPPORT
   48. 1 TO 76 BY 5 81 TO 89 PINNED
   49. UNIT METER
   50. LOAD 1
   51. ELEMENT LOAD
   52. 4 TO 64 BY 4 PR 50.0
   53. 3 TO 63 BY 4 PR 100.0
   54. 2 TO 62 BY 4 PR 150.0
   55. 1 TO 61 BY 4 PR 200.0
   56. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         89  NUMBER OF MEMBERS       0
    NUMBER OF PLATES         80  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     25
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     459
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   57. UNIT MMS
   58. PRINT JOINT DISPLACEMENTS LIST 5 25 45 65
 JOINT    DISPLACE LIST     5
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    3
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     5    1    -0.0103    0.0007   -0.0103    0.0002    0.0000   -0.0002
    25    1     0.0103    0.0007   -0.0103    0.0002    0.0000    0.0002
    45    1     0.0103    0.0007    0.0103   -0.0002    0.0000    0.0002
    65    1    -0.0103    0.0007    0.0103   -0.0002    0.0000   -0.0002
  ************** END OF LATEST ANALYSIS RESULT **************
   59. PRINT ELEM FORCE LIST 9 TO 16
 ELEM     FORCE    LIST     9
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    4
  ELEMENT FORCES    FORCE,LENGTH UNITS= KNS  MMS
  --------------
    GLOBAL CORNER FORCES
 JOINT     FX          FY          FZ          MX          MY          MZ
              ELE.NO.      9 FOR LOAD CASE     1
   11 -8.8840E+01 -5.4966E+01  2.9486E+02  1.3176E+05 -4.1177E+04  1.0696E+04
   12 -8.2763E+01 -6.0241E+00 -4.4118E+01  7.9554E+04  5.9496E+04 -1.6817E+04
   17  1.4375E+02  4.7401E+01 -5.6234E+01  1.2776E+05 -1.1029E+05  2.1861E+04
   16  2.7850E+01  1.3590E+01  2.5550E+02  1.4896E+05  5.3365E+04 -1.5740E+04
              ELE.NO.     10 FOR LOAD CASE     1
   12 -1.8729E+02  6.0241E+00  4.4118E+01 -7.9554E+04  1.0833E+05  4.7435E+04
   13 -1.9921E+02 -3.6766E+01  2.0250E+01  6.9592E+04  1.0234E+05 -5.8037E+04
   18  2.2995E+02 -1.3963E+01  1.1488E+02  5.8552E+04 -5.3896E+04  6.1870E+04
   17  1.5655E+02  4.4705E+01  1.5825E+02  1.8395E+03 -2.0026E+02 -5.1268E+04
              ELE.NO.     11 FOR LOAD CASE     1
```

```
   13 -2.1146E+02  3.6766E+01 -2.0250E+01 -6.9592E+04  1.2194E+05  5.7366E+04
   14 -2.3180E+02 -1.4769E+01  3.2094E+01  2.6339E+04  1.0470E+05 -6.4752E+04
   19  2.0981E+02 -3.2909E+01  1.4009E+02 -1.5592E+04 -2.4195E+04  6.5369E+04
   18  2.3345E+02  1.0912E+01  7.3067E+01 -3.0678E+04 -5.1461E+04 -5.7982E+04
              ELE.NO.     12 FOR LOAD CASE     1
   14 -1.7545E+02  1.4769E+01 -3.2094E+01 -2.6339E+04  1.0553E+05  4.3931E+04
   15 -1.7917E+02 -1.7164E-07 -5.3352E-07  5.4335E-04  9.5913E+04 -4.9179E+04
   20  1.6440E+02 -2.5971E+00  9.3630E+01 -2.8746E+04 -2.4521E+04  4.5374E+04
   19  1.9022E+02 -1.2172E+01  5.0964E+01 -9.8543E+02 -4.4410E+04 -4.0126E+04
              ELE.NO.     13 FOR LOAD CASE     1
   16 -1.0576E+02 -8.2837E+01  9.5266E+01 -1.4733E+04  2.7047E+04  1.5133E+04
   17 -3.2926E+01 -6.8334E+01  3.5521E+01 -8.9147E+03 -4.0371E+03 -2.4466E+04
   22  1.8410E+02  2.6957E+01  1.9413E+02 -1.6851E+04  6.2037E+04  2.9884E+04
   21 -4.5408E+01  1.2421E+02  1.2508E+02  3.3522E+04  5.6274E+04 -2.0551E+04
              ELE.NO.     14 FOR LOAD CASE     1
   17 -2.6737E+02 -2.3772E+01 -1.3754E+02 -1.2068E+05  1.1453E+05  5.3873E+04
   18 -2.5271E+02 -5.3156E+00 -6.4895E+01  3.2784E+04  5.0765E+04 -5.2692E+04
   23  2.8179E+02  5.6045E+01  3.1168E+02 -8.0772E+04  2.0347E+05  6.3464E+04
   22  2.3828E+02 -2.6957E+01  2.2825E+02  5.1613E+04  1.8801E+05 -6.4645E+04
              ELE.NO.     15 FOR LOAD CASE     1
   18 -2.1070E+02  8.3668E+00 -1.2305E+02 -6.0658E+04  5.4592E+04  4.8803E+04
   19 -2.3137E+02  2.9518E+01 -9.1935E+01  4.3694E+04  4.6686E+04 -5.2443E+04
   24  1.9349E+02  1.8160E+01  2.2130E+02 -7.1791E+04  1.9591E+05  4.9794E+04
   23  2.4858E+02 -5.6045E+01  2.1869E+02  6.3463E+04  1.9404E+05 -4.6155E+04
              ELE.NO.     16 FOR LOAD CASE     1
   19 -1.6865E+02  1.5563E+01 -9.9117E+01 -2.7116E+04  2.1920E+04  2.7200E+04
   20 -1.6440E+02  2.5971E+00 -9.3630E+01  2.8746E+04  2.4521E+04 -4.5374E+04
   25  1.4624E+02 -1.1240E-06  1.4624E+02 -5.5416E+04  1.6204E+05  5.5416E+04
   24  1.8681E+02 -1.8160E+01  1.5900E+02  5.9238E+04  1.6501E+05 -3.7241E+04
   60. START CONCRETE DESIGN
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    5
CONCRETE DESIGN
   61. CODE BS8007
 PROGRAM CODE REVISION V1.0_8007_87/1
   62. DESIGN SLAB 9 12
     FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    6
 --------------------------------------------------------------------------
|  ELEMENT  DESIGN  TO  BS8007  AND  BS8110            ELEMENT NO.     9 |
|--------------------------------------------------------------------------|
|            A >                         e          f             g        |
|    l--------|---------k    Top__________________________________         |
|    |                  |      | o_______o_______o_______o_______o |        |
|    |        ! y       |      |       outer bars // to  x       | My     |
|    |        |         |      | o-------o-------o-------o-------o |        |
|    |       z+ --> x   |    Bot-----------------------------------         |
|    |                  |                Section A-A                       |
|    |                  |                                                  |
|    i--------|---------j    Depth=200 mm  Width=1000 mm  Cover=20 mm      |
|            A >                                                           |
|--------------------------------------------------------------------------|
| Ultimate Limit State |  12 mm Bars     |  16 mm Bars      |  20 mm Bars  |
| Max.Momnt. kNm/m  Lo |C/C  AS R. AS P.|C/C  AS R.  AS P.|C/C  AS R. AS P.|
|----------------------|---|------|-----|---|------|------|---|------|-----|
| Mx Top =   0.0     0 |200|  259 | 565 |200|  259 | 1065 |200|  259 |1572 |
| Mx Bot = -24.5     1 |200|  367 | 565 |200|  372 | 1065 |200|  376 |1572 |
| My Top =  25.1     1 |200|  404 | 565 |200|  420 | 1065 |200|  437 |1572 |
| My Bot =   0.0     0 |200|  259 | 565 |200|  259 | 1065 |200|  259 |1572 |
 --------------------------------------------------------------------------
```

```
 -------------------------------------------------------------------------------
|  SERVICEABILITY    LIMIT    STATE                         ELEMENT NO.     9   |
|-------------------------------------------------------------------------------|
| Longitudinal  Moments  Mx   kNm/m  | Transverse    Moments    My   kNm/m  |
| Flexural      Crack    Width   mm  | Flexural      Crack      Width   mm  |
|------------------------------------|------------------------------------------|
| Top=  0.0  L.   0 Bot= -24.5 L.   1| Top= 25.1  L.   1 Bot=   0.0 L.   0  |
|------------------------------------|------------------------------------------|
| 12 | 16 | 20 |  @  | 12 | 16 |  20 | 12  | 16 | 20 |  @  | 12 | 16 | 20  |
|----|----|----|-----|----|----|-----|-----|----|----|-----|----|----|-----|
|0.00|0.00|0.00|  e  |0.08|0.04|0.03 | 0.17|0.12|0.12|  e  |0.00|0.00|0.00  |
|0.00|0.00|0.00|  f  |0.20|0.11|0.08 | 0.27|0.17|0.15|  f  |0.00|0.00|0.00  |
|0.00|0.00|0.00|  g  |0.10|0.06|0.05 | 0.13|0.08|0.08|  g  |0.00|0.00|0.00  |
|-------------------------------------------------------------------------------|
|  Thermal   Crack   Width   Calculations  Based  On   Wmax=Smax*R*T1*Alfa |
|-------------------------------------------------------------------------------|
| Surface Type : Suspended    Constuction type : 1     Temp. Range = 30 C  |
|-------------------------------------------------------------------------------|
| Surface  Zones & ROWcrit | 8 mm bars |10 mm Bars |12 mm Bars |16 mm Bars  |
| Top  : 100  mm  350  mm2 |-----------|-----------|-----------|-----------|
| Bot. : 100  mm  350  mm2 | Top | Bot.| Top | Bot.| Top | Bot.| Top | Bot.|
|--------------------------|-----|-----|-----|-----|-----|-----|-----|-----|
| Smax                  mm | 765 | 765 | 957 | 957 |1148 |1148 |1531 |1531 |
| Wmax                  mm |0.14 |0.14 |0.17 |0.17 |0.21 |0.21 |0.28 |0.28 |
| Sp. For  Wmax = 0.20  mm | 208 | 208 | 260 | 260 | 312 | 312 | 416 | 416 |
 -------------------------------------------------------------------------------
    FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    7
 -------------------------------------------------------------------------------
|  ELEMENT  DESIGN  TO  BS8007  AND  BS8110              ELEMENT NO.    12   |
|-------------------------------------------------------------------------------|
|           A >                          e          f             g         |
|    l--------|---------k    Top___________________________________         |
|    |                  |       | o_______o_______o_______o_______o |        |
|    |        ! y       |       |         outer bars // to  x       | My     |
|    |        |         |       | o-------o-------o-------o-------o |        |
|    |       z+ --> x   |    Bot-----------------------------------         |
|    |                  |                     Section A-A                   |
|    |                  |                                                    |
|    i--------|---------j    Depth=200 mm  Width=1000 mm  Cover=20 mm       |
|           A >                                                              |
|-------------------------------------------------------------------------------|
| Ultimate Limit State |  12 mm Bars     |  16 mm Bars      |  20 mm Bars    |
| Max.Momnt. kNm/m  Lo |C/C  AS R. AS P.|C/C  AS R.  AS P.|C/C  AS R. AS P. |
|----------------------|---|------|-----|---|------|------|---|------|-----|
| Mx Top =   0.0     0 |200|  259 | 565 |200|  259 | 1065 |200|  259 |1572 |
| Mx Bot =  -0.5     1 |200|  259 | 565 |200|  259 | 1065 |200|  259 |1572 |
| My Top =  90.1     1 |***| 1621 | 565 |100| 1712 | 2011 |150| 1816 |2094 |
| My Bot =   0.0     0 |200|  259 | 565 |200|  259 | 1065 |200|  259 |1572 |
 -------------------------------------------------------------------------------
 -------------------------------------------------------------------------------
|  SERVICEABILITY    LIMIT    STATE                         ELEMENT NO.    12   |
|-------------------------------------------------------------------------------|
| Longitudinal  Moments  Mx   kNm/m  | Transverse    Moments    My   kNm/m  |
| Flexural      Crack    Width   mm  | Flexural      Crack      Width   mm  |
|------------------------------------|------------------------------------------|
| Top=  0.0  L.   0 Bot=  -0.5 L.   1| Top= 90.1  L.   1 Bot=   0.0 L.   0  |
|------------------------------------|------------------------------------------|
| 12 | 16 | 20 |  @  | 12 | 16 |  20 | 12  | 16 | 20 |  @  | 12 | 16 | 20  |
```

```
 |----|----|----|-----|----|----|-----|-----|----|----|-----|----|----|-----|
 |0.00|0.00|0.00|  e  |-.03|-.02|-.01 | 0.76|0.30|0.40|  e  |0.00|0.00|0.00 |
 |0.00|0.00|0.00|  f  |-.08|-.04|-.02 | 1.75|0.32|0.44|  f  |0.00|0.00|0.00 |
 |0.00|0.00|0.00|  g  |-.04|-.02|-.01 | 0.56|0.20|0.25|  g  |0.00|0.00|0.00 |
 |--------------------------------------------------------------------------|
 |  Thermal   Crack   Width   Calculations  Based  On   Wmax=Smax*R*T1*Alfa |
 |--------------------------------------------------------------------------|
 | Surface Type : Suspended    Constuction type : 1     Temp. Range = 30 C  |
 |--------------------------------------------------------------------------|
 | Surface  Zones & ROWcrit | 8 mm bars |10 mm Bars |12 mm Bars |16 mm Bars |
 | Top  : 100  mm  350  mm2 |-----------|-----------|-----------|-----------|
 | Bot. : 100  mm  350  mm2 | Top | Bot.| Top | Bot.| Top | Bot.| Top | Bot.|
 |--------------------------|-----|-----|-----|-----|-----|-----|-----|-----|
 | Smax                  mm | 765 | 765 | 957 | 957 |1148 |1148 |1531 |1531 |
 | Wmax                  mm |0.14 |0.14 |0.17 |0.17 |0.21 |0.21 |0.28 |0.28 |
 | Sp. For  Wmax = 0.20  mm | 208 | 208 | 260 | 260 | 312 | 312 | 416 | 416 |
  --------------------------------------------------------------------------
   ***************************END OF ELEMENT DESIGN**************************
    63. END CONCRETE DESIGN
    64. FINISH
      FINITE ELEMENT MODEL OF TANK STRUCTURE                   -- PAGE NO.    8
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44: 8 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

Related Links

- *TR.14.2 Element Mesh Generation* (on page 2229)
- *TR.11 Joint Coordinates Specification* (on page 2217)

EX. UK-11 Response Spectrum Analysis of a Frame

Dynamic analysis (Response Spectrum) is performed for a steel structure. Results of a static and dynamic analysis are combined. The combined results are then used for steel design.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-11 Response Spectrum Analysis of a Frame.STD` when you install the program.

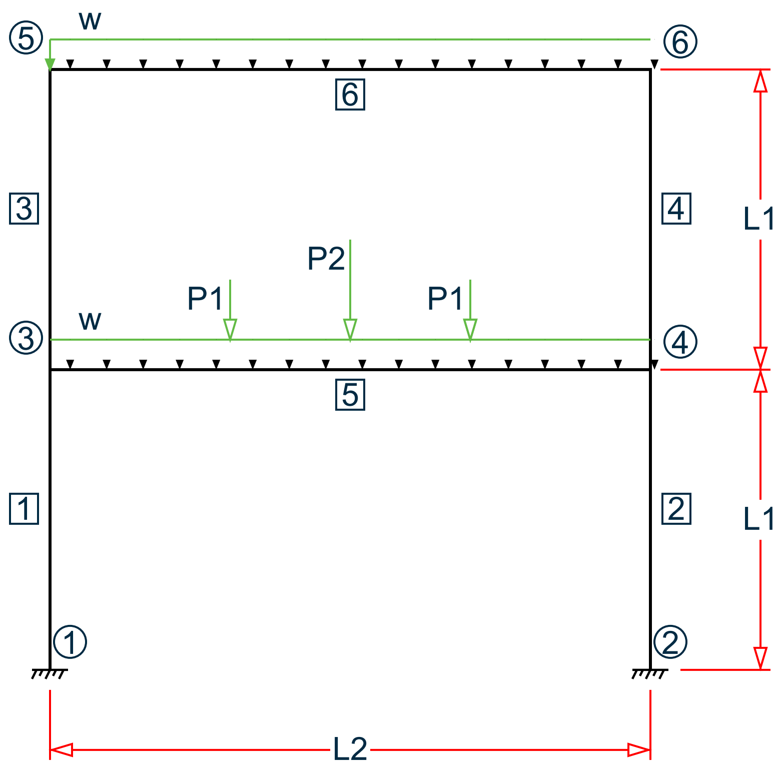


Figure 552: Example Problem No. 11

Where:

L1 = 3 m, L2 = 6 m
w = 22.5 kN/m
P1 = 25 kN, P2 = 37.5 kN

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
```

Every input has to start with the term STAAD. The term PLANE signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0
3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
```

Joint number followed by X, Y and Z coordinates are provided above. Since this is a plane structure, the Z coordinates are all the same, in this case, zeros.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTIES BRITISH
1 TO 4 TA ST UC254X254X73
5 TA ST UB305X165X54
6 TA ST UB203X133X30
```

Properties for all members are assigned from the British steel table. The word ST stands for standard single section.

```
SUPPORTS
1 2 FIXED
```

Fixed supports are specified at joints 1 and 2.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

Material constants such as E (modulus of elasticity), Poisson's ratio and density (DEN) are specified above. Length unit is changed from METER to MMS to facilitate the input.

```
CUT OFF MODE SHAPE 2
```

The number of mode shapes to be considered in dynamic analysis is set to 2. Without the above command, this will be set to the default. See TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344) .

```
* LOAD 1 WILL BE STATIC LOAD
UNIT METER
LOAD 1 DEAD AND LIVE LOADS
```

Load case 1 is initiated followed by a title. Prior to this, the length unit is changed to METER for specifying distributed member loads. A line starting with an asterisk (*) mark indicates a comment line.

```
SELFWEIGHT Y -1.0
```

The above command indicates that the selfweight of the structure acting in the global Y direction is part of this load case. The factor of -1.0 is meant to indicate that the load acts opposite to the positive direction of global Y, hence downwards.

```
MEMBER LOADS
5 CON GY -25.0 1.8
5 CON GY -37.5 3.0
5 CON GY -25.0 4.2
5 6 UNI Y -22.5
```

Load 1 contains member loads also. GY indicates that the load is in the global Y direction while Y indicates local Y direction. The word UNI stands for uniformly distributed load while CON stands for concentrated load. GY is followed by the value of the load and the distance at which it is applied.

```
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
```

The two lines which begin with the asterisk are comment lines which tell us the purpose of the next load case. Load case 2 is then initiated along with an optional title. This will be a dynamic load case. Permanent masses will be provided in the form of loads. These masses (in terms of loads) will be considered for the eigensolution. Internally, the program converts these loads to masses, hence it is best to specify them as absolute values (without a negative sign). Also, the direction (X, Y, Z etc.) of the loads will correspond to the dynamic degrees of freedom in which the masses are capable of vibrating. In a PLANE frame, only X and Y directions need to be considered. In a SPACE frame, masses (loads) should be provided in all three (X, Y and Z) directions if they are active along all three. The user has the freedom to restrict one or more directions.

```
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
```

The above commands indicate that the selfweight of the structure acting in the global X and Y directions with a factor of 1.0 is taken into consideration for the mass matrix.

```
MEMBER LOADS
5 CON GX 25.0 1.8
5 CON GY 25.0 1.8
5 CON GX 37.5 3.0
5 CON GY 37.5 3.0
5 CON GX 25.0 4.2
5 CON GY 25.0 4.2
```

The mass matrix will also consist of terms derived from the above member loads. GX and GY indicate that the load, and hence the resulting mass, is capable of vibration along the global X and Y directions. The word CON stands for concentrated load. Concentrated forces of 25, 37.5, and 25 kNs are located at 1.8m, 3.0m and 4.2m from the start of member 5.

```
SPECTRUM CQC EURO X 1.0 ACC DAMP 0.05
SOIL TYPE C ALPHA 0.15 Q 1.5
```

The `SPECTRUM` command specifies a Eurocode 8 2004 seismic response spectrum load. The modal responses will be combined using the CQC method. Here, the spectrum effect is in the global X direction with a factor of 1.0. EC8 2004 response spectra are always given in terms of acceleration (`ACC`). A damping ratio of 0.05 (5%) is used. The second line then gives the soil type along with the alpha ratio and behavior factor, Q.

```
LOAD COMBINATION 3
1 0.75 2 0.75
LOAD COMBINATION 4
1 0.75 2 -0.75
```

In a response spectrum analysis, the sign of the forces cannot be determined, and hence are absolute numbers. Consequently, to account for the fact that the force could be positive or negative, it is necessary to create 2 load combination cases. That is what is being done above. Load combination case no. 3 consists of the sum of the static load case (1) with the positive direction of the dynamic load case (2). Load combination case no. 4 consists of the sum of the static load case (1) with the negative direction of the dynamic load case (2). In both cases, the result is factored by 0.75.

```
PERFORM ANALYSIS PRINT MODE SHAPES
```

This command instructs the program to proceed with the analysis. The `PRINT` command instructs the program to print mode shape values.

```
    PRINT ANALYSIS RESULTS
```

Displacements, reactions and member forces are recorded in the output file using the above command.

```
    LOAD LIST 1 3 4
    PARAMETER
    CODE EN 1993-1-1:2005
    NA 1
    SELECT ALL
```

A steel design in the form of a member selection is performed based on the rules of the Eurocode 8 code using the UK national annex. Only the member forces resulting from load cases 1, 3 and 4 will be considered for these calculations.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE RESPONSE SPECTRUM ANALYSIS
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0
3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
MEMBER INCIDENCES
1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
5 3 4 ; 6 5 6
MEMBER PROPERTIES BRITISH
1 TO 4 TA ST UC254X254X73
5 TA ST UB305X165X54
6 TA ST UB203X133X30
SUPPORTS
1 2 FIXED
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
CUT OFF MODE SHAPE 2
*LOAD 1 WILL BE STATIC LOAD
UNIT METER
LOAD 1 DEAD AND LIVE LOADS
SELFWEIGHT Y -1.0
MEMBER LOADS
5 CON GY -25.0 1.8
5 CON GY -37.5 3.0
5 CON GY -25.0 4.2
```

```
5 6 UNI Y -22.5
* NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
* WITH MASSES PROVIDED IN TERMS OF LOAD.
LOAD 2 SEISMIC LOADING
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
MEMBER LOADS
5 CON GX 25.0 1.8
5 CON GY 25.0 1.8
5 CON GX 37.5 3.0
5 CON GY 37.5 3.0
5 CON GX 25.0 4.2
5 CON GY 25.0 4.2
SPECTRUM CQC EURO X 1.0 ACC DAMP 0.05
SOIL TYPE C ALPHA 0.15 Q 1.5
LOAD COMBINATION 3
1 0.75 2 0.75
LOAD COMBINATION 4
1 0.75 2 -0.75
PERFORM ANALYSIS PRINT MODE SHAPES
PRINT ANALYSIS RESULTS
LOAD LIST 1 3 4
PARAMETER
CODE EN 1993-1-1:2005
NA 1
SELECT ALL
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:10                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE RESPONSE SPECTRUM ANALYSIS
INPUT FILE: UK-11 Response Spectrum Analysis of a Frame.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 ; 2 6.0 0.0 0.0
     5. 3 0.0 3.0 0.0 ; 4 6.0 3.0 0.0
     6. 5 0.0 6.0 0.0 ; 6 6.0 6.0 0.0
     7. MEMBER INCIDENCES
     8. 1 1 3 ; 2 2 4 ; 3 3 5 ; 4 4 6
     9. 5 3 4 ; 6 5 6
    10. MEMBER PROPERTIES BRITISH
    11. 1 TO 4 TA ST UC254X254X73
    12. 5 TA ST UB305X165X54
    13. 6 TA ST UB203X133X30
    14. SUPPORTS
    15. 1 2 FIXED
```

```
   16. UNIT MMS
   17. DEFINE MATERIAL START
   18. ISOTROPIC STEEL
   19. E 210
   20. POISSON 0.3
   21. DENSITY 7.6977E-008
   22. ALPHA 6E-006
   23. DAMP 0.03
   24. TYPE STEEL
   25. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
   26. END DEFINE MATERIAL
   27. CONSTANTS
   28. MATERIAL STEEL ALL
   29. CUT OFF MODE SHAPE 2
   30. *LOAD 1 WILL BE STATIC LOAD
   31. UNIT METER
   32. LOAD 1 DEAD AND LIVE LOADS
   33. SELFWEIGHT Y -1.0
   34. MEMBER LOADS
   35. 5 CON GY -25.0 1.8
   36. 5 CON GY -37.5 3.0
   37. 5 CON GY -25.0 4.2
   38. 5 6 UNI Y -22.5
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    2
   39. * NEXT LOAD WILL BE RESPONSE SPECTRUM LOAD
   40. * WITH MASSES PROVIDED IN TERMS OF LOAD.
   41. LOAD 2 SEISMIC LOADING
   42. SELFWEIGHT X 1.0
   43. SELFWEIGHT Y 1.0
   44. MEMBER LOADS
   45. 5 CON GX 25.0 1.8
   46. 5 CON GY 25.0 1.8
   47. 5 CON GX 37.5 3.0
   48. 5 CON GY 37.5 3.0
   49. 5 CON GX 25.0 4.2
   50. 5 CON GY 25.0 4.2
   51. SPECTRUM CQC EURO X 1.0 ACC DAMP 0.05
   52. SOIL TYPE C ALPHA 0.15 Q 1.5
   53. LOAD COMBINATION 3
   54. 1 0.75 2 0.75
   55. LOAD COMBINATION 4
   56. 1 0.75 2 -0.75
   57. PERFORM ANALYSIS PRINT MODE SHAPES
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          6  NUMBER OF MEMBERS       6
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      12
  TOTAL LOAD COMBINATION  CASES =     2  SO FAR.
  ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED             =     2
NUMBER OF EXISTING MASSES IN THE MODEL =     8
```

```
 NUMBER OF MODES THAT WILL BE USED      =     2
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    3
               CALCULATED FREQUENCIES FOR LOAD CASE       2
       MODE            FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
         1                       5.178                  0.19312
         2                      19.435                  0.05145
      RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    4
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     1    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     1    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     1    0.58345   0.00218   0.00000  0.000E+00  0.000E+00 -3.862E-03
     4     1    0.58345  -0.00218   0.00000  0.000E+00  0.000E+00 -3.862E-03
     5     1    1.00000   0.00251   0.00000  0.000E+00  0.000E+00 -2.794E-03
     6     1    1.00000  -0.00251   0.00000  0.000E+00  0.000E+00 -2.794E-03
   MODE SHAPES
   -----------
  JOINT  MODE   X-TRANS   Y-TRANS   Z-TRANS    X-ROTAN    Y-ROTAN    Z-ROTAN
     1     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     2     2    0.00000   0.00000   0.00000  0.000E+00  0.000E+00  0.000E+00
     3     2   -0.07065   0.00282   0.00000  0.000E+00  0.000E+00 -2.798E-03
     4     2   -0.07065  -0.00282   0.00000  0.000E+00  0.000E+00 -2.798E-03
     5     2    1.00000   0.00400   0.00000  0.000E+00  0.000E+00 -9.739E-03
     6     2    1.00000  -0.00400   0.00000  0.000E+00  0.000E+00 -9.739E-03
  RESPONSE SPECTRUM LOAD     2
     RESPONSE LOAD CASE      2
            MODAL WEIGHT (MODAL MASS TIMES g) IN KNS          GENERALIZED
      MODE           X            Y             Z               WEIGHT
         1       9.711083E+01  9.577115E-18  0.000000E+00    3.624621E+01
         2       1.780698E+00  1.662500E-16  0.000000E+00    4.389139E+00
     CQC          MODAL COMBINATION METHOD USED.
     DYNAMIC WEIGHT X Y Z   9.889183E+01  9.889183E+01  0.000000E+00 KNS
     MISSING WEIGHT X Y Z  -3.075424E-04 -9.889183E+01  0.000000E+00 KNS
       MODAL WEIGHT X Y Z   9.889153E+01  1.758271E-16  0.000000E+00 KNS
      RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    5
           MODE                 ACCELERATION-G     DAMPING
           ----                 --------------     -------
              1                      0.22191       0.05000
              2                      0.15815       0.05000
 MODAL BASE ACTIONS
  MODAL BASE ACTIONS        FORCES IN KNS  LENGTH IN METE
  -----------------------------------------------------------
                                                            MOMENTS ARE ABOUT THE
ORIGIN
   MODE     PERIOD        FX          FY          FZ          MX          MY
MZ
      1      0.193       21.55        0.00        0.00        0.00        0.00
-69.15
      2      0.051        0.28       -0.00        0.00        0.00        0.00
0.42
 PARTICIPATION FACTORS
           MASS  PARTICIPATION FACTORS IN PERCENT        BASE SHEAR IN KNS
           --------------------------------------        ------------------
 MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z       X        Y        Z
   1   98.20  0.00  0.00   98.199    0.000    0.000     21.55     0.00     0.00
   2    1.80  0.00  0.00  100.000    0.000    0.000      0.28     0.00     0.00
```

```
                                                        ---------------------------
                                  TOTAL SRSS  SHEAR     21.55     0.00     0.00
                                  TOTAL 10PCT SHEAR     21.55     0.00     0.00
                                  TOTAL ABS   SHEAR     21.83     0.00     0.00
                                  TOTAL CQC   SHEAR     21.55     0.00     0.00
   ***WARNING: NO RIGID FLOOR DIAPHRAGM EXISTS FOR THE STRUCTURE.
               RESULTS ON EARTHQUAKE MODE IN USER-INTERFACE MAY BE APPROXIMATE.
   WARNING : NO WELL DEFINED FLOOR LEVEL EXISTS FOR "STAAD SPACE" MODEL.
             CALCULATION OF STOREY SHEAR DUE TO MISSING MASS OR TORSION IGNORED.
    58. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    6
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           4     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      2    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
           4     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
      3    1    -0.0042   -0.0283    0.0000    0.0000    0.0000   -0.0017
           2     0.1963    0.0007    0.0000    0.0000    0.0000    0.0005
           3     0.1441   -0.0207    0.0000    0.0000    0.0000   -0.0009
           4    -0.1504   -0.0218    0.0000    0.0000    0.0000   -0.0017
      4    1     0.0042   -0.0283    0.0000    0.0000    0.0000    0.0017
           2     0.1963    0.0007    0.0000    0.0000    0.0000    0.0005
           3     0.1504   -0.0207    0.0000    0.0000    0.0000    0.0017
           4    -0.1441   -0.0218    0.0000    0.0000    0.0000    0.0009
      5    1     0.0163   -0.0390    0.0000    0.0000    0.0000   -0.0017
           2     0.3366    0.0008    0.0000    0.0000    0.0000    0.0004
           3     0.2647   -0.0286    0.0000    0.0000    0.0000   -0.0010
           4    -0.2402   -0.0299    0.0000    0.0000    0.0000   -0.0015
      6    1    -0.0163   -0.0390    0.0000    0.0000    0.0000    0.0017
           2     0.3366    0.0008    0.0000    0.0000    0.0000    0.0004
           3     0.2402   -0.0286    0.0000    0.0000    0.0000    0.0015
           4    -0.2647   -0.0299    0.0000    0.0000    0.0000    0.0010
      RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    7
   SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1      23.47    185.52      0.00      0.00      0.00    -21.57
           2      10.78      4.78      0.00      0.00      0.00     20.25
           3      25.69    142.72      0.00      0.00      0.00     -1.00
           4       9.52    135.56      0.00      0.00      0.00    -31.37
      2    1     -23.47    185.52      0.00      0.00      0.00     21.57
           2      10.78      4.78      0.00      0.00      0.00     20.25
           3      -9.52    142.72      0.00      0.00      0.00     31.37
           4     -25.69    135.56      0.00      0.00      0.00      1.00
      RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.    8
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1    185.52    -23.47     0.00      0.00      0.00     -21.57
                 3   -183.37     23.47     0.00      0.00      0.00     -48.85
```

```
         2     1      4.78     10.78     0.00      0.00      0.00      20.25
               3      4.78     10.78     0.00      0.00      0.00      12.08
         3     1    142.72     -9.52     0.00      0.00      0.00      -1.00
               3   -141.11      9.52     0.00      0.00      0.00     -45.70
         4     1    135.56    -25.69     0.00      0.00      0.00     -31.37
               3   -133.94     25.69     0.00      0.00      0.00     -27.58
    2    1     2    185.52     23.47     0.00      0.00      0.00      21.57
               4   -183.37    -23.47     0.00      0.00      0.00      48.85
         2     2      4.78     10.78     0.00      0.00      0.00      20.25
               4      4.78     10.78     0.00      0.00      0.00      12.08
         3     2    142.72     25.69     0.00      0.00      0.00      31.37
               4   -141.11    -25.69     0.00      0.00      0.00      27.58
         4     2    135.56      9.52     0.00      0.00      0.00       1.00
               4   -133.94     -9.52     0.00      0.00      0.00      45.70
    3    1     3     70.53    -43.63     0.00      0.00      0.00     -65.87
               5    -68.38     43.63     0.00      0.00      0.00     -65.02
         2     3      0.74      0.74     0.00      0.00      0.00       0.44
               5      0.74      0.74     0.00      0.00      0.00       2.20
         3     3     53.45    -32.17     0.00      0.00      0.00     -49.07
               5    -51.84     32.17     0.00      0.00      0.00     -50.42
         4     3     52.35    -33.28     0.00      0.00      0.00     -49.73
               5    -50.74     33.28     0.00      0.00      0.00     -47.12
    4    1     4     70.53     43.63     0.00      0.00      0.00      65.87
               6    -68.38    -43.63     0.00      0.00      0.00      65.02
         2     4      0.74      0.74     0.00      0.00      0.00       0.44
               6      0.74      0.74     0.00      0.00      0.00       2.20
         3     4     53.45     33.28     0.00      0.00      0.00      49.73
               6    -51.84    -33.28     0.00      0.00      0.00      47.12
         4     4     52.35     32.17     0.00      0.00      0.00      49.07
               6    -50.74    -32.17     0.00      0.00      0.00      50.42
    5    1     3    -20.16    112.84     0.00      0.00      0.00     114.72
               4     20.16    112.84     0.00      0.00      0.00    -114.72
         2     3      0.00      4.01     0.00      0.00      0.00      12.02
               4      0.00      4.01     0.00      0.00      0.00      12.02
         3     3    -15.12     87.63     0.00      0.00      0.00      95.05
               4     15.12     81.62     0.00      0.00      0.00     -95.05
    RESPONSE SPECTRUM ANALYSIS                              -- PAGE NO.    9
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
         4     3    -15.12     81.62     0.00      0.00      0.00      77.03
               4     15.12     87.63     0.00      0.00      0.00     -77.03
    6    1     5     43.63     68.38     0.00      0.00      0.00      65.02
               6    -43.63     68.38     0.00      0.00      0.00     -65.02
         2     5      0.00      0.73     0.00      0.00      0.00       2.20
               6      0.00      0.73     0.00      0.00      0.00       2.20
         3     5     32.72     51.84     0.00      0.00      0.00      50.42
               6    -32.72     50.74     0.00      0.00      0.00     -50.42
         4     5     32.72     50.74     0.00      0.00      0.00      47.12
               6    -32.72     51.84     0.00      0.00      0.00     -47.12
 ************** END OF LATEST ANALYSIS RESULT **************
  59. LOAD LIST 1 3 4
  60. PARAMETER
  61. CODE EN 1993-1-1:2005
  62. NA 1
  63. SELECT ALL
STEEL DESIGN
```

```
                         STAADPRO MEMBER SELECTION - BS EN 1993-1-1:2005
                         ********************************************
                         NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.   10
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      1 ST  UC152X152X37 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.901        1
                     183.37 C         0.00         -48.85        3.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      2 ST  UC152X152X37 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.901        1
                     183.37 C         0.00          48.85        3.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      3 ST  UC152X152X44 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.851        1
                      70.53 C         0.00         -65.87        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      4 ST  UC152X152X44 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.851        1
                      70.53 C         0.00          65.87        0.00
     RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.   11
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      5 ST  UB305X165X54 (BRITISH SECTIONS)
                          PASS     EC-6.3.2 LTB       0.980        1
                      20.16 T         0.00         114.72        0.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
      6 ST  UB305X165X40 (BRITISH SECTIONS)
                          PASS     EC-6.3.3-662       0.972        1
                      43.63 C         0.00          65.02        0.00
  ************** END OF TABULATED RESULT OF DESIGN **************
   64. FINISH
  ***************************************************************************
  **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
              IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
              ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
              TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
              FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
              FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
              COMMANDS.
```

```
 ****************************************************************************
    RESPONSE SPECTRUM ANALYSIS                               -- PAGE NO.   12
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:44:10 ****
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

EX. UK-12 Moving Load Generation on a Bridge Deck

This example demonstrates generation of load cases for the type of loading known as a moving load. This type of loading occurs classically when the load-causing units move on the structure, as in the case of trucks on a bridge deck. The mobile loads are discretized into several individual immobile load cases at discrete positions. During this process, enormous number of load cases may be created resulting in plenty of output to be sorted. To avoid looking into a lot of output, the maximum force envelope is requested for a few specific members.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-12 Moving Load Generation on a Bridge Deck.STD` when you install the program.

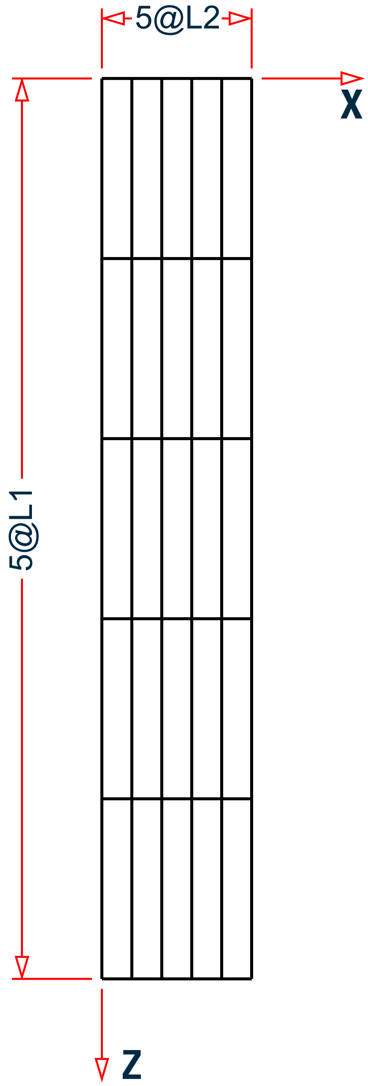


Figure 553: Example Problem No. 12

Where:

L1 = 9 m
L2 = 1.5 m

Actual input is shown in bold lettering followed by explanation.

```
STAAD FLOOR A SIMPLE BRIDGE DECK
```

Every input has to start with the term `STAAD`. The word `FLOOR` signifies that the structure is a floor structure and the geometry is defined through X and Z axis.

```
UNITS METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 6 7.5 0 0
R 5 0 0 9.0
```

Joint number followed by X, Y, and Z coordinates are provided above. Since this is a floor structure, the Y coordinates are all the same (in this case, zero). The first line generates joints 1 through 6. With the repeat (R) command, the coordinates of the next 30 joints are generated by repeating the pattern of the coordinates of the first 6 joints 5 times with X, Y and Z increments of 0,0, and 9 respectively.

```
MEMBER INCIDENCES
1 1 7 6
7 1 2 11
R A 4 11 6
56 31 32 60
```

Defines the members by the joints to which they are connected. The fourth number indicates the final member number up to which they will be generated. Repeat all (abbreviated as `R A`) will create members by repeating the member incidence pattern of the previous 11 members. The number of repetitions to be carried out is provided after the R A command and the member increment and joint increment are defined as 11 and 6 respectively. The fifth line of input defines the member incidences for members 56 to 60.

```
MEMBER PROPERTIES BRITISH
1 TO 60 TA ST UB305X165X40
```

Properties for all members are assigned from the British steel table. The word ST stands for standard single section.

```
SUPPORTS
1 TO 6 31 TO 36 PINNED
```

Pinned supports are specified at the above joints. A pinned support is one which can resist only translational forces.

```
UNITS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The units of length are changed from METER to MMS.

```
UNIT METER KNS
DEFINE MOVING LOAD
TYPE 1 LOAD 90.0 90.0 45.0
DISTANCE 3.0 1.5 WIDTH 3.0
```

The characteristics of the vehicle are defined above in METER and KNS units. The above lines represent the first out of two sets of data required in moving load generation. The type number (1) is a label for identification of the load-causing unit, such as a truck. Three axles (90 90 45) are specified with the LOAD command. The spacing between the axles in the direction of movement (longitudinal direction) is specified after the DISTANCE command. WIDTH is the spacing in the transverse direction, that is, it is the distance between the 2 prongs of an axle of the truck.

```
    LOAD 1
```

Load case 1 is initiated.

```
    SELF Y -1.0
```

Selfweight of the structure acting in the negative (due to the factor -1.0) global Y direction is the only component of load case 1.

```
    LOAD GENERATION 10
    TYPE 1 2.25 0. 0. ZI 3.0
```

This constitutes the second of the two sets of data required for moving load generation. 10 load cases are generated using the Type 1 vehicle whose characteristics were described earlier. For the first of these load cases, the X, Y and Z location of the reference load (see TR.31.1 Definition of Moving Load System (on page 2346)) have been specified after the command TYPE 1. The Z Increment of 3.0m denotes that the vehicle moves along the Z direction and the individual positions which are 3.0m apart will be used to generate the remaining 9 load cases.

When defining a moving load in STAAD.Pro, the reference wheel is on the last axle. The first load case which is generated will be the one for which the first axle is just about to enter the bridge. The last load case should be the one for which the last axle is just about to exit the bridge. Thus, the total distance travelled by the reference load will be the length of the vehicle (distance from first axle to last axle) plus the span of the bridge. In this problem, that comes to

$$(3.0+1.5) + 45 = 49.5 \text{ m.}$$

This example uses 3.0 m increments and generates 10 load cases.

However, if you want the vehicle to move forward in 4.5 m increments (each 4.5 m increment will create a discrete position of the truck on the bridge), it would required (49.5/4.5)+1 = 12 cases to be generated.

```
    PERFORM ANALYSIS PRINT LOAD
```

The above command instructs the program to proceed with the analysis and print the values and positions of all the generated load cases.

```
    PRINT MAXFORCE ENVELOP LIST 3 41 42
```

A maximum force envelope consisting of the highest forces for each degree of freedom on the listed members will be written into the output file.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD FLOOR A SIMPLE BRIDGE DECK
UNITS METER KNS
JOINT COORDINATES
1 0.0 0.0 0.0 6 7.5 0.0 0.0
R 5 0.0 0.0 9.0
MEMBER INCIDENCES
1 1 7 6
7 1 2 11
R A 4 11 6
56 31 32 60
MEMBER PROPERTIES BRITISH
```

```
1 TO 60 TA ST UB305X165X40
SUPPORTS
1 TO 6 31 TO 36 PINNED
UNITS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER KNS
DEFINE MOVING LOAD
TYPE 1 LOAD 90.0 90.0 45.0 DISTANCE 3.0 1.5 WIDTH 3.0
LOAD 1
SELF Y -1.0
LOAD GENERATION 10
TYPE 1 2.25 0. 0. ZI 3.0
PERFORM ANALYSIS PRINT LOAD
PRINT MAXFORCE ENVELOP LIST 3 41 42
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:12                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD FLOOR A SIMPLE BRIDGE DECK
INPUT FILE: UK-12 Moving Load Generation on a Bridge Deck.STD
     2. UNITS METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 0.0 6 7.5 0.0 0.0
     5. R 5 0.0 0.0 9.0
     6. MEMBER INCIDENCES
     7. 1 1 7 6
     8. 7 1 2 11
     9. R A 4 11 6
    10. 56 31 32 60
    11. MEMBER PROPERTIES BRITISH
    12. 1 TO 60 TA ST UB305X165X40
    13. SUPPORTS
    14. 1 TO 6 31 TO 36 PINNED
    15. UNITS MMS
    16. DEFINE MATERIAL START
```

```
  17. ISOTROPIC STEEL
  18. E 210
  19. POISSON 0.3
  20. DENSITY 7.6977E-008
  21. ALPHA 6E-006
  22. DAMP 0.03
  23. TYPE STEEL
  24. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
  25. END DEFINE MATERIAL
  26. CONSTANTS
  27. MATERIAL STEEL ALL
  28. UNIT METER KNS
  29. DEFINE MOVING LOAD
  30. TYPE 1 LOAD 90.0 90.0 45.0 DISTANCE 3.0 1.5 WIDTH 3.0
  31. LOAD 1
  32. SELF Y -1.0
  33. LOAD GENERATION 10
  34. TYPE 1 2.25 0. 0. ZI 3.0
  35. PERFORM ANALYSIS PRINT LOAD
    A SIMPLE BRIDGE DECK                                     -- PAGE NO.    2
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS         36  NUMBER OF MEMBERS      60
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     12
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =    11, TOTAL DEGREES OF FREEDOM =      96
 TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    A SIMPLE BRIDGE DECK                                     -- PAGE NO.    3
 LOADING     1
 -----------
     SELFWEIGHT  Y   -1.000
    ACTUAL WEIGHT OF THE STRUCTURE =     124.391 KNS
 LOADING     2
 -----------
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    8                           -90.0000 GY   0.75
   10                           -90.0000 GY   0.75
    3                           -45.0000 GY   3.00
    2                           -45.0000 GY   3.00
    5                           -45.0000 GY   3.00
    4                           -45.0000 GY   3.00
    3                           -22.5000 GY   4.50
    2                           -22.5000 GY   4.50
    5                           -22.5000 GY   4.50
    4                           -22.5000 GY   4.50
 LOADING     3
 -----------
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    3                           -45.0000 GY   3.00
    2                           -45.0000 GY   3.00
    5                           -45.0000 GY   3.00
    4                           -45.0000 GY   3.00
    3                           -45.0000 GY   6.00
    2                           -45.0000 GY   6.00
```

```
    5                          -45.0000 GY   6.00
    4                          -45.0000 GY   6.00
    3                          -22.5000 GY   7.50
    2                          -22.5000 GY   7.50
    5                          -22.5000 GY   7.50
    4                          -22.5000 GY   7.50
 LOADING     4
 -----------
    A SIMPLE BRIDGE DECK                                  -- PAGE NO.    4
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    3                          -45.0000 GY   6.00
    2                          -45.0000 GY   6.00
    5                          -45.0000 GY   6.00
    4                          -45.0000 GY   6.00
   19                          -90.0000 GY   0.75
   21                          -90.0000 GY   0.75
   14                          -22.5000 GY   1.50
   13                          -22.5000 GY   1.50
   16                          -22.5000 GY   1.50
   15                          -22.5000 GY   1.50
 LOADING     5
 -----------
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
   19                          -90.0000 GY   0.75
   21                          -90.0000 GY   0.75
   14                          -45.0000 GY   3.00
   13                          -45.0000 GY   3.00
   16                          -45.0000 GY   3.00
   15                          -45.0000 GY   3.00
   14                          -22.5000 GY   4.50
   13                          -22.5000 GY   4.50
   16                          -22.5000 GY   4.50
   15                          -22.5000 GY   4.50
 LOADING     6
 -----------
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
   14                          -45.0000 GY   3.00
   13                          -45.0000 GY   3.00
   16                          -45.0000 GY   3.00
   15                          -45.0000 GY   3.00
   14                          -45.0000 GY   6.00
   13                          -45.0000 GY   6.00
   16                          -45.0000 GY   6.00
   15                          -45.0000 GY   6.00
   14                          -22.5000 GY   7.50
   13                          -22.5000 GY   7.50
    A SIMPLE BRIDGE DECK                                  -- PAGE NO.    5
   16                          -22.5000 GY   7.50
   15                          -22.5000 GY   7.50
 LOADING     7
 -----------
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
   14                          -45.0000 GY   6.00
   13                          -45.0000 GY   6.00
```

```
  16                           -45.0000 GY   6.00
  15                           -45.0000 GY   6.00
  30                           -90.0000 GY   0.75
  32                           -90.0000 GY   0.75
  25                           -22.5000 GY   1.50
  24                           -22.5000 GY   1.50
  27                           -22.5000 GY   1.50
  26                           -22.5000 GY   1.50
LOADING     8
-----------
MEMBER LOAD - UNIT KNS  METE
MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
  30                           -90.0000 GY   0.75
  32                           -90.0000 GY   0.75
  25                           -45.0000 GY   3.00
  24                           -45.0000 GY   3.00
  27                           -45.0000 GY   3.00
  26                           -45.0000 GY   3.00
  25                           -22.5000 GY   4.50
  24                           -22.5000 GY   4.50
  27                           -22.5000 GY   4.50
  26                           -22.5000 GY   4.50
LOADING     9
-----------
MEMBER LOAD - UNIT KNS  METE
MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
  25                           -45.0000 GY   3.00
  24                           -45.0000 GY   3.00
  27                           -45.0000 GY   3.00
  26                           -45.0000 GY   3.00
   A SIMPLE BRIDGE DECK                                       -- PAGE NO.    6
  25                           -45.0000 GY   6.00
  24                           -45.0000 GY   6.00
  27                           -45.0000 GY   6.00
  26                           -45.0000 GY   6.00
  25                           -22.5000 GY   7.50
  24                           -22.5000 GY   7.50
  27                           -22.5000 GY   7.50
  26                           -22.5000 GY   7.50
LOADING    10
-----------
MEMBER LOAD - UNIT KNS  METE
MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
  25                           -45.0000 GY   6.00
  24                           -45.0000 GY   6.00
  27                           -45.0000 GY   6.00
  26                           -45.0000 GY   6.00
  41                           -90.0000 GY   0.75
  43                           -90.0000 GY   0.75
  36                           -22.5000 GY   1.50
  35                           -22.5000 GY   1.50
  38                           -22.5000 GY   1.50
  37                           -22.5000 GY   1.50
LOADING    11
-----------
MEMBER LOAD - UNIT KNS  METE
MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
  41                           -90.0000 GY   0.75
```

```
   43                           -90.0000 GY   0.75
   36                           -45.0000 GY   3.00
   35                           -45.0000 GY   3.00
   38                           -45.0000 GY   3.00
   37                           -45.0000 GY   3.00
   36                           -22.5000 GY   4.50
   35                           -22.5000 GY   4.50
   38                           -22.5000 GY   4.50
   37                           -22.5000 GY   4.50
 ************ END OF DATA FROM INTERNAL STORAGE ************
  36. PRINT MAXFORCE ENVELOP LIST 3 41 42
MAXFORCE ENVELOP  LIST     3
    A SIMPLE BRIDGE DECK                                    -- PAGE NO.    7
       MEMBER FORCE ENVELOPE
       ---------------------
       ALL UNITS ARE KNS  METE
 MAX AND MIN FORCE VALUES AMONGST ALL SECTION LOCATIONS
MEMB          FY/    DIST  LD        MZ/    DIST  LD
              FZ     DIST  LD        MY     DIST  LD        FX     DIST  LD
   3 MAX     81.16   0.00    3       0.03   0.00    4
              0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
     MIN    -31.34   9.00    3    -504.76   9.00    5
              0.00   9.00   11       0.00   9.00   11      0.00     9.00   11
  41 MAX     73.48   0.00   10       9.18   1.50    5
              0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
     MIN    -18.36   1.50   11    -147.24   0.75   10
              0.00   1.50   11       0.00   1.50   11      0.00     1.50   11
  42 MAX      0.30   0.00    1       9.17   0.00    5
              0.00   0.00    1       0.00   0.00    1      0.00     0.00    1
     MIN     -0.30   1.50    1    -134.84   1.50   10
              0.00   1.50   11       0.00   1.50   11      0.00     1.50   11
 ********** END OF FORCE ENVELOPE FROM INTERNAL STORAGE **********
  37. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:44:13 ****
    A SIMPLE BRIDGE DECK                                    -- PAGE NO.    8
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

Related Links

- *(Moving) Load Generation Type dialog* (on page 2832)
- *TR.32.12.1 Generation of Moving Loads* (on page 2596)
- *M. To add vehicles to the load generation* (on page 874)
- *TR.31.1 Definition of Moving Load System* (on page 2346)
- *V. Moving Load Generator* (on page 3597)

EX. UK-13 Section Displacements for a Frame

Calculation of displacements at intermediate points of members of a plane frame is demonstrated in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-13 Section Displacements for a Frame.STD` when you install the program.

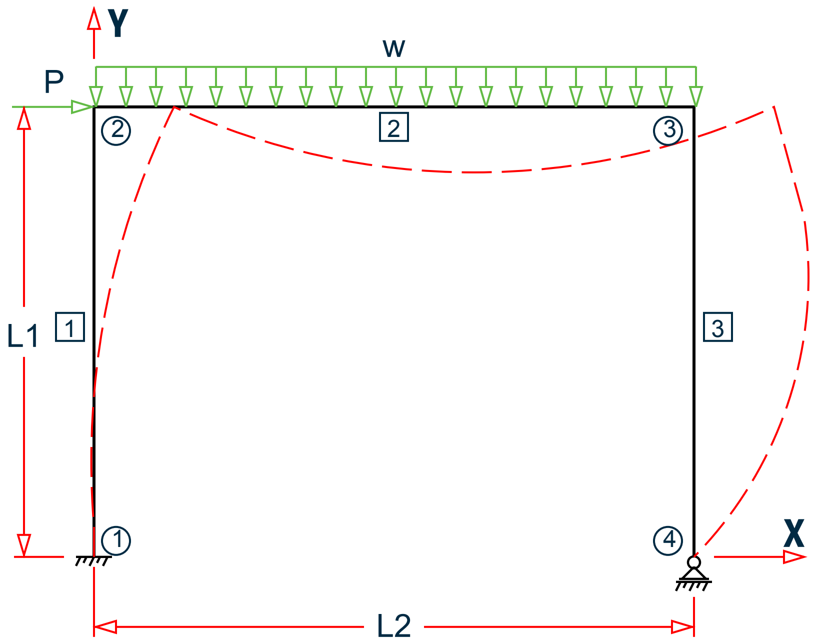


Figure 554: Example Problem No. 13

Where:

L1 = 4.5 m, L2 = 6.0 m
P = 25 kN
w = 45 kN/m

The dashed line represents the deflected shape of the structure. The shape is generated on the basis of displacements at the ends plus several intermediate points of the members.

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE TEST FOR SECTION DISPLACEMENT
```

Every input has to start with the term `STAAD`. The term `PLANE` signifies that the structure is a plane frame structure and the geometry is defined through X and Y axes.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 ; 2 0. 4.5 ; 3 6 4.5 ; 4 6 0.
```

Joint number followed by X and Y coordinates are provided above. Since this is a plane structure, the Z coordinates need not be provided.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
```

Defines the members by the joints to which they are connected.

```
MEMBER PROPERTY BRITISH
1 3 TABLE ST UC203X203X46
2 TABLE ST UB305X165X40
```

Member properties are specified from the British steel table. The word `ST` stands for standard single section.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

The length unit is changed from METER to MMS. The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 FIXED ; 4 PINNED
```

A fixed support is specified at Joint 1 and a pinned support at Joint 4.

```
UNIT METER
LOADING 1 DEAD + LIVE + WIND
```

Load case 1 is initiated followed by an optional title.

```
JOINT LOAD
2 FX 25.0
```

Load 1 contains a joint load of 25KN at node 2. `FX` indicates that the load is a force in the global X direction.

```
MEMBER LOAD
2 UNI GY -45.0
```

Load 1 contains a member load also. `GY` indicates that the load is in the global Y direction. The word `UNI` stands for uniformly distributed load.

```
PERFORM ANALYSIS
```

This command instructs the program to proceed with the analysis.

```
PRINT MEMBER FORCES
```

The above `PRINT` command is self-explanatory.

```
*
* FOLLOWING PRINT COMMAND WILL PRINT
```

```
    * DISPLACEMENTS OF THE MEMBERS
    * CONSIDERING EVERY TWELFTH INTERMEDIATE
    * POINT (THAT IS TOTAL OF 13 POINTS). THESE
    * DISPLACEMENTS ARE MEASURED IN GLOBAL X
    * Y Z COORDINATE SYSTEM AND THE VALUES
    * ARE FROM ORIGINAL COORDINATES (UNDEFLECTED
    * POSITION) OF CORRESPONDING TWELFTH
    * POINTS.
    *
    * MAX LOCAL DISPLACEMENT IS ALSO PRINTED.
    * THE LOCATION OF MAXIMUM INTERMEDIATE
    * DISPLACEMENT IS DETERMINED. THIS VALUE IS
    * MEASURED FROM ABOVE LOCATION TO THE
    * STRAIGHT LINE JOINING START AND END
    * JOINTS OF THE DEFLECTED MEMBER.
    *
    PRINT SECTION DISPLACEMENT
```

Above PRINT command is explained in the comment lines above.

```
    FINISH
```

This command terminates the STAAD run.

Input File

```
STAAD PLANE TEST FOR SECTION DISPLACEMENT
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 ; 2 0.0 4.5 ; 3 6.0 4.5 ; 4 6.0 0.0
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4
MEMBER PROPERTY BRITISH
1 3 TABLE ST UC203X203X46
2 TABLE ST UB305X165X40
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210.0
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 FIXED ; 4 PINNED
UNIT METER
LOADING 1 DEAD + LIVE + WIND
JOINT LOAD
2 FX 25.0
MEMBER LOAD
2 UNI GY -45.0
PERFORM ANALYSIS
PRINT MEMBER FORCES
*
```

```
* FOLLOWING PRINT COMMAND WILL PRINT
* DISPLACEMENTS OF THE MEMBERS CONSIDERING EVERY
* TWELFTH INTERMEDIATE POINT (THAT IS TOTAL OF 13 POINTS).
* THESE DISPLACEMENTS ARE MEASURED IN GLOBAL X Y Z
* COORDINATE SYSTEM AND THE VALUES ARE FROM
* ORIGINAL COORDINATES (UNDEFLECTED POSITION)
* OF THE CORRESPONDING TWELFTH POINTS.
*
* MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE
* LOCATION OF MAXIMUM INTERMEDIATE DISPLACEMENT
* IS DETERMINED. THIS VALUE IS MEASURED FROM ABOVE
* LOCATION TO THE STRAIGHT LINE JOINING START AND
* END JOINTS OF THE DEFLECTED MEMBER.
*
PRINT SECTION DISPLACEMENT
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:15                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE TEST FOR SECTION DISPLACEMENT
INPUT FILE: UK-13 Section Displacements for a Frame.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 ; 2 0.0 4.5 ; 3 6.0 4.5 ; 4 6.0 0.0
     5. MEMBER INCIDENCE
     6. 1 1 2 ; 2 2 3 ; 3 3 4
     7. MEMBER PROPERTY BRITISH
     8. 1 3 TABLE ST UC203X203X46
     9. 2 TABLE ST UB305X165X40
    10. UNIT MMS
    11. DEFINE MATERIAL START
    12. ISOTROPIC STEEL
    13. E 210.0
    14. POISSON 0.3
    15. DENSITY 7.6977E-008
    16. ALPHA 6E-006
    17. DAMP 0.03
    18. TYPE STEEL
    19. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
    20. END DEFINE MATERIAL
    21. CONSTANTS
    22. MATERIAL STEEL ALL
    23. SUPPORT
    24. 1 FIXED ; 4 PINNED
    25. UNIT METER
    26. LOADING 1 DEAD + LIVE + WIND
```

```
  27. JOINT LOAD
  28. 2 FX 25.0
  29. MEMBER LOAD
  30. 2 UNI GY -45.0
  31. PERFORM ANALYSIS
    TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    2
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =       7
 TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  32. PRINT MEMBER FORCES
MEMBER   FORCES
    TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    3
 MEMBER END FORCES    STRUCTURE TYPE = PLANE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1    121.84      1.65     0.00      0.00      0.00      33.54
               2   -121.84     -1.65     0.00      0.00      0.00     -26.10
    2    1     2     23.35    121.84     0.00      0.00      0.00      26.10
               3    -23.35    148.16     0.00      0.00      0.00    -105.06
    3    1     3    148.16     23.35     0.00      0.00      0.00     105.06
               4   -148.16    -23.35     0.00      0.00      0.00      -0.00
 ************** END OF LATEST ANALYSIS RESULT **************
  33. *
  34. * FOLLOWING PRINT COMMAND WILL PRINT
  35. * DISPLACEMENTS OF THE MEMBERS CONSIDERING EVERY
  36. * TWELFTH INTERMEDIATE POINT (THAT IS TOTAL OF 13 POINTS).
  37. * THESE DISPLACEMENTS ARE MEASURED IN GLOBAL X Y Z
  38. * COORDINATE SYSTEM AND THE VALUES ARE FROM
  39. * ORIGINAL COORDINATES (UNDEFLECTED POSITION)
  40. * OF THE CORRESPONDING TWELFTH POINTS.
  41. *
  42. * MAX LOCAL DISPLACEMENT IS ALSO PRINTED. THE
  43. * LOCATION OF MAXIMUM INTERMEDIATE DISPLACEMENT
  44. * IS DETERMINED. THIS VALUE IS MEASURED FROM ABOVE
  45. * LOCATION TO THE STRAIGHT LINE JOINING START AND
  46. * END JOINTS OF THE DEFLECTED MEMBER.
  47. *
  48. PRINT SECTION DISPLACEMENT
SECTION  DISPLACE
    TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    4
   MEMBER SECTION DISPLACEMENTS
   ----------------------------
   UNITS ARE - CM
   MEMB  LOAD     GLOBAL X,Y,Z DISPL FROM START TO END JOINTS AT 1/12TH PTS
     1     1       0.0000    0.0000    0.0000    0.0249   -0.0037    0.0000
                   0.0981   -0.0074    0.0000    0.2187   -0.0111    0.0000
                   0.3856   -0.0148    0.0000    0.5981   -0.0185    0.0000
                   0.8552   -0.0222    0.0000    1.1560   -0.0260    0.0000
                   1.4996   -0.0297    0.0000    1.8850   -0.0334    0.0000
                   2.3115   -0.0371    0.0000    2.7780   -0.0408    0.0000
                   3.2836   -0.0445    0.0000
```

```
     MAX LOCAL  DISP =    0.78662   AT     225.00  LOAD    1   L/DISP=    572
       2     1       3.2836   -0.0445    0.0000    3.2825   -0.7367    0.0000
                     3.2814   -1.3907    0.0000    3.2804   -1.9447    0.0000
                     3.2793   -2.3529    0.0000    3.2782   -2.5850    0.0000
                     3.2771   -2.6266    0.0000    3.2760   -2.4791    0.0000
                     3.2749   -2.1595    0.0000    3.2739   -1.7007    0.0000
                     3.2728   -1.1514    0.0000    3.2717   -0.5758    0.0000
                     3.2706   -0.0541    0.0000
     MAX LOCAL  DISP =    2.57728   AT     300.00  LOAD    1   L/DISP=    232
       3     1       3.2706   -0.0541    0.0000    3.5390   -0.0496    0.0000
                     3.6663   -0.0451    0.0000    3.6653   -0.0406    0.0000
                     3.5488   -0.0361    0.0000    3.3297   -0.0316    0.0000
                     3.0208   -0.0271    0.0000    2.6349   -0.0225    0.0000
                     2.1849   -0.0180    0.0000    1.6836   -0.0135    0.0000
                     1.1438   -0.0090    0.0000    0.5783   -0.0045    0.0000
                     0.0000    0.0000    0.0000
     MAX LOCAL  DISP =    1.42182   AT     187.50  LOAD    1   L/DISP=    316
         ************ END OF SECT DISPL RESULTS ***********
    49. FINISH
      TEST FOR SECTION DISPLACEMENT                            -- PAGE NO.    5
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44:16 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. UK-14 P-Delta Analysis of a Frame Under Seismic Loads

A space frame is analyzed for seismic loads. The seismic loads are generated using the procedures of the building code. A P-Delta analysis is performed to obtain the secondary effects of the lateral and vertical loads acting simultaneously.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-14 P-Delta Analysis of a Frame Under Seismic Loads.STD` when you install the program.

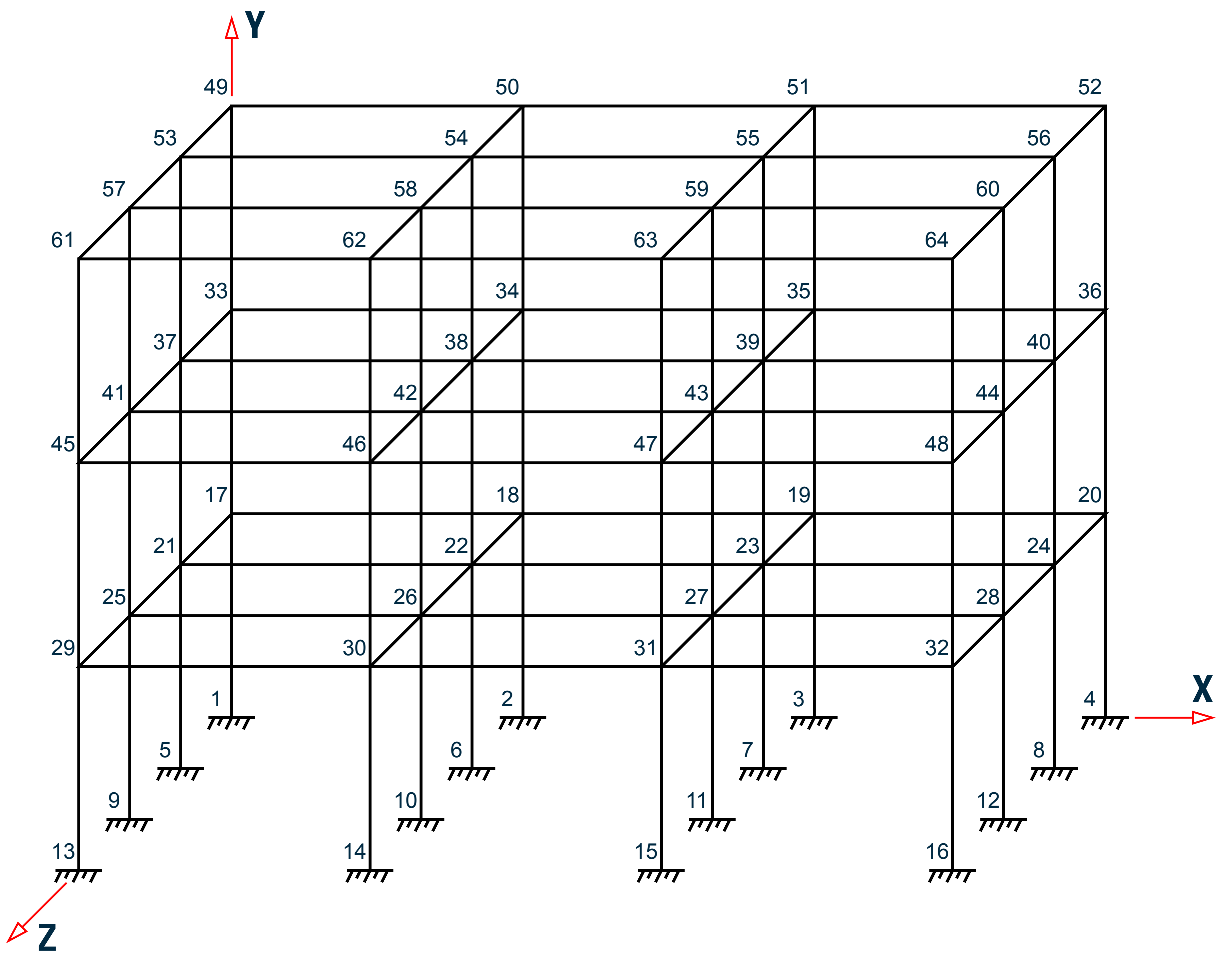


Figure 555: Example Problem No. 14

```
STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame.

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 4 10.5 0 0
REPEAT 3 0 0 3.5
REPEAT ALL 3 0 3.5 0
```

The X, Y and Z coordinates of the joints are specified here. First, coordinates of joints 1 through 4 are generated by taking advantage of the fact that they are equally spaced. Then, this pattern is REPEATed 3 times with a Z increment of 3.5 m for each repetition to generate joints 5 to 16. The REPEAT ALL command will then repeat 3 times, the pattern of joints 1 to 16 to generate joints 17 to 64.

```
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
107 25 26 109
```

```
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
```

Defines the members by the joints to which they are connected. Following the specification of incidences for members 101 to 112, the REPEAT ALL command is used to repeat the pattern and generate incidences for members 113 through 136. A similar logic is used in specification of incidences of members 201 through 212 and generation of incidences for members 213 to 236. Finally, members incidences of columns 301 to 348 are specified.

```
MEMBER PROPERTIES BRITISH
101 TO 136 201 TO 236 PRIS YD 0.40 ZD 0.30
301 TO 348 TA ST UB457X152X52
```

The beam members have prismatic member property specification (YD & ZD) while the columns (members 301 to 348) have their properties called from the built-in British steel table.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

Tip: You may see these values with the help of the command `PRINT MATERIAL PROPERTIES` following the preceding commands.

```
SUPPORT
1 TO 16 FIXED
```

Indicates the joints where the supports are located as well as the type of support restraints.

```
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7.0
49 TO 64 WEIGHT 3.5
```

There are two stages in a static seismic load. The first stage is to define the code-specified load parameters along witht he vertical loads (weights) from whic the base shear will be calculated. The vertical loads may be specified in the form of selfweight, joint weights and/or member weights. Member weights are not shown in this example. It is important to note that these vertical loads are used purely in the determination of the horizontal base shear only. In other words, the structure is not analyzed for these vertical loads.

```
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -7.0
49 TO 64 FY -3.5
```

This is the second stage in which the static seimsic load is applied with the help of a load case number, corresponding direction (X in the above case) and a factor by which the generated horizontal loads should be multiplied. Along with the seismic lateral load, deadweight is also added to the same load case. Since we will be doing second-order (PDELTA) analysis, it is important that we include horizontal and vertical loads in the same load case.

```
LOAD 2
UBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY –7.0
49 TO 64 FY –3.5
```

In load case 2, the static seimsic load is being applied in the Z direction. Vertical loads are part of this case, also.

```
PDELTA ANALYSIS PRINT LOAD DATA
```

We are requesting a second-order analysis by specifying the command `PDELTA ANALYSIS`. `PRINT LOAD DATA` is used to obtain a report of all the applied and generated loadings.

```
PRINT SUPPORT REACTIONS
FINISH
```

The above commands are self-explanatory.

Input File

```
STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 4 10.5 0 0
REPEAT 3 0 0 3.5
REPEAT ALL 3 0 3.5 0
MEMBER INCIDENCES
* beams in x direction
101 17 18 103
104 21 22 106
107 25 26 109
```

```
110 29 30 112
REPEAT ALL 2 12 16
* beams in z direction
201 17 21 204
205 21 25 208
209 25 29 212
REPEAT ALL 2 12 16
* columns
301 1 17 348
MEMBER PROPERTIES BRITISH
101 TO 136 201 TO 236 PRIS YD 0.40 ZD 0.30
301 TO 348 TA ST UB457X152X52
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 301 TO 348
MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
UNIT METER
SUPPORT
1 TO 16 FIXED
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7.0
49 TO 64 WEIGHT 3.5
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOAD
17 TO 48 FY -7.0
49 TO 64 FY -3.5
LOAD 2
UBC LOAD Z 0.75
SELFWEIGHT Y -1.0
JOINT LOAD
17 TO 48 FY -7.0
49 TO 64 FY -3.5
PDELTA ANALYSIS PRINT LOAD DATA
```

```
PRINT SUPPORT REACTIONS
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:18                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE PROBLEM FOR UBC LOAD
INPUT FILE: UK-14 P-Delta Analysis of a Frame Under Seismic Loads.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0 0 0 4 10.5 0 0
     5. REPEAT 3 0 0 3.5
     6. REPEAT ALL 3 0 3.5 0
     7. MEMBER INCIDENCES
     8. * BEAMS IN X DIRECTION
     9. 101 17 18 103
    10. 104 21 22 106
    11. 107 25 26 109
    12. 110 29 30 112
    13. REPEAT ALL 2 12 16
    14. * BEAMS IN Z DIRECTION
    15. 201 17 21 204
    16. 205 21 25 208
    17. 209 25 29 212
    18. REPEAT ALL 2 12 16
    19. * COLUMNS
    20. 301 1 17 348
    21. MEMBER PROPERTIES BRITISH
    22. 101 TO 136 201 TO 236 PRIS YD 0.40 ZD 0.30
    23. 301 TO 348 TA ST UB457X152X52
    24. UNIT MMS
    25. DEFINE MATERIAL START
    26. ISOTROPIC STEEL
    27. E 210
    28. POISSON 0.3
    29. DENSITY 7.6977E-008
    30. ALPHA 6E-006
    31. DAMP 0.03
    32. TYPE STEEL
    33. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
    34. ISOTROPIC CONCRETE
    35. E 21.0
    36. POISSON 0.17
    37. DENSITY 2.36158E-008
    38. ALPHA 5E-006
      EXAMPLE PROBLEM FOR UBC LOAD                              -- PAGE NO.    2
    39. DAMP 0.05
```

```
 40. G 9.25
 41. TYPE CONCRETE
 42. STRENGTH FCU 0.0275
 43. END DEFINE MATERIAL
 44. CONSTANTS
 45. MATERIAL STEEL MEMB 301 TO 348
 46. MATERIAL CONCRETE MEMB 101 TO 136 201 TO 236
 47. UNIT METER
 48. SUPPORT
 49. 1 TO 16 FIXED
 50. DEFINE UBC LOAD
 51. ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
 52. SELFWEIGHT
 53. JOINT WEIGHT
 54. 17 TO 48 WEIGHT 7.0
 55. 49 TO 64 WEIGHT 3.5
 56. LOAD 1
 57. UBC LOAD X 0.75
 58. SELFWEIGHT Y -1.0
 59. JOINT LOAD
 60. 17 TO 48 FY -7.0
 61. 49 TO 64 FY -3.5
 62. LOAD 2
 63. UBC LOAD Z 0.75
 64. SELFWEIGHT Y -1.0
 65. JOINT LOAD
 66. 17 TO 48 FY -7.0
 67. 49 TO 64 FY -3.5
 68. PDELTA ANALYSIS PRINT LOAD DATA
         P R O B L E M   S T A T I S T I C S
         -----------------------------------
  NUMBER OF JOINTS         64  NUMBER OF MEMBERS     120
  NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
  NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     16
        Using 64-bit analysis engine.
        SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     288
TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    3
   EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    4
LOADING     1
-----------
    SELFWEIGHT  Y   -1.000
   ACTUAL WEIGHT OF THE STRUCTURE =     800.269 KNS
JOINT LOAD - UNIT KNS  METE
JOINT   FORCE-X   FORCE-Y    FORCE-Z     MOM-X     MOM-Y     MOM-Z
   17      0.00     -7.00       0.00      0.00      0.00      0.00
   18      0.00     -7.00       0.00      0.00      0.00      0.00
   19      0.00     -7.00       0.00      0.00      0.00      0.00
   20      0.00     -7.00       0.00      0.00      0.00      0.00
   21      0.00     -7.00       0.00      0.00      0.00      0.00
   22      0.00     -7.00       0.00      0.00      0.00      0.00
   23      0.00     -7.00       0.00      0.00      0.00      0.00
   24      0.00     -7.00       0.00      0.00      0.00      0.00
   25      0.00     -7.00       0.00      0.00      0.00      0.00
   26      0.00     -7.00       0.00      0.00      0.00      0.00
   27      0.00     -7.00       0.00      0.00      0.00      0.00
   28      0.00     -7.00       0.00      0.00      0.00      0.00
```

```
   29      0.00     -7.00         0.00      0.00      0.00      0.00
   30      0.00     -7.00         0.00      0.00      0.00      0.00
   31      0.00     -7.00         0.00      0.00      0.00      0.00
   32      0.00     -7.00         0.00      0.00      0.00      0.00
   33      0.00     -7.00         0.00      0.00      0.00      0.00
   34      0.00     -7.00         0.00      0.00      0.00      0.00
   35      0.00     -7.00         0.00      0.00      0.00      0.00
   36      0.00     -7.00         0.00      0.00      0.00      0.00
   37      0.00     -7.00         0.00      0.00      0.00      0.00
   38      0.00     -7.00         0.00      0.00      0.00      0.00
   39      0.00     -7.00         0.00      0.00      0.00      0.00
   40      0.00     -7.00         0.00      0.00      0.00      0.00
   41      0.00     -7.00         0.00      0.00      0.00      0.00
   42      0.00     -7.00         0.00      0.00      0.00      0.00
   43      0.00     -7.00         0.00      0.00      0.00      0.00
   44      0.00     -7.00         0.00      0.00      0.00      0.00
   45      0.00     -7.00         0.00      0.00      0.00      0.00
   46      0.00     -7.00         0.00      0.00      0.00      0.00
   47      0.00     -7.00         0.00      0.00      0.00      0.00
   48      0.00     -7.00         0.00      0.00      0.00      0.00
   49      0.00     -3.50         0.00      0.00      0.00      0.00
   50      0.00     -3.50         0.00      0.00      0.00      0.00
   51      0.00     -3.50         0.00      0.00      0.00      0.00
   52      0.00     -3.50         0.00      0.00      0.00      0.00
   53      0.00     -3.50         0.00      0.00      0.00      0.00
   54      0.00     -3.50         0.00      0.00      0.00      0.00
   55      0.00     -3.50         0.00      0.00      0.00      0.00
   56      0.00     -3.50         0.00      0.00      0.00      0.00
   57      0.00     -3.50         0.00      0.00      0.00      0.00
   58      0.00     -3.50         0.00      0.00      0.00      0.00
   EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    5
   59      0.00     -3.50         0.00      0.00      0.00      0.00
   60      0.00     -3.50         0.00      0.00      0.00      0.00
   61      0.00     -3.50         0.00      0.00      0.00      0.00
   62      0.00     -3.50         0.00      0.00      0.00      0.00
   63      0.00     -3.50         0.00      0.00      0.00      0.00
   64      0.00     -3.50         0.00      0.00      0.00      0.00
LOADING     2
-----------
    SELFWEIGHT  Y   -1.000
   ACTUAL WEIGHT OF THE STRUCTURE =     800.269 KNS
JOINT LOAD - UNIT KNS  METE
JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
   17      0.00     -7.00         0.00      0.00      0.00      0.00
   18      0.00     -7.00         0.00      0.00      0.00      0.00
   19      0.00     -7.00         0.00      0.00      0.00      0.00
   20      0.00     -7.00         0.00      0.00      0.00      0.00
   21      0.00     -7.00         0.00      0.00      0.00      0.00
   22      0.00     -7.00         0.00      0.00      0.00      0.00
   23      0.00     -7.00         0.00      0.00      0.00      0.00
   24      0.00     -7.00         0.00      0.00      0.00      0.00
   25      0.00     -7.00         0.00      0.00      0.00      0.00
   26      0.00     -7.00         0.00      0.00      0.00      0.00
   27      0.00     -7.00         0.00      0.00      0.00      0.00
   28      0.00     -7.00         0.00      0.00      0.00      0.00
   29      0.00     -7.00         0.00      0.00      0.00      0.00
   30      0.00     -7.00         0.00      0.00      0.00      0.00
   31      0.00     -7.00         0.00      0.00      0.00      0.00
```

```
    32      0.00     -7.00        0.00      0.00      0.00      0.00
    33      0.00     -7.00        0.00      0.00      0.00      0.00
    34      0.00     -7.00        0.00      0.00      0.00      0.00
    35      0.00     -7.00        0.00      0.00      0.00      0.00
    36      0.00     -7.00        0.00      0.00      0.00      0.00
    37      0.00     -7.00        0.00      0.00      0.00      0.00
    38      0.00     -7.00        0.00      0.00      0.00      0.00
    39      0.00     -7.00        0.00      0.00      0.00      0.00
    40      0.00     -7.00        0.00      0.00      0.00      0.00
    41      0.00     -7.00        0.00      0.00      0.00      0.00
    42      0.00     -7.00        0.00      0.00      0.00      0.00
    43      0.00     -7.00        0.00      0.00      0.00      0.00
    44      0.00     -7.00        0.00      0.00      0.00      0.00
    45      0.00     -7.00        0.00      0.00      0.00      0.00
    46      0.00     -7.00        0.00      0.00      0.00      0.00
    47      0.00     -7.00        0.00      0.00      0.00      0.00
    48      0.00     -7.00        0.00      0.00      0.00      0.00
    49      0.00     -3.50        0.00      0.00      0.00      0.00
    50      0.00     -3.50        0.00      0.00      0.00      0.00
    51      0.00     -3.50        0.00      0.00      0.00      0.00
    52      0.00     -3.50        0.00      0.00      0.00      0.00
    EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    6
    53      0.00     -3.50        0.00      0.00      0.00      0.00
    54      0.00     -3.50        0.00      0.00      0.00      0.00
    55      0.00     -3.50        0.00      0.00      0.00      0.00
    56      0.00     -3.50        0.00      0.00      0.00      0.00
    57      0.00     -3.50        0.00      0.00      0.00      0.00
    58      0.00     -3.50        0.00      0.00      0.00      0.00
    59      0.00     -3.50        0.00      0.00      0.00      0.00
    60      0.00     -3.50        0.00      0.00      0.00      0.00
    61      0.00     -3.50        0.00      0.00      0.00      0.00
    62      0.00     -3.50        0.00      0.00      0.00      0.00
    63      0.00     -3.50        0.00      0.00      0.00      0.00
    64      0.00     -3.50        0.00      0.00      0.00      0.00
**WARNING: IF THIS UBC/IBC ANALYSIS HAS TENSION/COMPRESSION
OR REPEAT LOAD OR RE-ANALYSIS OR SELECT OPTIMIZE, THEN EACH
UBC/IBC CASE SHOULD BE FOLLOWED BY PERFORM ANALYSIS & CHANGE.
 ***********************************************************
 *                                                         *
 *  X  DIRECTION : Ta =  0.455 Tb =  0.285 Tuser =  0.000  *
 *  C =   2.7500, LOAD FACTOR =  0.750                     *
 * UBC TYPE = 94                                           *
 * UBC FACTOR V = 0.0611 x       1080.27 =      66.02 KNS  *
 *                                                         *
 ***********************************************************
 ***********************************************************
 *                                                         *
 *  Z  DIRECTION : Ta =  0.455 Tb =  1.092 Tuser =  0.000  *
 *  C =   2.7500, LOAD FACTOR =  0.750                     *
 * UBC TYPE = 94                                           *
 * UBC FACTOR V = 0.0611 x       1080.27 =      66.02 KNS  *
 *                                                         *
 ***********************************************************
  JOINT             LATERAL          TORSIONAL           LOAD -    1
                    LOAD (KNS )      MOMENT (KNS -METE) FACTOR -   0.750
  -----             -------          ---------
    17     FX        0.449    MY        0.000
    18     FX        0.569    MY        0.000
```

```
      19     FX        0.569    MY        0.000
      20     FX        0.449    MY        0.000
      21     FX        0.569    MY        0.000
      22     FX        0.688    MY        0.000
      23     FX        0.688    MY        0.000
      24     FX        0.569    MY        0.000
      25     FX        0.569    MY        0.000
      26     FX        0.688    MY        0.000
      27     FX        0.688    MY        0.000
      28     FX        0.569    MY        0.000
      29     FX        0.449    MY        0.000
      EXAMPLE PROBLEM FOR UBC LOAD                            -- PAGE NO.    7
      30     FX        0.569    MY        0.000
      31     FX        0.569    MY        0.000
      32     FX        0.449    MY        0.000
                 -----------        -----------
          TOTAL =      9.097              0.000 AT LEVEL     3.500 METE
      33     FX        0.899    MY        0.000
      34     FX        1.137    MY        0.000
      35     FX        1.137    MY        0.000
      36     FX        0.899    MY        0.000
      37     FX        1.137    MY        0.000
      38     FX        1.375    MY        0.000
      39     FX        1.375    MY        0.000
      40     FX        1.137    MY        0.000
      41     FX        1.137    MY        0.000
      42     FX        1.375    MY        0.000
      43     FX        1.375    MY        0.000
      44     FX        1.137    MY        0.000
      45     FX        0.899    MY        0.000
      46     FX        1.137    MY        0.000
      47     FX        1.137    MY        0.000
      48     FX        0.899    MY        0.000
                 -----------        -----------
          TOTAL =     18.194              0.000 AT LEVEL     7.000 METE
      49     FX        1.032    MY        0.000
      50     FX        1.389    MY        0.000
      51     FX        1.389    MY        0.000
      52     FX        1.032    MY        0.000
      53     FX        1.389    MY        0.000
      54     FX        1.746    MY        0.000
      55     FX        1.746    MY        0.000
      56     FX        1.389    MY        0.000
      57     FX        1.389    MY        0.000
      58     FX        1.746    MY        0.000
      59     FX        1.746    MY        0.000
      60     FX        1.389    MY        0.000
      61     FX        1.032    MY        0.000
      62     FX        1.389    MY        0.000
      63     FX        1.389    MY        0.000
      64     FX        1.032    MY        0.000
                 -----------        -----------
          TOTAL =     22.222              0.000 AT LEVEL    10.500 METE
    JOINT           LATERAL          TORSIONAL            LOAD -    2
                    LOAD (KNS )      MOMENT (KNS -METE) FACTOR -   0.750
    -----           -------          ---------
      17     FZ        0.449    MY        0.000
      18     FZ        0.569    MY        0.000
```

```
    19     FZ        0.569    MY        0.000
    20     FZ        0.449    MY        0.000
    21     FZ        0.569    MY        0.000
    22     FZ        0.688    MY        0.000
    23     FZ        0.688    MY        0.000
    EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    8
    24     FZ        0.569    MY        0.000
    25     FZ        0.569    MY        0.000
    26     FZ        0.688    MY        0.000
    27     FZ        0.688    MY        0.000
    28     FZ        0.569    MY        0.000
    29     FZ        0.449    MY        0.000
    30     FZ        0.569    MY        0.000
    31     FZ        0.569    MY        0.000
    32     FZ        0.449    MY        0.000
               -----------        -----------
        TOTAL =      9.097              0.000 AT LEVEL     3.500 METE
    33     FZ        0.899    MY        0.000
    34     FZ        1.137    MY        0.000
    35     FZ        1.137    MY        0.000
    36     FZ        0.899    MY        0.000
    37     FZ        1.137    MY        0.000
    38     FZ        1.375    MY        0.000
    39     FZ        1.375    MY        0.000
    40     FZ        1.137    MY        0.000
    41     FZ        1.137    MY        0.000
    42     FZ        1.375    MY        0.000
    43     FZ        1.375    MY        0.000
    44     FZ        1.137    MY        0.000
    45     FZ        0.899    MY        0.000
    46     FZ        1.137    MY        0.000
    47     FZ        1.137    MY        0.000
    48     FZ        0.899    MY        0.000
               -----------        -----------
        TOTAL =     18.194              0.000 AT LEVEL     7.000 METE
    49     FZ        1.032    MY        0.000
    50     FZ        1.389    MY        0.000
    51     FZ        1.389    MY        0.000
    52     FZ        1.032    MY        0.000
    53     FZ        1.389    MY        0.000
    54     FZ        1.746    MY        0.000
    55     FZ        1.746    MY        0.000
    56     FZ        1.389    MY        0.000
    57     FZ        1.389    MY        0.000
    58     FZ        1.746    MY        0.000
    59     FZ        1.746    MY        0.000
    60     FZ        1.389    MY        0.000
    61     FZ        1.032    MY        0.000
    62     FZ        1.389    MY        0.000
    63     FZ        1.389    MY        0.000
    64     FZ        1.032    MY        0.000
               -----------        -----------
        TOTAL =     22.222              0.000 AT LEVEL    10.500 METE
 ++ Adjusting Displacements.
    EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.    9
 ************ END OF DATA FROM INTERNAL STORAGE ************
  69. PRINT SUPPORT REACTIONS
SUPPORT  REACTION
```

```
      EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.   10
   SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     -2.21     43.45      0.06      0.07     -0.00      5.81
           2      0.46     41.46     -3.04     -5.63      0.00     -0.50
      2    1     -3.40     65.65      0.06      0.07     -0.00      7.09
           2      0.01     56.21     -3.01     -5.63      0.00     -0.01
      3    1     -3.43     64.48      0.06      0.07     -0.00      7.13
           2     -0.01     56.21     -3.01     -5.63     -0.00      0.01
      4    1     -3.13     57.17      0.06      0.07     -0.00      6.80
           2     -0.46     41.46     -3.04     -5.63     -0.00      0.50
      5    1     -2.31     62.90     -0.01     -0.01     -0.00      6.02
           2      0.46     73.97     -3.14     -5.82      0.00     -0.50
      6    1     -3.52     85.32     -0.01     -0.01     -0.00      7.33
           2      0.01     88.73     -3.11     -5.83      0.00     -0.02
      7    1     -3.54     84.12     -0.01     -0.01     -0.00      7.37
           2     -0.01     88.73     -3.11     -5.83     -0.00      0.02
      8    1     -3.22     77.05     -0.01     -0.01     -0.00      7.02
           2     -0.46     73.97     -3.14     -5.82     -0.00      0.50
      9    1     -2.31     62.90      0.01      0.01      0.00      6.02
           2      0.46     65.97     -3.14     -5.81      0.00     -0.50
     10    1     -3.52     85.32      0.01      0.01      0.00      7.33
           2      0.01     80.72     -3.11     -5.81      0.00     -0.02
     11    1     -3.54     84.12      0.01      0.01      0.00      7.37
           2     -0.01     80.72     -3.11     -5.81     -0.00      0.02
     12    1     -3.22     77.05      0.01      0.01      0.00      7.02
           2     -0.46     65.97     -3.14     -5.81     -0.00      0.50
     13    1     -2.21     43.45     -0.06     -0.07      0.00      5.81
           2      0.46     59.15     -3.12     -5.76      0.00     -0.50
     14    1     -3.40     65.65     -0.06     -0.07      0.00      7.09
           2      0.01     73.91     -3.09     -5.76      0.00     -0.02
     15    1     -3.43     64.48     -0.06     -0.07      0.00      7.13
           2     -0.01     73.91     -3.09     -5.76     -0.00      0.02
     16    1     -3.13     57.17     -0.06     -0.07      0.00      6.80
           2     -0.46     59.15     -3.12     -5.76     -0.00      0.50
   ************** END OF LATEST ANALYSIS RESULT **************
    70. FINISH
      EXAMPLE PROBLEM FOR UBC LOAD                             -- PAGE NO.   11
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44:18 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. UK-15 Wind and Floor Load Generation on a Space Frame

A space frame is analyzed for loads generated using the built-in wind and floor load generation facilities.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-15 Wind and Floor Load Generation on a Space Frame.STD` when you install the program.

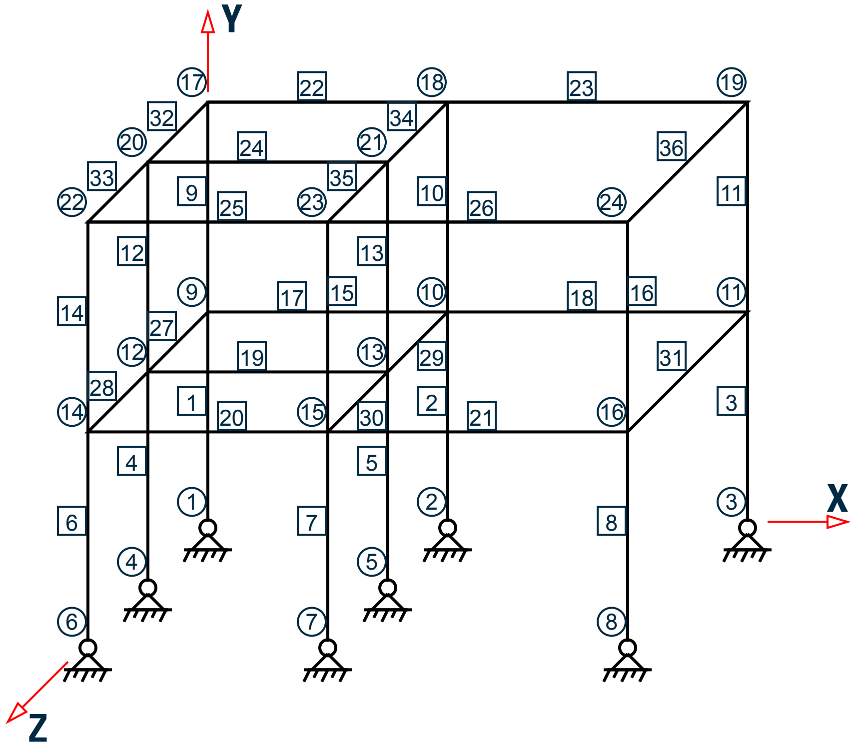


Figure 556: Example Problem No. 15

```
STAAD SPACE - WIND AND FLOOR LOAD GENERATION
```

This is a space frame analysis problem. Every STAAD input has to start with the command STAAD. The SPACE specification is used to denote a space (3D) frame.

```
UNIT METER KNS
```

The UNIT specification is used to specify the length and/or force units to be used.

```
JOINT COORDINATES
1 0 0 0
2 4 0 0
3 9 0 0
4 0 0 4
5 4 0 4
6 0 0 8
7 4 0 8
8 9 0 8
REPEAT ALL 2 0 3.5 0
```

The `JOINT COORDINATE` specification is used to specify the X, Y, and Z coordinates of the joints. Note that the `REPEAT ALL` command has been used to generate joints for the two upper stories each with a Y increment of 3.5 m.

```
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
```

The `MEMBER INCIDENCE` specification is used for specifying member connectivities.

```
MEMBER PROPERTIES BRITISH
1 TO 16 TA ST UB457X191X74
17 TO 26 TA ST UB457X152X52
27 TO 36 TA ST UB457X152X52
```

Properties for all members are specified from the built-in `BRITISH` steel table. Three different sections have been used.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 8 FIXED BUT MX MZ
```

The supports of the structure are defined through the `SUPPORT` specification. Here all the supports are `FIXED` with releases specified in the `MX` (rotation about global X-axis) and `MZ` (rotation about global Z-axis) directions.

```
DEFINE WIND LOAD
TYPE 1
INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
EXPOSURE 0.90 YRANGE 6.0 8.0
EXPOSURE 0.85 JOINT 9 12 14
```

When a structure has to be analyzed for wind loading, the engineer is confronted with the task of first converting an abstract quantity like wind velocity or wind pressure into concentrated loads at joints, distributed loads on members, or pressure loads on plates. The large number of calculations involved in this conversion can be avoided by making use of STAAD's wind load generation utility. This utility takes wind pressure at various heights as the input, and converts them to values that can then be used as concentrated forces known as joint loads in specific load cases. The input specification is done in two stages. The first stage is initiated above through the `DEFINE WIND LOAD` command. The basic parameters of the WIND loading are specified here. All values need to be provided in the current UNIT system. Each wind category is identified with a `TYPE` number (an identification mark) which is used later to specify load cases.

In this example, two different wind intensities (1.0 KN/sq. m and 1.5 KN/sq. m) are specified for two different height zones (0 to 3.5m and 3.5 to 7.0m). The `EXPOSURE` specification is used to mitigate or magnify the effect at specific nodes due to special considerations like openings in the structure. In this case, two different exposure factors are specified. The first EXPOSURE specification specifies the exposure factor as 0.9 for all joints within the height range (defined as global Y-range) of 6.0m – 8.0m. The second EXPOSURE specification specifies the exposure factor as 0.85 for joints 9, 12 and 14. In the EXPOSURE factor specification, the joints may be specified directly or through a vertical range specification.

```
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
```

This is the second stage of input specification for the wind load generation. The term `WIND LOAD` and the direction term that follows are used to specify the wind loading in a particular lateral direction. In this case, WIND loading `TYPE 1`, defined previously, is being applied in the global X-direction with a positive multiplication factor of 1.2 .

```
LOAD 2 FLOOR LOAD @ Y = 3.5M AND 7M
FLOOR LOAD
YRANGE 3.4 3.6 FLOAD -5.0 XRANGE 0.0 4.0 ZRANGE 0.0 8.0
YRANGE 3.4 3.6 FLOAD -2.5 XRANGE 4.0 9.0 ZRANGE 0.0 8.0
YRANGE 6.9 7.1 FLOAD -2.5
```

In load case 2 in this problem, a floor load generation is performed. In a floor load generation, a pressure load (force per unit area) is converted by the program into specific points forces and distributed forces on the members located in that region. The `YRANGE`, `XRANGE`, and `ZRANGE` specifications are used to define the area of the structure on which the pressure is acting. The `FLOAD` specification is used to specify the value of that pressure. All values need to be provided in the current UNIT system. For example, in the first line in the above FLOOR LOAD specification, the region is defined as being located within the bounds YRANGE of 3.4 – 3.6 m, XRANGE of 0.0 - 4.0 m and ZRANGE of 0.0 - 8.0 m. The –5.0 signifies that the pressure is 5.0 KN/sq.m. in the negative global Y direction.

The program will identify the members lying within the specified region and derive member loads on these members based on two-way load distribution.

```
PERFORM ANALYSIS PRINT LOAD DATA
```

We can view the values and position of the generated loads with the help of the `PRINT LOAD DATA` command used above along with the `PERFORM ANALYSIS` command.

```
PRINT SUPPORT REACTION
FINISH
```

Above commands are self-explanatory.

Input File

```
STAAD SPACE - WIND AND FLOOR LOAD GENERATION
UNIT METER KNS
JOINT COORDINATES
1 0 0 0
2 4 0 0
```

```
3 9 0 0
4 0 0 4
5 4 0 4
6 0 0 8
7 4 0 8
8 9 0 8
REPEAT ALL 2 0 3.5 0
MEMBER INCIDENCES
* Columns
1 1 9 16
* Beams in the X direction
17 9 10 18
19 12 13
20 14 15 21
22 17 18 23
24 20 21
25 22 23 26
* Beams in the Z direction
27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
MEMBER PROPERTIES BRITISH
1 TO 16 TA ST UB457X191X74
17 TO 26 TA ST UB457X152X52
27 TO 36 TA ST UB457X152X52
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
SUPPORT
1 TO 8 FIXED BUT MX MZ
DEFINE WIND LOAD
TYPE 1
INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
EXPOSURE 0.90 YRANGE 6.0 8.0
EXPOSURE 0.85 JOINT 9 12 14
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
LOAD 2 FLOOR LOAD AT Y = 3.5M AND 7M
FLOOR LOAD
YRANGE 3.4 3.6 FLOAD -5.0 XRANGE 0.0 4.0 ZRANGE 0.0 8.0
YRANGE 3.4 3.6 FLOAD -2.5 XRANGE 4.0 9.0 ZRANGE 0.0 8.0
YRANGE 6.9 7.1 FLOAD -2.5
PERFORM ANALYSIS PRINT LOAD DATA
PRINT SUPPORT REACTION
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:20                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE - WIND AND FLOOR LOAD GENERATION
INPUT FILE: UK-15 Wind and Floor Load Generation on a Space Frame.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0 0 0
     5. 2 4 0 0
     6. 3 9 0 0
     7. 4 0 0 4
     8. 5 4 0 4
     9. 6 0 0 8
    10. 7 4 0 8
    11. 8 9 0 8
    12. REPEAT ALL 2 0 3.5 0
    13. MEMBER INCIDENCES
    14. * COLUMNS
    15. 1 1 9 16
    16. * BEAMS IN THE X DIRECTION
    17. 17 9 10 18
    18. 19 12 13
    19. 20 14 15 21
    20. 22 17 18 23
    21. 24 20 21
    22. 25 22 23 26
    23. * BEAMS IN THE Z DIRECTION
    24. 27 9 12 ; 28 12 14 ; 29 10 13 ; 30 13 15 ; 31 11 16
    25. 32 17 20 ; 33 20 22 ; 34 18 21 ; 35 21 23 ; 36 19 24
    26. MEMBER PROPERTIES BRITISH
    27. 1 TO 16 TA ST UB457X191X74
    28. 17 TO 26 TA ST UB457X152X52
    29. 27 TO 36 TA ST UB457X152X52
    30. UNIT MMS
    31. DEFINE MATERIAL START
    32. ISOTROPIC STEEL
    33. E 210
    34. POISSON 0.3
    35. DENSITY 7.6977E-008
    36. ALPHA 6E-006
    37. DAMP 0.03
    38. TYPE STEEL
      - WIND AND FLOOR LOAD GENERATION                        -- PAGE NO.    2
    39. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
    40. END DEFINE MATERIAL
    41. CONSTANTS
```

```
   42. MATERIAL STEEL ALL
   43. UNIT METER
   44. SUPPORT
   45. 1 TO 8 FIXED BUT MX MZ
   46. DEFINE WIND LOAD
  *** NOTE: If any floor diaphragm is present in the model Wind Load definition
            should be defined after Floor Diaphragm definition. Otherwise wind
            load generation may be unsuccessful during analysis.
   47. TYPE 1
   48. INTENSITY 1.0 1.5 HEIGHT 3.5 7.0
   49. EXPOSURE 0.90 YRANGE 6.0 8.0
   50. EXPOSURE 0.85 JOINT 9 12 14
   51. LOAD 1 WIND LOAD IN X-DIRECTION
   52. WIND LOAD X 1.2 TYPE 1
   53. LOAD 2 FLOOR LOAD AT Y = 3.5M AND 7M
   54. FLOOR LOAD
   55. YRANGE 3.4 3.6 FLOAD -5.0 XRANGE 0.0 4.0 ZRANGE 0.0 8.0
**NOTE** about Floor/OneWay Loads/Weights.
  Please note that depending on the shape of the floor you may
  have to break up the FLOOR/ONEWAY LOAD into multiple commands.
  For details please refer to Technical Reference Manual
  Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
   56. YRANGE 3.4 3.6 FLOAD -2.5 XRANGE 4.0 9.0 ZRANGE 0.0 8.0
   57. YRANGE 6.9 7.1 FLOAD -2.5
   58. PERFORM ANALYSIS PRINT LOAD DATA
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
    NUMBER OF JOINTS         24  NUMBER OF MEMBERS      36
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      8
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     112
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
    - WIND AND FLOOR LOAD GENERATION                          -- PAGE NO.    3
    - WIND AND FLOOR LOAD GENERATION                          -- PAGE NO.    4
  LOADING     1  WIND LOAD IN X-DIRECTION
  -----------
  JOINT LOAD - UNIT KNS  METE
  JOINT   FORCE-X   FORCE-Y     FORCE-Z     MOM-X     MOM-Y     MOM-Z
      1      4.20      0.00        0.00      0.00      0.00      0.00
      4      8.40      0.00        0.00      0.00      0.00      0.00
      6      4.20      0.00        0.00      0.00      0.00      0.00
      9      8.93      0.00        0.00      0.00      0.00      0.00
     12     17.85      0.00        0.00      0.00      0.00      0.00
     14      8.93      0.00        0.00      0.00      0.00      0.00
     17      5.67      0.00        0.00      0.00      0.00      0.00
     20     11.34      0.00        0.00      0.00      0.00      0.00
     22      5.67      0.00        0.00      0.00      0.00      0.00
  LOADING     2  FLOOR LOAD AT Y = 3.5M AND 7M
  -----------
  MEMBER LOAD - UNIT KNS  METE
  MEMBER     UDL      L1     L2       CON       L     LIN1     LIN2
    17                            -0.1563 GY   0.17
    17                            -0.4688 GY   0.39
    17                            -0.7812 GY   0.63
    17                            -1.0938 GY   0.88
    17                            -1.4062 GY   1.13
```

```
     17                            -1.7188 GY   1.38
     17                            -2.0312 GY   1.63
     17                            -2.3438 GY   1.88
     17                            -2.3438 GY   2.12
     17                            -2.0312 GY   2.37
     17                            -1.7188 GY   2.62
     17                            -1.4062 GY   2.87
     17                            -1.0938 GY   3.12
     17                            -0.7812 GY   3.37
     17                            -0.4688 GY   3.61
     17                            -0.1563 GY   3.83
     29                            -0.1563 GY   0.17
     29                            -0.4688 GY   0.39
     29                            -0.7812 GY   0.63
     29                            -1.0938 GY   0.88
     29                            -1.4062 GY   1.13
     29                            -1.7188 GY   1.38
     29                            -2.0312 GY   1.63
     29                            -2.3438 GY   1.88
     29                            -2.3438 GY   2.12
     29                            -2.0312 GY   2.37
     29                            -1.7188 GY   2.62
      - WIND AND FLOOR LOAD GENERATION                        -- PAGE NO.    5
     29                            -1.4062 GY   2.87
     29                            -1.0938 GY   3.12
     29                            -0.7812 GY   3.37
     29                            -0.4688 GY   3.61
     29                            -0.1563 GY   3.83
     19                            -0.1563 GY   0.17
     19                            -0.4688 GY   0.39
     19                            -0.7812 GY   0.63
     19                            -1.0938 GY   0.88
     19                            -1.4062 GY   1.13
     19                            -1.7188 GY   1.38
     19                            -2.0312 GY   1.63
     19                            -2.3438 GY   1.88
     19                            -2.3438 GY   2.12
     19                            -2.0312 GY   2.37
     19                            -1.7188 GY   2.62
     19                            -1.4062 GY   2.87
     19                            -1.0938 GY   3.12
     19                            -0.7812 GY   3.37
     19                            -0.4688 GY   3.61
     19                            -0.1563 GY   3.83
     27                            -0.1563 GY   0.17
     27                            -0.4688 GY   0.39
     27                            -0.7812 GY   0.63
     27                            -1.0938 GY   0.88
     27                            -1.4062 GY   1.13
     27                            -1.7188 GY   1.38
     27                            -2.0312 GY   1.63
     27                            -2.3438 GY   1.88
     27                            -2.3438 GY   2.12
     27                            -2.0312 GY   2.37
     27                            -1.7188 GY   2.62
     27                            -1.4062 GY   2.87
     27                            -1.0938 GY   3.12
     27                            -0.7812 GY   3.37
```

```
     27                           -0.4688 GY   3.61
     27                           -0.1563 GY   3.83
     19                           -0.1563 GY   0.17
     19                           -0.4688 GY   0.39
     19                           -0.7812 GY   0.63
     19                           -1.0938 GY   0.88
     19                           -1.4062 GY   1.13
     19                           -1.7188 GY   1.38
     19                           -2.0312 GY   1.63
     19                           -2.3438 GY   1.88
     19                           -2.3438 GY   2.12
     19                           -2.0312 GY   2.37
     19                           -1.7188 GY   2.62
     19                           -1.4062 GY   2.87
     19                           -1.0938 GY   3.12
     19                           -0.7812 GY   3.37
     19                           -0.4688 GY   3.61
     19                           -0.1563 GY   3.83
     30                           -0.1563 GY   0.17
     30                           -0.4688 GY   0.39
     30                           -0.7812 GY   0.63
      - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.    6
     30                           -1.0938 GY   0.88
     30                           -1.4062 GY   1.13
     30                           -1.7188 GY   1.38
     30                           -2.0312 GY   1.63
     30                           -2.3438 GY   1.88
     30                           -2.3438 GY   2.12
     30                           -2.0312 GY   2.37
     30                           -1.7188 GY   2.62
     30                           -1.4062 GY   2.87
     30                           -1.0938 GY   3.12
     30                           -0.7812 GY   3.37
     30                           -0.4688 GY   3.61
     30                           -0.1563 GY   3.83
     20                           -0.1563 GY   0.17
     20                           -0.4688 GY   0.39
     20                           -0.7812 GY   0.63
     20                           -1.0938 GY   0.88
     20                           -1.4062 GY   1.13
     20                           -1.7188 GY   1.38
     20                           -2.0312 GY   1.63
     20                           -2.3438 GY   1.88
     20                           -2.3438 GY   2.12
     20                           -2.0312 GY   2.37
     20                           -1.7188 GY   2.62
     20                           -1.4062 GY   2.87
     20                           -1.0938 GY   3.12
     20                           -0.7812 GY   3.37
     20                           -0.4688 GY   3.61
     20                           -0.1563 GY   3.83
     28                           -0.1563 GY   0.17
     28                           -0.4688 GY   0.39
     28                           -0.7812 GY   0.63
     28                           -1.0938 GY   0.88
     28                           -1.4062 GY   1.13
     28                           -1.7188 GY   1.38
     28                           -2.0312 GY   1.63
```

```
    28                            -2.3438 GY   1.88
    28                            -2.3438 GY   2.12
    28                            -2.0312 GY   2.37
    28                            -1.7188 GY   2.62
    28                            -1.4062 GY   2.87
    28                            -1.0938 GY   3.12
    28                            -0.7812 GY   3.37
    28                            -0.4688 GY   3.61
    28                            -0.1563 GY   3.83
  MEMBER LOAD - UNIT KNS  METE
  MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
    18                            -0.1221 GY   0.21
    18                            -0.3662 GY   0.49
    18                            -0.6104 GY   0.79
    18                            -0.8545 GY   1.10
    18                            -1.0986 GY   1.41
     - WIND AND FLOOR LOAD GENERATION                        -- PAGE NO.    7
    18                            -1.3428 GY   1.72
    18                            -1.5869 GY   2.04
    18                            -1.8311 GY   2.35
    18                            -1.8311 GY   2.65
    18                            -1.5869 GY   2.96
    18                            -1.3428 GY   3.28
    18                            -1.0986 GY   3.59
    18                            -0.8545 GY   3.90
    18                            -0.6104 GY   4.21
    18                            -0.3662 GY   4.51
    18                            -0.1221 GY   4.79
    31                            -0.1221 GY   0.21
    31                            -0.3662 GY   0.49
    31                            -0.6104 GY   0.79
    31                            -0.8545 GY   1.10
    31                            -1.0986 GY   1.41
    31                            -1.3428 GY   1.72
    31                            -1.5869 GY   2.04
    31                            -1.8311 GY   2.35
    31    -6.2500 GY   2.50    5.50
    31                            -1.8311 GY   5.65
    31                            -1.5869 GY   5.96
    31                            -1.3428 GY   6.28
    31                            -1.0986 GY   6.59
    31                            -0.8545 GY   6.90
    31                            -0.6104 GY   7.21
    31                            -0.3662 GY   7.51
    31                            -0.1221 GY   7.79
    21                            -0.1221 GY   0.21
    21                            -0.3662 GY   0.49
    21                            -0.6104 GY   0.79
    21                            -0.8545 GY   1.10
    21                            -1.0986 GY   1.41
    21                            -1.3428 GY   1.72
    21                            -1.5869 GY   2.04
    21                            -1.8311 GY   2.35
    21                            -1.8311 GY   2.65
    21                            -1.5869 GY   2.96
    21                            -1.3428 GY   3.28
    21                            -1.0986 GY   3.59
    21                            -0.8545 GY   3.90
```

```
   21                           -0.6104 GY   4.21
   21                           -0.3662 GY   4.51
   21                           -0.1221 GY   4.79
   30                           -1.8311 GY   1.65
   30                           -1.5869 GY   1.96
   30                           -1.3428 GY   2.28
   30                           -1.0986 GY   2.59
   30                           -0.8545 GY   2.90
   30                           -0.6104 GY   3.21
   30                           -0.3662 GY   3.51
   30                           -0.1221 GY   3.79
   30    -6.2500 GY   0.00    1.50
   29                           -0.1221 GY   0.21
   29                           -0.3662 GY   0.49
   29                           -0.6104 GY   0.79
    - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.    8
   29                           -0.8545 GY   1.10
   29                           -1.0986 GY   1.41
   29                           -1.3428 GY   1.72
   29                           -1.5869 GY   2.04
   29                           -1.8311 GY   2.35
   29    -6.2500 GY   2.50    4.00
 MEMBER LOAD - UNIT KNS  METE
 MEMBER     UDL      L1     L2      CON       L     LIN1     LIN2
   22                           -0.0781 GY   0.17
   22                           -0.2344 GY   0.39
   22                           -0.3906 GY   0.63
   22                           -0.5469 GY   0.88
   22                           -0.7031 GY   1.13
   22                           -0.8594 GY   1.38
   22                           -1.0156 GY   1.63
   22                           -1.1719 GY   1.88
   22                           -1.1719 GY   2.12
   22                           -1.0156 GY   2.37
   22                           -0.8594 GY   2.62
   22                           -0.7031 GY   2.87
   22                           -0.5469 GY   3.12
   22                           -0.3906 GY   3.37
   22                           -0.2344 GY   3.61
   22                           -0.0781 GY   3.83
   34                           -0.0781 GY   0.17
   34                           -0.2344 GY   0.39
   34                           -0.3906 GY   0.63
   34                           -0.5469 GY   0.88
   34                           -0.7031 GY   1.13
   34                           -0.8594 GY   1.38
   34                           -1.0156 GY   1.63
   34                           -1.1719 GY   1.88
   34                           -1.1719 GY   2.12
   34                           -1.0156 GY   2.37
   34                           -0.8594 GY   2.62
   34                           -0.7031 GY   2.87
   34                           -0.5469 GY   3.12
   34                           -0.3906 GY   3.37
   34                           -0.2344 GY   3.61
   34                           -0.0781 GY   3.83
   24                           -0.0781 GY   0.17
   24                           -0.2344 GY   0.39
```

```
     24                            -0.3906 GY   0.63
     24                            -0.5469 GY   0.88
     24                            -0.7031 GY   1.13
     24                            -0.8594 GY   1.38
     24                            -1.0156 GY   1.63
     24                            -1.1719 GY   1.88
     24                            -1.1719 GY   2.12
     24                            -1.0156 GY   2.37
     24                            -0.8594 GY   2.62
     24                            -0.7031 GY   2.87
      - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.    9
     24                            -0.5469 GY   3.12
     24                            -0.3906 GY   3.37
     24                            -0.2344 GY   3.61
     24                            -0.0781 GY   3.83
     32                            -0.0781 GY   0.17
     32                            -0.2344 GY   0.39
     32                            -0.3906 GY   0.63
     32                            -0.5469 GY   0.88
     32                            -0.7031 GY   1.13
     32                            -0.8594 GY   1.38
     32                            -1.0156 GY   1.63
     32                            -1.1719 GY   1.88
     32                            -1.1719 GY   2.12
     32                            -1.0156 GY   2.37
     32                            -0.8594 GY   2.62
     32                            -0.7031 GY   2.87
     32                            -0.5469 GY   3.12
     32                            -0.3906 GY   3.37
     32                            -0.2344 GY   3.61
     32                            -0.0781 GY   3.83
     23                            -0.1221 GY   0.21
     23                            -0.3662 GY   0.49
     23                            -0.6104 GY   0.79
     23                            -0.8545 GY   1.10
     23                            -1.0986 GY   1.41
     23                            -1.3428 GY   1.72
     23                            -1.5869 GY   2.04
     23                            -1.8311 GY   2.35
     23                            -1.8311 GY   2.65
     23                            -1.5869 GY   2.96
     23                            -1.3428 GY   3.28
     23                            -1.0986 GY   3.59
     23                            -0.8545 GY   3.90
     23                            -0.6104 GY   4.21
     23                            -0.3662 GY   4.51
     23                            -0.1221 GY   4.79
     36                            -0.1221 GY   0.21
     36                            -0.3662 GY   0.49
     36                            -0.6104 GY   0.79
     36                            -0.8545 GY   1.10
     36                            -1.0986 GY   1.41
     36                            -1.3428 GY   1.72
     36                            -1.5869 GY   2.04
     36                            -1.8311 GY   2.35
     36    -6.2500 GY   2.50    5.50
     36                            -1.8311 GY   5.65
     36                            -1.5869 GY   5.96
```

```
     36                            -1.3428 GY   6.28
     36                            -1.0986 GY   6.59
     36                            -0.8545 GY   6.90
     36                            -0.6104 GY   7.21
     36                            -0.3662 GY   7.51
     36                            -0.1221 GY   7.79
     26                            -0.1221 GY   0.21
     26                            -0.3662 GY   0.49
     26                            -0.6104 GY   0.79
      - WIND AND FLOOR LOAD GENERATION                          -- PAGE NO.   10
     26                            -0.8545 GY   1.10
     26                            -1.0986 GY   1.41
     26                            -1.3428 GY   1.72
     26                            -1.5869 GY   2.04
     26                            -1.8311 GY   2.35
     26                            -1.8311 GY   2.65
     26                            -1.5869 GY   2.96
     26                            -1.3428 GY   3.28
     26                            -1.0986 GY   3.59
     26                            -0.8545 GY   3.90
     26                            -0.6104 GY   4.21
     26                            -0.3662 GY   4.51
     26                            -0.1221 GY   4.79
     35                            -1.8311 GY   1.65
     35                            -1.5869 GY   1.96
     35                            -1.3428 GY   2.28
     35                            -1.0986 GY   2.59
     35                            -0.8545 GY   2.90
     35                            -0.6104 GY   3.21
     35                            -0.3662 GY   3.51
     35                            -0.1221 GY   3.79
     35    -6.2500 GY   0.00    1.50
     34                            -0.1221 GY   0.21
     34                            -0.3662 GY   0.49
     34                            -0.6104 GY   0.79
     34                            -0.8545 GY   1.10
     34                            -1.0986 GY   1.41
     34                            -1.3428 GY   1.72
     34                            -1.5869 GY   2.04
     34                            -1.8311 GY   2.35
     34    -6.2500 GY   2.50    4.00
     24                            -0.0781 GY   0.17
     24                            -0.2344 GY   0.39
     24                            -0.3906 GY   0.63
     24                            -0.5469 GY   0.88
     24                            -0.7031 GY   1.13
     24                            -0.8594 GY   1.38
     24                            -1.0156 GY   1.63
     24                            -1.1719 GY   1.88
     24                            -1.1719 GY   2.12
     24                            -1.0156 GY   2.37
     24                            -0.8594 GY   2.62
     24                            -0.7031 GY   2.87
     24                            -0.5469 GY   3.12
     24                            -0.3906 GY   3.37
     24                            -0.2344 GY   3.61
     24                            -0.0781 GY   3.83
     35                            -0.0781 GY   0.17
```

```
     35                            -0.2344 GY   0.39
     35                            -0.3906 GY   0.63
     35                            -0.5469 GY   0.88
     35                            -0.7031 GY   1.13
     35                            -0.8594 GY   1.38
     35                            -1.0156 GY   1.63
     35                            -1.1719 GY   1.88
     35                            -1.1719 GY   2.12
      - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.   11
     35                            -1.0156 GY   2.37
     35                            -0.8594 GY   2.62
     35                            -0.7031 GY   2.87
     35                            -0.5469 GY   3.12
     35                            -0.3906 GY   3.37
     35                            -0.2344 GY   3.61
     35                            -0.0781 GY   3.83
     25                            -0.0781 GY   0.17
     25                            -0.2344 GY   0.39
     25                            -0.3906 GY   0.63
     25                            -0.5469 GY   0.88
     25                            -0.7031 GY   1.13
     25                            -0.8594 GY   1.38
     25                            -1.0156 GY   1.63
     25                            -1.1719 GY   1.88
     25                            -1.1719 GY   2.12
     25                            -1.0156 GY   2.37
     25                            -0.8594 GY   2.62
     25                            -0.7031 GY   2.87
     25                            -0.5469 GY   3.12
     25                            -0.3906 GY   3.37
     25                            -0.2344 GY   3.61
     25                            -0.0781 GY   3.83
     33                            -0.0781 GY   0.17
     33                            -0.2344 GY   0.39
     33                            -0.3906 GY   0.63
     33                            -0.5469 GY   0.88
     33                            -0.7031 GY   1.13
     33                            -0.8594 GY   1.38
     33                            -1.0156 GY   1.63
     33                            -1.1719 GY   1.88
     33                            -1.1719 GY   2.12
     33                            -1.0156 GY   2.37
     33                            -0.8594 GY   2.62
     33                            -0.7031 GY   2.87
     33                            -0.5469 GY   3.12
     33                            -0.3906 GY   3.37
     33                            -0.2344 GY   3.61
     33                            -0.0781 GY   3.83
   ************ END OF DATA FROM INTERNAL STORAGE ************
    59. PRINT SUPPORT REACTION
  SUPPORT  REACTION
      - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.   12
   SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      1    1     -9.58    -11.46     -0.01      0.00     -0.01      0.00
           2      0.80     25.69      0.16      0.00     -0.00      0.00
      2    1     -6.70      3.60      0.01      0.00     -0.00      0.00
```

```
         2     -0.15     52.75      0.26      0.00     -0.00      0.00
    3    1     -5.01      7.95      0.00      0.00      0.00      0.00
         2     -0.71     49.30      1.16      0.00      0.00      0.00
    4    1    -20.54    -28.15      0.00      0.00      0.00      0.00
         2      1.72     66.85     -0.00      0.00      0.00      0.00
    5    1    -12.05     27.97      0.00      0.00      0.00      0.00
         2     -1.61    117.68     -0.00      0.00      0.00      0.00
    6    1     -9.58    -11.46      0.01      0.00      0.01      0.00
         2      0.80     25.69     -0.16      0.00      0.00      0.00
    7    1     -6.70      3.60     -0.01      0.00      0.00      0.00
         2     -0.15     52.75     -0.26      0.00      0.00      0.00
    8    1     -5.01      7.95     -0.00      0.00     -0.00      0.00
         2     -0.71     49.30     -1.16      0.00     -0.00      0.00
************** END OF LATEST ANALYSIS RESULT **************
 60. FINISH
          *********** END OF THE STAAD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:44:21 ****
   - WIND AND FLOOR LOAD GENERATION                         -- PAGE NO.   13
     ************************************************************
     *   For technical assistance on STAAD.Pro, please visit    *
     *   http://www.bentley.com/en/support/                     *
     *                                                          *
     *   Details about additional assistance from               *
     *   Bentley and Partners can be found at program menu      *
     *   Help->Technical Support                                *
     *                                                          *
     *           Copyright (c) Bentley Systems, Inc.            *
     *                http://www.bentley.com                    *
     ************************************************************
```

EX. UK-16 Time History Analysis for Forcing Function and Ground Motion

Dynamic Analysis (Time History) is performed for a 3 span beam with concentrated and distributed masses. The structure is subjected to “forcing function” and “ground motion” loading. The maxima of joint displacements, member end forces and support reactions are determined.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-16 Time History Analysis for Forcing Function and Ground Motion.STD` when you install the program.

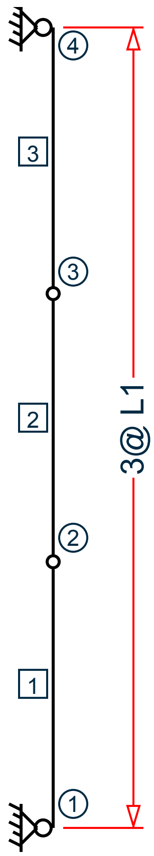


Figure 557: Example Problem No. 16

Where:

L1 = 1.2 m

```
STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
```

Every input file has to start with the word STAAD. The term PLANE signifies that the structure is a plane frame.

```
UNITS CMS KNS
```

Specifies the units to be used.

```
JOINT COORDINATES
1  0.0      0.0  0.0
2  0.0  120.0  0.0
3  0.0  240.0  0.0
4  0.0  360.0  0.0
```

Joint number followed by the X, Y and Z coordinates are specified above.

```
MEMBER INCIDENCES
1 1 2 3
```

Incidences of members 1 to 3 are specified above.

```
MEMBER PROPERTIES
1 2 3 PRIS AX 100.0 IZ 833.33
```

All the members have PRISMATIC property specification. Since this is a plane frame, Area of cross section (AX) and Moment of Inertia (IZ) about the Z axis are adequate for the analysis.

```
SUPPORTS
1 4 PINNED
```

Pinned supports are located at nodes 1 and 4.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2850
POISSON 0.17
DENSITY 25e-006
ALPHA 5e-006
DAMP 0.05
G 925
TYPE CONCRETE
STRENGTH FCU 2.75
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members.

```
UNIT NEWTON METER
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
TYPE 2 ACCELERATION
0.0 0.1 0.5 -0.25 1.0 -0.5 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
ARRIVAL TIMES
0.0
DAMPING 0.075
```

There are two stages in the command specification required for a time history analysis. The first stage is defined above. First the characteristics of the time varying load are provided. The loading type may be a forcing function (vibrating machinery) or ground motion (earthquake). The former is input in the form of time-force pairs while the latter is in the form of time-acceleration pairs. Following this data, all possible arrival times for these loads on the structure as well as the modal damping ratio are specified. In this example, the damping ratio is the same (7.5%) for all modes.

```
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 500.0
```

Load case 1 above is a static load. A uniformly distributed force of 500 Newton/m acts along the global X direction on all 3 members.

```
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 4000.0
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1
```

This is the second stage in the command specification for time history analysis. This involves the application of the time varying load on the structure. The masses that constitute the mass matrix of the structure are specified through the selfweight and joint load commands. The program will extract the lumped masses from these weights. Following that, both the `TIME LOAD` and `GROUND MOTION` are applied simultaneously.

Note: This example is only for illustration purposes and that it may be unlikely that a `TIME LOAD` and `GROUND MOTION` both act on the structure at the same time.

The Time load command is used to apply the Type 1 force, acting in the global X direction, at arrival time number 1, at nodes 2 and 3. The Ground motion, namely, the Type 2 time history loading, is also in the global X direction at arrival time 1.

```
    PERFORM ANALYSIS
```

The above command initiates the analysis process.

```
    PRINT JOINT DISPLACEMENTS
```

During the analysis, the program calculates joint displacements for every time step. The absolute maximum value of the displacement for every joint is then extracted from this joint displacement history. So, the value printed using the above command is the absolute maximum value for each of the six degrees of freedom at each node.

```
    UNIT KNS METER
    PRINT MEMBER FORCES
    PRINT SUPPORT REACTION
```

The member forces and support reactions too are calculated for every time step. For each degree of freedom, the maximum value of the member force and support reaction is extracted from these histories and reported in the output file using the above command.

```
    FINISH
```

Input File

```
STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
UNITS CMS KNS
JOINT COORDINATES
1  0.0     0.0  0.0
2  0.0   120.0  0.0
3  0.0   240.0  0.0
4  0.0   360.0  0.0
MEMBER INCIDENCES
1 1 2 3
MEMBER PROPERTIES
1 2 3 PRIS AX 100.0 IZ 833.33
SUPPORTS
1 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2850
POISSON 0.17
DENSITY 25e-006
ALPHA 5e-006
DAMP 0.05
G 925
TYPE CONCRETE
STRENGTH FCU 2.75
END DEFINE MATERIAL
CONSTANTS
```

```
MATERIAL CONCRETE ALL
UNIT NEWTON METER
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
TYPE 2 ACCELERATION
0.0 0.1 0.5 -0.25 1.0 -0.5 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
ARRIVAL TIMES
0.0
DAMPING 0.075
LOAD 1 STATIC LOAD
MEMBER LOAD
1 2 3 UNI GX 500.0
LOAD 2 TIME HISTORY LOAD
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
JOINT LOAD
2 3 FX 4000.0
TIME LOAD
2 3 FX 1 1
GROUND MOTION X 2 1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS
UNIT KNS METER
PRINT MEMBER FORCES
PRINT SUPPORT REACTION
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:23                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR TIME HISTORY ANALYSIS
INPUT FILE: UK-16 Time History Analysis for Forcing Function and Ground Motion.STD
     2. UNITS CMS KNS
     3. JOINT COORDINATES
     4. 1  0.0     0.0  0.0
     5. 2  0.0   120.0  0.0
     6. 3  0.0   240.0  0.0
     7. 4  0.0   360.0  0.0
     8. MEMBER INCIDENCES
     9. 1 1 2 3
    10. MEMBER PROPERTIES
    11. 1 2 3 PRIS AX 100.0 IZ 833.33
    12. SUPPORTS
    13. 1 4 PINNED
    14. DEFINE MATERIAL START
    15. ISOTROPIC CONCRETE
```

```
   16. E 2850
   17. POISSON 0.17
   18. DENSITY 25E-006
   19. ALPHA 5E-006
   20. DAMP 0.05
   21. G 925
   22. TYPE CONCRETE
   23. STRENGTH FCU 2.75
   24. END DEFINE MATERIAL
   25. CONSTANTS
   26. MATERIAL CONCRETE ALL
   27. UNIT NEWTON METER
   28. DEFINE TIME HISTORY
   29. TYPE 1 FORCE
   30. 0.0 -20.0 0.5 100.0 1.0 200.0 1.5 500.0 2.0 800.0 2.5 500.0 3.0 70.0
   31. TYPE 2 ACCELERATION
   32. 0.0 0.1 0.5 -0.25 1.0 -0.5 1.5 -0.9 2.0 -1.3 2.5 -1.0 3.0 -0.7
   33. ARRIVAL TIMES
   34. 0.0
   35. DAMPING 0.075
   36. LOAD 1 STATIC LOAD
   37. MEMBER LOAD
   38. 1 2 3 UNI GX 500.0
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    2
   39. LOAD 2 TIME HISTORY LOAD
   40. SELFWEIGHT X 1.0
   41. SELFWEIGHT Y 1.0
   42. JOINT LOAD
   43. 2 3 FX 4000.0
   44. TIME LOAD
   45. 2 3 FX 1 1
   46. GROUND MOTION X 2 1
   47. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS          4  NUMBER OF MEMBERS       3
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =       8
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
MORE MODES WERE REQUESTED THAN THERE ARE FREE MASSES.
NUMBER OF MODES REQUESTED             =     6
NUMBER OF EXISTING MASSES IN THE MODEL =     4
NUMBER OF MODES THAT WILL BE USED      =     4
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    3
              CALCULATED FREQUENCIES FOR LOAD CASE       2
      MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                       3.087                 0.32397
        2                      11.955                 0.08365
        3                     443.457                 0.00226
        4                     768.090                 0.00130
           MODAL WEIGHT (MODAL MASS TIMES g) IN NEWT         GENERALIZED
     MODE           X             Y             Z              WEIGHT
```

```
        1       8.600000E+03  0.000000E+00  0.000000E+00    8.600000E+03
        2       3.840329E-24  0.000000E+00  0.000000E+00    8.600000E+03
        3       0.000000E+00  6.000000E+02  0.000000E+00    6.000000E+02
        4       0.000000E+00  1.748000E-27  0.000000E+00    6.000000E+02
 MASS PARTICIPATION FACTORS
                     MASS  PARTICIPATION FACTORS IN PERCENT
                     --------------------------------------
           MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
             1   100.00   0.00   0.00  100.000    0.000    0.000
             2     0.00   0.00   0.00  100.000    0.000    0.000
             3     0.00 100.00   0.00  100.000  100.000    0.000
             4     0.00   0.00   0.00  100.000  100.000    0.000
    A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
    MODE       DAMPING
       1      0.07500000
       2      0.07500000
       3      0.07500000
       4      0.07500000
   TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
   NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =    2
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    4
   WARNING-NUMBER OF MODES LIMITED TO A FREQUENCY OF   360.0 DUE TO THE DT VALUE
ENTERED.
   TIME DURATION OF TIME HISTORY ANALYSIS =      3.000 SECONDS
   NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =     2160
    48. PRINT JOINT DISPLACEMENTS
  BASE SHEAR UNITS ARE -- NEWT METE
  MAXIMUM BASE SHEAR  X=  -2.777266E+03  Y=   0.000000E+00  Z=   0.000000E+00
      AT TIMES               2.054167           0.000000           0.000000
  JOINT    DISPLACE
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    5
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0041
           2     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0084
      2    1     0.4002    0.0000    0.0000    0.0000    0.0000   -0.0020
           2     0.8420    0.0000    0.0000    0.0000    0.0000   -0.0042
      3    1     0.4002    0.0000    0.0000    0.0000    0.0000    0.0020
           2     0.8420    0.0000    0.0000    0.0000    0.0000    0.0042
      4    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0041
           2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0084
   ************** END OF LATEST ANALYSIS RESULT **************
    49. UNIT KNS METER
    50. PRINT MEMBER FORCES
  MEMBER   FORCES
      EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    6
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
      1    1     1      0.00      0.90     0.00      0.00      0.00      -0.00
                 2      0.00     -0.30     0.00      0.00      0.00       0.72
           2     1      0.00      1.39     0.00      0.00      0.00       0.00
                 2      0.00     -1.39     0.00      0.00      0.00       1.67
      2    1     2      0.00      0.30     0.00      0.00      0.00      -0.72
                 3      0.00      0.30     0.00      0.00      0.00       0.72
           2     2      0.00      0.00     0.00      0.00      0.00      -1.67
```

```
               3      0.00      0.00     0.00      0.00      0.00       1.67
     3    1    3      0.00     -0.30     0.00      0.00      0.00      -0.72
               4      0.00      0.90     0.00      0.00      0.00       0.00
          2    3      0.00     -1.39     0.00      0.00      0.00      -1.67
               4      0.00      1.39     0.00      0.00      0.00       0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   51. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    7
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -0.90      0.00      0.00      0.00      0.00      0.00
          2     -1.39      0.00      0.00      0.00      0.00      0.00
     4    1     -0.90      0.00      0.00      0.00      0.00      0.00
          2     -1.39      0.00      0.00      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   52. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:23 ****
     EXAMPLE FOR TIME HISTORY ANALYSIS                        -- PAGE NO.    8
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-17 User-Provided Tables

The usage of user-provided steel tables is illustrated in this example for the analysis and design of a plane frame. User-provided tables allow you to specify property data for sections not found in the built-in steel section tables.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-17 User-Provided Tables.STD` when you install the program.

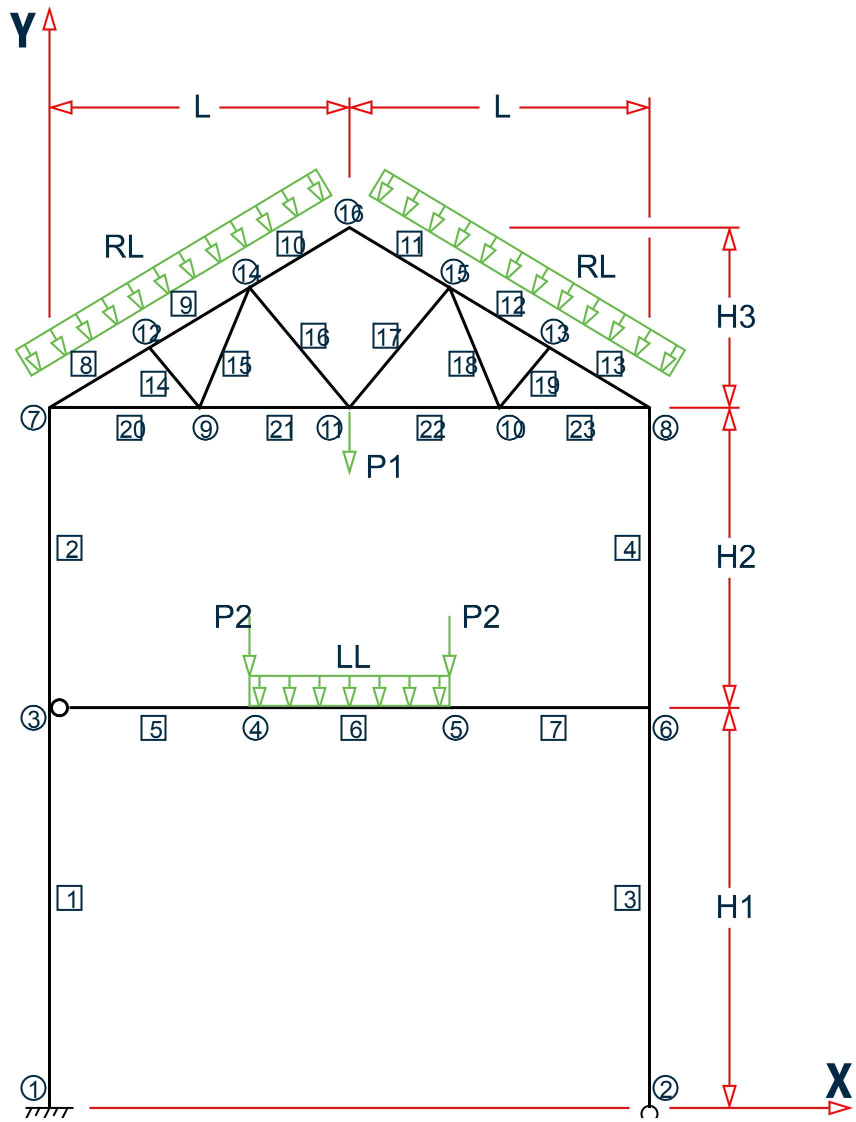


Figure 558: Example Problem No. 17

Where:

L1 = 4.5 m, L2 = 6.0 m
P = 25 kN
w = 45 kN/m

Actual input is shown in bold lettering followed by explanation.

```
STAAD PLANE EXAMPLE FOR USER TABLE
```

Every input file has to start with the command `STAAD`. The `PLANE` command is used to designate the structure as a plane frame.

```
UNIT METER KNS
```

The UNIT command sets the length and force units to be used.

```
JOINT COORDINATES
1 0. 0. ; 2 9 0 ; 3 0 6 0 6 9 6 0
7 0 10.5 ; 8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3.0 12.3 ; 15 6.0 12.3; 16 4.5 13.2
```

The above set of data is used to provide joint coordinates for the various joints of the structure. The Cartesian system is being used here. The data consists of the joint number followed by global X and Y coordinates. Note that for a space frame, the Z coordinate(s) need to be provided also.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
```

The above data set contains the member incidence information or the joint connectivity data for each member. This completes the geometry of the structure.

```
START USER TABLE
```

This command is utilized to set up a user-provided steel table. All user-provided steel tables must start with this command.

```
TABLE 1
```

Each table needs an unique numerical identification. The above command starts setting up Table no. 1. Up to twenty tables may be specified per run.

```
UNIT CM
WIDE FLANGE
```

This command is used to specify the section-type as WIDE FLANGE in this table. Note that several section-types such as WIDE FLANGE, CHANNEL, ANGLE, TEE etc. are available for specification (see TR.19 User Steel Table Specification (on page 2241)).

```
BEAM250
32.2 25.5 0.6 10.2 0.85 3400 150 6.2 15.7 15.4
BEAM300
47.5 30.4 0.72 12.4 1.07 7160 335 14.3 21.9 23.8
BEAM350
64.6 35.6 0.73 17.2 1.15 14150 970 22.8 26.0 35.5
```

The above data set is used to specify the properties of three wide flange sections. The data for each section consists of two parts. In the first line, the section-name is provided. You are allowed to provide any section name within twelve characters. The second line contains the section properties required for the particular section-type. Each section-type requires a certain number of data (area of cross-section, depth, moment of inertias etc.) provided in a certain order. For example, in this case, for wide flanges, ten different properties are required.

```
TABLE 2
UNIT CM
ANGLES
L30305
3.0 3.0 0.5 0.58 1.0 1.0
L40405
```

```
4.0 4.0 0.5 0.78 1.33 1.33
L50505
5.0 5.0 0.5 0.98 1.67 1.67
```

The above command and data lines set up another user provided table consisting of angle sections.

```
END
```

This command signifies the end of the user provided table data set. All user provided table related input must be terminated with this command.

```
MEMBER PROPERTIES
1 3 4 UPT 1 BEAM350
2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
8 TO 13 UPT 1 BEAM250
14 TO 19 UPT 2 L30305
20 TO 23 UPT 2 L40405
```

In the above command lines, the member properties are being assigned from the user provided tables created earlier. The word UPT signifies that the properties are from the user-provided table. This is followed by the table number and then the section name as specified in the use- provided table. The numbers 1 or 2 following the word UPT indicate the table from which section names are fetched.

```
MEMBER TRUSS
14 TO 23
```

The above command is used to designate members 14 to 23 as truss members.

```
MEMBER RELEASE
5 START MZ
```

The MEMBER RELEASE command is used to release the MZ moment at the start joint of member no. 5.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
```

The above command set is used to specify modulus of elasticity, density, Poisson's ratio and beta angle values. Built-in default value of steel is used for the material constants.

```
UNIT KNS METER
```

The force unit is reset to KNS, and length unit to METER using this command.

```
SUPPORT
1 FIXED ; 2 PINNED
```

The above command set is used to designate supports. Here, joint 1 is designated as a fixed support and joint 2 is designated as a pinned support.

```
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
```

```
JOINT LOAD
4 5 FY -65. ; 11 FY -155.
MEMB LOAD
8 TO 13 UNI Y –13.5 ; 6 UNI GY –17.5
```

The above command set is used to specify the loadings on the structure. In this case, dead and live loads are provided through load case 1. It consists of selfweight, concentrated loads at joints 4, 5 and 11, and distributed loads on some members.

```
PERFORM ANALYSIS
```

This command instructs the program to execute the analysis at this point.

```
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 1.0 ALL
NSF 0.85 ALL
KY 1.2 MEMB 3 4
```

The above commands are used to specify parameters for steel design.

```
CHECK CODE MEMBER 3 19
SELECT MEMBER 20
```

This command will perform a code check on members 3 and 19 per the Eurocode 3 steel design code with the UK National Annex. A member selection too is performed for member 20. For each member, the member selection will be performed from the table that was originally used for the specification of the member property. In this case, the selection will be from the respective user tables from which the properties were initially assigned. It may be noted that properties may be provided (and selection may be performed) from built-in steel tables and user provided tables in the same data file.

```
FINISH
```

This command terminates a STAAD run.

Input File

```
STAAD PLANE EXAMPLE FOR USER TABLE
UNIT METER KNS
JOINT COORD
1 0. 0. ; 2 9 0 ; 3 0 6 0 6 9 6 0
7 0 10.5 ; 8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
14 3.0 12.3 ; 15 6.0 12.3 ; 16 4.5 13.2
MEMBER INCIDENCES
1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
18 10 15 ; 19 10 13 ; 20 7 9
21 9 11 ; 22 10 11 ; 23 8 10
START USER TABLE
TABLE 1
UNIT CM
WIDE FLANGE
BEAM250
32.2 25.5 0.6 10.2 0.85 3400 150 6.2 15.7 15.4
BEAM300
47.5 30.4 0.72 12.4 1.07 7160 335 14.3 21.9 23.8
```

```
BEAM350
64.6 35.6 0.73 17.2 1.15 14150 970 22.8 26.0 35.5
TABLE 2
UNIT CM
ANGLES
L30305
3.0 3.0 0.5 0.58 1.0 1.0
L40405
4.0 4.0 0.5 0.78 1.33 1.33
L50505
5.0 5.0 0.5 0.98 1.67 1.67
END
MEMBER PROPERTIES
1 3 4 UPT 1 BEAM350
2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
8 TO 13 UPT 1 BEAM250
14 TO 19 UPT 2 L30305
20 TO 23 UPT 2 L40405
*MEMBER TRUSS
*14 TO 23
MEMB RELEASE
5 START MZ
14 to 23 start MPY 0.99 MPZ 0.99
14 to 23 end MPY 0.99 MPZ 0.99
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.68e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 90.0 MEMB 3 4
UNIT KNS METER
SUPPORT
1 FIXED ; 2 PINNED
LOADING 1 DEAD AND LIVE LOAD
SELFWEIGHT Y -1.0
JOINT LOAD
4 5 FY -65. ; 11 FY -155
MEMB LOAD
8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
PERFORM ANALYSIS
PARAMETER
CODE EN 1993-1-1:2005
NA 1
BEAM 3 ALL
NSF 0.85 ALL
KY 1.2 MEMB 3 4
CHECK CODE MEMBER 3 19
SELECT MEMB 20
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
            ****************************************************
            *                                                  *
            *           STAAD.Pro CONNECT Edition              *
            *           Version  22.10.00.***                  *
            *           Proprietary Program of                 *
            *           Bentley Systems, Inc.                  *
            *           Date=    MAR 24, 2022                  *
            *           Time=     9:44:25                      *
            *                                                  *
            *  Licensed to: Bentley Systems Inc                *
            ****************************************************
     1. STAAD PLANE EXAMPLE FOR USER TABLE
INPUT FILE: UK-17 User-Provided Tables.STD
     2. UNIT METER KNS
     3. JOINT COORD
     4. 1 0. 0. ; 2 9 0 ; 3 0 6 0 6 9 6 0
     5. 7 0 10.5 ; 8 9 10.5 ; 9 2.25 10.5 ; 10 6.75 10.5
     6. 11 4.5 10.5 ; 12 1.5 11.4 ; 13 7.5 11.4
     7. 14 3.0 12.3 ; 15 6.0 12.3 ; 16 4.5 13.2
     8. MEMBER INCIDENCES
     9. 1 1 3 ; 2 3 7 ; 3 2 6 ; 4 6 8 ; 5 3 4
    10. 6 4 5 ; 7 5 6 ; 8 7 12 ; 9 12 14
    11. 10 14 16 ; 11 15 16 ; 12 13 15 ; 13 8 13
    12. 14 9 12 ; 15 9 14 ; 16 11 14 ; 17 11 15
    13. 18 10 15 ; 19 10 13 ; 20 7 9
    14. 21 9 11 ; 22 10 11 ; 23 8 10
    15. START USER TABLE
    16. TABLE 1
    17. UNIT CM
    18. WIDE FLANGE
    19. BEAM250
    20. 32.2 25.5 0.6 10.2 0.85 3400 150 6.2 15.7 15.4
    21. BEAM300
    22. 47.5 30.4 0.72 12.4 1.07 7160 335 14.3 21.9 23.8
    23. BEAM350
    24. 64.6 35.6 0.73 17.2 1.15 14150 970 22.8 26.0 35.5
    25. TABLE 2
    26. UNIT CM
    27. ANGLES
    28. L30305
    29. 3.0 3.0 0.5 0.58 1.0 1.0
    30. L40405
    31. 4.0 4.0 0.5 0.78 1.33 1.33
    32. L50505
    33. 5.0 5.0 0.5 0.98 1.67 1.67
    34. END
    35. MEMBER PROPERTIES
    36. 1 3 4 UPT 1 BEAM350
    37. 2 UPT 1 BEAM300 ; 5 6 7 UPT 1 BEAM250
    38. 8 TO 13 UPT 1 BEAM250
      EXAMPLE FOR USER TABLE                                 -- PAGE NO.    2
    39. 14 TO 19 UPT 2 L30305
    40. 20 TO 23 UPT 2 L40405
    41. *MEMBER TRUSS
```

```
   42. *14 TO 23
   43. MEMB RELEASE
   44. 5 START MZ
   45. 14 TO 23 START MPY 0.99 MPZ 0.99
   46. 14 TO 23 END MPY 0.99 MPZ 0.99
   47. UNIT MMS
   48. DEFINE MATERIAL START
   49. ISOTROPIC STEEL
   50. E 210
   51. POISSON 0.3
   52. DENSITY 7.68E-008
   53. ALPHA 6E-006
   54. DAMP 0.03
   55. TYPE STEEL
   56. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
   57. END DEFINE MATERIAL
   58. CONSTANTS
   59. MATERIAL STEEL ALL
   60. BETA 90.0 MEMB 3 4
   61. UNIT KNS METER
   62. SUPPORT
   63. 1 FIXED ; 2 PINNED
   64. LOADING 1 DEAD AND LIVE LOAD
   65. SELFWEIGHT Y -1.0
   66. JOINT LOAD
   67. 4 5 FY -65. ; 11 FY -155
   68. MEMB LOAD
   69. 8 TO 13 UNI Y -13.5 ; 6 UNI GY -17.5
   70. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         16  NUMBER OF MEMBERS      23
    NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      43
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   71. PARAMETER
   72. CODE EN 1993-1-1:2005
     EXAMPLE FOR USER TABLE                                 -- PAGE NO.    3
   73. NA 1
   74. BEAM 3 ALL
   75. NSF 0.85 ALL
   76. KY 1.2 MEMB 3 4
   77. CHECK CODE MEMBER 3 19
STEEL DESIGN
                        STAAD.PRO CODE CHECKING - BS EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     EXAMPLE FOR USER TABLE                                 -- PAGE NO.    4
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY             MZ       LOCATION
=======================================================================
*     3 ST   BEAM350     (UPT)
                          FAIL     EC-6.3.3-662       2.339         1
```

```
                     243.71 C        52.86          0.00        6.00
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY            MZ       LOCATION
=======================================================================
*    19 ST   L30305      (UPT)
                          FAIL     EC-6.3.3-661       1.557         1
                      18.18 C        0.00          -0.00        0.59
  ************** END OF TABULATED RESULT OF DESIGN **************
   78. SELECT MEMB 20
STEEL DESIGN
                        STAAD.PRO MEMBER SELECTION - BS EN 1993-1-1:2005
                        ********************************************
                        NATIONAL ANNEX - NA to BS EN 1993-1-1:2005
PROGRAM CODE REVISION V1.14 BS_EC3_2005/1
     EXAMPLE FOR USER TABLE                                  -- PAGE NO.    5
ALL UNITS ARE - KNS  METE (UNLESS OTHERWISE Noted)
MEMBER     TABLE       RESULT/   CRITICAL COND/     RATIO/     LOADING/
                         FX            MY            MZ       LOCATION
=======================================================================
     20 ST   L40405      (UPT)
                          PASS     EC-6.2.9.2/3       0.763         1
                      62.38 T        0.00          -0.02        1.12
  ************** END OF TABULATED RESULT OF DESIGN **************
   79. FINISH
  *****************************************************************************
  **WARNING** SOME MEMBER SIZES HAVE CHANGED SINCE LAST ANALYSIS.
              IN THE POST PROCESSOR, MEMBER QUERIES WILL USE THE LAST
              ANALYSIS FORCES WITH THE UPDATED MEMBER SIZES.
              TO CORRECT THIS INCONSISTENCY, PLEASE DO ONE MORE ANALYSIS.
              FROM THE UPPER MENU, PRESS RESULTS, UPDATE PROPERTIES, THEN
              FILE SAVE; THEN ANALYZE AGAIN WITHOUT THE GROUP OR SELECT
              COMMANDS.
  *****************************************************************************
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:26 ****
     EXAMPLE FOR USER TABLE                                  -- PAGE NO.    6
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-18 Stress Calculation for Plate Elements

This is an example which demonstrates the calculation of principal stresses on a finite element.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-18 Stress Calculation for Plate Elements.STD` when you install the program.

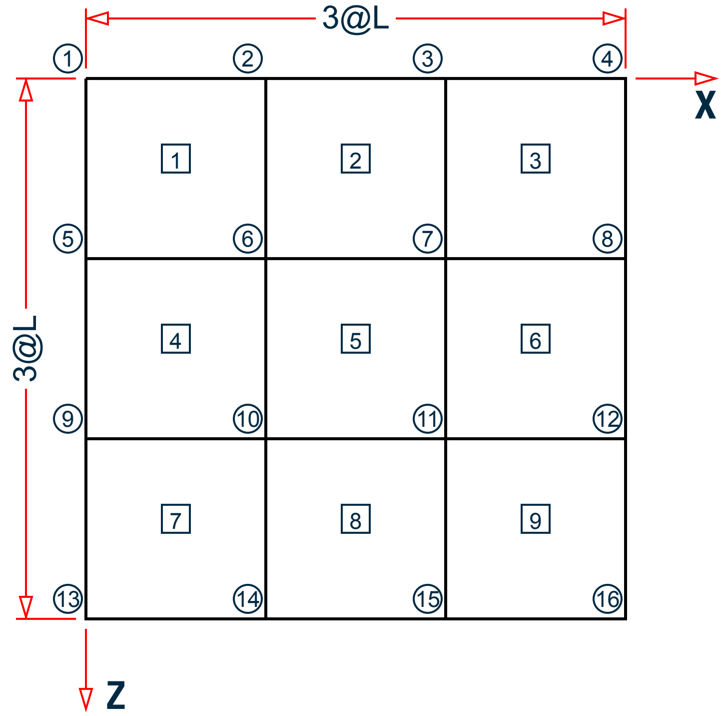


Figure 559: Example Problem No. 18

Where:

L = 3 m

Fixed Supports at Joints 1, 2, 3, 4, 5, 9, 13

Load intensity = 2000 Kn/sq.m in - Y direction

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
```

Every input has to start with the term STAAD. The word SPACE signifies that the structure is a space frame (3-D structure).

```
UNIT METER KNS
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 0 0 4 9 0 0
REPEAT 3 0 0 3
```

Joint number followed by X, Y and Z coordinates are provided above. The REPEAT command is used to generate coordinates of joints 5 to 16 based on the pattern of joints 1 to 4.

```
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
```

Element connectivities of elements 1 to 3 are defined first, based on which, the connectivities of elements 4 to 9 are generated.

```
ELEMENT PROPERTIES
1 TO 9 THICK 0.25
```

Elements 1 to 9 have a thickness of 0.25 m.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 TO 4 5 9 13 FIXED
```

Fixed support conditions exist at the above mentioned joints.

```
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -2000.0
```

A uniform pressure of 2000 Kn/sq.m is applied on all the elements. In the absence of an explicit direction specification, the load is assumed to act along the local Z axis. The negative value indicates that the load acts opposite to the positive direction of the local Z.

```
PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```
PRINT SUPPORT REACTION
```

The above command is self-explanatory.

```
UNIT MMS
PRINT ELEMENT STRESSES LIST 4
```

Element stresses at the centroid of the element are printed using the above command. The output includes membrane stresses, shear stresses, bending moments per unit width and principal stresses. The change of length unit from meter to mms indicates that the values will be printed in KN and MMs units.

```
FINISH
```

The STAAD run is terminated.

Calculation of principal stresses for element 4

Calculations are presented for the top surface only.

SX = 0.0 kN/mm^2

SY = 0.0 kN/mm^2

SXY = 0.0 kN/mm^2

MX = 2,111.84 kN-mm/mm

MY = 10,726.22 kN-mm/m

MXY = 4,553.97 kN-mm/mm

S = 1/6t^2 = 1/(6 ·250^2) = 10,416.67mm^2 (Section Modulus)

σ_x = SX + MX/S = 0.0 + 2,111.84/10,416.67 = 0.2027 kN/mm^2

σ_y = SY + MY/S = 0.0 + 10,726.22/10,416.67 = 1.0297 kN/mm^2

τ_{xy} = SXY + MXY/S = 0.0 + 4,553.97/10,416.67 = 0.4372 kN/mm^2

$$T_{Max} = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2} = \sqrt{\frac{(0.2027 - 1.0297)^2}{4} + 0.4372^2} = 0.6018 KN \Big/ mm^2$$

$S_{Max} = (\sigma_x + \sigma_y)/2 + T_{Max}$ = (0.2027 + 1.0297)/2 + 0.6018 = 1.218 kN/mm^2

Say 1.22 kN/mm^2

$S_{Min} = (\sigma_x + \sigma_y)/2 - T_{Max}$ = (0.2027 + 1.0297)/2 - 0.6018 = 0.0144 kN/mm^2

Say 0.01 kN/ mm^2

$$\text{Angle} = \frac{1}{2}\tan^{-1}\left(\frac{2\tau_{xy}}{\sigma_x - \sigma_y}\right) = \frac{1}{2}\tan^{-1}\left\{\frac{2(0.4372)}{0.2027 - 1.0297}\right\} = -23.30\,°$$

VON =

$$0.707\sqrt{(S_{Max} - S_{Min})^2 + S_{Max}^2 + S_{Min}^2} = 0.707\sqrt{(1.218 - 0.0144)^2 + 1.218^2 + 0.0144^2} = 1.2107 kN \Big/ mm^2$$

Input File

```
STAAD SPACE SAMPLE CALCULATION FOR
* ELEMENT STRESSES
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 4 9 0 0
REPEAT 3 0 0 3
ELEMENT INCIDENCE
1 1 5 6 2 TO 3
REPEAT 2 3 4
ELEMENT PROPERTIES
1 TO 9 THICK 0.25
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
```

```
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
SUPPORT
1 TO 4 5 9 13 FIXED
LOAD 1
ELEMENT LOAD
1 TO 9 PRESSURE -2000.0
PERFORM ANALYSIS
PRINT SUPPORT REACTION
UNIT MMS
PRINT ELEMENT STRESSES LIST 4
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:28                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE SAMPLE CALCULATION FOR
INPUT FILE: UK-18 Stress Calculation for Plate Elements.STD
     2. * ELEMENT STRESSES
     3. UNIT METER KNS
     4. JOINT COORDINATES
     5. 1 0 0 0 4 9 0 0
     6. REPEAT 3 0 0 3
     7. ELEMENT INCIDENCE
     8. 1 1 5 6 2 TO 3
     9. REPEAT 2 3 4
    10. ELEMENT PROPERTIES
    11. 1 TO 9 THICK 0.25
    12. UNIT MMS
    13. DEFINE MATERIAL START
    14. ISOTROPIC CONCRETE
    15. E 21.0
    16. POISSON 0.17
    17. DENSITY 2.36158E-008
    18. ALPHA 5E-006
    19. DAMP 0.05
    20. G 9.25
    21. TYPE CONCRETE
    22. STRENGTH FCU 0.0275
    23. END DEFINE MATERIAL
    24. CONSTANTS
    25. MATERIAL CONCRETE ALL
    26. UNIT METER
    27. SUPPORT
    28. 1 TO 4 5 9 13 FIXED
```

```
   29. LOAD 1
   30. ELEMENT LOAD
   31. 1 TO 9 PRESSURE -2000.0
   32. PERFORM ANALYSIS
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    2
   * ELEMENT STRESSES
         P R O B L E M   S T A T I S T I C S
         -----------------------------------
    NUMBER OF JOINTS         16  NUMBER OF MEMBERS       0
    NUMBER OF PLATES          9  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      7
        Using 64-bit analysis engine.
        SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      54
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   33. PRINT SUPPORT REACTION
 SUPPORT  REACTION
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    3
   * ELEMENT STRESSES
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      0.00  -1291.48      0.00   -348.06      0.00    348.06
     2    1      0.00   8700.82      0.00 -26485.08      0.00   -384.11
     3    1      0.00  37742.16      0.00 -87763.47      0.00   3265.14
     4    1      0.00  35202.76      0.00 -66094.57      0.00 -23952.47
     5    1      0.00   8700.82      0.00    384.11      0.00  26485.08
     9    1      0.00  37742.16      0.00  -3265.14      0.00  87763.47
    13    1      0.00  35202.76      0.00  23952.47      0.00  66094.57
  ************** END OF LATEST ANALYSIS RESULT **************
   34. UNIT MMS
   35. PRINT ELEMENT STRESSES LIST 4
 ELEMENT  STRESSES LIST     4
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    4
   * ELEMENT STRESSES
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KNS  MMS
  ----------------
         STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
     4      1         0.02       -0.02     2080.05    10634.76     4648.47
                      1.21        1.21        0.00        0.00        0.00
                      1.22        1.22
      TOP : SMAX=       1.22 SMIN=       0.00 TMAX=       0.61 ANGLE= 66.3
      BOTT: SMAX=      -0.00 SMIN=      -1.22 TMAX=       0.61 ANGLE=-23.7
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         1.216739E+00 -1.216739E+00  6.064286E-01  1.214803E+00  1.216739E+00
PLATE NO.       4             4             4             4             4
CASE  NO.       1             1             1             1             1
  ********************END OF ELEMENT FORCES********************
   36. FINISH
     SAMPLE CALCULATION FOR                                  -- PAGE NO.    5
   * ELEMENT STRESSES
```

```
            *********** END OF THE STAADD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:28 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-19 Inclined Supports

This example demonstrates the usage of inclined supports. The word `INCLINED` refers to the fact that the restraints at a joint where such a support is specified are along a user-specified axis system instead of along the default directions of the global axis system. STAAD.Pro offers a few different methods for assigning inclined supports, and we examine those in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-19 Inclined Supports.STD` when you install the program.

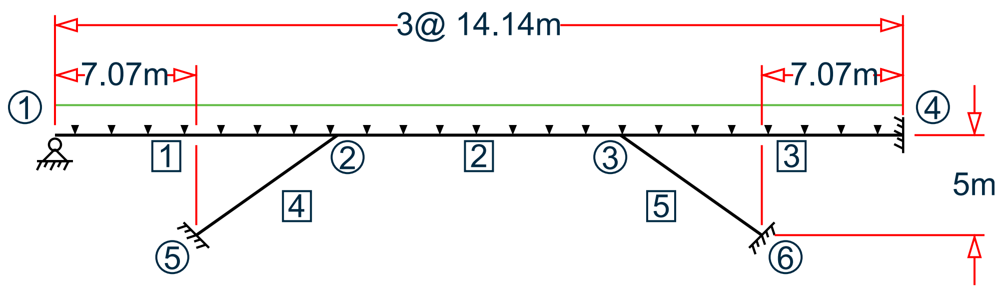


Figure 560: Example Problem No. 19

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE
INPUT WIDTH 79
```

Every input has to start with the term `STAAD`. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y, and Z coordinates.

```
UNIT METER KN
```

Defines the input units for the data that follows.

```
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
```

Joint number followed by X, Y and Z coordinates are provided above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
```

Defines the members by the joints to which they are connected.

```
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
```

Properties for all members of the model are provided using the `PRISMATIC` option (abbreviated to `PRIS` here). `YD` and `ZD` stand for depth and width. If `ZD` is not provided, a circular shape with diameter = YD is assumed for that cross section. All properties required for the analysis, such as, Area, Moments of Inertia, etc. are calculated automatically from these dimensions unless these are explicitly defined. The values are provided in MMS unit.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.72
POISSON 0.17
DENSITY 2.35615e-008
ALPHA 5e-006
DAMP 0.05
G 9.281
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

Material constants like E (modulus of elasticity) and Poisson's ratio are specified following the command `CONSTANTS`.

```
UNIT METER KN
SUPPORTS
5 INCLINED REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INCLINED REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INCLINED 1 0 1 FIXED BUT FX MX MY MZ
```

We assign supports (restraints) at 4 nodes - 5, 6, 1 and 4. For 3 of those, namely, 5, 6 and 4, the node number is followed by the keyword `INCLINED`, signifying that an INCLINED support is defined there. For the remaining one - node 1 - that keyword is missing. Hence, the support at node 1 is a global direction support.

The most important aspect of inclined supports is their axis system. Each node where an inclined support is defined has its own distinct local X, local Y and local Z axes. In order to define the axis system, we first have to define a datum point. The support node and the datum point together help define the axis system.

Three different methods are shown in the above 3 instances for defining the datum point.

- At node 5, notice the keyword `REF` followed by the numbers (10,5,10). This means that the datum point associated with node 5 is one which has the global coordinates of (10m, 5m, 10m). Coincidentally, this happens to be node 2.
- At node 6, the keyword `REFJT` is used followed by the number 3. This means that the datum point for support node 6 is the joint number 3 of the model. The coordinates of the datum point are hence those of node 3, namely, (20m, 5m and 20m).

- At node 4, the word `INCLINED` is merely followed by 3 numbers (1,0,1). In the absence of the words REF and REFJT, the program sets the datum point to be the following. It takes the coordinates of node 4, which are (30m,5m,30m) and adds to them, the 3 numbers which comes after the word INCLINED. Thus, the datum point becomes (31m, 5m and 31m).

Once the datum point is established, the local axis system is defined as follows. Local X is a straight line (vector) pointing from the support node towards the datum point. Local Z is the vector obtained by the cross product of local X and the global Y axis (unless the SET Z UP command is used in which case one would use global Z instead of global Y and that would yield local Y). Local Y is the vector resulting from the cross product of local Z and local X. The right hand rule must be used when performing these cross products.

Notice the unique nature of these datum points. The one for node 5 tells us that a line connecting nodes 5 to 2 is the local X axis, and is hence along the axis of member 4. By defining a KFX spring at that one, we are saying that the lower end of member 4 can move along its axis like the piston of a car engine. Think of a pile bored into rock with a certain amount of freedom to expand and contract axially.

The same is true for the support at the bottom of member 5. The local X axis of that support is along the axis of member 5. That also happens to be the case for the supported end of member 3. The line going from node 4 to the datum point (31,5,31) happens to be coincident with the axis of the member, or the traffic direction. The expression FIXED BUT FX MX MY MZ for that support indicates that it is free to translate along local X, suggesting that it is an expansion joint - free to expand or contract along the axis of member 3.

Since MX, MY, and MZ are all released at these supports, no moment will be resisted by these supports.

```
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
```

Three load cases followed by the instruction for the type of analysis are specified. The `PRINT STATICS CHECK` option will instruct the program to produce a report consisting of total applied load versus total reactions from the supports for each primary load case.

```
PRINT SUPPORT REACTION
```

By default, support reactions are printed in the global axis directions. The above command is an instruction for such a report.

```
SET INCLINED REACTION
PRINT SUPPORT REACTION
```

Just earlier, we saw how to obtain support reactions in the global axis system. What if we need them in the inclined axis system? The "SET INCLINED REACTION" is a switch for that purpose. It tells the program that reactions should be reported in the inclined axis system instead of the global axis system. This has to be followed by the PRINT SUPPORT REACTIONS command.

```
PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH
```

Member forces are reported in the local axis system of the members. Joint displacements at all joints are reported in the global axis system. Following this, the STAAD run is terminated.

Input File

```
STAAD SPACE
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3;
UNIT MMS KN
MEMBER PROPERTY AMERICAN
4 5 PRIS YD 800
1 TO 3 PRIS YD 750 ZD 500
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER KN
SUPPORTS
5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
1 PINNED
4 INC 1 0 1 FIXED BUT FX MX MY MZ
LOAD 1 DEAD LOAD
SELFWEIGHT Y -1.2
LOAD 2 LIVE LOAD
MEMBER LOAD
1 TO 3 UNI GY -6
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STATICS CHECK
PRINT SUPPORT REACTION
SET INCLINED REACTION
PRINT SUPPORT REACTION
PRINT MEMBER FORCES
PRINT JOINT DISP
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
```

```
             *           Time=     9:44:30                    *
             *                                                *
             *  Licensed to: Bentley Systems Inc              *
             **************************************************
     1. STAAD SPACE
INPUT FILE: UK-19 Inclined Supports.STD
     2. INPUT WIDTH 79
     3. UNIT METER KN
     4. JOINT COORDINATES
     5. 1 0 5 0; 2 10 5 10; 3 20 5 20; 4 30 5 30; 5 5 0 5; 6 25 0 25
     6. MEMBER INCIDENCES
     7. 1 1 2; 2 2 3; 3 3 4; 4 5 2; 5 6 3
     8. UNIT MMS KN
     9. MEMBER PROPERTY AMERICAN
    10. 4 5 PRIS YD 800
    11. 1 TO 3 PRIS YD 750 ZD 500
    12. DEFINE MATERIAL START
    13. ISOTROPIC CONCRETE
    14. E 21.0
    15. POISSON 0.17
    16. DENSITY 2.36158E-008
    17. ALPHA 5E-006
    18. DAMP 0.05
    19. G 9.25
    20. TYPE CONCRETE
    21. STRENGTH FCU 0.0275
    22. END DEFINE MATERIAL
    23. CONSTANTS
    24. MATERIAL CONCRETE ALL
    25. UNIT METER KN
    26. SUPPORTS
    27. 5 INC REF 10 5 10 FIXED BUT MX MY MZ KFX 30000
    28. 6 INC REFJT 3 FIXED BUT MX MY MZ KFX 30000
    29. 1 PINNED
    30. 4 INC 1 0 1 FIXED BUT FX MX MY MZ
    31. LOAD 1 DEAD LOAD
    32. SELFWEIGHT Y -1.2
    33. LOAD 2 LIVE LOAD
    34. MEMBER LOAD
    35. 1 TO 3 UNI GY -6
    36. LOAD COMB 3
    37. 1 1.0 2 1.0
    38. PERFORM ANALYSIS PRINT STATICS CHECK
      STAAD SPACE                                              -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS       5
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      27
   TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
      STAAD SPACE                                              -- PAGE NO.    3
          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
          DEAD LOAD
           CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
         (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
```

```
                       X =  0.150000000E+02
                       Y =  0.411580006E+01
                       Z =  0.150000000E+02
 TOTAL APPLIED LOAD     1
   ***TOTAL APPLIED LOAD ( KN   METE ) SUMMARY (LOADING     1 )
       SUMMATION FORCE-X =           0.00
       SUMMATION FORCE-Y =        -697.60
       SUMMATION FORCE-Z =           0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=       10463.94  MY=           0.00  MZ=      -10463.94
 TOTAL REACTION LOAD    1
   ***TOTAL REACTION LOAD( KN   METE ) SUMMARY (LOADING     1 )
       SUMMATION FORCE-X =          -0.00
       SUMMATION FORCE-Y =         697.60
       SUMMATION FORCE-Z =          -0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=      -10463.94  MY=          -0.00  MZ=       10463.94
   MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      1)
             MAXIMUMS    AT NODE
      X = -8.02030E-01       5
      Y = -2.50768E+00       3
      Z = -8.02030E-01       5
      RX= -2.71938E-03       4
      RY=  8.02194E-18       1
      RZ=  2.71938E-03       4
          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
          LIVE LOAD
           CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
         (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.149999999E+02
                       Y =  0.500000017E+01
                       Z =  0.149999999E+02
      STAAD SPACE                                            -- PAGE NO.    4
 TOTAL APPLIED LOAD     2
   ***TOTAL APPLIED LOAD ( KN   METE ) SUMMARY (LOADING     2 )
       SUMMATION FORCE-X =           0.00
       SUMMATION FORCE-Y =        -254.56
       SUMMATION FORCE-Z =           0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=        3818.38  MY=           0.00  MZ=       -3818.38
 TOTAL REACTION LOAD    2
   ***TOTAL REACTION LOAD( KN   METE ) SUMMARY (LOADING     2 )
       SUMMATION FORCE-X =          -0.00
       SUMMATION FORCE-Y =         254.56
       SUMMATION FORCE-Z =          -0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=       -3818.38  MY=          -0.00  MZ=        3818.38
   MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      2)
             MAXIMUMS    AT NODE
      X = -2.97766E-01       5
      Y = -9.34280E-01       3
      Z = -2.97766E-01       5
      RX= -1.21481E-03       4
      RY= -3.94177E-18       4
      RZ=  1.21481E-03       4
   ************ END OF DATA FROM INTERNAL STORAGE ************
    39. PRINT SUPPORT REACTION
  SUPPORT  REACTION
```

```
     STAADSPACE                                              -- PAGE NO.    5
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     5    1    216.27    289.28    216.27      0.00      0.00      0.00
          2     86.61     94.78     86.61      0.00      0.00      0.00
          3    302.88    384.06    302.88      0.00      0.00      0.00
     6    1   -213.07    287.50   -213.07      0.00      0.00      0.00
          2    -85.33     94.06    -85.33      0.00      0.00      0.00
          3   -298.40    381.56   -298.40      0.00      0.00      0.00
     1    1     -3.20     60.34     -3.20      0.00      0.00      0.00
          2     -1.28     32.84     -1.28      0.00      0.00      0.00
          3     -4.48     93.18     -4.48      0.00      0.00      0.00
     4    1      0.00     60.47      0.00      0.00      0.00      0.00
          2      0.00     32.89      0.00      0.00      0.00      0.00
          3      0.00     93.36      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    40. SET INCLINED REACTION
    41. PRINT SUPPORT REACTION
  SUPPORT  REACTION
     STAAD SPACE                                              -- PAGE NO.    6
   SUPPORT REACTIONS -UNIT KN   METE    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     5    1    416.75     59.61      0.00      0.00      0.00      0.00
          2    154.72      6.67      0.00      0.00      0.00      0.00
          3    571.47     66.28      0.00      0.00      0.00      0.00
     6    1    412.02     60.77      0.00      0.00      0.00      0.00
          2    152.83      7.13      0.00      0.00      0.00      0.00
          3    564.85     67.90      0.00      0.00      0.00      0.00
     1    1     -3.20     60.34     -3.20      0.00      0.00      0.00
          2     -1.28     32.84     -1.28      0.00      0.00      0.00
          3     -4.48     93.18     -4.48      0.00      0.00      0.00
     4    1      0.00     60.47      0.00      0.00      0.00      0.00
          2      0.00     32.89      0.00      0.00      0.00      0.00
          3      0.00     93.36      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    42. PRINT MEMBER FORCES
  MEMBER   FORCES
     STAAD SPACE                                              -- PAGE NO.    7
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KN   METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1      -4.52     60.34     0.00      0.00     -0.00      -0.00
                2       4.52     89.95    -0.00      0.00     -0.00    -209.31
          2     1      -1.81     32.84     0.00     -0.00     -0.00       0.00
                2       1.81     52.02    -0.00      0.00     -0.00    -135.62
          3     1      -6.33     93.18     0.00     -0.00     -0.00      -0.00
                2       6.33    141.96    -0.00      0.00     -0.00    -344.93
     2    1     2     301.33     75.98    -0.00      0.00      0.00     129.19
                3    -301.33     74.31     0.00     -0.00     -0.00    -117.44
          2     2     120.67     42.76    -0.00      0.00      0.00      77.86
                3    -120.67     42.09     0.00     -0.00     -0.00     -73.16
          3     2     422.00    118.73    -0.00      0.00      0.00     207.05
                3    -422.00    116.41     0.00     -0.00     -0.00    -190.59
     3    1     3       0.00     89.82    -0.00      0.00      0.00     207.54
                4       0.00     60.47     0.00      0.00      0.00       0.00
```

```
        2     3      0.00     51.97    -0.00     -0.00      0.00    134.91
              4     -0.00     32.89     0.00      0.00      0.00      0.00
        3     3      0.00    141.79    -0.00     -0.00      0.00    342.45
              4     -0.00     93.36     0.00      0.00      0.00      0.00
   4    1     5    416.75     59.61     0.00     -0.00      0.00     -0.00
              2   -345.52     41.11    -0.00      0.00     -0.00     80.12
        2     5    154.72      6.67     0.00      0.00      0.00      0.00
              2   -154.72     -6.67    -0.00      0.00      0.00     57.76
        3     5    571.47     66.28     0.00     -0.00      0.00      0.00
              2   -500.25     34.44    -0.00      0.00     -0.00    137.88
   5    1     6    412.02     60.77     0.00     -0.00     -0.00     -0.00
              3   -340.80     39.96    -0.00      0.00     -0.00     90.10
        2     6    152.83      7.13     0.00     -0.00     -0.00      0.00
              3   -152.83     -7.13    -0.00      0.00     -0.00     61.75
        3     6    564.85     67.90     0.00     -0.00     -0.00      0.00
              3   -493.63     32.83    -0.00      0.00     -0.00    151.86
 ************** END OF LATEST ANALYSIS RESULT **************
  43. PRINT JOINT DISP
    STAAD SPACE                                              -- PAGE NO.    8
JOINT    DISP
    STAAD SPACE                                              -- PAGE NO.    9
 JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
 ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    1    1     0.0000    0.0000    0.0000    0.0027    0.0000   -0.0027
         2     0.0000    0.0000    0.0000    0.0012    0.0000   -0.0012
         3     0.0000    0.0000    0.0000    0.0039    0.0000   -0.0039
    2    1     0.0006   -2.4614    0.0006    0.0007    0.0000   -0.0007
         2     0.0002   -0.9157    0.0002    0.0003    0.0000   -0.0003
         3     0.0008   -3.3771    0.0008    0.0011    0.0000   -0.0011
    3    1    -0.0377   -2.5077   -0.0377   -0.0007    0.0000    0.0007
         2    -0.0151   -0.9343   -0.0151   -0.0003    0.0000    0.0003
         3    -0.0528   -3.4420   -0.0528   -0.0011    0.0000    0.0011
    4    1    -0.0377    0.0000   -0.0377   -0.0027    0.0000    0.0027
         2    -0.0151    0.0000   -0.0151   -0.0012    0.0000    0.0012
         3    -0.0528    0.0000   -0.0528   -0.0039    0.0000    0.0039
    5    1    -0.8020   -0.8020   -0.8020    0.0024    0.0000   -0.0024
         2    -0.2978   -0.2978   -0.2978    0.0007    0.0000   -0.0007
         3    -1.0998   -1.0998   -1.0998    0.0031    0.0000   -0.0031
    6    1     0.7929   -0.7929    0.7929   -0.0024    0.0000    0.0024
         2     0.2941   -0.2941    0.2941   -0.0008    0.0000    0.0008
         3     1.0871   -1.0871    1.0871   -0.0032    0.0000    0.0032
 ************** END OF LATEST ANALYSIS RESULT **************
  44. FINISH
           *********** END OF THE STAAD.Pro RUN ***********
             **** DATE= MAR 24,2022   TIME=  9:44:31 ****
    STAAD SPACE                                              -- PAGE NO.   10
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

Related Links

- *M. To assign an inclined support* (on page 823)
- *TR.27.2 Inclined Support Specification* (on page 2318)
- *Create Support dialog* (on page 2815)

EX. UK-20 Generating a Structure in Cylindrical Coordinates

This example generates the geometry of a cylindrical tank structure using the cylindrical coordinate system. The tank lies on its side in this example.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-20 Generating a Structure in Cylindrical Coordinates.STD` when you install the program.

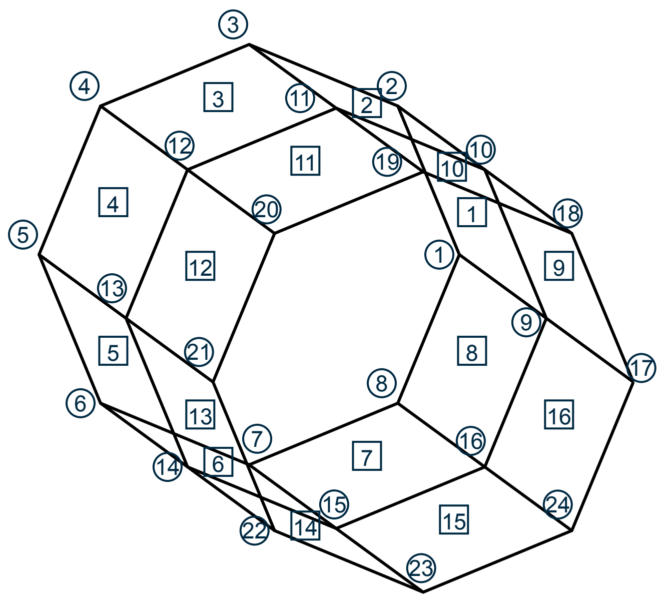


Figure 561: Example Problem No. 20

In this example, a cylindrical tank is modeled using finite elements. The radial direction is in the XY plane and longitudinal direction is along the Z-axis. Hence, the coordinates in the XY plane are generated using the cylindrical coordinate system.

```
STAAD SPACE
```

```
UNIT METER KN
```

The type of structure (space frame) and length and force units for data to follow are specified.

```
JOINT COORD CYLINDRICAL
```

The above command instructs the program that the coordinate data that follows is in the cylindrical coordinate system (r,theta,z).

```
1 3.5 0 0 8 3.5 315 0
```

Joint 1 has an 'r' of 3.5 meters, theta of 0 degrees and Z of 0 ft. Joint 8 has an 'r' of 3.5 meters, theta of 315 degrees and Z of 0 ft. The 315 degrees angle is measured counter-clockwise from the +ve direction of the X-axis. Joints 2 to 7 are generated by equal increments of the coordinate values between joints 1 and 8.

```
    REPEAT 2 0 0 3.0
```

The REPEAT command is used to generate joints 9 through 24 by repeating twice, the pattern of joints 1 to 8 at Z-increments of 3.0 meters for each `REPEAT`.

```
    PRINT JOINT COORD
```

The above command is used to produce a report consisting of the coordinates of all the joints in the Cartesian coordinate system. Note that even though the input data was in the cylindrical coordinate system, the output is in the Cartesian coordinate system.

```
    ELEMENT INCIDENCES
    1 1 2 10 9 TO 7 1 1
    8 8 1 9 16
    REPEAT ALL 1 8 8
```

The above 4 lines identify the element incidences of all 16 elements. Incidences of element 1 is defined as 1 2 10 9. Incidences of element 2 is generated by incrementing the joint numbers of element 1 by 1, incidences of element 3 is generated by incrementing the incidences of element 2 by 1 and so on up to element 7. Incidences of element 8 has been defined above as 8 1 9 16. The `REPEAT ALL` command states that the pattern of all the elements defined by the previous 2 lines, namely elements 1 to 8, must be repeated once with an element number increment of 8 and a joint number increment of 8 to generate elements 9 through 16.

```
    PRINT ELEMENT INFO
```

The above command is self-explanatory.

```
    FINISH
```

Input File

```
STAAD SPACE
UNIT METER KN
JOINT COORD CYLINDRICAL
1 3.5 0 0 8 3.5 315 0
REPEAT 2 0 0 3.0
PRINT JOINT COORD
ELEMENT INCIDENCES
1 1 2 10 9 TO 7 1 1
8 8 1 9 16
REPEAT ALL 1 8 8
PRINT ELEMENT INFO
FINISH
```

STAAD Output File

```
                                                                  PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:36                      *
             *                                                  *
```

```
            *  Licensed to: Bentley Systems Inc                *
            ****************************************************
     1. STAAD SPACE
INPUT FILE: UK-20 Generating a Structure in Cylindrical Coordinates.STD
     2. UNIT METER KN
     3. JOINT COORD CYLINDRICAL
     4. 1 3.5 0 0 8 3.5 315 0
     5. REPEAT 2 0 0 3.0
     6. PRINT JOINT COORD
  JOINT    COORD
      STAAD SPACE                                             -- PAGE NO.    2
   JOINT COORDINATES
   -----------------
   COORDINATES ARE METE UNIT
   JOINT            X              Y              Z
      1           3.500          0.000          0.000
      2           2.475          2.475          0.000
      3           0.000          3.500          0.000
      4          -2.475          2.475          0.000
      5          -3.500          0.000          0.000
      6          -2.475         -2.475          0.000
      7          -0.000         -3.500          0.000
      8           2.475         -2.475          0.000
      9           3.500          0.000          3.000
     10           2.475          2.475          3.000
     11           0.000          3.500          3.000
     12          -2.475          2.475          3.000
     13          -3.500          0.000          3.000
     14          -2.475         -2.475          3.000
     15          -0.000         -3.500          3.000
     16           2.475         -2.475          3.000
     17           3.500          0.000          6.000
     18           2.475          2.475          6.000
     19           0.000          3.500          6.000
     20          -2.475          2.475          6.000
     21          -3.500          0.000          6.000
     22          -2.475         -2.475          6.000
     23          -0.000         -3.500          6.000
     24           2.475         -2.475          6.000
   ************ END OF DATA FROM INTERNAL STORAGE ************
     7. ELEMENT INCIDENCES
     8. 1 1 2 10 9 TO 7 1 1
     9. 8 8 1 9 16
    10. REPEAT ALL 1 8 8
    11. PRINT ELEMENT INFO
  ELEMENT  INFO
      STAAD SPACE                                             -- PAGE NO.    3
   ELEMENT INFORMATION
   -------------------
  ELEMENT           INCIDENCES         THICK   POISS    E1/E2        G1/G2        AREA
    NO.                                (METE)
      1       1      2     10      9    0.000   0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                      0.000000E+00 0.000000E+00
      2       2      3     11     10    0.000   0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                      0.000000E+00 0.000000E+00
      3       3      4     12     11    0.000   0.000 0.000000E+00 0.000000E+00
```

```
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      4       4      5     13     12    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      5       5      6     14     13    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      6       6      7     15     14    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      7       7      8     16     15    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      8       8      1      9     16    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
      9       9     10     18     17    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     10      10     11     19     18    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     11      11     12     20     19    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     12      12     13     21     20    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     13      13     14     22     21    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     14      14     15     23     22    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     15      15     16     24     23    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
     16      16      9     17     24    0.000  0.000 0.000000E+00 0.000000E+00
8.036351E+00
                                                    0.000000E+00 0.000000E+00
   ******************END OF ELEMENT INFO******************
    12. FINISH
      STAAD SPACE                                              -- PAGE NO.    4
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44:36 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. UK-21 Analysis of a Structure with Tension-Only Members

This example illustrates the modeling of tension-only members using the `MEMBER TENSION` command.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-21 Analysis of a Structure with Tension-Only Members.STD` when you install the program.

It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of "active" members (and hence the stiffness matrix) is load case dependent.

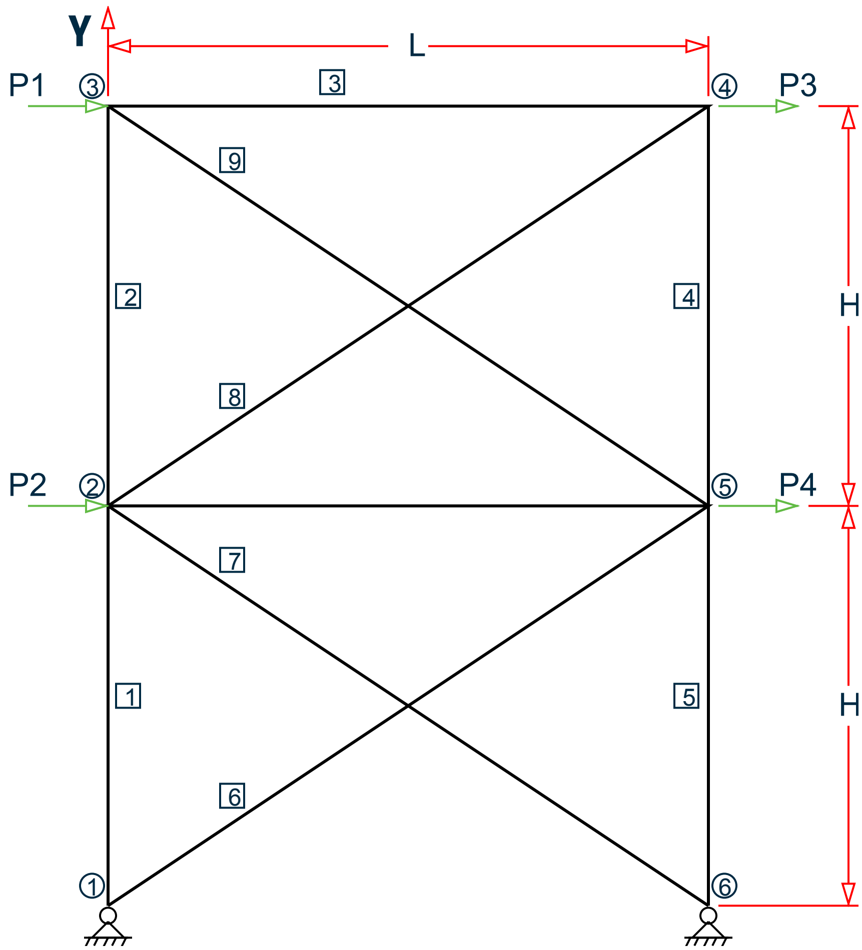


Figure 562: Example Problem No. 21

where:

L = 5.25 m, H = 3.5 m
Load case 1: P1 = 45 kN & P2 = 70 kN
Load case 2: P3 = -45 kN & P4 = -70 kN

```
STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
```

The input data is initiated with the word STAAD. This structure is a PLANE frame.

```
UNIT METER KNS
```

Units for the commands to follow are defined above.

```
JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
```

Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

Incidences of members 1 to 10 are defined.

```
MEMBER TENSION
6 TO 9
```

Members 6 to 9 are defined as tension-only members using the MEMBER TENSION command. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a compressive force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
```

All members have been assigned a UC section from the British table.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
```

The DEFINE MATERIAL command is used to specify material properties and the CONSTANT is used to assign the material to all members. The length units have been changed from meters to millimeters to facilitate the input of these values.

```
SUPPORT
1 6 PINNED
```

The supports are defined above.

```
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
```

Load 1 is defined above and consists of joint loads at joints 2 and 3.

```
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
6 TO 9
```

One or more among the members 6 to 9 may have been inactivated in the previous analysis.

```
    LOAD 2
    JOINT LOAD
    4 FX -45
    5 FX -70
```

Load case 2 is described above.

```
    LOAD 3
    REPEAT LOAD
    1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously.

```
    PRINT ANALYSIS RESULTS
    FINI
```

The analysis results are printed and the run terminated.

Input File

```
STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
UNIT METER KNS
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
MEMBER TENSION
6 TO 9
MEMBER PROPERTY BRITISH
1 TO 10 TABLE ST UC152X152X30
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
SUPPORT
1 6 PINNED
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
6 TO 9
LOAD 2
```

```
JOINT LOAD
4 FX -45
5 FX -70
PERFORM ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST 1 2 3
PRINT ANALYSIS RESULTS
FINI
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:38                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE EXAMPLE FOR TENSION-ONLY MEMBERS
INPUT FILE: UK-21 Analysis of a Structure with Tension-Only Members.STD
     2. UNIT METER KNS
     3. SET NL 3
     4. JOINT COORDINATES
     5. 1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
     6. MEMBER INCIDENCES
     7. 1 1 2 5
     8. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
     9. MEMBER TENSION
    10. 6 TO 9
    11. MEMBER PROPERTY BRITISH
    12. 1 TO 10 TABLE ST UC152X152X30
    13. UNIT MMS
    14. DEFINE MATERIAL START
    15. ISOTROPIC STEEL
    16. E 210
    17. POISSON 0.3
    18. DENSITY 7.6977E-008
    19. ALPHA 6E-006
    20. DAMP 0.03
    21. TYPE STEEL
    22. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
    23. END DEFINE MATERIAL
    24. CONSTANTS
    25. MATERIAL STEEL ALL
    26. SUPPORT
    27. 1 6 PINNED
    28. LOAD 1
    29. JOINT LOAD
```

```
    30. 2 FX 70
    31. 3 FX 45
    32. PERFORM ANALYSIS
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    2
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    1
    33. CHANGE
    34. MEMBER TENSION
    35. 6 TO 9
    36. LOAD 2
    37. JOINT LOAD
    38. 4 FX -45
    39. 5 FX -70
    40. PERFORM ANALYSIS
*** LOAD CASE     2 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    2
    41. CHANGE
    42. LOAD 3
    43. REPEAT LOAD
    44. 1 1.0 2 1.0
    45. PERFORM ANALYSIS
*** LOAD CASE     3 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    3
    46. CHANGE
    47. LOAD LIST 1 2 3
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    3
    48. PRINT ANALYSIS RESULTS
  ANALYSIS RESULTS
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    4
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0008
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0005
           3    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0001
      2    1    0.2355    0.0132    0.0000    0.0000    0.0000   -0.0004
           2   -0.1609   -0.0464    0.0000    0.0000    0.0000    0.0004
           3    0.0228    0.0000    0.0000    0.0000    0.0000   -0.0000
      3    1    0.3633    0.0133    0.0000    0.0000    0.0000   -0.0002
           2   -0.3341   -0.0594    0.0000    0.0000    0.0000    0.0004
           3    0.0147    0.0000    0.0000    0.0000    0.0000    0.0000
      4    1    0.3341   -0.0594    0.0000    0.0000    0.0000   -0.0004
           2   -0.3633    0.0133    0.0000    0.0000    0.0000    0.0002
           3   -0.0147    0.0000    0.0000    0.0000    0.0000   -0.0000
      5    1    0.1609   -0.0464    0.0000    0.0000    0.0000   -0.0004
           2   -0.2355    0.0132    0.0000    0.0000    0.0000    0.0004
           3   -0.0228    0.0000    0.0000    0.0000    0.0000    0.0000
      6    1    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0005
           2    0.0000    0.0000    0.0000    0.0000    0.0000    0.0008
```

```
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    5
   SUPPORT REACTIONS -UNIT KNS  MMS     STRUCTURE TYPE = PLANE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1   -114.92   -106.67      0.00      0.00      0.00      0.00
          2      0.08    106.67      0.00      0.00      0.00      0.00
          3     -0.05      0.00      0.00      0.00      0.00      0.00
     6    1     -0.08    106.67      0.00      0.00      0.00      0.00
          2    114.92   -106.67      0.00      0.00      0.00      0.00
          3      0.05      0.00      0.00      0.00      0.00      0.00
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    6
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  MMS      (LOCAL )
  MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1    -30.23      0.26     0.00      0.00      0.00       0.00
                2     30.23     -0.26     0.00      0.00      0.00     894.16
          2     1    106.67     -0.08     0.00      0.00      0.00       0.00
                2   -106.67      0.08     0.00      0.00      0.00    -294.48
          3     1     -0.00      0.05     0.00      0.00      0.00      -0.00
                2      0.00     -0.05     0.00      0.00      0.00     158.18
     2    1     2     -0.25      0.21     0.00      0.00      0.00     200.03
                3      0.25     -0.21     0.00      0.00      0.00     545.92
          2     2     29.82     -0.44     0.00      0.00      0.00    -770.68
                3    -29.82      0.44     0.00      0.00      0.00    -756.76
          3     2     -0.00     -0.05     0.00      0.00      0.00    -137.99
                3      0.00      0.05     0.00      0.00      0.00     -43.08
     3    1     3     44.79     -0.25     0.00      0.00      0.00    -545.92
                4    -44.79      0.25     0.00      0.00      0.00    -756.76
          2     3     44.79      0.25     0.00      0.00      0.00     756.76
                4    -44.79     -0.25     0.00      0.00      0.00     545.92
          3     3     45.05     -0.00     0.00      0.00      0.00      43.08
                4    -45.05      0.00     0.00      0.00      0.00     -43.08
     4    1     4     29.82      0.44     0.00      0.00      0.00     756.76
                5    -29.82     -0.44     0.00      0.00      0.00     770.68
          2     4     -0.25     -0.21     0.00      0.00      0.00    -545.92
                5      0.25      0.21     0.00      0.00      0.00    -200.03
          3     4      0.00      0.05     0.00      0.00      0.00      43.08
                5     -0.00     -0.05     0.00      0.00      0.00     137.99
     5    1     5    106.67      0.08     0.00      0.00      0.00     294.48
                6   -106.67     -0.08     0.00      0.00      0.00       0.00
          2     5    -30.23     -0.26     0.00      0.00      0.00    -894.16
                6     30.23      0.26     0.00      0.00      0.00       0.00
          3     5      0.00     -0.05     0.00      0.00      0.00    -158.18
                6     -0.00      0.05     0.00      0.00      0.00       0.00
     6    1     1   -137.80      0.00     0.00      0.00      0.00       0.00
                5    137.80      0.00     0.00      0.00      0.00       0.00
          2     1      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
          3     1      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
     7    1     2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
      EXAMPLE FOR TENSION-ONLY MEMBERS                         -- PAGE NO.    7
   MEMBER END FORCES    STRUCTURE TYPE = PLANE
   -----------------
   ALL UNITS ARE -- KNS  MMS      (LOCAL )
```

```
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           2    2   -137.80      0.00     0.00      0.00      0.00       0.00
                6    137.80      0.00     0.00      0.00      0.00       0.00
           3    2      0.00      0.00     0.00      0.00      0.00       0.00
                6      0.00      0.00     0.00      0.00      0.00       0.00
      8    1    2    -53.30      0.00     0.00      0.00      0.00       0.00
                4     53.30      0.00     0.00      0.00      0.00       0.00
           2    2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
           3    2      0.00      0.00     0.00      0.00      0.00       0.00
                4      0.00      0.00     0.00      0.00      0.00       0.00
      9    1    3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
           2    3    -53.30      0.00     0.00      0.00      0.00       0.00
                5     53.30      0.00     0.00      0.00      0.00       0.00
           3    3      0.00      0.00     0.00      0.00      0.00       0.00
                5      0.00      0.00     0.00      0.00      0.00       0.00
     10    1    2    114.31     -0.41     0.00      0.00      0.00   -1094.19
                5   -114.31      0.41     0.00      0.00      0.00   -1065.16
           2    2    114.31      0.41     0.00      0.00      0.00    1065.16
                5   -114.31     -0.41     0.00      0.00      0.00    1094.19
           3    2     69.90     -0.00     0.00      0.00      0.00     -20.19
                5    -69.90      0.00     0.00      0.00      0.00      20.19
  ************** END OF LATEST ANALYSIS RESULT **************
   49. FINI
     EXAMPLE FOR TENSION-ONLY MEMBERS                          -- PAGE NO.    8
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:38 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-22 Time History Analysis for Sinusoidal Loading

A space frame structure is subjected to a sinusoidal (dynamic) loading. The commands necessary to describe the sine function are demonstrated in this example. Time History analysis is performed on this model.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-22 Time History Analysis for Sinusoidal Loading.STD` when you install the program.

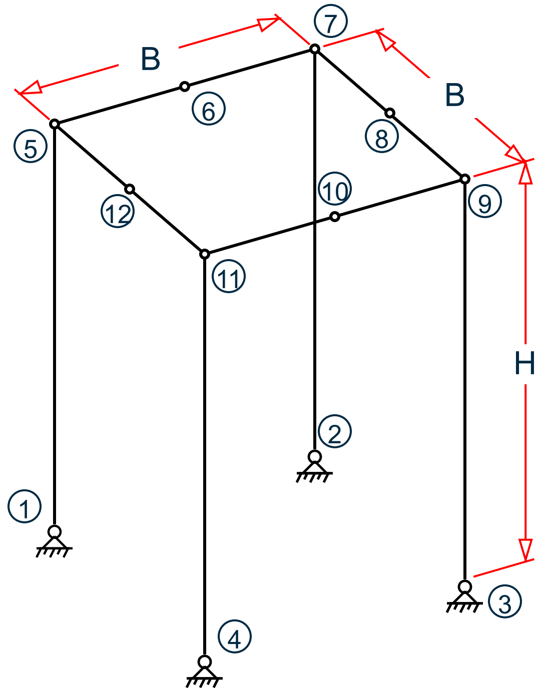


Figure 563: Example Problem No. 22

```
STAAD SPACE
*EXAMPLE FOR HARMONIC LOADING GENERATOR
```

Every STAAD input file has to begin with the word STAAD.

The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

```
UNIT KNS METER
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
5 0 7 0 ; 6 2.5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
9 5 7 5 ; 10 2.5 7 5 ; 11 0 7 5
12 0 7 2.5 ; 13 2.5 7 2.5
```

The joint number followed by the X, Y and Z coordinates are specified above.

Note: Semicolons (;) are used as line separators to allow for input of multiple sets of data on one line.

```
MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
```

The members are defined by the joints to which they are connected.

```
UNIT MMS
MEMBER PROPERTIES
```

```
1 TO 4 PRIS YD 600 ZD 600
5 TO 16 PRIS YD 450 ZD 450
```

Members 1 to 16 are defined as PRISmatic sections with width and depth values provided using the YD and ZD options. The UNIT command is specified to change the units for length from METER to MMS.

```
SUPPORTS
1 TO 4 PINNED
```

Joints 1 to 4 are declared to be pinned-supported.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The modulus of elasticity (E), density and Poisson's ratio are specified following the command CONSTANTS. Built-in default values for concrete are used.

```
DEFINE TIME HISTORY
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 30 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
```

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the sinusoidal loading are provided.

Each data set is individually identified by the number that follows the `TYPE` command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word `FORCE` that follows the `TYPE 1` command signifies that this data set is for a forcing function. (If you want to specify an earthquake motion, an `ACCELERATION` may be specified.)

The command `FUNCTION SINE` indicates that instead of providing the data set as discrete TIME-FORCE pairs, a sinusoidal function, which describes the variation of force with time, is provided.

The parameters of the sine function, such as `FREQUENCY`, `AMPLITUDE`, and number of `CYCLES` of application are then defined. STAAD internally generates discrete TIME-FORCE pairs of data from the sine function in steps of time defined by the default value (refer to TR.31.4 Definition of Time History Load (on page 2441) for more information). The arrival time value indicates the relative value of time at which the force begins to act upon the structure. The modal damping ratio for all the modes is set to 0.075.

```
UNIT METER
LOAD 1
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -10.0
```

The above data describe a static load case. A uniformly distributed load of 10 kN/m acting in the negative global Y direction is applied on some members.

```
    LOAD 2
    SELFWEIGHT X 1.0
    SELFWEIGHT Y 1.0
    SELFWEIGHT Z 1.0
    JOINT LOAD
    8 12 FX 15.0
    8 12 FY 15.0
    8 12 FZ 15.0
    TIME LOAD
    8 12 FX 1 1
```

This is the second stage of command specification for time history analysis. The two sets of data specified here are:

a. the weights for generation of the mass matrix
b. the application of the time varying loads on the structure.

The weights (from which the masses for the mass matrix are obtained) are specified in the form of selfweight and joint loads.

Following that, the sinusoidal force is applied using the `TIME LOAD` command. The forcing function described by the TYPE 1 load is applied on joints 8 and 12 and it starts to act starting at a time defined by the 1st arrival time number.

```
    PERFORM ANALYSIS
    PRINT ANALYSIS RESULTS
    FINI
```

The above commands are self explanatory. The `FINISH` command terminates the STAAD run.

Input File

```
STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
UNIT KNS METER
JOINT COORDINATES
1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
5 0 7 0 ; 6 2.5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
9 5 7 5 ; 10 2.5 7 5 ; 11 0 7 5
12 0 7 2.5 ; 13 2.5 7 2.5
MEMBER INCIDENCES
1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
UNIT MMS
MEMBER PROPERTIES
1 TO 4 PRIS YD 600 ZD 600
5 TO 16 PRIS YD 450 ZD 450
SUPPORTS
1 TO 4 PINNED
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
```

```
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
DEFINE TIME HISTORY
TYPE 1 FORCE
* FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
FUNCTION SINE
AMPLITUDE 30 FREQUENCY 60 CYCLES 100
*
ARRIVAL TIMES
0.0
DAMPING 0.075
UNIT METER
LOAD 1
MEMBER LOAD
5 6 7 8 9 10 11 12 UNI GY -10.0
LOAD 2
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
8 12 FX 15.0
8 12 FY 15.0
8 12 FZ 15.0
TIME LOAD
8 12 FX 1 1
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
FINI
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:40                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE FOR HARMONIC LOADING GENERATOR
INPUT FILE: UK-22 Time History Analysis for Sinusoidal Loading.STD
     2. UNIT KNS METER
     3. JOINT COORDINATES
     4. 1 0 0 0 ; 2 5 0 0 ; 3 5 0 5 ; 4 0 0 5
     5. 5 0 7 0 ; 6 2.5 7 0 ; 7 5 7 0 ; 8 5 7 2.5
     6. 9 5 7 5 ; 10 2.5 7 5 ; 11 0 7 5
     7. 12 0 7 2.5 ; 13 2.5 7 2.5
     8. MEMBER INCIDENCES
     9. 1 1 5 ; 2 2 7 ; 3 3 9 ; 4 4 11 ; 5 5 6 ; 6 6 7
    10. 7 7 8 ; 8 8 9 ; 9 9 10 ; 10 10 11 ; 11 11 12 ; 12 12 5
```

```
  11. 13 6 13 ; 14 13 10 ; 15 8 13 ; 16 13 12
  12. UNIT MMS
  13. MEMBER PROPERTIES
  14. 1 TO 4 PRIS YD 600 ZD 600
  15. 5 TO 16 PRIS YD 450 ZD 450
  16. SUPPORTS
  17. 1 TO 4 PINNED
  18. DEFINE MATERIAL START
  19. ISOTROPIC CONCRETE
  20. E 21.0
  21. POISSON 0.17
  22. DENSITY 2.36158E-008
  23. ALPHA 5E-006
  24. DAMP 0.05
  25. G 9.25
  26. TYPE CONCRETE
  27. STRENGTH FCU 0.0275
  28. END DEFINE MATERIAL
  29. CONSTANTS
  30. MATERIAL CONCRETE ALL
  31. DEFINE TIME HISTORY
  32. TYPE 1 FORCE
  33. * FOLLOWING LINES FOR HARMONIC LOADING GENERATOR
  34. FUNCTION SINE
  35. AMPLITUDE 30 FREQUENCY 60 CYCLES 100
    EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    2
 FOR SEQUENTIAL HARMONIC FORCING CURVE NUMBER=             1
 NUMBER OF POINTS IN DIGITIZED HARMONIC FUNCTION=       1201
 NUMBER OF POINTS PER QUARTER CYCLE OF HARMONIC FUNCTION=  3
 FORCE STEP DELTA TIME PER POINT                  1.38889E-03
 ENDING TIME FOR THIS DIGITIZED HARMONIC FUNCTION  1.66667E+00
  36. *
  37. ARRIVAL TIMES
  38. 0.0
  39. DAMPING 0.075
  40. UNIT METER
  41. LOAD 1
  42. MEMBER LOAD
  43. 5 6 7 8 9 10 11 12 UNI GY -10.0
  44. LOAD 2
  45. SELFWEIGHT X 1.0
  46. SELFWEIGHT Y 1.0
  47. SELFWEIGHT Z 1.0
  48. JOINT LOAD
  49. 8 12 FX 15.0
  50. 8 12 FY 15.0
  51. 8 12 FZ 15.0
  52. TIME LOAD
  53. 8 12 FX 1 1
  54. PERFORM ANALYSIS
          P R O B L E M   S T A T I S T I C S
          -----------------------------------
   NUMBER OF JOINTS         13  NUMBER OF MEMBERS      16
   NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
   NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
         Using 64-bit analysis engine.
         SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
 TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =      66
```

```
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  ***NOTE: MASSES DEFINED UNDER LOAD#       2 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    3
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED             =     6
NUMBER OF EXISTING MASSES IN THE MODEL =    27
NUMBER OF MODES THAT WILL BE USED      =     6
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    4
              CALCULATED FREQUENCIES FOR LOAD CASE       2
      MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                       1.832                 0.54584
        2                       1.833                 0.54558
        3                       2.179                 0.45894
        4                      18.014                 0.05551
        5                      18.981                 0.05268
        6                      23.162                 0.04317
           MODAL WEIGHT (MODAL MASS TIMES g) IN KNS          GENERALIZED
     MODE           X             Y             Z              WEIGHT
        1       2.924784E+02  3.675141E-24  1.333421E-21    2.902617E+02
        2       1.338687E-21  3.119688E-30  2.924834E+02    2.907889E+02
        3       1.457481E-25  3.230663E-26  6.769659E-21    4.344889E+02
        4       1.151691E-20  3.984483E-21  1.962646E-19    3.771789E+02
        5       3.567685E-14  1.271663E+02  3.475675E-20    5.768673E+01
        6       8.511521E-03  2.884108E-21  4.373936E-20    6.607558E+01
MASS PARTICIPATION FACTORS
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
         MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
           1   100.00   0.00   0.00   99.996    0.000    0.000
           2     0.00   0.00 100.00   99.996    0.000   99.998
           3     0.00   0.00   0.00   99.996    0.000   99.998
           4     0.00   0.00   0.00   99.996    0.000   99.998
           5     0.00  43.48   0.00   99.996   43.477   99.998
           6     0.00   0.00   0.00   99.999   43.477   99.998
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    5
   A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
   MODE       DAMPING
      1      0.07500000
      2      0.07500000
      3      0.07500000
      4      0.07500000
      5      0.07500000
      6      0.07500000
  TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
  NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =    6
  TIME DURATION OF TIME HISTORY ANALYSIS =      1.667 SECONDS
  NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =     1200
   55. PRINT ANALYSIS RESULTS
 BASE SHEAR UNITS ARE -- KNS  METE
 MAXIMUM BASE SHEAR  X=  -1.639967E+00  Y=  -2.384186E-07  Z=  -9.313226E-10
     AT TIMES               0.127778           0.034722           0.063889
 ANALYSIS RESULTS
     EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    6
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
```

```
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1    0.0000    0.0000    0.0000   -0.0001    0.0000    0.0001
          2    0.0000    0.0000    0.0000    0.0000    0.0000   -0.0001
     2    1    0.0000    0.0000    0.0000   -0.0001    0.0000   -0.0001
          2    0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0001
     3    1    0.0000    0.0000    0.0000    0.0001    0.0000   -0.0001
          2    0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0001
     4    1    0.0000    0.0000    0.0000    0.0001    0.0000    0.0001
          2    0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0001
     5    1    0.0001   -0.0046    0.0001    0.0002    0.0000   -0.0002
          2    0.0414    0.0001    0.0000   -0.0000    0.0000   -0.0000
     6    1    0.0000   -0.0487    0.0000    0.0001    0.0000    0.0000
          2    0.0414    0.0000    0.0000    0.0000    0.0000    0.0000
     7    1   -0.0001   -0.0046    0.0001    0.0002    0.0000    0.0002
          2    0.0414   -0.0001   -0.0000    0.0000    0.0000   -0.0000
     8    1   -0.0000   -0.0487    0.0000    0.0000    0.0000    0.0001
          2    0.0416   -0.0005    0.0000    0.0000    0.0000   -0.0000
     9    1   -0.0001   -0.0046   -0.0001   -0.0002    0.0000    0.0002
          2    0.0414   -0.0001    0.0000   -0.0000   -0.0000   -0.0000
    10    1    0.0000   -0.0487   -0.0000   -0.0001    0.0000    0.0000
          2    0.0414    0.0000    0.0000    0.0000   -0.0000    0.0000
    11    1    0.0001   -0.0046   -0.0001   -0.0002    0.0000   -0.0002
          2    0.0414    0.0001   -0.0000    0.0000   -0.0000   -0.0000
    12    1    0.0000   -0.0487    0.0000    0.0000    0.0000   -0.0001
          2    0.0416    0.0005    0.0000    0.0000    0.0000   -0.0000
    13    1    0.0000   -0.0604    0.0000    0.0000    0.0000    0.0000
          2    0.0416   -0.0000    0.0000    0.0000    0.0000    0.0000
      EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    7
   SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
   -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1      2.15     50.00      2.15      0.00      0.00      0.00
          2     -0.41     -1.13     -0.02      0.00      0.00      0.00
     2    1     -2.15     50.00      2.15      0.00      0.00      0.00
          2     -0.41      1.13      0.02      0.00      0.00      0.00
     3    1     -2.15     50.00     -2.15      0.00      0.00      0.00
          2     -0.41      1.13     -0.02      0.00      0.00      0.00
     4    1      2.15     50.00     -2.15      0.00      0.00      0.00
          2     -0.41     -1.13      0.02      0.00      0.00      0.00
      EXAMPLE FOR HARMONIC LOADING GENERATOR                   -- PAGE NO.    8
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KNS  METE     (LOCAL )
  MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1     50.00     -2.15     2.15      0.00      0.00       0.00
                5    -50.00      2.15    -2.15      0.00    -15.03     -15.03
          2     1     -1.13      0.41    -0.02      0.00      0.00       0.00
                5      1.13     -0.41     0.02      0.00      0.12       2.87
     2    1     2     50.00      2.15     2.15      0.00      0.00       0.00
                7    -50.00     -2.15    -2.15      0.00    -15.03      15.03
          2     2      1.13      0.41     0.02      0.00      0.00       0.00
                7     -1.13     -0.41    -0.02      0.00     -0.12       2.87
     3    1     3     50.00      2.15    -2.15      0.00     -0.00       0.00
                9    -50.00     -2.15     2.15      0.00     15.03      15.03
          2     3      1.13      0.41    -0.02      0.00      0.00       0.00
                9     -1.13     -0.41     0.02      0.00      0.12       2.87
     4    1     4     50.00     -2.15    -2.15      0.00     -0.00       0.00
               11    -50.00      2.15     2.15      0.00     15.03     -15.03
```

```
        2    4     -1.13      0.41     0.02      0.00      0.00       0.00
            11      1.13     -0.41    -0.02      0.00     -0.12       2.87
    5   1    5      2.09     25.00     0.06      1.34     -0.07      16.37
             6     -2.09     -0.00    -0.06     -1.34     -0.07      14.88
        2    5     -0.10     -1.03    -0.05     -0.03      0.08      -2.49
             6      0.10      1.03     0.05      0.03      0.06      -0.08
    6   1    6      2.09      0.00    -0.06     -1.34      0.07     -14.88
             7     -2.09     25.00     0.06      1.34      0.07     -16.37
        2    6      0.10     -1.03    -0.05     -0.03      0.06      -0.08
             7     -0.10      1.03     0.05      0.03      0.08      -2.49
    7   1    7      2.09     25.00     0.06      1.34     -0.07      16.37
             8     -2.09      0.00    -0.06     -1.34     -0.07      14.88
        2    7     -0.04      0.14     0.08     -0.38     -0.08       0.14
             8      0.04     -0.14    -0.08      0.38     -0.12       0.21
    8   1    8      2.09      0.00    -0.06     -1.34      0.07     -14.88
             9     -2.09     25.00     0.06      1.34      0.07     -16.37
        2    8     -0.04     -0.14    -0.08      0.38      0.12      -0.21
             9      0.04      0.14     0.08     -0.38      0.08      -0.14
    9   1    9      2.09     25.00     0.06      1.34     -0.07      16.37
            10     -2.09      0.00    -0.06     -1.34     -0.07      14.88
    EXAMPLE FOR HARMONIC LOADING GENERATOR                    -- PAGE NO.    9
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KNS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
        2    9      0.10      1.03     0.05      0.03     -0.08       2.49
            10     -0.10     -1.03    -0.05     -0.03     -0.06       0.08
   10   1   10      2.09     -0.00    -0.06     -1.34      0.07     -14.88
            11     -2.09     25.00     0.06      1.34      0.07     -16.37
        2   10     -0.10      1.03     0.05      0.03     -0.06       0.08
            11      0.10     -1.03    -0.05     -0.03     -0.08       2.49
   11   1   11      2.09     25.00     0.06      1.34     -0.07      16.37
            12     -2.09      0.00    -0.06     -1.34     -0.07      14.88
        2   11      0.04     -0.14    -0.08      0.38      0.08      -0.14
            12     -0.04      0.14     0.08     -0.38      0.12      -0.21
   12   1   12      2.09      0.00    -0.06     -1.34      0.07     -14.88
             5     -2.09     25.00     0.06      1.34      0.07     -16.37
        2   12      0.04      0.14     0.08     -0.38     -0.12       0.21
             5     -0.04     -0.14    -0.08      0.38     -0.08       0.14
   13   1    6      0.12     -0.00     0.00      0.00     -0.00      -2.67
            13     -0.12      0.00    -0.00     -0.00     -0.00       2.67
        2    6      0.00      0.00     0.10      0.17     -0.11       0.00
            13     -0.00     -0.00    -0.10     -0.17     -0.13       0.00
   14   1   13      0.12      0.00    -0.00     -0.00      0.00      -2.67
            10     -0.12     -0.00     0.00      0.00      0.00       2.67
        2   13      0.00     -0.00    -0.10     -0.17      0.13      -0.00
            10     -0.00      0.00     0.10      0.17      0.11      -0.00
   15   1    8      0.12     -0.00    -0.00      0.00     -0.00      -2.67
            13     -0.12      0.00     0.00     -0.00      0.00       2.67
        2    8     -0.03      0.24     0.00     -0.00     -0.00       0.76
            13      0.03     -0.24    -0.00      0.00     -0.00      -0.17
   16   1   13      0.12      0.00    -0.00     -0.00      0.00      -2.67
            12     -0.12     -0.00     0.00      0.00      0.00       2.67
        2   13      0.03      0.24     0.00     -0.00     -0.00      -0.17
            12     -0.03     -0.24    -0.00      0.00     -0.00       0.76
 ************** END OF LATEST ANALYSIS RESULT **************
  56. FINI
    EXAMPLE FOR HARMONIC LOADING GENERATOR                    -- PAGE NO.   10
```

```
          *********** END OF THE STAADD.Pro RUN ***********
            **** DATE= MAR 24,2022   TIME=  9:44:41 ****
     ************************************************************
     *   For technical assistance on STAAD.Pro, please visit    *
     *   http://www.bentley.com/en/support/                     *
     *                                                          *
     *   Details about additional assistance from               *
     *   Bentley and Partners can be found at program menu      *
     *   Help->Technical Support                                *
     *                                                          *
     *           Copyright (c) Bentley Systems, Inc.            *
     *                http://www.bentley.com                    *
     ************************************************************
```

EX. UK-23 Spring Support Generation for a Slab on Grade

This example illustrates the usage of commands necessary to automatically generate spring supports for a slab on grade. The slab is subjected to pressure loading and analysis of the structure is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-23 Spring Support Generation for a Slab on Grade.STD` when you install the program.

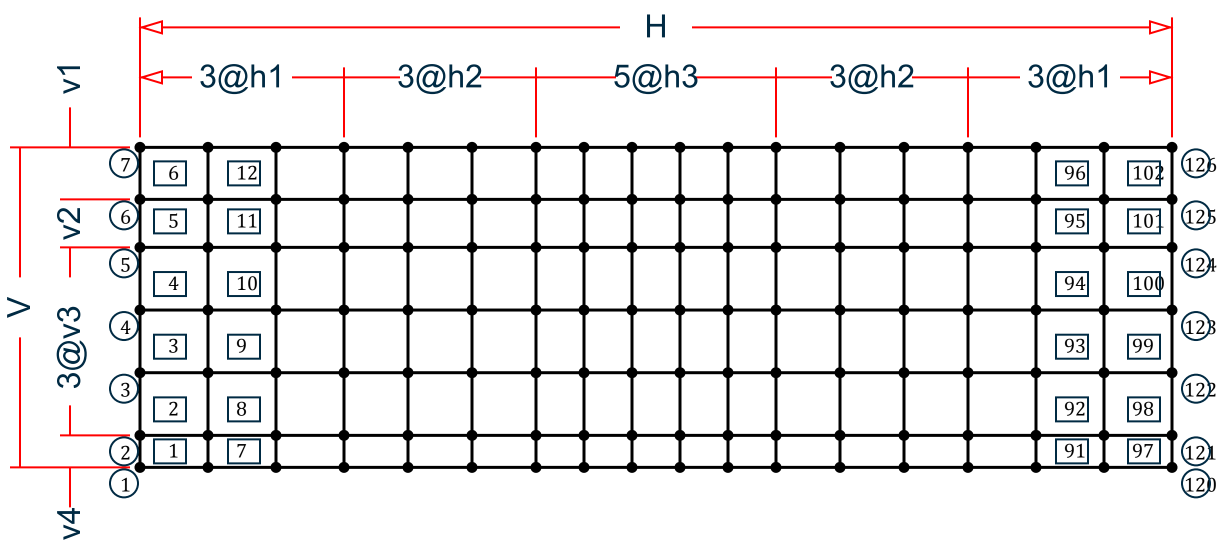


Figure 564: Example Problem No. 23

where:

H = 43m, h1 = 2.83m, h2 = 2.67m, h3 = 2m
V = 13.33m, v1 = 2.17m, v2 = 2m, v3 = 2.61m, v4 = 1.33m

```
STAAD SPACE SLAB ON GRADE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The remainder of the words form a title to identify this project.

```
UNIT METER KNS
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0.0 0.0 13.33
2 0.0 0.0 12.0
3 0.0 0.0  9.39
4 0.0 0.0  6.78
5 0.0 0.0  4.17
6 0.0 0.0  2.17
7 0.0 0.0  0.0
REPEAT ALL 3 2.83 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 5 2 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 3 2.83 0.0 0.0
```

For joints 1 through 7, the joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these seven joints by 2.83 meters, three times. `REPEAT` commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and that data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first six elements is then used to generate the incidences of 96 (= 16 x 6) more elements using the `REPEAT` command.

```
UNIT CM
ELEMENT PROPERTIES
1 TO 102 TH 14.0
```

The thickness of elements 1 to 102 is specified as 14 cms following the command ELEMENT PROPERTIES.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUB 1570.
```

The above command is used to instruct STAAD to generate supports with springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade modulus of the soil is specified as

1570 KN/cu.m. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the FY degree of freedom at the joint. See TR. 27.3 Automatic Spring Support Generator for Foundations (on page 2319) .

```
PRINT SUPP INFO
```

This command will enable us to obtain the details of the support conditions which were generated using the earlier commands.

```
LOAD 1 WEIGHT OF MAT & EARTH
ELEMENT LOAD
1 TO 102 PR GY –74.2
```

The above data describe a static load case. A pressure load of 74.2 kN/sq.m. acting in the negative global Y direction is applied on all the 102 elements.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2  FY -965.
8 9  FY -485.
5 FY -1373.
6 FY -2746.
22 23 FY -1824.
29 30 FY -912.
26 FY -2414.
27 FY -4828.
43 44 50 51 71 72 78 79 FY -1368.
47 54 82 FY -1175.
48 55 76 83 FY -2350.
92 93 FY -912.
99 100 FY -1824.
103 FY -2166.
104 FY -4333.
113 114 FY -485.
120 121 FY -965.
124 FY -1216.
125 FY -2431.
```

Load case 2 consists of several joint loads acting in the negative global Y direction.

```
LOADING COMBINATION 101 TOTAL LOAD
1 1. 2 1.
```

A load combination case, identified with load case number 101, is specified above. It instructs STAAD to factor loads 1 and 2 by a value of 1.0 and then algebraically add the results.

```
PERFORM ANALYSIS
```

The analysis is initiated using the above command.

```
UNIT CM
    LOAD LIST 101
    PRINT JOINT DISPLACEMENTS LIST 33 56
    PRINT ELEMENT STRESSES LIST 34 67
```

Joint displacements for joints 33 and 56, and element stresses for elements 34 and 67, for load case 101, is obtained with the help of the above commands.

```
FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE SLAB ON GRADE
UNIT METER KNS
JOINT COORDINATES
1 0.0 0.0 13.33
2 0.0 0.0 12.0
3 0.0 0.0 9.39
4 0.0 0.0 6.78
5 0.0 0.0 4.17
6 0.0 0.0  2.17
7 0.0 0.0  0.0
REPEAT ALL 3 2.83 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 5 2 0.0 0.0
REPEAT 3 2.67 0.0 0.0
REPEAT 3 2.83 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT CM
ELEMENT PROPERTIES
1 TO 102 TH 14.0
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
UNIT METER
SUPPORTS
1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 1570.
PRINT SUPP INFO
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -74.2
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2   FY -965.
8 9   FY -485.
5     FY -1373.
6     FY -2746.
22 23 FY -1824.
29 30 FY -912.
26    FY -2414.
27    FY -4828.
43 44 50 51 71 72 78 79 FY -1368.
47 54 82    FY -1175.
48 55 76 83 FY -2350.
```

```
92 93       FY -912.
99 100      FY -1824.
103         FY -2166.
104         FY -4333.
113 114     FY -485.
120 121     FY -965.
124         FY -1216.
125         FY -2431.
LOADING COMBINATION 101 TOTAL LOAD
1 1. 2 1.
PERFORM ANALYSIS
UNIT CM
LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67
FINISH
```

STAAD Output File

```
                                                               PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:43                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE SLAB ON GRADE
INPUT FILE: UK-23 Spring Support Generation for a Slab on Grade.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0.0 0.0 13.33
     5. 2 0.0 0.0 12.0
     6. 3 0.0 0.0 9.39
     7. 4 0.0 0.0 6.78
     8. 5 0.0 0.0 4.17
     9. 6 0.0 0.0  2.17
    10. 7 0.0 0.0  0.0
    11. REPEAT ALL 3 2.83 0.0 0.0
    12. REPEAT 3 2.67 0.0 0.0
    13. REPEAT 5 2 0.0 0.0
    14. REPEAT 3 2.67 0.0 0.0
    15. REPEAT 3 2.83 0.0 0.0
    16. ELEMENT INCIDENCES
    17. 1 1 8 9 2 TO 6
    18. REPEAT 16 6 7
    19. UNIT CM
    20. ELEMENT PROPERTIES
    21. 1 TO 102 TH 14.0
    22. UNIT MMS
    23. DEFINE MATERIAL START
    24. ISOTROPIC CONCRETE
    25. E 21.0
    26. POISSON 0.17
```

```
  27. DENSITY 2.36158E-008
  28. ALPHA 5E-006
  29. DAMP 0.05
  30. G 9.25
  31. TYPE CONCRETE
  32. STRENGTH FCU 0.0275
  33. END DEFINE MATERIAL
  34. CONSTANTS
  35. MATERIAL CONCRETE ALL
  36. UNIT METER
  37. SUPPORTS
  38. 1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 1570.
    SLAB ON GRADE                                               -- PAGE NO.    2
  39. PRINT SUPP INFO
SUPP     INFO
 SUPPORT INFORMATION (1=FIXED, 0=RELEASED)
 -------------------
 UNITS FOR SPRING CONSTANTS ARE KNS  METE DEGREES
 JOINT   FORCE-X/   FORCE-Y/   FORCE-Z/   MOM-X/   MOM-Y/   MOM-Z/
           KFX        KFY        KFZ       KMX      KMY      KMZ
    1       1          0          1         0        1        0
               0.0     1477.3       0.0      0.0      0.0      0.0
    2       1          0          1         0        1        0
               0.0     4376.5       0.0      0.0      0.0      0.0
    3       1          0          1         0        1        0
               0.0     5798.2       0.0      0.0      0.0      0.0
    4       1          0          1         0        1        0
               0.0     5798.2       0.0      0.0      0.0      0.0
    5       1          0          1         0        1        0
               0.0     5120.7       0.0      0.0      0.0      0.0
    6       1          0          1         0        1        0
               0.0     4631.9       0.0      0.0      0.0      0.0
    7       1          0          1         0        1        0
               0.0     2410.4       0.0      0.0      0.0      0.0
    8       1          0          1         0        1        0
               0.0     2954.7       0.0      0.0      0.0      0.0
    9       1          0          1         0        1        0
               0.0     8752.9       0.0      0.0      0.0      0.0
   10       1          0          1         0        1        0
               0.0    11596.5       0.0      0.0      0.0      0.0
   11       1          0          1         0        1        0
               0.0    11596.5       0.0      0.0      0.0      0.0
   12       1          0          1         0        1        0
               0.0    10241.3       0.0      0.0      0.0      0.0
   13       1          0          1         0        1        0
               0.0     9263.9       0.0      0.0      0.0      0.0
   14       1          0          1         0        1        0
               0.0     4820.8       0.0      0.0      0.0      0.0
   15       1          0          1         0        1        0
               0.0     2954.7       0.0      0.0      0.0      0.0
   16       1          0          1         0        1        0
               0.0     8752.9       0.0      0.0      0.0      0.0
   17       1          0          1         0        1        0
               0.0    11596.5       0.0      0.0      0.0      0.0
   18       1          0          1         0        1        0
               0.0    11596.5       0.0      0.0      0.0      0.0
   19       1          0          1         0        1        0
               0.0    10241.3       0.0      0.0      0.0      0.0
```

```
     20       1         0          1         0        1        0
                0.0     9263.9        0.0      0.0      0.0      0.0
     21       1         0          1         0        1        0
                0.0     4820.8        0.0      0.0      0.0      0.0
     22       1         0          1         0        1        0
                0.0     2871.1        0.0      0.0      0.0      0.0
      SLAB ON GRADE                                               -- PAGE NO.    3
     23       1         0          1         0        1        0
                0.0     8505.5        0.0      0.0      0.0      0.0
     24       1         0          1         0        1        0
                0.0    11268.7        0.0      0.0      0.0      0.0
     25       1         0          1         0        1        0
                0.0    11268.7        0.0      0.0      0.0      0.0
     26       1         0          1         0        1        0
                0.0     9951.8        0.0      0.0      0.0      0.0
     27       1         0          1         0        1        0
                0.0     9002.0        0.0      0.0      0.0      0.0
     28       1         0          1         0        1        0
                0.0     4684.5        0.0      0.0      0.0      0.0
     29       1         0          1         0        1        0
                0.0     2787.6        0.0      0.0      0.0      0.0
     30       1         0          1         0        1        0
                0.0     8258.0        0.0      0.0      0.0      0.0
     31       1         0          1         0        1        0
                0.0    10940.9        0.0      0.0      0.0      0.0
     32       1         0          1         0        1        0
                0.0    10940.9        0.0      0.0      0.0      0.0
     33       1         0          1         0        1        0
                0.0     9662.3        0.0      0.0      0.0      0.0
     34       1         0          1         0        1        0
                0.0     8740.1        0.0      0.0      0.0      0.0
     35       1         0          1         0        1        0
                0.0     4548.2        0.0      0.0      0.0      0.0
     36       1         0          1         0        1        0
                0.0     2787.6        0.0      0.0      0.0      0.0
     37       1         0          1         0        1        0
                0.0     8258.0        0.0      0.0      0.0      0.0
     38       1         0          1         0        1        0
                0.0    10940.9        0.0      0.0      0.0      0.0
     39       1         0          1         0        1        0
                0.0    10940.9        0.0      0.0      0.0      0.0
     40       1         0          1         0        1        0
                0.0     9662.3        0.0      0.0      0.0      0.0
     41       1         0          1         0        1        0
                0.0     8740.1        0.0      0.0      0.0      0.0
     42       1         0          1         0        1        0
                0.0     4548.2        0.0      0.0      0.0      0.0
     43       1         0          1         0        1        0
                0.0     2437.9        0.0      0.0      0.0      0.0
     44       1         0          1         0        1        0
                0.0     7221.9        0.0      0.0      0.0      0.0
     45       1         0          1         0        1        0
                0.0     9568.1        0.0      0.0      0.0      0.0
     46       1         0          1         0        1        0
                0.0     9568.1        0.0      0.0      0.0      0.0
     47       1         0          1         0        1        0
                0.0     8450.0        0.0      0.0      0.0      0.0
     48       1         0          1         0        1        0
```

```
              0.0     7643.5         0.0       0.0       0.0       0.0
     49       1       0          1         0         1         0
              0.0     3977.6         0.0       0.0       0.0       0.0
     50       1       0          1         0         1         0
              0.0     2088.1         0.0       0.0       0.0       0.0
      SLAB ON GRADE                                            -- PAGE NO.    4
     51       1       0          1         0         1         0
              0.0     6185.8         0.0       0.0       0.0       0.0
     52       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     53       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     54       1       0          1         0         1         0
              0.0     7237.7         0.0       0.0       0.0       0.0
     55       1       0          1         0         1         0
              0.0     6546.9         0.0       0.0       0.0       0.0
     56       1       0          1         0         1         0
              0.0     3406.9         0.0       0.0       0.0       0.0
     57       1       0          1         0         1         0
              0.0     2088.1         0.0       0.0       0.0       0.0
     58       1       0          1         0         1         0
              0.0     6185.8         0.0       0.0       0.0       0.0
     59       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     60       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     61       1       0          1         0         1         0
              0.0     7237.7         0.0       0.0       0.0       0.0
     62       1       0          1         0         1         0
              0.0     6546.9         0.0       0.0       0.0       0.0
     63       1       0          1         0         1         0
              0.0     3406.9         0.0       0.0       0.0       0.0
     64       1       0          1         0         1         0
              0.0     2088.1         0.0       0.0       0.0       0.0
     65       1       0          1         0         1         0
              0.0     6185.8         0.0       0.0       0.0       0.0
     66       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     67       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     68       1       0          1         0         1         0
              0.0     7237.7         0.0       0.0       0.0       0.0
     69       1       0          1         0         1         0
              0.0     6546.9         0.0       0.0       0.0       0.0
     70       1       0          1         0         1         0
              0.0     3406.9         0.0       0.0       0.0       0.0
     71       1       0          1         0         1         0
              0.0     2088.1         0.0       0.0       0.0       0.0
     72       1       0          1         0         1         0
              0.0     6185.8         0.0       0.0       0.0       0.0
     73       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     74       1       0          1         0         1         0
              0.0     8195.4         0.0       0.0       0.0       0.0
     75       1       0          1         0         1         0
              0.0     7237.7         0.0       0.0       0.0       0.0
     76       1       0          1         0         1         0
              0.0     6546.9         0.0       0.0       0.0       0.0
```

```
     77       1          0          1         0        1        0
                 0.0     3406.9        0.0      0.0      0.0      0.0
     78       1          0          1         0        1        0
                 0.0     2437.9        0.0      0.0      0.0      0.0
      SLAB ON GRADE                                             -- PAGE NO.    5
     79       1          0          1         0        1        0
                 0.0     7221.9        0.0      0.0      0.0      0.0
     80       1          0          1         0        1        0
                 0.0     9568.1        0.0      0.0      0.0      0.0
     81       1          0          1         0        1        0
                 0.0     9568.1        0.0      0.0      0.0      0.0
     82       1          0          1         0        1        0
                 0.0     8450.0        0.0      0.0      0.0      0.0
     83       1          0          1         0        1        0
                 0.0     7643.5        0.0      0.0      0.0      0.0
     84       1          0          1         0        1        0
                 0.0     3977.6        0.0      0.0      0.0      0.0
     85       1          0          1         0        1        0
                 0.0     2787.6        0.0      0.0      0.0      0.0
     86       1          0          1         0        1        0
                 0.0     8258.0        0.0      0.0      0.0      0.0
     87       1          0          1         0        1        0
                 0.0    10940.9        0.0      0.0      0.0      0.0
     88       1          0          1         0        1        0
                 0.0    10940.9        0.0      0.0      0.0      0.0
     89       1          0          1         0        1        0
                 0.0     9662.3        0.0      0.0      0.0      0.0
     90       1          0          1         0        1        0
                 0.0     8740.1        0.0      0.0      0.0      0.0
     91       1          0          1         0        1        0
                 0.0     4548.2        0.0      0.0      0.0      0.0
     92       1          0          1         0        1        0
                 0.0     2787.6        0.0      0.0      0.0      0.0
     93       1          0          1         0        1        0
                 0.0     8258.0        0.0      0.0      0.0      0.0
     94       1          0          1         0        1        0
                 0.0    10940.9        0.0      0.0      0.0      0.0
     95       1          0          1         0        1        0
                 0.0    10940.9        0.0      0.0      0.0      0.0
     96       1          0          1         0        1        0
                 0.0     9662.3        0.0      0.0      0.0      0.0
     97       1          0          1         0        1        0
                 0.0     8740.1        0.0      0.0      0.0      0.0
     98       1          0          1         0        1        0
                 0.0     4548.2        0.0      0.0      0.0      0.0
     99       1          0          1         0        1        0
                 0.0     2871.1        0.0      0.0      0.0      0.0
    100       1          0          1         0        1        0
                 0.0     8505.5        0.0      0.0      0.0      0.0
    101       1          0          1         0        1        0
                 0.0    11268.7        0.0      0.0      0.0      0.0
    102       1          0          1         0        1        0
                 0.0    11268.7        0.0      0.0      0.0      0.0
    103       1          0          1         0        1        0
                 0.0     9951.8        0.0      0.0      0.0      0.0
    104       1          0          1         0        1        0
                 0.0     9002.0        0.0      0.0      0.0      0.0
    105       1          0          1         0        1        0
```

```
               0.0    4684.5        0.0      0.0      0.0      0.0
    106       1         0         1         0        1        0
               0.0    2954.7        0.0      0.0      0.0      0.0
     SLAB ON GRADE                                          -- PAGE NO.    6
    107       1         0         1         0        1        0
               0.0    8752.9        0.0      0.0      0.0      0.0
    108       1         0         1         0        1        0
               0.0   11596.5        0.0      0.0      0.0      0.0
    109       1         0         1         0        1        0
               0.0   11596.5        0.0      0.0      0.0      0.0
    110       1         0         1         0        1        0
               0.0   10241.3        0.0      0.0      0.0      0.0
    111       1         0         1         0        1        0
               0.0    9263.9        0.0      0.0      0.0      0.0
    112       1         0         1         0        1        0
               0.0    4820.8        0.0      0.0      0.0      0.0
    113       1         0         1         0        1        0
               0.0    2954.7        0.0      0.0      0.0      0.0
    114       1         0         1         0        1        0
               0.0    8752.9        0.0      0.0      0.0      0.0
    115       1         0         1         0        1        0
               0.0   11596.5        0.0      0.0      0.0      0.0
    116       1         0         1         0        1        0
               0.0   11596.5        0.0      0.0      0.0      0.0
    117       1         0         1         0        1        0
               0.0   10241.3        0.0      0.0      0.0      0.0
    118       1         0         1         0        1        0
               0.0    9263.9        0.0      0.0      0.0      0.0
    119       1         0         1         0        1        0
               0.0    4820.8        0.0      0.0      0.0      0.0
    120       1         0         1         0        1        0
               0.0    1477.3        0.0      0.0      0.0      0.0
    121       1         0         1         0        1        0
               0.0    4376.5        0.0      0.0      0.0      0.0
    122       1         0         1         0        1        0
               0.0    5798.2        0.0      0.0      0.0      0.0
    123       1         0         1         0        1        0
               0.0    5798.2        0.0      0.0      0.0      0.0
    124       1         0         1         0        1        0
               0.0    5120.7        0.0      0.0      0.0      0.0
    125       1         0         1         0        1        0
               0.0    4631.9        0.0      0.0      0.0      0.0
    126       1         0         1         0        1        0
               0.0    2410.4        0.0      0.0      0.0      0.0
   ************ END OF DATA FROM INTERNAL STORAGE ************
    40. LOAD 1 'WEIGHT OF MAT & EARTH'
    41. ELEMENT LOAD
    42. 1 TO 102 PR GY -74.2
    43. LOAD 2 'COLUMN LOAD-DL+LL'
    44. JOINT LOADS
    45. 1 2   FY -965.
    46. 8 9   FY -485.
    47. 5     FY -1373.
    48. 6     FY -2746.
     SLAB ON GRADE                                          -- PAGE NO.    7
    49. 22 23 FY -1824.
    50. 29 30 FY -912.
    51. 26    FY -2414.
```

```
   52. 27    FY -4828.
   53. 43 44 50 51 71 72 78 79 FY -1368.
   54. 47 54 82    FY -1175.
   55. 48 55 76 83 FY -2350.
   56. 92 93       FY -912.
   57. 99 100      FY -1824.
   58. 103         FY -2166.
   59. 104         FY -4333.
   60. 113 114     FY -485.
   61. 120 121     FY -965.
   62. 124         FY -1216.
   63. 125         FY -2431.
   64. LOADING COMBINATION 101 TOTAL LOAD
   65. 1 1. 2 1.
   66. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
    NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     2, TOTAL DEGREES OF FREEDOM =     378
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    254 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
   67. UNIT CM
   68. LOAD LIST 101
   69. PRINT JOINT DISPLACEMENTS LIST 33 56
 JOINT    DISPLACE LIST     33
     SLAB ON GRADE                                            -- PAGE NO.    8
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
    33  101     0.0000  -10.5679    0.0000   -0.0259    0.0000    0.0552
    56  101     0.0000  -12.1989    0.0000    0.0660    0.0000    0.0288
  ************** END OF LATEST ANALYSIS RESULT **************
   70. PRINT ELEMENT STRESSES LIST 34 67
 ELEMENT  STRESSES LIST     34
     SLAB ON GRADE                                            -- PAGE NO.    9
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KNS  CM
  ----------------
           STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY           MX           MY           MXY
                     VONT       VONB          SX           SY           SXY
                     TRESCAT    TRESCAB
    34    101       -0.02       -0.03        2.22        17.75        41.14
                     2.24        2.24        0.00         0.00         0.00
                     2.56        2.56
      TOP : SMAX=       1.59 SMIN=      -0.98 TMAX=       1.28 ANGLE= 50.3
      BOTT: SMAX=       0.98 SMIN=      -1.59 TMAX=       1.28 ANGLE=-39.7
    67    101        0.28        0.03       62.65        39.76        88.62
                     4.99        4.99        0.00         0.00         0.00
                     5.47        5.47
      TOP : SMAX=       4.30 SMIN=      -1.17 TMAX=       2.74 ANGLE= 41.3
      BOTT: SMAX=       1.17 SMIN=      -4.30 TMAX=       2.74 ANGLE=-48.7
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
```

```
            STRESS        STRESS        STRESS        STRESS        STRESS
         4.302792E+00 -4.302792E+00  2.735344E+00  4.990313E+00  5.470689E+00
PLATE NO.      67            67            67            67            67
CASE  NO.     101           101           101           101           101
  ********************END OF ELEMENT FORCES********************
   71. FINISH
     SLAB ON GRADE                                            -- PAGE NO.   10
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:44 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *M. To assign a foundation support* (on page 824)
- *TR.27.3 Automatic Spring Support Generator for Foundations* (on page 2319)
- *Create Support dialog* (on page 2815)

EX. UK-24 Analysis of a Concrete Block Using Solid Elements

This is an example of the analysis of a structure modeled using solid finite elements. This example also illustrates the method for applying an enforced displacement on the structure.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-24 Analysis of a Concrete Block Using Solid Elements.STD` when you install the program.

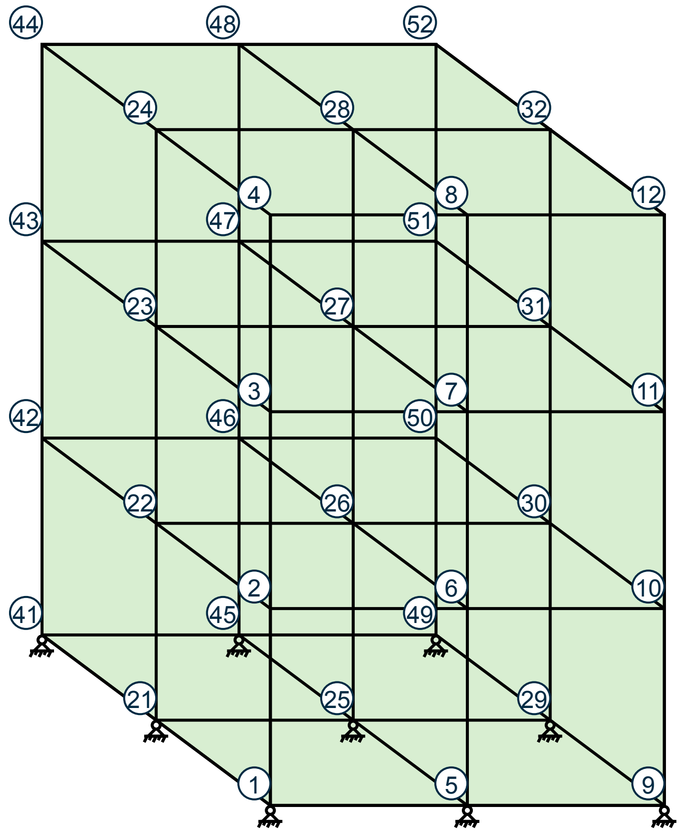


Figure 565: Example Problem No. 24

```
STAAD SPACE
*EXAMPLE PROBLEM USING SOLID ELEMENTS
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. The comment line which begins with an asterisk is an optional title to identify this project.

```
UNIT KNS MET
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
  1  0.0  0.0  2.0   4  0.0  3.0  2.0
  5  1.0  0.0  2.0   8  1.0  3.0  2.0
  9  2.0  0.0  2.0  12  2.0  3.0  2.0
 21  0.0  0.0  1.0  24  0.0  3.0  1.0
 25  1.0  0.0  1.0  28  1.0  3.0  1.0
 29  2.0  0.0  1.0  32  2.0  3.0  1.0
 41  0.0  0.0  0.0  44  0.0  3.0  0.0
 45  1.0  0.0  0.0  48  1.0  3.0  0.0
 49  2.0  0.0  0.0  52  2.0  3.0  0.0
```

The joint number followed by the X, Y, and Z coordinates are specified above. The coordinates of some of those nodes are generated utilizing the fact that they are equally spaced between the extremities.

```
ELEMENT INCIDENCES SOLID
  1    1   5   6   2  21  25  26  22   TO   3
```

```
     4   21  25  26  22  41  45  46  42   TO   6  1 1
     7    5   9  10   6  25  29  30  26   TO   9  1 1
    10   25  29  30  26  45  49  50  46   TO  12  1 1
```

The incidences of solid elements are defined above. The word `SOLID` is used to signify that these are 8-node solid elements as opposed to 3-noded or 4-noded plate elements. Each line contains the data for generating 3 elements. For example, element number 1 is first defined by all of its 8 nodes. Then, increments of 1 to the joint number and 1 to the element number (the defaults) are used for generating incidences for elements 2 and 3. Similarly, incidences of elements 4, 7 and 10 are defined while those of 5, 6, 8, 9, 11 and 12 are generated.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METERL
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
PRINT ELEMENT INFO SOLID LIST 1 TO 5
```

This command will enable us to obtain, in a tabular form, the details of the incidences and material property values of elements 1 to 5.

```
SUPPORTS
1 5 21 25 29 41 45 49 PINNED
9 ENFORCED
```

The above lines contain the data for supports for the model. The ENFORCED support condition is used to declare a point at which an enforced displacement load is applied later (see load case 3).

```
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
```

The above data describe a static load case. It consists of selfweight loading and a joint load, both in the negative global Y direction.

```
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24 42 TO 44 FX 100.0
```

Load case 2 consists of several joint loads acting in the positive global X direction.

```
LOAD 3
SUPPORT DISPLACEMENT
9 FX 0.0011
```

Load case 3 consists of an enforced displacement along the global X direction at node 9. The displacement in the other enforced support degrees of freedom will default to zero.

```
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
```

In Load case 4, a pressure load of 500 pounds/sq.ft is applied on Face # 4 of solid elements 3, 6, 9 and 12. Face 4 is defined as shown in the following table :

Face Number	Surface Joints			
	f_1	f_2	f_3	f_4
1 front	Jt 1	Jt 4	Jt 3	Jt 2
2 bottom	Jt 1	Jt 2	Jt 6	Jt 5
3 left	Jt 1	Jt 5	Jt 8	Jt 4
4 top	Jt 4	Jt 8	Jt 7	Jt 3
5 right	Jt 2	Jt 3	Jt 7	Jt 6
6 back	Jt 5	Jt 6	Jt 7	Jt 8

The above table, and other details of this type of loading can be found in TR.32.3.2 Element Load Specification - Solids (on page 2472) .

```
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
```

Load case 5 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We want the program to analyze the structure for loads from cases 1 through 4 acting simultaneously. In other words, the above instruction is the same as the following:

```
LOAD 5
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
2 TO 4 22 TO 24 42 TO 44 FX 100.0
SUPPORT DISPLACEMENT
9 FX .0011
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
LOAD COMB 10
1 1.0 2 1.0
```

Load case 10 is a combination load case, which combines the effects of cases 1 & 2. While the syntax of this might look very similar to that of the `REPEAT LOAD` case shown in case 5, there is a fundamental difference. In a `REPEAT LOAD` case, the program computes the displacements by multiplying the inverted stiffness matrix by the load vector built for the `REPEAT LOAD` case. But in solving load combination cases, the program merely calculates the end results (displacements, forces, reactions) by gathering up the corresponding values from the individual components of the combination case, factoring them, and then algebraically summing them up. This difference in approach is quite important in that non-linear problems such as `PDELTA ANALYSIS`, `MEMBER TENSION`, and `MEMBER COMPRESSION` situations, changes in support conditions etc. should be handled using `REPEAT LOAD` cases, *not* load combination cases.

```
    PERFORM ANALYSIS PRINT STATICS CHECK
```

A static equilibrium report, consisting of total applied loading and total support reactions from each primary load case is requested along with the instructions to carry out a linear static analysis.

```
    PRINT JOINT DISPLACEMENTS LIST 8 9
```

Global displacements at nodes 8 and 9 are obtained using the above command.

```
    UNIT KNS METER
    PRINT SUPPORT REACTIONS
```

Reactions at the supports are obtained using the above command.

```
    UNIT NEWTON MMS
    PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
```

This command requests the program to provide the element stress results at the nodes of elements 4 and 6. The results will be printed for all the load cases. The word SOLID is used to signify that these are solid elements as opposed to plate or shell elements.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
UNIT KNS MET
JOINT COORDINATES
  1  0.0  0.0  2.0   4  0.0  3.0  2.0
  5  1.0  0.0  2.0   8  1.0  3.0  2.0
  9  2.0  0.0  2.0  12  2.0  3.0  2.0
 21  0.0  0.0  1.0  24  0.0  3.0  1.0
 25  1.0  0.0  1.0  28  1.0  3.0  1.0
 29  2.0  0.0  1.0  32  2.0  3.0  1.0
 41  0.0  0.0  0.0  44  0.0  3.0  0.0
 45  1.0  0.0  0.0  48  1.0  3.0  0.0
 49  2.0  0.0  0.0  52  2.0  3.0  0.0
ELEMENT INCIDENCES SOLID
  1    1   5   6   2  21  25  26  22   TO   3
  4   21  25  26  22  41  45  46  42   TO   6  1 1
  7    5   9  10   6  25  29  30  26   TO   9  1 1
 10   25  29  30  26  45  49  50  46   TO  12  1 1
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.25
DENSITY 7.5e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
```

```
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
PRINT ELEMENT INFO SOLID LIST 1 TO 5
SUPPORTS
1  5   21  25  29  41  45  49  PINNED
9 ENFORCED BUT MX MY MZ
LOAD 1
SELF Y -1.0
JOINT LOAD
28 FY -1000.0
LOAD 2
JOINT LOADS
2 TO 4 22 TO 24  42 TO 44 FX 100.0
LOAD 3
SUPPORT DISPLACEMENT
9 FX .0011
UNIT POUND FEET
LOAD 4
ELEMENT LOAD SOLIDS
3 6 9 12 FACE 4 PRE GY -500.0
UNIT KNS MMS
LOAD 5
REPEAT LOAD
1 1.0 2 1.0 3 1.0 4 1.0
LOAD COMB 10
1 1.0 2 1.0
PERFORM ANALYSIS PRINT STAT CHECK
PRINT JOINT DISPLACEMENTS LIST 8 9
UNIT KNS METER
PRINT SUPPORT REACTIONS
UNIT NEWTON MMS
PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:46                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE EXAMPLE PROBLEM USING SOLID ELEMENTS
INPUT FILE: UK-24 Analysis of a Concrete Block Using Solid Elements.STD
     2. UNIT KNS MET
     3. JOINT COORDINATES
     4. 1  0.0  0.0  2.0   4  0.0  3.0  2.0
     5. 5  1.0  0.0  2.0   8  1.0  3.0  2.0
     6. 9  2.0  0.0  2.0  12  2.0  3.0  2.0
```

```
    7. 21  0.0  0.0  1.0  24  0.0  3.0  1.0
    8. 25  1.0  0.0  1.0  28  1.0  3.0  1.0
    9. 29  2.0  0.0  1.0  32  2.0  3.0  1.0
   10. 41  0.0  0.0  0.0  44  0.0  3.0  0.0
   11. 45  1.0  0.0  0.0  48  1.0  3.0  0.0
   12. 49  2.0  0.0  0.0  52  2.0  3.0  0.0
   13. ELEMENT INCIDENCES SOLID
   14. 1    1   5   6   2  21  25  26  22   TO   3
   15. 4   21  25  26  22  41  45  46  42   TO   6  1 1
   16. 7    5   9  10   6  25  29  30  26   TO   9  1 1
   17. 10   25  29  30  26  45  49  50  46   TO  12  1 1
   18. UNIT MMS
   19. DEFINE MATERIAL START
   20. ISOTROPIC STEEL
   21. E 210
   22. POISSON 0.25
   23. DENSITY 7.5E-008
   24. ALPHA 6E-006
   25. DAMP 0.03
   26. TYPE STEEL
   27. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
   28. END DEFINE MATERIAL
   29. CONSTANTS
   30. MATERIAL STEEL ALL
   31. UNIT METER
   32. PRINT ELEMENT INFO SOLID LIST 1 TO 5
 ELEMENT  INFO     SOLID    LIST
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    2
ELEMENT NODE-1  NODE-2  NODE-3  NODE-4  NODE-5  NODE-6  NODE-7  NODE-8
      1      1       5       6       2      21      25      26      22
      2      2       6       7       3      22      26      27      23
      3      3       7       8       4      23      27      28      24
      4     21      25      26      22      41      45      46      42
      5     22      26      27      23      42      46      47      43
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    3
  MATERIAL PROPERTIES.
  --------------------
  ALL UNITS ARE - KNS  METE
ELEMENT   YOUNG'S MODULUS   MODULUS OF RIGIDITY    DENSITY        ALPHA
      1      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
      2      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
      3      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
      4      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
      5      2.1000002E+08         0.0000000E+00  7.5000E+01   6.0000E-06
   33. SUPPORTS
   34. 1  5   21  25  29  41  45  49  PINNED
   35. 9 ENFORCED BUT MX MY MZ
   36. LOAD 1
   37. SELF Y -1.0
   38. JOINT LOAD
   39. 28 FY -1000.0
   40. LOAD 2
   41. JOINT LOADS
   42. 2 TO 4 22 TO 24  42 TO 44 FX 100.0
   43. LOAD 3
   44. SUPPORT DISPLACEMENT
   45. 9 FX .0011
   46. UNIT POUND FEET
```

```
   47. LOAD 4
   48. ELEMENT LOAD SOLIDS
   49. 3 6 9 12 FACE 4 PRE GY -500.0
   50. UNIT KNS MMS
   51. LOAD 5
   52. REPEAT LOAD
   53. 1 1.0 2 1.0 3 1.0 4 1.0
   54. LOAD COMB 10
   55. 1 1.0 2 1.0
   56. PERFORM ANALYSIS PRINT STAT CHECK
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         36  NUMBER OF MEMBERS       0
    NUMBER OF PLATES          0  NUMBER OF SOLIDS       12
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      9
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    4
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     5, TOTAL DEGREES OF FREEDOM =      84
  TOTAL LOAD COMBINATION  CASES =     1  SO FAR.
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    5
  *** NOTE: CAPACITY FOR MAXIMUM #    260 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.999999993E+03
                       Y =  0.228947364E+04
                       Z =  0.999999993E+03
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =      -1900.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     1900000.15  MY=           0.00  MZ=    -1900000.15
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =       1900.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=    -1900000.15  MY=          -0.00  MZ=     1900000.15
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      1)
           MAXIMUMS    AT NODE
     X = -1.21106E-04      23
     Y = -1.15439E-03      28
     Z =  1.21106E-04       7
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
          CENTER OF FORCE BASED ON X FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.000000000E+00
                       Y =  0.199999999E+04
                       Z =  0.999999993E+03
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    6
TOTAL APPLIED LOAD     2
```

```
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =        900.00
      SUMMATION FORCE-Y =          0.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=      900000.03  MZ=    -1800000.06
TOTAL REACTION LOAD    2
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =       -900.00
      SUMMATION FORCE-Y =         -0.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=           0.00  MY=     -900000.03  MZ=     1800000.06
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      2)
            MAXIMUMS    AT NODE
     X =  2.22892E-03       4
     Y =  7.83934E-04      44
     Z =  9.49033E-05      10
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     3
TOTAL APPLIED LOAD     3
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  0.0000000E+00
      SUMMATION FORCE-Y =  0.0000000E+00
      SUMMATION FORCE-Z =  0.0000000E+00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=  0.0000000E+00  MY=  0.0000000E+00  MZ=  0.0000000E+00
TOTAL REACTION LOAD    3
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =  1.6182536E-11
      SUMMATION FORCE-Y = -5.0570426E-12
      SUMMATION FORCE-Z =  2.6296621E-11
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=  5.5900954E-08  MY=  3.9459497E-08  MZ= -3.2882914E-09
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    7
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      3)
            MAXIMUMS    AT NODE
     X =  1.10000E-01       9
     Y = -1.21497E-02       6
     Z =  1.61372E-02      24
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     4
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                      X =  0.999999993E+03
                      Y =  0.299999998E+04
                      Z =  0.999999993E+03
TOTAL APPLIED LOAD     4
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     4 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =        -95.76
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=       95760.52  MY=           0.00  MZ=      -95760.52
```

```
TOTAL REACTION LOAD    4
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     4 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =         95.76
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      -95760.52  MY=           0.00  MZ=       95760.52
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      4)
            MAXIMUMS    AT NODE
     X =  3.17652E-06      50
     Y = -3.35288E-05      28
     Z = -3.17652E-06      50
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     5
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    8
          CENTER OF FORCE BASED ON X FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                     X =  0.000000000E+00
                     Y =  0.199999999E+04
                     Z =  0.999999993E+03
          CENTER OF FORCE BASED ON Y FORCES ONLY (MMS ).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                     X =  0.999999993E+03
                     Y =  0.232356609E+04
                     Z =  0.999999993E+03
TOTAL APPLIED LOAD     5
  ***TOTAL APPLIED LOAD ( KNS  MMS  ) SUMMARY (LOADING     5 )
      SUMMATION FORCE-X =        900.00
      SUMMATION FORCE-Y =      -1995.76
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     1995760.67  MY=      900000.03  MZ=    -3795760.73
TOTAL REACTION LOAD    5
  ***TOTAL REACTION LOAD( KNS  MMS  ) SUMMARY (LOADING     5 )
      SUMMATION FORCE-X =       -900.00
      SUMMATION FORCE-Y =       1995.76
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=    -1995760.67  MY=     -900000.03  MZ=     3795760.73
  MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      5)
            MAXIMUMS    AT NODE
     X =  1.10000E-01       9
     Y = -1.23568E-02       6
     Z =  1.61372E-02      24
     RX=  0.00000E+00       0
     RY=  0.00000E+00       0
     RZ=  0.00000E+00       0
  ************ END OF DATA FROM INTERNAL STORAGE ************
   57. PRINT JOINT DISPLACEMENTS LIST 8 9
 JOINT    DISPLACE LIST     8
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.    9
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     8    1     0.0000   -0.0002   -0.0001    0.0000    0.0000    0.0000
          2     0.0020    0.0000   -0.0000    0.0000    0.0000    0.0000
```

```
          3     0.0193   -0.0049    0.0089    0.0000    0.0000    0.0000
          4     0.0000   -0.0000    0.0000    0.0000    0.0000    0.0000
          5     0.0213   -0.0052    0.0088    0.0000    0.0000    0.0000
         10     0.0020   -0.0002   -0.0001    0.0000    0.0000    0.0000
     9    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          3     0.1100    0.0000   -0.0000    0.0000    0.0000    0.0000
          4     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
          5     0.1100    0.0000   -0.0000    0.0000    0.0000    0.0000
         10     0.0000    0.0000    0.0000    0.0000    0.0000    0.0000
  ************** END OF LATEST ANALYSIS RESULT **************
   58. UNIT KNS METER
   59. PRINT SUPPORT REACTIONS
 SUPPORT  REACTION
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   10
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     27.47    128.97    -27.47      0.00      0.00      0.00
          2    -72.24   -232.67     42.18      0.00      0.00      0.00
          3  -2022.70   -302.04  -1192.39      0.00      0.00      0.00
          4      1.52      6.63     -1.52      0.00      0.00      0.00
          5  -2065.94   -399.11  -1179.21      0.00      0.00      0.00
         10    -44.76   -103.70     14.70      0.00      0.00      0.00
     5    1      0.00    236.52    -54.44      0.00      0.00      0.00
          2    -62.32     11.42     -0.05      0.00      0.00      0.00
          3 -16410.02   7434.80  -2287.95      0.00      0.00      0.00
          4      0.00     11.97     -2.98      0.00      0.00      0.00
          5 -16472.33   7694.71  -2345.41      0.00      0.00      0.00
         10    -62.32    247.94    -54.49      0.00      0.00      0.00
    21    1     54.44    236.52     -0.00      0.00      0.00      0.00
          2   -159.92   -450.84     -0.00      0.00      0.00      0.00
          3  -3341.67  -2923.60  -1877.00      0.00      0.00      0.00
          4      2.98     11.97     -0.00      0.00      0.00      0.00
          5  -3444.18  -3125.95  -1877.00      0.00      0.00      0.00
         10   -105.49   -214.32     -0.00      0.00      0.00      0.00
    25    1     -0.00    438.06     -0.00      0.00      0.00      0.00
          2   -138.00      9.51     -0.00      0.00      0.00      0.00
          3 -19197.98   5248.66 -10975.25      0.00      0.00      0.00
          4     -0.00     21.34     -0.00      0.00      0.00      0.00
          5 -19335.98   5717.57 -10975.25      0.00      0.00      0.00
         10   -138.00    447.56     -0.00      0.00      0.00      0.00
    29    1    -54.44    236.52      0.00      0.00      0.00      0.00
          2   -170.27    431.34      0.00      0.00      0.00      0.00
          3   3902.73    512.05   3842.64      0.00      0.00      0.00
          4     -2.98     11.97      0.00      0.00      0.00      0.00
          5   3675.05   1191.87   3842.64      0.00      0.00      0.00
         10   -224.70    667.85      0.00      0.00      0.00      0.00
    41    1     27.47    128.97     27.47      0.00      0.00      0.00
          2    -72.24   -232.67    -42.18      0.00      0.00      0.00
          3   -891.15  -2739.86  -1598.54      0.00      0.00      0.00
          4      1.52      6.63      1.52      0.00      0.00      0.00
          5   -934.39  -2836.93  -1611.72      0.00      0.00      0.00
         10    -44.76   -103.70    -14.70      0.00      0.00      0.00
    45    1     -0.00    236.52     54.44      0.00      0.00      0.00
          2    -62.32     11.42      0.05      0.00      0.00      0.00
          3   -430.44   -752.46   -237.57      0.00      0.00      0.00
          4     -0.00     11.97      2.98      0.00      0.00      0.00
```

```
          5   -492.75   -492.56   -180.10      0.00      0.00      0.00
         10    -62.32    247.94     54.49      0.00      0.00      0.00
    49    1    -27.47    128.97     27.47      0.00      0.00      0.00
          2    -81.35    226.24     45.03      0.00      0.00      0.00
          3   -778.26   2073.77   1192.39      0.00      0.00      0.00
          4     -1.52      6.63      1.52      0.00      0.00      0.00
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   11
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
          5   -888.61   2435.62   1266.41      0.00      0.00      0.00
         10   -108.83    355.21     72.50      0.00      0.00      0.00
     9    1    -27.47    128.97    -27.47      0.00      0.00      0.00
          2    -81.35    226.24    -45.03      0.00      0.00      0.00
          3  39169.49  -8551.31  13133.66      0.00      0.00      0.00
          4     -1.52      6.63     -1.52      0.00      0.00      0.00
          5  39059.14  -8189.46  13059.64      0.00      0.00      0.00
         10   -108.83    355.21    -72.50      0.00      0.00      0.00
  ************** END OF LATEST ANALYSIS RESULT **************
   60. UNIT NEWTON MMS
   61. PRINT ELEMENT JOINT STRESS SOLID LIST 4 6
 ELEMENT  JOINT    STRESS   SOLID
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   12
ELEMENT STRESSES        UNITS= NEWTMMS
-------------------------------------------------------------------------------
             NODE/           NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
-------------------------------------------------------------------------------
    4    1     21     -0.144     -0.449     -0.155     -0.006     -0.011      0.000
    4    1     25     -0.132     -0.368     -0.132     -0.011     -0.011      0.005
    4    1     26     -0.009     -0.377     -0.009     -0.003     -0.003      0.005
    4    1     22     -0.012     -0.449     -0.005      0.002     -0.018      0.009
    4    1     41     -0.152     -0.484     -0.152     -0.015     -0.015     -0.005
    4    1     45     -0.155     -0.449     -0.144     -0.011     -0.006     -0.000
    4    1     46     -0.005     -0.449     -0.012     -0.018      0.002      0.009
    4    1     42      0.007     -0.475      0.007     -0.023     -0.023      0.014
    4    1 CENTER     -0.075     -0.437     -0.075     -0.011     -0.011      0.005
             S1=     -0.070   S2=     -0.080   S3=     -0.438   SE=      0.363
             DC=      0.707     -0.041      0.707     -0.707     -0.000      0.707
    4    2     21      0.176      1.021      0.284      0.217      0.014      0.005
    4    2     25      0.154     -0.006      0.022      0.251      0.014     -0.029
    4    2     26     -0.028      0.053     -0.015      0.253      0.016     -0.002
    4    2     22     -0.054      1.031      0.103      0.219      0.012     -0.036
    4    2     41      0.189      1.034      0.321      0.258      0.038      0.029
    4    2     45      0.162     -0.006      0.054      0.223     -0.010     -0.005
    4    2     46     -0.225     -0.016     -0.051      0.221     -0.008     -0.026
    4    2     42     -0.247      0.976      0.071      0.255      0.036     -0.060
    4    2 CENTER      0.016      0.511      0.099      0.237      0.014     -0.015
             S1=      0.606   S2=      0.101   S3=     -0.082   SE=      0.617
             DC=      0.372      0.928      0.014     -0.106      0.027      0.994
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   13
ELEMENT STRESSES        UNITS= NEWTMMS
-------------------------------------------------------------------------------
             NODE/           NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
-------------------------------------------------------------------------------
    4    3     21      0.900      5.181      1.884      4.989      5.058      0.396
    4    3     25     -0.893     -5.740     -1.294      6.615      1.429     -1.229
```

```
    4    3     26      5.251     -3.282      3.647      5.654      0.468      3.274
    4    3     22      5.379      5.974      1.830      4.029      6.019      1.649
    4    3     41      2.148      9.507      2.550      0.107      5.891      1.229
    4    3     45      2.276      4.348      1.292     -1.518      0.596     -0.396
    4    3     46     -1.334      3.555      2.982     -0.558     -0.364      2.442
    4    3     42     -3.127      7.049     -0.756      1.067      6.851      0.816
    4    3 CENTER      1.325      3.324      1.517      2.548      3.244      1.023
             S1=      7.030   S2=      0.411   S3=     -1.275   SE=       7.604
             DC=       0.425      0.744      0.516      0.809     -0.055     -0.586
    4    4     21     -0.008     -0.024     -0.008     -0.001     -0.001     -0.000
    4    4     25     -0.008     -0.022     -0.008     -0.001     -0.001      0.000
    4    4     26      0.001     -0.022      0.001     -0.001     -0.001      0.000
    4    4     22      0.001     -0.024      0.001     -0.001     -0.001      0.000
    4    4     41     -0.008     -0.026     -0.008     -0.001     -0.001     -0.000
    4    4     45     -0.008     -0.024     -0.008     -0.001     -0.001     -0.000
    4    4     46      0.001     -0.024      0.001     -0.001     -0.001      0.000
    4    4     42      0.001     -0.026      0.001     -0.001     -0.001      0.000
    4    4 CENTER     -0.004     -0.024     -0.004     -0.001     -0.001      0.000
             S1=     -0.003   S2=     -0.004   S3=     -0.024   SE=       0.021
             DC=       0.705     -0.070      0.705     -0.707      0.000      0.707
    4    5     21      0.925      5.729      2.005      5.199      5.061      0.402
    4    5     25     -0.878     -6.136     -1.412      6.854      1.431     -1.254
    4    5     26      5.215     -3.629      3.624      5.903      0.481      3.277
    4    5     22      5.313      6.532      1.928      4.248      6.011      1.622
     EXAMPLE PROBLEM USING SOLID ELEMENTS                    -- PAGE NO.   14
ELEMENT STRESSES        UNITS= NEWTMMS
-----------------------------------------------------------------------------
             NODE/           NORMAL STRESSES               SHEAR STRESSES
ELEMENT LOAD CENTER      SXX        SYY        SZZ        SXY        SYZ        SZX
-----------------------------------------------------------------------------
    4    5     41      2.176     10.031      2.710      0.349      5.913      1.254
    4    5     45      2.275      3.869      1.194     -1.307      0.579     -0.402
    4    5     46     -1.564      3.066      2.920     -0.356     -0.371      2.425
    4    5     42     -3.366      7.524     -0.677      1.299      6.864      0.770
    4    5 CENTER      1.262      3.373      1.537      2.774      3.246      1.012
             S1=      7.193   S2=      0.379   S3=     -1.400   SE=       7.856
             DC=       0.435      0.745      0.505     -0.764      0.008      0.645
    4   10     21      0.032      0.572      0.129      0.211      0.004      0.005
    4   10     25      0.022     -0.374     -0.110      0.240      0.004     -0.024
    4   10     26     -0.038     -0.325     -0.024      0.250      0.013      0.003
    4   10     22     -0.067      0.582      0.098      0.221     -0.006     -0.027
    4   10     41      0.036      0.550      0.168      0.242      0.023      0.024
    4   10     45      0.007     -0.455     -0.090      0.213     -0.016     -0.005
    4   10     46     -0.230     -0.465     -0.063      0.203     -0.006     -0.017
    4   10     42     -0.240      0.501      0.078      0.233      0.013     -0.046
    4   10 CENTER     -0.060      0.073      0.023      0.227      0.004     -0.011
             S1=      0.243   S2=      0.024   S3=     -0.230   SE=       0.410
             DC=       0.600      0.800     -0.017     -0.024      0.039      0.999
    6    1     23      0.329      0.394      0.413     -0.043     -0.127     -0.060
    6    1     27     -0.071     -1.739     -0.071     -0.099     -0.099     -0.005
    6    1     28     -0.676     -1.849     -0.676     -0.553     -0.553     -0.005
    6    1     24     -0.166      0.394      0.140     -0.498      0.328      0.051
    6    1     43     -0.097     -0.200     -0.097     -0.182     -0.182     -0.115
    6    1     47      0.413      0.394      0.329     -0.127     -0.043     -0.060
    6    1     48      0.140      0.394     -0.166      0.328     -0.498      0.051
     EXAMPLE PROBLEM USING SOLID ELEMENTS                    -- PAGE NO.   15
ELEMENT STRESSES        UNITS= NEWTMMS
-----------------------------------------------------------------------------
```

```
            NODE/          NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ       SZX
-------------------------------------------------------------------------------
    6    1     44      -0.259    -0.089    -0.259     0.273     0.273     0.106
    6    1 CENTER      -0.049    -0.287    -0.049    -0.113    -0.113    -0.005
            S1=       0.027   S2=     -0.044   S3=     -0.368   SE=       0.365
            DC=        0.631    -0.451     0.631    -0.707    -0.000     0.707
    6    2     23      -0.032     0.112    -0.001     0.030    -0.002     0.016
    6    2     27      -0.001    -0.025    -0.046     0.073    -0.013    -0.027
    6    2     28      -0.096    -0.003    -0.065     0.083    -0.003    -0.035
    6    2     24      -0.085     0.177     0.109     0.040    -0.012    -0.078
    6    2     43      -0.152     0.158     0.052     0.136    -0.023    -0.005
    6    2     47      -0.140    -0.041    -0.013     0.092     0.008    -0.049
    6    2     48      -0.496    -0.105    -0.119     0.082     0.019    -0.014
    6    2     44      -0.464     0.136     0.076     0.125    -0.033    -0.057
    6    2 CENTER      -0.183     0.051    -0.001     0.083    -0.007    -0.031
            S1=       0.081   S2=     -0.001   S3=     -0.213   SE=       0.263
            DC=        0.314     0.928    -0.202    -0.060     0.232     0.971
    6    3     23      -2.744    -0.535    -0.041    -0.327    -0.468     0.408
    6    3     27      -3.140    -0.556    -1.018     0.642     0.296    -0.560
    6    3     28       1.815     0.568     0.607     0.402     0.056     0.214
    6    3     24       1.900     0.279     0.654    -0.567    -0.228    -0.755
    6    3     43       0.636    -0.478     0.687    -0.031    -0.313     0.563
    6    3     47       0.721     0.942     0.191    -0.999     0.141    -0.405
    6    3     48      -0.136     0.128    -0.121    -0.759    -0.099     0.058
    6    3     44      -0.531    -1.602    -0.555     0.210    -0.073    -0.910
    6    3 CENTER      -0.185    -0.157     0.050    -0.179    -0.086    -0.173
            S1=       0.143   S2=     -0.010   S3=     -0.424   SE=       0.508
            DC=       -0.484     0.038     0.874    -0.507     0.802    -0.316
    EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   16
ELEMENT STRESSES        UNITS= NEWTMMS
-------------------------------------------------------------------------------
            NODE/          NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ       SZX
-------------------------------------------------------------------------------
    6    4     23       0.000    -0.024     0.000    -0.000    -0.000    -0.000
    6    4     27       0.000    -0.024     0.000    -0.000    -0.000    -0.000
    6    4     28      -0.000    -0.024    -0.000    -0.000    -0.000    -0.000
    6    4     24      -0.000    -0.024    -0.000    -0.000    -0.000     0.000
    6    4     43       0.000    -0.024     0.000    -0.000    -0.000    -0.000
    6    4     47       0.000    -0.024     0.000    -0.000    -0.000    -0.000
    6    4     48      -0.000    -0.024    -0.000    -0.000    -0.000     0.000
    6    4     44      -0.000    -0.024    -0.000    -0.000    -0.000     0.000
    6    4 CENTER       0.000    -0.024     0.000    -0.000    -0.000    -0.000
            S1=       0.000   S2=     -0.000   S3=     -0.024   SE=       0.024
            DC=       -0.707     0.000     0.707     0.707    -0.002     0.707
    6    5     23      -2.448    -0.052     0.370    -0.340    -0.596     0.364
    6    5     27      -3.211    -2.343    -1.135     0.616     0.185    -0.592
    6    5     28       1.043    -1.309    -0.134    -0.068    -0.500     0.174
    6    5     24       1.649     0.826     0.902    -1.025     0.089    -0.782
    6    5     43       0.387    -0.545     0.642    -0.077    -0.518     0.443
    6    5     47       0.994     1.271     0.506    -1.034     0.106    -0.514
    6    5     48      -0.492     0.393    -0.406    -0.349    -0.578     0.096
    6    5     44      -1.255    -1.580    -0.739     0.608     0.167    -0.861
    6    5 CENTER      -0.417    -0.417     0.001    -0.209    -0.206    -0.209
            S1=       0.117   S2=     -0.208   S3=     -0.741   SE=       0.750
            DC=       -0.265    -0.255     0.930     0.705    -0.710     0.006
    6   10     23       0.296     0.507     0.412    -0.013    -0.128    -0.044
```

```
    6   10     27     -0.071    -1.764    -0.117    -0.025    -0.112    -0.032
    6   10     28     -0.773    -1.853    -0.741    -0.470    -0.556    -0.039
    6   10     24     -0.251     0.572     0.249    -0.458     0.316    -0.027
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   17
ELEMENT STRESSES        UNITS= NEWTMMS
-------------------------------------------------------------------------------
             NODE/           NORMAL STRESSES              SHEAR STRESSES
ELEMENT LOAD CENTER      SXX       SYY       SZZ       SXY       SYZ       SZX
-------------------------------------------------------------------------------
    6   10     43     -0.249    -0.043    -0.045    -0.046    -0.205    -0.121
    6   10     47      0.272     0.354     0.315    -0.034    -0.035    -0.109
    6   10     48     -0.356     0.289    -0.285     0.410    -0.479     0.037
    6   10     44     -0.724     0.047    -0.184     0.398     0.239     0.050
    6   10 CENTER     -0.232    -0.236    -0.050    -0.030    -0.120    -0.036
             S1=      0.011   S2=     -0.212   S3=     -0.316   SE=      0.289
             DC=     -0.080    -0.428     0.900     0.888    -0.441    -0.131
   62. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:46 ****
     EXAMPLE PROBLEM USING SOLID ELEMENTS                     -- PAGE NO.   18
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Related Links

- *M. To assign an enforced support* (on page 820)
- *TR.27.1 Global Support Specification* (on page 2316)
- *Create Support dialog* (on page 2815)

EX. UK-25 Analysis of a Structure with Compression-Only Members

This example demonstrates the usage of compression-only members. Since the structural condition is load dependent, the `PERFORM ANALYSIS` command is specified once for each primary load case.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-25 Analysis of a Structure with Compression-Only Members.STD` when you install the program.

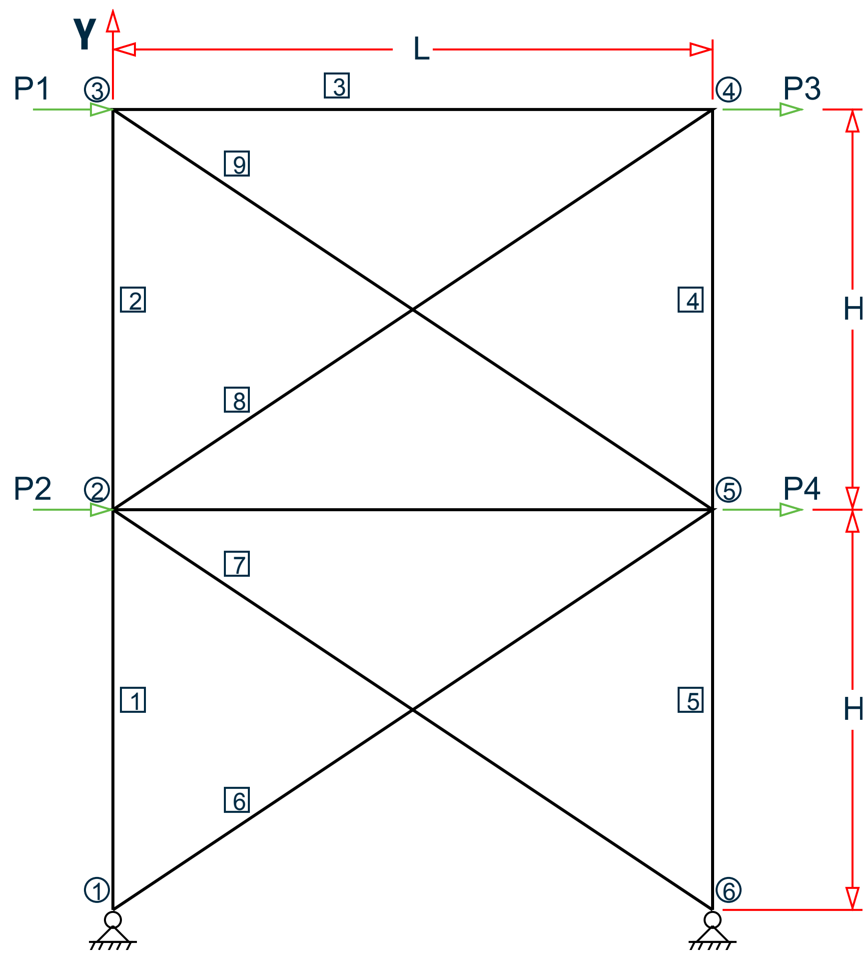


Figure 566: Example Problem No. 25

This example has been created to illustrate the command specification for a structure with certain members capable of carrying compressive force only. It is important to note that the analysis can be done for only 1 load case at a time. This is because, the set of "active" members (and hence the stiffness matrix) is load case dependent.

where:

L = 5.25 m, H = 3.5 m
Load case 1: P1 = 45 kN & P2 = 70 kN
Load case 2: P3 = -45 kN & P4 = -70 kN

```
STAAD PLANE
* EXAMPLE FOR COMPRESSION-ONLY MEMBERS
```

The input data is initiated with the word STAAD. This structure is a PLANE frame. The second line is an optional comment line.

```
UNIT METER KNS
```

Units for the commands to follow are specified above.

```
JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
```

Joint coordinates of joints 1 to 6 are defined above.

```
MEMBER INCIDENCES
1 1 2 5 ; 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

The members 1 to 10 are defined along with the joints to which they are connected.

```
MEMBER COMPRESSION
6 TO 9
```

Members 6 to 9 are defined as COMPRESSION-only members. Hence for each load case, if during the analysis, any of the members 6 to 9 is found to be carrying a tensile force, it is disabled from the structure and the analysis is carried out again with the modified structure.

```
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
```

Members 1 to 10 are assigned the UC152X152X30 section from the British steel table.

```
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members.

```
SUPPORT
1 6 PINNED
```

Joints 1 and 6 are declared as pinned-supported.

```
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
```

Load 1 is defined above and consists of joint loads in the global X direction at joints 2 and 3.

```
PERFORM ANALYSIS
```

The above structure is analyzed for load case 1.

```
CHANGE
MEMBER COMPRESSION
6 TO 9
```

One or more among the members 6 to 9 may have been in-activated in the previous analysis. The `CHANGE` command restores the original structure to prepare it for the analysis for the next primary load case. The members with the compression-only attribute are specified again.

```
LOAD 2
JOINT LOAD
```

```
    4 FX -45
    5 FX -70
```

In load case 2, joint loads are applied in the negative global X direction at joints 4 and 5.

```
    PERFORM ANALYSIS
    CHANGE
```

The instruction to analyze the structure is specified again. Next, any compression-only members that were inactivated during the second analysis (due to the fact that they were subjected to tensile axial forces) are re-activated with the `CHANGE` command. Without the re-activation, these members cannot be accessed for further processing.

```
    LOAD 3
    REPEAT LOAD
    1 1.0 2 1.0
```

Load case 3 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases already specified earlier. We would like the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. In other words, the above instruction is the same as the following:

```
    LOAD 3
    JOINT LOAD
    2 FX 70
    3 FX 45
    4 FX -45
    5 FX -70
    PERFORM ANALYSIS
```

The analysis is carried out for load case 3.

```
    CHANGE
```

The members inactivated during the analysis of load case 3 are re-activated for further processing.

```
    LOAD LIST ALL
```

At the end of any analysis, only those load cases for which the analysis was done most recently, are recognized as the “active” load cases. The `LOAD LIST ALL` command enables all the load cases in the structure to be made active for further processing.

```
    PRINT ANALYSIS RESULTS
```

The program is instructed to write the joint displacements, support reactions and member forces to the output file.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD PLANE
* EXAMPLE FOR COMPRESSION-ONLY MEMBERS
UNIT METER KNS
SET NL 3
JOINT COORDINATES
1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
MEMBER INCIDENCES
1 1 2 5
6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
```

```
MEMBER COMPRESSION
6 TO 9
MEMBER PROPERTY BRITISH
1 TO 10 TA ST UC152X152X30
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 210
POISSON 0.3
DENSITY 7.6977e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
UNIT METER
SUPPORT
1 6 PINNED
LOAD 1
JOINT LOAD
2 FX 70
3 FX 45
PERFORM ANALYSIS
CHANGE
MEMBER COMPRESSION
6 TO 9
LOAD 2
JOINT LOAD
4 FX -45
5 FX -70
PERFORM ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.0 2 1.0
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ANALYSIS RESULTS
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:48                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD PLANE
```

```
INPUT FILE: UK-25 Analysis of a Structure with Compression-Only Members.STD
     2. * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
     3. UNIT METER KNS
     4. SET NL 3
     5. JOINT COORDINATES
     6. 1 0 0 ; 2 0 3.5 ; 3 0 7.0 ; 4 5.25 7.0 ; 5 5.25 3.5 ; 6 5.25 0
     7. MEMBER INCIDENCES
     8. 1 1 2 5
     9. 6 1 5 ; 7 2 6 ; 8 2 4 ; 9 3 5 ; 10 2 5
    10. MEMBER COMPRESSION
    11. 6 TO 9
    12. MEMBER PROPERTY BRITISH
    13. 1 TO 10 TA ST UC152X152X30
    14. UNIT MMS
    15. DEFINE MATERIAL START
    16. ISOTROPIC STEEL
    17. E 210
    18. POISSON 0.3
    19. DENSITY 7.6977E-008
    20. ALPHA 6E-006
    21. DAMP 0.03
    22. TYPE STEEL
    23. STRENGTH FY 0.25 FU 0.4 RY 1.5 RT 1.2
    24. END DEFINE MATERIAL
    25. CONSTANTS
    26. MATERIAL STEEL ALL
    27. UNIT METER
    28. SUPPORT
    29. 1 6 PINNED
    30. LOAD 1
    31. JOINT LOAD
    32. 2 FX 70
    33. 3 FX 45
    34. PERFORM ANALYSIS
      STAAD PLANE                                              -- PAGE NO.    2
    * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS          6  NUMBER OF MEMBERS      10
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      2
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =      14
   TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
*** LOAD CASE     1 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    1
    35. CHANGE
    36. MEMBER COMPRESSION
    37. 6 TO 9
    38. LOAD 2
    39. JOINT LOAD
    40. 4 FX -45
    41. 5 FX -70
    42. PERFORM ANALYSIS
*** LOAD CASE     2 -- START ITERATION NO.   2
  **NOTE-Tension/Compression converged after  2 iterations, Case=    2
    43. CHANGE
```

```
   44. LOAD 3
   45. REPEAT LOAD
   46. 1 1.0 2 1.0
   47. PERFORM ANALYSIS
 **NOTE-Tension/Compression converged after  1 iterations, Case=    3
   48. CHANGE
   49. LOAD LIST ALL
   50. PRINT ANALYSIS RESULTS
     STAAD PLANE                                             -- PAGE NO.    3
   * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
 ANALYSIS RESULTS
     STAAD PLANE                                             -- PAGE NO.    4
   * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = PLANE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     1    1     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0005
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0007
          3     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0001
     2    1     0.1609    0.0464    0.0000    0.0000    0.0000   -0.0003
          2    -0.1900   -0.0132    0.0000    0.0000    0.0000    0.0003
          3     0.0138    0.0030    0.0000    0.0000    0.0000   -0.0000
     3    1     0.2889    0.0594    0.0000    0.0000    0.0000   -0.0003
          2    -0.2889   -0.0132    0.0000    0.0000    0.0000    0.0002
          3     0.0091    0.0080    0.0000    0.0000    0.0000    0.0000
     4    1     0.2889   -0.0132    0.0000    0.0000    0.0000   -0.0002
          2    -0.2889    0.0594    0.0000    0.0000    0.0000    0.0003
          3    -0.0091    0.0080    0.0000    0.0000    0.0000   -0.0000
     5    1     0.1900   -0.0132    0.0000    0.0000    0.0000   -0.0003
          2    -0.1609    0.0464    0.0000    0.0000    0.0000    0.0003
          3    -0.0138    0.0030    0.0000    0.0000    0.0000    0.0000
     6    1     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0007
          2     0.0000    0.0000    0.0000    0.0000    0.0000    0.0005
          3     0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
     STAAD PLANE                                             -- PAGE NO.    5
   * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
  SUPPORT REACTIONS -UNIT KNS  METE    STRUCTURE TYPE = PLANE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
     1    1     -0.13   -106.67      0.00      0.00      0.00      0.00
          2    114.87    106.67      0.00      0.00      0.00      0.00
          3     10.38      0.00      0.00      0.00      0.00      0.00
     6    1   -114.87    106.67      0.00      0.00      0.00      0.00
          2      0.13   -106.67      0.00      0.00      0.00      0.00
          3    -10.38      0.00      0.00      0.00      0.00      0.00
     STAAD PLANE                                             -- PAGE NO.    6
   * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
  MEMBER END FORCES    STRUCTURE TYPE = PLANE
  -----------------
  ALL UNITS ARE -- KNS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
     1    1     1   -106.67      0.13     0.00      0.00      0.00      -0.00
                2    106.67     -0.13     0.00      0.00      0.00       0.45
          2     1     30.22     -0.21     0.00      0.00      0.00       0.00
                2    -30.22      0.21     0.00      0.00      0.00      -0.73
          3     1     -6.94      0.03     0.00      0.00      0.00      -0.00
                2      6.94     -0.03     0.00      0.00      0.00       0.09
     2    1     2    -29.91      0.24     0.00      0.00      0.00       0.38
```

```
                 3     29.91      -0.24      0.00      0.00      0.00       0.46
            2    2      0.15      -0.12      0.00      0.00      0.00      -0.08
                 3     -0.15       0.12      0.00      0.00      0.00      -0.33
            3    2    -11.49      -0.03      0.00      0.00      0.00      -0.08
                 3     11.49       0.03      0.00      0.00      0.00      -0.03
       3    1    3      0.12      -0.15      0.00      0.00      0.00      -0.46
                 4     -0.12       0.15      0.00      0.00      0.00      -0.33
            2    3      0.12       0.15      0.00      0.00      0.00       0.33
                 4     -0.12      -0.15      0.00      0.00      0.00       0.46
            3    3     27.79      -0.00      0.00      0.00      0.00       0.03
                 4    -27.79       0.00      0.00      0.00      0.00      -0.03
       4    1    4      0.15       0.12      0.00      0.00      0.00       0.33
                 5     -0.15      -0.12      0.00      0.00      0.00       0.08
            2    4    -29.91      -0.24      0.00      0.00      0.00      -0.46
                 5     29.91       0.24      0.00      0.00      0.00      -0.38
            3    4    -11.49       0.03      0.00      0.00      0.00       0.03
                 5     11.49      -0.03      0.00      0.00      0.00       0.08
       5    1    5     30.22       0.21      0.00      0.00      0.00       0.73
                 6    -30.22      -0.21      0.00      0.00      0.00       0.00
            2    5   -106.67      -0.13      0.00      0.00      0.00      -0.45
                 6    106.67       0.13      0.00      0.00      0.00       0.00
            3    5     -6.94      -0.03      0.00      0.00      0.00      -0.09
                 6      6.94       0.03      0.00      0.00      0.00       0.00
       6    1    1      0.00       0.00      0.00      0.00      0.00       0.00
                 5      0.00       0.00      0.00      0.00      0.00       0.00
            2    1    137.81       0.00      0.00      0.00      0.00       0.00
                 5   -137.81       0.00      0.00      0.00      0.00       0.00
            3    1     12.51       0.00      0.00      0.00      0.00       0.00
                 5    -12.51       0.00      0.00      0.00      0.00       0.00
       7    1    2    137.81       0.00      0.00      0.00      0.00       0.00
                 6   -137.81       0.00      0.00      0.00      0.00       0.00
       STAAD PLANE                                             -- PAGE NO.    7
     * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
    MEMBER END FORCES    STRUCTURE TYPE = PLANE
    -----------------
    ALL UNITS ARE -- KNS  METE     (LOCAL )
   MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
            2    2      0.00       0.00      0.00      0.00      0.00       0.00
                 6      0.00       0.00      0.00      0.00      0.00       0.00
            3    2     12.51       0.00      0.00      0.00      0.00       0.00
                 6    -12.51       0.00      0.00      0.00      0.00       0.00
       8    1    2      0.00       0.00      0.00      0.00      0.00       0.00
                 4      0.00       0.00      0.00      0.00      0.00       0.00
            2    2     53.66       0.00      0.00      0.00      0.00       0.00
                 4    -53.66       0.00      0.00      0.00      0.00       0.00
            3    2     20.72       0.00      0.00      0.00      0.00       0.00
                 4    -20.72       0.00      0.00      0.00      0.00       0.00
       9    1    3     53.66       0.00      0.00      0.00      0.00       0.00
                 5    -53.66       0.00      0.00      0.00      0.00       0.00
            2    3      0.00       0.00      0.00      0.00      0.00       0.00
                 5      0.00       0.00      0.00      0.00      0.00       0.00
            3    3     20.72       0.00      0.00      0.00      0.00       0.00
                 5    -20.72       0.00      0.00      0.00      0.00       0.00
      10    1    2    -44.55      -0.31      0.00      0.00      0.00      -0.82
                 5     44.55       0.31      0.00      0.00      0.00      -0.81
            2    2    -44.55       0.31      0.00      0.00      0.00       0.81
                 5     44.55      -0.31      0.00      0.00      0.00       0.82
            3    2     42.29       0.00      0.00      0.00      0.00      -0.01
```

```
                5    -42.29     -0.00     0.00      0.00      0.00       0.01
   ************** END OF LATEST ANALYSIS RESULT **************
    51. FINISH
      STAAD PLANE                                              -- PAGE NO.    8
    * EXAMPLE FOR COMPRESSION-ONLY MEMBERS
             *********** END OF THE STAAD.Pro RUN ***********
               **** DATE= MAR 24,2022   TIME=  9:44:48 ****
         ************************************************************
         *   For technical assistance on STAAD.Pro, please visit    *
         *   http://www.bentley.com/en/support/                     *
         *                                                          *
         *   Details about additional assistance from               *
         *   Bentley and Partners can be found at program menu      *
         *   Help->Technical Support                                *
         *                                                          *
         *           Copyright (c) Bentley Systems, Inc.            *
         *                http://www.bentley.com                    *
         ************************************************************
```

EX. UK-26 Modeling a Rigid Diaphragm Using Control-Dependent

The structure in this example is a building consisting of member columns as well as floors made up of beam members and plate elements. Using the control-dependent command, the floors are specified to be rigid diaphragms for in-plane actions but flexible for bending actions.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-26 Modeling a Rigid Diaphragm Using Control-Dependent.STD` when you install the program.

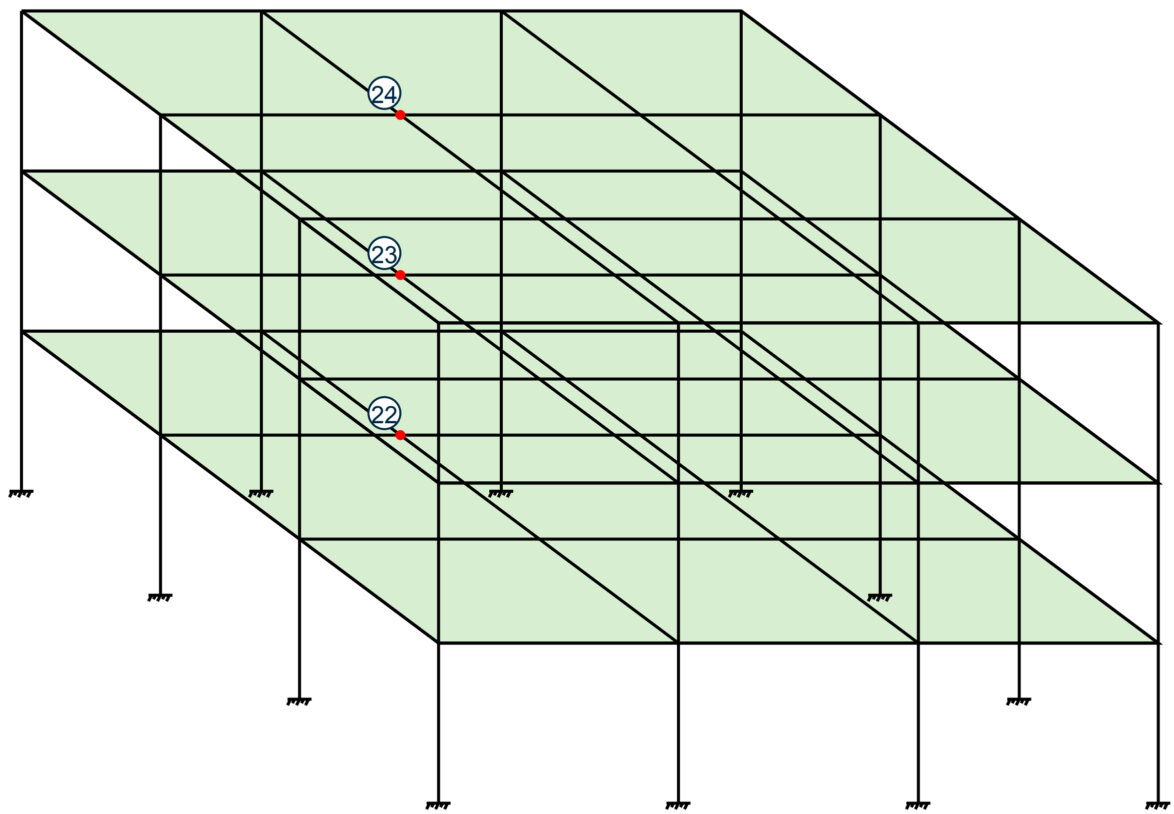


Figure 567: Example Problem No. 26

```
STAAD SPACE
*MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The second line is an optional title to identify this project.

```
UNITS KIP FT
```

Specify units for the following data.

```
JOINT COORD
1 0 0 0 4 0 48 0
REPEAT 3 24 0 0
REPEAT ALL 3 0 0 24
DELETE JOINT 21 25 37 41
```

The joint numbers and coordinates are specified above. The unwanted joints, created during the generation process used above, are then deleted.

```
MEMBER INCI
1 1 2 3 ; 4 5 6 6 ; 7 9 10 9 ; 10 13 14 12
13 17 18 15 ; 22 29 30 24 ; 25 33 34 27
34 45 46 36 ; 37 49 50 39 ; 40 53 54 42
43 57 58 45 ; 46 61 62 48 ; 49 2 6 51
52 6 10 54 ; 55 10 14 57 ; 58 18 22 60
61 22 26 63 ; 64 26 30 66 ; 67 34 38 69
```

```
70 38 42 72 ; 73 42 46 75 ; 76 50 54 78
79 54 58 81 ; 82 58 62 84 ; 85 18 2 87
88 22 6 90 ; 91 26 10 93 ; 94 30 14 96
97 34 18 99 ; 100 38 22 102 ; 103 42 26 105
106 46 30 108 ; 109 50 34 111 ; 112 54 38 114
115 58 42 117 ; 118 62 46 120
```

The `MEMBER INCIDENCE` specification is used for specifying MEMBER connectivities.

```
ELEMENT INCI
152 50 34 38 54 TO 154
155 54 38 42 58 TO 157
158 58 42 46 62 TO 160
161 34 18 22 38 TO 163
164 38 22 26 42 TO 166
167 42 26 30 46 TO 169
170 18 2 6 22 TO 172
173 22 6 10 26 TO 175
176 26 10 14 30 TO 178
```

The `ELEMENT INCIDENCE` specification is used for specifying plate element connectivities.

```
MEMBER PROPERTIES AMERICAN
1 TO 15 22 TO 27 34 TO 48 TA ST W14X90
49 TO 120 TABLE ST W27X84
```

Tip: The elements are *not* what provides the diaphragm action for this model. Rather, the control-dependent node relationships defined later in the model dictate that interaction.

All members are `WIDE FLANGE` sections whose properties are obtained from the built in American steel table.

```
ELEMENT PROP
152 TO 178 THICK 0.75
```

The thickness of the plate elements is specified above.

```
UNIT INCHES
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29000
POISSON 0.3
DENSITY 283e-006
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 36 FU 58 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
UNIT FEET
CONSTANTS
MATERIAL STEEL MEMB 1 TO 15 22 TO 27 34 TO 120
```

```
BETA 90.0 MEMB 13 14 15 22 TO 27 34 TO 39
MATERIAL CONCRETE MEMB 152 TO 178
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The orientation of some of the members is set using the `BETA` angle command.

```
SUPPORTS
1 TO 17 BY 4 29 33 45 TO 61 BY 4 FIXED
```

The supports at the above mentioned joints are declared as fixed.

```
DEPENDENT DIA ZX CONTROL 22 JOINTS YR 15.0 17.0
DEPENDENT DIA ZX CONTROL 23 JOINTS YR 31.0 33.0
DEPENDENT DIA ZX CONTROL 24 JOINTS YR 47.0 49.0
```

The three floors of the structure are specified to act as rigid diaphragms in the `ZX` plane with the corresponding control joint specified. The associated dependent joints in a floor are specified by the `YRANGE` parameter. The floors may still resist out-of-plane bending actions flexibly.

```
LOADING 1 LATERAL LOADS
JOINT LOADS
2 3 4 14 15 16 50 51 52 62 63 64 FZ 10.0
6 7 8 10 11 12 18 19 20 30 31 32 FZ 20.0
34 35 36 46 47 48 54 55 56 58 59 60 FZ 20.0
22 23 24 26 27 28 38 39 40 42 43 44 FZ 40.0
```

The above data describe a static load case. It consists of joint loads in the global Z direction.

```
LOADING 2 TORSIONAL LOADS
JOINT LOADS
2 3 4 50 51 52 FZ 5.0
14 15 16 62 63 64 FZ 15.0
6 7 8 18 19 20 FZ 10.0
10 11 12 30 31 32 FZ 30.0
34 35 36 54 55 56 FZ 10.0
46 47 48 58 59 60 FZ 30.0
22 23 24 38 39 40 FZ 20.0
26 27 28 42 43 44 FZ 60.0
```

The above data describe a static load case. It consists of joint loads that create a torsional loading on the structure.

```
LOADING 3 DEAD LOAD
ELEMENT LOAD
152 TO 178 PRESS GY -1.0
```

The above data describe a static load case. It consists of plate element pressure on a floor in the negative global Y direction.

```
PERFORM ANALYSIS
```

The above command instructs the program to proceed with the analysis.

```
PRINT JOINT DISP LIST 4 TO 60 BY 8
PRINT MEMBER FORCES LIST 116 115
PRINT SUPPORT REACTIONS LIST 9 57
```

Print displacements at selected joints, then print member forces for two members, then print support reactions at selected joints.

```
FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 10-Dec-18
END JOB INFORMATION
*MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 16 0; 3 0 32 0; 4 0 48 0; 5 24 0 0; 6 24 16 0; 7 24 32 0;
8 24 48 0; 9 48 0 0; 10 48 16 0; 11 48 32 0; 12 48 48 0; 13 72 0 0;
14 72 16 0; 15 72 32 0; 16 72 48 0; 17 0 0 24; 18 0 16 24; 19 0 32 24;
20 0 48 24; 22 24 16 24; 23 24 32 24; 24 24 48 24; 26 48 16 24;
27 48 32 24; 28 48 48 24; 29 72 0 24; 30 72 16 24; 31 72 32 24;
32 72 48 24; 33 0 0 48; 34 0 16 48; 35 0 32 48; 36 0 48 48; 38 24 16 48;
39 24 32 48; 40 24 48 48; 42 48 16 48; 43 48 32 48; 44 48 48 48;
45 72 0 48; 46 72 16 48; 47 72 32 48; 48 72 48 48; 49 0 0 72;
50 0 16 72; 51 0 32 72; 52 0 48 72; 53 24 0 72; 54 24 16 72;
55 24 32 72; 56 24 48 72; 57 48 0 72; 58 48 16 72; 59 48 32 72;
60 48 48 72; 61 72 0 72; 62 72 16 72; 63 72 32 72; 64 72 48 72;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 9 10; 8 10 11; 9 11 12;
10 13 14; 11 14 15; 12 15 16; 13 17 18; 14 18 19; 15 19 20; 22 29 30;
23 30 31; 24 31 32; 25 33 34; 26 34 35; 27 35 36; 34 45 46; 35 46 47;
36 47 48; 37 49 50; 38 50 51; 39 51 52; 40 53 54; 41 54 55; 42 55 56;
43 57 58; 44 58 59; 45 59 60; 46 61 62; 47 62 63; 48 63 64; 49 2 6;
50 3 7; 51 4 8; 52 6 10; 53 7 11; 54 8 12; 55 10 14; 56 11 15; 57 12 16;
58 18 22; 59 19 23; 60 20 24; 61 22 26; 62 23 27; 63 24 28; 64 26 30;
65 27 31; 66 28 32; 67 34 38; 68 35 39; 69 36 40; 70 38 42; 71 39 43;
72 40 44; 73 42 46; 74 43 47; 75 44 48; 76 50 54; 77 51 55; 78 52 56;
79 54 58; 80 55 59; 81 56 60; 82 58 62; 83 59 63; 84 60 64; 85 18 2;
86 19 3; 87 20 4; 88 22 6; 89 23 7; 90 24 8; 91 26 10; 92 27 11;
93 28 12; 94 30 14; 95 31 15; 96 32 16; 97 34 18; 98 35 19; 99 36 20;
100 38 22; 101 39 23; 102 40 24; 103 42 26; 104 43 27; 105 44 28;
106 46 30; 107 47 31; 108 48 32; 109 50 34; 110 51 35; 111 52 36;
112 54 38; 113 55 39; 114 56 40; 115 58 42; 116 59 43; 117 60 44;
118 62 46; 119 63 47; 120 64 48;
ELEMENT INCIDENCES SHELL
152 50 34 38 54; 153 51 35 39 55; 154 52 36 40 56; 155 54 38 42 58;
156 55 39 43 59; 157 56 40 44 60; 158 58 42 46 62; 159 59 43 47 63;
160 60 44 48 64; 161 34 18 22 38; 162 35 19 23 39; 163 36 20 24 40;
164 38 22 26 42; 165 39 23 27 43; 166 40 24 28 44; 167 42 26 30 46;
168 43 27 31 47; 169 44 28 32 48; 170 18 2 6 22; 171 19 3 7 23;
172 20 4 8 24; 173 22 6 10 26; 174 23 7 11 27; 175 24 8 12 28;
176 26 10 14 30; 177 27 11 15 31; 178 28 12 16 32;
MEMBER PROPERTY AMERICAN
1 TO 15 22 TO 27 34 TO 48 TABLE ST W14X90
49 TO 120 TABLE ST W27X84
ELEMENT PROPERTY
152 TO 178 THICKNESS 0.75
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
```

```
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC CONCRETE
E 453600
POISSON 0.17
DENSITY 0.14999
ALPHA 5.5e-06
DAMP 0.05
TYPE CONCRETE
STRENGTH FCU 576
END DEFINE MATERIAL
CONSTANTS
BETA 90 MEMB 13 TO 15 22 TO 27 34 TO 39
MATERIAL STEEL_50_KSI MEMB 1 TO 15 22 TO 27 34 TO 120
MATERIAL CONCRETE MEMB 152 TO 178
SUPPORTS
1 5 9 13 17 29 33 45 49 53 57 61 FIXED
DEPENDENT ZX CONTROL 22 JOINT YR 15 17
DEPENDENT ZX CONTROL 23 JOINT YR 31 33
DEPENDENT ZX CONTROL 24 JOINT YR 47 49
LOAD 1 LATERAL LOADS
JOINT LOAD
2 TO 4 14 TO 16 50 TO 52 62 TO 64 FZ 10
6 TO 8 10 TO 12 18 TO 20 30 TO 32 FZ 20
34 TO 36 46 TO 48 54 TO 56 58 TO 60 FZ 20
22 TO 24 26 TO 28 38 TO 40 42 TO 44 FZ 40
LOAD 2 TORSIONAL LOADS
JOINT LOAD
2 TO 4 50 TO 52 FZ 5
14 TO 16 62 TO 64 FZ 15
6 TO 8 18 TO 20 FZ 10
10 TO 12 30 TO 32 FZ 30
34 TO 36 54 TO 56 FZ 10
46 TO 48 58 TO 60 FZ 30
22 TO 24 38 TO 40 FZ 20
26 TO 28 42 TO 44 FZ 60
LOAD 3 DEAD LOAD
ELEMENT LOAD
152 TO 178 PR GY -1
PERFORM ANALYSIS
PRINT JOINT DISPLACEMENTS LIST 4 12 20 28 36 44 52 60
PRINT MEMBER FORCES LIST 115 116
PRINT SUPPORT REACTION LIST 9 57
FINISH
```

STAAD Output File

```
                                                                 PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.08.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    OCT 22, 2021                  *
             *           Time=    17:16: 5                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
```

```
               ****************************************************
     1. STAAD SPACE
INPUT FILE: UK-26 Modeling a Rigid Diaphragm Using Control-Dependent.STD
     2. START JOB INFORMATION
     3. ENGINEER DATE 10-DEC-18
     4. END JOB INFORMATION
     5. *MODELING RIGID DIAPHRAGMS USING CONTROL DEPENDENT
     6. UNIT FEET KIP
     7. JOINT COORDINATES
     8. 1 0 0 0; 2 0 16 0; 3 0 32 0; 4 0 48 0; 5 24 0 0; 6 24 16 0; 7 24 32 0
     9. 8 24 48 0; 9 48 0 0; 10 48 16 0; 11 48 32 0; 12 48 48 0; 13 72 0 0
    10. 14 72 16 0; 15 72 32 0; 16 72 48 0; 17 0 0 24; 18 0 16 24; 19 0 32 24
    11. 20 0 48 24; 22 24 16 24; 23 24 32 24; 24 24 48 24; 26 48 16 24
    12. 27 48 32 24; 28 48 48 24; 29 72 0 24; 30 72 16 24; 31 72 32 24
    13. 32 72 48 24; 33 0 0 48; 34 0 16 48; 35 0 32 48; 36 0 48 48; 38 24 16 48
    14. 39 24 32 48; 40 24 48 48; 42 48 16 48; 43 48 32 48; 44 48 48 48
    15. 45 72 0 48; 46 72 16 48; 47 72 32 48; 48 72 48 48; 49 0 0 72
    16. 50 0 16 72; 51 0 32 72; 52 0 48 72; 53 24 0 72; 54 24 16 72
    17. 55 24 32 72; 56 24 48 72; 57 48 0 72; 58 48 16 72; 59 48 32 72
    18. 60 48 48 72; 61 72 0 72; 62 72 16 72; 63 72 32 72; 64 72 48 72
    19. MEMBER INCIDENCES
    20. 1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 9 10; 8 10 11; 9 11 12
    21. 10 13 14; 11 14 15; 12 15 16; 13 17 18; 14 18 19; 15 19 20; 22 29 30
    22. 23 30 31; 24 31 32; 25 33 34; 26 34 35; 27 35 36; 34 45 46; 35 46 47
    23. 36 47 48; 37 49 50; 38 50 51; 39 51 52; 40 53 54; 41 54 55; 42 55 56
    24. 43 57 58; 44 58 59; 45 59 60; 46 61 62; 47 62 63; 48 63 64; 49 2 6
    25. 50 3 7; 51 4 8; 52 6 10; 53 7 11; 54 8 12; 55 10 14; 56 11 15; 57 12 16
    26. 58 18 22; 59 19 23; 60 20 24; 61 22 26; 62 23 27; 63 24 28; 64 26 30
    27. 65 27 31; 66 28 32; 67 34 38; 68 35 39; 69 36 40; 70 38 42; 71 39 43
    28. 72 40 44; 73 42 46; 74 43 47; 75 44 48; 76 50 54; 77 51 55; 78 52 56
    29. 79 54 58; 80 55 59; 81 56 60; 82 58 62; 83 59 63; 84 60 64; 85 18 2
    30. 86 19 3; 87 20 4; 88 22 6; 89 23 7; 90 24 8; 91 26 10; 92 27 11
    31. 93 28 12; 94 30 14; 95 31 15; 96 32 16; 97 34 18; 98 35 19; 99 36 20
    32. 100 38 22; 101 39 23; 102 40 24; 103 42 26; 104 43 27; 105 44 28
    33. 106 46 30; 107 47 31; 108 48 32; 109 50 34; 110 51 35; 111 52 36
    34. 112 54 38; 113 55 39; 114 56 40; 115 58 42; 116 59 43; 117 60 44
    35. 118 62 46; 119 63 47; 120 64 48
    36. ELEMENT INCIDENCES SHELL
    37. 152 50 34 38 54; 153 51 35 39 55; 154 52 36 40 56; 155 54 38 42 58
    38. 156 55 39 43 59; 157 56 40 44 60; 158 58 42 46 62; 159 59 43 47 63
      STAAD SPACE                                              -- PAGE NO.    2
    39. 160 60 44 48 64; 161 34 18 22 38; 162 35 19 23 39; 163 36 20 24 40
    40. 164 38 22 26 42; 165 39 23 27 43; 166 40 24 28 44; 167 42 26 30 46
    41. 168 43 27 31 47; 169 44 28 32 48; 170 18 2 6 22; 171 19 3 7 23
    42. 172 20 4 8 24; 173 22 6 10 26; 174 23 7 11 27; 175 24 8 12 28
    43. 176 26 10 14 30; 177 27 11 15 31; 178 28 12 16 32
    44. MEMBER PROPERTY AMERICAN
    45. 1 TO 15 22 TO 27 34 TO 48 TABLE ST W14X90
    46. 49 TO 120 TABLE ST W27X84
    47. ELEMENT PROPERTY
    48. 152 TO 178 THICKNESS 0.75
    49. DEFINE MATERIAL START
    50. ISOTROPIC STEEL_50_KSI
    51. E 4.176E+06
    52. POISSON 0.3
    53. DENSITY 0.489024
    54. ALPHA 6.5E-06
    55. DAMP 0.03
```

```
   56. TYPE STEEL
   57. STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
   58. ISOTROPIC CONCRETE
   59. E 453600
   60. POISSON 0.17
   61. DENSITY 0.14999
   62. ALPHA 5.5E-06
   63. DAMP 0.05
   64. TYPE CONCRETE
   65. STRENGTH FCU 576
   66. END DEFINE MATERIAL
   67. CONSTANTS
   68. BETA 90 MEMB 13 TO 15 22 TO 27 34 TO 39
   69. MATERIAL STEEL_50_KSI MEMB 1 TO 15 22 TO 27 34 TO 120
   70. MATERIAL CONCRETE MEMB 152 TO 178
   71. SUPPORTS
   72. 1 5 9 13 17 29 33 45 49 53 57 61 FIXED
   73. DEPENDENT ZX CONTROL 22 JOINT YR 15 17
   74. DEPENDENT ZX CONTROL 23 JOINT YR 31 33
   75. DEPENDENT ZX CONTROL 24 JOINT YR 47 49
   76. LOAD 1 LATERAL LOADS
   77. JOINT LOAD
   78. 2 TO 4 14 TO 16 50 TO 52 62 TO 64 FZ 10
   79. 6 TO 8 10 TO 12 18 TO 20 30 TO 32 FZ 20
   80. 34 TO 36 46 TO 48 54 TO 56 58 TO 60 FZ 20
   81. 22 TO 24 26 TO 28 38 TO 40 42 TO 44 FZ 40
   82. LOAD 2 TORSIONAL LOADS
   83. JOINT LOAD
   84. 2 TO 4 50 TO 52 FZ 5
   85. 14 TO 16 62 TO 64 FZ 15
   86. 6 TO 8 18 TO 20 FZ 10
   87. 10 TO 12 30 TO 32 FZ 30
   88. 34 TO 36 54 TO 56 FZ 10
   89. 46 TO 48 58 TO 60 FZ 30
   90. 22 TO 24 38 TO 40 FZ 20
   91. 26 TO 28 42 TO 44 FZ 60
   92. LOAD 3 DEAD LOAD
   93. ELEMENT LOAD
   94. 152 TO 178 PR GY -1.
     STAAD SPACE                                              -- PAGE NO.    3
   95. PERFORM ANALYSIS
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS         60  NUMBER OF MEMBERS     108
    NUMBER OF PLATES         27  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     12
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =     153
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
   96. PRINT JOINT DISPLACEMENTS LIST 4 12 20 28 36 44 52 60
 JOINT    DISPLACE LIST     4
     STAAD SPACE                                              -- PAGE NO.    4
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
     4    1    0.23216   0.04609   8.13343   0.00108  -0.00056  -0.00008
          2    1.49692   0.04919   6.87507   0.00090  -0.00363  -0.00046
```

```
           3    0.02679  -0.19715  -0.32918   0.00792  -0.00041  -0.00625
     12    1    0.23216   0.02166   8.45820   0.00159  -0.00056   0.00014
           2    1.49692   0.02717   8.96788   0.00166  -0.00363  -0.00000
           3    0.02679  -0.86714  -0.09026   0.07454  -0.00041   0.00495
     20    1    0.06978  -0.00054   8.13343   0.00120  -0.00056  -0.00025
           2    0.45052   0.00140   6.87507   0.00103  -0.00363  -0.00031
           3   -0.09267  -0.88243  -0.32918   0.00452  -0.00041  -0.07454
     28    1    0.06978  -0.07793   8.45820  -0.00058  -0.00056   0.00024
           2    0.45052  -0.07823   8.96788  -0.00059  -0.00363   0.00028
           3   -0.09267 -21.50271  -0.09026   0.04716  -0.00041   0.04703
     36    1   -0.09260   0.02065   8.13343   0.00102  -0.00056   0.00030
           2   -0.59589   0.01536   6.87507   0.00088  -0.00363   0.00036
           3   -0.21213  -0.86781  -0.32918  -0.00503  -0.00041  -0.07452
     44    1   -0.09260   0.08469   8.45820  -0.00057  -0.00056  -0.00028
           2   -0.59589   0.08129   8.96788  -0.00059  -0.00363  -0.00031
           3   -0.21213 -21.51369  -0.09026  -0.04712  -0.00041   0.04704
     52    1   -0.25499  -0.06556   8.13343   0.00245  -0.00056  -0.00002
           2   -1.64229  -0.06312   6.87507   0.00207  -0.00363   0.00017
           3   -0.33159  -0.19363  -0.32918  -0.00649  -0.00041  -0.00791
     60    1   -0.25499  -0.02115   8.45820   0.00162  -0.00056  -0.00014
           2   -1.64229  -0.02678   8.96788   0.00167  -0.00363   0.00001
           3   -0.33159  -0.86677  -0.09026  -0.07468  -0.00041   0.00504
 ************** END OF LATEST ANALYSIS RESULT **************
  97. PRINT MEMBER FORCES LIST 115 116
MEMBER   FORCES   LIST     115
    STAAD SPACE                                              -- PAGE NO.    5
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KIP  FEET     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
  115    1    58      0.00      7.70     0.00     -0.01      0.00     322.16
              42      0.00     -7.70     0.00      0.01      0.00    -137.43
         2    58      0.00      8.32     0.00     -0.01     -0.00     336.90
              42      0.00     -8.32     0.00      0.01      0.00    -137.14
         3    58      0.00    125.39    -0.00      0.34      0.00    1173.78
              42      0.00   -125.39     0.00     -0.34      0.00    1835.57
  116    1    59      0.00      4.73     0.00     -0.00      0.00     198.85
              43      0.00     -4.73     0.00      0.00      0.00     -85.39
         2    59      0.00      5.10     0.00     -0.01     -0.00     208.24
              43      0.00     -5.10    -0.00      0.01     -0.00     -85.84
         3    59      0.00    129.34     0.00      0.32      0.00    1407.23
              43      0.00   -129.34    -0.00     -0.32      0.00    1696.94
 ************** END OF LATEST ANALYSIS RESULT **************
  98. PRINT SUPPORT REACTION LIST 9 57
SUPPORT  REACTION LIST     9
    STAAD SPACE                                              -- PAGE NO.    6
 SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
 -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    9    1      -6.98    -54.60    -54.87   -470.16      0.01     50.68
         2     -28.54    -69.18    -58.55   -500.67      0.03    231.30
         3     -14.10   1732.38     92.25    487.24      0.00     70.67
   57    1       7.65     53.36    -54.77   -469.60      0.01    -55.89
         2      31.74     68.15    -58.52   -500.51      0.03   -257.50
         3     -11.82   1731.54    -91.91   -483.94      0.00     51.07
 ************** END OF LATEST ANALYSIS RESULT **************
  99. FINISH
    STAAD SPACE                                              -- PAGE NO.    7
```

```
           *********** END OF THE STAADD.Pro RUN ***********
             **** DATE= OCT 22,2021   TIME= 17:16: 6 ****
       ************************************************************
       *   For technical assistance on STAAD.Pro, please visit    *
       *   http://www.bentley.com/en/support/                     *
       *                                                          *
       *   Details about additional assistance from               *
       *   Bentley and Partners can be found at program menu      *
       *   Help->Technical Support                                *
       *                                                          *
       *           Copyright (c) Bentley Systems, Inc.            *
       *                http://www.bentley.com                    *
       ************************************************************
```

Related Links

- *M. To display control nodes* (on page 652)

EX. UK-27 Modeling Soil Springs for a Slab on Grade

This example illustrates the usage of commands necessary to apply the compression only attribute to spring supports for a slab on grade. The spring supports themselves are generated utilizing the built-in support generation facility. The slab is subjected to pressure and overturning loading. A tension/compression only analysis of the structure is performed.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-27 Modeling Soil Springs for a Slab on Grade.STD` when you install the program.

The numbers shown in the diagram below are the element numbers.

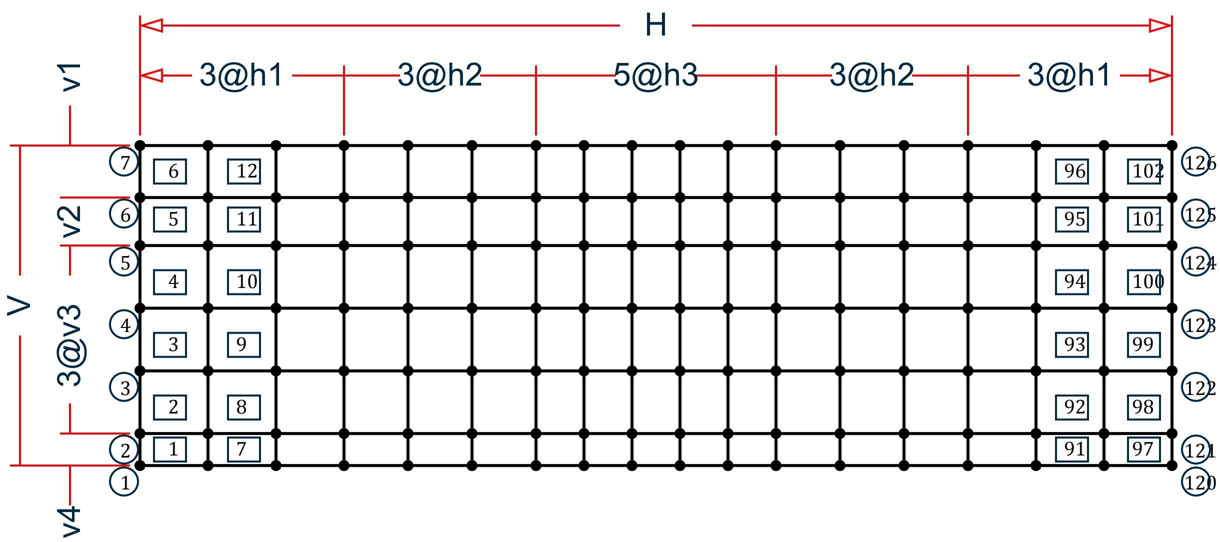


Figure 568: Example Problem No. 27

where:

H = 129', h1 = 8'-6", h2 = 8', h3 = 6'

V = 40', v1 = 6'-6", v2 = 6', v3 = 7'-10", v4 = 4'

```
STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y, and Z axes. An optional title to identify this project is provided in the second line.

```
SET NL 3
```

This structure has to be analyzed for 3 primary load cases. Consequently, the modeling of our problem requires us to define 3 sets of data, with each set containing a load case and an associated analysis command. Also, the supports which get switched off in the analysis for any load case have to be restored for the analysis for the subsequent load case. To accommodate these requirements, it is necessary to have 2 commands: SET NL and CHANGE. The SET NL command is used above to indicate the total number of primary load cases that the file contains. The CHANGE command will come in later (after the PERFORM ANALYSIS command).

```
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0 6.5
7 0.0 0.0 0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
```

For joints 1 through 7, the joint number followed by the X, Y and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these 7 joints by 8.5 feet, 3 times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and this data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first 6 elements is then used to generate the incidences of 96 more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
```

The thickness of elements 1 to 102 is specified as 8.0 inches following the command ELEMENT PROPERTIES.

```
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
```

```
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The modulus of elasticity (E) and Poisson's Ratio are specified following the command `CONSTANTS`.

```
SPRING COMPRESSION
1 TO 126 KFY
```

The above two lines declare the spring supports at nodes 1 to 126 as having the compression-only attribute. The supports themselves are being generated later (see the `ELASTIC MAT` command which appears later).

```
UNIT FEET
SUPPORTS
1 TO 126 ELASTIC MAT DIRECTION Y SUBGRADE 12.0
```

The above command is used to instruct STAAD to generate supports with compression-only springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade reaction of the soil is specified as 12 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the subgrade reaction to arrive at the spring stiffness for the "FY" degree of freedom at the joint. Units for length are changed to `FEET` to facilitate the input of subgrade reaction of soil. See TR.27.3 Automatic Spring Support Generator for Foundations (on page 2319) .

```
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
```

The above data describe a static load case. A pressure load of $1.50\ kip/ft$ acting in the negative global Y direction is applied on all the elements.

```
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
```

Tension/compression cases must each be followed by `PERFORM ANALYSIS` and `CHANGE` commands. The `CHANGE` command restores the original structure to prepare it for the analysis for the next primary load case.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2  FY -217.
8 9  FY -109.
5    FY -308.7
6    FY -617.4
22 23 FY -410.
29 30 FY -205.
26   FY -542.7
27   FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82   FY -264.2
48 55 76 83 FY -528.3
92 93      FY -205.0
99 100     FY -410.0
103        FY -487.0
104        FY -974.0
113 114    FY -109.0
120 121    FY -217.0
124        FY -273.3
FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
```

Load case 2 consists of several joint loads acting in the negative global Y direction. This is followed by another ANALYSIS command. The CHANGE command restores the original structure once again for the forthcoming load case.

```
    LOAD 3 'COLUMN OVERTURNING LOAD'
    ELEMENT LOAD
    1 TO 102 PR GY -1.50
    JOINT LOADS
    1 2  FY -100.
    8 9  FY -50.
    5    FY -150.7
    6    FY -310.4
    22 23 FY -205.
    29 30 FY -102.
    26   FY -271.7
    27    FY -542.4
    43 44 50 51 71 72 78 79 FY -153.5
    47 54 82   FY -132.2
    48 55 76 83 FY -264.3
    92 93      FY 102.0
    99 100     FY 205.0
    103        FY 243.0
    104        FY 487.0
    113 114    FY 54.0
    120 121    FY 108.0
    124        FY 136.3
    125        FY 273.6
    PERFORM ANALYSIS PRINT STATICS CHECK
```

Load case 3 consists of several joint loads acting in the upward direction at one end and downward on the other end to apply an overturning moment that will lift off one end. The CHANGE command is not needed after the last analysis.

```
    LOAD LIST 3
    PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
    PRINT ELEMENT STRESSES LIST 34 67
    PRINT SUPPORT REACTIONS LIST 5 6 12 13
```

A list of joint displacements, element stresses for elements 34 and 67, and support reactions at a list of joints for load case 3, are obtained with the help of the above commands.

```
    FINISH
```

The STAAD run is terminated.

Input File

```
STAAD SPACE SLAB ON GRADE
* SPRING COMPRESSION EXAMPLE
SET NL 3
UNIT FEET KIP
JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
```

```
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 8.0
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 4000.0
POISSON 0.12
DENSITY 8.68e-005
ALPHA 5e-006
DAMP 0.05
G 1346.15
TYPE CONCRETE
STRENGTH FCU 4
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SPRING COMPRESSION
1 TO 126 KFY
UNIT FEET
SUPPORTS
1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 12.0
LOAD 1 'WEIGHT OF MAT & EARTH'
ELEMENT LOAD
1 TO 102 PR GY -1.50
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2   FY -217.
8 9   FY -109.
5     FY -308.7
6     FY -617.4
22 23 FY -410.
29 30 FY -205.
26    FY -542.7
27    FY -1085.4
43 44 50 51 71 72 78 79 FY -307.5
47 54 82    FY -264.2
48 55 76 83 FY -528.3
92 93       FY -205.0
99 100      FY -410.0
103         FY -487.0
104         FY -974.0
113 114     FY -109.0
120 121     FY -217.0
124         FY -273.3
125         FY -546.6
PERFORM ANALYSIS PRINT STATICS CHECK
CHANGE
LOAD 3 'COLUMN OVERTURNING LOAD'
```

```
ELEMENT LOAD
1 TO 102 PR GY -1.50
JOINT LOADS
1 2   FY -100.
8 9   FY -50.
5     FY -150.7
6     FY -310.4
22 23 FY -205.
29 30 FY -102.
26    FY -271.7
27    FY -542.4
43 44 50 51 71 72 78 79 FY -153.5
47 54 82    FY -132.2
48 55 76 83 FY -264.3
92 93       FY 102.0
99 100      FY 205.0
103         FY 243.0
104         FY 487.0
113 114     FY 54.0
120 121     FY 108.0
124         FY 136.3
125         FY 273.6
PERFORM ANALYSIS PRINT STATICS CHECK
LOAD LIST 3
PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
PRINT ELEMENT STRESSES LIST 34 67
PRINT SUPPORT REACTIONS LIST 5 6 12 13
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:52                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE SLAB ON GRADE
INPUT FILE: UK-27 Modeling Soil Springs for a Slab on Grade.STD
     2. * SPRING COMPRESSION EXAMPLE
     3. SET NL 3
     4. UNIT FEET KIP
     5. JOINT COORDINATES
     6. 1 0.0 0.0 40.0
     7. 2 0.0 0.0 36.0
     8. 3 0.0 0.0 28.167
     9. 4 0.0 0.0 20.333
    10. 5 0.0 0.0 12.5
    11. 6 0.0 0.0  6.5
    12. 7 0.0 0.0  0.0
    13. REPEAT ALL 3 8.5 0.0 0.0
    14. REPEAT 3 8.0 0.0 0.0
```

```
   15. REPEAT 5 6.0 0.0 0.0
   16. REPEAT 3 8.0 0.0 0.0
   17. REPEAT 3 8.5 0.0 0.0
   18. ELEMENT INCIDENCES
   19. 1 1 8 9 2 TO 6
   20. REPEAT 16 6 7
   21. UNIT INCH
   22. ELEMENT PROPERTIES
   23. 1 TO 102 TH 8.0
   24. DEFINE MATERIAL START
   25. ISOTROPIC CONCRETE
   26. E 4000.0
   27. POISSON 0.12
   28. DENSITY 8.68E-005
   29. ALPHA 5E-006
   30. DAMP 0.05
   31. G 1346.15
   32. TYPE CONCRETE
   33. STRENGTH FCU 4
   34. END DEFINE MATERIAL
   35. CONSTANTS
   36. MATERIAL CONCRETE ALL
   37. SPRING COMPRESSION
   38. 1 TO 126 KFY
     SLAB ON GRADE                                           -- PAGE NO.    2
   * SPRING COMPRESSION EXAMPLE
   39. UNIT FEET
   40. SUPPORTS
   41. 1 TO 126  ELASTIC MAT DIRECTION Y SUBGRADE 12.0
   42. LOAD 1 'WEIGHT OF MAT & EARTH'
   43. ELEMENT LOAD
   44. 1 TO 102 PR GY -1.50
   45. PERFORM ANALYSIS PRINT STATICS CHECK
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS        126  NUMBER OF MEMBERS       0
    NUMBER OF PLATES        102  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS    126
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =     378
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
     SLAB ON GRADE                                           -- PAGE NO.    3
   * SPRING COMPRESSION EXAMPLE
  *** NOTE: CAPACITY FOR MAXIMUM #    253 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
 **NOTE-Tension/Compression converged after  1 iterations, Case=    1
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     1
         'WEIGHT OF MAT & EARTH'
          CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                       X =  0.645000019E+02
                       Y =  0.000000000E+00
                       Z =  0.200000006E+02
TOTAL APPLIED LOAD     1
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       -7740.00
      SUMMATION FORCE-Z =           0.00
```

```
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      154800.01  MY=           0.00  MZ=     -499230.03
TOTAL REACTION LOAD    1
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     1 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =       7740.00
      SUMMATION FORCE-Z =          0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     -154800.01  MY=           0.00  MZ=      499230.03
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      1)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y = -1.50000E+00       1
     Z =  0.00000E+00       0
     RX=  9.79653E-10     121
     RY=  0.00000E+00       0
     RZ=  4.32018E-10      45
  ************ END OF DATA FROM INTERNAL STORAGE ************
   46. CHANGE
   47. LOAD 2 'COLUMN LOAD-DL+LL'
   48. JOINT LOADS
   49. 1 2   FY -217.
   50. 8 9   FY -109.
   51. 5     FY -308.7
   52. 6     FY -617.4
   53. 22 23 FY -410.
     SLAB ON GRADE                                         -- PAGE NO.    4
   * SPRING COMPRESSION EXAMPLE
   54. 29 30 FY -205.
   55. 26    FY -542.7
   56. 27    FY -1085.4
   57. 43 44 50 51 71 72 78 79 FY -307.5
   58. 47 54 82    FY -264.2
   59. 48 55 76 83 FY -528.3
   60. 92 93       FY -205.0
   61. 99 100      FY -410.0
   62. 103         FY -487.0
   63. 104         FY -974.0
   64. 113 114     FY -109.0
   65. 120 121     FY -217.0
   66. 124         FY -273.3
   67. 125         FY -546.6
   68. PERFORM ANALYSIS PRINT STATICS CHECK
     SLAB ON GRADE                                         -- PAGE NO.    5
   * SPRING COMPRESSION EXAMPLE
 **NOTE-Tension/Compression converged after  1 iterations, Case=    2
         STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     2
         'COLUMN LOAD-DL+LL'
          CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                        X =  0.633725605E+02
                        Y =  0.000000000E+00
                        Z =  0.215722031E+02
TOTAL APPLIED LOAD     2
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     2 )
      SUMMATION FORCE-X =          0.00
      SUMMATION FORCE-Y =     -13964.90
      SUMMATION FORCE-Z =          0.00
```

```
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=      301253.66  MY=           0.00  MZ=     -884991.47
 TOTAL REACTION LOAD    2
   ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     2 )
       SUMMATION FORCE-X =           0.00
       SUMMATION FORCE-Y =       13964.90
       SUMMATION FORCE-Z =           0.00
      SUMMATION OF MOMENTS AROUND THE ORIGIN-
      MX=     -301253.66  MY=           0.00  MZ=      884991.47
   MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      2)
             MAXIMUMS    AT NODE
      X =  0.00000E+00       0
      Y = -1.12956E+01     120
      Z =  0.00000E+00       0
      RX=  8.11932E-02      99
      RY=  0.00000E+00       0
      RZ=  1.00569E-01       6
   ************ END OF DATA FROM INTERNAL STORAGE ************
    69. CHANGE
    70. LOAD 3 'COLUMN OVERTURNING LOAD'
    71. ELEMENT LOAD
    72. 1 TO 102 PR GY -1.50
    73. JOINT LOADS
    74. 1 2   FY -100.
    75. 8 9   FY -50.
    76. 5     FY -150.7
    77. 6     FY -310.4
      SLAB ON GRADE                                            -- PAGE NO.    6
    * SPRING COMPRESSION EXAMPLE
    78. 22 23 FY -205.
    79. 29 30 FY -102.
    80. 26    FY -271.7
    81. 27    FY -542.4
    82. 43 44 50 51 71 72 78 79 FY -153.5
    83. 47 54 82    FY -132.2
    84. 48 55 76 83 FY -264.3
    85. 92 93       FY 102.0
    86. 99 100      FY 205.0
    87. 103         FY 243.0
    88. 104         FY 487.0
    89. 113 114     FY 54.0
    90. 120 121     FY 108.0
    91. 124         FY 136.3
    92. 125         FY 273.6
    93. PERFORM ANALYSIS PRINT STATICS CHECK
      SLAB ON GRADE                                            -- PAGE NO.    7
    * SPRING COMPRESSION EXAMPLE
*** LOAD CASE     3 -- START ITERATION NO.   2
*** LOAD CASE     3 -- START ITERATION NO.   3
*** LOAD CASE     3 -- START ITERATION NO.   4
*** LOAD CASE     3 -- START ITERATION NO.   5
  **NOTE-Tension/Compression converged after  5 iterations, Case=    3
          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.     3
          'COLUMN OVERTURNING LOAD'
           CENTER OF FORCE BASED ON Y FORCES ONLY (FEET).
         (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)
                        X =  0.454460478E+02
                        Y =  0.000000000E+00
```

```
                       Z =  0.202712741E+02
TOTAL APPLIED LOAD     3
  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =      -10533.10
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=      213519.36  MY=           0.00  MZ=     -478687.78
TOTAL REACTION LOAD    3
  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     3 )
      SUMMATION FORCE-X =           0.00
      SUMMATION FORCE-Y =       10533.10
      SUMMATION FORCE-Z =           0.00
     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=     -213519.36  MY=           0.00  MZ=      478687.78
  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      3)
            MAXIMUMS    AT NODE
     X =  0.00000E+00       0
     Y =  3.06386E+01     120
     Z =  0.00000E+00       0
     RX= -1.35247E-01     120
     RY=  0.00000E+00       0
     RZ=  1.12342E-01     125
  ************ END OF DATA FROM INTERNAL STORAGE ************
   94. LOAD LIST 3
   95. PRINT JOINT DISPLACEMENTS LIST 113 114 120 121
 JOINT    DISPLACE LIST     113
     SLAB ON GRADE                                              -- PAGE NO.    8
   * SPRING COMPRESSION EXAMPLE
  JOINT DISPLACEMENT (INCH RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
   113    3    0.00000  20.81368   0.00000  -0.10277   0.00000   0.07382
   114    3    0.00000  15.82565   0.00000  -0.10134   0.00000   0.06799
   120    3    0.00000  30.63862   0.00000  -0.13525   0.00000   0.10711
   121    3    0.00000  24.13106   0.00000  -0.12907   0.00000   0.09243
  ************** END OF LATEST ANALYSIS RESULT **************
   96. PRINT ELEMENT STRESSES LIST 34 67
 ELEMENT  STRESSES LIST     34
     SLAB ON GRADE                                              -- PAGE NO.    9
   * SPRING COMPRESSION EXAMPLE
  ELEMENT STRESSES    FORCE,LENGTH UNITS= KIP  FEET
  ----------------
         STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                     VONT       VONB         SX          SY          SXY
                     TRESCAT    TRESCAB
    34      3       -4.66       -6.79        2.34        7.95        5.74
                   164.71      164.71        0.00        0.00        0.00
                   172.44      172.44
      TOP : SMAX=     155.71 SMIN=     -16.73 TMAX=      86.22 ANGLE= 58.0
      BOTT: SMAX=      16.73 SMIN=    -155.71 TMAX=      86.22 ANGLE=-32.0
    67      3       32.82        6.55      -63.47        5.72       36.48
                  1238.45     1238.45        0.00        0.00        0.00
                  1357.37     1357.37
      TOP : SMAX=     288.91 SMIN=   -1068.46 TMAX=     678.69 ANGLE= 66.7
      BOTT: SMAX=    1068.46 SMIN=    -288.91 TMAX=     678.69 ANGLE=-23.3
         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
```

```
            MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
           PRINCIPAL     PRINCIPAL       SHEAR       VONMISES       TRESCA
            STRESS        STRESS        STRESS        STRESS        STRESS
         1.068459E+03 -1.068459E+03  6.786865E+02  1.238454E+03  1.357373E+03
 PLATE NO.      67            67            67            67            67
 CASE  NO.       3             3             3             3             3
   ********************END OF ELEMENT FORCES*******************
    97. PRINT SUPPORT REACTIONS LIST 5 6 12 13
  SUPPORT  REACTION LIST     5
      SLAB ON GRADE                                           -- PAGE NO.   10
    * SPRING COMPRESSION EXAMPLE
   SUPPORT REACTIONS -UNIT KIP  FEET    STRUCTURE TYPE = SPACE
   -----------------
 JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
      5    3      0.00    149.37      0.00      0.00      0.00      0.00
      6    3      0.00    170.57      0.00      0.00      0.00      0.00
     12    3      0.00    148.42      0.00      0.00      0.00      0.00
     13    3      0.00    152.40      0.00      0.00      0.00      0.00
   ************** END OF LATEST ANALYSIS RESULT **************
    98. FINISH
      SLAB ON GRADE                                           -- PAGE NO.   11
    * SPRING COMPRESSION EXAMPLE
            *********** END OF THE STAADD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:53 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

EX. UK-28 Calculation of Modes and Frequencies of a Bridge

This example demonstrates the input required for obtaining the modes and frequencies of the skewed bridge shown in the figure below. The structure consists of piers, pier-cap girders and a deck slab.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-28 Calculation of Modes and Frequencies of a Bridge.std` when you install the program.

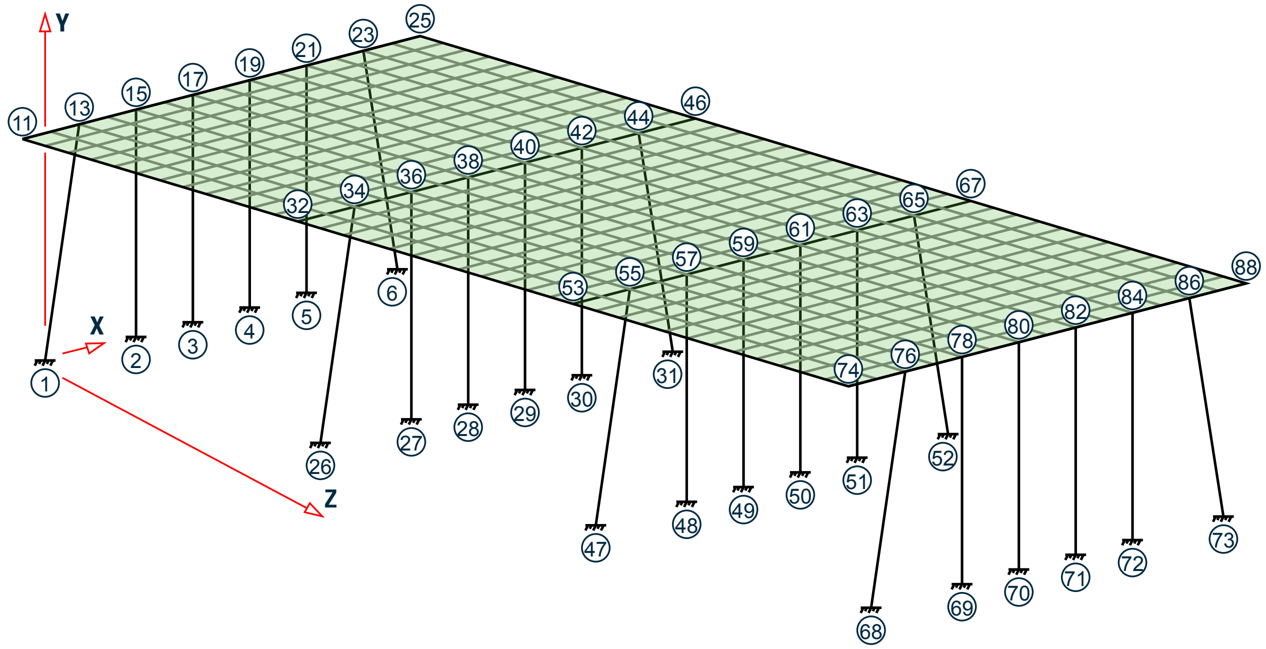


Figure 569: Example Problem No. 28

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words forms a title to identify this project.

```
IGNORE LIST
```

Further below in this file, we will call element lists in which some element numbers may not actually be present in the structure. We do so because it minimizes the effort involved in fetching the desired elements and reduces the size of the respective commands. To prevent the program from treating that condition (referring to elements which do not exist) as an error, the above command is required.

```
UNIT METER KN
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
```

For joints 1 through 6, the joint number followed by the X, Y and Z coordinates are specified first.

Next, using the coordinates of joints 11 and 25 as the basis, joints 12 through 24 are generated using linear interpolation.

Following this, using the data of these 21 joints (1 through 6 and 11 through 25), 63 new joints are generated. To achieve this, the X coordinate of these 21 joints is incremented by 4 meters and the Z coordinate is incremented by 14 meters, in 3 successive operations.

The `REPEAT ALL` command is used for the generation. Details of this command is available in TR.11 Joint Coordinates Specification (on page 2217) . The results of the generation may be visually verified using STAAD.Pro's graphical viewing facilities.

```
MEMBER INCI
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
```

The member connectivity data (joint numbers between which members are connected) is specified for the 24 columns for the structure. The above method, where the member number is followed by the 2 node numbers, is the explicit definition method. No generation is involved here.

```
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
```

The member connectivity data is specified for the pier cap beams for the structure. The above method is a combination of explicit definition and generation. For example, member 101 is defined as connected between 11 & 12. Then, by incrementing those nodes by 1 unit at a time (which is the default increment), the incidences of members 102 to 114 are generated. Similarly, we create members 202 to 215, 303 to 316, and, 404 to 417.

```
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
```

The next step is to generate the deck slab which will be modeled using plate elements. For this, we use a technique called mesh generation. Mesh generation is a process of generating several "child" elements from a "parent" or "super" element. The above set of commands defines the corner nodes of the super-element. Details of the above can be found in TR.14.2 Element Mesh Generation (on page 2229).

Note that instead of elaborately defining the coordinates of the corner nodes of the super-elements, we have taken advantage of the fact that the coordinates of these joints (A through H) have already been defined or generated earlier. Thus, A is the same as joint 11 while D is the same as joint 32. Alternatively, we could have defined the super-element nodes as A -1 10 0 ; B 16.5 10 0 ; C 20.5 10 14 ; D 3 10 14 ; etc.

```
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEGH 14 12
```

The above lines are the instructions for generating the “child” elements from the super-elements. For example, from the super-element bound by the corners A, B, C and D (which in turn are nodes 11, 25, 46 and 32), we generate a total of 14X12=168 elements, with 14 divisions along the edges AB and CD, and 12 along the edges BC and DA. These are the elements which make up the first span.

Similarly, 168 elements are created for the 2nd span, and another 168 for the 3rd span.

It may be noted here that we have taken great care to ensure that the resulting elements and the piercap beams form a perfect fit. In other words, there is no overlap between the two in a manner that nodes of the beams are at a different point in space than nodes of elements. At every node along their common boundary, plates and beams are properly connected. This is absolutely essential to ensure proper transfer of load and stiffness from beams to plates and vice versa. The tools in the user interface may be used to confirm that beam-plate connectivity is proper for this model.

```
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
```

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the piercap beams are being grouped to a name called GIRDERS, the pier columns are assigned the name PIERS, and so on. For the deck, a few selected elements are chosen into a few selective groups. The reason is that these elements happen to be right beneath wheels of vehicles whose weight will be used in the frequency calculation.

```
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
```

Member properties are assigned as prismatic rectangular sections for the girders, and prismatic circular sections for the columns.

```
ELEMENT PROPERTY
YRA 9 11 TH 0.375
```

The plate elements of the deck slab, which happen to be at a Y elevation of 10 metres (between a `YRANGE` of 9 metres and 11 metres) are assigned a thickness of 375 mms.

```
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.36158e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
```

The Modulus of elasticity (E) is set to 21000 N/sq.mm for all members. The keyword `CONSTANTS` has to precede this data. Built-in default value for Poisson's ratio for concrete is also assigned to ALL members and elements.

```
UNIT KNS METER
CONSTANTS
DENSITY 24 ALL
```

Following a change of units, density of concrete is specified.

```
SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
```

The base nodes of the piers are fully restrained (FIXED supports).

```
CUT OFF MODE SHAPE 65
```

Theoretically, a structure has as many modes of vibration as the number of degrees of freedom in the model. However, the limitations of the mathematical process used in extracting modes may limit the number of modes that can actually be extracted. In a large structure, the extraction process can also be very time consuming. Further, not all modes are of equal importance. (One measure of the importance of modes is the participation factor of that mode.) In many cases, the first few modes may be sufficient to obtain a significant portion of the total dynamic response.

Due to these reasons, in the absence of any explicit instruction, STAAD calculates only the first 6 modes. This is like saying that the command `CUT OFF MODE SHAPE 6` has been specified.

If the inspection of the first 6 modes reveals that the overall vibration pattern of the structure has not been obtained, one may ask STAAD to compute a larger (or smaller) number of modes with the help of this command. The number that follows this command is the number of modes being requested. In our example, we are asking for 65 modes by specifying `CUT OFF MODE SHAPE 65`.

```
UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700
```

The mathematical method that STAAD uses is called the eigen extraction method. Some information on this is available in G.17.3 Dynamic Analysis (on page 2154) . The method involves 2 matrices - the stiffness matrix, and the mass matrix.

The stiffness matrix, usually called the [K] matrix, is assembled using data such as member and element lengths, member and element properties, modulus of elasticity, Poisson's ratio, member and element releases, member offsets, support information, etc.

For assembling the mass matrix, called the [M] matrix, STAAD uses the load data specified in the load case in which the MODAL CAL REQ command is specified. So, some of the important aspects to bear in mind are :

1. The input you specify is weights, not masses. Internally, STAAD will convert weights to masses by dividing the input by "g", the acceleration due to gravity.
2. If the structure is declared as a PLANE frame, there are 2 possible directions of vibration - global X, and global Y. If the structure is declared as a SPACE frame, there are 3 possible directions - global X, global Y and global Z. However, this does not guarantee that STAAD will automatically consider the masses for vibration in all the available directions.

   You have control over and are responsible for specifying the directions in which the masses ought to vibrate. In other words, if a weight is not specified along a certain direction, the corresponding degrees of freedom (such as for example, global X at node 34 hypothetically) will not receive a contribution in the mass matrix. The mass matrix is assembled using only the masses from the weights and directions you specify.

   In our example, notice that we are specifying the selfweight along global X, Y and Z directions. Similarly, a 200 kg/sq.m pressure load is also specified along all 3 directions on the deck.

   But for the truck loads, we choose to apply it on just a few elements in the global Y and Z directions only. The reasoning is something like - for the X direction, the mass is not capable of vibrating because the tires allow the truck to roll along X. Remember, this is just a demonstration example, not necessarily what you may want to do.

   The point we want to illustrate is that if you want to restrict a certain weight to certain directions only, all you need to do is not provide the directions in which those weights cannot vibrate in.
3. As much as possible, provide absolute values for the weights. STAAD is programmed to algebraically add the weights at nodes. So, if some weights are specified as positive numbers and others as negative, the total weight at a given node is the algebraic summation of all the weights in the global directions at that node and the mass is then derived from this algebraic resultant.

```
MODAL CALCULATION REQUESTED
```

This is the command which tells the program that frequencies and modes should be calculated. It is specified inside a load case. In other words, this command accompanies the loads that are to be used in generating the mass matrix.

Frequencies and modes have to be calculated also when dynamic analysis such as response spectrum or time history analysis is carried out. But in such analyses, the `MODAL CALCULATION REQUESTED` command is not explicitly required. When STAAD encounters the commands for response spectrum (see example 11) and time history (see examples 16 and 22), it automatically will carry out a frequency extraction without the help of the MODAL .. command.

```
PERFORM ANALYSIS
```

This initiates the processes which are required to obtain the frequencies. Frequencies, periods and participation factors are automatically reported in the output file when the operation is completed.

```
FINISH
```

This terminates the STAAD run.

Input File

```
STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
IGNORE LIST
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0;
11 -1 10 0 25 16.5 10 0
REPEAT ALL 3 4 0 14
MEMBER INCI
```

```
1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
101 11 12 114
202 32 33 215
303 53 54 316
404 74 75 417
DEFINE MESH
A JOINT 11
B JOINT 25
C JOINT 46
D JOINT 32
E JOINT 67
F JOINT 53
G JOINT 88
H JOINT 74
GENERATE ELEMENT
MESH ABCD 14 12
MESH DCEF 14 12
MESH FEGH 14 12
START GROUP DEFINITION
MEMBER
_GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
_PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
ELEMENT
_P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
_P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
_P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
_P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
_P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
_P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
END GROUP DEFINITION
MEMBER PROPERTY
_GIRDERS PRIS YD 0.6 ZD 0.6
_PIERS PRIS YD 1.0
ELEMENT PROPERTY
YRA 9 11 TH 0.375
UNIT MMS
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 21.0
POISSON 0.17
DENSITY 2.4e-008
ALPHA 5e-006
DAMP 0.05
G 9.25
TYPE CONCRETE
STRENGTH FCU 0.0275
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE ALL
SUPPORTS
1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
CUT OFF MODE SHAPE 65
UNIT KGS METER
LOAD 1 FREQUENCY CALCULATION
SELFWEIGHT X 1.0
```

```
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
* PERMANENT WEIGHTS ON DECK
ELEMENT LOAD
YRA 9 11 PR GX 200
YRA 9 11 PR GY 200
YRA 9 11 PR GZ 200
* VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
ELEMENT LOAD
_P1 PR GY 700
_P2 PR GY 700
_P3 PR GY 700
_P4 PR GY 700
_P5 PR GY 700
_P6 PR GY 700
_P1 PR GZ 700
_P2 PR GZ 700
_P3 PR GZ 700
_P4 PR GZ 700
_P5 PR GZ 700
_P6 PR GZ 700
MODAL CALCULATION REQUESTED
PERFORM ANALYSIS
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:44:55                      *
             *                                                  *
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE
INPUT FILE: UK-28 Calculation of Modes and Frequencies of a Bridge.STD
     2. IGNORE LIST
     3. UNIT METER KN
     4. JOINT COORDINATES
     5. 1 0 0 0; 2 4 0 0; 3 6.5 0 0; 4 9 0 0; 5 11.5 0 0; 6 15.5 0 0
     6. 11 -1 10 0 25 16.5 10 0
     7. REPEAT ALL 3 4 0 14
     8. MEMBER INCI
     9. 1 1 13 ; 2 2 15 ; 3 3 17 ; 4 4 19 ; 5 5 21 ; 6 6 23
    10. 26 26 34 ; 27 27 36 ; 28 28 38 ; 29 29 40 ; 30 30 42 ; 31 31 44
    11. 47 47 55 ; 48 48 57 ; 49 49 59 ; 50 50 61 ; 51 51 63 ; 52 52 65
    12. 68 68 76 ; 69 69 78 ; 70 70 80 ; 71 71 82 ; 72 72 84 ; 73 73 86
    13. 101 11 12 114
    14. 202 32 33 215
    15. 303 53 54 316
    16. 404 74 75 417
    17. DEFINE MESH
    18. A JOINT 11
```

```
    19. B JOINT 25
    20. C JOINT 46
    21. D JOINT 32
    22. E JOINT 67
    23. F JOINT 53
    24. G JOINT 88
    25. H JOINT 74
    26. GENERATE ELEMENT
    27. MESH ABCD 14 12
    28. MESH DCEF 14 12
    29. MESH FEGH 14 12
    30. START GROUP DEFINITION
    31. MEMBER
    32. _GIRDERS 101 TO 114 202 TO 215 303 TO 316 404 TO 417
    33. _PIERS 1 TO 6 26 TO 31 47 TO 52 68 TO 73
    34. ELEMENT
    35. _P1 447 TO 450 454 TO 457 461 TO 464 468 TO 471
    36. _P2 531 TO 534 538 TO 541 545 TO 548 552 TO 555
    37. _P3 615 TO 618 622 TO 625 629 TO 632 636 TO 639
    38. _P4 713 TO 716 720 TO 723 727 TO 730 734 TO 737
      FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    2
    39. _P5 783 TO 786 790 TO 793 797 TO 800 804 TO 807
    40. _P6 881 TO 884 888 TO 891 895 TO 898 902 TO 905
    41. END GROUP DEFINITION
    42. MEMBER PROPERTY
    43. _GIRDERS PRIS YD 0.6 ZD 0.6
    44. _PIERS PRIS YD 1.0
    45. ELEMENT PROPERTY
    46. YRA 9 11 TH 0.375
    47. UNIT MMS
    48. DEFINE MATERIAL START
    49. ISOTROPIC CONCRETE
    50. E 21.0
    51. POISSON 0.17
    52. DENSITY 2.4E-008
    53. ALPHA 5E-006
    54. DAMP 0.05
    55. G 9.25
    56. TYPE CONCRETE
    57. STRENGTH FCU 0.0275
    58. END DEFINE MATERIAL
    59. CONSTANTS
    60. MATERIAL CONCRETE ALL
    61. SUPPORTS
    62. 1 TO 6 26 TO 31 47 TO 52 68 TO 73 FIXED
    63. CUT OFF MODE SHAPE 65
    64. UNIT KGS METER
    65. LOAD 1 FREQUENCY CALCULATION
    66. SELFWEIGHT X 1.0
    67. SELFWEIGHT Y 1.0
    68. SELFWEIGHT Z 1.0
    69. * PERMANENT WEIGHTS ON DECK
    70. ELEMENT LOAD
    71. YRA 9 11 PR GX 200
    72. YRA 9 11 PR GY 200
    73. YRA 9 11 PR GZ 200
    74. * VEHICLES ON SPANS - ONLY Y & Z EFFECT CONSIDERED
    75. ELEMENT LOAD
```

```
   76. _P1 PR GY 700
   77. _P2 PR GY 700
   78. _P3 PR GY 700
   79. _P4 PR GY 700
   80. _P5 PR GY 700
   81. _P6 PR GY 700
   82. _P1 PR GZ 700
   83. _P2 PR GZ 700
   84. _P3 PR GZ 700
   85. _P4 PR GZ 700
   86. _P5 PR GZ 700
   87. _P6 PR GZ 700
   88. MODAL CALCULATION REQUESTED
   89. PERFORM ANALYSIS
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    3
           P R O B L E M   S T A T I S T I C S
           -----------------------------------
    NUMBER OF JOINTS        579  NUMBER OF MEMBERS      80
    NUMBER OF PLATES        504  NUMBER OF SOLIDS        0
    NUMBER OF SURFACES        0  NUMBER OF SUPPORTS     24
          Using 64-bit analysis engine.
          SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
  TOTAL      PRIMARY LOAD CASES =     1, TOTAL DEGREES OF FREEDOM =    3330
  TOTAL LOAD COMBINATION  CASES =     0  SO FAR.
  *** NOTE: CAPACITY FOR MAXIMUM #    252 LOAD CASES IS ASSIGNED FOR PLATE LOAD.
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ** WARNING: PRESSURE LOADS ON ELEMENTS OTHER THAN PLATE ELEMENTS
              ARE IGNORED. ELEM.NO.   101
  ***NOTE: MASSES DEFINED UNDER LOAD#       1 WILL FORM
           THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
EIGEN METHOD   : SUBSPACE
-------------------------
NUMBER OF MODES REQUESTED             =    65
NUMBER OF EXISTING MASSES IN THE MODEL =  1665
NUMBER OF MODES THAT WILL BE USED      =    65
  ***  EIGENSOLUTION : ADVANCED METHOD ***
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    4
              CALCULATED FREQUENCIES FOR LOAD CASE       1
      MODE            FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
        1                       1.637                 0.61071
        2                       2.604                 0.38399
        3                       2.886                 0.34652
        4                       3.760                 0.26597
        5                       4.083                 0.24494
        6                       4.396                 0.22747
        7                       4.533                 0.22062
        8                       4.696                 0.21293
        9                       5.049                 0.19806
       10                       7.232                 0.13828
       11                       7.281                 0.13734
       12                       7.407                 0.13501
       13                      10.360                 0.09653
       14                      10.758                 0.09296
       15                      11.195                 0.08932
       16                      11.310                 0.08841
```

```
      17                       11.605                  0.08617
      18                       11.872                  0.08423
      19                       11.956                  0.08364
      20                       12.134                  0.08241
      21                       12.542                  0.07974
      22                       13.727                  0.07285
      23                       14.719                  0.06794
      24                       14.831                  0.06743
      25                       15.202                  0.06578
      26                       17.387                  0.05751
      27                       17.560                  0.05695
      28                       17.828                  0.05609
      29                       19.795                  0.05052
      30                       19.988                  0.05003
      31                       20.596                  0.04855
      32                       20.688                  0.04834
      33                       20.901                  0.04785
      34                       21.212                  0.04714
      35                       21.502                  0.04651
      36                       21.873                  0.04572
      37                       22.161                  0.04512
      38                       23.248                  0.04301
      39                       23.593                  0.04239
      40                       24.075                  0.04154
      41                       24.730                  0.04044
      42                       25.535                  0.03916
      43                       26.120                  0.03828
      44                       26.537                  0.03768
      45                       26.938                  0.03712
      46                       27.433                  0.03645
      47                       27.884                  0.03586
      48                       29.090                  0.03438
      49                       29.686                  0.03369
      50                       29.927                  0.03341
      51                       31.110                  0.03214
    FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    5
             CALCULATED FREQUENCIES FOR LOAD CASE       1
     MODE             FREQUENCY(CYCLES/SEC)         PERIOD(SEC)
      52                       31.643                  0.03160
      53                       31.824                  0.03142
      54                       32.121                  0.03113
      55                       32.701                  0.03058
      56                       33.002                  0.03030
      57                       34.257                  0.02919
      58                       35.053                  0.02853
      59                       35.310                  0.02832
      60                       35.548                  0.02813
      61                       36.095                  0.02770
      62                       36.688                  0.02726
      63                       38.682                  0.02585
      64                       38.951                  0.02567
      65                       39.661                  0.02521
          MODAL WEIGHT (MODAL MASS TIMES g) IN KGS          GENERALIZED
    MODE            X             Y             Z             WEIGHT
       1       1.313672E+02  1.108398E+00  1.201129E+06    1.221103E+06
       2       1.105162E+06  2.912622E+00  2.323269E+02    1.098762E+06
       3       2.073649E-01  2.786898E+03  2.769101E+00    5.493268E+05
       4       1.462758E+00  3.955818E+04  6.224106E+00    2.014392E+05
```

```
     5       2.818490E-01  4.828385E+02  5.992502E+02    1.365709E+05
     6       5.598604E+02  6.953900E+02  2.771716E+02    9.013609E+04
     7       1.854187E+01  3.566982E+05  3.225212E+00    7.500848E+04
     8       5.356242E+00  2.742697E+05  8.417095E+00    7.765617E+04
     9       5.904817E+03  1.694393E+03  2.261244E+03    7.013562E+04
    10       4.430377E+00  1.534484E+03  4.903639E-01    4.388408E+04
    11       4.601393E+00  6.655514E+02  1.207885E+00    4.369367E+04
    12       2.289314E-01  4.228225E+02  7.787098E-01    3.999065E+04
    13       3.993157E+01  2.747181E+00  6.896335E+03    1.557242E+05
    14       2.161567E-01  1.315486E+02  1.086174E+02    7.218063E+04
    15       2.743077E-02  4.349509E+03  1.227626E+02    4.601683E+04
    16       7.932399E+00  7.722762E+01  1.715499E+02    5.132691E+04
    17       9.357269E+00  2.437766E+00  6.022200E-01    5.278384E+04
    18       1.310402E+02  1.245065E+01  1.558885E+02    8.975814E+04
    19       3.237358E-01  3.119274E+01  4.475092E+00    4.629568E+04
    20       9.860828E+00  7.537996E+01  1.815535E+01    7.571097E+04
    21       8.020853E-01  2.958744E+00  4.741305E+00    7.486690E+04
    22       1.257488E+02  1.348115E-01  1.747276E+00    4.145462E+05
    23       6.964422E+00  4.654827E+01  8.396615E+01    6.993260E+04
    24       4.645357E-01  1.888568E+02  6.969984E+00    5.688589E+04
    25       8.396620E+00  1.087438E+01  2.082024E+02    4.957080E+04
    26       3.970981E-01  2.273349E+03  6.502642E-03    2.916350E+04
    27       8.158171E+00  3.453441E+02  9.698586E-01    3.234756E+04
   FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE                -- PAGE NO.    6
    28       2.411181E+00  1.344046E+04  2.572347E-01    3.030211E+04
    29       6.757432E+02  6.222985E+03  5.878092E+01    3.533053E+04
    30       3.712278E+01  4.299742E+04  5.310814E+00    3.319031E+04
    31       2.171634E+02  2.603085E+02  6.963874E-01    2.870320E+04
    32       2.478604E+01  4.222519E+04  2.375592E-03    3.571038E+04
    33       9.426945E+00  2.074711E+03  7.006805E-01    3.546272E+04
    34       1.318033E+01  1.391960E+02  9.997939E+00    3.683261E+04
    35       2.655567E+00  5.108468E+03  2.202262E+00    4.277587E+04
    36       3.664638E-01  4.719298E+00  1.036419E+00    5.132263E+04
    37       1.618305E+00  2.671420E+04  8.032944E-01    4.385049E+04
    38       1.289408E+00  8.348047E+01  2.136199E-02    1.948813E+04
    39       1.090681E-02  3.990261E+02  4.500325E-01    1.839953E+04
    40       2.516875E-03  2.838393E+01  1.672893E-04    4.285791E+05
    41       1.800354E+00  1.729541E+03  1.115900E+01    4.982347E+04
    42       3.094704E-01  5.689236E+04  1.057619E-01    7.183174E+04
    43       4.237560E+00  7.117003E+02  1.709904E+00    7.941099E+04
    44       6.985981E-03  7.120621E+02  2.228543E-02    7.795531E+04
    45       1.090046E-02  2.815537E+02  8.068273E-04    1.854435E+05
    46       3.867976E-01  1.895886E+03  9.105723E-03    2.331780E+04
    47       1.245474E+01  2.154164E+00  1.786947E+00    1.432475E+04
    48       6.042358E-03  4.552571E+03  7.449196E-02    2.931614E+04
    49       3.089849E+00  1.585498E+01  2.356090E-01    4.790401E+04
    50       5.917700E-01  3.501022E-02  3.813271E+00    4.319210E+04
    51       7.531865E-03  1.363633E+02  1.039786E-01    2.739737E+04
    52       3.396939E-01  2.796655E+00  9.065380E-01    4.399495E+04
    53       7.633919E-01  1.251482E+03  2.100451E-01    3.694608E+04
    54       1.113975E+01  6.123655E+01  4.429348E+00    4.883033E+04
    55       4.296741E-01  2.619401E+02  5.431140E+00    2.644050E+04
    56       1.724619E-01  8.197998E+02  1.827260E-02    3.670345E+04
    57       5.833701E-01  9.933417E+01  1.283571E+00    3.868065E+04
    58       6.173715E+00  1.663166E+03  1.165825E+01    4.293976E+04
    59       3.968332E+00  1.254915E+01  1.776702E+01    3.983809E+04
    60       7.435025E+00  5.085940E+02  5.033208E+01    5.274970E+04
    61       5.168555E+01  1.721488E+03  3.628858E+01    1.212435E+04
```

```
       62       2.675386E+01  3.225244E+03  1.680691E+01    1.478013E+04
       63       1.898740E-01  5.111631E+02  4.371451E-01    7.704926E+04
       64       1.017851E-01  3.878617E+03  1.786230E-01    6.212678E+04
       65       7.558339E+00  2.180679E+03  1.533813E+01    3.321933E+04
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE             -- PAGE NO.    7
MASS PARTICIPATION FACTORS
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
          MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
            1     0.01   0.00  99.04    0.012    0.000   99.043
            2    99.14   0.00   0.02   99.153    0.000   99.063
            3     0.00   0.23   0.00   99.154    0.230   99.063
            4     0.00   3.26   0.00   99.154    3.492   99.063
            5     0.00   0.04   0.05   99.154    3.532   99.113
            6     0.05   0.06   0.02   99.204    3.589   99.136
            7     0.00  29.41   0.00   99.206   33.002   99.136
            8     0.00  22.62   0.00   99.206   55.618   99.137
            9     0.53   0.14   0.19   99.736   55.758   99.323
           10     0.00   0.13   0.00   99.736   55.884   99.323
           11     0.00   0.05   0.00   99.737   55.939   99.323
           12     0.00   0.03   0.00   99.737   55.974   99.323
           13     0.00   0.00   0.57   99.740   55.974   99.892
           14     0.00   0.01   0.01   99.740   55.985   99.901
           15     0.00   0.36   0.01   99.740   56.344   99.911
           16     0.00   0.01   0.01   99.741   56.350   99.925
           17     0.00   0.00   0.00   99.742   56.350   99.925
           18     0.01   0.00   0.01   99.753   56.351   99.938
           19     0.00   0.00   0.00   99.754   56.354   99.938
           20     0.00   0.01   0.00   99.754   56.360   99.940
           21     0.00   0.00   0.00   99.754   56.360   99.940
           22     0.01   0.00   0.00   99.766   56.360   99.940
           23     0.00   0.00   0.01   99.766   56.364   99.947
           24     0.00   0.02   0.00   99.766   56.380   99.948
           25     0.00   0.00   0.02   99.767   56.381   99.965
           26     0.00   0.19   0.00   99.767   56.568   99.965
           27     0.00   0.03   0.00   99.768   56.597   99.965
           28     0.00   1.11   0.00   99.768   57.705   99.965
           29     0.06   0.51   0.00   99.829   58.218   99.970
           30     0.00   3.55   0.00   99.832   61.763   99.970
           31     0.02   0.02   0.00   99.852   61.785   99.971
           32     0.00   3.48   0.00   99.854   65.267   99.971
           33     0.00   0.17   0.00   99.855   65.438   99.971
           34     0.00   0.01   0.00   99.856   65.449   99.971
           35     0.00   0.42   0.00   99.856   65.871   99.972
           36     0.00   0.00   0.00   99.856   65.871   99.972
           37     0.00   2.20   0.00   99.856   68.074   99.972
           38     0.00   0.01   0.00   99.856   68.081   99.972
           39     0.00   0.03   0.00   99.856   68.114   99.972
           40     0.00   0.00   0.00   99.856   68.116   99.972
           41     0.00   0.14   0.00   99.857   68.259   99.973
           42     0.00   4.69   0.00   99.857   72.950   99.973
           43     0.00   0.06   0.00   99.857   73.008   99.973
           44     0.00   0.06   0.00   99.857   73.067   99.973
           45     0.00   0.02   0.00   99.857   73.090   99.973
           46     0.00   0.16   0.00   99.857   73.247   99.973
           47     0.00   0.00   0.00   99.858   73.247   99.973
           48     0.00   0.38   0.00   99.858   73.622   99.973
           49     0.00   0.00   0.00   99.858   73.624   99.973
```

```
           50     0.00   0.00   0.00   99.858   73.624   99.973
           51     0.00   0.01   0.00   99.858   73.635   99.973
           52     0.00   0.00   0.00   99.858   73.635   99.973
           53     0.00   0.10   0.00   99.859   73.738   99.973
           54     0.00   0.01   0.00   99.860   73.743   99.974
           55     0.00   0.02   0.00   99.860   73.765   99.974
           56     0.00   0.07   0.00   99.860   73.833   99.974
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    8
                    MASS  PARTICIPATION FACTORS IN PERCENT
                    --------------------------------------
          MODE    X     Y     Z     SUMM-X   SUMM-Y   SUMM-Z
           57     0.00   0.01   0.00   99.860   73.841   99.974
           58     0.00   0.14   0.00   99.860   73.978   99.975
           59     0.00   0.00   0.00   99.861   73.979   99.977
           60     0.00   0.04   0.00   99.861   74.021   99.981
           61     0.00   0.14   0.00   99.866   74.163   99.984
           62     0.00   0.27   0.00   99.868   74.429   99.985
           63     0.00   0.04   0.00   99.868   74.471   99.985
           64     0.00   0.32   0.00   99.868   74.791   99.985
           65     0.00   0.18   0.00   99.869   74.971   99.987
   90. FINISH
     FREQUENCIES OF VIBRATION OF A SKEWED BRIDGE              -- PAGE NO.    9
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:44:58 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

Understanding the output

After the analysis is complete, look at the output file. (This file can be viewed by selecting the **Analysis Output** tool in the **View** group on the **Utilities** ribbon tab).

i. Mode number and corresponding frequencies and periods

Since we asked for 65 modes, we obtain a report, a portion of which is as shown:

Table 913: Calculated Frequencies for Load Case 1

Mode	Frequency (Cycles/Sec)	Period (Sec)	Accuracy
1	1.636	0.61111	1.344E-16
2	2.602	0.38433	0.000E+00
3	2.882	0.34695	8.666E-16

Mode	Frequency (Cycles/Sec)	Period (Sec)	Accuracy
4	3.754	0.26636	0.000E+00
5	4.076	0.24532	3.466E-16
6	4.373	0.22870	6.025E-16
7	4.519	0.22130	5.641E-16
8	4.683	0.21355	5.253E-16
9	5.028	0.19889	0.000E+00
10	7.189	0.13911	8.916E-16
11	7.238	0.13815	0.000E+00
12	7.363	0.13582	0.000E+00

ii. Participation factors in Percentage

Table 914: Mass Participation Factors in Percent

Mode	X	Y	Z	ΣX	ΣY	ΣZ
1	0.01	0.00	99.04	0.012	0.000	99.042
2	99.14	0.00	0.02	99.151	0.000	99.061
3	0.00	0.23	0.00	99.151	0.229	99.062
4	0.00	3.27	0.00	99.151	3.496	99.062
5	0.00	0.04	0.05	99.151	3.536	99.112
6	0.05	0.04	0.02	99.202	3.575	99.135
7	0.00	26.42	0.00	99.204	30.000	99.135
8	0.00	25.59	0.00	99.204	55.587	99.136
9	0.53	0.15	0.19	99.735	55.740	99.326
10	0.00	0.13	0.00	99.736	55.871	99.326
11	0.00	0.06	0.00	99.736	55.927	99.326
12	0.00	0.04	0.00	99.736	55.969	99.326

In the explanation earlier for the `CUT OFF MODE` command, we said that one measure of the importance of a mode is the participation factor of that mode. We can see from the above report that for vibration along Z direction, the first mode has a 99.04 percent participation. It is also apparent that the 7th mode is primarily a Y direction mode with a 26.42% participation along Y and 0 in X and Z.

The ΣX, ΣY and ΣZ columns show the cumulative value of the participation of all the modes up to and including a given mode (Corresponding to the SUMM-X, SUMM-Y, and SUMM-Z reported in the output, respectively). One can infer from those terms that if one is interested in 95% participation along X, the first 2 modes are sufficient.

But for the Y direction, even with 10 modes, we barely obtained 60%. The reason for this can be understood by an examination of the nature of the structure. The deck slab is capable of vibrating in several low energy and primarily vertical direction modes. The out-of-plane flexible nature of the slab enables it to vibrate in a manner resembling a series of wave like curves. Masses on either side of the equilibrium point have opposing eigenvector values leading to a lot of cancellation of the contribution from the respective masses. Localized modes, where small pockets in the structure undergo flutter due to their relative weak stiffness compared to the rest of the model, also result in small participation factors.

iii. After the analysis is completed, select Post-processing from the mode menu. This screen contains facilities for graphically examining the shape of the mode in static and animated views. The Dynamics page on the left side of the screen is available for viewing the shape of the mode statically. The Animation option of the Results menu can be used for animating the mode. The mode number can be selected from the **Loads and Results** tab of the **Diagrams** dialog box which opens when the Animation option is chosen. The size to which the mode is drawn is controlled using the **Scales** tab of the **Diagrams** dialog box.

Related Links

- *M. To calculate the structure frequency* (on page 843)
- *TR.34.2 Modal Calculation Command* (on page 2615)

EX. UK-29 Time History Analysis of a Frame for Seismic Loads

Analysis and design of a structure for seismic loads is demonstrated in this example. In this model, static load cases are solved along with the seismic load case. For the seismic case, the maximum values of displacements, forces and reactions are obtained. The results of the dynamic case are combined with those of the static cases and steel design is performed on the combined cases.

This problem is installed with the program by default to `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\UK\UK-29 Time History Analysis of a Frame for Seismic Loads.std` when you install the program.

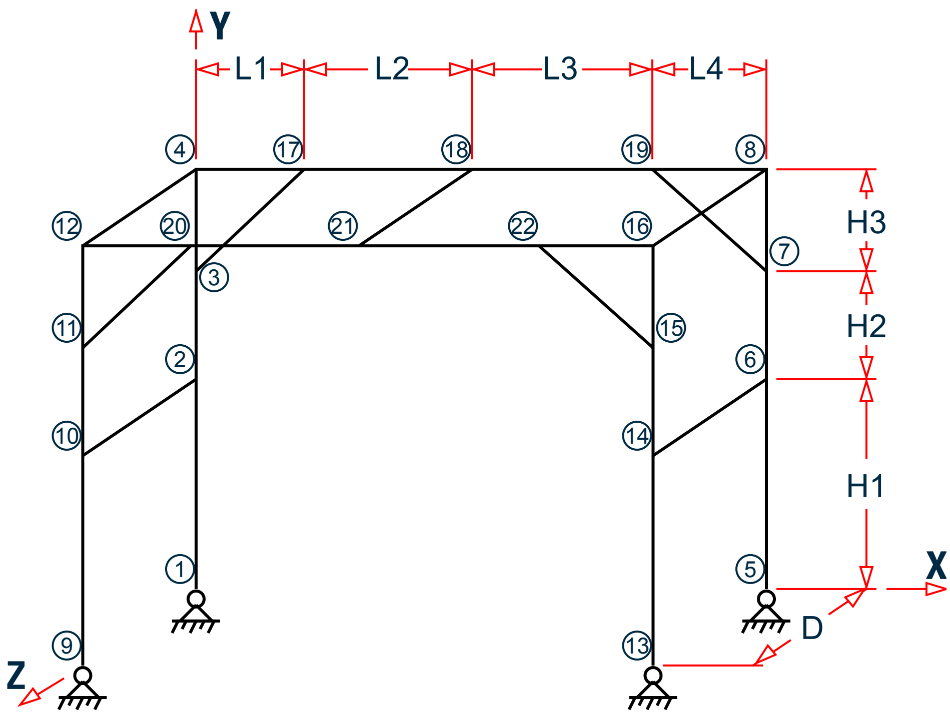


Figure 570: Example Problem No. 29

Actual input is shown in bold lettering followed by explanation.

```
STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
```

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words form a title to identify this project.

```
UNIT METER KNS
```

The units for the data that follows are specified above.

```
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
```

For joints 1 through 4, the joint number is followed by the X, Y and Z coordinates as specified above. The coordinates of these joints are used as a basis for generating 12 more joints by incrementing the X & Z coordinates by specific amounts. REPEAT ALL commands are used for the generation. Details of these commands are available in TR.11 Joint Coordinates Specification (on page 2217). Following this, another round of explicit definition (joints 17, 18 & 19) and generation (20, 21 & 22) is carried out. The results of the generation may be visually verified using graphical view features in STAAD.Pro.

```
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
```

```
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
```

A mixture of explicit definition and generation of member connectivity data (joint numbers between which members are connected) is used to generate 29 members for the structure.

```
START GROUP DEFINITION
MEMBER
_VERTICAL 1 TO 12
_XBEAM 13 TO 20
_ZBEAM 21 TO 24 29
_BRACE 25 TO 28
END GROUP DEFINITION
```

The above block of data is referred to as formation of groups. Group names are a mechanism by which a single moniker can be used to refer to a cluster of entities, such as members. For our structure, the columns are being grouped to a name called VERTICAL, the beams running along the X direction are assigned the name XBEAM, and so on.

```
MEMBER PROPERTIES CANADIAN
_VERTICAL TA ST W310X97
_XBEAM TA ST W250X39
_ZBEAM TA ST C200X17
_BRACE TA ST L152X152X13
```

Member properties are assigned from the Canadian steel table. The members which receive these properties are those embedded within the respective group names. The benefit of using the group name is apparent here. Just from the looks of the command, we can understand that the diagonal braces are being assigned a single angle. The alternative, which would be

```
25 TO 28 TA ST L152X152X13
```

would have required us to go to the graphical tools to get a sense of what members 25 to 28 are.

```
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 200
POISSON 0.3
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
```

The `DEFINE MATERIAL` command is used to specify material properties and the `CONSTANT` is used to assign the material to all members. The `BETA` angle for the channels along the left edge is set to 180 so their legs point toward the interior of the structure.

```
SUPPORTS
1 5 9 13 PINNED
```

The bottom ends of the columns of the platform are pinned supported.

```
CUT OFF MODE SHAPE 30
```

The above command is a critical command if you want to override the default number of modes computed and used in a dynamic analysis. The default, which is 6, may not always be sufficient to capture a significant portion of the structural response in a response spectrum or time history analysis, and hence the need to override the default. This command is explained in TR.30.1 Cut-Off Frequency, Mode Shapes, or Time (on page 2344).

```
UNIT METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
```

There are two stages in the command specification required for a time-history analysis. The first stage is defined above. Here, the parameters of the earthquake (ground acceleration) are provided.

Each data set is individually identified by the number that follows the TYPE command. In this file, only one data set is defined, which is apparent from the fact that only one TYPE is defined.

The word FORCE that follows the TYPE 1 command signifies that this data set is for a ground acceleration. (If you want to specify a forcing function, the keyword FORCE must be used instead.)

Notice the expression `READ EQDATA.TXT`. It means that we have chosen to specify the time vs. ground acceleration data in the file called `EQDATA.TXT`. That file must reside in the same folder as the one in which the data file for this structure resides. As explained in the small examples shown in TR.31.4 Definition of Time History Load (on page 2441) , the `EQDATA.TXT` file is a simple text file containing several pairs of time-acceleration data. A sample portion of that file is as shown below.

```
0.0000   0.006300
0.0200   0.003640
0.0400   0.000990
0.0600   0.004280
0.0800   0.007580
0.1000   0.010870
```

While it may not be apparent from the above numbers, it may also be noted that the geological data for the site the building sits on indicate that the above acceleration values are a fraction of "g", the acceleration due to gravity. Thus, for example, at 0.02 seconds, the acceleration is 0.00364 multiplied by 9.806 m/sec^2 (or 0.00364 multiplied by 32.2 ft/sec^2). Consequently, the burden of informing the program that the values need to be multiplied by "g" is upon us, and we shall do so at a later step.

The arrival time value indicates the relative value of time at which the earthquake begins to act upon the structure. We have chosen 0.0, as there is no other dynamic load on the structure from the relative time standpoint. The modal damping ratio for all the modes is set to 0.05.

```
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
```

The above data describe a static load case. The selfweight of the structure is acting in the negative global Y direction.

```
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
```

Load case 2 is also a static load case. At the Y=7.0m elevation, our structure has a floor slab. But, it is a non-structural entity which, though capable of carrying the loads acting on itself, is not meant to be an integral part of the framing system. It merely transmits the load to the beam-column grid.

There are uniform area loads on the floor (think of the load as wooden pallets supporting boxes of paper). Since the slab is not part of the structural model, how do we tell the program to transmit the imposed load from the slab to the beams without manually converting them to distributed beam loads ourselves? That is where the floor load utility comes in handy. It is a facility where we specify the load as a pressure, and the program converts the pressure to individual beam loads. Thus, the input required is very simple - load intensity in the form of pressure, and the region of the structure in terms of X, Y and Z coordinates in space, of the area over which the pressure acts.

In the process of converting the pressure to beam loads, STAAD will consider the empty space between criss-crossing beams (in plan view) to be panels, similar to the squares of a chess board. The load on each panel is then transferred to beams surrounding the panel, using a triangular or trapezoidal load distribution method.

```
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
```

Load case 3 is the dynamic load case, the one which contains the second part of the instruction set for a dynamic analysis to be performed. The data here are

a. loads which will yield the mass values which will populate the mass matrix
b. the directions of the loads, which will yield the degree of freedom numbers of the mass matrix for being populated.

Thus, the selfweight, as well as the imposed loads on the non-structural slab are to be considered as participating in the vibration along all the global directions.

```
GROUND MOTION X 1 1 9.806
```

The above command too is part of load case 3. Here we say that the seismic force, whose characteristics are defined by the TYPE 1 time history input data, acting at arrival time 1, is to be applied along the X direction. We mentioned earlier that the acceleration input data was specified as a fraction of “g”. The number 9.806 indicates the value which the acceleration data, as read from EQDATA.TXT are to be factored by before they are used.

```
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1 1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1 1.0 2 1.0 3 -1.0
```

In a time history analysis, the member forces FX thru MZ each have a value for every time step. If there are a 1000 time steps, there will be 1000 values of FX, 1000 for FY etc. for that load case. Not all of them can be used in a further calculation like a steel or concrete design. However, the maximum from among those time steps is available. If we want to do a design, one way to make sure that the structure is not under-designed is to create 2 load combination cases involving the dynamic case, a positive combination, and a negative combination.

That is what is being done above. Load combination case no. 11 consists of the sum of the static load cases (1 & 2) with the positive direction of the dynamic load case (3). Load combination case no. 12 consists of the sum of the static load cases (1 & 2) with the negative direction of the dynamic load case (3). The user has discretion on what load factors to use with these combinations. We have chosen the factors to be 1.0.

```
    PERFORM ANALYSIS
```

The above is the instruction to perform the analysis related calculations. That means, computing nodal displacements, support reactions, etc.

```
    PRINT ANALYSIS RESULTS
```

The above command is an instruction to the program to produce a report of the joint displacements, support reactions and member end forces in the output file. As mentioned earlier, for the dynamic case, these will be just the maximum values, not the ones generated for every time step. If you want to see the results for each time step, you may do so by using STAAD's Post-processing facilities.

```
    LOAD LIST 11 12
    PARAMETER
    CODE CANADIAN
    CHECK CODE ALL
```

A steel design code check is done according to the Canadian code for load cases 11 and 12.

```
    FINISH
```

Input File

```
STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
UNIT METER KNS
JOINT COORDINATES
1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
REPEAT ALL 1 9.5 0 0
REPEAT ALL 1 0 0 3
17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
REPEAT ALL 1 0 0 3
MEMBER INCIDENCES
1 1 2 3
REPEAT 1 3 4
7 9 10 9
10 13 14 12
13 4 17; 14 17 18; 15 18 19; 16 19 8
17 12 20; 18 20 21; 19 21 22; 20 22 16
21 2 10; 22 4 12; 23 6 14
24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
START GROUP DEFINITION
MEMBER
_VERTICAL 1 TO 12
_XBEAM 13 TO 20
_ZBEAM 21 TO 24 29
_BRACE 25 TO 28
END GROUP DEFINITION
MEMBER PROPERTIES CANADIAN
_VERTICAL TA ST W310X97
_XBEAM TA ST W250X39
_ZBEAM TA ST C200X17
_BRACE TA ST L152X152X13
UNIT KNS MMS
DEFINE MATERIAL START
ISOTROPIC STEEL
E 200
POISSON 0.3
```

```
DENSITY 7.8e-008
ALPHA 6e-006
DAMP 0.03
TYPE STEEL
STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL ALL
BETA 180 MEMB 21 22
SUPPORTS
1 5 9 13 PINNED
CUT OFF MODE SHAPE 30
UNIT KGS METER
DEFINE TIME HISTORY
TYPE 1 ACCELERATION
READ EQDATA.TXT
ARRIVAL TIME
0.0
DAMPING 0.05
LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
SELFWEIGHT Y -1.0
LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
FLOOR LOAD
YRA 6.9 7.1 FLOAD -500
LOAD 3 DYNAMIC LOAD
* MASSES
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
FLOOR LOAD
YRANGE 6.9 7.1 FLOAD 500 GX
YRANGE 6.9 7.1 FLOAD 500 GY
YRANGE 6.9 7.1 FLOAD 500 GZ
GROUND MOTION X 1 1 9.806
LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
1 1.0 2 1.0 3 1.0
LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
1 1.0 2 1.0 3 -1.0
PERFORM ANALYSIS
PRINT ANALYSIS RESULTS
LOAD LIST 11 12
PARAMETER
CODE CANADA
CHECK CODE ALL
FINISH
```

STAAD Output File

```
                                                                PAGE NO.    1
             ****************************************************
             *                                                  *
             *           STAAD.Pro CONNECT Edition              *
             *           Version  22.10.00.***                  *
             *           Proprietary Program of                 *
             *           Bentley Systems, Inc.                  *
             *           Date=    MAR 24, 2022                  *
             *           Time=     9:45: 0                      *
             *                                                  *
```

```
             *  Licensed to: Bentley Systems Inc                *
             ****************************************************
     1. STAAD SPACE DYNAMIC ANALYSIS FOR SEISMIC LOADS
INPUT FILE: UK-29 Time History Analysis of a Frame for Seismic Loads.STD
     2. UNIT METER KNS
     3. JOINT COORDINATES
     4. 1 0 0 0 ; 2 0 3.5 0 ; 3 0 5.3 0 ; 4 0 7 0
     5. REPEAT ALL 1 9.5 0 0
     6. REPEAT ALL 1 0 0 3
     7. 17 1.8 7 0 ; 18 4.6 7 0 ; 19 7.6 7 0
     8. REPEAT ALL 1 0 0 3
     9. MEMBER INCIDENCES
    10. 1 1 2 3
    11. REPEAT 1 3 4
    12. 7 9 10 9
    13. 10 13 14 12
    14. 13 4 17; 14 17 18; 15 18 19; 16 19 8
    15. 17 12 20; 18 20 21; 19 21 22; 20 22 16
    16. 21 2 10; 22 4 12; 23 6 14
    17. 24 8 16; 25 3 17; 26 7 19; 27 11 20; 28 15 22; 29 18 21
    18. START GROUP DEFINITION
    19. MEMBER
    20. _VERTICAL 1 TO 12
    21. _XBEAM 13 TO 20
    22. _ZBEAM 21 TO 24 29
    23. _BRACE 25 TO 28
    24. END GROUP DEFINITION
    25. MEMBER PROPERTIES CANADIAN
    26. _VERTICAL TA ST W310X97
    27. _XBEAM TA ST W250X39
    28. _ZBEAM TA ST C200X17
    29. _BRACE TA ST L152X152X13
    30. UNIT KNS MMS
    31. DEFINE MATERIAL START
    32. ISOTROPIC STEEL
    33. E 200
    34. POISSON 0.3
    35. DENSITY 7.8E-008
    36. ALPHA 6E-006
    37. DAMP 0.03
    38. TYPE STEEL
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    2
    39. STRENGTH FY 0.24821 FU 0.399894 RY 1.5 RT 1.2
    40. END DEFINE MATERIAL
    41. CONSTANTS
    42. MATERIAL STEEL ALL
    43. BETA 180 MEMB 21 22
    44. SUPPORTS
    45. 1 5 9 13 PINNED
    46. CUT OFF MODE SHAPE 30
    47. UNIT KGS METER
    48. DEFINE TIME HISTORY
    49. TYPE 1 ACCELERATION
    50. READ EQDATA.TXT
    51. ARRIVAL TIME
    52. 0.0
    53. DAMPING 0.05
    54. LOAD 1 WEIGHT OF STRUCTURE ACTING STATICALLY
```

```
    55. SELFWEIGHT Y -1.0
    56. LOAD 2 PLATFORM LEVEL LOAD ACTING STATICALLY
    57. FLOOR LOAD
    58. YRA 6.9 7.1 FLOAD -500
 **NOTE** about Floor/OneWay Loads/Weights.
   Please note that depending on the shape of the floor you may
   have to break up the FLOOR/ONEWAY LOAD into multiple commands.
   For details please refer to Technical Reference Manual
   Section 5.32.4.2 Note d and/or "5.32.4.3 Note f.
    59. LOAD 3 DYNAMIC LOAD
    60. * MASSES
    61. SELFWEIGHT X 1.0
    62. SELFWEIGHT Y 1.0
    63. SELFWEIGHT Z 1.0
    64. FLOOR LOAD
    65. YRANGE 6.9 7.1 FLOAD 500 GX
    66. YRANGE 6.9 7.1 FLOAD 500 GY
    67. YRANGE 6.9 7.1 FLOAD 500 GZ
    68. GROUND MOTION X 1 1 9.806
    69. LOAD COMBINATION 11 (STATIC + POSITIVE OF DYNAMIC)
    70. 1 1.0 2 1.0 3 1.0
    71. LOAD COMBINATION 12 (STATIC + NEGATIVE OF DYNAMIC)
    72. 1 1.0 2 1.0 3 -1.0
    73. PERFORM ANALYSIS
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    3
            P R O B L E M   S T A T I S T I C S
            -----------------------------------
     NUMBER OF JOINTS         22  NUMBER OF MEMBERS      29
     NUMBER OF PLATES          0  NUMBER OF SOLIDS        0
     NUMBER OF SURFACES        0  NUMBER OF SUPPORTS      4
           Using 64-bit analysis engine.
           SOLVER USED IS THE IN-CORE ADVANCED MATH SOLVER
   TOTAL      PRIMARY LOAD CASES =     3, TOTAL DEGREES OF FREEDOM =     120
   TOTAL LOAD COMBINATION  CASES =     2  SO FAR.
   ***NOTE: MASSES DEFINED UNDER LOAD#       3 WILL FORM
            THE FINAL MASS MATRIX FOR DYNAMIC ANALYSIS.
 EIGEN METHOD   : SUBSPACE
 -------------------------
 NUMBER OF MODES REQUESTED             =    30
 NUMBER OF EXISTING MASSES IN THE MODEL =    54
 NUMBER OF MODES THAT WILL BE USED      =    30
   ***  EIGENSOLUTION : ADVANCED METHOD ***
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    4
               CALCULATED FREQUENCIES FOR LOAD CASE       3
       MODE           FREQUENCY(CYCLES/SEC)        PERIOD(SEC)
         1                      0.693                 1.44295
         2                      1.215                 0.82296
         3                      1.365                 0.73265
         4                      1.561                 0.64059
         5                      2.077                 0.48142
         6                      3.044                 0.32846
         7                      4.217                 0.23712
         8                      4.273                 0.23404
         9                      5.538                 0.18058
        10                      5.543                 0.18039
        11                      5.728                 0.17457
        12                     12.732                 0.07854
        13                     12.741                 0.07849
```

```
       14                    15.103                 0.06621
       15                    15.160                 0.06596
       16                    16.354                 0.06115
       17                    16.361                 0.06112
       18                    45.292                 0.02208
       19                    45.315                 0.02207
       20                    48.670                 0.02055
       21                    48.698                 0.02053
       22                    51.820                 0.01930
       23                    51.966                 0.01924
       24                    54.942                 0.01820
       25                    55.877                 0.01790
       26                    55.892                 0.01789
       27                    65.315                 0.01531
       28                    65.477                 0.01527
       29                    87.675                 0.01141
       30                    87.698                 0.01140
          MODAL WEIGHT (MODAL MASS TIMES g) IN KGS          GENERALIZED
     MODE           X             Y             Z              WEIGHT
        1       8.428479E-17  3.973848E-17  1.501941E+04    8.845560E+03
        2       1.725302E+04  4.369967E-02  1.375553E-16    1.677680E+04
        3       1.921147E-11  1.030589E-14  8.419906E-01    1.266359E+04
        4       5.867172E-12  1.954666E-14  4.822830E+00    2.596549E+04
        5       3.639106E-16  1.767266E-19  2.366926E+03    6.440201E+03
        6       9.389162E-30  9.016470E-30  3.077720E-13    5.200768E+03
        7       7.448636E-29  8.510260E-28  1.081621E-12    4.804401E+03
        8       2.127649E-16  8.969083E-16  1.923308E+00    8.821168E+03
        9       6.201208E-03  9.100304E+03  1.700018E-10    6.018229E+03
       10       5.259372E-12  6.922816E-06  2.331645E-01    6.033932E+03
       11       1.221274E-16  1.806567E-11  4.085088E+01    8.734128E+03
       12       3.013838E+02  4.352953E+01  2.749213E-13    6.235822E+03
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    5
       13       9.397726E-08  1.312250E-08  8.607833E-04    6.238517E+03
       14       4.732753E-19  7.878291E-19  6.997098E+01    8.386677E+02
       15       1.000137E-15  1.586578E-14  9.354548E+01    8.424974E+02
       16       5.267608E+00  1.221315E+03  3.186588E-11    5.514818E+03
       17       2.898206E-09  6.827010E-07  5.604505E-02    5.516616E+03
       18       2.828040E-01  1.883040E+03  1.599798E-16    1.675764E+03
       19       5.315951E-12  3.899624E-08  7.850585E-06    1.675934E+03
       20       4.118067E+01  1.211323E+01  1.031537E-18    1.390516E+03
       21       3.229149E-10  9.662888E-11  1.550643E-07    1.389660E+03
       22       4.568585E-26  2.039624E-22  2.052465E-17    4.245108E+02
       23       9.871051E-26  1.307891E-22  3.766811E-17    4.220495E+02
       24       3.514037E-26  1.116731E-24  6.236122E-21    4.868892E+03
       25       8.054632E-03  3.932247E+02  3.819723E-16    5.355819E+03
       26       1.156172E-13  3.029329E-09  4.932334E-05    5.335906E+03
       27       3.125936E-20  5.889706E-16  2.044529E+00    5.603039E+02
       28       2.432179E-18  3.227255E-15  2.037832E+00    5.570734E+02
       29       7.267862E-01  1.713536E+03  7.387739E-16    2.807397E+03
       30       2.296865E-18  5.310772E-15  5.251737E-17    1.982699E+03
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    6
 MASS PARTICIPATION FACTORS
                   MASS  PARTICIPATION FACTORS IN PERCENT
                   --------------------------------------
          MODE    X      Y      Z     SUMM-X   SUMM-Y   SUMM-Z
            1     0.00   0.00  85.32    0.000    0.000   85.325
            2    98.01   0.00   0.00   98.014    0.000   85.325
            3     0.00   0.00   0.00   98.014    0.000   85.329
```

```
          4     0.00   0.00   0.03   98.014    0.000   85.357
          5     0.00   0.00  13.45   98.014    0.000   98.803
          6     0.00   0.00   0.00   98.014    0.000   98.803
          7     0.00   0.00   0.00   98.014    0.000   98.803
          8     0.00   0.00   0.01   98.014    0.000   98.814
          9     0.00  51.70   0.00   98.014   51.699   98.814
         10     0.00   0.00   0.00   98.014   51.699   98.815
         11     0.00   0.00   0.23   98.014   51.699   99.047
         12     1.71   0.25   0.00   99.726   51.946   99.047
         13     0.00   0.00   0.00   99.726   51.946   99.047
         14     0.00   0.00   0.40   99.726   51.946   99.445
         15     0.00   0.00   0.53   99.726   51.946   99.976
         16     0.03   6.94   0.00   99.756   58.884   99.976
         17     0.00   0.00   0.00   99.756   58.884   99.977
         18     0.00  10.70   0.00   99.757   69.582   99.977
         19     0.00   0.00   0.00   99.757   69.582   99.977
         20     0.23   0.07   0.00   99.991   69.650   99.977
         21     0.00   0.00   0.00   99.991   69.650   99.977
         22     0.00   0.00   0.00   99.991   69.650   99.977
         23     0.00   0.00   0.00   99.991   69.650   99.977
         24     0.00   0.00   0.00   99.991   69.650   99.977
         25     0.00   2.23   0.00   99.991   71.884   99.977
         26     0.00   0.00   0.00   99.991   71.884   99.977
         27     0.00   0.00   0.01   99.991   71.884   99.988
         28     0.00   0.00   0.01   99.991   71.884  100.000
         29     0.00   9.73   0.00   99.995   81.619  100.000
         30     0.00   0.00   0.00   99.995   81.619  100.000
 A C T U A L  MODAL  D A M P I N G  USED IN ANALYSIS
 MODE       DAMPING
    1      0.05000000
    2      0.05000000
    3      0.05000000
    4      0.05000000
    5      0.05000000
    6      0.05000000
    7      0.05000000
    8      0.05000000
    9      0.05000000
   10      0.05000000
   11      0.05000000
   12      0.05000000
   13      0.05000000
   14      0.05000000
   15      0.05000000
   16      0.05000000
   DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    7
 MODE       DAMPING
   17      0.05000000
   18      0.05000000
   19      0.05000000
   20      0.05000000
   21      0.05000000
   22      0.05000000
   23      0.05000000
   24      0.05000000
   25      0.05000000
   26      0.05000000
   27      0.05000000
```

```
     28      0.05000000
     29      0.05000000
     30      0.05000000
   TIME STEP USED IN TIME HISTORY ANALYSIS = 0.00139 SECONDS
   NUMBER OF MODES WHOSE CONTRIBUTION IS CONSIDERED =   30
   TIME DURATION OF TIME HISTORY ANALYSIS =     31.160 SECONDS
   NUMBER OF TIME STEPS IN THE SOLUTION PROCESS =    22435
    74. PRINT ANALYSIS RESULTS
  BASE SHEAR UNITS ARE -- KGS  METE
  MAXIMUM BASE SHEAR  X=  -9.498282E+03  Y=  -5.285202E+01  Z=   3.482067E-06
      AT TIMES               5.809722          2.445833           2.766667
  ANALYSIS RESULTS
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    8
   JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
   ------------------
 JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
      1    1     0.0000    0.0000    0.0000   -0.0000   -0.0000    0.0001
           2     0.0000    0.0000    0.0000    0.0000   -0.0000    0.0015
           3     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0173
          11     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0157
          12     0.0000    0.0000    0.0000    0.0000   -0.0000    0.0190
      2    1    -0.0290   -0.0012   -0.0000    0.0000   -0.0000    0.0000
           2    -0.4146   -0.0050   -0.0004   -0.0001   -0.0000    0.0004
           3     5.7319    0.0046    0.0000    0.0000   -0.0000   -0.0142
          11     5.2883   -0.0015   -0.0005   -0.0001   -0.0000   -0.0138
          12    -6.1755   -0.0108   -0.0005   -0.0001   -0.0000    0.0145
      3    1    -0.0249   -0.0016   -0.0002   -0.0000   -0.0000   -0.0001
           2    -0.3581   -0.0075   -0.0122   -0.0000   -0.0001   -0.0012
           3     7.9529    0.0070    0.0000    0.0000   -0.0000   -0.0101
          11     7.5699   -0.0021   -0.0124   -0.0000   -0.0001   -0.0113
          12    -8.3358   -0.0161   -0.0124   -0.0000   -0.0001    0.0088
      4    1    -0.0030   -0.0016    0.0000    0.0000   -0.0000   -0.0001
           2    -0.0451   -0.0076    0.0004    0.0002   -0.0000   -0.0020
           3     9.3683    0.0023    0.0000    0.0000    0.0000   -0.0083
          11     9.3201   -0.0069    0.0004    0.0002   -0.0000   -0.0104
          12    -9.4164   -0.0115    0.0004    0.0002   -0.0000    0.0062
      5    1     0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0001
           2     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0014
           3     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0174
          11     0.0000    0.0000    0.0000    0.0000    0.0000   -0.0189
          12     0.0000    0.0000    0.0000    0.0000    0.0000    0.0159
      6    1     0.0261   -0.0012   -0.0000    0.0000    0.0000   -0.0000
           2     0.3723   -0.0050   -0.0004   -0.0001    0.0000   -0.0002
           3     5.7484   -0.0046   -0.0000   -0.0000   -0.0000   -0.0142
          11     6.1468   -0.0108   -0.0005   -0.0001    0.0000   -0.0144
          12    -5.3500   -0.0015   -0.0005   -0.0001    0.0000    0.0139
      7    1     0.0206   -0.0016   -0.0002   -0.0000    0.0000    0.0001
           2     0.2940   -0.0075   -0.0122   -0.0000    0.0001    0.0013
           3     7.9667   -0.0070   -0.0000   -0.0000   -0.0000   -0.0100
          11     8.2813   -0.0161   -0.0124   -0.0000    0.0001   -0.0086
          12    -7.6521   -0.0021   -0.0124   -0.0000    0.0001    0.0114
      8    1    -0.0027   -0.0016    0.0000    0.0000    0.0000    0.0001
           2    -0.0398   -0.0077    0.0004    0.0002    0.0000    0.0021
           3     9.3696   -0.0026   -0.0000   -0.0000    0.0000   -0.0082
          11     9.3271   -0.0119    0.0004    0.0002    0.0000   -0.0060
          12    -9.4121   -0.0068    0.0004    0.0002    0.0000    0.0105
      9    1     0.0000    0.0000    0.0000    0.0000    0.0000    0.0001
           2     0.0000    0.0000    0.0000   -0.0000    0.0000    0.0015
```

```
          3     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0173
         11     0.0000    0.0000    0.0000   -0.0000    0.0000   -0.0157
         12     0.0000    0.0000    0.0000   -0.0000    0.0000    0.0190
    10    1    -0.0290   -0.0012    0.0000   -0.0000    0.0000    0.0000
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.    9
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
          2    -0.4146   -0.0050    0.0004    0.0001    0.0000    0.0004
          3     5.7319    0.0046    0.0000    0.0000   -0.0000   -0.0142
         11     5.2883   -0.0015    0.0005    0.0001    0.0000   -0.0138
         12    -6.1755   -0.0108    0.0005    0.0001    0.0000    0.0145
    11    1    -0.0249   -0.0016    0.0002    0.0000    0.0000   -0.0001
          2    -0.3581   -0.0075    0.0122    0.0000    0.0001   -0.0012
          3     7.9529    0.0070    0.0000    0.0000   -0.0000   -0.0101
         11     7.5699   -0.0021    0.0124    0.0000    0.0001   -0.0113
         12    -8.3358   -0.0161    0.0124    0.0000    0.0001    0.0088
    12    1    -0.0030   -0.0016   -0.0000   -0.0000    0.0000   -0.0001
          2    -0.0451   -0.0076   -0.0004   -0.0002    0.0000   -0.0020
          3     9.3683    0.0023    0.0000    0.0000    0.0000   -0.0083
         11     9.3201   -0.0069   -0.0004   -0.0002    0.0000   -0.0104
         12    -9.4164   -0.0115   -0.0004   -0.0002    0.0000    0.0062
    13    1     0.0000    0.0000    0.0000    0.0000   -0.0000   -0.0001
          2     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0014
          3     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0174
         11     0.0000    0.0000    0.0000   -0.0000   -0.0000   -0.0189
         12     0.0000    0.0000    0.0000   -0.0000   -0.0000    0.0159
    14    1     0.0261   -0.0012    0.0000   -0.0000   -0.0000   -0.0000
          2     0.3723   -0.0050    0.0004    0.0001   -0.0000   -0.0002
          3     5.7484   -0.0046   -0.0000   -0.0000   -0.0000   -0.0142
         11     6.1468   -0.0108    0.0005    0.0001   -0.0000   -0.0144
         12    -5.3500   -0.0015    0.0005    0.0001   -0.0000    0.0139
    15    1     0.0206   -0.0016    0.0002    0.0000   -0.0000    0.0001
          2     0.2940   -0.0075    0.0122    0.0000   -0.0001    0.0013
          3     7.9667   -0.0070   -0.0000   -0.0000   -0.0000   -0.0100
         11     8.2813   -0.0161    0.0124    0.0000   -0.0001   -0.0086
         12    -7.6521   -0.0021    0.0124    0.0000   -0.0001    0.0114
    16    1    -0.0027   -0.0016   -0.0000   -0.0000   -0.0000    0.0001
          2    -0.0398   -0.0077   -0.0004   -0.0002   -0.0000    0.0021
          3     9.3696   -0.0026   -0.0000   -0.0000    0.0000   -0.0082
         11     9.3271   -0.0119   -0.0004   -0.0002   -0.0000   -0.0060
         12    -9.4121   -0.0068   -0.0004   -0.0002   -0.0000    0.0105
    17    1    -0.0027   -0.0267    0.0000    0.0000    0.0000   -0.0002
          2    -0.0400   -0.3687    0.0008    0.0001    0.0000   -0.0025
          3     9.3513   -1.4019    0.0000   -0.0000    0.0000   -0.0033
         11     9.3086   -1.7973    0.0008    0.0001    0.0000   -0.0059
         12    -9.3941    1.0065    0.0008    0.0001    0.0000    0.0006
    18    1    -0.0029   -0.0644   -0.0000    0.0001    0.0000   -0.0000
          2    -0.0425   -0.9647   -0.0000    0.0038    0.0000   -0.0003
          3     9.3534   -0.0734    0.0000    0.0000    0.0000    0.0086
         11     9.3080   -1.1025   -0.0000    0.0039    0.0000    0.0083
         12    -9.3987   -0.9557   -0.0000    0.0039    0.0000   -0.0089
    19    1    -0.0031   -0.0296    0.0000    0.0000   -0.0000    0.0002
          2    -0.0451   -0.4115    0.0009    0.0001   -0.0000    0.0025
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   10
  JOINT DISPLACEMENT (CM   RADIANS)    STRUCTURE TYPE = SPACE
  ------------------
JOINT  LOAD   X-TRANS   Y-TRANS   Z-TRANS   X-ROTAN   Y-ROTAN   Z-ROTAN
```

```
         3     9.3514    1.4648   -0.0000    0.0000    0.0000   -0.0031
        11     9.3032    1.0237    0.0009    0.0001   -0.0000   -0.0004
        12    -9.3995   -1.9060    0.0009    0.0001   -0.0000    0.0057
   20    1    -0.0027   -0.0267   -0.0000   -0.0000   -0.0000   -0.0002
         2    -0.0400   -0.3687   -0.0008   -0.0001   -0.0000   -0.0025
         3     9.3513   -1.4019    0.0000   -0.0000    0.0000   -0.0033
        11     9.3086   -1.7973   -0.0008   -0.0001   -0.0000   -0.0059
        12    -9.3941    1.0065   -0.0008   -0.0001   -0.0000    0.0006
   21    1    -0.0029   -0.0644    0.0000   -0.0001   -0.0000   -0.0000
         2    -0.0425   -0.9647    0.0000   -0.0038   -0.0000   -0.0003
         3     9.3534   -0.0734    0.0000    0.0000    0.0000    0.0086
        11     9.3080   -1.1025    0.0000   -0.0039   -0.0000    0.0083
        12    -9.3987   -0.9557    0.0000   -0.0039   -0.0000   -0.0089
   22    1    -0.0031   -0.0296   -0.0000   -0.0000    0.0000    0.0002
         2    -0.0451   -0.4115   -0.0009   -0.0001    0.0000    0.0025
         3     9.3514    1.4648   -0.0000    0.0000    0.0000   -0.0031
        11     9.3032    1.0237   -0.0009   -0.0001    0.0000   -0.0004
        12    -9.3995   -1.9060   -0.0009   -0.0001    0.0000    0.0057
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   11
  SUPPORT REACTIONS -UNIT KGS  METE    STRUCTURE TYPE = SPACE
  -----------------
JOINT  LOAD   FORCE-X   FORCE-Y   FORCE-Z     MOM-X     MOM-Y     MOM Z
    1    1      61.39   1009.30      0.98      0.00      0.00      0.00
         2     872.77   3562.50    -19.74      0.00      0.00      0.00
         3   -2356.64  -3311.14     -0.00      0.00      0.00      0.00
        11   -1422.49   1260.65    -18.76      0.00      0.00      0.00
        12    3290.79   7882.94    -18.76      0.00      0.00      0.00
    5    1     -61.39   1009.45      0.98      0.00      0.00      0.00
         2    -872.77   3562.50    -19.74      0.00      0.00      0.00
         3   -2392.50   3308.29      0.00      0.00      0.00      0.00
        11   -3326.65   7880.23    -18.76      0.00      0.00      0.00
        12    1458.35   1263.66    -18.76      0.00      0.00      0.00
    9    1      61.39   1009.30     -0.98      0.00      0.00      0.00
         2     872.77   3562.50     19.74      0.00      0.00      0.00
         3   -2356.64  -3311.14     -0.00      0.00      0.00      0.00
        11   -1422.49   1260.65     18.76      0.00      0.00      0.00
        12    3290.79   7882.94     18.76      0.00      0.00      0.00
   13    1     -61.39   1009.45     -0.98      0.00      0.00      0.00
         2    -872.77   3562.50     19.74      0.00      0.00      0.00
         3   -2392.50   3308.29      0.00      0.00      0.00      0.00
        11   -3326.65   7880.23     18.76      0.00      0.00      0.00
        12    1458.35   1263.66     18.76      0.00      0.00      0.00
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   12
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KGS  METE     (LOCAL )
 MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    1    1     1    1009.30    -61.39     0.98      0.00      0.00       0.00
               2    -666.89     61.39    -0.98      0.00     -3.43    -214.85
         2     1    3562.50   -872.77   -19.74      0.00     -0.00      -0.00
               2   -3562.50    872.77    19.74      0.00     69.09   -3054.69
         3     1   -3311.14   2356.64    -0.00      0.00      0.00       0.00
               2    3311.14  -2356.64     0.00      0.00      0.00    8248.24
        11     1    1260.65   1422.49   -18.76      0.00     -0.00       0.00
               2    -918.24  -1422.49    18.76      0.00     65.66    4978.71
        12     1    7882.94  -3290.79   -18.76      0.00     -0.00      -0.00
               2   -7540.53   3290.79    18.76      0.00     65.66  -11517.78
    2    1     2     641.00    -61.39     5.97      0.01     -9.03     214.85
```

```
              3   -464.90     61.39    -5.97     -0.01     -1.72    -325.34
        2     2   3562.50   -872.77   112.31      0.24    -79.35    3054.69
              3  -3562.50    872.77  -112.31     -0.24   -122.82   -4625.67
        3     2  -3311.13   2253.63    -0.00     -0.00      0.00   -8248.24
              3   3311.13  -2253.63     0.00      0.00      0.00   12304.78
       11     2    892.36   1319.48   118.29      0.24    -88.38   -4978.71
              3   -716.27  -1319.48  -118.29     -0.24   -124.54    7353.77
       12     2   7514.63  -3187.79   118.29      0.24    -88.38   11517.78
              3  -7338.53   3187.79  -118.29     -0.24   -124.54  -17255.79
    3   1     3    191.68    186.66     5.96     -0.01      1.70     310.00
              4    -25.36   -186.66    -5.96      0.01    -11.84       7.31
        2     3     41.65   2832.69   111.63     -0.35    122.13    4570.39
              4    -41.65  -2832.69  -111.63      0.35   -311.90     245.19
        3     3   6886.37  -8900.35    -0.00     -0.00     -0.00  -12336.68
              4  -6886.37   8900.35     0.00      0.00      0.00   -2793.95
       11     3   7119.69  -5881.00   117.60     -0.36    123.83   -7456.29
              4  -6953.38   5881.00  -117.60      0.36   -323.74   -2541.45
       12     3  -6653.05  11919.69   117.60     -0.36    123.83   17217.07
              4   6819.36 -11919.69  -117.60      0.36   -323.74    3046.45
    4   1     5   1009.45     61.39     0.98      0.00      0.00      -0.00
              6   -667.04    -61.39    -0.98      0.00     -3.43     214.85
        2     5   3562.50    872.77   -19.74      0.00      0.00      -0.00
              6  -3562.50   -872.77    19.74      0.00     69.07    3054.69
        3     5   3308.29   2392.50     0.00      0.00      0.00       0.00
              6  -3308.29  -2392.50    -0.00      0.00     -0.00    8373.75
       11     5   7880.23   3326.65   -18.76      0.00      0.00       0.00
              6  -7537.82  -3326.65    18.76      0.00     65.65   11643.29
       12     5   1263.66  -1458.35   -18.76      0.00      0.00      -0.00
              6   -921.25   1458.35    18.76      0.00     65.65   -5104.22
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   13
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    5   1     6    641.15     61.39     5.97     -0.01     -9.03    -214.85
              7   -465.05    -61.39    -5.97      0.01     -1.72     325.34
        2     6   3562.50    872.77   112.28     -0.22    -79.33   -3054.69
              7  -3562.50   -872.77  -112.28      0.22   -122.76    4625.67
        3     6   3308.16   2288.65     0.00      0.00     -0.00   -8373.75
              7  -3308.16  -2288.65    -0.00     -0.00     -0.00   12493.31
       11     6   7511.81   3222.80   118.25     -0.23    -88.36  -11643.29
              7  -7335.71  -3222.80  -118.25      0.23   -124.49   17444.32
       12     6    895.48  -1354.49   118.25     -0.23    -88.36    5104.22
              7   -719.39   1354.49  -118.25      0.23   -124.49   -7542.31
    6   1     7    206.02   -182.87     5.96      0.01      1.71    -308.77
              8    -39.71    182.87    -5.96     -0.01    -11.84      -2.12
        2     7    257.08  -2789.79   111.63      0.33    122.14   -4566.17
              8   -257.08   2789.79  -111.63     -0.33   -311.91    -176.47
        3     7  -6533.99  -9068.05     0.00     -0.00      0.00  -12526.20
              8   6533.99   9068.05    -0.00      0.00     -0.00   -2889.65
       11     7  -6070.89 -12040.72   117.59      0.34    123.84  -17401.14
              8   6237.20  12040.72  -117.59     -0.34   -323.75   -3068.23
       12     7   6997.09   6095.39   117.59      0.34    123.84    7651.26
              8  -6830.78  -6095.39  -117.59     -0.34   -323.75    2711.06
    7   1     9   1009.30    -61.39    -0.98      0.00     -0.00       0.00
             10   -666.89     61.39     0.98      0.00      3.43    -214.85
        2     9   3562.50   -872.77    19.74      0.00      0.00       0.00
             10  -3562.50    872.77   -19.74      0.00    -69.09   -3054.69
```

```
          3     9  -3311.14   2356.64     -0.00       0.00       0.00      -0.00
               10   3311.14  -2356.64      0.00       0.00       0.00    8248.24
         11     9   1260.65   1422.49     18.76       0.00       0.00      -0.00
               10   -918.24  -1422.49    -18.76       0.00     -65.66    4978.71
         12     9   7882.94  -3290.79     18.76       0.00       0.00       0.00
               10  -7540.53   3290.79    -18.76       0.00     -65.66  -11517.78
      8   1    10    641.00    -61.39     -5.97      -0.01       9.03     214.85
               11   -464.90     61.39      5.97       0.01       1.72    -325.34
          2    10   3562.50   -872.77   -112.31      -0.24      79.35    3054.69
               11  -3562.50    872.77    112.31       0.24     122.82   -4625.67
          3    10  -3311.13   2253.63     -0.00      -0.00       0.00   -8248.24
               11   3311.13  -2253.63      0.00       0.00       0.00   12304.78
         11    10    892.36   1319.48   -118.29      -0.24      88.38   -4978.71
               11   -716.27  -1319.48    118.29       0.24     124.54    7353.77
         12    10   7514.63  -3187.79   -118.29      -0.24      88.38   11517.78
               11  -7338.53   3187.79    118.29       0.24     124.54  -17255.79
      9   1    11    191.68    186.66     -5.96       0.01      -1.70     310.00
               12    -25.36   -186.66      5.96      -0.01      11.84       7.31
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   14
   MEMBER END FORCES    STRUCTURE TYPE = SPACE
   -----------------
   ALL UNITS ARE -- KGS  METE     (LOCAL )
  MEMBER   LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          2    11     41.65   2832.69   -111.63       0.35    -122.13    4570.39
               12    -41.65  -2832.69    111.63      -0.35     311.90     245.19
          3    11   6886.37  -8900.35     -0.00      -0.00      -0.00  -12336.68
               12  -6886.37   8900.35      0.00       0.00       0.00   -2793.95
         11    11   7119.69  -5881.00   -117.60       0.36    -123.83   -7456.29
               12  -6953.38   5881.00    117.60      -0.36     323.74   -2541.45
         12    11  -6653.05  11919.69   -117.60       0.36    -123.83   17217.07
               12   6819.36 -11919.69    117.60      -0.36     323.74    3046.45
     10   1    13   1009.45     61.39     -0.98       0.00      -0.00      -0.00
               14   -667.04    -61.39      0.98       0.00       3.43     214.85
          2    13   3562.50    872.77     19.74       0.00       0.00       0.00
               14  -3562.50   -872.77    -19.74       0.00     -69.07    3054.69
          3    13   3308.29   2392.50      0.00       0.00       0.00      -0.00
               14  -3308.29  -2392.50     -0.00       0.00      -0.00    8373.75
         11    13   7880.23   3326.65     18.76       0.00       0.00      -0.00
               14  -7537.82  -3326.65    -18.76       0.00     -65.65   11643.29
         12    13   1263.66  -1458.35     18.76       0.00       0.00       0.00
               14   -921.25   1458.35    -18.76       0.00     -65.65   -5104.22
     11   1    14    641.15     61.39     -5.97       0.01       9.03    -214.85
               15   -465.05    -61.39      5.97      -0.01       1.72     325.34
          2    14   3562.50    872.77   -112.28       0.22      79.33   -3054.69
               15  -3562.50   -872.77    112.28      -0.22     122.76    4625.67
          3    14   3308.16   2288.65      0.00       0.00      -0.00   -8373.75
               15  -3308.16  -2288.65     -0.00      -0.00      -0.00   12493.31
         11    14   7511.81   3222.80   -118.25       0.23      88.36  -11643.29
               15  -7335.71  -3222.80    118.25      -0.23     124.49   17444.32
         12    14    895.48  -1354.49   -118.25       0.23      88.36    5104.22
               15   -719.39   1354.49    118.25      -0.23     124.49   -7542.31
     12   1    15    206.02   -182.87     -5.96      -0.01      -1.71    -308.77
               16    -39.71    182.87      5.96       0.01      11.84      -2.12
          2    15    257.08  -2789.79   -111.63      -0.33    -122.14   -4566.17
               16   -257.08   2789.79    111.63       0.33     311.91    -176.47
          3    15  -6533.99  -9068.05      0.00      -0.00       0.00  -12526.20
               16   6533.99   9068.05     -0.00       0.00      -0.00   -2889.65
         11    15  -6070.89 -12040.72   -117.59      -0.34    -123.84  -17401.14
```

```
                16   6237.20  12040.72   117.59      0.34    323.75   -3068.23
          12    15   6997.09   6095.39  -117.59     -0.34   -123.84    7651.26
                16  -6830.78  -6095.39   117.59      0.34    323.75    2711.06
    13     1     4   -186.66     -0.53    -0.01      0.00     -0.01      -7.31
                17    186.66     70.96     0.01     -0.00      0.03     -57.03
           2     4  -2832.69   -520.85    -0.69      0.09     -0.29    -245.19
                17   2832.69   1308.35     0.69     -0.09      1.54   -1176.10
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                        -- PAGE NO.   15
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
           3     4   9442.80   6886.14    -0.00      0.00     -0.00    2793.95
                17  -9442.80  -6886.14     0.00     -0.00      0.00    9601.11
          11     4   6423.46   6364.76    -0.70      0.09     -0.30    2541.45
                17  -6423.46  -5506.82     0.70     -0.09      1.57    8367.98
          12     4 -12462.15  -7407.52    -0.70      0.09     -0.30   -3046.45
                17  12462.15   8265.46     0.70     -0.09      1.57  -10834.23
    14     1    17     61.39    129.40    -0.00     -0.03      0.01      67.81
                18    -61.39    -19.82     0.00      0.03     -0.01     141.09
           2    17    872.77   2212.50    -0.01     -1.76      0.27    1193.12
                18   -872.77   -675.00     0.01      1.76     -0.24    2343.13
           3    17   -725.43  -3432.33    -0.00     -0.00     -0.00  -10005.50
                18    725.43   3432.33     0.00      0.00     -0.00     395.71
          11    17    208.72  -1090.44    -0.01     -1.79      0.28   -8744.57
                18   -208.72   2737.51     0.01      1.79     -0.24    2879.93
          12    17   1659.58   5774.23    -0.01     -1.79      0.28   11266.44
                18  -1659.58  -4127.16     0.01      1.79     -0.24    2088.51
    15     1    18     61.39     -6.07     0.00      0.03      0.01    -141.09
                19    -61.39    123.46    -0.00     -0.03     -0.01     -53.20
           2    18    872.77   -450.00     0.01      1.64      0.23   -2343.13
                19   -872.77   2137.50    -0.01     -1.64     -0.26    -975.62
           3    18    667.43  -3441.37     0.00     -0.00      0.00    -395.71
                19   -667.43   3441.37    -0.00      0.00     -0.00   -9929.10
          11    18   1601.58  -3897.43     0.01      1.67      0.24   -2879.93
                19  -1601.58   5702.33    -0.01     -1.67     -0.27  -10957.92
          12    18    266.73   2985.30     0.01      1.67      0.24   -2088.51
                19   -266.73  -1180.40    -0.01     -1.67     -0.27    8900.27
    16     1    19   -182.87     60.53     0.01     -0.00     -0.03      42.27
                 8    182.87     13.82    -0.01      0.00      0.01       2.12
           2    19  -2789.79   1167.92     0.66     -0.08     -1.52     970.08
                 8   2789.79   -305.42    -0.66      0.08      0.27     176.47
           3    19  -9627.97   6534.44    -0.00      0.00      0.00    9525.79
                 8   9627.97  -6534.44     0.00     -0.00     -0.00    2889.65
          11    19 -12600.64   7762.89     0.67     -0.09     -1.55   10538.13
                 8  12600.64  -6826.04    -0.67      0.09      0.28    3068.23
          12    19   6655.31  -5305.98     0.67     -0.09     -1.55   -8513.44
                 8  -6655.31   6242.83    -0.67      0.09      0.28   -2711.06
    17     1    12   -186.66     -0.53     0.01     -0.00      0.01      -7.31
                20    186.66     70.96    -0.01      0.00     -0.03     -57.03
           2    12  -2832.69   -520.85     0.69     -0.09      0.29    -245.19
                20   2832.69   1308.35    -0.69      0.09     -1.54   -1176.10
           3    12   9442.80   6886.14    -0.00      0.00     -0.00    2793.95
                20  -9442.80  -6886.14     0.00     -0.00      0.00    9601.11
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                        -- PAGE NO.   16
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
```

```
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          11   12   6423.46   6364.76     0.70     -0.09      0.30    2541.45
               20  -6423.46  -5506.82    -0.70      0.09     -1.57    8367.98
          12   12 -12462.15  -7407.52     0.70     -0.09      0.30   -3046.45
               20  12462.15   8265.46    -0.70      0.09     -1.57  -10834.23
     18    1   20     61.39    129.40     0.00      0.03     -0.01      67.81
               21    -61.39    -19.82    -0.00     -0.03      0.01     141.09
           2   20    872.77   2212.50     0.01      1.76     -0.27    1193.12
               21   -872.77   -675.00    -0.01     -1.76      0.24    2343.13
           3   20   -725.43  -3432.34    -0.00     -0.00     -0.00  -10005.50
               21    725.43   3432.34     0.00      0.00     -0.00     395.71
          11   20    208.72  -1090.44     0.01      1.79     -0.28   -8744.57
               21   -208.72   2737.51    -0.01     -1.79      0.24    2879.93
          12   20   1659.58   5774.23     0.01      1.79     -0.28   11266.44
               21  -1659.58  -4127.16    -0.01     -1.79      0.24    2088.51
     19    1   21     61.39     -6.07    -0.00     -0.03     -0.01    -141.09
               22    -61.39    123.46     0.00      0.03      0.01     -53.20
           2   21    872.77   -450.00    -0.01     -1.64     -0.23   -2343.13
               22   -872.77   2137.50     0.01      1.64      0.26    -975.62
           3   21    667.43  -3441.37     0.00     -0.00      0.00    -395.71
               22   -667.43   3441.37    -0.00      0.00     -0.00   -9929.10
          11   21   1601.58  -3897.43    -0.01     -1.67     -0.24   -2879.93
               22  -1601.58   5702.33     0.01      1.67      0.27  -10957.92
          12   21    266.73   2985.30    -0.01     -1.67     -0.24   -2088.51
               22   -266.73  -1180.40     0.01      1.67      0.27    8900.27
     20    1   22   -182.87     60.53    -0.01      0.00      0.03      42.27
               16    182.87     13.82     0.01     -0.00     -0.01       2.12
           2   22  -2789.79   1167.92    -0.66      0.08      1.52     970.08
               16   2789.79   -305.42     0.66     -0.08     -0.27     176.47
           3   22  -9627.98   6534.44    -0.00      0.00      0.00    9525.79
               16   9627.98  -6534.44     0.00     -0.00     -0.00    2889.65
          11   22 -12600.64   7762.89    -0.67      0.09      1.55   10538.13
               16  12600.64  -6826.04     0.67     -0.09     -0.28    3068.23
          12   22   6655.31  -5305.98    -0.67      0.09      1.55   -8513.44
               16  -6655.31   6242.84     0.67     -0.09     -0.28   -2711.06
     21    1    2     -5.00    -25.89     0.00     -0.00      0.01     -12.46
               10      5.00    -25.89    -0.00      0.00     -0.01      12.46
           2    2   -132.05     -0.00     0.00     -0.00      0.24     -10.26
               10    132.05      0.00    -0.00      0.00     -0.24      10.26
           3    2      0.00      0.00     0.00     -0.00     -0.00       0.00
               10      0.00     -0.00    -0.00      0.00     -0.00       0.00
          11    2   -137.05    -25.89     0.00     -0.00      0.24     -22.72
               10    137.05    -25.89    -0.00      0.00     -0.24      22.72
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                         -- PAGE NO.   17
  MEMBER END FORCES    STRUCTURE TYPE = SPACE
  -----------------
  ALL UNITS ARE -- KGS  METE     (LOCAL )
 MEMBER  LOAD  JT     AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
          12    2   -137.05    -25.89    -0.00      0.00      0.24     -22.72
               10    137.05    -25.89     0.00     -0.00     -0.24      22.72
     22    1    4      5.97    -25.89     0.00     -0.00      0.00     -11.85
               12     -5.97    -25.89    -0.00      0.00     -0.00      11.85
           2    4    112.33   -562.50     0.00     -0.00      0.06    -311.99
               12   -112.33   -562.50    -0.00      0.00     -0.06     311.99
           3    4      0.00      0.00     0.00     -0.00     -0.00       0.00
               12      0.00     -0.00    -0.00      0.00     -0.00       0.00
          11    4    118.30   -588.39     0.00     -0.00      0.06    -323.84
               12   -118.30   -588.39    -0.00      0.00     -0.06     323.84
```

```
         12     4    118.30   -588.39    -0.00      0.00      0.06   -323.84
               12   -118.30   -588.39     0.00     -0.00     -0.06    323.84
    23    1     6     -4.99     25.89    -0.00     -0.00      0.01     12.46
               14      4.99     25.89     0.00      0.00     -0.01    -12.46
          2     6   -132.01     -0.00    -0.00     -0.00      0.22     10.26
               14    132.01      0.00     0.00      0.00     -0.22    -10.26
          3     6      0.00      0.00     0.00     -0.00     -0.00      0.00
               14      0.00     -0.00    -0.00      0.00     -0.00      0.00
         11     6   -137.01     25.89     0.00     -0.00      0.23     22.71
               14    137.01     25.89    -0.00      0.00     -0.23    -22.71
         12     6   -137.01     25.89    -0.00      0.00      0.23     22.71
               14    137.01     25.89     0.00     -0.00     -0.23    -22.71
    24    1     8      5.97     25.89    -0.00     -0.00      0.00     11.85
               16     -5.97     25.89     0.00      0.00     -0.00    -11.85
          2     8    112.29    562.50    -0.00     -0.00      0.06    311.99
               16   -112.29    562.50     0.00      0.00     -0.06   -311.99
          3     8      0.00      0.00    -0.00     -0.00      0.00      0.00
               16      0.00     -0.00     0.00      0.00      0.00      0.00
         11     8    118.26    588.39    -0.00     -0.00      0.06    323.84
               16   -118.26    588.39     0.00      0.00     -0.06   -323.84
         12     8    118.26    588.39     0.00      0.00      0.06    323.84
               16   -118.26    588.39    -0.00     -0.00     -0.06   -323.84
    25    1     3    367.93     28.33     0.01     -0.00      0.02     15.34
               17   -317.90     24.65    -0.01      0.00     -0.05    -10.79
          2     3   5111.42     15.45     0.68     -0.10      0.90     55.28
               17  -5111.42    -15.45    -0.68      0.10     -2.59    -17.03
          3     3 -15037.77    176.23     0.00      0.00     -0.00     31.99
               17  15037.77   -176.23    -0.00     -0.00      0.00    404.40
         11     3  -9558.42    220.00     0.69     -0.10      0.93    102.61
               17   9608.45   -167.03    -0.69      0.10     -2.64    376.59
         12     3  20517.12   -132.45     0.69     -0.10      0.93     38.63
               17 -20467.09    185.42    -0.69      0.10     -2.64   -432.21
    DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   18
 MEMBER END FORCES    STRUCTURE TYPE = SPACE
 -----------------
 ALL UNITS ARE -- KGS  METE     (LOCAL )
MEMBER  LOAD  JT      AXIAL   SHEAR-Y  SHEAR-Z   TORSION     MOM-Y      MOM-Z
    26    1     7    354.75     30.17    -0.01      0.00     -0.02     16.58
               19   -304.72     25.75     0.01     -0.00      0.05    -10.94
          2     7   4933.53     21.16    -0.65      0.10     -0.83     59.49
               19  -4933.53    -21.16     0.65     -0.10      2.48     -5.54
          3     7  14950.07   -171.11    -0.00      0.00      0.00    -33.13
               19 -14950.07    171.11     0.00     -0.00      0.00   -403.31
         11     7  20238.35   -119.78    -0.66      0.10     -0.85     42.93
               19 -20188.32    175.70     0.66     -0.10      2.53   -419.79
         12     7  -9661.80    222.44    -0.66      0.10     -0.85    109.20
               19   9711.83   -166.53     0.66     -0.10      2.53    386.83
    27    1    11    367.93     28.33    -0.01      0.00     -0.02     15.34
               20   -317.90     24.65     0.01     -0.00      0.05    -10.79
          2    11   5111.42     15.45    -0.68      0.10     -0.90     55.28
               20  -5111.42    -15.45     0.68     -0.10      2.59    -17.03
          3    11 -15037.77    176.23     0.00      0.00     -0.00     31.99
               20  15037.77   -176.23    -0.00     -0.00      0.00    404.40
         11    11  -9558.42    220.00    -0.69      0.10     -0.93    102.61
               20   9608.45   -167.03     0.69     -0.10      2.64    376.59
         12    11  20517.12   -132.45    -0.69      0.10     -0.93     38.63
               20 -20467.09    185.42     0.69     -0.10      2.64   -432.21
    28    1    15    354.75     30.17     0.01     -0.00      0.02     16.58
```

```
                22   -304.72      25.75    -0.01      0.00     -0.05     -10.94
           2    15   4933.53      21.16     0.65     -0.10      0.83      59.49
                22  -4933.53     -21.16    -0.65      0.10     -2.48      -5.54
           3    15  14950.07    -171.11    -0.00      0.00      0.00     -33.13
                22 -14950.07     171.11     0.00     -0.00      0.00    -403.31
          11    15  20238.35    -119.78     0.66     -0.10      0.85      42.93
                22 -20188.32     175.70    -0.66      0.10     -2.53    -419.79
          12    15  -9661.80     222.44     0.66     -0.10      0.85     109.20
                22   9711.83    -166.53    -0.66      0.10     -2.53     386.83
     29    1    18     -0.00      25.89    -0.00      0.00      0.00       0.06
                21      0.00      25.89     0.00     -0.00     -0.00      -0.06
           2    18     -0.02    1125.00    -0.00      0.00      0.00       3.41
                21      0.02    1125.00     0.00     -0.00     -0.00      -3.41
           3    18      0.00       0.00     0.00      0.00     -0.00       0.00
                21      0.00      -0.00    -0.00     -0.00     -0.00       0.00
          11    18     -0.02    1150.89     0.00      0.00      0.00       3.47
                21      0.02    1150.89    -0.00     -0.00     -0.00      -3.47
          12    18     -0.02    1150.89    -0.00     -0.00      0.00       3.47
                21      0.02    1150.89     0.00      0.00     -0.00      -3.47
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   19
   ************** END OF LATEST ANALYSIS RESULT **************
    75. LOAD LIST 11 12
    76. PARAMETER
    77. CODE CANADA
    78. CHECK CODE ALL
 STEEL DESIGN
   WARNING: The current units are in KGS  METE. All the design results are in KN
MET  (unless otherwise noted).
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   20
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      1      SECTION: ST  W310X97         (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.321         LOADCASE   12      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      2      SECTION: ST  W310X97         (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   12      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      3      SECTION: ST  W310X97         (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.464         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      4      SECTION: ST  W310X97         (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.324         LOADCASE   11      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   21
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
   ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      5      SECTION: ST  W310X97         (CANADIAN SECTIONS)|
```

```
|  STATUS:        PASS    CRITICAL RATIO:  0.471         LOADCASE   11      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      6      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   11      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      7      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.321         LOADCASE   12      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:      8      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   12      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                         -- PAGE NO.   22
                    STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                  ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:      9      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.464         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:     10      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.324         LOADCASE   11      |
|  LOCATION:      3.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:     11      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.471         LOADCASE   11      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8.4         |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:     12      SECTION: ST  W310X97          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.467         LOADCASE   11      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|--------------------------------------------------------------------------|
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                         -- PAGE NO.   23
                    STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                  ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|--------------------------------------------------------------------------|
|   MEMBER NO.:     13      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.768         LOADCASE   12      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8           |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:     14      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.798         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8           |
|--------------------------------------------------------------------------|
|--------------------------------------------------------------------------|
|   MEMBER NO.:     15      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
```

```
|  STATUS:        PASS    CRITICAL RATIO:  0.776         LOADCASE   11      |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     16      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.747         LOADCASE   11      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   24
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:     17      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.768         LOADCASE   12      |
|  LOCATION:      1.80    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     18      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.798         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     19      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.776         LOADCASE   11      |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     20      SECTION: ST  W250X39          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.747         LOADCASE   11      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8           |
|---------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   25
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:     21      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.009         LOADCASE   12      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     22      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.118         LOADCASE   11      |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     23      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.009         LOADCASE   12      |
|  LOCATION:      3.00    CRITICAL CONDITION:            Cl. 13.9.1         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     24      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.118         LOADCASE   11      |
|  LOCATION:      0.00    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
      DYNAMIC ANALYSIS FOR SEISMIC LOADS                      -- PAGE NO.   26
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
```

```
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:     25      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.675         LOADCASE   12      |
|  LOCATION:      2.48    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     26      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.671         LOADCASE   11      |
|  LOCATION:      2.55    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     27      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.675         LOADCASE   12      |
|  LOCATION:      2.48    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
|---------------------------------------------------------------------------|
|   MEMBER NO.:     28      SECTION: ST  L152X152X13      (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.671         LOADCASE   11      |
|  LOCATION:      2.55    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
     DYNAMIC ANALYSIS FOR SEISMIC LOADS                       -- PAGE NO.   27
                       STAAD.PRO CODE CHECKING - CSA S16-19 (v1.0)
                     ***********************************************
  ALL UNITS ARE IN KN   MET   UNLESS OTHERWISE Noted
|---------------------------------------------------------------------------|
|   MEMBER NO.:     29      SECTION: ST  C200X17          (CANADIAN SECTIONS)|
|  STATUS:        PASS    CRITICAL RATIO:  0.444         LOADCASE   11      |
|  LOCATION:      1.50    CRITICAL CONDITION:            Cl. 13.8.4         |
|---------------------------------------------------------------------------|
  ***NOTE:OMEGA1 AND OMEGA2 ARE CALCULATED USING C/S MOMENT OF ANALYTICAL MEMBERS
   79. FINISH
            *********** END OF THE STAAD.Pro RUN ***********
              **** DATE= MAR 24,2022   TIME=  9:45: 2 ****
        ************************************************************
        *   For technical assistance on STAAD.Pro, please visit    *
        *   http://www.bentley.com/en/support/                     *
        *                                                          *
        *   Details about additional assistance from               *
        *   Bentley and Partners can be found at program menu      *
        *   Help->Technical Support                                *
        *                                                          *
        *           Copyright (c) Bentley Systems, Inc.            *
        *                http://www.bentley.com                    *
        ************************************************************
```

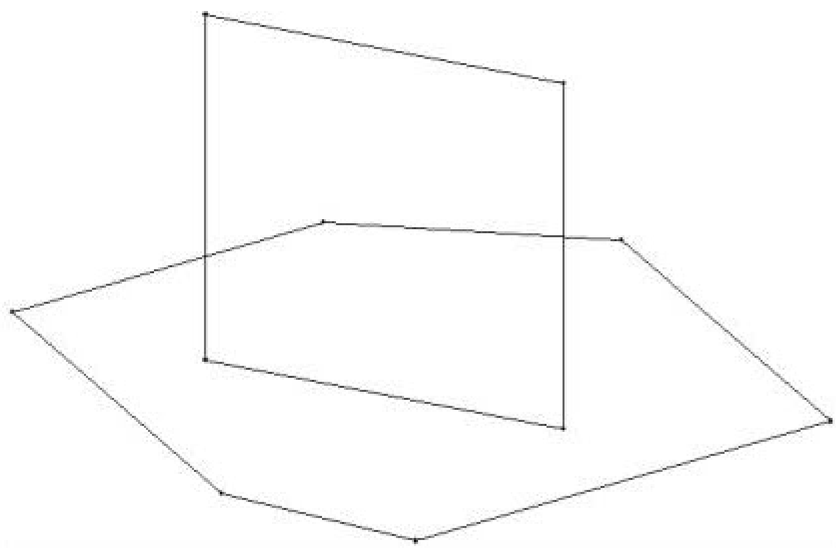
EX. Modeling Examples

These examples demonstrate using some of modeling features in STAAD.Pro.

EX. Meshed Wall-Slab Connection

This example demonstrates how to ensure finite element mesh compatibility between two adjoining meshed entities.

The Model Data



Use the **Copy to clipboard** button below to copy and paste this model data into an empty STAAD input file if you want to work through this example yourself.

```
STAAD SPACE
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 18 0 0; 3 30 0 9; 4 30 0 30; 5 6 0 30; 6 0 0 21; 7 6 0 9;
8 18 0 24; 9 6 20 9; 10 18 20 24;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 5 6; 6 6 1; 7 7 8; 8 9 10; 9 7 9; 10 8 10;
FINISH
```

Related Links

- *M. To define a slab/wall connection* (on page 675)

EX. To create a new view

To create a new view that isolates the slab members for meshing, use the following procedure.

1. On the **View** ribbon tab, select the **Front View** tool in the **Tools** group.

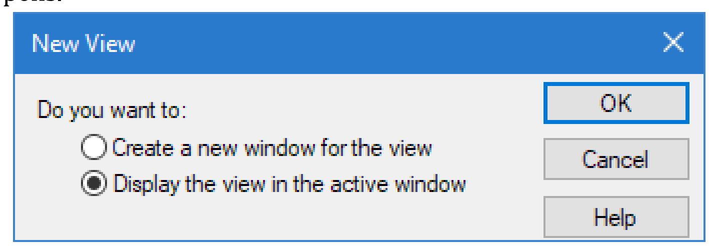
   

   The view rotates to this angle.
2. Click-and-drag the mouse pointer from the top-left towards the bottom-right of the slab members.

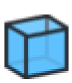
Note: You may need to select the **Beam Cursor** tool on the **Select** ribbon tab first.

3. On the **View** ribbon tab, select the **New View** tool in the **Views** group.

The **New View** dialog opens.



4. Select the **Display the view in the active window** option and then click **OK**.
The new view is displayed in the active view window. Only the selected slab members are currently displayed. The wall members are still contained in the model. They are just hidden from this view.
5. On the **View** ribbon tab, select the **Isometric View** tool in the **Tools** group.



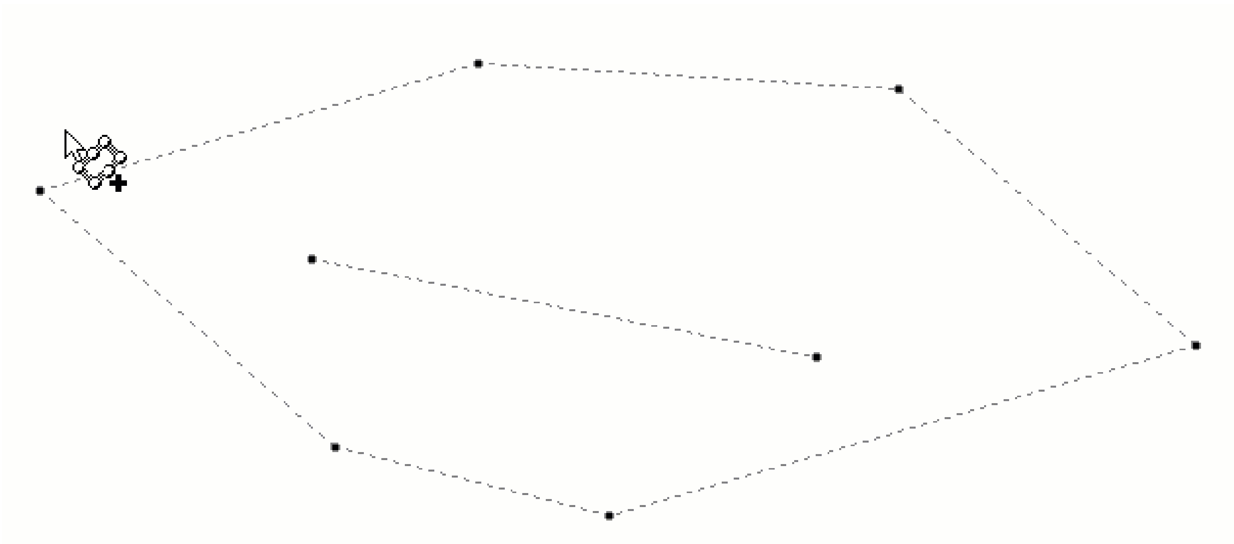
EX. To create a parametric model

To create a slab finite element mesh, use the following procedure.

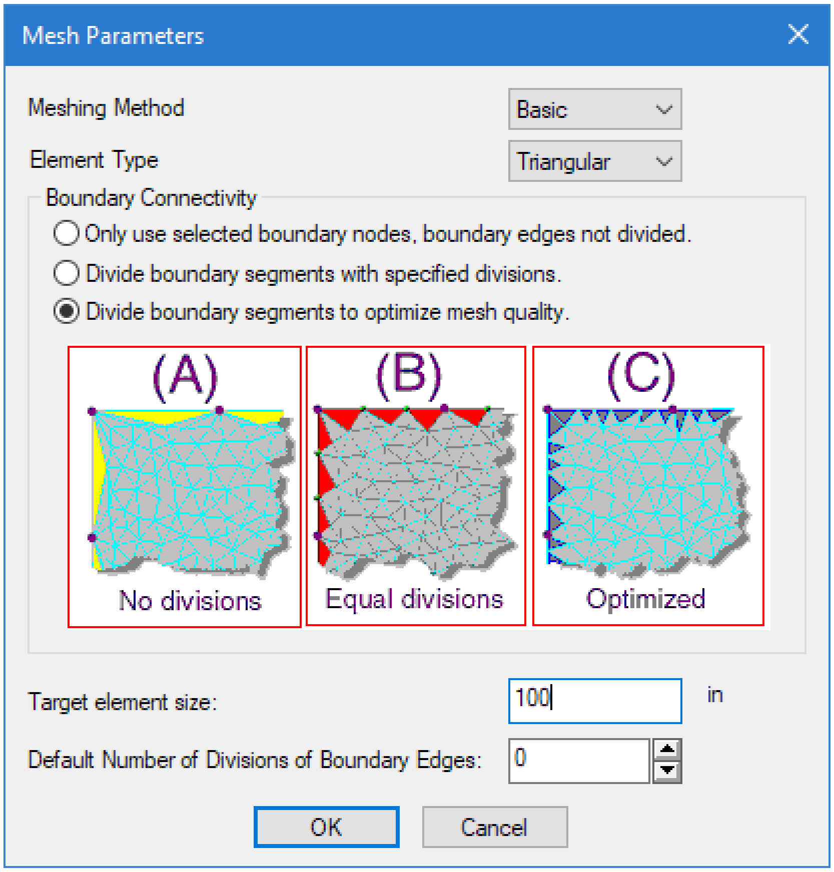
1. Select the **Parametric Models** tool in the Plate group on the **Geometry** ribbon tab.

The **Parametric Models** dialog opens. The structure is displayed as a series of dashed lines to signify this mode can be used to experiment with various settings.
2. Click **Add** in the **Parametric Models** dialog.
The mouse pointer changes to the add surface cursor .
3. Specify the parametric model parameters:
   a. (Optional) Type a **Name** for the slab mesh.

   b. Select **Slab** from the **Type** drop-down list.
   c. Set the **Use nodes and beams that occur on or inside the outer boundary as additional density lines and points?** option.
4. Click the corner nodes of the slab in a clockwise order, clicking on the first node selected to complete the boundary definition.



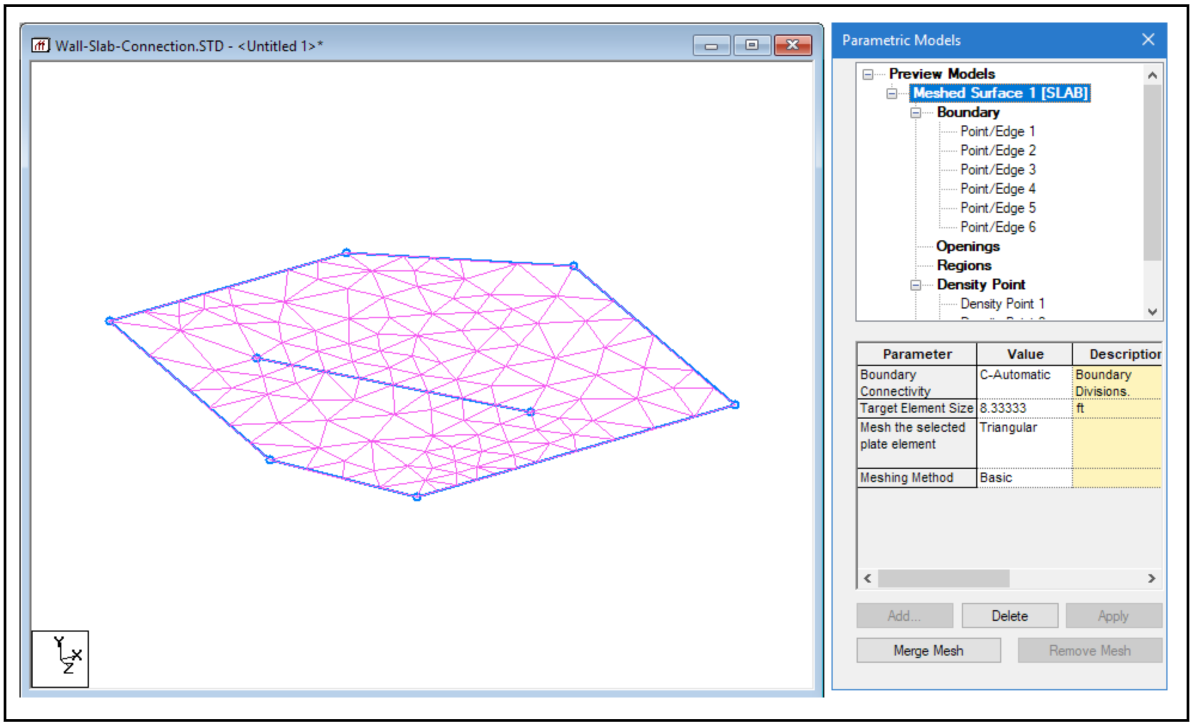
   The **Mesh Parameters** dialog opens.
5. Specify the meshing parameters:
   Leave the Element Type as Triangular, the Boundary method as (C) Optimized, and the Default Number of Divisions of Boundary Edges as zero.



a. Select **Basic** from the **Meshing Method** drop-down list.
b. Type a **Target element size** of `100` (in).
c. Click **OK**.

A message dialog prompts if you want to add any openings.

6. Click **No**.
   The preliminary mesh is displayed in the view window. The mesh parameters are displayed in the **Parametric Models** dialog.
7. In the **Parametric Models** dialog, select the **Meshed Surface 1 [Slab]** entry and then click **Merge Mesh**.



EX. To designate the wall-slab connection

To mesh the wall such that it is compatible with the slab mesh, use the following procedure.

1. In the Analytical Modeling workflow bar, select **Geometry**.
   The Parametric Modeling dialog closes and the plate model is displayed.
2. On the **View** ribbon tab, select the **Display Whole Structure** tool in the **Tools** group.

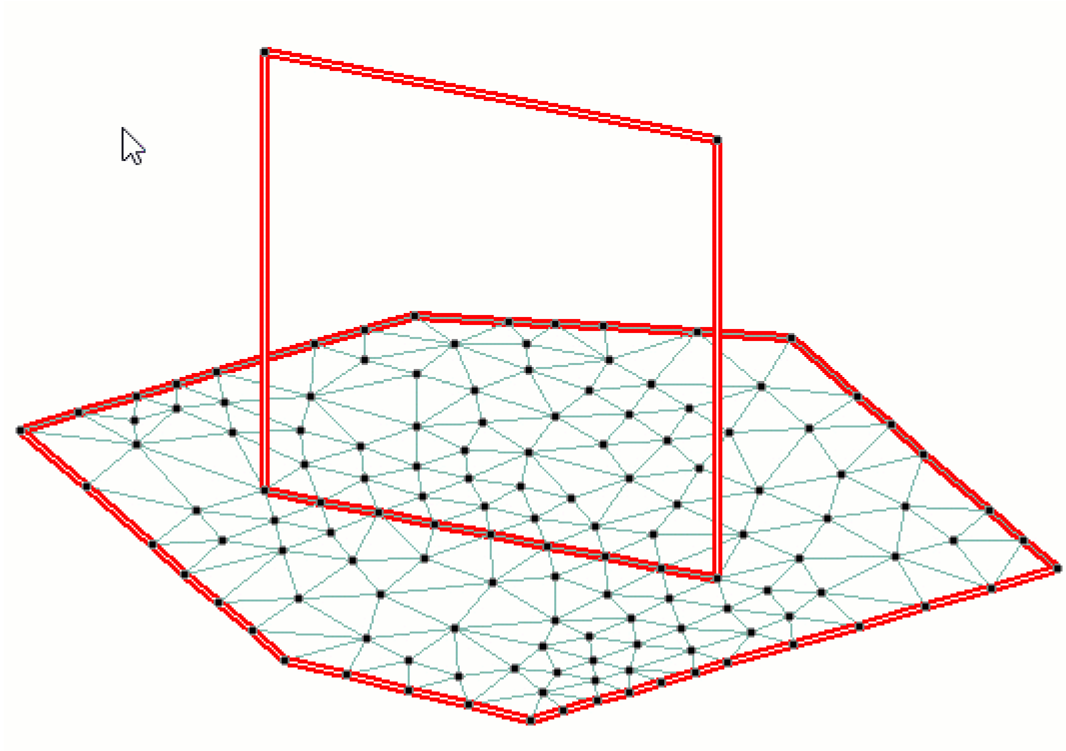
   The standard view is restored, displaying the hidden portions to the view.
3. (Optional) On the **Select** ribbon tab, select the **All** tool in the **Beams** group.

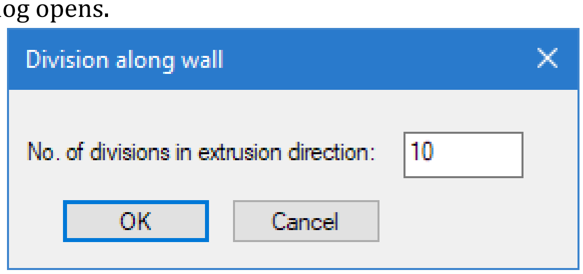
   Tip: It is not necessary to select the members in order to perform the next steps. However, this can server as an easy way to identify the edges of the slab-wall interface when selecting the vertices.
4. On the **Geometry** ribbon tab, select the **Wall/Slab Connection** tool in the **Structure** group.

5. Starting with the two nodes at the base, select the four corner nodes of the wall in a clockwise order, clicking on the first selected node a second time to complete the boundary definition.

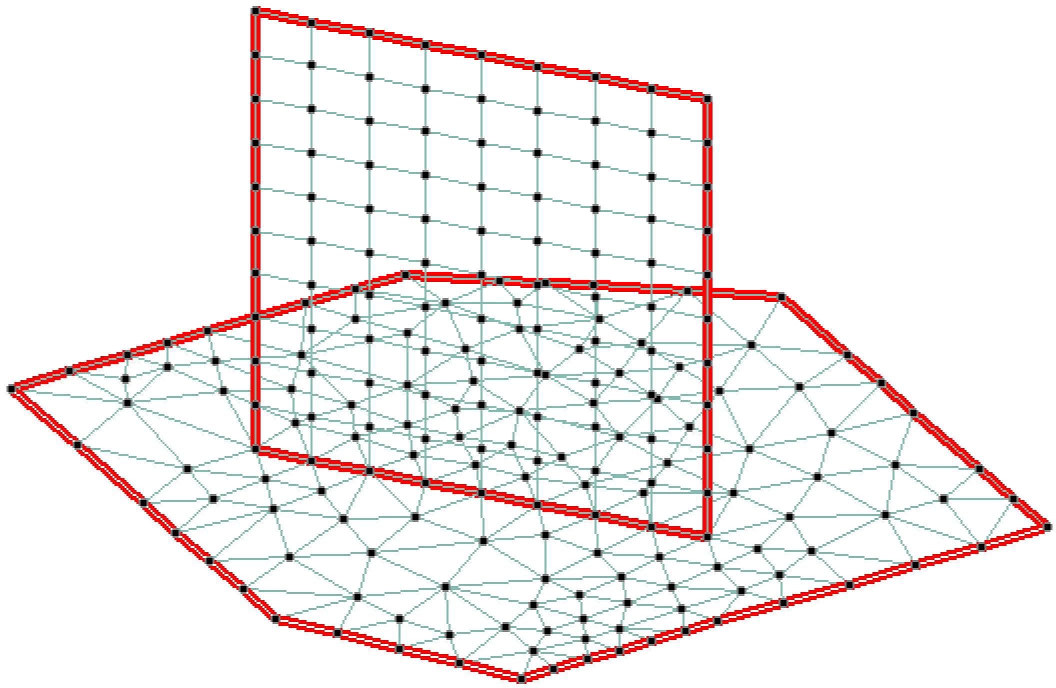


The **Division along wall** dialog opens.



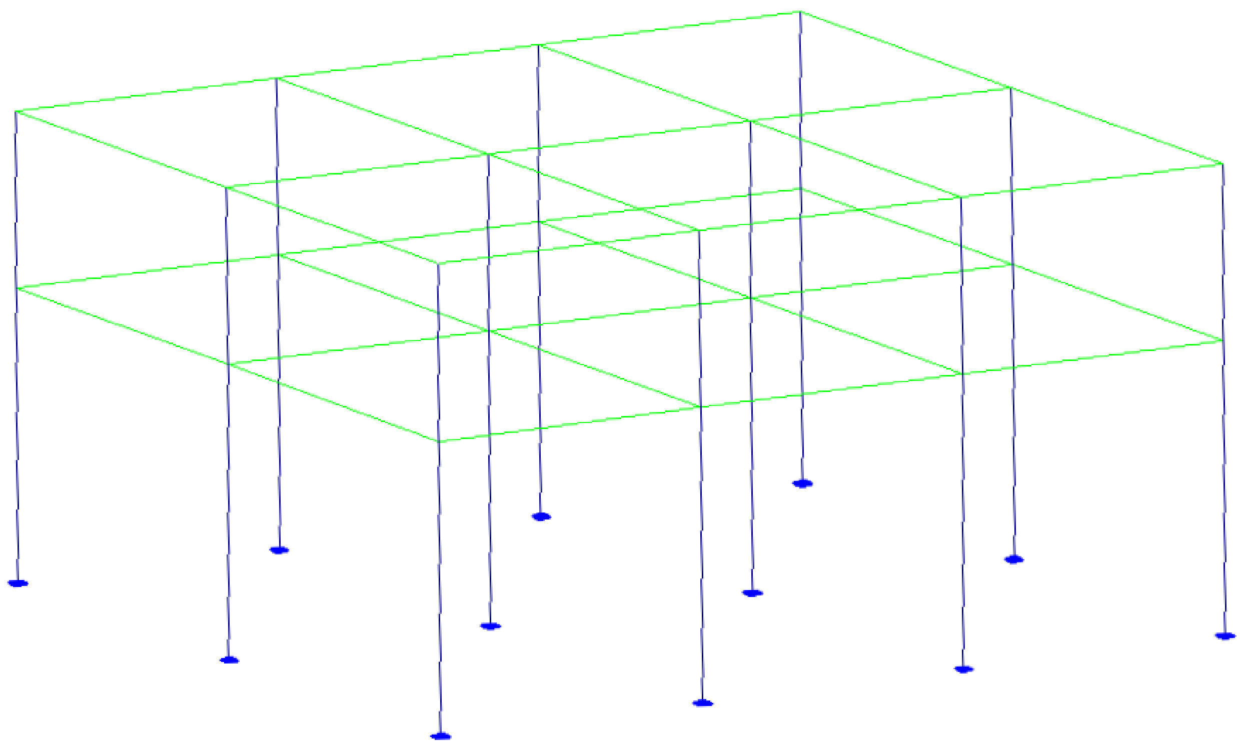
> **Note:** As the divisions along the base of the wall must be compatible with the slab density line, this value will control the meshing along the sides.

6. Click **OK**.
   The default value of 10 divisions is sufficient for this example.
7. On the **Utilities** ribbon tab, select the **Plate Tools** > **Plate Connectivity** tool I the **Geometry Tools** group.
   A message dialog opens indicating that no overlapping plates are found. This indicates also that the slab and wall are properly connected.



EX. Building Planner Workflow Example

This short tutorial will guide you through the create of a small two-story concrete frame building using the Building Planner workflow.



EX. Create the model and plan details

1. On the Start page, select **New**.
   The New page opens to the **Model Info** tab.
2. Type `Building Example` in the **File Name** field and then select a **Location** to store the file.
3. Select the **Building** option for the **Type** of model.
4. Select **Metric** for the **Units**.

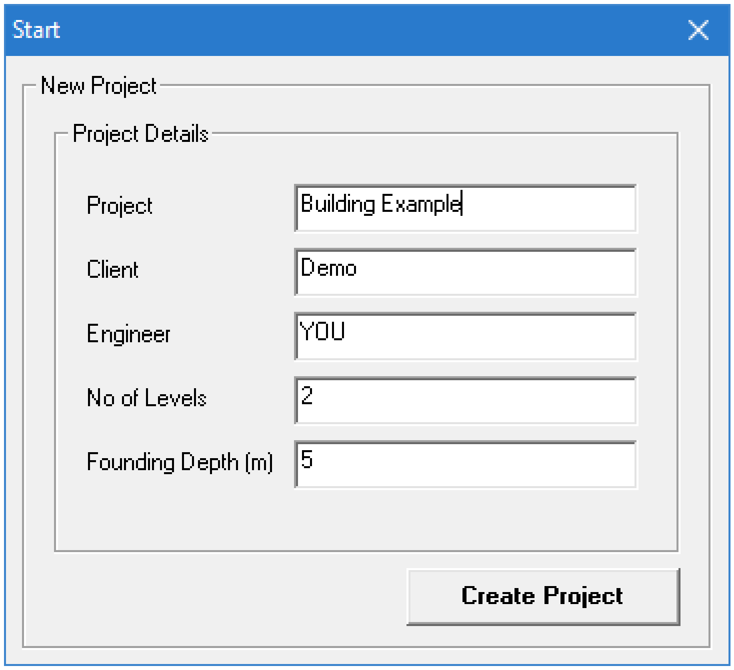
> **Note:** You can use either system of units in the STAAD.Pro Analytical Workflow once you generate the analysis model. However, the Building Planner application only operates in metric units.

5. (Optional) Select the **Job Info** tab to add project member names, dates, project description data, etc.
   You can also associate your STAAD project with a ProjectWise Project here.
6. Click **Create**.

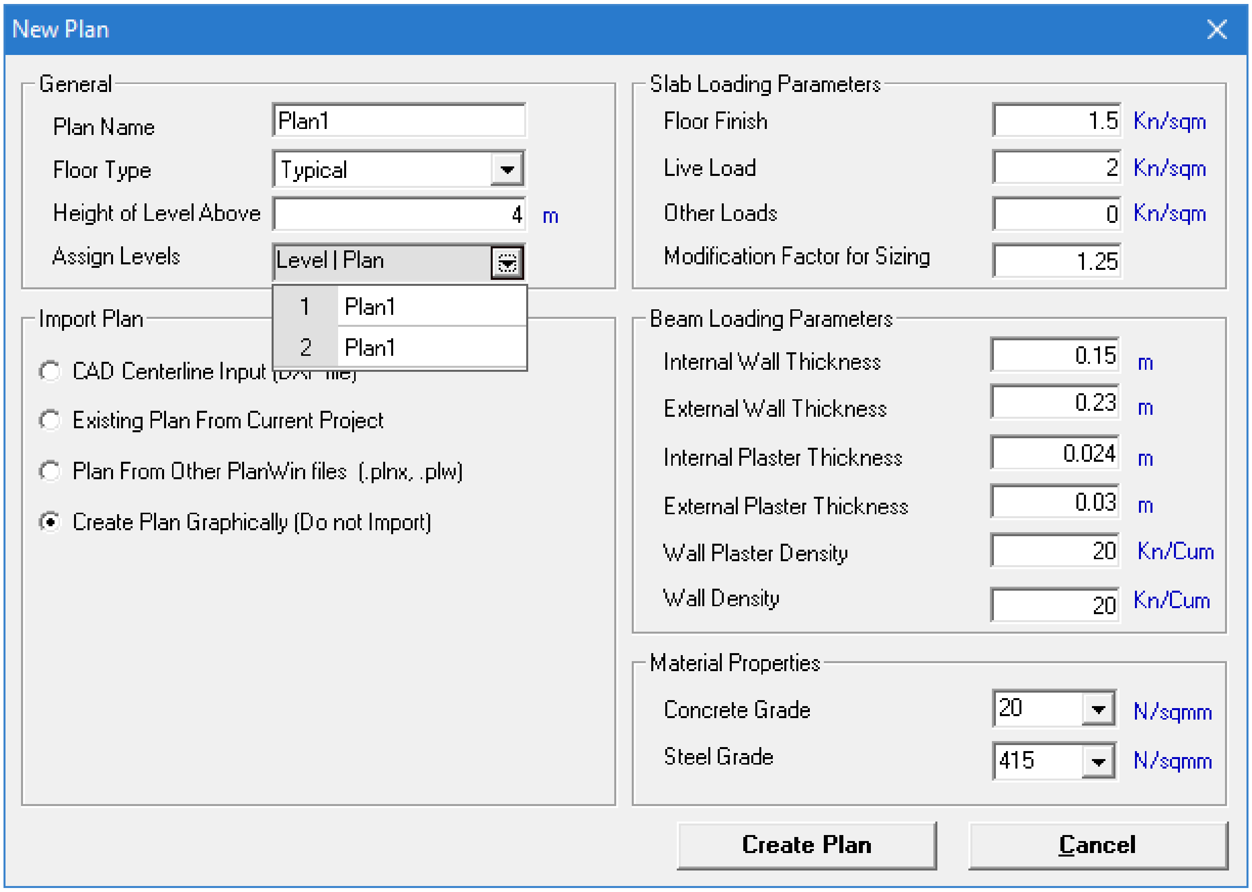
   The STAAD.Pro window closes and the STAAD Building Planner application opens. The **Start** dialog opens.
7. Provide the project details:
   a. Type `Building Example` in the **Project** field.
   b. Type `Demo` in the **Client** field.
   c. Type your name or initials in the **Engineer** field.
   d. Type 2 in the **No. of Levels** field.
   e. Type 5 (m) in the **Founding Depth** field.

This value is the distance from the top of foundations to the first floor.



8. Click **Create Project**.
   The **New Plan** dialog opens.
9. Enter in the plan details and assign it to the levels:
   a. Type 4 (m) in the **Height of Level Above** field.
   b. Click in the **Assign Levels** drop-down list and then click in the fields for both levels 1 and 2 such that **Plan1** is displayed there.
      This action assigns the typical plan to both levels.
   c. In the **Import Plan** group, select the **Create Plan Graphically** option.
      You will use the graphical interface to create a simple plan in this tutorial.

   Leave the remaining options in the dialog their default values.



10. Click **Create Plan**.
The **Slab Details (Rectangle)** dialog opens.

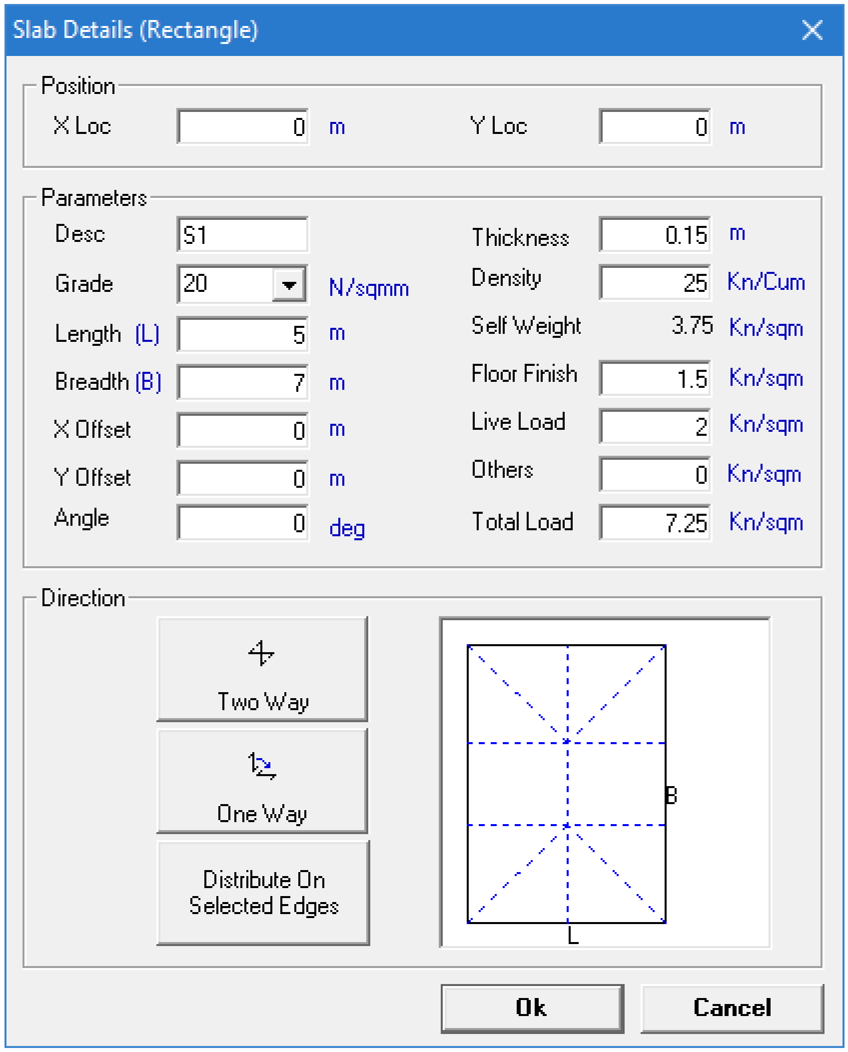
Proceed to the next set of steps to generate the slab.

EX. Create the slab layout

This procedure continues from the previous steps in which you created the model and plan details in building planner.

1. In the **Slab Details (Rectangle)** dialog, type 7 (m) in the **Breadth (B)** field.

   This updates the slab loading direction diagram in the dialog. Leave all other fields and options their defaults.



2. Click **OK**.
   A single slab section is added to the graphical view.
3. Select **Slab** > **Create Slab Rectangle**.
4. Click inside the existing slab near the top, left corner.
   A rectangular window is now pinned to the nearest corner and “rubber banded” to your mouse pointer.
5. Click at a point near where you want the opposite corner of the slab section placed.

   This should roughly be in line with the existing slab edge.

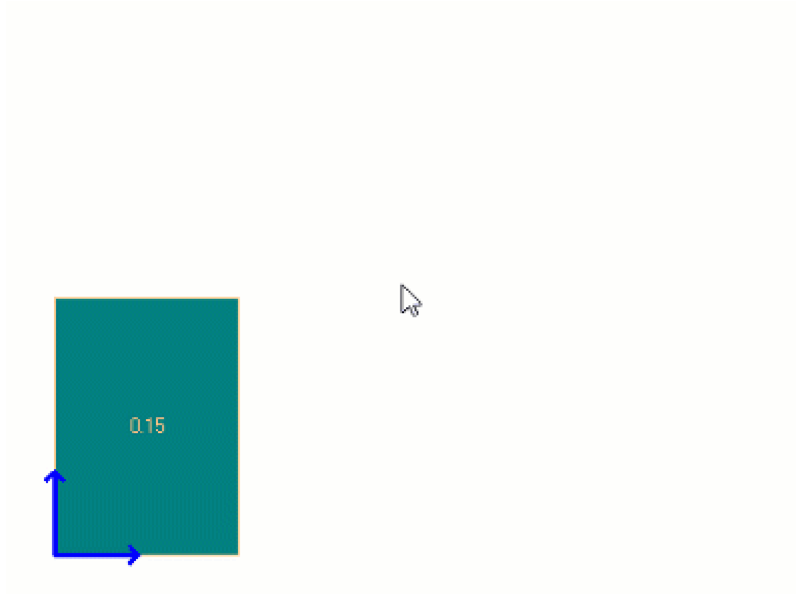
   The **Slab Details (Rectangle)** dialog opens. The same parameters from your first slab are entered.
6. Either:

   click **OK**

   or

   press **<Enter>**

   The new slab section is added adjacent to the existing slab edge.
7. Repeat steps 5 and 6 to complete the 2 bay by 3 bay slab area.



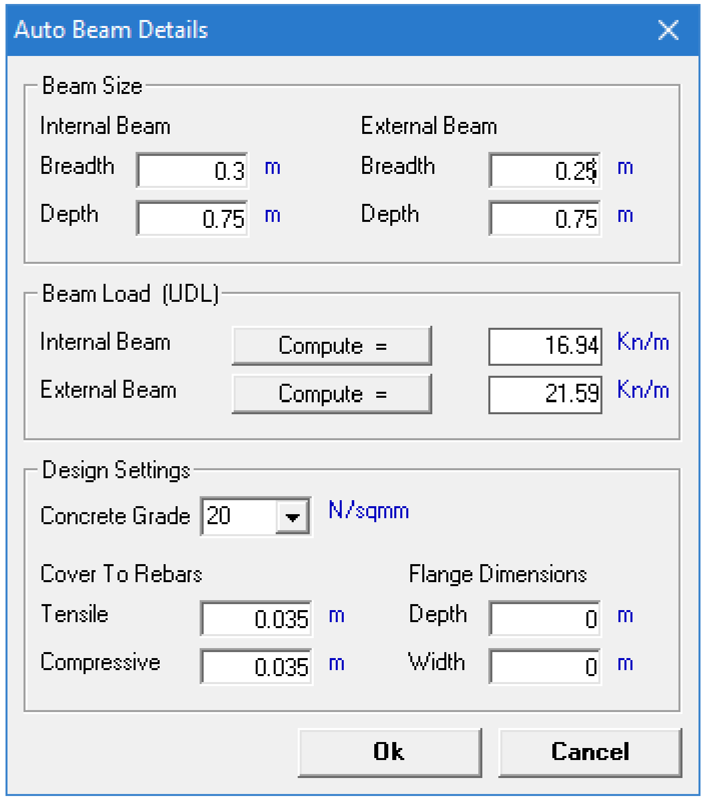
Tip: You do not need to reselect the tool from the menu as it remains active after you create a slab.

Click the **Save** tool to save your progress so far.

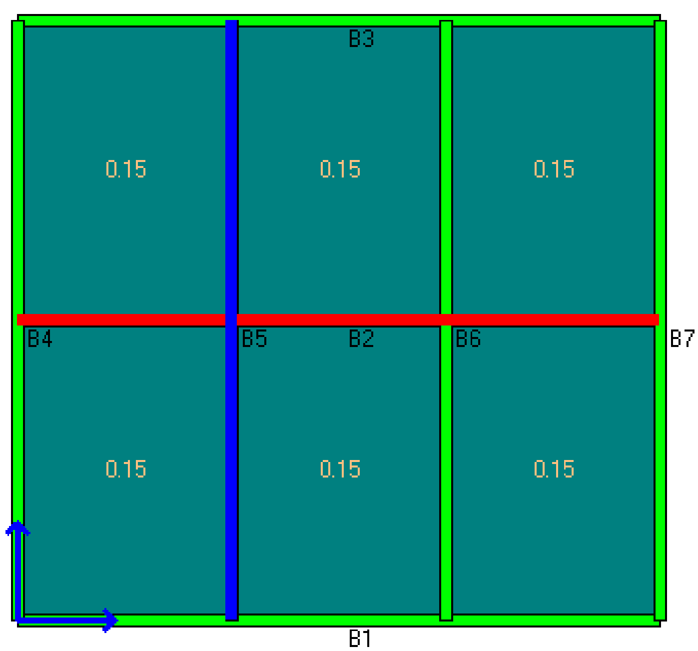
EX. Add the beams and columns

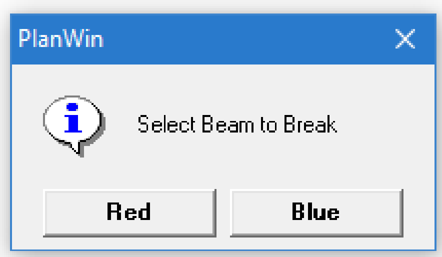
1. Select **Beam** > **Auto Beam**.
   The **Auto Beam Detail** dialog opens.
2. Customize the beam properties to use:
   a. Type 0.30 (m) fro the **Internal Beam Breadth**.
   b. Type 0.25 (m) fro the **External Beam Breadth.**

   Leave the remaining parameters their default values.



3. Click **OK**.
   The beams are automatically added around the slab panel edges.
4. If the Select Beam to Break dialog opens, break the beams spanning 7 m.
   That is, such that the girder spanning horizontally on the screen is continuous.



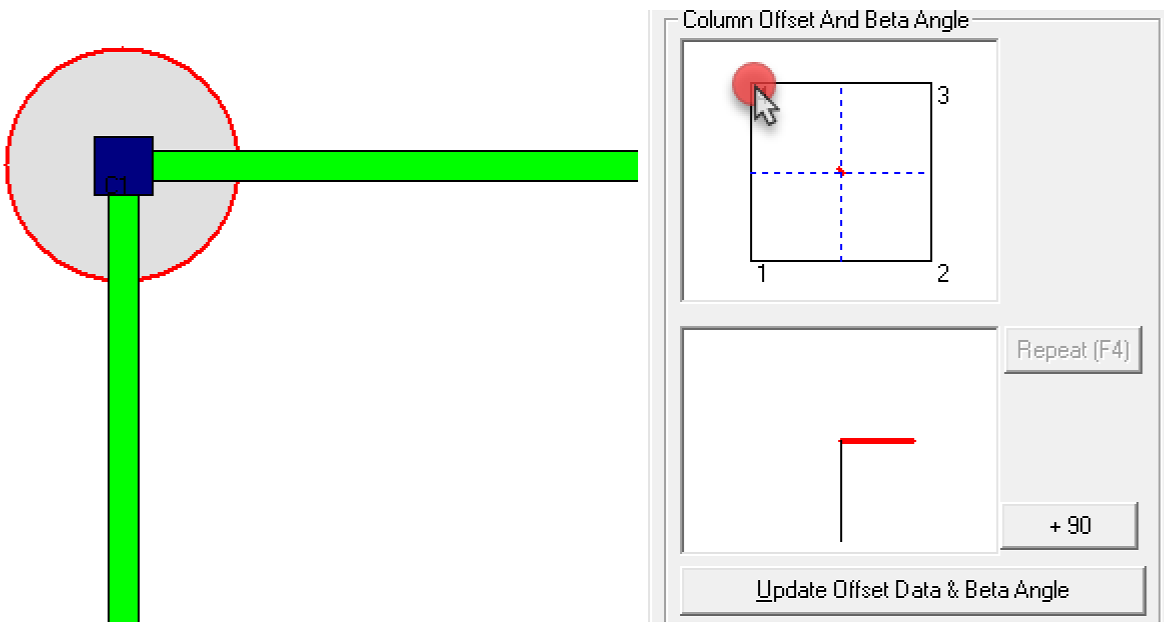


5. Repeat step 4 if necessary.
6. Select **Column** > **Auto Column**.
   Square columns are automatically added at each beam intersection.

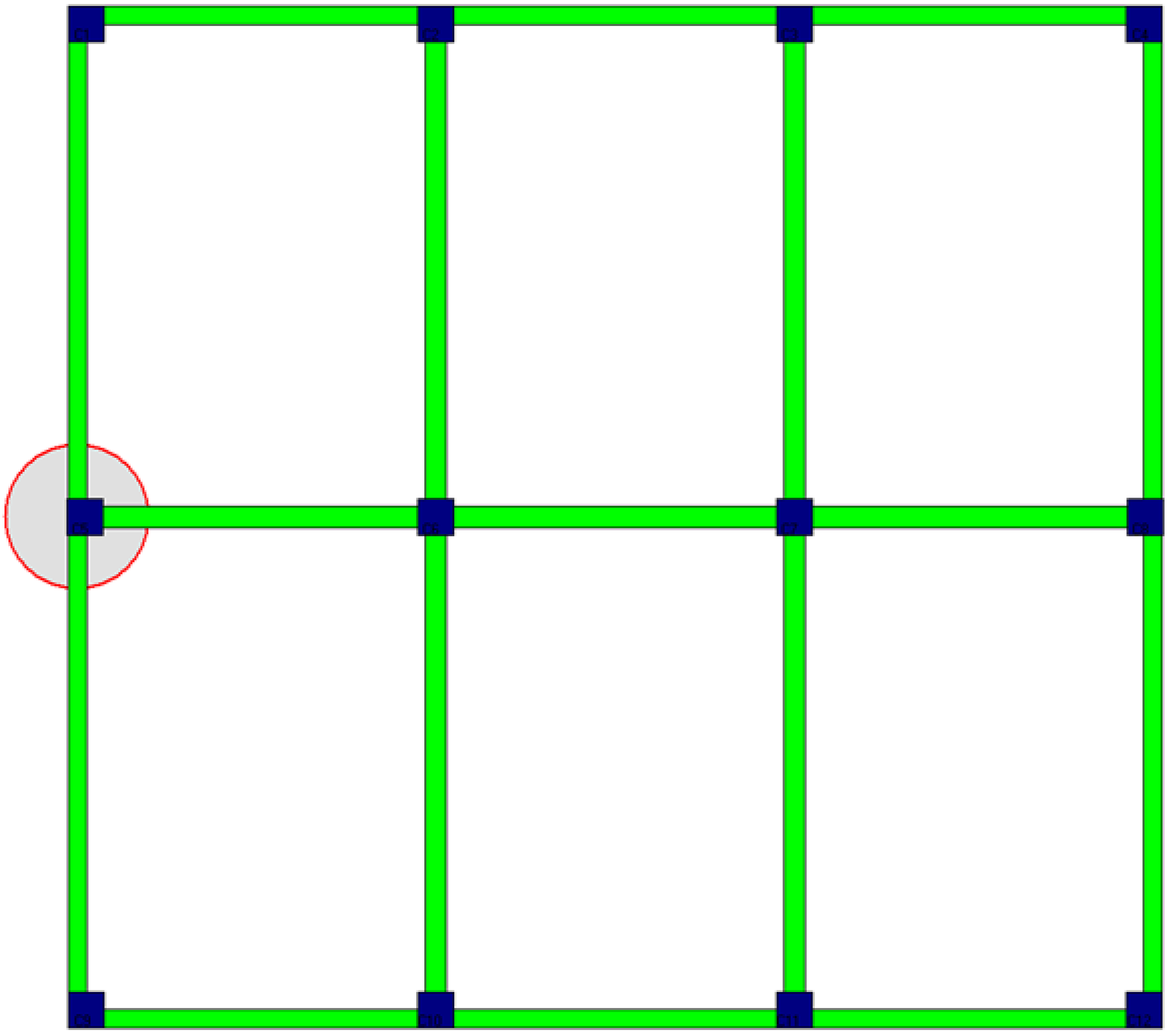
EX. Check for errors and align columns

1. Perform error checks:
   a. Select **Analysis** > **Slab Error Check**.
      This should result in no warnings or messages.
   b. Select **Analysis** > **Beam Error Check**.
      This should result in no warnings or messages.
2. Select **Analysis** > **Finalize Plan (with Continuity)**.
   A message dialog confirms the plan is finalized.
3. Select **Workspace** > **3D Frame**.
   If the **3D View** window opens, close this window.

   Note: You are now in the frame workspace, which has different menu options.
4. Select **Assign** > **Column Sizing and Orientation**.
   The **Column Sizing** window opens.
5. Set the column offset so that the faces of the columns align with the faces of the exterior beams:
   a. Click Column C1 to select it (the top, left column).
   b. In the Column Offset And Beta Angle diagram, click corner 4 (the top, left corner).
   c. Click **Update Offset Data & Beta Angle**.
      The column face is aligned to this point on the beams.



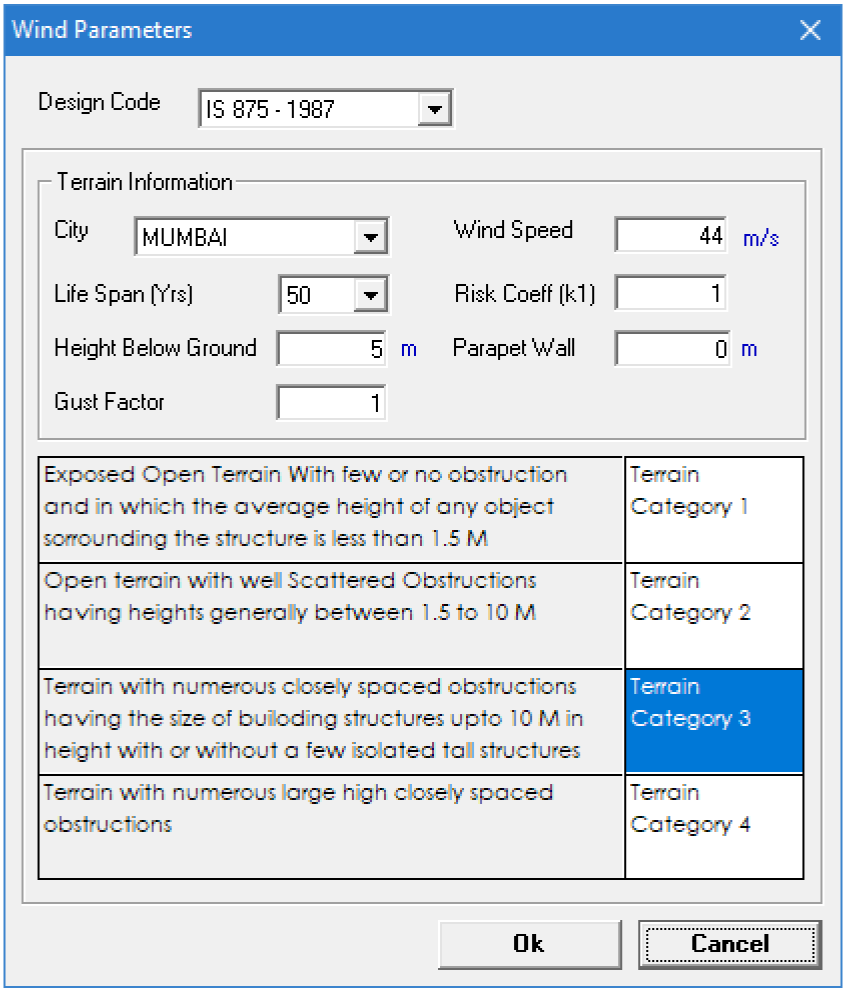
6. Repeat step 6 for each of the edge and corner columns, selecting the corresponding edge or corner for alignment.



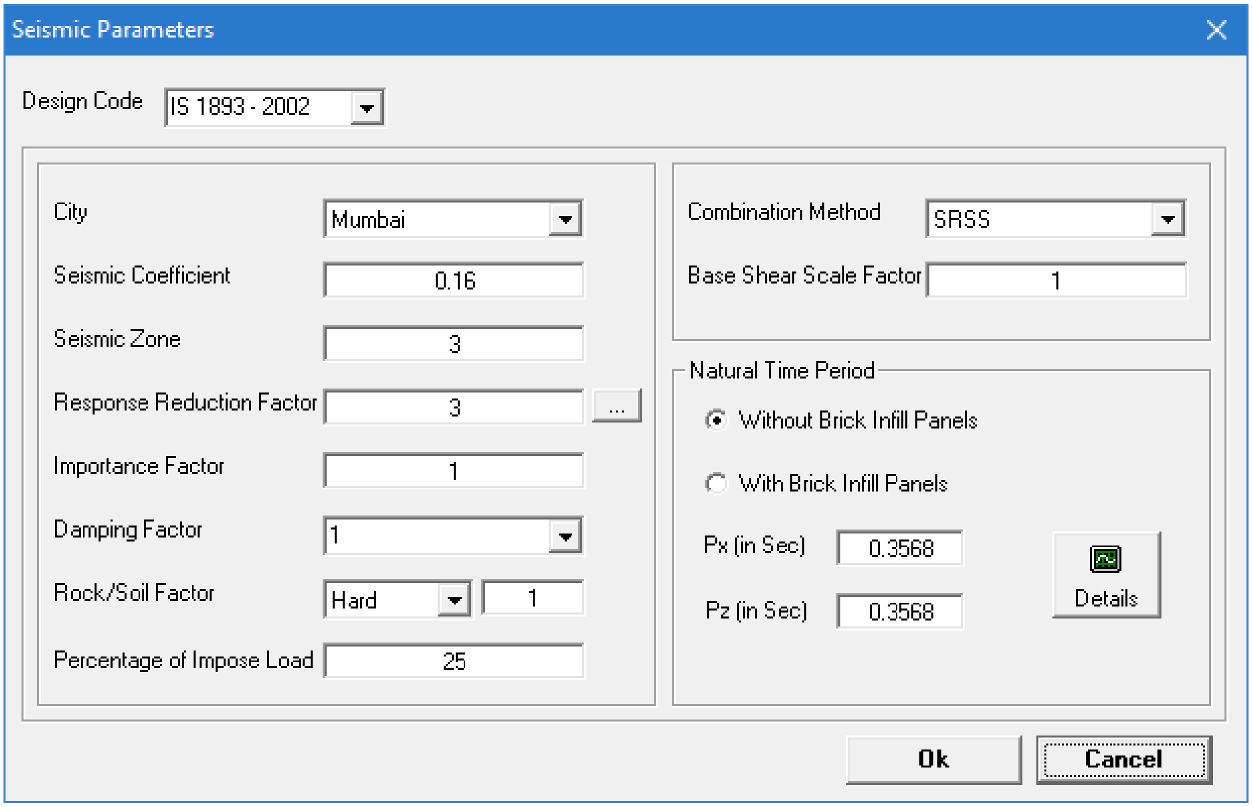
7. Close the **Column Sizing** window.

EX. Assign lateral load data and generate the analysis model

1. Select **Assign** > **Wind Parameters**.
   The **Wind Parameters** dialog opens. Here you can select the appropriate Indian or American code for wind loads and assign parameters. For this example, you will accept the default parameters.



2. Select **IS 875-1987** from the **Design Code** drop-down and then click **OK.**
3. Select **Assign** > **Seismic Parameters**.
   The **Seismic Parameters** dialog opens. Here you can select the appropriate Indian or American code for wind loads and assign parameters. For this example, you will accept the default parameters.



4. Select **Analysis** > **Generate Analysis File**.
   The **Space Frame Generation** dialog opens.
5. Click **Generate**.
   A warning dialog asks to overwrite the existing file, click **Yes**.

The STAAD Building Planner window closes and the analysis model opens in STAAD.Pro.

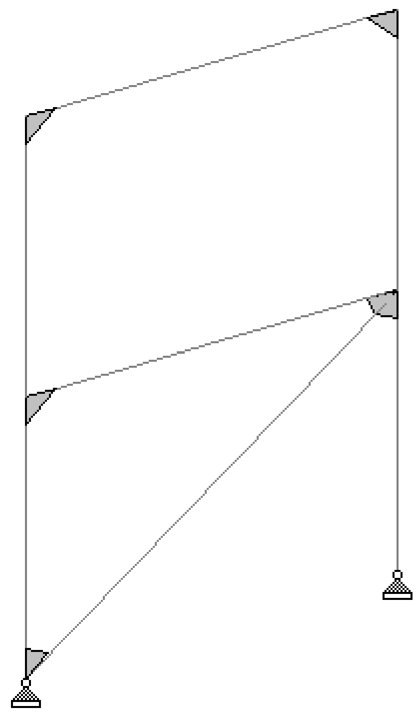
All gravity loads, lateral loads, load combinations, and analysis commands have been added to the STAAD.Pro model. You can run the analysis in the Analytical Modeling workflow with no further actions necessary.

EX. Steel Design Examples

EX. Connection Design Example

This short tutorial presents an overview of design and detailing several steel connections.

This example uses US Example no. 7, which is included with the installation of STAAD.Pro.



EX. To start the Connection Design workflow

To open the example file in the Connection Design workflow, use the following procedure.

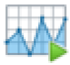
Using the Connection Design workflow requires a RAM Connection license. This license can incur additional cost to your organization.

1.
   a. Close any open model files in STAAD.Pro.
   b. On the Start page, check **RAM Connection** in the **Additional Licenses** panel.
      The **Additional License Selected** message opens to indicate that this can incur additional cost.
   c. Click Yes to activate this license for STAAD.Pro.
2. Open the file example file `US.7. Modeling Offset Connections in a Frame.std` in the **Analytical Modeling** workflow.
   This file is typically found in the
   `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\`
   directory.
3. (Optional) On the **View** ribbon tab, **3D Rendering** tool in the **Windows** group.

   This view is useful to quickly identify the orientation of the columns with respect to the framing beam members. Here, it is evident that the beams and brace frame into the column flanges (rather than the webs).
4. Either:

   Select the **Run Analysis** tool in the **Analysis** group on the **Analysis and Design** ribbon tab.



or

Press **<CTRL+F5>**

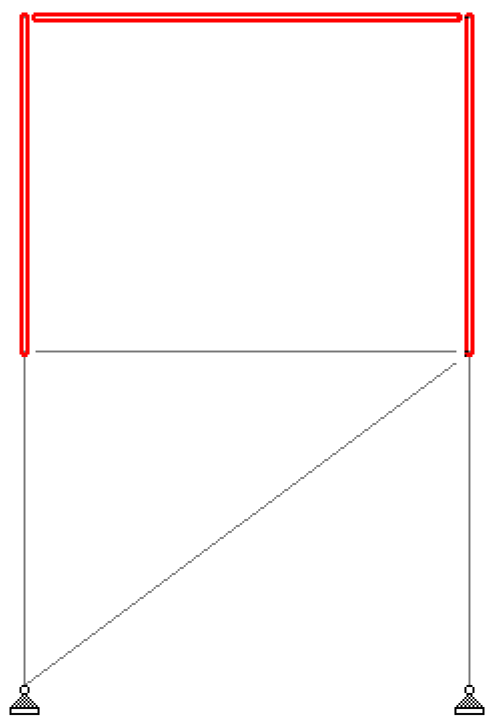
The **STAAD Analysis and Design** dialog opens.

During the analysis (and design, if specified), an output file is generated. This file may contain selected input data items, results and error messages. Optional print specifications can be used to include additional information in the output file.

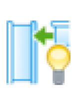
5. Click **Done**.
6. Select the **Connection Design** workflow.
   The **RAM Connection - Validation** dialog opens.
7. Click **Close**.

EX. To design the roof beam to column connections

1. Hold **<Ctrl>** and select both upper columns and the roof beam (members 3, 4, and 6).



2. On the **Connection Design** ribbon tab, select the **Smart Connection** tool in the **Assign Connections** group.

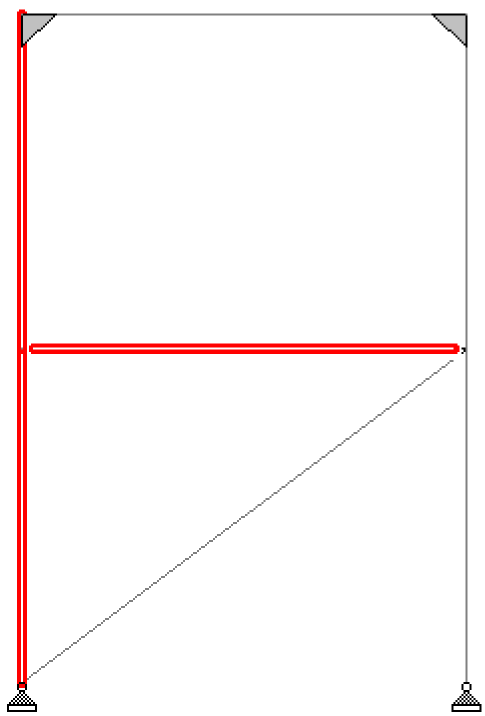


The **Smart Connections** dialog opens.

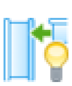
3. Select **AISC 360-10 (ASD)** as the **Design Code**.
   Leave all other options unchecked.
4. Select **Clip Angle BCF (Clip Angle, Beam-Column(Flange))** from the connection types drop-down list.
5. Double-click **DA BCF All bolted** in the **Available** list.
   It is added to the **Selected** list.
6. Click **OK**.
   The RAM Connection - Validation dialog opens to display the design results overview.
7. Click **Close**.
   The connection design is added to the **RAM Connection Input** table.

EX. To design the floor beam to column connection

1. Hold **<Ctrl>** and select both the left column sections and the floor beam (members 1, 3, and 5).



2. On the **Connection Design** ribbon tab, select the **Smart Connection** tool in the **Assign Connections** group.



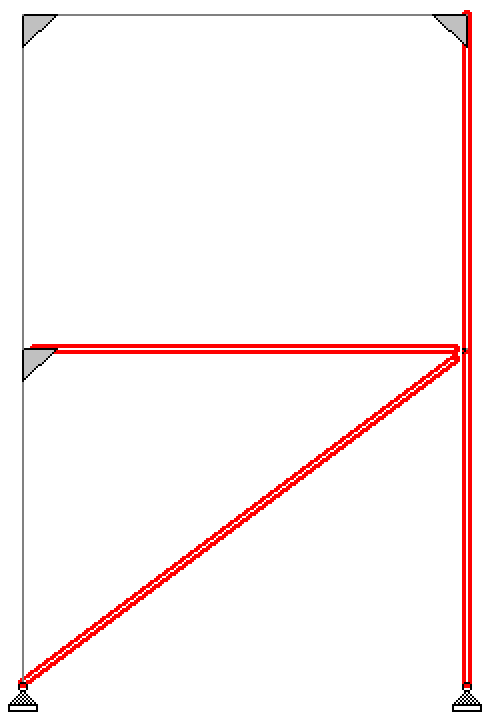
   The **Smart Connections** dialog opens.
3. Select **AISC 360-10 (ASD)** as the **Design Code**.
   Leave all other options unchecked.

4. Select **Unstiffened Seated BCF (UnStiffened Seated, Beam-Column(Flange))** from the connection drop-down list.
5. Double-click **BCF Bolted** in the **Available** list.
   It is added to the **Selected** list.
6. Click **OK**.
   The RAM Connection - Validation dialog opens to display the design results overview.
7. Click **Close**.
   The connection design is added to the **RAM Connection Input** table.

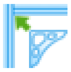
EX. To design the brace gusset plate connection

So far, you have selected members which form a joint for design. In this procedure, you will use some of the quick selection tools in the program which can be used to select all joints of a similar type.

1. On the **Connection Design** ribbon tab, select the **Select Joints** > **Select all Column-Beam-Brace Joints** tool in the **Assign Connections** group.
   Members 1, 2, 4, 5, and 7 are selected.
2. Hold **<Ctrl>** and then click the lower, left column section to remove it from the selection set.
   The lower end of the brace ends at a support so a separate connection design incorporating the base plate is needed.



3. On the **Connection Design** ribbon tab, select the **Gusset Connections** tool in the **Assign Connections** group.

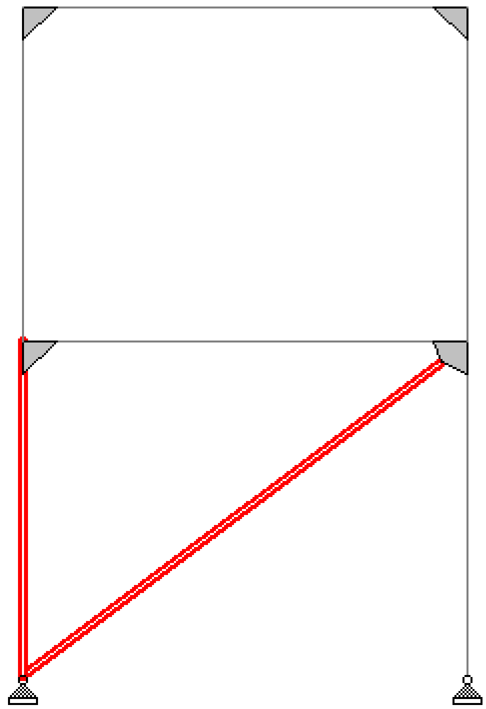


   The **Gusset Connections** dialog opens.

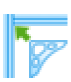
4. Select **AISC 360-10 (ASD)** as the **Design Code**.
   Leave all other options unchecked.
5. Select **Gusset Plate CBB (Gusset Plate, Column-Beam-Brace)** from the connection types drop-down list.
6. Double-click the **CBB_DW_CBF** entry in the Available list.
   This is a Column-Beam-Brace connection, directly welded, at the column and beam flanges.
   It is added to the **Selected** list.
7. Click **OK**.
   The RAM Connection - Validation dialog opens to display the design results overview.
8. Click **Close**.
   The connection design is added to the **RAM Connection Input** table.

EX. To design the gusset base plate connection

1. Hold **<Ctrl>** and select the lower-left column segment and the brace (members 1 and 7).



2. On the **Connection Design** ribbon tab, select the **Gusset Connections** tool in the **Assign Connections** group.

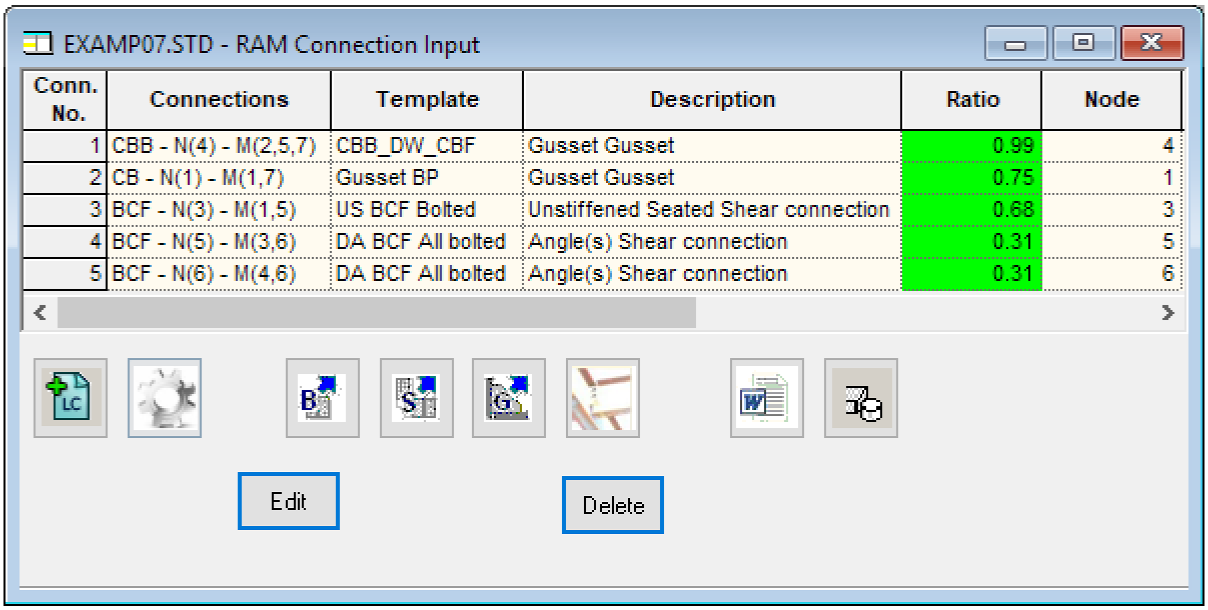


   The **Gusset Connections** dialog opens.
3. Select **AISC 360-10 (ASD)** as the **Design Code**.
   Leave all other options unchecked.
4. Select **Gusset Base Plate** from the connection types drop-down list.
5. Click **[>>]** to include all options in the **Selected** list.
   In this case, there is only a single **Gusset BP** type.

6. Click **OK**.
   The RAM Connection - Validation dialog opens to display the design results overview.
7. Click **Close**.
   The connection design is added to the **RAM Connection Input** table.

EX. To change detailing of a connection

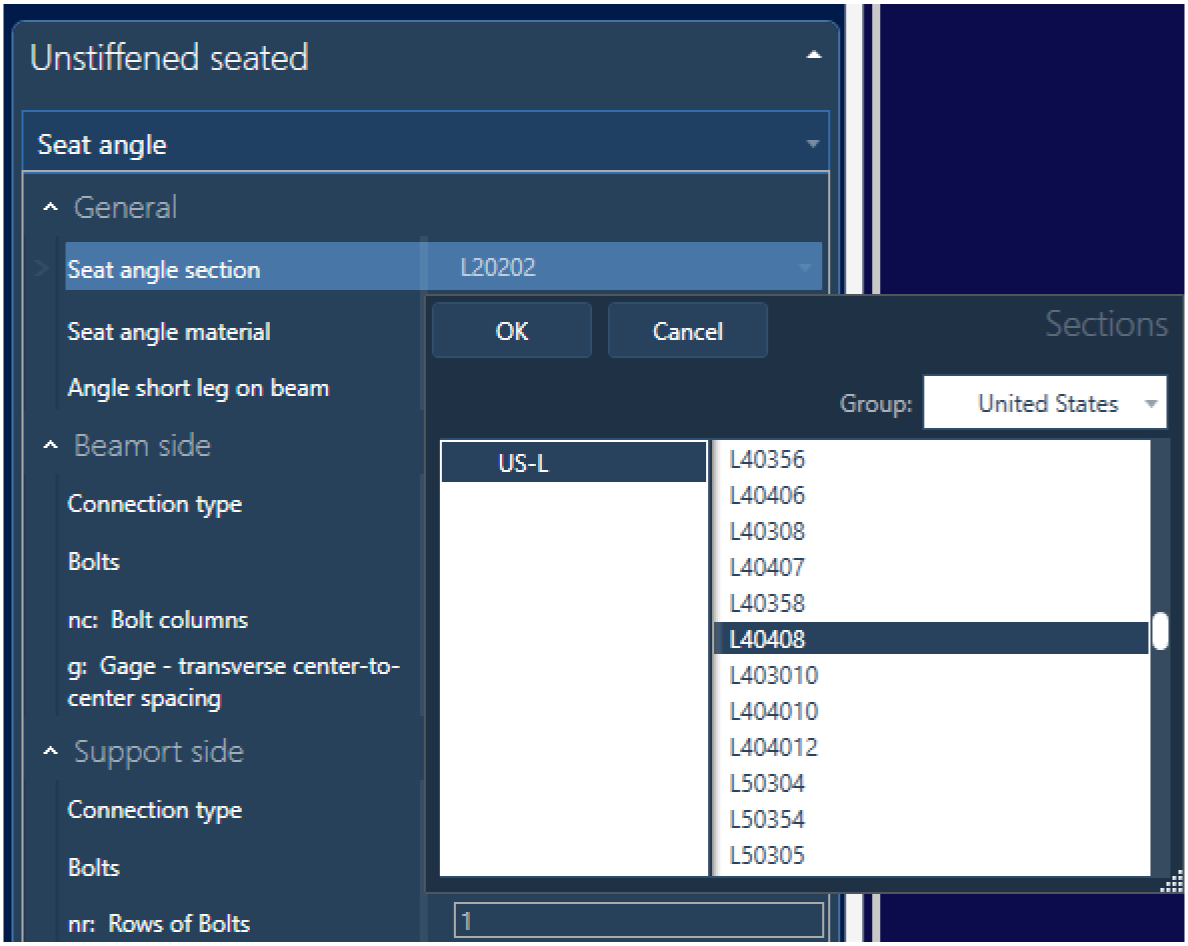
1. In the **RAM Connection Input** table, double-click the row for connection number 3 (connection BCF - N(3) - M(1,5)).



Conn. No.	Connections	Template	Description	Ratio	Node
1	CBB - N(4) - M(2,5,7)	CBB_DW_CBF	Gusset Gusset	0.99	4
2	CB - N(1) - M(1,7)	Gusset BP	Gusset Gusset	0.75	1
3	BCF - N(3) - M(1,5)	US BCF Bolted	Unstiffened Seated Shear connection	0.68	3
4	BCF - N(5) - M(3,6)	DA BCF All bolted	Angle(s) Shear connection	0.31	5
5	BCF - N(6) - M(4,6)	DA BCF All bolted	Angle(s) Shear connection	0.31	6

> **Tip:** You can also use the **Joint Cursor** tool in the **Connection Design** ribbon tab to double-click on a joint in the View window.

   The **Connection Pad** window opens. This window contains the details of the current connection design as well as a 3D rendering.
2. In the right details pane, select the Seat angle in the Unstiffened seated section.
   The default design is a relatively small seat angle and larger top angle.
3. Select an **L40408** for the **Seat angle section** and then click **OK**.



The design, connection detailing, and drawing update immediately.

4. Click the **Results** tool to display the design checks report.
5. In the Report window, select the show Formulas tool.
   The report updates with the calculations for each check.

> **Tip:** You can export either format to a Microsoft Office Word document. Including formulas significantly increases the length of the report.

6. Select the **Exit** tool to close the report window.
7. Select the **Save** tool.
8. Click the **[X]** to close the **Connection Pad** window.
   The design is updated in the **RAM Connection Input** table.

EX. To generate a report of the connection designs

1. On the **Connection Design** ribbon tab, select the **Connection Report** tool in the **Reports** group.

The **RAM Report Export** dialog opens.

2. Select the **Grouped by Identical Connections** options.
   The two BCF connections for the roof beam are grouped together.
3. Click **Select All**.
   Leave the other options their defaults: include Data Report, Results Report with Formula, and in a merged reports format.
4. Click **OK**.
   The report is generated and saved to a Microsoft Office Word document (`EXAMP07_report.doc`) in the same folder as the STAAD input file.
5. Click **OK**.

Tip: You can also add a summary table of the connection design results to your STAAD report.

EX. Connection Tags Example

This example demonstrates applying and checking connection tags.

Connection Tag Data

Use the **Copy to clipboard** button below to copy and paste this connection data into an empty text file and then save it as `Connection_Tag_Example.xml` to a convenient location.

Tip: The default location for connection tag data in STAAD.Pro can be found at `%LOCALAPPDATA%\Bentley\Engineering\STAAD.Pro CONNECT Edition\Default\Plugins \ConnectionTagLink`.

```
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile>
    <FileVersion value="1.0" />
    <Categories>
        <Category CategoryName="MOMENT">
            <CategoryDesc>End Moment Connection</CategoryDesc>
        </Category>
        <Category CategoryName="SHEAR">
            <CategoryDesc>Single Shear Connection</CategoryDesc>
        </Category>
    </Categories>
    <Equations>
        <Equation EquationID="Eq1" Equation="(abs([Mz])+([alpha]*abs([Fx])))/[Mz.cap]"
Condition="LT" Limit="1.0" />
        <Equation EquationID="Eq2" Equation="abs([Fy])/[Fy.cap]" Condition="LT"
Limit="1.0" />
        <Equation EquationID="Eq3" Equation="(abs([Fy])+([alpha]*abs([Fx])))/[Fy.cap]"
Condition="LT" Limit="1.0" />
    </Equations>
    <Tags>
        <Tag TagName="EM" CategoryName="MOMENT">
            <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
            <Capacities UnitSystem="IMPERIAL">
                <Beam Name="W12x26">
                    <BeamOrCol Name="W14x90" Mz.cap="700" Fy.cap="65" Fx.cap=""
alpha="0.10" />
                </Beam>
            </Capacities>
            <Checks>
```

```
                <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
                <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
            </Checks>
        </Tag>
        <Tag TagName="EMH" CategoryName="MOMENT">
            <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="0" MZ="0" />
            <Capacities UnitSystem="IMPERIAL">
                <Beam Name="W12x26">
                    <BeamOrCol Name="W14x90" Mz.cap="700" Fy.cap="65" Fx.cap=""
alpha="0.10" />
                </Beam>
            </Capacities>
            <Checks>
                <Check Type="MOMENT" Desc="Moment Check" EquationID="Eq1" />
                <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq2" />
            </Checks>
        </Tag>
        <Tag TagName="SS" CategoryName="SHEAR">
            <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="1" MZ="1" />
            <Capacities UnitSystem="IMPERIAL">
                <Beam  Name="W12x26">
                    <BeamOrCol Name="Column" Mz.cap="" Fy.cap="65" Fx.cap=""
alpha="1" />
                    <BeamOrCol Name="W14x90" Mz.cap="" Fy.cap="65" Fx.cap=""
alpha="0.75" />
                </Beam>
            </Capacities>
            <Checks>
                <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq3" />
            </Checks>
        </Tag>
        <Tag TagName="DS" CategoryName="SHEAR">
            <EndRelease FX="0" FY="0" FZ="0" MX="0" MY="1" MZ="1" />
            <Capacities UnitSystem="IMPERIAL">
                <Beam  Name="W12x26">
                    <BeamOrCol Name="Column" Mz.cap="" Fy.cap="65" Fx.cap=""
alpha="1" />
                    <BeamOrCol Name="W14x90" Mz.cap="" Fy.cap="65" Fx.cap=""
alpha="0.75" />
                </Beam>
            </Capacities>
            <Checks>
                <Check Type="SHEAR" Desc="Shear Check" EquationID="Eq3" />
            </Checks>
        </Tag>
    </Tags>
</ConnectionTagFile>
```


Note: This is an extremely short connection data example for the purposes of demonstration. A typical connection data file would contain a very large number of connection capacities for checking.

EX. To open the example and load example connection data

Copy and save the example connection tag data presented in the previous topic before proceeding.

1. Open the file example file `US.7. Modeling Offset Connections in a Frame.std` in the **Analytical Modeling** workflow.

This file is typically found in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models\US\` directory.

2. On the **Utilities** ribbon tab, select the **Connection Tags** > **Open Tab Specifications** tool in the **Tools** group.

   The **Select Connection Tag File** dialog opens.
3. Navigate to the `Connection_Tags_Example.xml` file and the click **Open**.

   This file can be found where you saved it in the previous topic.

EX. To assign the connection tag types to beam ends

1. Select both beams 5 and 6.

   Tip: Press **<Shift+B>** to display the beam numbers.
2. On the **Beam Tools** ribbon tab, select the **Assign** tool in the **Connection Tags** group.

   The **Assign Connection Tags** and **New Connection Tag** dialogs open.
3. In the **New Connection Tag** dialog, select the options to generate the tags:
   a. Select **EM** from the **Select Tags** drop-down list.
   b. Select both **Start** and **End** options for the **Location**.
   c. Click **Assign**.

   The dialog closes and the two tags are added to the **Assign Connection Tags** dialog, assigned to the members.

   Note: There are two tags generated, one for the beam start and another for the beam end.
4. Click **Close** on the **Assign Connection Tags** dialog.

EX. To check the connection tags

1. On the Utilities ribbon tab, select the **Connection Tags** > **Check Tags** tool in the **Tools** group.

   The **Assign Connection Tags** and **Check Connection Tags** dialogs open.
2. In the **Check Connection Tags** dialog:
   a. Check the option for the **1 - WIND LOAD** load case.
   b. Click **Check**.

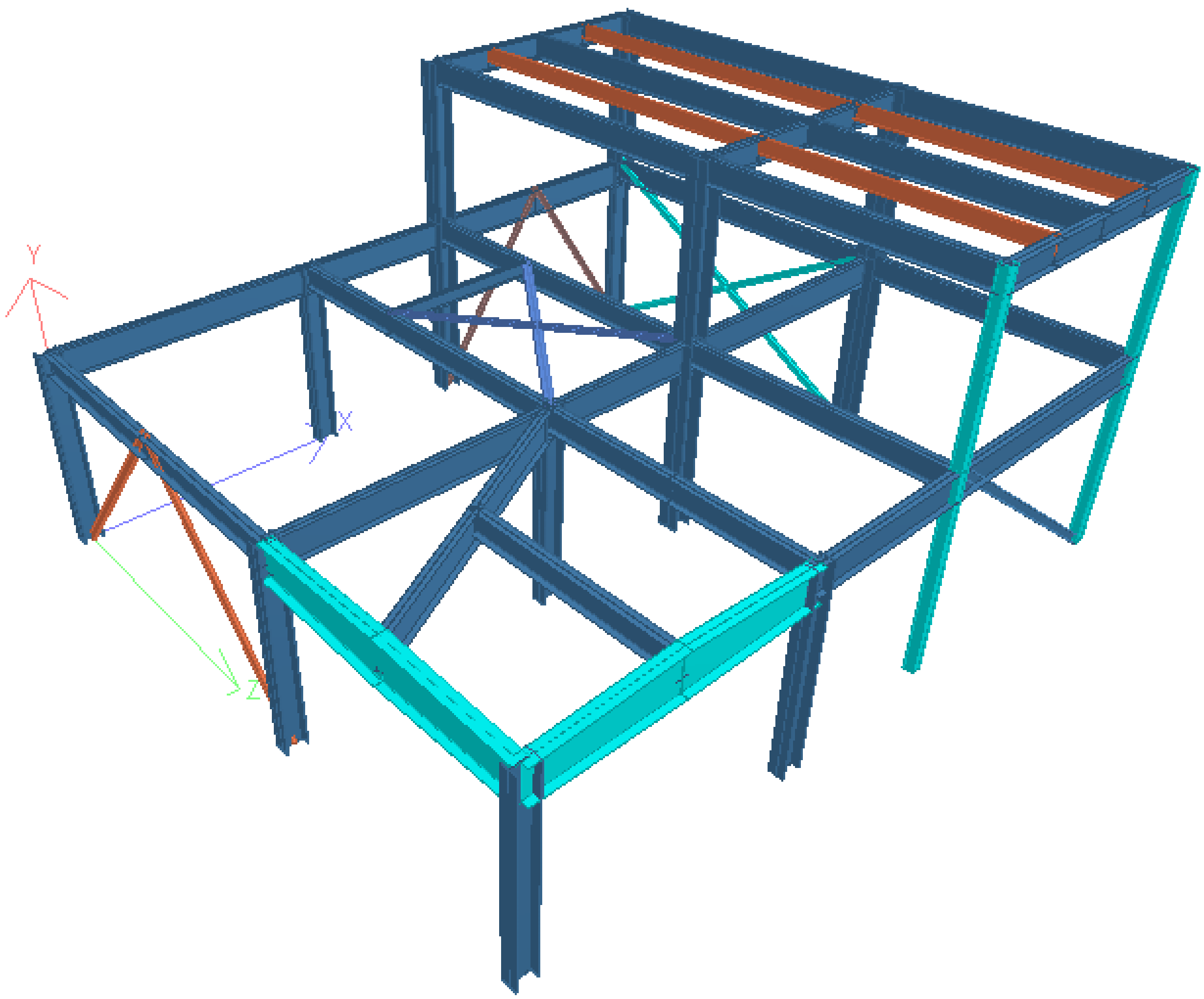
   The **Connection Tag Check Results** dialog opens.

3. (Optional) Click the **Print** tool to open the print Preview window, where you can save a copy of the connection tag results table.

EX. Connection Tags Example 02

This example demonstrates applying connection tags using RAM Connection templates for mapping connections designed per AISC code to connection tags.

An industrial steel frame structure consists of several member types, including W shapes, tubes, and channels.



EX. Data Files

Save the following data as files for use with this example.

Save the following file contents to a convenient location on your computer.

1. Copy and save the following data to a file named `ConnectionTagsExampl02.std`:

```
STAAD SPACE Industrial Structure
START JOB INFORMATION
ENGINEER DATE 03-Mar-21
JOB NAME Connection Tags Example 2
JOB CLIENT Integrated Connection Tags Workflow
JOB REV Rev 0
JOB COMMENT Connection Tags Pre-Assigned
ENGINEER NAME KR
CHECKER NAME KR
END JOB INFORMATION
INPUT WIDTH 79
UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 0 21; 3 16 0 0; 4 16 0 21; 5 26 0 0; 6 26 0 21; 7 42 0 0;
8 42 0 21; 9 0 12 0; 10 0 12 21; 11 16 12 0; 12 16 12 21; 13 26 12 0;
14 26 12 21; 15 42 12 0; 16 42 12 21; 21 26 22 0; 22 26 22 21; 23 42 22 0;
24 42 22 21; 25 16 12 8; 26 26 12 8; 27 34 12 0; 28 42 6 10.5; 29 21 12 14.5;
30 0 12 10.5; 31 30 22 0; 32 34 22 0; 33 38 22 0; 34 30 22 21; 35 34 22 21;
36 38 22 21; 37 0 0 37; 38 16 0 37; 39 26 0 37; 40 42 0 37; 41 0 12 37;
42 16 12 37; 43 26 12 37; 44 42 12 37; 45 26 22 37; 46 42 22 37; 47 30 22 37;
48 34 22 37; 49 38 22 37; 50 0 12 29; 51 16 12 29; 52 8 12 37; 53 8 12 25;
54 26 17 37; 55 42 17 37;
MEMBER INCIDENCES
1 1 9; 2 2 10; 3 3 11; 4 4 12; 5 5 13; 6 6 14; 7 7 15; 8 8 16; 13 13 21;
14 14 22; 15 15 23; 16 16 24; 17 9 11; 18 11 13; 19 13 27; 20 10 12; 21 12 14;
22 14 16; 23 9 30; 24 11 25; 25 13 26; 26 15 16; 27 25 12; 28 26 14; 29 25 26;
30 29 25; 31 29 26; 32 7 28; 33 8 28; 34 27 15; 35 5 27; 36 7 27; 37 28 16;
38 28 15; 39 21 22; 40 23 24; 41 21 31; 42 22 34; 43 29 14; 44 29 12; 45 30 10;
46 1 30; 47 2 30; 48 31 32; 49 32 33; 50 33 23; 51 34 35; 52 35 36; 53 36 24;
54 31 34; 55 32 35; 56 33 36; 57 10 50; 58 12 51; 59 14 43; 60 16 44; 61 22 45;
62 24 46; 63 34 47; 64 35 48; 65 36 49; 66 37 41; 67 42 38; 68 39 43; 69 40 44;
70 43 54; 71 44 55; 72 41 52; 73 42 43; 74 43 44; 75 45 47; 76 47 48; 77 48 49;
78 49 46; 79 41 50; 80 51 42; 81 12 53; 82 42 52; 83 53 50; 84 53 52; 85 43 40;
86 54 45; 87 55 46;
START GROUP DEFINITION
MEMBER
_BEAMS 23 TO 29 39 40 45 54 TO 65 79 80 84
_GIRDERS 17 TO 22 34 41 42 48 TO 53 72 TO 78 81 TO 83
_COLUMNS 1 TO 8 13 TO 16 66 TO 71 86 87
_BRACES 30 TO 33 35 TO 38 43 44 46 47 85
_ALL-BEAMS 1 TO 8 13 TO 29 34 39 TO 42 45 48 TO 84 86 87
FLOOR
_NO-H-BRACES 17 TO 29 34 45 57 TO 60 72 TO 74 79 TO 84
END GROUP DEFINITION
*
SUPPORTS
1 TO 6 37 TO 39 PINNED
7 8 40 FIXED
MEMBER RELEASE
23 TO 26 29 39 40 54 TO 56 61 TO 65 81 83 84 START MY MZ
26 TO 28 39 40 45 54 TO 56 61 TO 65 81 83 84 END MY MZ
74 START MY MZ
74 END MY MZ
29 END MY MZ
MEMBER TRUSS
30 TO 33 35 TO 38 43 44 46 47 85
*
```

```
DEFINE MATERIAL START
ISOTROPIC STEEL_50_KSI
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH FY 7200 FU 8928 RY 1.5 RT 1.2
ISOTROPIC STEEL
E 4.176e+06
POISSON 0.3
DENSITY 0.489024
ALPHA 6.5e-06
DAMP 0.03
TYPE STEEL
STRENGTH RY 1.5 RT 1.2
END DEFINE MATERIAL
CONSTANTS
BETA 180 MEMB 31 44 46 54 63
MATERIAL STEEL_50_KSI ALL
*
START USER TABLE
TABLE 1
UNIT FEET KIP
ISECTION
PG-1
2 0.045 2.5 0.75 0.0625 0.75 0.0625 0.084 0.094 3.25
END
*
MEMBER PROPERTY AMERICAN
1 TO 4 66 67 TABLE ST W14X74
5 6 8 13 TO 16 TABLE ST W14X99
7 23 TO 28 39 TO 42 45 48 TO 53 55 58 TO 62 64 75 TO 78 80 84 TABLE ST W16X36
17 TO 22 34 73 74 81 83 TABLE ST W21X44
35 36 TABLE ST HSSP4.500X0.375
32 33 37 38 TABLE ST HSST4X4X0.375
54 56 63 65 TABLE ST C10X20
68 TO 71 86 87 TABLE ST HSST7X7X0.625
29 TABLE ST W10X19
30 31 43 44 TABLE T W6X20
85 TABLE ST W6X12
* ABOVE MEMBERS INITIALLY 4x2x0.375 Double Angles *
46 47 TABLE ST C6X10
*
MEMBER PROPERTY
57 72 79 82 UPTABLE 1 PG-1
*
MEMBER OFFSET
17 TO 22 34 73 74 81 83 START 0 -0.875 0
17 TO 22 34 73 74 81 TO 83 END 0 -0.875 0
23 TO 28 39 TO 42 45 48 TO 53 55 58 TO 62 64 75 TO 78 84 START 0 -0.65 0
23 TO 28 39 TO 42 45 48 TO 53 55 59 TO 62 64 75 TO 78 80 84 END 0 -0.65 0
29 54 56 63 65 START 0 -0.42 0
29 54 56 63 65 END 0 -0.42 0
58 END 0 -0.65 0
80 START 0 -0.65 0
72 END 0 -0.875 0
```

```
57 72 79 82 START 0 -1 0
57 79 END 0 -1 0
*
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Mass  TITLE STRUCTURAL MASS
SELFWEIGHT X 1
SELFWEIGHT Y 1
SELFWEIGHT Z 1
ONEWAY LOAD
YRANGE 11 13 ONE 0.1 GX
YRANGE 11 13 ONE 0.1 GY
YRANGE 11 13 ONE 0.1 GZ
END DEFINE REFERENCE LOADS
FLOOR DIAPHRAGM
DIA 1 TYPE RIG HEI 12 JOINT 9 TO 16 25 TO 27
DIA 2 TYPE RIG HEI 22 JOINT 21 TO 24
*
DEFINE WIND LOAD
TYPE 1 ASCE-7 WIND LOAD
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2016:PARAMS 105.000 MPH 0.000 FT 4 2 2 0 0.000 FT 0.000 FT 0.000 FT 1 -
0 30.000 FT 20.000 FT 0.500 FT 0.300 2.000 0.010 0 0 0 0 0 0.982 1.000 1.000 -
0.850 1.000 0 0 0 0 0.888 1.600 0.000
!> END GENERATED DATA BLOCK
INT 0.0289345 0.0289345 0.0293895 0.0298195 0.0302274 0.0306157 0.0309863 -
0.031341 0.0316813 0.0320084 0.0323234 0.0326273 0.032921 0.0332051 -
0.0334805 HEIG 0 15 16.1539 17.3077 18.4615 19.6154 20.7692 21.9231 23.0769 -
24.2308 25.3846 26.5385 27.6923 28.8462 30
LOAD 101 LOADTYPE Dead  TITLE DEAD LOAD
SELFWEIGHT Y -1
ONEWAY LOAD
_NO-H-BRACES ONE -0.1 GY
FLOOR LOAD
YRANGE 21 23 FLOAD -0.025 GY
LOAD 102 LOADTYPE Live  TITLE LIVE LOAD
FLOOR LOAD
YRANGE 21 23 FLOAD -0.05 GY
ONEWAY LOAD
_NO-H-BRACES ONE -0.035 GY
JOINT LOAD
10 12 16 41 42 44 TO 46 MZ 2
51 53 FY -6
MEMBER LOAD
83 UNI GY -0.5
LOAD 103 LOADTYPE Wind  TITLE WIND +X
WIND LOAD X 1 TYPE 1 OPEN
JOINT LOAD
26 FX 15
29 54 55 FX 3
LOAD 104 LOADTYPE Wind  TITLE WIND +Z
WIND LOAD Z 1 TYPE 1 OPEN
JOINT LOAD
26 FZ 15
29 54 55 FZ 3
45 46 MX 2
LOAD 1 GENERATED ASCE-7-2016 LRFD 1
REPEAT LOAD
101 1.4
```

```
LOAD 2 GENERATED ASCE-7-2016 LRFD 2
REPEAT LOAD
101 1.2 102 1.6
LOAD 3 GENERATED ASCE-7-2016 LRFD 3
REPEAT LOAD
101 1.2 102 1.0
LOAD 4 GENERATED ASCE-7-2016 LRFD 4
REPEAT LOAD
101 1.2 103 0.5
LOAD 5 GENERATED ASCE-7-2016 LRFD 5
REPEAT LOAD
101 1.2 104 0.5
LOAD 6 GENERATED ASCE-7-2016 LRFD 6
REPEAT LOAD
101 1.2 102 1.0 103 1.0
LOAD 7 GENERATED ASCE-7-2016 LRFD 7
REPEAT LOAD
101 1.2 102 1.0 104 1.0
LOAD 8 GENERATED ASCE-7-2016 LRFD 8
REPEAT LOAD
101 0.9 103 1.0
LOAD 9 GENERATED ASCE-7-2016 LRFD 9
REPEAT LOAD
101 0.9 104 1.0
LOAD 10 GENERATED ASCE-7-2016 LRFD 10
REPEAT LOAD
101 0.9
PDELTA 10 ANALYSIS SMALLDELTA
FINISH
```

2. Copy and save the following data to a file named `CT_AISC-360-16-LRFD.xml`:

Note: This XML template data is specific to RAM Connection templates per the AISC 360-16 LRFD. Other mappings must be used for other editions or codes.

```
<?xml version="1.0" encoding="utf-8" ?>
<ConnectionTagFile>
    <FileVersion value="1.0" />
    <Categories>

        <Category CategoryName="BASIC-COL">
                <CategoryDesc>-Column Connections-</CategoryDesc>
        </Category>
        <Category CategoryName="BASIC-BEAM">
                <CategoryDesc>-Beam Connections -</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-COL">
                <CategoryDesc>-Column Connections-</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-BEAM">
                <CategoryDesc>-Beam Connections-</CategoryDesc>
        </Category>
        <Category CategoryName="BASIC-BCF">
                <CategoryDesc>Beam Column Flange</CategoryDesc>
        </Category>
        <Category CategoryName="BASIC-BCW">
                <CategoryDesc>Beam Column Web</CategoryDesc>
        </Category>
        <Category CategoryName="BASIC-BG">
```

```
                <CategoryDesc>Beam Girder</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-BCF">
                <CategoryDesc>Beam Column Flange</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-BCW">
                <CategoryDesc>Beam Column Web</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-BG">
                <CategoryDesc>Beam Girder</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-CBB">
                <CategoryDesc>Column Beam Brace</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-CVR">
                <CategoryDesc>Chevron Brace</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-VXB">
                <CategoryDesc>Vertical X Brace</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-BP">
                <CategoryDesc>Base Plate</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-HCBB">
                <CategoryDesc>Horizontal CBB</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-HBBB">
                <CategoryDesc>Horizontal BB Brace</CategoryDesc>
        </Category>
        <Category CategoryName="GUSSET-HXB">
                <CategoryDesc>Horizontal X Brace</CategoryDesc>
        </Category>
        <Category CategoryName="SMART-CHB">
                <CategoryDesc>Chord Branches</CategoryDesc>
        </Category>

       </Categories>

<Tags>

        <Tag TagName="Pin-BP" CategoryName="SMART-COL">

   </Tag>
        <Tag TagName="Fix-BP1" CategoryName="SMART-COL">

        </Tag>
        <Tag TagName="Cap-CC1" CategoryName="BASIC-COL">

        </Tag>
        <Tag TagName="SEP-BCF1" CategoryName="BASIC-BCF">

        </Tag>
        <Tag TagName="SEP-BCW1" CategoryName="BASIC-BCW">

        </Tag>
        <Tag TagName="SP-BG1" CategoryName="BASIC-BG">

        </Tag>
```

```
        <Tag TagName="SP-BG2" CategoryName="BASIC-BG">

        </Tag>
            <Tag TagName="MEP-BCW1" CategoryName="BASIC-BCW">

        </Tag>
        <Tag TagName="MEP-BCW1" CategoryName="BASIC-BCW">

    </Tag>



        <Tag TagName="SP-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="SP-BCWS" CategoryName="SMART-BCW">

    </Tag>
        <Tag TagName="SP-BG-S" CategoryName="SMART-BG">

    </Tag>

        <Tag TagName="EP-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="EP-BCWS" CategoryName="SMART-BCW">

    </Tag>
        <Tag TagName="EP-BG-S" CategoryName="SMART-BG">

    </Tag>
        <Tag TagName="EP-CS1" CategoryName="BASIC-COL">

    </Tag>

        <Tag TagName="DA-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="DA-BCWS" CategoryName="SMART-BCW">

    </Tag>
        <Tag TagName="DA-BG-S" CategoryName="SMART-BG">

    </Tag>

        <Tag TagName="ST-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="ST-BCW1" CategoryName="BASIC-BCW">

    </Tag>
        <Tag TagName="ST-BG-S" CategoryName="SMART-BG">

    </Tag>

        <Tag TagName="BP-BCF1" CategoryName="BASIC-BCF">

        </Tag>
```

```
        <Tag TagName="BP-BCW1" CategoryName="BASIC-BCW">

    </Tag>
        <Tag TagName="BP-BG-1" CategoryName="BASIC-BG">

    </Tag>

        <Tag TagName="SS-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="SS-BCWS" CategoryName="SMART-BCW">

    </Tag>
        <Tag TagName="US-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="US-BCWS" CategoryName="SMART-BCW">

    </Tag>

        <Tag TagName="TP-BCF1" CategoryName="BASIC-BCF">

        </Tag>
        <Tag TagName="TP-BCW1" CategoryName="BASIC-BCW">

    </Tag>



        <Tag TagName="FP-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="FP-BCWS" CategoryName="SMART-BCW">

    </Tag>
        <Tag TagName="FP-BG-S" CategoryName="SMART-BG">

    </Tag>

        <Tag TagName="DW-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="DW-BCWS" CategoryName="SMART-BCW">

    </Tag>

        <Tag TagName="MA-BCFS" CategoryName="SMART-BCF">

        </Tag>

        <Tag TagName="FW-BCFS" CategoryName="SMART-BCF">

        </Tag>

        <Tag TagName="ME-BCFS" CategoryName="SMART-BCF">

        </Tag>
        <Tag TagName="ME-BSe" CategoryName="SMART-BEAM">
```

```
    </Tag>
        <Tag TagName="ME-BCW1" CategoryName="BASIC-BCW">

    </Tag>
        <Tag TagName="MK-BCF1" CategoryName="BASIC-BCF">

    </Tag>




        <Tag TagName="DA-CBB" CategoryName="GUSSET-CBB">

    </Tag>
        <Tag TagName="SP-CBB" CategoryName="GUSSET-CBB">

    </Tag>
        <Tag TagName="DW1-CBB" CategoryName="GUSSET-CBB">

    </Tag>
        <Tag TagName="DW2-CBB" CategoryName="GUSSET-CBB">

    </Tag>
        <Tag TagName="G1-CVR" CategoryName="GUSSET-CVR">

    </Tag>
        <Tag TagName="G2-CVR" CategoryName="GUSSET-CVR">

    </Tag>
        <Tag TagName="G1-VXB" CategoryName="GUSSET-VXB">

    </Tag>
        <Tag TagName="GT-BP" CategoryName="GUSSET-BP">

    </Tag>
        <Tag TagName="G1-HCBB" CategoryName="GUSSET-HCBB">

    </Tag>
        <Tag TagName="G1-HBBB" CategoryName="GUSSET-HBBB">

    </Tag>
        <Tag TagName="G1-HXB" CategoryName="GUSSET-HXB">

    </Tag>
        <Tag TagName="TJ-CHB" CategoryName="SMART-CHB">

    </Tag>


</Tags>

    <Tag>
    <RCNXTemplates>



        <RCNXTemplate TagName="SP-BCFS" Code="AISC 360-16 (LRFD)" File="Single
```

```
Plate BCF.con" Template="SP BCF" />
        <RCNXTemplate TagName="SP-BCWS" Code="AISC 360-16 (LRFD)" File="Single
Plate BCW.con" Template="SP BCW" />
        <RCNXTemplate TagName="SP-BG-S" Code="AISC 360-16 (LRFD)" File="Single
Plate BG.con" Template="SP BG" />

        <RCNXTemplate TagName="EP-BCFS" Code="AISC 360-16 (LRFD)" File="Shear End
Plate BCF.con" Template="EP BCF Bolted" />
        <RCNXTemplate TagName="EP-BCWS" Code="AISC 360-16 (LRFD)" File="Shear End
Plate BCW.con" Template="EP BCW Bolted" />
        <RCNXTemplate TagName="EP-BG-S" Code="AISC 360-16 (LRFD)" File="Shear End
Plate BG.con" Template="EP BG Bolted" />

        <RCNXTemplate TagName="DA-BCFS" Code="AISC 360-16 (LRFD)" File="Clip Angle
BCF.con" Template="DA BCF All Bolted" />
        <RCNXTemplate TagName="DA-BCWS" Code="AISC 360-16 (LRFD)" File="Clip Angle
BCW.con" Template="DA BCW All Bolted" />
        <RCNXTemplate TagName="DA-BG-S" Code="AISC 360-16 (LRFD)" File="Clip Angle
BG.con" Template="DA BG All bolted" />

        <RCNXTemplate TagName="ST-BCFS" Code="AISC 360-16 (LRFD)" File="Standard
Tee BCF.con" Template="ST BCF All Bolted" />
        <RCNXTemplate TagName="ST-BCW1" Code="AISC 360-16 (LRFD)" File="Standard
Tee BCW.con" Template="ST_BCW_WT 5x16.5_3B3/4" />
        <RCNXTemplate TagName="ST-BG-S" Code="AISC 360-16 (LRFD)" File="Standard
Tee BG.con" Template="ST BG All bolted" />

        <RCNXTemplate TagName="BP-BCF1" Code="AISC 360-16 (LRFD)" File="Bent Plate
BCF.con" Template="BP_BCF_1/2PL_3B3/4" />
        <RCNXTemplate TagName="BP-BCW1" Code="AISC 360-16 (LRFD)" File="Bent Plate
BCW.con" Template="BP_BCW_1/2PL_3B3/4" />
        <RCNXTemplate TagName="BP-BG-1" Code="AISC 360-16 (LRFD)" File="Bent Plate
BG.con" Template="BP_BG_1/2PL_3B3/4" />

        <RCNXTemplate TagName="SS-BCFS" Code="AISC 360-16 (LRFD)" File="Stiffened
Seated BCF.con" Template="SST BCF Bolted" />
        <RCNXTemplate TagName="SS-BCWS" Code="AISC 360-16 (LRFD)" File="Stiffened
Seated BCW.con" Template="SST BCW Welded" />
        <RCNXTemplate TagName="US-BCFS" Code="AISC 360-16 (LRFD)" File="UnStiffened
Seated BCF.con" Template="US BCF Welded" />
        <RCNXTemplate TagName="US-BCWS" Code="AISC 360-16 (LRFD)" File="UnStiffened
Seated BCW.con" Template="US BCW Welded" />

        <RCNXTemplate TagName="TP-BCF1" Code="AISC 360-16 (LRFD)" File="Through
Plate BCF.con" Template="TP_BCF_1/2PL_3B1" />
        <RCNXTemplate TagName="TP-BCW1" Code="AISC 360-16 (LRFD)" File="Through
Plate BCW.con" Template="TP_BCW_1/2PL_3B1" />

        <RCNXTemplate TagName="FP-BCFS" Code="AISC 360-16 (LRFD)" File="Flange
Plate BCF.con" Template="FP BCF Bolted" />
        <RCNXTemplate TagName="FP-BCWS" Code="AISC 360-16 (LRFD)" File="Flange
Plate BCW.con" Template="FP BCW Bolted" />
        <RCNXTemplate TagName="FP-BG-S" Code="AISC 360-16 (LRFD)" File="Flange
Plate BG.con" Template="FP BG Bolted" />

        <RCNXTemplate TagName="DW-BCFS" Code="AISC 360-16 (LRFD)" File="Directly
Welded BCF.con" Template="DW BCF" />
        <RCNXTemplate TagName="DW-BCWS" Code="AISC 360-16 (LRFD)" File="Directly
```

```
Welded BCW.con" Template="DW BCW" />

        <RCNXTemplate TagName="MA-BCFS" Code="AISC 360-16 (LRFD)" File="Moment
Angle_Tee BCF.con" Template="MA BCF Bolted" />

        <RCNXTemplate TagName="FW-BCFS" Code="AISC 360-16 (LRFD)" File="Fully DW
BCF.con" Template="FullyDW_BCF" />

        <RCNXTemplate TagName="ME-BCFS" Code="AISC 360-16 (LRFD)" File="Moment End
Plate BCF.con" Template="MEP BCF DG16 Extended both ways" />
        <RCNXTemplate TagName="ME-BCW1" Code="AISC 360-16 (LRFD)" File="Moment End
Plate BCW.con" Template="MEP_BCW_1 1/4PL_4B3/4" />

        <RCNXTemplate TagName="MK-BCF1" Code="AISC 360-16 (LRFD)" File="MEP Knee
BCF.con" Template="MEP_KNEE_BCF_VERTICAL_EU_3/8_PL_3B_1B5/8" />

        <RCNXTemplate TagName="DA-CBB" Code="AISC 360-16 (LRFD)" File="Gusset Plate
CBB.con" Template="CBB_DA_cont" />

        <RCNXTemplate TagName="SP-CBB" Code="AISC 360-16 (LRFD)" File="Gusset Plate
CBB.con" Template="CBB_SP" />
        <RCNXTemplate TagName="DW1-CBB" Code="AISC 360-16 (LRFD)" File="Gusset
Plate CBB.con" Template="CBB_DW_CBF" />
        <RCNXTemplate TagName="DW2-CBB" Code="AISC 360-16 (LRFD)" File="Gusset
Plate CBB.con" Template="CBB_DW" />

        <RCNXTemplate TagName="G1-CVR" Code="AISC 360-16 (LRFD)" File="Gusset Plate
CVR.con" Template="CVR" />
        <RCNXTemplate TagName="G2-CVR" Code="AISC 360-16 (LRFD)" File="Gusset Plate
CVR.con" Template="CVR_BCF" />

        <RCNXTemplate TagName="G1-VXB" Code="AISC 360-16 (LRFD)" File="Gusset Plate
VXB.con" Template="VXB" />

        <RCNXTemplate TagName="GT-BP" Code="AISC 360-16 (LRFD)" File="Gusset Base
Plate.con" Template="Gusset BP" />

        <RCNXTemplate TagName="G1-HCBB" Code="AISC 360-16 (LRFD)" File="Gusset
Plate HCBB.con" Template="HCBB_CA" />

        <RCNXTemplate TagName="G1-HBBB" Code="AISC 360-16 (LRFD)" File="Gusset
Plate HBBB.con" Template="HBBB_CA" />

        <RCNXTemplate TagName="G1-HXB" Code="AISC 360-16 (LRFD)" File="Gusset Plate
HXB.con" Template="HXB" />

        <RCNXTemplate TagName="TJ-CHB" Code="AISC 360-16 (LRFD)" File="Chord
Branches.con" Template="CHB_FEXX70_6" />

        <RCNXTemplate TagName="Cap-CC1" Code="AISC 360-16 (LRFD)" File="Cap Plate
CC.con" Template="CP_3/4PL_2B3/4" />
        <RCNXTemplate TagName="SEP-BCF1" Code="AISC 360-16 (LRFD)" File="Shear End
Plate BCF.con" Template="EP_BCF_1/4PL_10 in_E70XX" />
        <RCNXTemplate TagName="SEP-BCW1" Code="AISC 360-16 (LRFD)" File="Shear End
Plate BCW.con" Template="EP_BCW_1/4PL_10 in_E70XX" />

        <RCNXTemplate TagName="MEP-BCW1" Code="AISC 360-16 (LRFD)" File="Moment End
Plate BCW.con" Template="MEP_BCW_1PL_4B7/8" />
```

```
        <RCNXTemplate TagName="MEP-BCW2" Code="AISC 360-16 (LRFD)" File="Moment End
Plate BCW.con" Template="MEP_BCW_3/4PL_4B3/4" />
        <RCNXTemplate TagName="SP-BG1" Code="AISC 360-16 (LRFD)" File="Single Plate
BG.con" Template="SP_BG_1/4PL_2B3/4" />
        <RCNXTemplate TagName="SP-BG2" Code="AISC 360-16 (LRFD)" File="Single Plate
BG.con" Template="SP_BG_1/4PL_4B3/4" />

        <RCNXTemplate TagName="ME-BSe" Code="AISC 360-16 (LRFD)" File="Moment End
Plate BS.con" Template="MEP BS Extended both ways" />

        <RCNXTemplate TagName="EP-CS1" Code="AISC 360-16 (LRFD)" File="End Plate CS
HSS.con" Template="EndPlate_CS_3/4PL_4B1/2" />

        <RCNXTemplate TagName="Pin-BP" Code="AISC 360-16 (LRFD)" File="Base
Plate.con" Template="Pinned BP" />
        <RCNXTemplate TagName="Fix-BP1" Code="AISC 360-16 (LRFD)" File="Base
Plate.con" Template="Fixed uniaxial major axis BP" />

    </RCNXTemplates>
    </Tag>
</ConnectionTagFile>
```

3. In STAAD.Pro, open the file `ConnectionTagsExampl02.std`.

EX. Add the connection tags definition file

1. On the **Utilities** ribbon tab, select the **Connection Tags** > **Open Tag Specifications** tool in the **Tools** group.
   The **Select Connection Tag File** dialog opens.
2. Navigate to and select the connection tags definition XML file, `ConnectionTagsData.xml`.

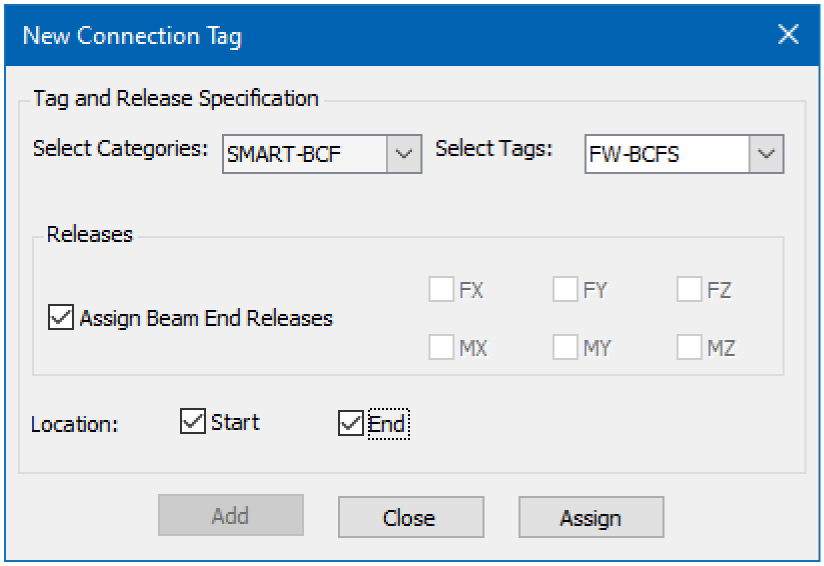
   Tip: This connection tags definition XML file contains the associated RAM Connection types to use for each connection tag.
3. Click **Open**.

EX. Assign connection tags to beam ends

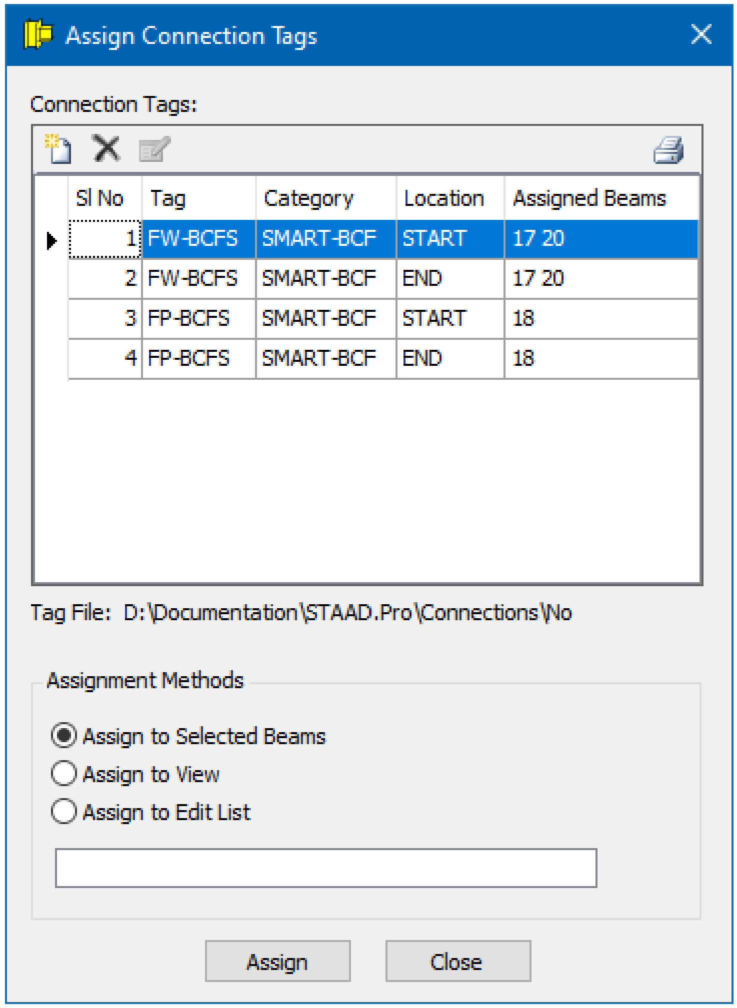
1. Select members 17 and 20.

   Hold **<Ctrl>** and then click on the members individually.

   Tip: Press **<Shift+B>** to display the member numbers.
2. On the **Beam Tools** ribbon tab, select the **Assign** tool in the **Connection Tags** group.
   The **New Connection Tag** and **Assign Connection Tag** dialogs both open.
3. In the **New Connection Tag** dialog, select **SMART-BCF** from the **Select Categories** drop-down list and then select **FW-BCFS** from the **Select Tags** drop-down list.



4. Check both the **Start** and **End** options.
5. Click **Assign**.
   The connection tag assignments are added to the table in the **Assign Connection Tags** dialog. There are two entries in the table, one for both the start and end connection.
6. In the View Window, click on member number 18 to select it.
7. In the **Assign Connection Tags** dialog, click the **New** tool.
   The **New Connection Tag** dialog opens.
8. Select **SMART-BCF** from the **Select Categories** drop-down list and then select **FP-BCFS** from the **Select Tags** drop-down list.
9. Check both the **Start** and **End** options.
10. Click **Assign**.
    The connection tag assignments are added to the table in the **Assign Connection Tags** dialog.



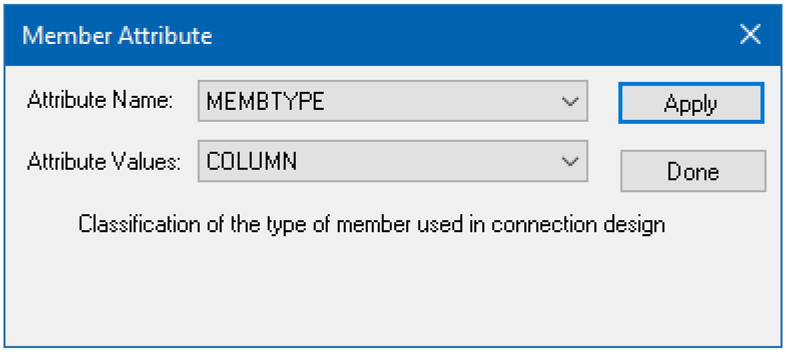
Repeat this procedure to assign the following connection tags in the model:

Member Numbers	Categories	Tags	Locations
41 42 48 TO 53	SMART-BCF	DW-BCFS	Start and End
19	SMART-BCF	MA-BCFS	Start only
34	SMART-BCF	MA-BCFS	End only
22	SMART-BCF	ME-BCFS	Start and End
24	SMART-BCW	EP-BCWS	Start only
27	SMART-BCW	EP-BCWS	End only
55	SMART-BG	DA-BG-S	Start and End
56	SMART-BG	SP-BG-S	Start and End

Member Numbers	Categories	Tags	Locations
54	SMART-BG	EP-BG-S	Start and End
57 59 79	SMART-BCW	FP-BCWS	Start and End
60	SMART-BCW	DW-BCWS	Start and End
58	BASIC-BCW	ST-BCW1	Start only
80	BASIC-BCW	MEP-BCW2	End only
81	BASIC-BCF	BP-BCF1	Start only
83	BASIC-BG	BP-BG-1	End only
84	BASIC-BG	BP-BG-1	Start only
61 62	BASIC-BCW	TP-BCW1	End only
74	SMART-BCF	US-BCF1	End only
66 67 72 82	BASIC-BCF	MK-BCF1	Start only
80	SMART-BEAM	ME-BSE	Start only

EX. Assign gusset connection tags to columns

1. Assign the column member types:
   a. On the **Select** ribbon tab, select the **Parallel** > **Y** tool in the **Beams** group.
   All the vertical members (parallel to the global Y axis) are selected.
   b. On the **Beam Tools** ribbon tab, select the **Add Attribute** tool in the **Plugins** group.
   The **Member Attribute** dialog opens.
   c. Select **MEMBTYPE** from the **Attribute Name** drop-down list and then select **COLUMN** from the **Attribute Value** drop-down list.



   d. Click **Apply**.
2. Assign the brace member types:
   a. On the **Select** ribbon tab, select the **List** tool in the **Beams** group.
   The **Select Beams** dialog opens.

b. Select the **Select using typed list** option.
c. Type the following list in the text field: `30 TO 33 35 TO 38 43 44 46 47 85`.

Tip: You can also copy and past this list using keyboard commands.

d. Click **Select Listed Entities**.
e. Add the **MEMBTYPE** attribute of **BRACE** to these members, similar to what you did in step 1.

3. Select members 7 8 26 37 and 38.

Tip: You can hold the **<Ctrl>** key which clicking on these members individually or use the list selection option from Step 2.

4. On the **Beam Tools** ribbon tab, select the **Assign** tool in the **Connection Tags** group.
   The **New Connection Tag** and **Assign Connection Tag** dialogs both open.
5. In the **New Connection Tag** dialog, select **GUSSET-CBB** from the **Select Categories** drop-down list and then select **DA-CBB** from the **Select Tags** drop-down list.
6. Check the **End** option only.
7. Click **Assign**.

Repeat this procedure (steps 3 through 7) to assign the following connection tags in the model:

Member Numbers	Categories	Tags	Locations
7 8 26 37 38	GUSSET-CBB	DA-CBB	End only
85	GUSSET-CBB	DW1-CBB	Start only
85	GUSSET-BP	GT-BP	End only
32 33 35 46 47	GUSSET-BP	GT-BP	Start only
33	GUSSET-VXB	G1-VXB	End only
34 45	GUSSET-CVR	G1-CVR	Start only
70 71	BASIC-COL	EP-CS1	End only
30	GUSSET-HXB	G1-HXB	Start only
30	GUSSET-HBBB	G1-HBBB	End only
21 43	GUSSET-HCBB	G1-HCBB	End only

EX. Analyze, design, and export connection data

1. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.
   The **STAAD Analysis and Design** dialog opens and the analysis is performed.
2. Click **Done**.
3. Select the **Connection Design** workflow.
   The **Materials Mapping** dialog opens.
4. Click **OK**.

The connections are designed based on the connection tag mappings in the XML file. The results of the connection design process is displayed in the RAM Connection Input table.

You can double-click any row in this table to open the connection pad. From there, you can review the connection design details and make changes as needed.

5. Select **File** > **ISM** and then select **Create Repository**.
   The **STAAD.Pro CONNECT Edition ISM Module** dialog opens and then you are prompted to save the repository to the same base name and file location as the STAAD input file.
6. Click **Save**.
7. Check that the **Open Structural Synchronizer** option and **Include Member Offsets** option are both checked.
8. Click the **Run** tool.
   If you are prompted again to create the repository file, click **Yes**.
9. In the **iTwin Analytical Synchronizer** application, select the **Select All** tool for Objects and then click **Update**.
10. Type an option commit message and then click **OK**.
    The repository is created and the model data is pushed to it. The **iTwin Analytical Synchronizer** application closes.
11. In the **STAAD.Pro CONNECT Edition ISM Module** dialog , click **OK** in the prompt and the close the dialog.

You now have an ISM repository that can be used to transmit connection detailing designed in STAAD.Pro and RAM Connection to detailing software, such as ProStructures.

EX. Interactive Concrete Design Examples

This is an example of using the interactive Concrete Design workflow to design an example concrete structure.

EX. Advanced Concrete Design Tutorial

Getting Started

STAAD Advanced Concrete Design is performed using the RCDC application, which is installed separately from STAAD.Pro. This program can be launched from the STAAD.Pro Building Designer mode or as a standalone application.

RCDC is a highly automated design tool for concrete building structures. This tutorial guides you through the process of designing the elements in a concrete building. It will also indicate where to change the design settings to customize design and detailing to meet the requirements of your project or organization.

Tutorial File

This tutorial uses example files that are installed with STAAD.Pro V8i. However, you will be making changes to this example file so you may wish to make a backup copy to preserve the example as installed with STAAD.Pro. A copy of the original input file is also included as Tutorial STAAD Input File (on page 6934) of this tutorial.

Starting in STAAD.Pro

This example includes a fully modeled concrete structure, including loads. The modeling and application of loads per a building code specification are outside the scope of this tutorial. Refer to the STAAD.Prohelp contents for additional information.

Using RCDC as a Standalone Application

You can also use RCDC as a standalone application for concrete design. You will still need to have a analytical model created in STAAD.Pro for the basis of the physical model as well as for analytical results.

Design and detailing of foundation elements such as footings and pilecaps must be performed by starting RCDC as a standalone application and creating a project associated with an analytical model.

To open the model and generate analysis

Tip: It is recommended that you first make a backup copy of all files with the name `Typical Ground +3 Story Building` in the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models \Building Planner\` folder. You can do this I. To create an archive (on page 2067) in STAAD.Pro.

1. Open the STAAD input file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\Sample Models \Building Planner\Typical Ground +3 Story Building.std`.

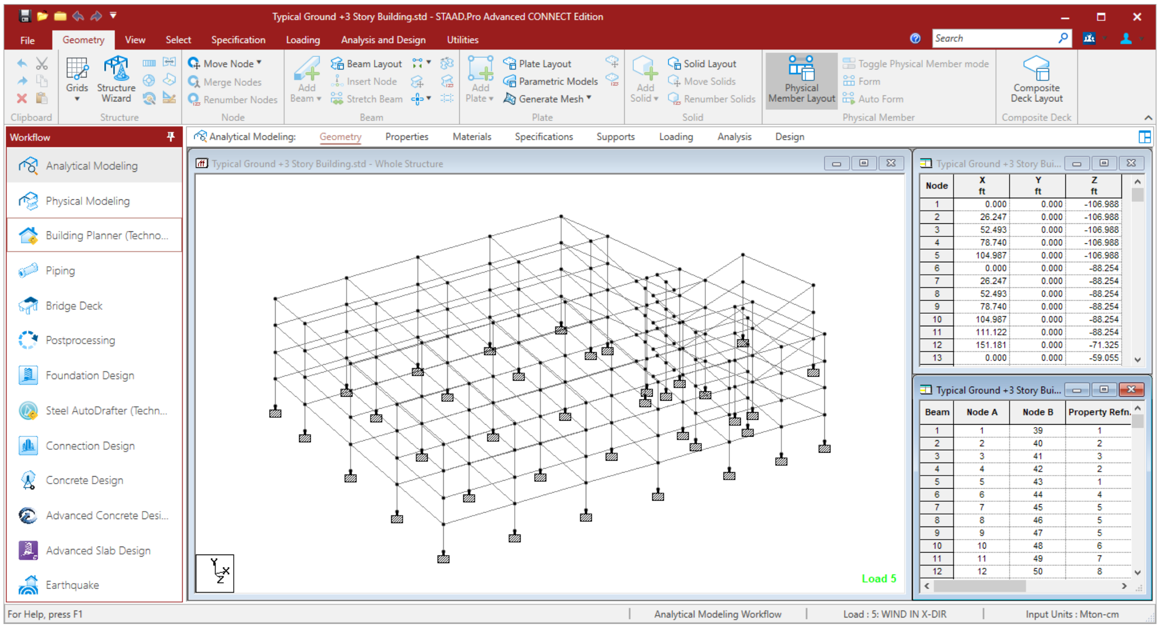
   Tip: If you have a ProjectWise Project you want to associate with this input file, you can do so. This is not necessary, though, for this tutorial.
2. (Optional) On the **File** ribbon tab **Info** tab, add job information:

   type `Tutorial` in the **Job** field
   type your organization's name in the **Client** field
   type your initials in the **Engineer** field
3. Either:

   on the **Analysis and Design** ribbon tab, select the **Run Analyze** tool in the **Analysis** group

   or

   press **<Ctrl+F5>**
   The analysis is performed.
4. Select the **Stay in Modeling Mode** option and click **Done**.



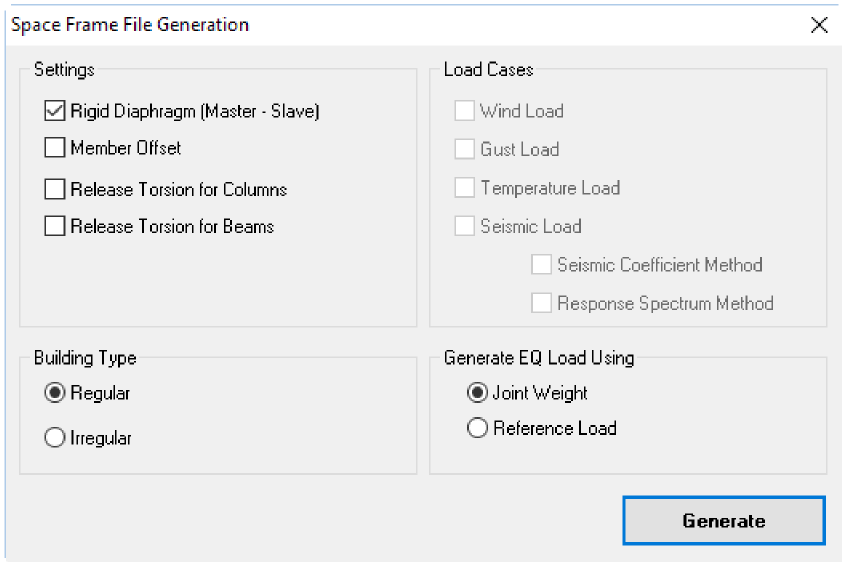
5. Select the **Building Planner** workflow.
   The STAAD Building Planner window opens.

The STAAD Building Planner will be used to layout the structure.

To create a building model from floor plan

In order to demonstrate column design in RCDC, you will use the Building Planner model rapidly model a building from a single floor plan.

1. Select the **Frame | Geometry** page.
2. Click **Edit** in the **Level Details** dialog.
   The **Level Information** dialog opens
3. Type 6 (m) in the **Height** cell for row No 0-1 and press **<Enter>**.
4. Click **Add Level** and then click **Yes** in the confirmation dialog.
   A new row is added representing a new level.
5. Select **Plan1** from the **Plan** drop-down list for the second level.
   Leave the default heights and other values for this level.
6. Click **OK**.
7. In the **Level Details** dialog, click **Generate Model** and then click **Yes** to confirm you want to update the STAAD input model..
   The **Space Frame File Generation** dialog opens.



8. Click **Generate**.
   For this tutorial, you will use the default settings, load settings, and building type. Refer to the STAAD.Pro User Interface help for details on these settings.
   The new input file is generated and opened.
9. Either:

   select **Analyze** > **Run Analysis**

   or

   press **<Ctrl+F5>**

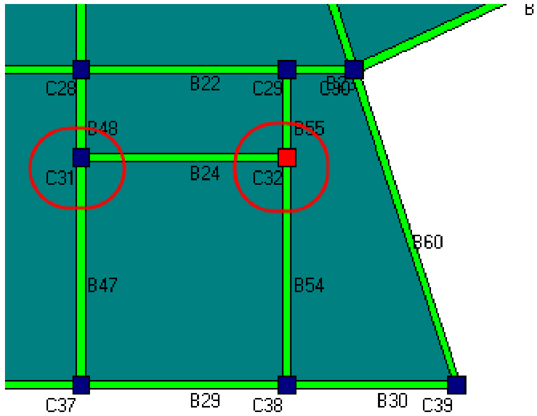
   The analysis is performed.
10. Select the **Stay in Modeling Mode** option and click **Done**.

To add a shear wall

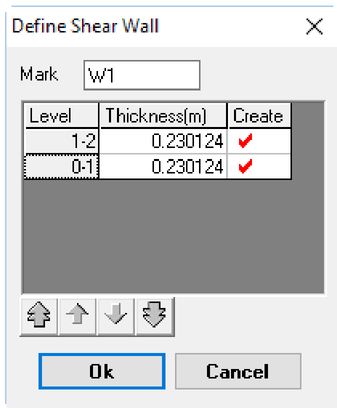
To add a shear wall along one side of the elevators, do the following.

For the purpose of this tutorial, you will only add a single shear wall. In a typical building, other shear walls would be present.

1. In the Building Planner mode, select the **Frame | Shear Wall** page.
2. In the **Shear Wall Details** dialog, click **Edit**.
3. Click on column C31 and then on C32.



The **Define Shear Wall** dialog opens.



4. Click **OK**.
   For this tutorial, use the default thickness. The shear wall is added on both levels using this plan by default.

You have now finished all the necessary modifications to this example for this tutorial. The next sections cover the design and detailing of structural elements using RCDC.

Slab Design

This section of the tutorial demonstrates how to design and detail a slab, including openings and stairs.

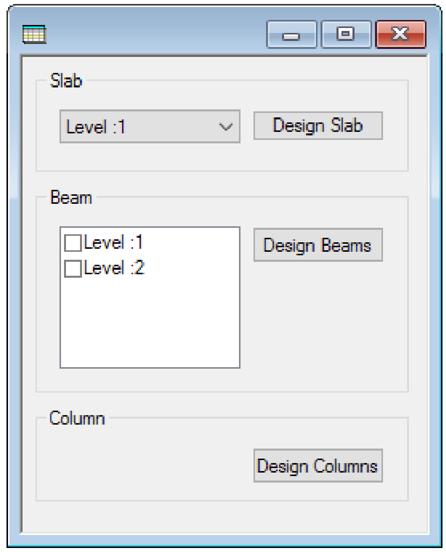
Slab continuity is based on edge conditions where are automatically determined by the program. You can manually change the end conditions of a slab. You can also change a slab to a cutout, raised slab, or depressed slab which also affect the edge conditions.

Slab properties are imported from the analysis model where specified. The program will evaluate missing properties (e.g., slab thickness) during design when not specified.

To start the slab design

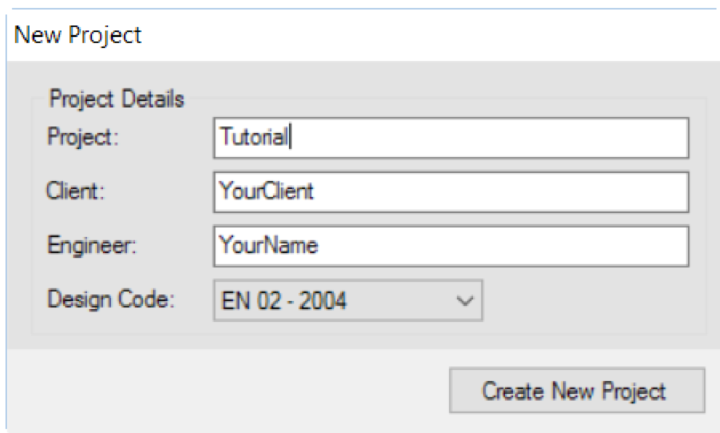
Tip: When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, select **Level :1** from the drop-down list and then click **Design Slab.**



RCDC opens and the **New Project** dialog opens.

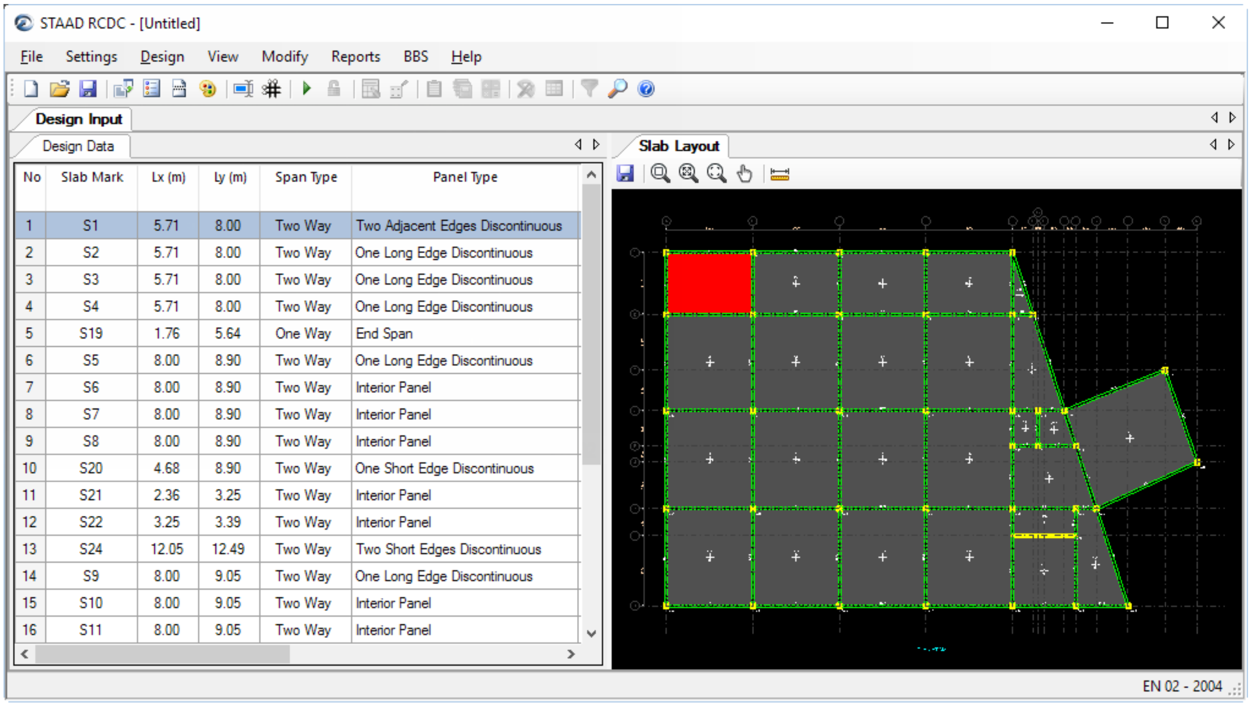
2. Enter the project data:



   a. Type `Tutorial` in the **Project** field.
   b. (Optional) Type **Client** and **Engineer** name.

      This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.
   c. Select **EN 02 - 2004** from the **Design Code** drop-down list.
      This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

The slab edges are automatically detected and continuity is established.



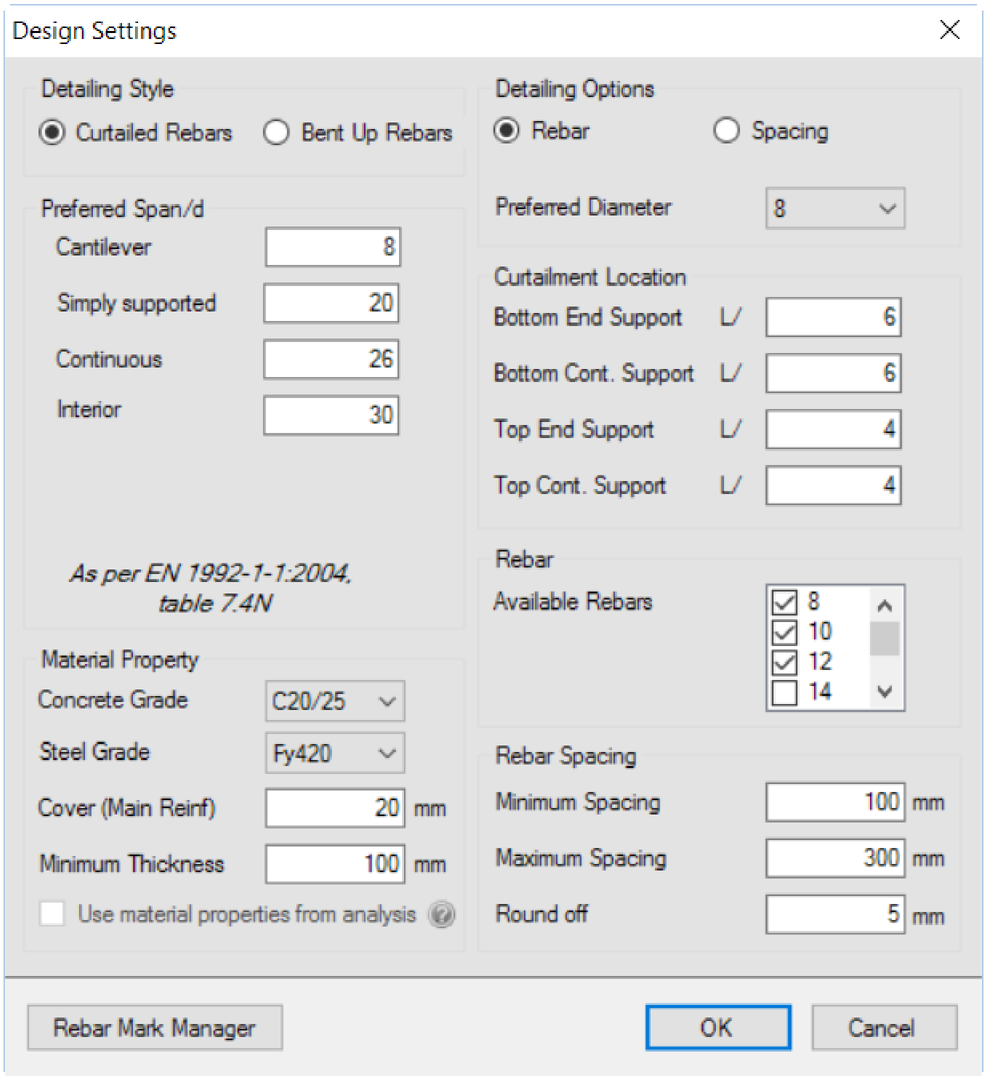
3. Either:

select the **Design Parameters** tool

or

select **Settings** > **Design Parameters**

The **Design Settings** dialog opens.



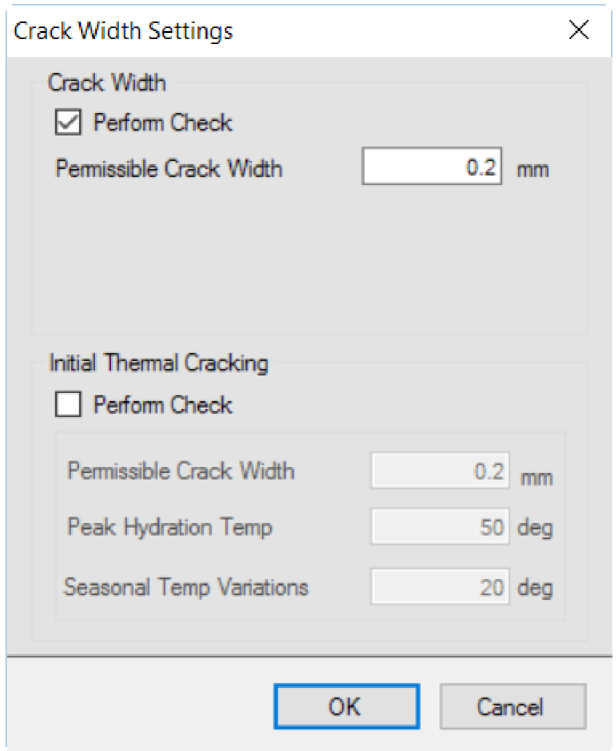
4. In the Available Rebar list, clear the check mark by **25** (mm).
   This tutorial will limit the rebar size.

   Note: All other design settings are left as their defaults, but you can make many design and detailing changes here.
5. Click **OK**.
6. Either:

   select the **Crack Width Setting** tool 

   or

   select **Settings** > **Crack Width Settings**
   The **Crack Width Settings** dialog opens.



7. Select the **Perform Check** option in the **Crack Width** group.

   Note: Leave the Permissible Crack Width as the default value and leave the Perform Check option in the Initial Thermal Cracking group unchecked for this tutorial.

8. Click **OK**.

To specify elevator openings

Since the analysis model does not contain openings in the slab, you can add those in RCDC.

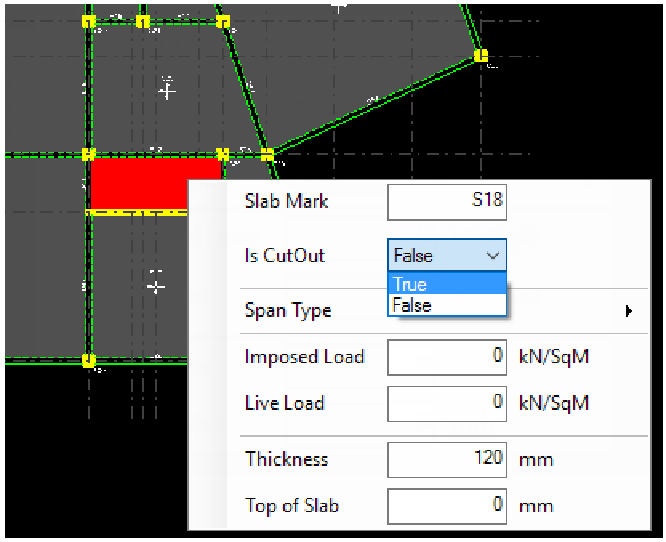
Note: RCDC can design and detail concrete stairs with the slab. Do *not* create an opening for the stairs if you plan to use this feature. Refer to the RCDC help topic **Slab** > **Staircase Section** for details on this feature.

1. In the Design Data table, select row 19 (for Slab 18).
   The slab is highlighted in the Slab Layout view window.
2. Do either of the following:

   select **True** in the **Is Cut Out** cell for this row in the table

   or

   right-click on the slab in the view window and select **True** from the **Is Cut Out** drop-down in the pop-up menu



The slab is now marked as a cutout.

3. Select **Modify** > **Re-evaluate Slab continuity**.

> **Note:** This will change the continuity of surrounding slabs. If you do not perform this step, you will be prompted to automatically re-evaluate the continuity before performing an autodesign.

To perform the design

Slabs in RCDC are designed for the **Imposed Load** and **Live Load** values input for each slab on the **Design Data** table. These values are imported (along with thickness, etc.) from the slab parameters specified in STAAD.Pro Building Planner mode.

1. Either:

   select the **Autodesign** tool 

   or

   select **Design** > **Autodesign**.

   A message dialog opens confirming you want to proceed since the continuity must be reevaluated if you have not already done so.
2. If you did not previously reevaluate the continuity, click **Yes**.
   The Design Output page opens.

   A warning message opens stating that two of the slabs have failed the design checks. The same slab is marked with a red row number in the table.
3. Select the row for the first failed slab (row 13, S24) in the table and then either:

   select the **Design** tool 

   or

   select **Modify** > **Design**

   or

   right-click and select **Design** from the pop-up menu

The **Design Slab** dialog for this slab opens at the bottom of the program window.

4. Type `325` (mm) in the **Thickness** field in the **Design Parameters** group.
5. Select the **Design** tool in the **Design Slab** dialog.
   The slab now passes the design checks.
6. Select the **Accept** tool to update the design with the changes.
7. Repeat steps 3 through 6 for S17 (row 24) with a thickness of `200` (mm).

To generate construction and detail drawings

1. Generate a GA drawing for the slab:
   a. Select **Reports** > **General Arrangement Plan**.
      The **GA Drawing** view window opens.
   b. Click the **Save** tool in the view window.
   c. Type `Sample_Slab_GA.dxf` and click **Save** to save the GA drawing as a DXF file.
2. Generate a schedule:
   a. Either:

      select the **Text Schedule** tool

      or

      Select **Reports** > **Text Schedule**.

      A dialog opens to ask if you want to group similar slabs.
   b. Click **Yes**.
      The **Text Schedule** view window opens.
   c. Click the **Save** tool in the view window.
   d. Type `Sample_Slab_Sched.dxf` and click **Save** to save the schedule as a DXF file.
3. Generate the reinforcement layout drawings:
   a. Either:

      select the **In-Plan Detailing** tool

      or

      Select **Reports** > **In-Plan Detailing**.

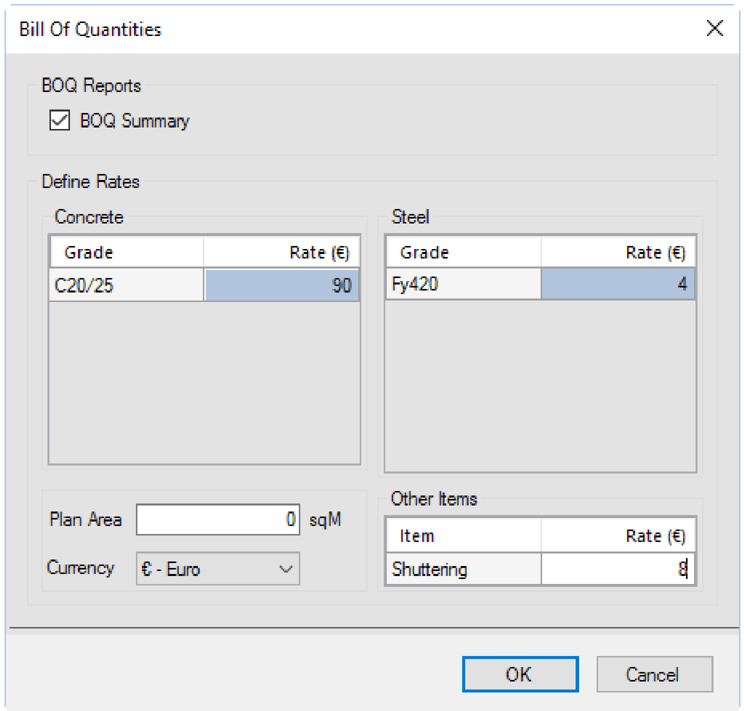
      The Top Reinforcement and Bottom Reinforcement drawings open.
   b. Click the **Save** tool in the view window.
   c. Type `Sample_Slab_TopReinfPlan.dxf` and click **Save** to save the schedule as a DXF file.
   d. Repeat steps 4b and 4c to save the bottom reinforcement plan as `Sample_Slab_BotReinfPlan.dxf`.
4. Generate a bill of quantities:
   a. Either:

      select the **Bill of Quantities** tool

      or

      Select **Reports** > **Bill of Quantities**.

      The **Bill Of Quantities** dialog opens.



b. Select the **BOQ Summary** option.

c. Select **€ - Euro** from the **Currency** drop-down list.

d. Type `90` (€/m^3) for Grade C20/25 concrete and `4` (€/kg) for Grade Fy420 steel. Type `8` (€/m^2) for the Shuttering value.

e. Click **OK**.
The **BOQ Summary** opens.

f. Either:

click the **Print** tool to print out a copy of this report

or

click the **Save** tool to save an HTML copy of this report

To generate design calculation reports

1. Either:

   select the **Design Summary** tool

   or

   select **Reports** > **Design Summary**
   The **Slab Design Summary** opens.

2. Either:

   select the **Design Calculations** tool

   or

   select **Reports** > **Design Calculations**

The **Select Slabs** dialog opens.

3. Check the boxes for **S1** and **S12**.
   For this tutorial you will only output the detailed design for two slabs. You can check the All Slabs option to include the full design report.
4. Click **OK**.
   The **Slab Design** opens.
5. Either:

   click the **Print** tool to print out a copy of this report

   or

   click the **Save** tool to save an HTML copy of this report

To save the project and exit

1. Either:

   select the **Save** tool (on the main program toolbar)

   or

   select **File** > **Save**
   The **Save As** dialog opens.
2. Type `Sample1_Slab_tutorial.rcdx` in the filename and click **Save**.
3. Select **File** > **Exit**.
   RCDC closes and you are returned to STAAD.Pro.

RCDC Settings Files

When you save your project in RCDC, in addition to the project file (`.rcdx`) a project settings file is also created (`.rcps`). You can select **File** > **Import Project Settings** to open the project settings for a previous project once you have established project or office standards you want to re-use.

Beam Design

This section of the tutorial demonstrates how to design and detail continuous concrete beams.

RCDC will automatically look for continuous beam members in the STAAD model and group these as a single physical beam. This way if you have a beam which continuous through columns or one which was sub-divided into STAAD.Pro to facilitate support of a transverse member, it will be designed as a single beam entity in RCDC.

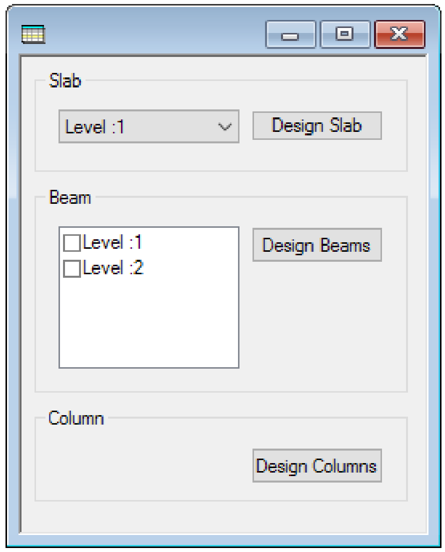
In this example, you will use most of the default values for all designs.

To start the beam design

In STAAD.Pro, ensure you are on the **Frame | Design** page in the **Building Planner** mode.

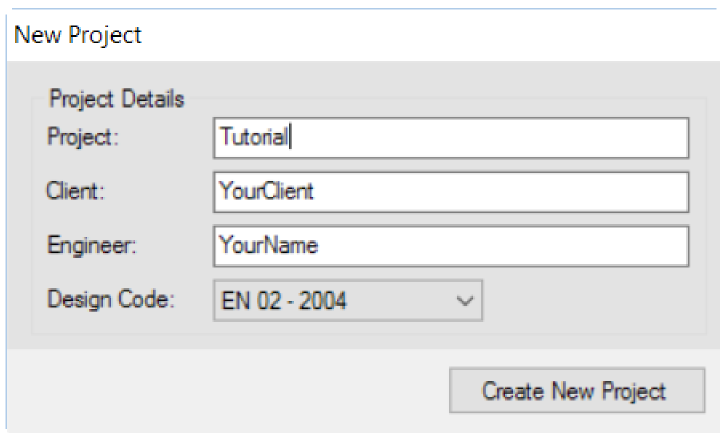
Tip: When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, check the **Level :1** option and then click **Design Beams**.



RCDC opens and the **New Project** dialog opens.

2. Enter the project data and select the design code:



   a. Type `Tutorial` in the **Project** field.
   b. (Optional) Type **Client** and **Engineer** name.

      This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.
   c. Select **EN 02 - 2004** from the **Design Code** drop-down list.
      This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

   The beam continuity is automatically detected as the analytical model data is read.

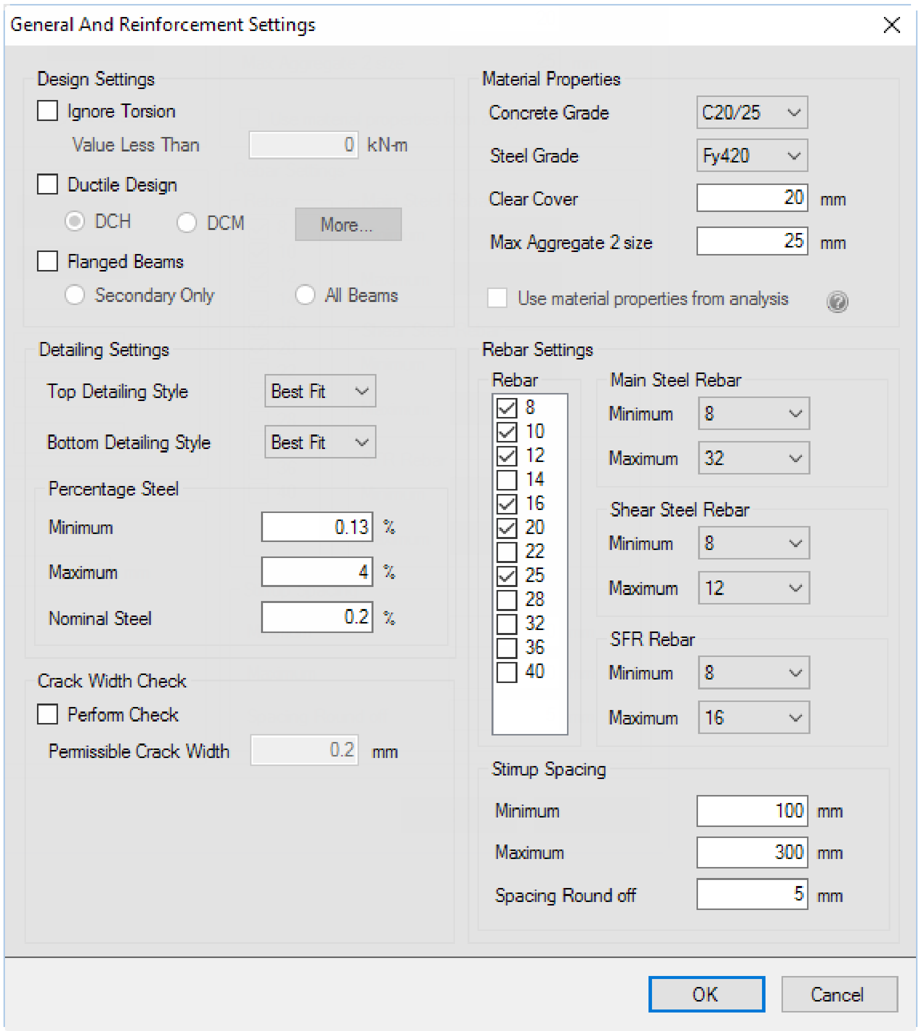
3. Either:

   select the **General and Reinforcement Settings** tool

   or

   select **Settings** > **General and Reinforcement Settings**

   The **General and Reinforcement Settings** dialog opens.



4. In the **Rebar** list, clear the check mark by both **32** (mm) and **40** (mm).
   The initial design will be evaluated using smaller size bars where possible.

   Note: All other design settings are left as their defaults, but you can make many design and detailing changes here.

5. Click **OK**.

Note: There are numerous other detailing and drawing settings in RCDC that allow you to customize the concrete detailing for your client or organization's needs. For details on these options, refer to the RCDC help by pressing **<F1>** or selecting **Help** > **Contents**.

To specify load cases and combinations

1. Select **Settings** > **Basic Load Cases**
   The **Basic Load Cases** dialog opens.
2. (Optional) If the **Load Type** selection is not already specified for each load case, select it from the drop-down list:

LOAD 9: DEAD LOAD	Dead Load
LOAD 10: LIVE LOAD	Live Load

3. (Optional) If both loads are not already added to the Primary Load Cases list, click the **[+]** button when a load case is selected in the Analysis Load Cases list to add it.

   > **Note:** You can import previously saved load cases and combinations to use by clicking the **Import Load Cases & Combinations** button and selecting an `.ldsx` file. Refer to the following steps on how to create a load settings file.

4. Click **OK**.
   The **Load Combinations** dialog opens.
5. Click **Add From Analysis**.
   The load combination from the analysis file is added for strength design checks.
6. Add the load combination for use in evaluating crack width:
   a. Select the **Crack Width** tab.
   b. Click **Add From Template** > **For Regular Structure**.
   A service level load combination is added from the RCDC load combinations template.

   > **Note:** Typically, a different (service level) load combination is used for evaluating crack widths. If you already have appropriate service load combinations in your analytical model, you can add those instead and select them for use here.

   c. Check the options for **Crack Width** and **Stress Limit** in the load load combination in the table.
7. Export the load cases and combinations for later use:
   a. Click **Export Load Cases & Combinations**.
   The **Save As** dialog opens.
   b. Type `Sample1_Loads` and click **Save**.
8. Click **OK**.

To split beam group at staircase

1. Select the **Design Input | Beam Continuum** tab.
2. Select the third beam group in the chart.

   > **Note:** The beams on grid line 6 are highlighted in the Layout view.

3. Right-click on the blue box labeled C17 (column 17) and select **Split Group** from the pop-up menu.
   The beam group is split into two physical members at this location.
4. Select the **Design Input | Design Grouping** tab.
5. Select **G19** in the table or click on the newly created beam group in the Layout view.
6. Type `-900` in the **Top** field for both beam segments in G19 (B14 and B15).
   This represents a top of beam elevation of 0.9m below the floor elevation.

To perform the beam design

1. Either:

   select the **Autodesign** tool 

   or

select **Design** > **Autodesign.**

An exceptions dialog opens indicating that two beams have failed the design checks.

2. Click **OK.**
3. Either:

   select the **Failure Diagnostics** tool

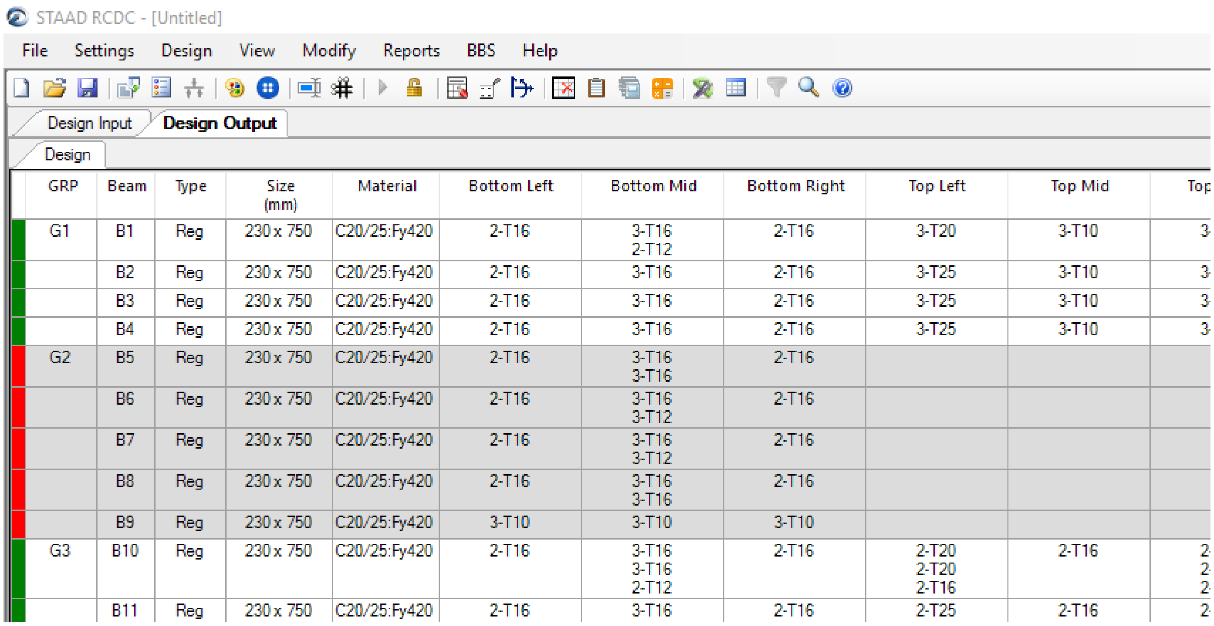
   or

   select **Reports** > **Failure Diagnostics**

   The **Failure Diagnostics** tab opens. The table indicates that many beams are failing due detailing checks.
4. Evaluate some options for beam group 2:

   a. Select any of the beam segments in **G2** in the Design Output table.

   The beam segments in this group are marked with a red tag to indicate the group failed one or more checks.



GRP	Beam	Type	Size (mm)	Material	Bottom Left	Bottom Mid	Bottom Right	Top Left	Top Mid	Top
G1	B1	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 2-T12	2-T16	3-T20	3-T10	3
	B2	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16	2-T16	3-T25	3-T10	3
	B3	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16	2-T16	3-T25	3-T10	3
	B4	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16	2-T16	3-T25	3-T10	3
G2	B5	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 3-T16	2-T16			
	B6	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 3-T12	2-T16			
	B7	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 3-T12	2-T16			
	B8	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 3-T16	2-T16			
	B9	Reg	230 x 750	C20/25:Fy420	3-T10	3-T10	3-T10			
G3	B10	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16 3-T16 2-T12	2-T16	2-T20 2-T20 2-T16	2-T16	2 2 2
	B11	Reg	230 x 750	C20/25:Fy420	2-T16	3-T16	2-T16	2-T25	2-T16	2

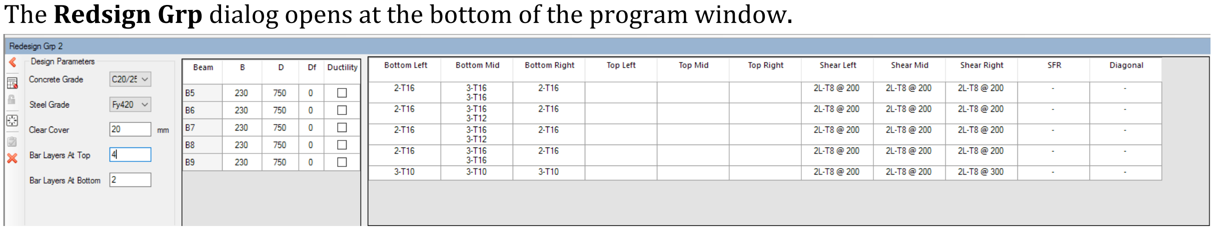
   b. Either:

   select the **Design** tool

   or

   select **Modify** > **Design**

   The **Redsign Grp** dialog opens at the bottom of the program window.



Beam	B	D	Df	Ductility
B5	230	750	0	☐
B6	230	750	0	☐
B7	230	750	0	☐
B8	230	750	0	☐
B9	230	750	0	☐

Bottom Left	Bottom Mid	Bottom Right	Top Left	Top Mid	Top Right	Shear Left	Shear Mid	Shear Right	SFR	Diagonal
2-T16	3-T16 3-T16	2-T16				2L-T8 @ 200	2L-T8 @ 200	2L-T8 @ 200	-	-
2-T16	3-T16 3-T12	2-T16				2L-T8 @ 200	2L-T8 @ 200	2L-T8 @ 200	-	-
2-T16	3-T16 3-T12	2-T16				2L-T8 @ 200	2L-T8 @ 200	2L-T8 @ 200	-	-
2-T16	3-T16 3-T16	2-T16				2L-T8 @ 200	2L-T8 @ 200	2L-T8 @ 200	-	-
3-T10	3-T10	3-T10				2L-T8 @ 200	2L-T8 @ 200	2L-T8 @ 300	-	-

   c. Type 3 in the **Bar Layers at Top** field.

   d. Select the **Redesign** tool in the **Redesign Grp** dialog.

The beam now passes design and crack width checks.

e. Select the **Accept** tool.

The Design Output table now shows green for the G2 beam indicating it passes.

5. Repeat step 4 to redesign G7 with 3 layers of top bars as well.
Optionally, you can use the **UnLock** tool to unlock the current design and then change the initial settings to use larger reinforcement sizes.

6. Review the detailing on the demand and capacity curve for a beam:

a. Select beam group G1 and either:

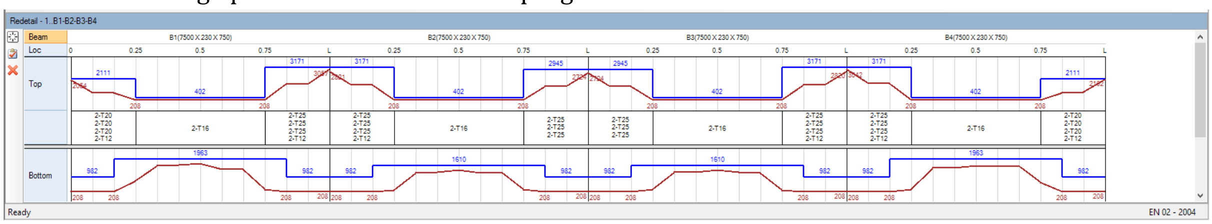
select the **Detail** tool

or

select **Modify** > **Detail**

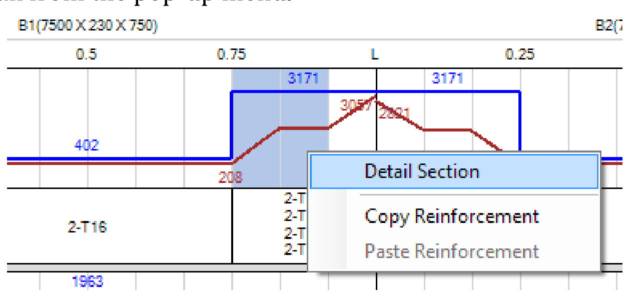
or

right-click and select **Detail** from the pop-up menu
The Redtail dialog opens at the bottom of the program window.



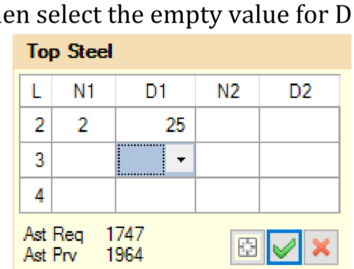
b. Click-and-drag your mouse cursor over the beam subdivisions starting at 0.75L of the first beam span in G1.

c. Right-click and select detail from the pop-up menu.



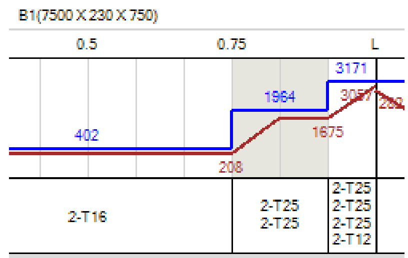
The Top Steel pop-up dialog opens.

d. Delete the N1 value for layer 4 and then select the empty value for D1 for the same layer.



This will remove the fourth layer of bars from the top steel in this segment, creating a new reinforcement detail for the selected segment.

e. Click OK.
The section of the capacity curve for the top steel updates to reflect the detailing change.



f. Click Cancel.

> **Tip:** You can use select the beam segments or entire reinforcement zones by their labels just below the diagram. You can copy and paste reinforcement details between zones or selections.

7. (Optional) Check for any resized beams during design by selecting **Reports** > **Beam Size Changed**.
A report including any resized beams during design opens. For this example, none of the beams were resized.

To generate beam schedules and drawings

1. Generate a text schedule of beam reinforcement:
   a. Either:

      select the **Text Schedule** tool

      or

      Select **Reports** > **Text Schedule**.

      The **Schedule Type** dialog opens.
   b. Select the **Type 1** option.
   c. Select the **Group Beams** option.
   d. Click **OK**.
      The **Text Schedule** view window opens.
   e. Click the **Save** tool in the view window.
   f. Type `Tutorial_Beam_Sched.dxf` and click **Save** to save the schedule as a DXF file.
2. Generate an elevation for any beam group:
   a. In the **Design Output** table, right-click on any beam group and select **Quick Elevation** from the pop-up window.
      The elevation drawing for this beam group opens in the lower section of the program window.
   b. (Optional) Select the **Save** tool in the **Elevation** view to save this view as a DXF file.
   c. Select the **Cancel** tool to close the elevation view
3. Generate the elevation and sections for a beam group:
   a. Either:

      select the **Elevation and Section** tool

      or

      Select **Reports** > **Elevation and Section**.

      The **Select Beams** dialog opens.
   b. Select group **1.B1-B2-B3-B4** in the beams list.
   c. Select the following Draw options:

Elevation
Mark Anchorage
Cross section
Show Section Mark

> **Tip:** You can modify the drawing and detailing options by clicking the **Detailing & Drawing Settings** button (also by selecting **Settings** > **Detailing & Drawing Settings** before using this tool). You can also customize rebar mark styles from this dialog.

d. Click **OK**.
The Elevation for the selected beam group opens.

e. Click the **Save** tool in the view window.

f. Type `Tutorial_Beam_Sched.dxf` and click **Save** to save the schedule as a DXF file.

4. Generate a bill of quantities:

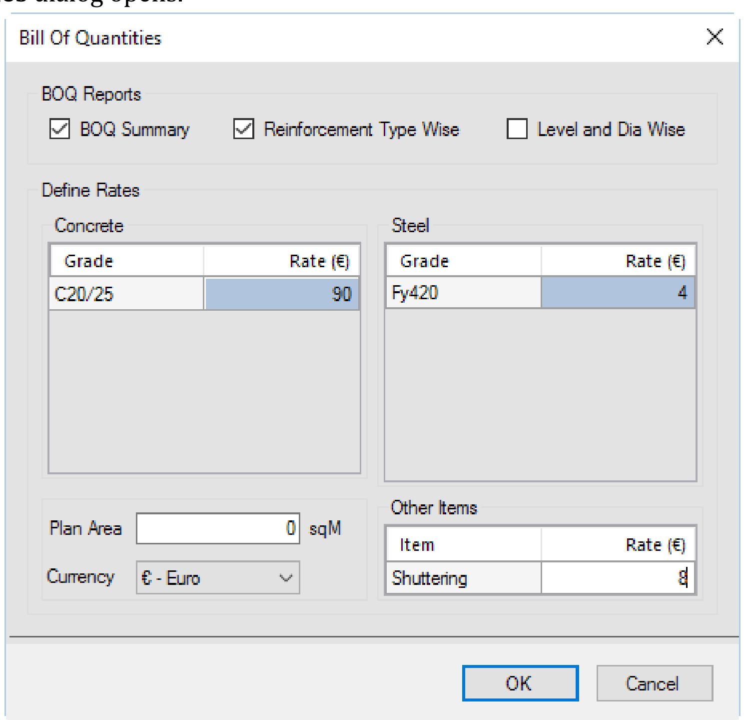
a. Either:

select the **Bill of Quantities** tool

or

Select **Reports** > **Bill of Quantities.**

The **Bill Of Quantities** dialog opens.



b. Select the **BOQ Summary** and **Reinforcement Type Wise** options.

c. Select **€ - Euro** from the **Currency** drop-down list.

d. Type `90` (€/m^3) for Grade C20/25 concrete and `4` (€/kg) for Grade Fy420 steel. Type `8` (€/m^2) for the Shuttering value.

e. Click **OK**.
The **BOQ Summary** and **BOQ Detailed - Reinf Type Wise** tabs open.

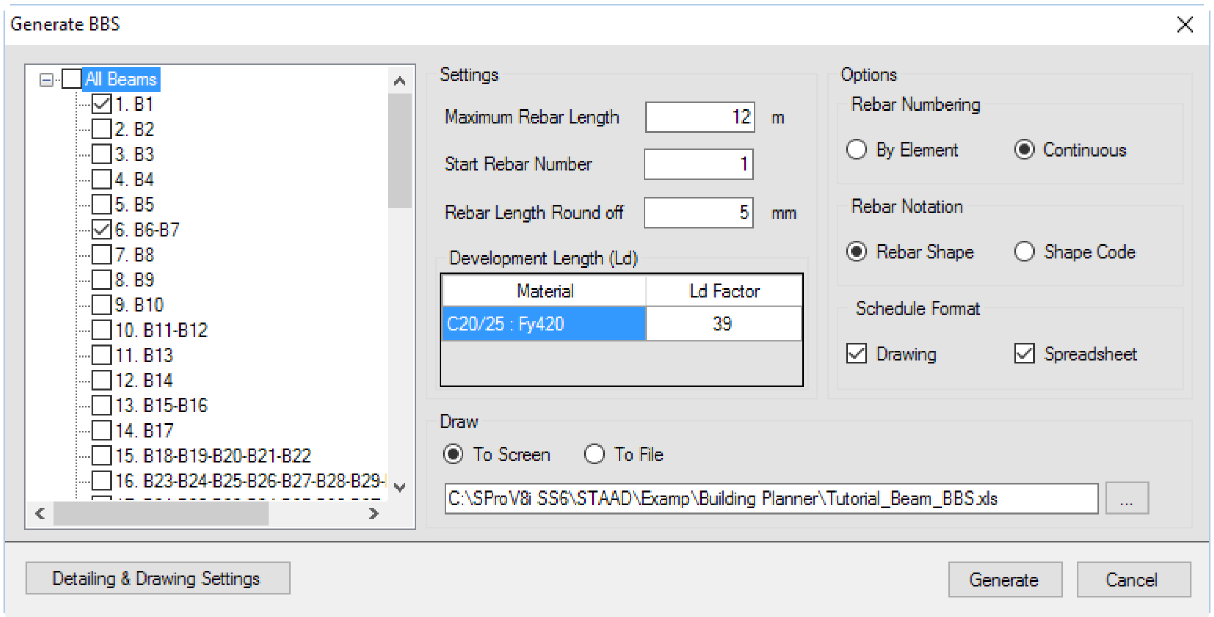
f. Either:

click the **Print** tool to print out a copy of this report

or

click the **Save** tool to save an HTML copy of this report

5. Generate bar bending schedule:
   a. Select **BBS** > **Generate BBS**.
   The **Generate BBS** dialog opens.



   b. Select **1.B1-B2-B3-B4** and **2. B5-B6-B7-B8-B9** in the list of beam groups.
   c. Select the **Continuous** option in the **Rebar Numbering** group.
   d. Select both the **Drawing** and **Spreadsheet** options in the **Schedule Format** group.
   e. Click inside the file path field, type `Tutorial_Beam_BBS` in the **Save As** dialog that opens, and then click **Save**.
   f. Click **Generate**.
   The BBS opens containing a text schedule of the bar bending schedule. If you have Microsoft® Office Excel® installed (or another spreadsheet program cable of opening `.xslx` format spreadsheet files), the spreadsheet of the bar bending schedule opens.
   g. Close your spreadsheet application.
   h. Click the **Save** tool in the view window.
   i. Type `Tutorial_Beam_BBS.dxf` and click **Save** to save the schedule as a DXF file.

To generate beam design calculations

1. Either:

   select the **Design Summary** tool

   or

   select **Reports** > **Design Summary**

   The **Design Summary** opens.
2. Either:

select the **Design Calculations** tool

or

select **Reports** > **Design Calculations**

The **Select Beams** opens.

3. Select **1.B1-B2-B3-B4** and **2. B5-B6-B7-B8-B9** in the list of beam groups.
   For this tutorial you will only output the detailed design for two beam groups. You can check the **All Beams** option to include the full design report.
4. Click **OK**.
5. Either:

   click the **Print** tool to print out a copy of this report

   or

   click the **Save** tool to save an HTML copy of this report

To save the project and exit

1. Either:

   select the **Save** tool (on the main program toolbar)

   or

   select **File** > **Save**

   The **Save As** dialog opens.
2. Type `Sample1_Beam_tutorial.rcdx` in the filename and click **Save**.
3. Select **File** > **Exit**.
   RCDC closes and you are returned to STAAD.Pro.

Column and Wall Design

This section of the tutorial demonstrates how to design and detail continuous concrete columns.

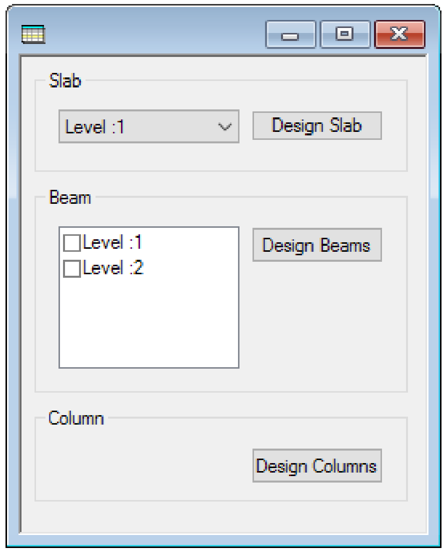
RCDC will make continuous physical columns out of collinear analytical columns modeled in the Building Planner mode in STAAD.Pro.

To start the column design

In STAAD.Pro, ensure you are on the **Frame | Design** page in the **Building Planner** mode.

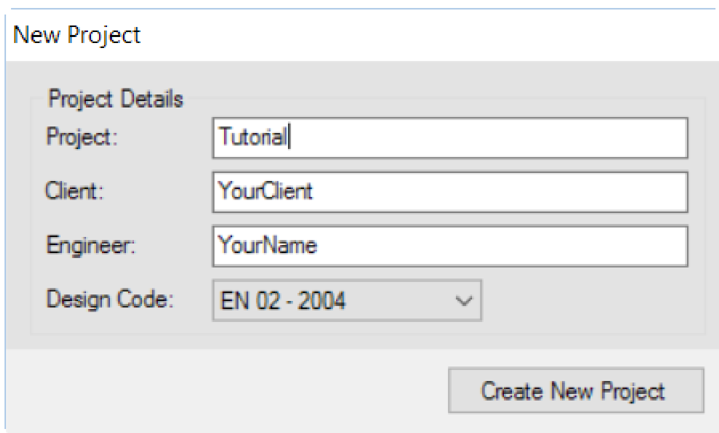
Tip: When you import STAAD.Pro model data into RCDC, grids are automatically detected and assigned. You can manually edit both the exact locations of grids as well as grid labels in RCDC for drawing generation.

1. In the top-right dialog, click **Design Columns**.



RCDC opens and the **New Project** dialog opens.

2. Enter the project data:



   a. Type `Tutorial` in the **Project** field.

   b. (Optional) Type **Client** and **Engineer** name.

      This information is used as the header for all reports. It will be automatically imported from the STAAD.Pro input file Job Information block if available.

   c. Select **EN 02 - 2004** from the **Design Code** drop-down list.
      This tutorial uses the Eurocode. However, you can select any available code. Just note that some design settings and features may vary based on the code selection made here.

   The column continuity is automatically detected as the analytical model data is read.

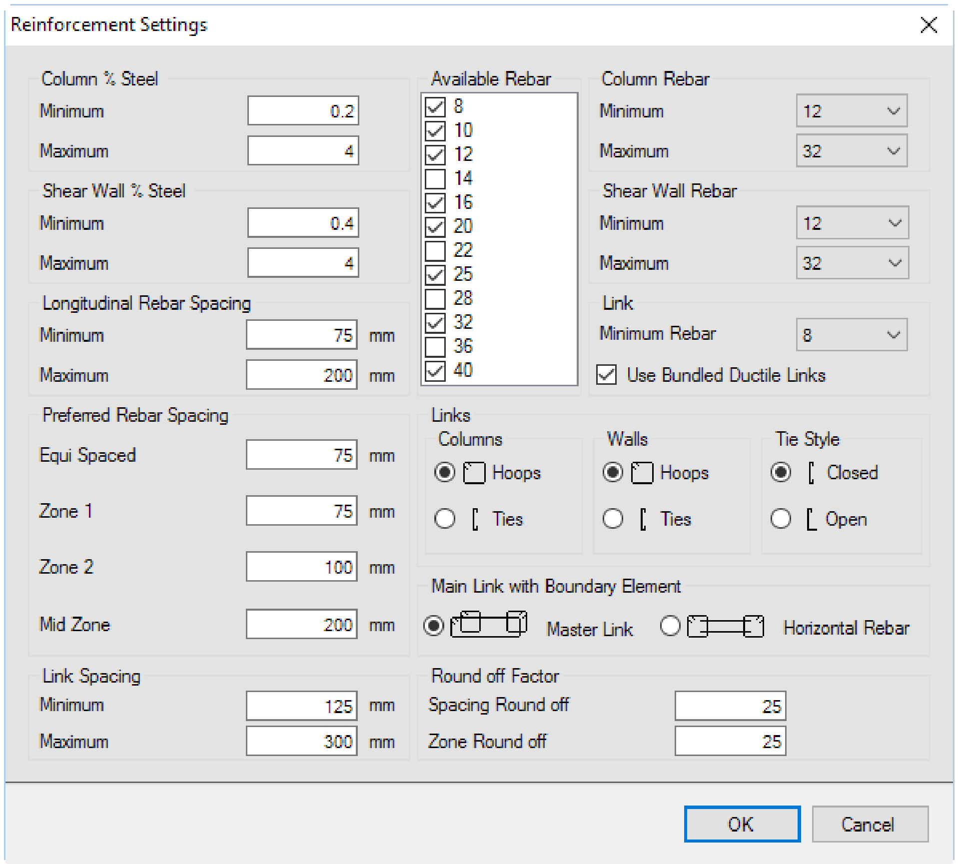
3. Either:

   select the **Reinforcement Settings** tool

   or

   select **Settings** > **Reinforcement Settings**

   The **Reinforcement Settings** dialog opens.



4. In the **Rebar** list, clear the check mark by both **32** (mm) and **40** (mm).
   The initial design will be evaluated using smaller size bars where possible.
5. Click **OK**.

Note: There are numerous other detailing and drawing settings in RCDC that allow you to customize the concrete detailing for your client or organization's needs. For details on these options, refer to the RCDC help by pressing **<F1>** or selecting **Help** > **Contents**.

To import load cases and combinations

In this procedure, you will import the load settings file you saved when To specify load cases and combinations (on page 6918).

1. Select **Settings** > **Basic Load Cases**.
   The **Basic Load Cases** dialog opens.
2. Click **Import Load Cases & Combinations**.
   The **Open** dialog opens.
3. Select the file `Sample1_Loads.ldsx` and then click **Open**.
   If you saved this file under a different name or in a different location, locate and select the file you created in the previous section.
   The previously selected load cases are added to the Primary Load Cases list.
4. Click **OK**.

The **Load Combinations** dialog opens. The previously selected load combinations from the analysis model are also already added.

5. Click **OK**.

To perform the column design

1. Either:

   select the **Autodesign** tool

   or

   select **Design** > **Autodesign**.

   The **Failed Columns Dialog** dialog opens indicating that two columns have failed the design checks.
2. Click **OK**.
3. Either:

   select the **Failure Diagnostics** tool

   or

   select **Reports** > **Failure Diagnostics**

   The **Failure Diagnostics** tab opens. The table indicates that the two beams have failed a detailing check.
4. Evaluate some options for column 12:

   a. Select the row for column **C12** from 6m TO 12m in the column design table.

   This column is marked with a red tag to indicate it failed one or more checks.

   b. Either:

   select the **Redesign Section** tool

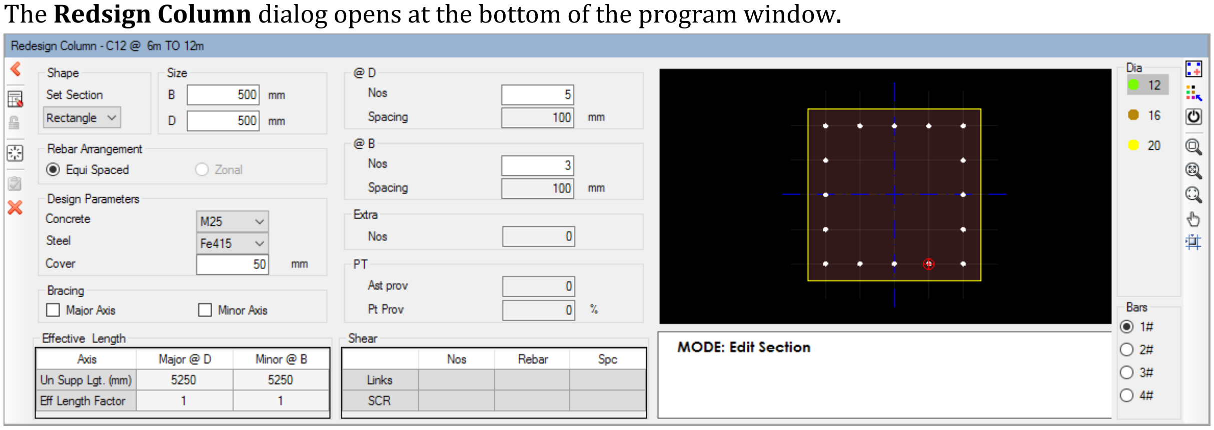
   or

   select **Modify** > **Redesign Section**

   or

   right-click on the column row and select **Redesign Section** from the pop-up menu

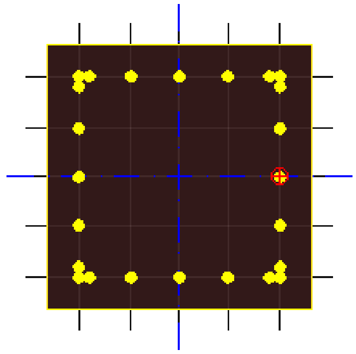
   The **Redsign Column** dialog opens at the bottom of the program window.



   c. In the Dia list, select **20** (mm) and then in the Bars list select **3#**.

   This will place 20mm bars in groups of three in the section.

d. Click on any of the *corner* bars in the cross section.
   The color-coded size of the grouped bars are added to each corner.
e. Repeat steps 4c and 4d to add single (1#) 20mm bars along the faces.



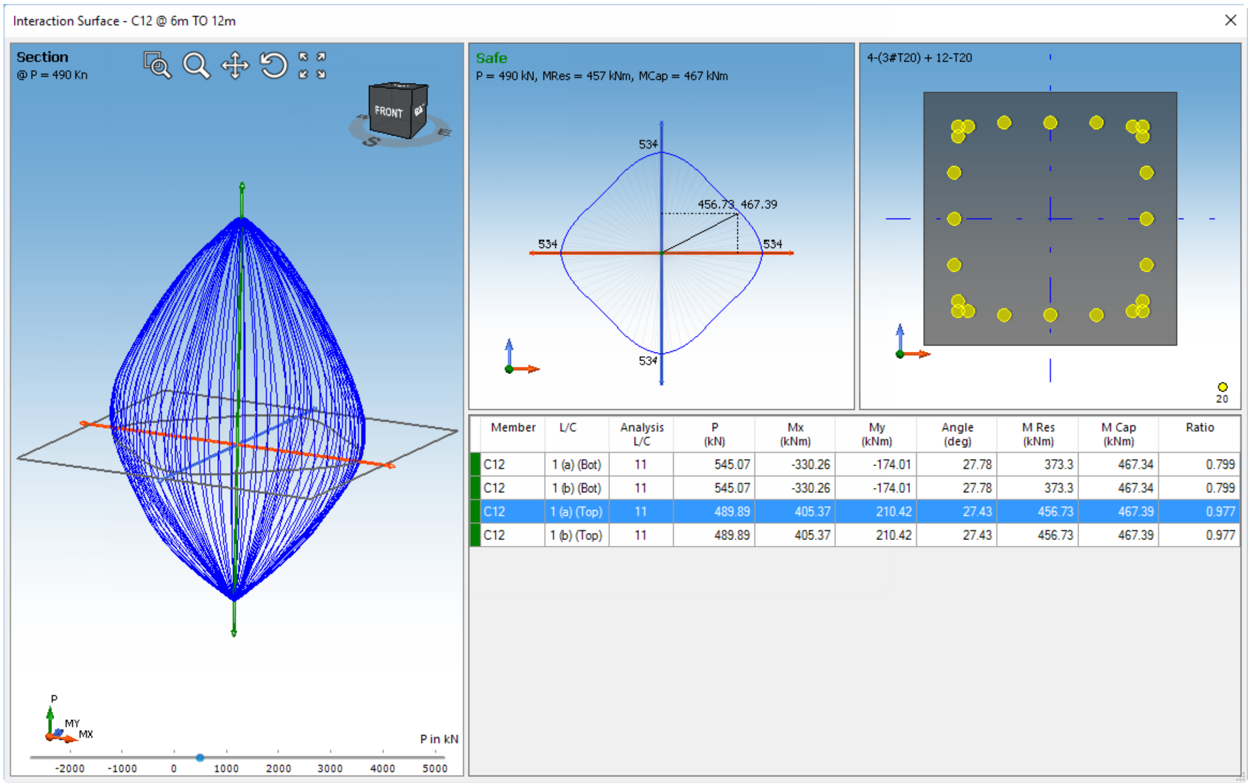
> **Note:** Once you select number of bars, you only need to click twice in the section to update all the face bars.

f. Select the **Redesign** tool in the **Redesign Grp** dialog.
   The column now passes design and detailing checks.
g. Select the **Accept** tool.

> **Tip:** Right-clicking on a wall and selecting **Redesign Section** allows you to graphically detail a wall section in the same manner. Try this with W1 in the Column Design table as an extra excercise.

The Design Output table now shows green for the G1 beam indicating it passes.

5. Repeat step 4 to redesign the other failing column (C23).
6. Right-click on the second level column 12 (C12: 6m TO 12m) and select **Interaction Surface from the pop-up menu.**
   The **Interaction Surface** dialog opens.



7. Review the interaction surface:

Options	**Description**
To display the interaction at critical points for a load combination	select that combination from the table
To view the interaction at any point along the column height	drag the slider to a specific load value
To control the view of the interaction surface	use the view controls at the top of the interaction surface view window

8. Click the **X** in top-right to close the **Interaction Surface** window.

To generate column schedules and drawings

1. Generate a text schedule of column reinforcement:
   a. Either:

      select the **Text Schedule** tool

      or

      Select **Reports** > **Text Schedule** > **Type 1 (Ductile) Section**.

      The **Text Schedule** view window opens.

   b. Click the **Save** tool in the view window.

   c. Type `Tutorial_Column_Sched.dxf` and click **Save** to save the schedule as a DXF file.
2. Generate detail drawings of columns:

a. Either:

   select the **Detailed Drawing** tool

   or

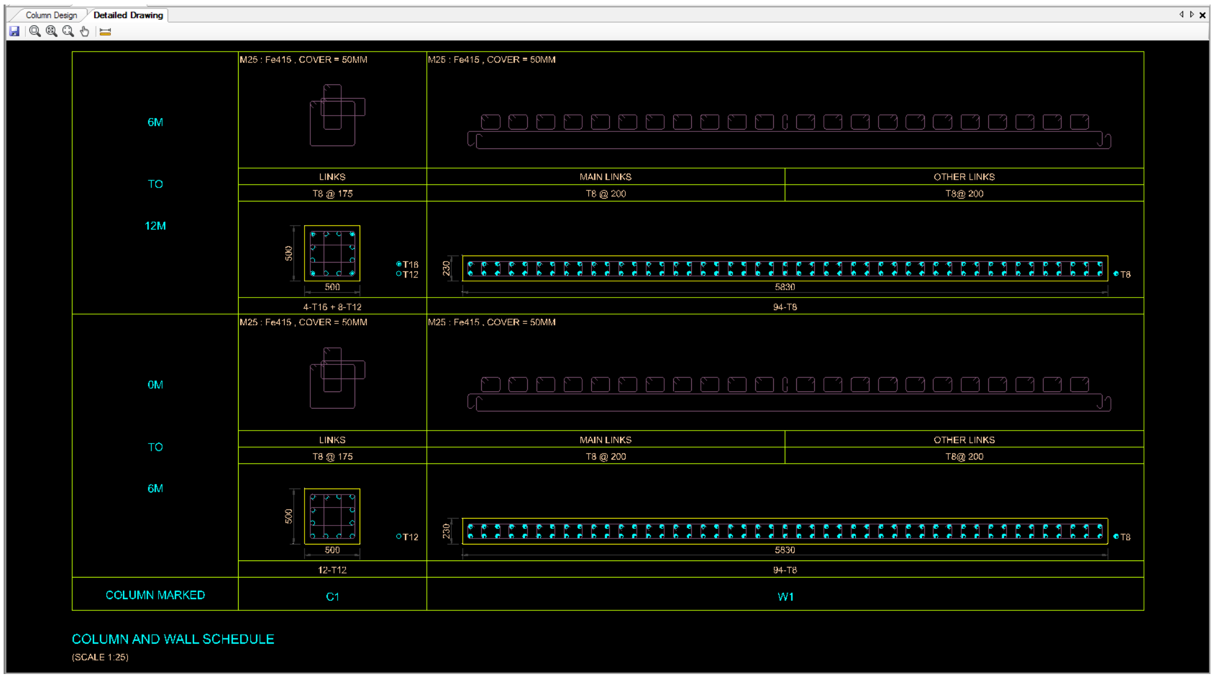
   Select **Reports** > **Detailed Drawing**.

   The **Select Columns** dialog opens.

b. Select 0m to 12m for the range of heights.

c. Select **C1** and **W1** in the columns list.

d. Click **OK**.
   The **Detailed Drawing** opens with the Column and Wall Schedule.



e. Click the **Save** tool in the view window.

f. Type `Tutorial_Col_Detail.dxf` and click **Save** to save the schedule as a DXF file.

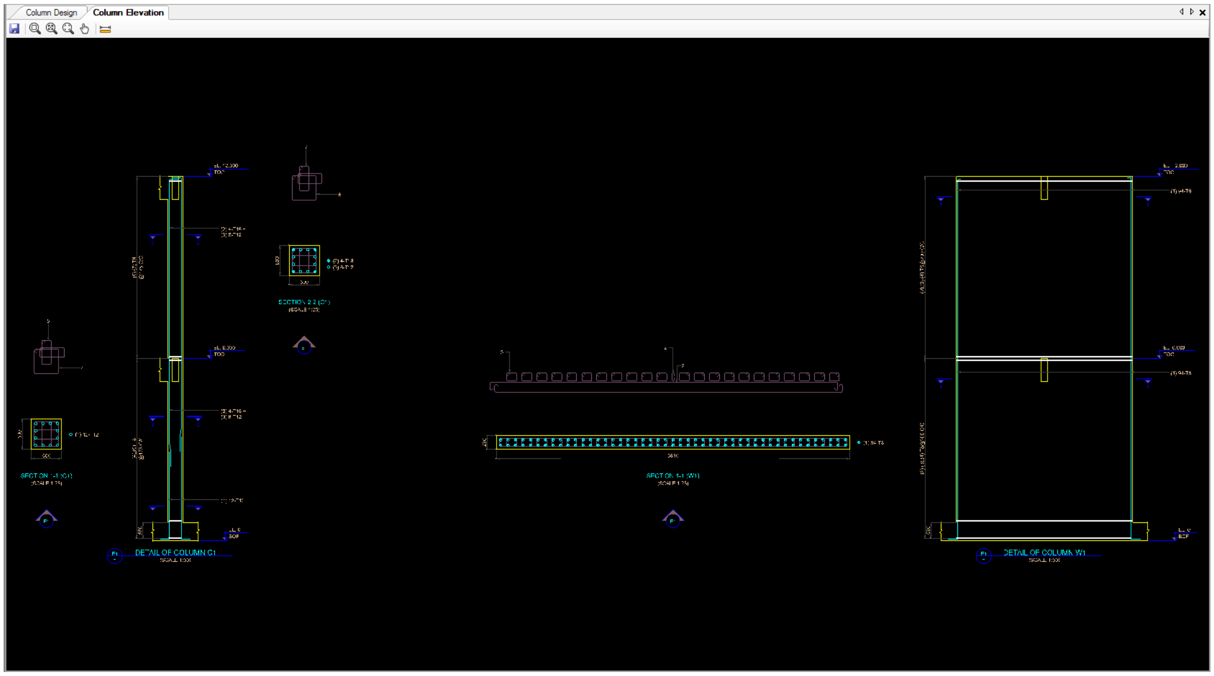
3. Generate the elevation and sections for a column:

   a. Select **Reports** > **Elevation**.
      The **Select Columns** dialog opens.

   b. Select **C1** and **W1** in the columns list.

   Tip: You can modify the drawing and detailing options by clicking the **Detailing & Drawing Settings** button (also by selecting **Settings** > **Detailing & Drawing Settings** before using this tool). You can also customize rebar mark styles from this dialog.

   c. Click **OK**.
      The elevation for the selected column and wall opens. This drawing contains section cuts at each reinforcement arrangement.



d. Click the **Save** tool in the view window.

e. Type `Tutorial_Col_Elev.dxf` and click **Save** to save the schedule as a DXF file.

4. Generate a bill of quantities:

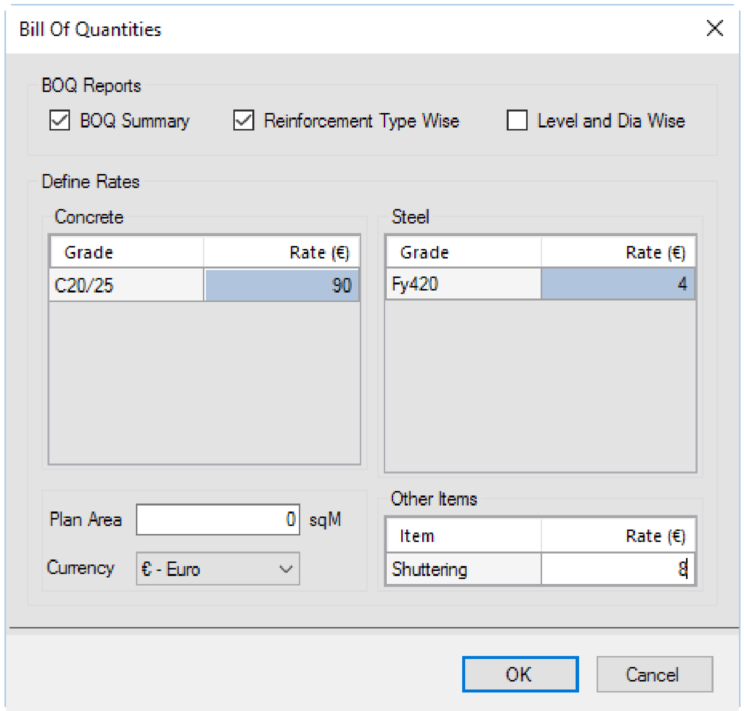
   a. Either:

      select the **Bill of Quantities** tool

      or

      Select **Reports** > **Bill of Quantities**.

      The **Bill Of Quantities** dialog opens.



b. Select the **BOQ Summary** and **Reinforcement Type Wise** options.

c. Select **€ - Euro** from the **Currency** drop-down list.

d. Type `90` (€/m^3) for Grade C20/25 concrete and `4` (€/kg) for Grade Fy420 steel. Type `8` (€/m^2) for the Shuttering value.

e. Click **OK**.
The **BOQ Summary** and **BOQ Detailed - Reinf Type Wise** tabs open.

f. Either:

click the **Print** tool to print out a copy of this report

or

click the **Save** tool to save an HTML copy of this report

5. Generate bar bending schedule:

a. Select **BBS** > **Generate BBS**.
The **Generate BBS** dialog opens.

b. Select **C1** and **W1** in the list of columns groups.

c. Select the **Continuous** option in the **Rebar Numbering** group.

d. Select both the **Drawing** and **Spreadsheet** options in the **Schedule Format** group.

e. Click inside the file path field, type `Sample1_Col_BBS` in the **Save As** dialog that opens, and then click **Save**.

f. Click **Generate**.
The BBS opens containing a text schedule of the bar bending schedule. If you have Microsoft® Office Excel® installed (or another spreadsheet program cable of opening `.xslx` format spreadsheet files), the spreadsheet of the bar bending schedule opens.

g. Close your spreadsheet application.

h. Click the **Save** tool in the view window.

i. Type `Tutorial_Col_BBS.dxf` and click **Save** to save the schedule as a DXF file.

To generate column design calculations

1. Either:

   select the **Design Summary** tool

   or

   select **Reports** > **Design Summary**

   The **Design Summary** opens.
2. Either:

   select the **Design Calculations** tool

   or

   select **Reports** > **Design Calculations**

   The **Select Columns** dialog opens.
3. Select columns **C1** and **W1** in the columns list.
   For this tutorial you will only output the detailed design for one column and one wall. You can check the **All Columns** option to include the full design report. Or, if you only need the calculations for one level of a column or wall, you can expand that entry and select the level of interest.
4. Click **OK**.
5. Either:

   click the **Print** tool to print out a copy of this report

   or

   click the **Save** tool to save an HTML copy of this report

To save the project and exit

1. Either:

   select the **Save** tool (on the main program toolbar)

   or

   select **File** > **Save**

   The **Save As** dialog opens.
2. Type `Sample1_Col_tutorial.rcdx` in the filename and click **Save**.
3. Select **File** > **Exit**.
   RCDC closes and you are returned to STAAD.Pro.

Tutorial STAAD Input File

The following input file for STAAD.Pro is included with the installation of the product. It is included here as well.

```
STAAD SPACE ...\Typical Ground +3 Storey Building.std
** FILE CREATED ON : (19-09-2018, 02:49:43 PM)
** FILE IS GENERATED USING PlanWin
** PlanWin FILE : C:\Users\Public\Documents\STAAD.Pro CONNECT Edition\Samples\Sample
Models\Building Planner\Typical Ground +3 Storey Building.plnx
UNIT METERS MTONS
JOINT COORDINATES
```

```
1          0.0000    0.0000  -32.6100
2          8.0000    0.0000  -32.6100
3         16.0000    0.0000  -32.6100
4         24.0000    0.0000  -32.6100
5         32.0000    0.0000  -32.6100
6          0.0000    0.0000  -26.9000
7          8.0000    0.0000  -26.9000
8         16.0000    0.0000  -26.9000
9         24.0000    0.0000  -26.9000
10        32.0000    0.0000  -26.9000
11        33.8700    0.0000  -26.9000
12        46.0800    0.0000  -21.7400
13         0.0000    0.0000  -18.0000
14         8.0000    0.0000  -18.0000
15        16.0000    0.0000  -18.0000
16        24.0000    0.0000  -18.0000
17        36.7900    0.0000  -18.0000
18        37.8500    0.0000  -14.7500
19        49.0600    0.0000  -13.2500
20         0.0000    0.0000   -8.9500
21         8.0000    0.0000   -8.9500
22        16.0000    0.0000   -8.9500
23        24.0000    0.0000   -8.9500
24        32.0000    0.0000   -8.9500
25        37.8300    0.0000   -8.9500
26        39.7500    0.0000   -8.9500
27        32.0000    0.0000   -6.4600
28        37.8300    0.0000   -6.4600
29         0.0000    0.0000    0.0000
30         8.0000    0.0000    0.0000
31        16.0000    0.0000    0.0000
32        24.0000    0.0000    0.0000
33        32.0000    0.0000    0.0000
34        37.8300    0.0000    0.0000
35        42.6800    0.0000    0.0000
36        32.0005    0.0000  -16.3750
37        33.1805    0.0000  -18.0000
38        34.3605    0.0000  -16.3750
39         0.0000    3.0000  -32.6100
40         8.0000    3.0000  -32.6100
41        16.0000    3.0000  -32.6100
42        24.0000    3.0000  -32.6100
43        32.0000    3.0000  -32.6100
44         0.0000    3.0000  -26.9000
45         8.0000    3.0000  -26.9000
46        16.0000    3.0000  -26.9000
47        24.0000    3.0000  -26.9000
48        32.0000    3.0000  -26.9000
49        33.8700    3.0000  -26.9000
50        46.0800    3.0000  -21.7400
51         0.0000    3.0000  -18.0000
52         8.0000    3.0000  -18.0000
53        16.0000    3.0000  -18.0000
54        24.0000    3.0000  -18.0000
55        32.0010    3.0000  -18.0000
56        34.3600    3.0000  -18.0000
57        36.7900    3.0000  -18.0000
58        32.0000    3.0000  -14.7500
```

```
59        34.3610    3.0000  -14.7500
60        37.8500    3.0000  -14.7500
61        49.0600    3.0000  -13.2500
62         0.0000    3.0000   -8.9500
63         8.0000    3.0000   -8.9500
64        16.0000    3.0000   -8.9500
65        24.0000    3.0000   -8.9500
66        32.0000    3.0000   -8.9500
67        37.8300    3.0000   -8.9500
68        39.7500    3.0000   -8.9500
69        32.0000    3.0000   -6.4600
70        37.8300    3.0000   -6.4600
71         0.0000    3.0000    0.0000
72         8.0000    3.0000    0.0000
73        16.0000    3.0000    0.0000
74        24.0000    3.0000    0.0000
75        32.0000    3.0000    0.0000
76        37.8300    3.0000    0.0000
77        42.6800    3.0000    0.0000
78        32.0005    3.0000  -16.3750
79        33.1805    3.0000  -18.0000
80        34.3605    3.0000  -16.3750
81         0.0000    6.0000  -32.6100
82         8.0000    6.0000  -32.6100
83        16.0000    6.0000  -32.6100
84        24.0000    6.0000  -32.6100
85        32.0000    6.0000  -32.6100
86         0.0000    6.0000  -26.9000
87         8.0000    6.0000  -26.9000
88        16.0000    6.0000  -26.9000
89        24.0000    6.0000  -26.9000
90        32.0000    6.0000  -26.9000
91        33.8700    6.0000  -26.9000
92        46.0800    6.0000  -21.7400
93         0.0000    6.0000  -18.0000
94         8.0000    6.0000  -18.0000
95        16.0000    6.0000  -18.0000
96        24.0000    6.0000  -18.0000
97        32.0010    6.0000  -18.0000
98        34.3600    6.0000  -18.0000
99        36.7900    6.0000  -18.0000
100       32.0000    6.0000  -14.7500
101       34.3610    6.0000  -14.7500
102       37.8500    6.0000  -14.7500
103       49.0600    6.0000  -13.2500
104        0.0000    6.0000   -8.9500
105        8.0000    6.0000   -8.9500
106       16.0000    6.0000   -8.9500
107       24.0000    6.0000   -8.9500
108       32.0000    6.0000   -8.9500
109       37.8300    6.0000   -8.9500
110       39.7500    6.0000   -8.9500
111       32.0000    6.0000   -6.4600
112       37.8300    6.0000   -6.4600
113        0.0000    6.0000    0.0000
114        8.0000    6.0000    0.0000
115       16.0000    6.0000    0.0000
116       24.0000    6.0000    0.0000
```

```
117       32.0000    6.0000    0.0000
118       37.8300    6.0000    0.0000
119       42.6800    6.0000    0.0000
120       32.0005    6.0000  -16.3750
121       33.1805    6.0000  -18.0000
122       34.3605    6.0000  -16.3750
123        0.0000    9.0000  -32.6100
124        8.0000    9.0000  -32.6100
125       16.0000    9.0000  -32.6100
126       24.0000    9.0000  -32.6100
127       32.0000    9.0000  -32.6100
128        0.0000    9.0000  -26.9000
129        8.0000    9.0000  -26.9000
130       16.0000    9.0000  -26.9000
131       24.0000    9.0000  -26.9000
132       32.0000    9.0000  -26.9000
133       33.8700    9.0000  -26.9000
134       46.0800    9.0000  -21.7400
135        0.0000    9.0000  -18.0000
136        8.0000    9.0000  -18.0000
137       16.0000    9.0000  -18.0000
138       24.0000    9.0000  -18.0000
139       32.0010    9.0000  -18.0000
140       34.3600    9.0000  -18.0000
141       36.7900    9.0000  -18.0000
142       32.0000    9.0000  -14.7500
143       34.3610    9.0000  -14.7500
144       37.8500    9.0000  -14.7500
145       49.0600    9.0000  -13.2500
146        0.0000    9.0000   -8.9500
147        8.0000    9.0000   -8.9500
148       16.0000    9.0000   -8.9500
149       24.0000    9.0000   -8.9500
150       32.0000    9.0000   -8.9500
151       37.8300    9.0000   -8.9500
152       39.7500    9.0000   -8.9500
153       32.0000    9.0000   -6.4600
154       37.8300    9.0000   -6.4600
155        0.0000    9.0000    0.0000
156        8.0000    9.0000    0.0000
157       16.0000    9.0000    0.0000
158       24.0000    9.0000    0.0000
159       32.0000    9.0000    0.0000
160       37.8300    9.0000    0.0000
161       42.6800    9.0000    0.0000
162       32.0005    9.0000  -16.3750
163       33.1805    9.0000  -18.0000
164       34.3605    9.0000  -16.3750
165        0.0000   12.0000  -32.6100
166        8.0000   12.0000  -32.6100
167       16.0000   12.0000  -32.6100
168       24.0000   12.0000  -32.6100
169       32.0000   12.0000  -32.6100
170        0.0000   12.0000  -26.9000
171        8.0000   12.0000  -26.9000
172       16.0000   12.0000  -26.9000
173       24.0000   12.0000  -26.9000
174       32.0000   12.0000  -26.9000
```

```
 175      33.8700   12.0000  -26.9000
 176       0.0000   12.0000  -18.0000
 177       8.0000   12.0000  -18.0000
 178      16.0000   12.0000  -18.0000
 179      24.0000   12.0000  -18.0000
 180      32.0010   12.0000  -18.0000
 181      34.3600   12.0000  -18.0000
 182      36.7900   12.0000  -18.0000
 183      32.0000   12.0000  -14.7500
 184      34.3610   12.0000  -14.7500
 185      37.8500   12.0000  -14.7500
 186       0.0000   12.0000   -8.9500
 187       8.0000   12.0000   -8.9500
 188      16.0000   12.0000   -8.9500
 189      24.0000   12.0000   -8.9500
 190      32.0000   12.0000   -8.9500
 191      37.8300   12.0000   -8.9500
 192      39.7500   12.0000   -8.9500
 193      32.0000   12.0000   -6.4600
 194      37.8300   12.0000   -6.4600
 195       0.0000   12.0000    0.0000
 196       8.0000   12.0000    0.0000
 197      16.0000   12.0000    0.0000
 198      24.0000   12.0000    0.0000
 199      32.0000   12.0000    0.0000
 200      37.8300   12.0000    0.0000
 201      42.6800   12.0000    0.0000
 202      32.0005   12.0000  -16.3750
 203      33.1805   12.0000  -18.0000
 204      34.3605   12.0000  -16.3750
MEMBER INCIDENCES
*Columns at Level 1
 1 1  39
 2 2  40
 3 3  41
 4 4  42
 5 5  43
 6 6  44
 7 7  45
 8 8  46
 9 9  47
 10 10  48
 11 11  49
 12 12  50
 13 13  51
 14 14  52
 15 15  53
 16 16  54
 17 17  57
 18 18  60
 19 19  61
 20 20  62
 21 21  63
 22 22  64
 23 23  65
 24 24  66
 25 25  67
 26 26  68
```

```
 27 27  69
 28 28  70
 29 29  71
 30 30  72
 31 31  73
 32 32  74
 33 33  75
 34 34  76
 35 35  77
 36 36  78
 37 37  79
 38 38  80
*Columns at Level 2
 39 39  81
 40 40  82
 41 41  83
 42 42  84
 43 43  85
 44 44  86
 45 45  87
 46 46  88
 47 47  89
 48 48  90
 49 49  91
 50 50  92
 51 51  93
 52 52  94
 53 53  95
 54 54  96
 55 57  99
 56 60  102
 57 61  103
 58 62  104
 59 63  105
 60 64  106
 61 65  107
 62 66  108
 63 67  109
 64 68  110
 65 69  111
 66 70  112
 67 71  113
 68 72  114
 69 73  115
 70 74  116
 71 75  117
 72 76  118
 73 77  119
 74 78  120
 75 79  121
 76 80  122
*Columns at Level 3
 77 81  123
 78 82  124
 79 83  125
 80 84  126
 81 85  127
 82 86  128
```

```
 83 87  129
 84 88  130
 85 89  131
 86 90  132
 87 91  133
 88 92  134
 89 93  135
 90 94  136
 91 95  137
 92 96  138
 93 99  141
 94 102  144
 95 103  145
 96 104  146
 97 105  147
 98 106  148
 99 107  149
 100 108  150
 101 109  151
 102 110  152
 103 111  153
 104 112  154
 105 113  155
 106 114  156
 107 115  157
 108 116  158
 109 117  159
 110 118  160
 111 119  161
 112 120  162
 113 121  163
 114 122  164
*Columns at Level 4
 115 123  165
 116 124  166
 117 125  167
 118 126  168
 119 127  169
 120 128  170
 121 129  171
 122 130  172
 123 131  173
 124 132  174
 125 133  175
 126 135  176
 127 136  177
 128 137  178
 129 138  179
 130 141  182
 131 144  185
 132 146  186
 133 147  187
 134 148  188
 135 149  189
 136 150  190
 137 151  191
 138 152  192
 139 153  193
```

```
 140 154  194
 141 155  195
 142 156  196
 143 157  197
 144 158  198
 145 159  199
 146 160  200
 147 161  201
 148 162  202
 149 163  203
 150 164  204
*Beams at Level 1
 151 39  40
 152 40  41
 153 41  42
 154 42  43
 155 44  45
 156 45  46
 157 46  47
 158 47  48
 159 48  49
 160 51  52
 161 52  53
 162 53  54
 163 54  55
 164 55  79
 165 56  57
 166 58  59
 167 59  60
 168 62  63
 169 63  64
 170 64  65
 171 65  66
 172 66  67
 173 67  68
 174 69  70
 175 71  72
 176 72  73
 177 73  74
 178 74  75
 179 75  76
 180 76  77
 181 71  62
 182 62  51
 183 51  44
 184 44  39
 185 72  63
 186 63  52
 187 52  45
 188 45  40
 189 73  64
 190 64  53
 191 53  46
 192 46  41
 193 74  65
 194 65  54
 195 54  47
 196 47  42
```

```
 197 75  69
 198 69  66
 199 66  58
 200 58  78
 201 55  48
 202 48  43
 203 59  80
 204 76  70
 205 70  67
 206 57  50
 207 68  61
 208 57  60
 209 43  49
 210 68  77
 211 60  68
 212 49  57
 213 50  61
 214 78  55
 215 79  56
 216 80  56
*Beams at Level 2
 217 81  82
 218 82  83
 219 83  84
 220 84  85
 221 86  87
 222 87  88
 223 88  89
 224 89  90
 225 90  91
 226 93  94
 227 94  95
 228 95  96
 229 96  97
 230 97  121
 231 98  99
 232 100  101
 233 101  102
 234 104  105
 235 105  106
 236 106  107
 237 107  108
 238 108  109
 239 109  110
 240 111  112
 241 113  114
 242 114  115
 243 115  116
 244 116  117
 245 117  118
 246 118  119
 247 113  104
 248 104  93
 249 93  86
 250 86  81
 251 114  105
 252 105  94
 253 94  87
```

```
 254 87  82
 255 115  106
 256 106  95
 257 95  88
 258 88  83
 259 116  107
 260 107  96
 261 96  89
 262 89  84
 263 117  111
 264 111  108
 265 108  100
 266 100  120
 267 97  90
 268 90  85
 269 101  122
 270 118  112
 271 112  109
 272 99  92
 273 110  103
 274 99  102
 275 85  91
 276 110  119
 277 102  110
 278 91  99
 279 92  103
 280 120  97
 281 121  98
 282 122  98
*Beams at Level 3
 283 123  124
 284 124  125
 285 125  126
 286 126  127
 287 128  129
 288 129  130
 289 130  131
 290 131  132
 291 132  133
 292 135  136
 293 136  137
 294 137  138
 295 138  139
 296 139  163
 297 140  141
 298 142  143
 299 143  144
 300 146  147
 301 147  148
 302 148  149
 303 149  150
 304 150  151
 305 151  152
 306 153  154
 307 155  156
 308 156  157
 309 157  158
 310 158  159
```

```
 311 159  160
 312 160  161
 313 155  146
 314 146  135
 315 135  128
 316 128  123
 317 156  147
 318 147  136
 319 136  129
 320 129  124
 321 157  148
 322 148  137
 323 137  130
 324 130  125
 325 158  149
 326 149  138
 327 138  131
 328 131  126
 329 159  153
 330 153  150
 331 150  142
 332 142  162
 333 139  132
 334 132  127
 335 143  164
 336 160  154
 337 154  151
 338 141  134
 339 152  145
 340 141  144
 341 127  133
 342 152  161
 343 144  152
 344 133  141
 345 134  145
 346 162  139
 347 163  140
 348 164  140
*Beams at Level 4
 349 165  166
 350 166  167
 351 167  168
 352 168  169
 353 170  171
 354 171  172
 355 172  173
 356 173  174
 357 174  175
 358 176  177
 359 177  178
 360 178  179
 361 179  180
 362 180  203
 363 181  182
 364 183  184
 365 184  185
 366 186  187
 367 187  188
```

```
 368 188  189
 369 189  190
 370 190  191
 371 191  192
 372 193  194
 373 195  196
 374 196  197
 375 197  198
 376 198  199
 377 199  200
 378 200  201
 379 195  186
 380 186  176
 381 176  170
 382 170  165
 383 196  187
 384 187  177
 385 177  171
 386 171  166
 387 197  188
 388 188  178
 389 178  172
 390 172  167
 391 198  189
 392 189  179
 393 179  173
 394 173  168
 395 199  193
 396 193  190
 397 190  183
 398 183  202
 399 180  174
 400 174  169
 401 184  204
 402 200  194
 403 194  191
 404 182  185
 405 169  175
 406 192  201
 407 185  192
 408 175  182
 409 202  180
 410 203  181
 411 204  181
DEFINE MATERIAL START
 ISOTROPIC M20
 E 2236067.97749979
 POISSON 0.17
 DENSITY 2.5
 ALPHA 1e-005
 DAMP 0.05
 ISOTROPIC RL20
 E 2236067.97749979
 POISSON 0.17
 DENSITY 0
 ALPHA 1e-005
 DAMP 0.05
END DEFINE MATERIAL
```

```
MEMBER PROPERTIES
*Columns
1 5 35 39 43 73 PRIS YD 0.23 ZD 0.3
2 4 33 40 42 71 103 139 PRIS YD 0.23 ZD 0.7
3 41 95 104 140 PRIS YD 0.23 ZD 0.55
6 44 107 143 PRIS YD 0.23 ZD 0.85
7 TO 9 14 TO 16 21 TO 23 45 TO 47 52 TO 54 59 TO 61 -
 PRIS YD 0.6 ZD 0.6
10 48 PRIS YD 0.23 ZD 1.3
11 25 49 63 77 81 87 101 111 115 119 125 137 147 PRIS YD 0.23 ZD 0.25
12 50 78 80 109 116 118 145 PRIS YD 0.23 ZD 0.6
13 32 51 70 96 132 PRIS YD 0.23 ZD 1
17 55 82 120 PRIS YD 0.23 ZD 0.75
18 56 79 117 PRIS YD 0.23 ZD 0.45
19 28 57 66 93 130 PRIS YD 0.23 ZD 0.65
20 58 PRIS YD 0.23 ZD 1.1
24 26 62 64 89 108 126 144 PRIS YD 0.23 ZD 0.9
27 65 100 102 136 138 PRIS YD 0.23 ZD 0.8
29 34 67 72 88 PRIS YD 0.23 ZD 0.5
30 68 PRIS YD 0.23 ZD 1.05
31 69 106 142 PRIS YD 0.23 ZD 0.95
36 38 74 76 112 114 148 150 PRIS YD 0.23 ZD 3.25000015367718
37 75 113 149 PRIS YD 0.23 ZD 2.35900115966797
83 TO 85 90 TO 92 97 TO 99 PRIS YD 0.55 ZD 0.55
86 124 PRIS YD 0.23 ZD 1.2
94 105 110 131 141 146 PRIS YD 0.23 ZD 0.4
121 TO 123 127 TO 129 133 TO 135 PRIS YD 0.5 ZD 0.5

*Beams
151 TO 163 165 TO 199 201 TO 202 204 TO 213 217 TO 229 -
 231 TO 265 267 TO 268 270 TO 279 283 TO 295 297 TO 331 -
 333 TO 334 336 TO 345 349 TO 361 363 TO 397 399 TO 400 -
 402 TO 408 PRIS YD 0.75 ZD 0.23
164 200 203 214 TO 216 230 266 269 280 TO 282 296 332 -
 335 346 TO 348 362 398 401 409 TO 411 PRIS YD 3 ZD 0.23

MEMBER RELEASE
CONSTANTS
 MATERIAL M20 MEMB 1 TO 150
 MATERIAL M20 MEMB 151 TO 163
 MATERIAL M20 MEMB 165 TO 199
 MATERIAL M20 MEMB 201 TO 202
 MATERIAL M20 MEMB 204 TO 213
 MATERIAL M20 MEMB 217 TO 229
 MATERIAL M20 MEMB 231 TO 265
 MATERIAL M20 MEMB 267 TO 268
 MATERIAL M20 MEMB 270 TO 279
 MATERIAL M20 MEMB 283 TO 295
 MATERIAL M20 MEMB 297 TO 331
 MATERIAL M20 MEMB 333 TO 334
 MATERIAL M20 MEMB 336 TO 345
 MATERIAL M20 MEMB 349 TO 361
 MATERIAL M20 MEMB 363 TO 397
 MATERIAL M20 MEMB 399 TO 400
 MATERIAL M20 MEMB 402 TO 408
 MATERIAL RL20 MEMB 164
 MATERIAL RL20 MEMB 200
 MATERIAL RL20 MEMB 203
```

```
 MATERIAL RL20 MEMB 214 TO 216
 MATERIAL RL20 MEMB 230
 MATERIAL RL20 MEMB 266
 MATERIAL RL20 MEMB 269
 MATERIAL RL20 MEMB 280 TO 282
 MATERIAL RL20 MEMB 296
 MATERIAL RL20 MEMB 332
 MATERIAL RL20 MEMB 335
 MATERIAL RL20 MEMB 346 TO 348
 MATERIAL RL20 MEMB 362
 MATERIAL RL20 MEMB 398
 MATERIAL RL20 MEMB 401
 MATERIAL RL20 MEMB 409 TO 411
*Columns Beta Angle
 BETA 270 MEM 1
 BETA 180 MEM 2 TO 4
 BETA 198 MEM 5
 BETA 270 MEM 6 TO 10
 BETA 198 MEM 11
 BETA 112 MEM 12
 BETA 270 MEM 13 TO 16
 BETA 18 MEM 17 TO 18
 BETA 115 MEM 19
 BETA 270 MEM 20 TO 25
 BETA 18 MEM 26
 BETA 270 MEM 27 TO 29
 BETA 18 MEM 35
 BETA 270 MEM 37
 BETA 180 MEM 38
 BETA 270 MEM 39
 BETA 180 MEM 40 TO 42
 BETA 198 MEM 43
 BETA 270 MEM 44 TO 48
 BETA 198 MEM 49
 BETA 112 MEM 50
 BETA 270 MEM 51 TO 54
 BETA 18 MEM 55 TO 56
 BETA 115 MEM 57
 BETA 270 MEM 58 TO 63
 BETA 18 MEM 64
 BETA 270 MEM 65 TO 67
 BETA 18 MEM 73
 BETA 270 MEM 75
 BETA 180 MEM 76
 BETA 270 MEM 77
 BETA 180 MEM 78 TO 80
 BETA 198 MEM 81
 BETA 270 MEM 82 TO 86
 BETA 198 MEM 87
 BETA 112 MEM 88
 BETA 270 MEM 89 TO 92
 BETA 18 MEM 93 TO 94
 BETA 115 MEM 95
 BETA 270 MEM 96 TO 101
 BETA 18 MEM 102
 BETA 270 MEM 103 TO 105
 BETA 18 MEM 111
 BETA 270 MEM 113
```

```
 BETA 180 MEM 114
 BETA 270 MEM 115
 BETA 180 MEM 116 TO 118
 BETA 198 MEM 119
 BETA 270 MEM 120 TO 124
 BETA 198 MEM 125
 BETA 270 MEM 126 TO 129
 BETA 18 MEM 130 TO 131
 BETA 270 MEM 132 TO 137
 BETA 18 MEM 138
 BETA 270 MEM 139 TO 141
 BETA 18 MEM 147
 BETA 270 MEM 149
 BETA 180 MEM 150
*All Beam Beta Angle=0
 BETA 0 MEM 151 TO 411
SUPPORTS
 1 FIXED
 2 FIXED
 3 FIXED
 4 FIXED
 5 FIXED
 6 FIXED
 7 FIXED
 8 FIXED
 9 FIXED
 10 FIXED
 11 FIXED
 12 FIXED
 13 FIXED
 14 FIXED
 15 FIXED
 16 FIXED
 17 FIXED
 18 FIXED
 19 FIXED
 20 FIXED
 21 FIXED
 22 FIXED
 23 FIXED
 24 FIXED
 25 FIXED
 26 FIXED
 27 FIXED
 28 FIXED
 29 FIXED
 30 FIXED
 31 FIXED
 32 FIXED
 33 FIXED
 34 FIXED
 35 FIXED
 36 FIXED
 37 FIXED
 38 FIXED
LOAD 5 LOADTYPE Wind TITLE WIND IN X-DIR
JOINT LOAD
 39 FX 5.768697E-02
```

```
1 FX 5.768697E-02
40 FX 5.768697E-02
2 FX 5.768697E-02
41 FX 5.768697E-02
3 FX 5.768697E-02
42 FX 5.768697E-02
4 FX 5.768697E-02
43 FX 5.768697E-02
5 FX 5.768697E-02
44 FX 9.151451E-02
6 FX 9.151451E-02
45 FX 9.151451E-02
7 FX 9.151451E-02
46 FX 9.151451E-02
8 FX 9.151451E-02
47 FX 9.151451E-02
9 FX 9.151451E-02
48 FX 9.151451E-02
10 FX 9.151451E-02
49 FX 9.151451E-02
11 FX 9.151451E-02
50 FX 0.4495745
12 FX 0.4495745
51 FX 5.044183E-02
13 FX 5.044183E-02
52 FX 5.044183E-02
14 FX 5.044183E-02
53 FX 5.044183E-02
15 FX 5.044183E-02
54 FX 5.044183E-02
16 FX 5.044183E-02
57 FX 5.044183E-02
17 FX 5.044183E-02
60 FX 7.998049E-02
18 FX 7.998049E-02
61 FX 0.2929811
19 FX 0.2929811
62 FX 4.899858E-02
20 FX 4.899858E-02
63 FX 4.899858E-02
21 FX 4.899858E-02
64 FX 4.899858E-02
22 FX 4.899858E-02
65 FX 4.899858E-02
23 FX 4.899858E-02
66 FX 4.899858E-02
24 FX 4.899858E-02
67 FX 4.899858E-02
25 FX 4.899858E-02
68 FX 4.899858E-02
26 FX 4.899858E-02
69 FX 0.2260501
27 FX 0.2260501
70 FX 0.2260501
28 FX 0.2260501
71 FX 0.0466172
29 FX 0.0466172
72 FX 0.0466172
```

```
30 FX 0.0466172
73 FX 0.0466172
31 FX 0.0466172
74 FX 0.0466172
32 FX 0.0466172
75 FX 0.0466172
33 FX 0.0466172
76 FX 0.0466172
34 FX 0.0466172
77 FX 0.0466172
35 FX 0.0466172
78 FX 7.998049E-02
78 FX 7.998049E-02
79 FX 5.044182E-02
79 FX 5.044182E-02
80 FX 0.1304223
80 FX 0.1304223
81 FX 8.653045E-02
39 FX 8.653045E-02
82 FX 8.653045E-02
40 FX 8.653045E-02
83 FX 8.653045E-02
41 FX 8.653045E-02
84 FX 8.653045E-02
42 FX 8.653045E-02
85 FX 8.653045E-02
43 FX 8.653045E-02
86 FX 0.1372718
44 FX 0.1372718
87 FX 0.1372718
45 FX 0.1372718
88 FX 0.1372718
46 FX 0.1372718
89 FX 0.1372718
47 FX 0.1372718
90 FX 0.1372718
48 FX 0.1372718
91 FX 0.1372718
49 FX 0.1372718
92 FX 0.6743617
50 FX 0.6743617
93 FX 7.566274E-02
51 FX 7.566274E-02
94 FX 7.566274E-02
52 FX 7.566274E-02
95 FX 7.566274E-02
53 FX 7.566274E-02
96 FX 7.566274E-02
54 FX 7.566274E-02
99 FX 7.566274E-02
57 FX 7.566274E-02
102 FX 0.1199707
60 FX 0.1199707
103 FX 0.4394716
61 FX 0.4394716
104 FX 7.349786E-02
62 FX 7.349786E-02
105 FX 7.349786E-02
```

```
63 FX 7.349786E-02
106 FX 7.349786E-02
64 FX 7.349786E-02
107 FX 7.349786E-02
65 FX 7.349786E-02
108 FX 7.349786E-02
66 FX 7.349786E-02
109 FX 7.349786E-02
67 FX 7.349786E-02
110 FX 7.349786E-02
68 FX 7.349786E-02
111 FX 0.3390752
69 FX 0.3390752
112 FX 0.3390752
70 FX 0.3390752
113 FX 6.992579E-02
71 FX 6.992579E-02
114 FX 6.992579E-02
72 FX 6.992579E-02
115 FX 6.992579E-02
73 FX 6.992579E-02
116 FX 6.992579E-02
74 FX 6.992579E-02
117 FX 6.992579E-02
75 FX 6.992579E-02
118 FX 6.992579E-02
76 FX 6.992579E-02
119 FX 6.992579E-02
77 FX 6.992579E-02
120 FX 0.1199707
120 FX 0.1199707
121 FX 7.566273E-02
121 FX 7.566273E-02
122 FX 0.1956334
122 FX 0.1956334
123 FX 8.653045E-02
81 FX 8.653045E-02
124 FX 8.653045E-02
82 FX 8.653045E-02
125 FX 8.653045E-02
83 FX 8.653045E-02
126 FX 8.653045E-02
84 FX 8.653045E-02
127 FX 8.653045E-02
85 FX 8.653045E-02
128 FX 0.1372718
86 FX 0.1372718
129 FX 0.1372718
87 FX 0.1372718
130 FX 0.1372718
88 FX 0.1372718
131 FX 0.1372718
89 FX 0.1372718
132 FX 0.1372718
90 FX 0.1372718
133 FX 0.1372718
91 FX 0.1372718
134 FX 0.6743617
```

```
92 FX 0.6743617
135 FX 7.566274E-02
93 FX 7.566274E-02
136 FX 7.566274E-02
94 FX 7.566274E-02
137 FX 7.566274E-02
95 FX 7.566274E-02
138 FX 7.566274E-02
96 FX 7.566274E-02
141 FX 7.566274E-02
99 FX 7.566274E-02
144 FX 0.1199707
102 FX 0.1199707
145 FX 0.4394716
103 FX 0.4394716
146 FX 7.349786E-02
104 FX 7.349786E-02
147 FX 7.349786E-02
105 FX 7.349786E-02
148 FX 7.349786E-02
106 FX 7.349786E-02
149 FX 7.349786E-02
107 FX 7.349786E-02
150 FX 7.349786E-02
108 FX 7.349786E-02
151 FX 7.349786E-02
109 FX 7.349786E-02
152 FX 7.349786E-02
110 FX 7.349786E-02
153 FX 0.3390752
111 FX 0.3390752
154 FX 0.3390752
112 FX 0.3390752
155 FX 6.992579E-02
113 FX 6.992579E-02
156 FX 6.992579E-02
114 FX 6.992579E-02
157 FX 6.992579E-02
115 FX 6.992579E-02
158 FX 6.992579E-02
116 FX 6.992579E-02
159 FX 6.992579E-02
117 FX 6.992579E-02
160 FX 6.992579E-02
118 FX 6.992579E-02
161 FX 6.992579E-02
119 FX 6.992579E-02
162 FX 0.1199707
162 FX 0.1199707
163 FX 7.566273E-02
163 FX 7.566273E-02
164 FX 0.1956334
164 FX 0.1956334
165 FX 9.330964E-02
123 FX 0.0878863
166 FX 9.330964E-02
124 FX 0.0878863
167 FX 9.330964E-02
```

```
125 FX 0.0878863
168 FX 9.330964E-02
126 FX 0.0878863
169 FX 9.330964E-02
127 FX 0.0878863
170 FX 0.1989571
128 FX 0.1873933
171 FX 0.1989571
129 FX 0.1873933
172 FX 0.1989571
130 FX 0.1873933
173 FX 0.1989571
131 FX 0.1873933
174 FX 0.1989571
132 FX 0.1873933
175 FX 0.1989571
133 FX 0.1873933
176 FX 0.1418204
135 FX 0.1335775
177 FX 0.1418204
136 FX 0.1335775
178 FX 0.1418204
137 FX 0.1335775
179 FX 0.1418204
138 FX 0.1335775
182 FX 0.1418204
141 FX 0.1335775
185 FX 0.2464834
144 FX 0.2321573
186 FX 9.676471E-02
146 FX 9.114055E-02
187 FX 9.676471E-02
147 FX 9.114055E-02
188 FX 9.676471E-02
148 FX 9.114055E-02
189 FX 9.676471E-02
149 FX 9.114055E-02
190 FX 9.676471E-02
150 FX 9.114055E-02
191 FX 9.676471E-02
151 FX 9.114055E-02
192 FX 9.676471E-02
152 FX 9.114055E-02
193 FX 0.3656399
153 FX 0.3443882
194 FX 0.3656399
154 FX 0.3443882
195 FX 7.540409E-02
155 FX 7.102145E-02
196 FX 7.540409E-02
156 FX 7.102145E-02
197 FX 7.540409E-02
157 FX 7.102145E-02
198 FX 7.540409E-02
158 FX 7.102145E-02
199 FX 7.540409E-02
159 FX 7.102145E-02
200 FX 7.540409E-02
```

```
 160 FX 7.102145E-02
 201 FX 7.540409E-02
 161 FX 7.102145E-02
 202 FX 0.2464834
 202 FX 0.2321573
 203 FX 0.1418204
 203 FX 0.1335775
 204 FX 0.3883038
 204 FX 0.3657348
LOAD 6 LOADTYPE Wind TITLE WIND IN Z-DIR
JOINT LOAD
 39 FZ -8.758195E-02
 1 FZ -8.758195E-02
 40 FZ -0.1751639
 2 FZ -0.1751639
 41 FZ -0.1751639
 3 FZ -0.1751639
 42 FZ -0.1751639
 4 FZ -0.1751639
 43 FZ -0.0771816
 5 FZ -0.0771816
 44 FZ -8.758195E-02
 6 FZ -8.758195E-02
 45 FZ -0.1751639
 7 FZ -0.1751639
 46 FZ -0.1751639
 8 FZ -0.1751639
 47 FZ -0.1751639
 9 FZ -0.1751639
 48 FZ -0.0771816
 10 FZ -0.0771816
 49 FZ -0.1291834
 11 FZ -0.1291834
 50 FZ -0.3492331
 12 FZ -0.3492331
 51 FZ -8.758195E-02
 13 FZ -8.758195E-02
 52 FZ -0.1751639
 14 FZ -0.1751639
 53 FZ -0.1751639
 15 FZ -0.1751639
 54 FZ -0.1751639
 16 FZ -0.1751639
 57 FZ -0.1899434
 17 FZ -0.1899434
 60 FZ -4.050666E-02
 18 FZ -4.050666E-02
 61 FZ -0.1631214
 19 FZ -0.1631214
 62 FZ -8.758195E-02
 20 FZ -8.758195E-02
 63 FZ -0.1751639
 21 FZ -0.1751639
 64 FZ -0.1751639
 22 FZ -0.1751639
 65 FZ -0.1751639
 23 FZ -0.1751639
 66 FZ -0.0771816
```

```
24 FZ -0.0771816
67 FZ -4.050666E-02
25 FZ -4.050666E-02
68 FZ -0.2654828
26 FZ -0.2654828
69 FZ -0.0771816
27 FZ -0.0771816
70 FZ -4.050666E-02
28 FZ -4.050666E-02
71 FZ -8.758195E-02
29 FZ -8.758195E-02
72 FZ -0.1751639
30 FZ -0.1751639
73 FZ -0.1751639
31 FZ -0.1751639
74 FZ -0.1751639
32 FZ -0.1751639
75 FZ -0.0771816
33 FZ -0.0771816
76 FZ -4.050666E-02
34 FZ -4.050666E-02
77 FZ -0.3464961
35 FZ -0.3464961
78 FX -0.0771816
78 FX -0.0771816
79 FX -0.0771816
79 FX -0.0771816
80 FX -0.1598371
80 FX -0.1598371
81 FZ -0.1313729
39 FZ -0.1313729
82 FZ -0.2627459
40 FZ -0.2627459
83 FZ -0.2627459
41 FZ -0.2627459
84 FZ -0.2627459
42 FZ -0.2627459
85 FZ -0.1157724
43 FZ -0.1157724
86 FZ -0.1313729
44 FZ -0.1313729
87 FZ -0.2627459
45 FZ -0.2627459
88 FZ -0.2627459
46 FZ -0.2627459
89 FZ -0.2627459
47 FZ -0.2627459
90 FZ -0.1157724
48 FZ -0.1157724
91 FZ -0.1937751
49 FZ -0.1937751
92 FZ -0.5238495
50 FZ -0.5238495
93 FZ -0.1313729
51 FZ -0.1313729
94 FZ -0.2627459
52 FZ -0.2627459
95 FZ -0.2627459
```

```
53 FZ -0.2627459
96 FZ -0.2627459
54 FZ -0.2627459
99 FZ -0.2849151
57 FZ -0.2849151
102 FZ -6.075998E-02
60 FZ -6.075998E-02
103 FZ -0.2446821
61 FZ -0.2446821
104 FZ -0.1313729
62 FZ -0.1313729
105 FZ -0.2627459
63 FZ -0.2627459
106 FZ -0.2627459
64 FZ -0.2627459
107 FZ -0.2627459
65 FZ -0.2627459
108 FZ -0.1157724
66 FZ -0.1157724
109 FZ -6.075998E-02
67 FZ -6.075998E-02
110 FZ -0.3982242
68 FZ -0.3982242
111 FZ -0.1157724
69 FZ -0.1157724
112 FZ -6.075998E-02
70 FZ -6.075998E-02
113 FZ -0.1313729
71 FZ -0.1313729
114 FZ -0.2627459
72 FZ -0.2627459
115 FZ -0.2627459
73 FZ -0.2627459
116 FZ -0.2627459
74 FZ -0.2627459
117 FZ -0.1157724
75 FZ -0.1157724
118 FZ -6.075998E-02
76 FZ -6.075998E-02
119 FZ -0.5197442
77 FZ -0.5197442
120 FX -0.1157724
120 FX -0.1157724
121 FX -0.1157724
121 FX -0.1157724
122 FX -0.2397556
122 FX -0.2397556
123 FZ -0.1313729
81 FZ -0.1313729
124 FZ -0.2627459
82 FZ -0.2627459
125 FZ -0.2627459
83 FZ -0.2627459
126 FZ -0.2627459
84 FZ -0.2627459
127 FZ -0.1157724
85 FZ -0.1157724
128 FZ -0.1313729
```

```
86 FZ -0.1313729
129 FZ -0.2627459
87 FZ -0.2627459
130 FZ -0.2627459
88 FZ -0.2627459
131 FZ -0.2627459
89 FZ -0.2627459
132 FZ -0.1157724
90 FZ -0.1157724
133 FZ -0.1937751
91 FZ -0.1937751
134 FZ -0.5238495
92 FZ -0.5238495
135 FZ -0.1313729
93 FZ -0.1313729
136 FZ -0.2627459
94 FZ -0.2627459
137 FZ -0.2627459
95 FZ -0.2627459
138 FZ -0.2627459
96 FZ -0.2627459
141 FZ -0.2849151
99 FZ -0.2849151
144 FZ -6.075998E-02
102 FZ -6.075998E-02
145 FZ -0.2446821
103 FZ -0.2446821
146 FZ -0.1313729
104 FZ -0.1313729
147 FZ -0.2627459
105 FZ -0.2627459
148 FZ -0.2627459
106 FZ -0.2627459
149 FZ -0.2627459
107 FZ -0.2627459
150 FZ -0.1157724
108 FZ -0.1157724
151 FZ -6.075998E-02
109 FZ -6.075998E-02
152 FZ -0.3982242
110 FZ -0.3982242
153 FZ -0.1157724
111 FZ -0.1157724
154 FZ -6.075998E-02
112 FZ -6.075998E-02
155 FZ -0.1313729
113 FZ -0.1313729
156 FZ -0.2627459
114 FZ -0.2627459
157 FZ -0.2627459
115 FZ -0.2627459
158 FZ -0.2627459
116 FZ -0.2627459
159 FZ -0.1157724
117 FZ -0.1157724
160 FZ -6.075998E-02
118 FZ -6.075998E-02
161 FZ -0.5197442
```

```
119 FZ -0.5197442
162 FX -0.1157724
162 FX -0.1157724
163 FX -0.1157724
163 FX -0.1157724
164 FX -0.2397556
164 FX -0.2397556
165 FZ -0.1416653
123 FZ -0.1334314
166 FZ -0.2833306
124 FZ -0.2668628
167 FZ -0.2833306
125 FZ -0.2668628
168 FZ -0.2833306
126 FZ -0.2668628
169 FZ -0.1248425
127 FZ -0.1175864
170 FZ -0.1416653
128 FZ -0.1334314
171 FZ -0.2833306
129 FZ -0.2668628
172 FZ -0.2833306
130 FZ -0.2668628
173 FZ -0.2833306
131 FZ -0.2668628
174 FZ -0.1248425
132 FZ -0.1175864
175 FZ -0.2089563
133 FZ -0.1968113
176 FZ -0.1416653
135 FZ -0.1334314
177 FZ -0.2833306
136 FZ -0.2668628
178 FZ -0.2833306
137 FZ -0.2668628
179 FZ -0.2833306
138 FZ -0.2668628
182 FZ -0.3072366
141 FZ -0.2893794
185 FZ -6.552019E-02
144 FZ -6.171203E-02
186 FZ -0.1416653
146 FZ -0.1334314
187 FZ -0.2833306
147 FZ -0.2668628
188 FZ -0.2833306
148 FZ -0.2668628
189 FZ -0.2833306
149 FZ -0.2668628
190 FZ -0.1248425
150 FZ -0.1175864
191 FZ -6.552019E-02
151 FZ -6.171203E-02
192 FZ -0.4294229
152 FZ -0.4044639
193 FZ -0.1248425
153 FZ -0.1175864
194 FZ -6.552019E-02
```

```
 154 FZ -6.171203E-02
 195 FZ -0.1416653
 155 FZ -0.1334314
 196 FZ -0.2833306
 156 FZ -0.2668628
 197 FZ -0.2833306
 157 FZ -0.2668628
 198 FZ -0.2833306
 158 FZ -0.2668628
 199 FZ -0.1248425
 159 FZ -0.1175864
 200 FZ -6.552019E-02
 160 FZ -6.171203E-02
 201 FZ -0.2594245
 161 FZ -0.2443463
 202 FX -0.1248425
 202 FX -0.1175864
 203 FX -0.1248425
 203 FX -0.1175864
 204 FX -0.2585391
 204 FX -0.2435123
LOAD 9 LOADTYPE Dead TITLE DEAD LOAD
 SELFWEIGHT Y -1
JOINT LOAD
 92 FY  -8.15622E-14 MX  -4.841474E-21 MZ  -1.202599E-20
 103 FY  -1.06989E-13 MX  1.604572E-20 MZ  -5.632066E-21
 110 FY  -5.36661E-15 MX  1.305736E-38 MZ  -1.06625E-22
 119 FY  -2.300696E-14 MX  2.239106E-37 MZ  -1.828429E-21
 134 FY  -8.15622E-14 MX  -4.841474E-21 MZ  -1.202599E-20
 145 FY  -1.06989E-13 MX  1.604572E-20 MZ  -5.632066E-21
 152 FY  -5.36661E-15 MX  1.305736E-38 MZ  -1.06625E-22
 161 FY  -2.300696E-14 MX  2.239106E-37 MZ  -1.828429E-21
 192 FY  -5.36661E-15 MX  1.305736E-38 MZ  -1.06625E-22
 201 FY  -2.300696E-14 MX  2.239106E-37 MZ  -1.828429E-21
MEMBER LOAD
 151 UNI  Y  -0.77875
 152 UNI  Y  -0.77875
 153 UNI  Y  -0.77875
 154 UNI  Y  -0.77875
 155 UNI  Y  -0.39175
 156 UNI  Y  -0.39175
 157 UNI  Y  -0.39175
 158 UNI  Y  -0.39175
 159 UNI  Y  -0.39175
 160 UNI  Y  -0.39175
 161 UNI  Y  -0.39175
 162 UNI  Y  -0.39175
 163 UNI  Y  -0.39175
 165 UNI  Y  -0.39175
 166 UNI  Y  -0.39175
 167 UNI  Y  -0.39175
 168 UNI  Y  -0.39175
 169 UNI  Y  -0.39175
 170 UNI  Y  -0.39175
 171 UNI  Y  -0.39175
 172 UNI  Y  -0.39175
 173 UNI  Y  -0.39175
 174 UNI  Y  -0.39175
```

```
175 UNI  Y  -0.77875
176 UNI  Y  -0.77875
177 UNI  Y  -0.77875
178 UNI  Y  -0.77875
179 UNI  Y  -0.77875
180 UNI  Y  -0.77875
181 UNI  Y  -0.77875
182 UNI  Y  -0.77875
183 UNI  Y  -0.77875
184 UNI  Y  -0.77875
185 UNI  Y  -0.39175
186 UNI  Y  -0.39175
187 UNI  Y  -0.39175
188 UNI  Y  -0.39175
189 UNI  Y  -0.39175
190 UNI  Y  -0.39175
191 UNI  Y  -0.39175
192 UNI  Y  -0.39175
193 UNI  Y  -0.39175
194 UNI  Y  -0.39175
195 UNI  Y  -0.39175
196 UNI  Y  -0.39175
197 UNI  Y  -0.39175
198 UNI  Y  -0.39175
199 UNI  Y  -0.39175
201 UNI  Y  -0.39175
202 UNI  Y  -0.39175
204 UNI  Y  -0.39175
205 UNI  Y  -0.39175
206 UNI  Y  -0.77875
207 UNI  Y  -0.77875
208 UNI  Y  -0.39175
209 UNI  Y  -0.77875
210 UNI  Y  -0.77875
211 UNI  Y  -0.39175
212 UNI  Y  -0.77875
213 UNI  Y  -0.77875
217 UNI  Y  -0.77875
217 TRAP  Y  -1.642 0 5.145 7.998
217 UNI  Y  -1.642 2.855 5.143
217 TRAP  Y  0 -1.642 0 2.854
218 UNI  Y  -0.77875
218 TRAP  Y  -1.642 0 5.145 7.998
218 UNI  Y  -1.642 2.855 5.143
218 TRAP  Y  0 -1.642 0 2.854
219 UNI  Y  -0.77875
219 TRAP  Y  -1.642 0 5.145 7.998
219 UNI  Y  -1.642 2.855 5.143
219 TRAP  Y  0 -1.642 0 2.854
220 UNI  Y  -0.77875
220 TRAP  Y  -1.642 0 5.145 7.998
220 UNI  Y  -1.642 2.855 5.143
220 TRAP  Y  0 -1.642 0 2.854
221 UNI  Y  -0.39175
221 TRAP  Y  -1.642 0 5.145 7.998
221 UNI  Y  -1.642 2.855 5.143
221 TRAP  Y  0 -1.642 0 2.854
221 TRAP  Y  0 -2.8 0 3.998
```

```
221 TRAP  Y  -2.8 0 4 7.998
222 UNI  Y  -0.39175
222 TRAP  Y  -1.642 0 5.145 7.998
222 UNI  Y  -1.642 2.855 5.143
222 TRAP  Y  0 -1.642 0 2.854
222 TRAP  Y  0 -2.8 0 3.998
222 TRAP  Y  -2.8 0 4 7.998
223 UNI  Y  -0.39175
223 TRAP  Y  -1.642 0 5.145 7.998
223 UNI  Y  -1.642 2.855 5.143
223 TRAP  Y  0 -1.642 0 2.854
223 TRAP  Y  0 -2.8 0 3.998
223 TRAP  Y  -2.8 0 4 7.998
224 UNI  Y  -0.39175
224 TRAP  Y  -1.642 0 5.145 7.998
224 UNI  Y  -1.642 2.855 5.143
224 TRAP  Y  0 -1.642 0 2.854
224 TRAP  Y  0 -2.8 0 3.998
224 TRAP  Y  -2.8 0 4 7.998
225 UNI  Y  -0.39175
225 TRAP  Y  0 -0.761 0 0.874
225 TRAP  Y  -0.761 0 0.876 1.869
225 TRAP  Y  -2.805 0 0.961 1.869
225 TRAP  Y  0 -2.805 0 0.96
226 UNI  Y  -0.39175
226 TRAP  Y  0 -2.8 0 3.998
226 TRAP  Y  -2.8 0 4 7.998
226 TRAP  Y  0 -2.8 0 3.998
226 TRAP  Y  -2.8 0 4 7.998
227 UNI  Y  -0.39175
227 TRAP  Y  0 -2.8 0 3.998
227 TRAP  Y  -2.8 0 4 7.998
227 TRAP  Y  0 -2.8 0 3.998
227 TRAP  Y  -2.8 0 4 7.998
228 UNI  Y  -0.39175
228 TRAP  Y  0 -2.8 0 3.998
228 TRAP  Y  -2.8 0 4 7.998
228 TRAP  Y  0 -2.8 0 3.998
228 TRAP  Y  -2.8 0 4 7.998
229 UNI  Y  -0.39175
229 TRAP  Y  0 -2.8 0 3.998
229 TRAP  Y  -2.8 0 4 7.998
229 TRAP  Y  0 -2.8 0 3.998
229 TRAP  Y  -2.8 0 4 7.998
230 TRAP  Y  0 -1.111 0 1.179
230 TRAP  Y  0 -0.472 0 1.178
231 UNI  Y  -0.39175
231 TRAP  Y  -1.976 0 0 2.429
231 TRAP  Y  -0.689 0 1.307 2.428
231 TRAP  Y  0 -0.689 0 1.306
232 UNI  Y  -0.39175
232 TRAP  Y  0 -0.472 0 1.178
232 TRAP  Y  -0.472 0 1.18 2.358
232 TRAP  Y  0 -1.302 0 2.358
233 UNI  Y  -0.39175
233 TRAP  Y  0 -0.611 0 1.603
233 TRAP  Y  -0.611 0 1.605 3.489
233 TRAP  Y  -1.745 0 0.803 3.488
```

```
233 TRAP  Y  -1.302 -1.745 0 0.801
234 UNI  Y  -0.39175
234 TRAP  Y  0 -2.8 0 3.998
234 TRAP  Y  -2.8 0 4 7.998
234 TRAP  Y  0 -2.8 0 3.998
234 TRAP  Y  -2.8 0 4 7.998
235 UNI  Y  -0.39175
235 TRAP  Y  0 -2.8 0 3.998
235 TRAP  Y  -2.8 0 4 7.998
235 TRAP  Y  0 -2.8 0 3.998
235 TRAP  Y  -2.8 0 4 7.998
236 UNI  Y  -0.39175
236 TRAP  Y  0 -2.8 0 3.998
236 TRAP  Y  -2.8 0 4 7.998
236 TRAP  Y  0 -2.8 0 3.998
236 TRAP  Y  -2.8 0 4 7.998
237 UNI  Y  -0.39175
237 TRAP  Y  0 -2.8 0 3.998
237 TRAP  Y  -2.8 0 4 7.998
237 TRAP  Y  0 -2.8 0 3.998
237 TRAP  Y  -2.8 0 4 7.998
238 UNI  Y  -0.39175
238 UNI  Y  -0.56 0 5.828
238 TRAP  Y  0 -1.59 0 3.574
238 TRAP  Y  -1.59 -0.731 3.576 5.828
239 UNI  Y  -0.39175
239 TRAP  Y  -2.816 0 0.988 1.918
239 TRAP  Y  0 -2.816 0 0.986
239 TRAP  Y  -0.731 0 0 1.918
240 UNI  Y  -0.39175
240 TRAP  Y  0 -1.676 0 2.913
240 TRAP  Y  -1.676 0 2.915 5.828
240 UNI  Y  -0.56 0 5.828
241 UNI  Y  -0.77875
241 TRAP  Y  0 -2.8 0 3.998
241 TRAP  Y  -2.8 0 4 7.998
242 UNI  Y  -0.77875
242 TRAP  Y  0 -2.8 0 3.998
242 TRAP  Y  -2.8 0 4 7.998
243 UNI  Y  -0.77875
243 TRAP  Y  0 -2.8 0 3.998
243 TRAP  Y  -2.8 0 4 7.998
244 UNI  Y  -0.77875
244 TRAP  Y  0 -2.8 0 3.998
244 TRAP  Y  -2.8 0 4 7.998
245 UNI  Y  -0.77875
245 TRAP  Y  0 -1.676 0 2.913
245 TRAP  Y  -1.676 0 2.915 5.828
246 UNI  Y  -0.77875
246 TRAP  Y  0 -2.106 0 2.247
246 TRAP  Y  -2.106 0 2.248 4.848
247 UNI  Y  -0.77875
247 TRAP  Y  0 -2.8 0 3.998
247 UNI  Y  -2.8 4 4.948
247 TRAP  Y  -2.8 0 4.95 8.948
248 UNI  Y  -0.77875
248 TRAP  Y  0 -2.8 0 3.998
248 UNI  Y  -2.8 4 5.048
```

```
248 TRAP  Y  -2.8 0 5.05 9.049
249 UNI  Y  -0.77875
249 TRAP  Y  0 -2.8 0 3.998
249 UNI  Y  -2.8 4 4.898
249 TRAP  Y  -2.8 0 4.9 8.899
250 UNI  Y  -0.77875
250 TRAP  Y  -1.642 0 2.855 5.709
250 TRAP  Y  0 -1.642 0 2.854
251 UNI  Y  -0.39175
251 TRAP  Y  0 -2.8 0 3.998
251 UNI  Y  -2.8 4 4.948
251 TRAP  Y  -2.8 0 4.95 8.948
251 TRAP  Y  0 -2.8 0 3.998
251 UNI  Y  -2.8 4 4.948
251 TRAP  Y  -2.8 0 4.95 8.948
252 UNI  Y  -0.39175
252 TRAP  Y  0 -2.8 0 3.998
252 UNI  Y  -2.8 4 5.048
252 TRAP  Y  -2.8 0 5.05 9.049
252 TRAP  Y  0 -2.8 0 3.998
252 UNI  Y  -2.8 4 5.048
252 TRAP  Y  -2.8 0 5.05 9.049
253 UNI  Y  -0.39175
253 TRAP  Y  0 -2.8 0 3.998
253 UNI  Y  -2.8 4 4.898
253 TRAP  Y  -2.8 0 4.9 8.899
253 TRAP  Y  0 -2.8 0 3.998
253 UNI  Y  -2.8 4 4.898
253 TRAP  Y  -2.8 0 4.9 8.899
254 UNI  Y  -0.39175
254 TRAP  Y  -1.642 0 2.855 5.709
254 TRAP  Y  0 -1.642 0 2.854
254 TRAP  Y  -1.642 0 2.855 5.709
254 TRAP  Y  0 -1.642 0 2.854
255 UNI  Y  -0.39175
255 TRAP  Y  0 -2.8 0 3.998
255 UNI  Y  -2.8 4 4.948
255 TRAP  Y  -2.8 0 4.95 8.948
255 TRAP  Y  0 -2.8 0 3.998
255 UNI  Y  -2.8 4 4.948
255 TRAP  Y  -2.8 0 4.95 8.948
256 UNI  Y  -0.39175
256 TRAP  Y  0 -2.8 0 3.998
256 UNI  Y  -2.8 4 5.048
256 TRAP  Y  -2.8 0 5.05 9.049
256 TRAP  Y  0 -2.8 0 3.998
256 UNI  Y  -2.8 4 5.048
256 TRAP  Y  -2.8 0 5.05 9.049
257 UNI  Y  -0.39175
257 TRAP  Y  0 -2.8 0 3.998
257 UNI  Y  -2.8 4 4.898
257 TRAP  Y  -2.8 0 4.9 8.899
257 TRAP  Y  0 -2.8 0 3.998
257 UNI  Y  -2.8 4 4.898
257 TRAP  Y  -2.8 0 4.9 8.899
258 UNI  Y  -0.39175
258 TRAP  Y  -1.642 0 2.855 5.709
258 TRAP  Y  0 -1.642 0 2.854
```

```
258 TRAP  Y  -1.642 0 2.855 5.709
258 TRAP  Y  0 -1.642 0 2.854
259 UNI  Y  -0.39175
259 TRAP  Y  0 -2.8 0 3.998
259 UNI  Y  -2.8 4 4.948
259 TRAP  Y  -2.8 0 4.95 8.948
259 TRAP  Y  0 -2.8 0 3.998
259 UNI  Y  -2.8 4 4.948
259 TRAP  Y  -2.8 0 4.95 8.948
260 UNI  Y  -0.39175
260 TRAP  Y  0 -2.8 0 3.998
260 UNI  Y  -2.8 4 5.048
260 TRAP  Y  -2.8 0 5.05 9.049
260 TRAP  Y  0 -2.8 0 3.998
260 UNI  Y  -2.8 4 5.048
260 TRAP  Y  -2.8 0 5.05 9.049
261 UNI  Y  -0.39175
261 TRAP  Y  0 -2.8 0 3.998
261 UNI  Y  -2.8 4 4.898
261 TRAP  Y  -2.8 0 4.9 8.899
261 TRAP  Y  0 -2.8 0 3.998
261 UNI  Y  -2.8 4 4.898
261 TRAP  Y  -2.8 0 4.9 8.899
262 UNI  Y  -0.39175
262 TRAP  Y  -1.642 0 2.855 5.709
262 TRAP  Y  0 -1.642 0 2.854
262 TRAP  Y  -1.642 0 2.855 5.709
262 TRAP  Y  0 -1.642 0 2.854
263 UNI  Y  -0.39175
263 TRAP  Y  0 -2.8 0 3.998
263 UNI  Y  -2.8 4 4.948
263 TRAP  Y  -2.8 -1.743 4.95 6.458
263 TRAP  Y  0 -1.676 0 2.913
263 UNI  Y  -1.676 2.915 3.543
263 TRAP  Y  -1.676 0 3.545 6.459
264 UNI  Y  -0.39175
264 TRAP  Y  -1.743 0 0 2.489
265 UNI  Y  -0.39175
265 TRAP  Y  0 -2.8 0 3.998
265 UNI  Y  -2.8 4 5.048
265 TRAP  Y  -2.8 -2.275 5.05 5.799
265 TRAP  Y  -1.968 0 2.844 5.798
265 TRAP  Y  0 -1.968 0 2.842
266 TRAP  Y  -2.275 -1.138 0 1.624
266 UNI  Y  -0.472 1.18 1.624
266 TRAP  Y  0 -0.472 0 1.178
267 UNI  Y  -0.39175
267 TRAP  Y  0 -2.8 0 3.998
267 UNI  Y  -2.8 4 4.898
267 TRAP  Y  -2.8 0 4.9 8.899
267 TRAP  Y  -0.974 0 3.89 8.899
267 TRAP  Y  0 -0.974 0 3.889
268 UNI  Y  -0.39175
268 TRAP  Y  -1.642 0 2.855 5.709
268 TRAP  Y  0 -1.642 0 2.854
268 TRAP  Y  -0.249 0 1.952 5.709
268 TRAP  Y  0 -0.249 0 1.95
269 UNI  Y  -0.472 1.18 1.624
```

```
269 TRAP  Y  0 -0.472 0 1.178
269 TRAP  Y  -0.598 -0.58 1.572 1.624
269 TRAP  Y  0 -0.598 0 1.571
270 UNI  Y  -0.39175
270 TRAP  Y  0 -1.676 0 2.913
270 UNI  Y  -1.676 2.915 3.543
270 TRAP  Y  -1.676 0 3.545 6.459
270 TRAP  Y  -0.989 -0.49 3.921 6.459
270 TRAP  Y  0 -0.989 0 3.919
271 UNI  Y  -0.39175
271 TRAP  Y  -0.49 0 0 2.489
272 UNI  Y  -0.77875
272 TRAP  Y  0 -3.711 0 4.841
272 TRAP  Y  -3.711 0 4.842 10.013
273 UNI  Y  -0.77875
273 TRAP  Y  -3.672 0 5.373 10.253
273 TRAP  Y  0 -3.672 0 5.371
274 UNI  Y  -0.39175
274 TRAP  Y  -0.569 0 1.498 3.417
274 TRAP  Y  0 -0.569 0 1.496
274 TRAP  Y  0 -3.005 0 3.417
275 UNI  Y  -0.77875
275 TRAP  Y  -0.237 0 3.779 6.007
275 TRAP  Y  0 -0.237 0 3.778
276 UNI  Y  -0.77875
276 TRAP  Y  -0.94 0 4.815 9.416
276 TRAP  Y  0 -0.94 0 4.813
277 UNI  Y  -0.39175
277 TRAP  Y  -3.999 0 1.131 6.102
277 TRAP  Y  -3.005 -3.999 0 1.13
277 TRAP  Y  -1.87 0 2.629 6.102
277 TRAP  Y  0 -1.87 0 2.628
278 UNI  Y  -0.77875
278 TRAP  Y  -0.926 0 4.8 9.365
278 TRAP  Y  0 -0.926 0 4.799
279 UNI  Y  -0.77875
279 TRAP  Y  0 -4.08 0 4.661
279 TRAP  Y  -4.08 0 4.663 8.996
280 TRAP  Y  -1.138 0 0 1.624
280 TRAP  Y  -0.472 0 0.445 1.623
280 UNI  Y  -0.472 0 0.443
281 TRAP  Y  -1.111 -2.09 0 1.038
281 TRAP  Y  -2.09 -1.976 1.04 1.178
281 TRAP  Y  -0.472 0 0 1.178
282 TRAP  Y  -0.472 0 0.445 1.623
282 UNI  Y  -0.472 0 0.443
282 TRAP  Y  -0.58 0 0 1.624
283 UNI  Y  -0.77875
283 TRAP  Y  -1.642 0 5.145 7.998
283 UNI  Y  -1.642 2.855 5.143
283 TRAP  Y  0 -1.642 0 2.854
284 UNI  Y  -0.77875
284 TRAP  Y  -1.642 0 5.145 7.998
284 UNI  Y  -1.642 2.855 5.143
284 TRAP  Y  0 -1.642 0 2.854
285 UNI  Y  -0.77875
285 TRAP  Y  -1.642 0 5.145 7.998
285 UNI  Y  -1.642 2.855 5.143
```

```
285 TRAP  Y  0 -1.642 0 2.854
286 UNI  Y  -0.77875
286 TRAP  Y  -1.642 0 5.145 7.998
286 UNI  Y  -1.642 2.855 5.143
286 TRAP  Y  0 -1.642 0 2.854
287 UNI  Y  -0.39175
287 TRAP  Y  -1.642 0 5.145 7.998
287 UNI  Y  -1.642 2.855 5.143
287 TRAP  Y  0 -1.642 0 2.854
287 TRAP  Y  0 -2.8 0 3.998
287 TRAP  Y  -2.8 0 4 7.998
288 UNI  Y  -0.39175
288 TRAP  Y  -1.642 0 5.145 7.998
288 UNI  Y  -1.642 2.855 5.143
288 TRAP  Y  0 -1.642 0 2.854
288 TRAP  Y  0 -2.8 0 3.998
288 TRAP  Y  -2.8 0 4 7.998
289 UNI  Y  -0.39175
289 TRAP  Y  -1.642 0 5.145 7.998
289 UNI  Y  -1.642 2.855 5.143
289 TRAP  Y  0 -1.642 0 2.854
289 TRAP  Y  0 -2.8 0 3.998
289 TRAP  Y  -2.8 0 4 7.998
290 UNI  Y  -0.39175
290 TRAP  Y  -1.642 0 5.145 7.998
290 UNI  Y  -1.642 2.855 5.143
290 TRAP  Y  0 -1.642 0 2.854
290 TRAP  Y  0 -2.8 0 3.998
290 TRAP  Y  -2.8 0 4 7.998
291 UNI  Y  -0.39175
291 TRAP  Y  0 -0.761 0 0.874
291 TRAP  Y  -0.761 0 0.876 1.869
291 TRAP  Y  -2.805 0 0.961 1.869
291 TRAP  Y  0 -2.805 0 0.96
292 UNI  Y  -0.39175
292 TRAP  Y  0 -2.8 0 3.998
292 TRAP  Y  -2.8 0 4 7.998
292 TRAP  Y  0 -2.8 0 3.998
292 TRAP  Y  -2.8 0 4 7.998
293 UNI  Y  -0.39175
293 TRAP  Y  0 -2.8 0 3.998
293 TRAP  Y  -2.8 0 4 7.998
293 TRAP  Y  0 -2.8 0 3.998
293 TRAP  Y  -2.8 0 4 7.998
294 UNI  Y  -0.39175
294 TRAP  Y  0 -2.8 0 3.998
294 TRAP  Y  -2.8 0 4 7.998
294 TRAP  Y  0 -2.8 0 3.998
294 TRAP  Y  -2.8 0 4 7.998
295 UNI  Y  -0.39175
295 TRAP  Y  0 -2.8 0 3.998
295 TRAP  Y  -2.8 0 4 7.998
295 TRAP  Y  0 -2.8 0 3.998
295 TRAP  Y  -2.8 0 4 7.998
296 TRAP  Y  0 -1.111 0 1.179
296 TRAP  Y  0 -0.472 0 1.178
297 UNI  Y  -0.39175
297 TRAP  Y  -1.976 0 0 2.429
```

```
297 TRAP  Y  -0.689 0 1.307 2.428
297 TRAP  Y  0 -0.689 0 1.306
298 UNI  Y  -0.39175
298 TRAP  Y  0 -0.472 0 1.178
298 TRAP  Y  -0.472 0 1.18 2.358
298 TRAP  Y  0 -1.302 0 2.358
299 UNI  Y  -0.39175
299 TRAP  Y  0 -0.611 0 1.603
299 TRAP  Y  -0.611 0 1.605 3.489
299 TRAP  Y  -1.745 0 0.803 3.488
299 TRAP  Y  -1.302 -1.745 0 0.801
300 UNI  Y  -0.39175
300 TRAP  Y  0 -2.8 0 3.998
300 TRAP  Y  -2.8 0 4 7.998
300 TRAP  Y  0 -2.8 0 3.998
300 TRAP  Y  -2.8 0 4 7.998
301 UNI  Y  -0.39175
301 TRAP  Y  0 -2.8 0 3.998
301 TRAP  Y  -2.8 0 4 7.998
301 TRAP  Y  0 -2.8 0 3.998
301 TRAP  Y  -2.8 0 4 7.998
302 UNI  Y  -0.39175
302 TRAP  Y  0 -2.8 0 3.998
302 TRAP  Y  -2.8 0 4 7.998
302 TRAP  Y  0 -2.8 0 3.998
302 TRAP  Y  -2.8 0 4 7.998
303 UNI  Y  -0.39175
303 TRAP  Y  0 -2.8 0 3.998
303 TRAP  Y  -2.8 0 4 7.998
303 TRAP  Y  0 -2.8 0 3.998
303 TRAP  Y  -2.8 0 4 7.998
304 UNI  Y  -0.39175
304 UNI  Y  -0.56 0 5.828
304 TRAP  Y  0 -1.59 0 3.574
304 TRAP  Y  -1.59 -0.731 3.576 5.828
305 UNI  Y  -0.39175
305 TRAP  Y  -2.816 0 0.988 1.918
305 TRAP  Y  0 -2.816 0 0.986
305 TRAP  Y  -0.731 0 0 1.918
306 UNI  Y  -0.39175
306 TRAP  Y  0 -1.676 0 2.913
306 TRAP  Y  -1.676 0 2.915 5.828
306 UNI  Y  -0.56 0 5.828
307 UNI  Y  -0.77875
307 TRAP  Y  0 -2.8 0 3.998
307 TRAP  Y  -2.8 0 4 7.998
308 UNI  Y  -0.77875
308 TRAP  Y  0 -2.8 0 3.998
308 TRAP  Y  -2.8 0 4 7.998
309 UNI  Y  -0.77875
309 TRAP  Y  0 -2.8 0 3.998
309 TRAP  Y  -2.8 0 4 7.998
310 UNI  Y  -0.77875
310 TRAP  Y  0 -2.8 0 3.998
310 TRAP  Y  -2.8 0 4 7.998
311 UNI  Y  -0.77875
311 TRAP  Y  0 -1.676 0 2.913
311 TRAP  Y  -1.676 0 2.915 5.828
```

```
312 UNI  Y  -0.77875
312 TRAP  Y  0 -2.106 0 2.247
312 TRAP  Y  -2.106 0 2.248 4.848
313 UNI  Y  -0.77875
313 TRAP  Y  0 -2.8 0 3.998
313 UNI  Y  -2.8 4 4.948
313 TRAP  Y  -2.8 0 4.95 8.948
314 UNI  Y  -0.77875
314 TRAP  Y  0 -2.8 0 3.998
314 UNI  Y  -2.8 4 5.048
314 TRAP  Y  -2.8 0 5.05 9.049
315 UNI  Y  -0.77875
315 TRAP  Y  0 -2.8 0 3.998
315 UNI  Y  -2.8 4 4.898
315 TRAP  Y  -2.8 0 4.9 8.899
316 UNI  Y  -0.77875
316 TRAP  Y  -1.642 0 2.855 5.709
316 TRAP  Y  0 -1.642 0 2.854
317 UNI  Y  -0.39175
317 TRAP  Y  0 -2.8 0 3.998
317 UNI  Y  -2.8 4 4.948
317 TRAP  Y  -2.8 0 4.95 8.948
317 TRAP  Y  0 -2.8 0 3.998
317 UNI  Y  -2.8 4 4.948
317 TRAP  Y  -2.8 0 4.95 8.948
318 UNI  Y  -0.39175
318 TRAP  Y  0 -2.8 0 3.998
318 UNI  Y  -2.8 4 5.048
318 TRAP  Y  -2.8 0 5.05 9.049
318 TRAP  Y  0 -2.8 0 3.998
318 UNI  Y  -2.8 4 5.048
318 TRAP  Y  -2.8 0 5.05 9.049
319 UNI  Y  -0.39175
319 TRAP  Y  0 -2.8 0 3.998
319 UNI  Y  -2.8 4 4.898
319 TRAP  Y  -2.8 0 4.9 8.899
319 TRAP  Y  0 -2.8 0 3.998
319 UNI  Y  -2.8 4 4.898
319 TRAP  Y  -2.8 0 4.9 8.899
320 UNI  Y  -0.39175
320 TRAP  Y  -1.642 0 2.855 5.709
320 TRAP  Y  0 -1.642 0 2.854
320 TRAP  Y  -1.642 0 2.855 5.709
320 TRAP  Y  0 -1.642 0 2.854
321 UNI  Y  -0.39175
321 TRAP  Y  0 -2.8 0 3.998
321 UNI  Y  -2.8 4 4.948
321 TRAP  Y  -2.8 0 4.95 8.948
321 TRAP  Y  0 -2.8 0 3.998
321 UNI  Y  -2.8 4 4.948
321 TRAP  Y  -2.8 0 4.95 8.948
322 UNI  Y  -0.39175
322 TRAP  Y  0 -2.8 0 3.998
322 UNI  Y  -2.8 4 5.048
322 TRAP  Y  -2.8 0 5.05 9.049
322 TRAP  Y  0 -2.8 0 3.998
322 UNI  Y  -2.8 4 5.048
322 TRAP  Y  -2.8 0 5.05 9.049
```

```
323 UNI  Y  -0.39175
323 TRAP  Y  0 -2.8 0 3.998
323 UNI  Y  -2.8 4 4.898
323 TRAP  Y  -2.8 0 4.9 8.899
323 TRAP  Y  0 -2.8 0 3.998
323 UNI  Y  -2.8 4 4.898
323 TRAP  Y  -2.8 0 4.9 8.899
324 UNI  Y  -0.39175
324 TRAP  Y  -1.642 0 2.855 5.709
324 TRAP  Y  0 -1.642 0 2.854
324 TRAP  Y  -1.642 0 2.855 5.709
324 TRAP  Y  0 -1.642 0 2.854
325 UNI  Y  -0.39175
325 TRAP  Y  0 -2.8 0 3.998
325 UNI  Y  -2.8 4 4.948
325 TRAP  Y  -2.8 0 4.95 8.948
325 TRAP  Y  0 -2.8 0 3.998
325 UNI  Y  -2.8 4 4.948
325 TRAP  Y  -2.8 0 4.95 8.948
326 UNI  Y  -0.39175
326 TRAP  Y  0 -2.8 0 3.998
326 UNI  Y  -2.8 4 5.048
326 TRAP  Y  -2.8 0 5.05 9.049
326 TRAP  Y  0 -2.8 0 3.998
326 UNI  Y  -2.8 4 5.048
326 TRAP  Y  -2.8 0 5.05 9.049
327 UNI  Y  -0.39175
327 TRAP  Y  0 -2.8 0 3.998
327 UNI  Y  -2.8 4 4.898
327 TRAP  Y  -2.8 0 4.9 8.899
327 TRAP  Y  0 -2.8 0 3.998
327 UNI  Y  -2.8 4 4.898
327 TRAP  Y  -2.8 0 4.9 8.899
328 UNI  Y  -0.39175
328 TRAP  Y  -1.642 0 2.855 5.709
328 TRAP  Y  0 -1.642 0 2.854
328 TRAP  Y  -1.642 0 2.855 5.709
328 TRAP  Y  0 -1.642 0 2.854
329 UNI  Y  -0.39175
329 TRAP  Y  0 -2.8 0 3.998
329 UNI  Y  -2.8 4 4.948
329 TRAP  Y  -2.8 -1.743 4.95 6.458
329 TRAP  Y  0 -1.676 0 2.913
329 UNI  Y  -1.676 2.915 3.543
329 TRAP  Y  -1.676 0 3.545 6.459
330 UNI  Y  -0.39175
330 TRAP  Y  -1.743 0 0 2.489
331 UNI  Y  -0.39175
331 TRAP  Y  0 -2.8 0 3.998
331 UNI  Y  -2.8 4 5.048
331 TRAP  Y  -2.8 -2.275 5.05 5.799
331 TRAP  Y  -1.968 0 2.844 5.798
331 TRAP  Y  0 -1.968 0 2.842
332 TRAP  Y  -2.275 -1.138 0 1.624
332 UNI  Y  -0.472 1.18 1.624
332 TRAP  Y  0 -0.472 0 1.178
333 UNI  Y  -0.39175
333 TRAP  Y  0 -2.8 0 3.998
```

```
333 UNI  Y  -2.8 4 4.898
333 TRAP  Y  -2.8 0 4.9 8.899
333 TRAP  Y  -0.974 0 3.89 8.899
333 TRAP  Y  0 -0.974 0 3.889
334 UNI  Y  -0.39175
334 TRAP  Y  -1.642 0 2.855 5.709
334 TRAP  Y  0 -1.642 0 2.854
334 TRAP  Y  -0.249 0 1.952 5.709
334 TRAP  Y  0 -0.249 0 1.95
335 UNI  Y  -0.472 1.18 1.624
335 TRAP  Y  0 -0.472 0 1.178
335 TRAP  Y  -0.598 -0.58 1.572 1.624
335 TRAP  Y  0 -0.598 0 1.571
336 UNI  Y  -0.39175
336 TRAP  Y  0 -1.676 0 2.913
336 UNI  Y  -1.676 2.915 3.543
336 TRAP  Y  -1.676 0 3.545 6.459
336 TRAP  Y  -0.989 -0.49 3.921 6.459
336 TRAP  Y  0 -0.989 0 3.919
337 UNI  Y  -0.39175
337 TRAP  Y  -0.49 0 0 2.489
338 UNI  Y  -0.77875
338 TRAP  Y  0 -3.711 0 4.841
338 TRAP  Y  -3.711 0 4.842 10.013
339 UNI  Y  -0.77875
339 TRAP  Y  -3.672 0 5.373 10.253
339 TRAP  Y  0 -3.672 0 5.371
340 UNI  Y  -0.39175
340 TRAP  Y  -0.569 0 1.498 3.417
340 TRAP  Y  0 -0.569 0 1.496
340 TRAP  Y  0 -3.005 0 3.417
341 UNI  Y  -0.77875
341 TRAP  Y  -0.237 0 3.779 6.007
341 TRAP  Y  0 -0.237 0 3.778
342 UNI  Y  -0.77875
342 TRAP  Y  -0.94 0 4.815 9.416
342 TRAP  Y  0 -0.94 0 4.813
343 UNI  Y  -0.39175
343 TRAP  Y  -3.999 0 1.131 6.102
343 TRAP  Y  -3.005 -3.999 0 1.13
343 TRAP  Y  -1.87 0 2.629 6.102
343 TRAP  Y  0 -1.87 0 2.628
344 UNI  Y  -0.77875
344 TRAP  Y  -0.926 0 4.8 9.365
344 TRAP  Y  0 -0.926 0 4.799
345 UNI  Y  -0.77875
345 TRAP  Y  0 -4.08 0 4.661
345 TRAP  Y  -4.08 0 4.663 8.996
346 TRAP  Y  -1.138 0 0 1.624
346 TRAP  Y  -0.472 0 0.445 1.623
346 UNI  Y  -0.472 0 0.443
347 TRAP  Y  -1.111 -2.09 0 1.038
347 TRAP  Y  -2.09 -1.976 1.04 1.178
347 TRAP  Y  -0.472 0 0 1.178
348 TRAP  Y  -0.472 0 0.445 1.623
348 UNI  Y  -0.472 0 0.443
348 TRAP  Y  -0.58 0 0 1.624
349 TRAP  Y  -1.642 0 5.145 7.998
```

```
349 UNI  Y  -1.642 2.855 5.143
349 TRAP  Y  0 -1.642 0 2.854
350 TRAP  Y  -1.642 0 5.145 7.998
350 UNI  Y  -1.642 2.855 5.143
350 TRAP  Y  0 -1.642 0 2.854
351 TRAP  Y  -1.642 0 5.145 7.998
351 UNI  Y  -1.642 2.855 5.143
351 TRAP  Y  0 -1.642 0 2.854
352 TRAP  Y  -1.642 0 5.145 7.998
352 UNI  Y  -1.642 2.855 5.143
352 TRAP  Y  0 -1.642 0 2.854
353 TRAP  Y  -1.642 0 5.145 7.998
353 UNI  Y  -1.642 2.855 5.143
353 TRAP  Y  0 -1.642 0 2.854
353 TRAP  Y  0 -2.8 0 3.998
353 TRAP  Y  -2.8 0 4 7.998
354 TRAP  Y  -1.642 0 5.145 7.998
354 UNI  Y  -1.642 2.855 5.143
354 TRAP  Y  0 -1.642 0 2.854
354 TRAP  Y  0 -2.8 0 3.998
354 TRAP  Y  -2.8 0 4 7.998
355 TRAP  Y  -1.642 0 5.145 7.998
355 UNI  Y  -1.642 2.855 5.143
355 TRAP  Y  0 -1.642 0 2.854
355 TRAP  Y  0 -2.8 0 3.998
355 TRAP  Y  -2.8 0 4 7.998
356 TRAP  Y  -1.642 0 5.145 7.998
356 UNI  Y  -1.642 2.855 5.143
356 TRAP  Y  0 -1.642 0 2.854
356 TRAP  Y  0 -2.8 0 3.998
356 TRAP  Y  -2.8 0 4 7.998
357 TRAP  Y  0 -0.761 0 0.874
357 TRAP  Y  -0.761 0 0.876 1.869
357 TRAP  Y  -2.805 0 0.961 1.869
357 TRAP  Y  0 -2.805 0 0.96
358 TRAP  Y  0 -2.8 0 3.998
358 TRAP  Y  -2.8 0 4 7.998
358 TRAP  Y  0 -2.8 0 3.998
358 TRAP  Y  -2.8 0 4 7.998
359 TRAP  Y  0 -2.8 0 3.998
359 TRAP  Y  -2.8 0 4 7.998
359 TRAP  Y  0 -2.8 0 3.998
359 TRAP  Y  -2.8 0 4 7.998
360 TRAP  Y  0 -2.8 0 3.998
360 TRAP  Y  -2.8 0 4 7.998
360 TRAP  Y  0 -2.8 0 3.998
360 TRAP  Y  -2.8 0 4 7.998
361 TRAP  Y  0 -2.8 0 3.998
361 TRAP  Y  -2.8 0 4 7.998
361 TRAP  Y  0 -2.8 0 3.998
361 TRAP  Y  -2.8 0 4 7.998
362 TRAP  Y  0 -1.111 0 1.179
362 TRAP  Y  0 -0.472 0 1.178
363 TRAP  Y  -1.976 0 0 2.429
363 TRAP  Y  -0.689 0 1.307 2.428
363 TRAP  Y  0 -0.689 0 1.306
364 TRAP  Y  0 -0.472 0 1.178
364 TRAP  Y  -0.472 0 1.18 2.358
```

```
364 TRAP  Y  0 -1.302 0 2.358
365 TRAP  Y  0 -0.611 0 1.603
365 TRAP  Y  -0.611 0 1.605 3.489
365 TRAP  Y  -1.745 0 0.803 3.488
365 TRAP  Y  -1.302 -1.745 0 0.801
366 TRAP  Y  0 -2.8 0 3.998
366 TRAP  Y  -2.8 0 4 7.998
366 TRAP  Y  0 -2.8 0 3.998
366 TRAP  Y  -2.8 0 4 7.998
367 TRAP  Y  0 -2.8 0 3.998
367 TRAP  Y  -2.8 0 4 7.998
367 TRAP  Y  0 -2.8 0 3.998
367 TRAP  Y  -2.8 0 4 7.998
368 TRAP  Y  0 -2.8 0 3.998
368 TRAP  Y  -2.8 0 4 7.998
368 TRAP  Y  0 -2.8 0 3.998
368 TRAP  Y  -2.8 0 4 7.998
369 TRAP  Y  0 -2.8 0 3.998
369 TRAP  Y  -2.8 0 4 7.998
369 TRAP  Y  0 -2.8 0 3.998
369 TRAP  Y  -2.8 0 4 7.998
370 UNI  Y  -0.56 0 5.828
370 TRAP  Y  0 -1.59 0 3.574
370 TRAP  Y  -1.59 -0.731 3.576 5.828
371 TRAP  Y  -2.816 0 0.988 1.918
371 TRAP  Y  0 -2.816 0 0.986
371 TRAP  Y  -0.731 0 0 1.918
372 TRAP  Y  0 -1.676 0 2.913
372 TRAP  Y  -1.676 0 2.915 5.828
372 UNI  Y  -0.56 0 5.828
373 TRAP  Y  0 -2.8 0 3.998
373 TRAP  Y  -2.8 0 4 7.998
374 TRAP  Y  0 -2.8 0 3.998
374 TRAP  Y  -2.8 0 4 7.998
375 TRAP  Y  0 -2.8 0 3.998
375 TRAP  Y  -2.8 0 4 7.998
376 TRAP  Y  0 -2.8 0 3.998
376 TRAP  Y  -2.8 0 4 7.998
377 TRAP  Y  0 -1.676 0 2.913
377 TRAP  Y  -1.676 0 2.915 5.828
378 TRAP  Y  0 -2.106 0 2.247
378 TRAP  Y  -2.106 0 2.248 4.848
379 TRAP  Y  0 -2.8 0 3.998
379 UNI  Y  -2.8 4 4.948
379 TRAP  Y  -2.8 0 4.95 8.948
380 TRAP  Y  0 -2.8 0 3.998
380 UNI  Y  -2.8 4 5.048
380 TRAP  Y  -2.8 0 5.05 9.049
381 TRAP  Y  0 -2.8 0 3.998
381 UNI  Y  -2.8 4 4.898
381 TRAP  Y  -2.8 0 4.9 8.899
382 TRAP  Y  -1.642 0 2.855 5.709
382 TRAP  Y  0 -1.642 0 2.854
383 TRAP  Y  0 -2.8 0 3.998
383 UNI  Y  -2.8 4 4.948
383 TRAP  Y  -2.8 0 4.95 8.948
383 TRAP  Y  0 -2.8 0 3.998
383 UNI  Y  -2.8 4 4.948
```

```
383 TRAP  Y  -2.8 0 4.95 8.948
384 TRAP  Y  0 -2.8 0 3.998
384 UNI  Y  -2.8 4 5.048
384 TRAP  Y  -2.8 0 5.05 9.049
384 TRAP  Y  0 -2.8 0 3.998
384 UNI  Y  -2.8 4 5.048
384 TRAP  Y  -2.8 0 5.05 9.049
385 TRAP  Y  0 -2.8 0 3.998
385 UNI  Y  -2.8 4 4.898
385 TRAP  Y  -2.8 0 4.9 8.899
385 TRAP  Y  0 -2.8 0 3.998
385 UNI  Y  -2.8 4 4.898
385 TRAP  Y  -2.8 0 4.9 8.899
386 TRAP  Y  -1.642 0 2.855 5.709
386 TRAP  Y  0 -1.642 0 2.854
386 TRAP  Y  -1.642 0 2.855 5.709
386 TRAP  Y  0 -1.642 0 2.854
387 TRAP  Y  0 -2.8 0 3.998
387 UNI  Y  -2.8 4 4.948
387 TRAP  Y  -2.8 0 4.95 8.948
387 TRAP  Y  0 -2.8 0 3.998
387 UNI  Y  -2.8 4 4.948
387 TRAP  Y  -2.8 0 4.95 8.948
388 TRAP  Y  0 -2.8 0 3.998
388 UNI  Y  -2.8 4 5.048
388 TRAP  Y  -2.8 0 5.05 9.049
388 TRAP  Y  0 -2.8 0 3.998
388 UNI  Y  -2.8 4 5.048
388 TRAP  Y  -2.8 0 5.05 9.049
389 TRAP  Y  0 -2.8 0 3.998
389 UNI  Y  -2.8 4 4.898
389 TRAP  Y  -2.8 0 4.9 8.899
389 TRAP  Y  0 -2.8 0 3.998
389 UNI  Y  -2.8 4 4.898
389 TRAP  Y  -2.8 0 4.9 8.899
390 TRAP  Y  -1.642 0 2.855 5.709
390 TRAP  Y  0 -1.642 0 2.854
390 TRAP  Y  -1.642 0 2.855 5.709
390 TRAP  Y  0 -1.642 0 2.854
391 TRAP  Y  0 -2.8 0 3.998
391 UNI  Y  -2.8 4 4.948
391 TRAP  Y  -2.8 0 4.95 8.948
391 TRAP  Y  0 -2.8 0 3.998
391 UNI  Y  -2.8 4 4.948
391 TRAP  Y  -2.8 0 4.95 8.948
392 TRAP  Y  0 -2.8 0 3.998
392 UNI  Y  -2.8 4 5.048
392 TRAP  Y  -2.8 0 5.05 9.049
392 TRAP  Y  0 -2.8 0 3.998
392 UNI  Y  -2.8 4 5.048
392 TRAP  Y  -2.8 0 5.05 9.049
393 TRAP  Y  0 -2.8 0 3.998
393 UNI  Y  -2.8 4 4.898
393 TRAP  Y  -2.8 0 4.9 8.899
393 TRAP  Y  0 -2.8 0 3.998
393 UNI  Y  -2.8 4 4.898
393 TRAP  Y  -2.8 0 4.9 8.899
394 TRAP  Y  -1.642 0 2.855 5.709
```

```
 394 TRAP  Y  0 -1.642 0 2.854
 394 TRAP  Y  -1.642 0 2.855 5.709
 394 TRAP  Y  0 -1.642 0 2.854
 395 TRAP  Y  0 -2.8 0 3.998
 395 UNI  Y  -2.8 4 4.948
 395 TRAP  Y  -2.8 -1.743 4.95 6.458
 395 TRAP  Y  0 -1.676 0 2.913
 395 UNI  Y  -1.676 2.915 3.543
 395 TRAP  Y  -1.676 0 3.545 6.459
 396 TRAP  Y  -1.743 0 0 2.489
 397 TRAP  Y  0 -2.8 0 3.998
 397 UNI  Y  -2.8 4 5.048
 397 TRAP  Y  -2.8 -2.275 5.05 5.799
 397 TRAP  Y  -1.968 0 2.844 5.798
 397 TRAP  Y  0 -1.968 0 2.842
 398 TRAP  Y  -2.275 -1.138 0 1.624
 398 UNI  Y  -0.472 1.18 1.624
 398 TRAP  Y  0 -0.472 0 1.178
 399 TRAP  Y  0 -2.8 0 3.998
 399 UNI  Y  -2.8 4 4.898
 399 TRAP  Y  -2.8 0 4.9 8.899
 399 TRAP  Y  -0.974 0 3.89 8.899
 399 TRAP  Y  0 -0.974 0 3.889
 400 TRAP  Y  -1.642 0 2.855 5.709
 400 TRAP  Y  0 -1.642 0 2.854
 400 TRAP  Y  -0.249 0 1.952 5.709
 400 TRAP  Y  0 -0.249 0 1.95
 401 UNI  Y  -0.472 1.18 1.624
 401 TRAP  Y  0 -0.472 0 1.178
 401 TRAP  Y  -0.598 -0.58 1.572 1.624
 401 TRAP  Y  0 -0.598 0 1.571
 402 TRAP  Y  0 -1.676 0 2.913
 402 UNI  Y  -1.676 2.915 3.543
 402 TRAP  Y  -1.676 0 3.545 6.459
 402 TRAP  Y  -0.989 -0.49 3.921 6.459
 402 TRAP  Y  0 -0.989 0 3.919
 403 TRAP  Y  -0.49 0 0 2.489
 404 TRAP  Y  -0.569 0 1.498 3.417
 404 TRAP  Y  0 -0.569 0 1.496
 405 TRAP  Y  -0.237 0 3.779 6.007
 405 TRAP  Y  0 -0.237 0 3.778
 406 TRAP  Y  -0.94 0 4.815 9.416
 406 TRAP  Y  0 -0.94 0 4.813
 407 TRAP  Y  -1.87 0 2.629 6.102
 407 TRAP  Y  0 -1.87 0 2.628
 408 TRAP  Y  -0.926 0 4.8 9.365
 408 TRAP  Y  0 -0.926 0 4.799
 409 TRAP  Y  -1.138 0 0 1.624
 409 TRAP  Y  -0.472 0 0.445 1.623
 409 UNI  Y  -0.472 0 0.443
 410 TRAP  Y  -1.111 -2.09 0 1.038
 410 TRAP  Y  -2.09 -1.976 1.04 1.178
 410 TRAP  Y  -0.472 0 0 1.178
 411 TRAP  Y  -0.472 0 0.445 1.623
 411 UNI  Y  -0.472 0 0.443
 411 TRAP  Y  -0.58 0 0 1.624
LOAD 10 LOADTYPE Live TITLE LIVE LOAD
JOINT LOAD
```

```
 92 FY  -2.039E-14 MX  -1.210336E-21 MZ  -3.006417E-21
 103 FY  -2.67466E-14 MX  4.011332E-21 MZ  -1.407982E-21
 110 FY  -1.951477E-15 MX  4.748088E-39 MZ  -3.877237E-23
 119 FY  -8.366266E-15 MX  8.142299E-38 MZ  -6.64891E-22
 134 FY  -2.039E-14 MX  -1.210336E-21 MZ  -3.006417E-21
 145 FY  -2.67466E-14 MX  4.011332E-21 MZ  -1.407982E-21
 152 FY  -1.951477E-15 MX  4.748088E-39 MZ  -3.877237E-23
 161 FY  -8.366266E-15 MX  8.142299E-38 MZ  -6.64891E-22
 192 FY  -1.951477E-15 MX  4.748088E-39 MZ  -3.877237E-23
 201 FY  -8.366266E-15 MX  8.142299E-38 MZ  -6.64891E-22
MEMBER LOAD
 217 TRAP  Y  -0.571 0 5.145 7.9985
 217 UNI  Y  -0.571 2.855 5.1435
 217 TRAP  Y  0 -0.571 0 2.8535
 218 TRAP  Y  -0.571 0 5.145 7.9985
 218 UNI  Y  -0.571 2.855 5.1435
 218 TRAP  Y  0 -0.571 0 2.8535
 219 TRAP  Y  -0.571 0 5.145 7.9985
 219 UNI  Y  -0.571 2.855 5.1435
 219 TRAP  Y  0 -0.571 0 2.8535
 220 TRAP  Y  -0.571 0 5.145 7.9985
 220 UNI  Y  -0.571 2.855 5.1435
 220 TRAP  Y  0 -0.571 0 2.8535
 221 TRAP  Y  -0.571 0 5.145 7.9985
 221 UNI  Y  -0.571 2.855 5.1435
 221 TRAP  Y  0 -0.571 0 2.8535
 221 TRAP  Y  0 -0.8 0 3.9985
 221 TRAP  Y  -0.8 0 4 7.9985
 222 TRAP  Y  -0.571 0 5.145 7.9985
 222 UNI  Y  -0.571 2.855 5.1435
 222 TRAP  Y  0 -0.571 0 2.8535
 222 TRAP  Y  0 -0.8 0 3.9985
 222 TRAP  Y  -0.8 0 4 7.9985
 223 TRAP  Y  -0.571 0 5.145 7.9985
 223 UNI  Y  -0.571 2.855 5.1435
 223 TRAP  Y  0 -0.571 0 2.8535
 223 TRAP  Y  0 -0.8 0 3.9985
 223 TRAP  Y  -0.8 0 4 7.9985
 224 TRAP  Y  -0.571 0 5.145 7.9985
 224 UNI  Y  -0.571 2.855 5.1435
 224 TRAP  Y  0 -0.571 0 2.8535
 224 TRAP  Y  0 -0.8 0 3.9985
 224 TRAP  Y  -0.8 0 4 7.9985
 225 TRAP  Y  0 -0.381 0 0.874
 225 TRAP  Y  -0.381 0 0.876 1.8685
 225 TRAP  Y  -1.02 0 0.961 1.8685
 225 TRAP  Y  0 -1.02 0 0.96
 226 TRAP  Y  0 -0.8 0 3.9985
 226 TRAP  Y  -0.8 0 4 7.9985
 226 TRAP  Y  0 -0.8 0 3.9985
 226 TRAP  Y  -0.8 0 4 7.9985
 227 TRAP  Y  0 -0.8 0 3.9985
 227 TRAP  Y  -0.8 0 4 7.9985
 227 TRAP  Y  0 -0.8 0 3.9985
 227 TRAP  Y  -0.8 0 4 7.9985
 228 TRAP  Y  0 -0.8 0 3.9985
 228 TRAP  Y  -0.8 0 4 7.9985
 228 TRAP  Y  0 -0.8 0 3.9985
```

```
228 TRAP  Y  -0.8 0 4 7.9985
229 TRAP  Y  0 -0.8 0 3.9985
229 TRAP  Y  -0.8 0 4 7.9985
229 TRAP  Y  0 -0.8 0 3.9985
229 TRAP  Y  -0.8 0 4 7.9985
230 TRAP  Y  0 -0.404 0 1.179
230 TRAP  Y  0 -0.236 0 1.1785
231 TRAP  Y  -0.719 0 0 2.4285
231 TRAP  Y  -0.344 0 1.307 2.4285
231 TRAP  Y  0 -0.344 0 1.3057
232 TRAP  Y  0 -0.236 0 1.1785
232 TRAP  Y  -0.236 0 1.18 2.3585
232 TRAP  Y  0 -0.453 0 2.3585
233 TRAP  Y  0 -0.306 0 1.6032
233 TRAP  Y  -0.306 0 1.605 3.4885
233 TRAP  Y  -0.607 0 0.803 3.4885
233 TRAP  Y  -0.453 -0.607 0 0.8012
234 TRAP  Y  0 -0.8 0 3.9985
234 TRAP  Y  -0.8 0 4 7.9985
234 TRAP  Y  0 -0.8 0 3.9985
234 TRAP  Y  -0.8 0 4 7.9985
235 TRAP  Y  0 -0.8 0 3.9985
235 TRAP  Y  -0.8 0 4 7.9985
235 TRAP  Y  0 -0.8 0 3.9985
235 TRAP  Y  -0.8 0 4 7.9985
236 TRAP  Y  0 -0.8 0 3.9985
236 TRAP  Y  -0.8 0 4 7.9985
236 TRAP  Y  0 -0.8 0 3.9985
236 TRAP  Y  -0.8 0 4 7.9985
237 TRAP  Y  0 -0.8 0 3.9985
237 TRAP  Y  -0.8 0 4 7.9985
237 TRAP  Y  0 -0.8 0 3.9985
237 TRAP  Y  -0.8 0 4 7.9985
238 UNI  Y  -0.249 0 5.8285
238 TRAP  Y  0 -0.553 0 3.5743
238 TRAP  Y  -0.553 -0.254 3.576 5.8285
239 TRAP  Y  -1.024 0 0.988 1.9185
239 TRAP  Y  0 -1.024 0 0.9863
239 TRAP  Y  -0.254 0 0 1.9185
240 TRAP  Y  0 -0.583 0 2.9135
240 TRAP  Y  -0.583 0 2.915 5.8285
240 UNI  Y  -0.249 0 5.8285
241 TRAP  Y  0 -0.8 0 3.9985
241 TRAP  Y  -0.8 0 4 7.9985
242 TRAP  Y  0 -0.8 0 3.9985
242 TRAP  Y  -0.8 0 4 7.9985
243 TRAP  Y  0 -0.8 0 3.9985
243 TRAP  Y  -0.8 0 4 7.9985
244 TRAP  Y  0 -0.8 0 3.9985
244 TRAP  Y  -0.8 0 4 7.9985
245 TRAP  Y  0 -0.583 0 2.9135
245 TRAP  Y  -0.583 0 2.915 5.8285
246 TRAP  Y  0 -0.766 0 2.2466
246 TRAP  Y  -0.766 0 2.248 4.8485
247 TRAP  Y  0 -0.8 0 3.9985
247 UNI  Y  -0.8 4 4.9485
247 TRAP  Y  -0.8 0 4.95 8.9485
248 TRAP  Y  0 -0.8 0 3.9985
```

```
248 UNI  Y  -0.8 4 5.0485
248 TRAP  Y  -0.8 0 5.05 9.0485
249 TRAP  Y  0 -0.8 0 3.9985
249 UNI  Y  -0.8 4 4.8985
249 TRAP  Y  -0.8 0 4.9 8.8985
250 TRAP  Y  -0.571 0 2.855 5.7085
250 TRAP  Y  0 -0.571 0 2.8535
251 TRAP  Y  0 -0.8 0 3.9985
251 UNI  Y  -0.8 4 4.9485
251 TRAP  Y  -0.8 0 4.95 8.9485
251 TRAP  Y  0 -0.8 0 3.9985
251 UNI  Y  -0.8 4 4.9485
251 TRAP  Y  -0.8 0 4.95 8.9485
252 TRAP  Y  0 -0.8 0 3.9985
252 UNI  Y  -0.8 4 5.0485
252 TRAP  Y  -0.8 0 5.05 9.0485
252 TRAP  Y  0 -0.8 0 3.9985
252 UNI  Y  -0.8 4 5.0485
252 TRAP  Y  -0.8 0 5.05 9.0485
253 TRAP  Y  0 -0.8 0 3.9985
253 UNI  Y  -0.8 4 4.8985
253 TRAP  Y  -0.8 0 4.9 8.8985
253 TRAP  Y  0 -0.8 0 3.9985
253 UNI  Y  -0.8 4 4.8985
253 TRAP  Y  -0.8 0 4.9 8.8985
254 TRAP  Y  -0.571 0 2.855 5.7085
254 TRAP  Y  0 -0.571 0 2.8535
254 TRAP  Y  -0.571 0 2.855 5.7085
254 TRAP  Y  0 -0.571 0 2.8535
255 TRAP  Y  0 -0.8 0 3.9985
255 UNI  Y  -0.8 4 4.9485
255 TRAP  Y  -0.8 0 4.95 8.9485
255 TRAP  Y  0 -0.8 0 3.9985
255 UNI  Y  -0.8 4 4.9485
255 TRAP  Y  -0.8 0 4.95 8.9485
256 TRAP  Y  0 -0.8 0 3.9985
256 UNI  Y  -0.8 4 5.0485
256 TRAP  Y  -0.8 0 5.05 9.0485
256 TRAP  Y  0 -0.8 0 3.9985
256 UNI  Y  -0.8 4 5.0485
256 TRAP  Y  -0.8 0 5.05 9.0485
257 TRAP  Y  0 -0.8 0 3.9985
257 UNI  Y  -0.8 4 4.8985
257 TRAP  Y  -0.8 0 4.9 8.8985
257 TRAP  Y  0 -0.8 0 3.9985
257 UNI  Y  -0.8 4 4.8985
257 TRAP  Y  -0.8 0 4.9 8.8985
258 TRAP  Y  -0.571 0 2.855 5.7085
258 TRAP  Y  0 -0.571 0 2.8535
258 TRAP  Y  -0.571 0 2.855 5.7085
258 TRAP  Y  0 -0.571 0 2.8535
259 TRAP  Y  0 -0.8 0 3.9985
259 UNI  Y  -0.8 4 4.9485
259 TRAP  Y  -0.8 0 4.95 8.9485
259 TRAP  Y  0 -0.8 0 3.9985
259 UNI  Y  -0.8 4 4.9485
259 TRAP  Y  -0.8 0 4.95 8.9485
260 TRAP  Y  0 -0.8 0 3.9985
```

```
260 UNI  Y  -0.8 4 5.0485
260 TRAP  Y  -0.8 0 5.05 9.0485
260 TRAP  Y  0 -0.8 0 3.9985
260 UNI  Y  -0.8 4 5.0485
260 TRAP  Y  -0.8 0 5.05 9.0485
261 TRAP  Y  0 -0.8 0 3.9985
261 UNI  Y  -0.8 4 4.8985
261 TRAP  Y  -0.8 0 4.9 8.8985
261 TRAP  Y  0 -0.8 0 3.9985
261 UNI  Y  -0.8 4 4.8985
261 TRAP  Y  -0.8 0 4.9 8.8985
262 TRAP  Y  -0.571 0 2.855 5.7085
262 TRAP  Y  0 -0.571 0 2.8535
262 TRAP  Y  -0.571 0 2.855 5.7085
262 TRAP  Y  0 -0.571 0 2.8535
263 TRAP  Y  0 -0.8 0 3.9985
263 UNI  Y  -0.8 4 4.9485
263 TRAP  Y  -0.8 -0.498 4.95 6.4585
263 TRAP  Y  0 -0.583 0 2.9135
263 UNI  Y  -0.583 2.915 3.5435
263 TRAP  Y  -0.583 0 3.545 6.4585
264 TRAP  Y  -0.498 0 0 2.4885
265 TRAP  Y  0 -0.8 0 3.9985
265 UNI  Y  -0.8 4 5.0485
265 TRAP  Y  -0.8 -0.65 5.05 5.7985
265 TRAP  Y  -0.684 0 2.844 5.7985
265 TRAP  Y  0 -0.684 0 2.8422
266 TRAP  Y  -0.65 -0.325 0 1.6235
266 UNI  Y  -0.236 1.18 1.6235
266 TRAP  Y  0 -0.236 0 1.1785
267 TRAP  Y  0 -0.8 0 3.9985
267 UNI  Y  -0.8 4 4.8985
267 TRAP  Y  -0.8 0 4.9 8.8985
267 TRAP  Y  -0.354 0 3.89 8.8985
267 TRAP  Y  0 -0.354 0 3.8887
268 TRAP  Y  -0.571 0 2.855 5.7085
268 TRAP  Y  0 -0.571 0 2.8535
268 TRAP  Y  -0.125 0 1.952 5.7085
268 TRAP  Y  0 -0.125 0 1.9501
269 UNI  Y  -0.236 1.18 1.6235
269 TRAP  Y  0 -0.236 0 1.1785
269 UNI  Y  -0.299 1.572 1.6235
269 TRAP  Y  0 -0.299 0 1.5711
270 TRAP  Y  0 -0.583 0 2.9135
270 UNI  Y  -0.583 2.915 3.5435
270 TRAP  Y  -0.583 0 3.545 6.4585
270 TRAP  Y  -0.36 -0.178 3.921 6.4585
270 TRAP  Y  0 -0.36 0 3.9192
271 TRAP  Y  -0.178 0 0 2.4885
272 TRAP  Y  0 -0.928 0 4.8406
272 TRAP  Y  -0.928 0 4.842 10.0131
273 TRAP  Y  -0.918 0 5.373 10.2535
273 TRAP  Y  0 -0.918 0 5.3713
274 TRAP  Y  -0.284 0 1.498 3.417
274 TRAP  Y  0 -0.284 0 1.4963
274 TRAP  Y  0 -0.751 0 3.417
275 TRAP  Y  -0.119 0 3.779 6.0069
275 TRAP  Y  0 -0.119 0 3.7777
```

```
276 TRAP  Y  -0.342 0 4.815 9.4159
276 TRAP  Y  0 -0.342 0 4.8131
277 TRAP  Y  -1 0 1.131 6.1018
277 TRAP  Y  -0.751 -1 0 1.1295
277 TRAP  Y  -0.65 0 2.629 6.1018
277 TRAP  Y  0 -0.65 0 2.6277
278 TRAP  Y  -0.337 0 4.8 9.3653
278 TRAP  Y  0 -0.337 0 4.7989
279 TRAP  Y  0 -1.02 0 4.6612
279 TRAP  Y  -1.02 0 4.663 8.9963
280 TRAP  Y  -0.325 0 0 1.6235
280 TRAP  Y  -0.236 0 0.445 1.6235
280 UNI  Y  -0.236 0 0.4435
281 TRAP  Y  -0.404 -0.76 0 1.0382
281 TRAP  Y  -0.76 -0.719 1.04 1.178
281 TRAP  Y  -0.236 0 0 1.178
282 TRAP  Y  -0.236 0 0.445 1.6235
282 UNI  Y  -0.236 0 0.4435
282 TRAP  Y  -0.29 0 0 1.6235
283 TRAP  Y  -0.571 0 5.145 7.9985
283 UNI  Y  -0.571 2.855 5.1435
283 TRAP  Y  0 -0.571 0 2.8535
284 TRAP  Y  -0.571 0 5.145 7.9985
284 UNI  Y  -0.571 2.855 5.1435
284 TRAP  Y  0 -0.571 0 2.8535
285 TRAP  Y  -0.571 0 5.145 7.9985
285 UNI  Y  -0.571 2.855 5.1435
285 TRAP  Y  0 -0.571 0 2.8535
286 TRAP  Y  -0.571 0 5.145 7.9985
286 UNI  Y  -0.571 2.855 5.1435
286 TRAP  Y  0 -0.571 0 2.8535
287 TRAP  Y  -0.571 0 5.145 7.9985
287 UNI  Y  -0.571 2.855 5.1435
287 TRAP  Y  0 -0.571 0 2.8535
287 TRAP  Y  0 -0.8 0 3.9985
287 TRAP  Y  -0.8 0 4 7.9985
288 TRAP  Y  -0.571 0 5.145 7.9985
288 UNI  Y  -0.571 2.855 5.1435
288 TRAP  Y  0 -0.571 0 2.8535
288 TRAP  Y  0 -0.8 0 3.9985
288 TRAP  Y  -0.8 0 4 7.9985
289 TRAP  Y  -0.571 0 5.145 7.9985
289 UNI  Y  -0.571 2.855 5.1435
289 TRAP  Y  0 -0.571 0 2.8535
289 TRAP  Y  0 -0.8 0 3.9985
289 TRAP  Y  -0.8 0 4 7.9985
290 TRAP  Y  -0.571 0 5.145 7.9985
290 UNI  Y  -0.571 2.855 5.1435
290 TRAP  Y  0 -0.571 0 2.8535
290 TRAP  Y  0 -0.8 0 3.9985
290 TRAP  Y  -0.8 0 4 7.9985
291 TRAP  Y  0 -0.381 0 0.874
291 TRAP  Y  -0.381 0 0.876 1.8685
291 TRAP  Y  -1.02 0 0.961 1.8685
291 TRAP  Y  0 -1.02 0 0.96
292 TRAP  Y  0 -0.8 0 3.9985
292 TRAP  Y  -0.8 0 4 7.9985
292 TRAP  Y  0 -0.8 0 3.9985
```

```
292 TRAP  Y  -0.8 0 4 7.9985
293 TRAP  Y  0 -0.8 0 3.9985
293 TRAP  Y  -0.8 0 4 7.9985
293 TRAP  Y  0 -0.8 0 3.9985
293 TRAP  Y  -0.8 0 4 7.9985
294 TRAP  Y  0 -0.8 0 3.9985
294 TRAP  Y  -0.8 0 4 7.9985
294 TRAP  Y  0 -0.8 0 3.9985
294 TRAP  Y  -0.8 0 4 7.9985
295 TRAP  Y  0 -0.8 0 3.9985
295 TRAP  Y  -0.8 0 4 7.9985
295 TRAP  Y  0 -0.8 0 3.9985
295 TRAP  Y  -0.8 0 4 7.9985
296 TRAP  Y  0 -0.404 0 1.179
296 TRAP  Y  0 -0.236 0 1.1785
297 TRAP  Y  -0.719 0 0 2.4285
297 TRAP  Y  -0.344 0 1.307 2.4285
297 TRAP  Y  0 -0.344 0 1.3057
298 TRAP  Y  0 -0.236 0 1.1785
298 TRAP  Y  -0.236 0 1.18 2.3585
298 TRAP  Y  0 -0.453 0 2.3585
299 TRAP  Y  0 -0.306 0 1.6032
299 TRAP  Y  -0.306 0 1.605 3.4885
299 TRAP  Y  -0.607 0 0.803 3.4885
299 TRAP  Y  -0.453 -0.607 0 0.8012
300 TRAP  Y  0 -0.8 0 3.9985
300 TRAP  Y  -0.8 0 4 7.9985
300 TRAP  Y  0 -0.8 0 3.9985
300 TRAP  Y  -0.8 0 4 7.9985
301 TRAP  Y  0 -0.8 0 3.9985
301 TRAP  Y  -0.8 0 4 7.9985
301 TRAP  Y  0 -0.8 0 3.9985
301 TRAP  Y  -0.8 0 4 7.9985
302 TRAP  Y  0 -0.8 0 3.9985
302 TRAP  Y  -0.8 0 4 7.9985
302 TRAP  Y  0 -0.8 0 3.9985
302 TRAP  Y  -0.8 0 4 7.9985
303 TRAP  Y  0 -0.8 0 3.9985
303 TRAP  Y  -0.8 0 4 7.9985
303 TRAP  Y  0 -0.8 0 3.9985
303 TRAP  Y  -0.8 0 4 7.9985
304 UNI  Y  -0.249 0 5.8285
304 TRAP  Y  0 -0.553 0 3.5743
304 TRAP  Y  -0.553 -0.254 3.576 5.8285
305 TRAP  Y  -1.024 0 0.988 1.9185
305 TRAP  Y  0 -1.024 0 0.9863
305 TRAP  Y  -0.254 0 0 1.9185
306 TRAP  Y  0 -0.583 0 2.9135
306 TRAP  Y  -0.583 0 2.915 5.8285
306 UNI  Y  -0.249 0 5.8285
307 TRAP  Y  0 -0.8 0 3.9985
307 TRAP  Y  -0.8 0 4 7.9985
308 TRAP  Y  0 -0.8 0 3.9985
308 TRAP  Y  -0.8 0 4 7.9985
309 TRAP  Y  0 -0.8 0 3.9985
309 TRAP  Y  -0.8 0 4 7.9985
310 TRAP  Y  0 -0.8 0 3.9985
310 TRAP  Y  -0.8 0 4 7.9985
```

```
311 TRAP  Y  0 -0.583 0 2.9135
311 TRAP  Y  -0.583 0 2.915 5.8285
312 TRAP  Y  0 -0.766 0 2.2466
312 TRAP  Y  -0.766 0 2.248 4.8485
313 TRAP  Y  0 -0.8 0 3.9985
313 UNI  Y  -0.8 4 4.9485
313 TRAP  Y  -0.8 0 4.95 8.9485
314 TRAP  Y  0 -0.8 0 3.9985
314 UNI  Y  -0.8 4 5.0485
314 TRAP  Y  -0.8 0 5.05 9.0485
315 TRAP  Y  0 -0.8 0 3.9985
315 UNI  Y  -0.8 4 4.8985
315 TRAP  Y  -0.8 0 4.9 8.8985
316 TRAP  Y  -0.571 0 2.855 5.7085
316 TRAP  Y  0 -0.571 0 2.8535
317 TRAP  Y  0 -0.8 0 3.9985
317 UNI  Y  -0.8 4 4.9485
317 TRAP  Y  -0.8 0 4.95 8.9485
317 TRAP  Y  0 -0.8 0 3.9985
317 UNI  Y  -0.8 4 4.9485
317 TRAP  Y  -0.8 0 4.95 8.9485
318 TRAP  Y  0 -0.8 0 3.9985
318 UNI  Y  -0.8 4 5.0485
318 TRAP  Y  -0.8 0 5.05 9.0485
318 TRAP  Y  0 -0.8 0 3.9985
318 UNI  Y  -0.8 4 5.0485
318 TRAP  Y  -0.8 0 5.05 9.0485
319 TRAP  Y  0 -0.8 0 3.9985
319 UNI  Y  -0.8 4 4.8985
319 TRAP  Y  -0.8 0 4.9 8.8985
319 TRAP  Y  0 -0.8 0 3.9985
319 UNI  Y  -0.8 4 4.8985
319 TRAP  Y  -0.8 0 4.9 8.8985
320 TRAP  Y  -0.571 0 2.855 5.7085
320 TRAP  Y  0 -0.571 0 2.8535
320 TRAP  Y  -0.571 0 2.855 5.7085
320 TRAP  Y  0 -0.571 0 2.8535
321 TRAP  Y  0 -0.8 0 3.9985
321 UNI  Y  -0.8 4 4.9485
321 TRAP  Y  -0.8 0 4.95 8.9485
321 TRAP  Y  0 -0.8 0 3.9985
321 UNI  Y  -0.8 4 4.9485
321 TRAP  Y  -0.8 0 4.95 8.9485
322 TRAP  Y  0 -0.8 0 3.9985
322 UNI  Y  -0.8 4 5.0485
322 TRAP  Y  -0.8 0 5.05 9.0485
322 TRAP  Y  0 -0.8 0 3.9985
322 UNI  Y  -0.8 4 5.0485
322 TRAP  Y  -0.8 0 5.05 9.0485
323 TRAP  Y  0 -0.8 0 3.9985
323 UNI  Y  -0.8 4 4.8985
323 TRAP  Y  -0.8 0 4.9 8.8985
323 TRAP  Y  0 -0.8 0 3.9985
323 UNI  Y  -0.8 4 4.8985
323 TRAP  Y  -0.8 0 4.9 8.8985
324 TRAP  Y  -0.571 0 2.855 5.7085
324 TRAP  Y  0 -0.571 0 2.8535
324 TRAP  Y  -0.571 0 2.855 5.7085
```

```
324 TRAP  Y  0 -0.571 0 2.8535
325 TRAP  Y  0 -0.8 0 3.9985
325 UNI  Y  -0.8 4 4.9485
325 TRAP  Y  -0.8 0 4.95 8.9485
325 TRAP  Y  0 -0.8 0 3.9985
325 UNI  Y  -0.8 4 4.9485
325 TRAP  Y  -0.8 0 4.95 8.9485
326 TRAP  Y  0 -0.8 0 3.9985
326 UNI  Y  -0.8 4 5.0485
326 TRAP  Y  -0.8 0 5.05 9.0485
326 TRAP  Y  0 -0.8 0 3.9985
326 UNI  Y  -0.8 4 5.0485
326 TRAP  Y  -0.8 0 5.05 9.0485
327 TRAP  Y  0 -0.8 0 3.9985
327 UNI  Y  -0.8 4 4.8985
327 TRAP  Y  -0.8 0 4.9 8.8985
327 TRAP  Y  0 -0.8 0 3.9985
327 UNI  Y  -0.8 4 4.8985
327 TRAP  Y  -0.8 0 4.9 8.8985
328 TRAP  Y  -0.571 0 2.855 5.7085
328 TRAP  Y  0 -0.571 0 2.8535
328 TRAP  Y  -0.571 0 2.855 5.7085
328 TRAP  Y  0 -0.571 0 2.8535
329 TRAP  Y  0 -0.8 0 3.9985
329 UNI  Y  -0.8 4 4.9485
329 TRAP  Y  -0.8 -0.498 4.95 6.4585
329 TRAP  Y  0 -0.583 0 2.9135
329 UNI  Y  -0.583 2.915 3.5435
329 TRAP  Y  -0.583 0 3.545 6.4585
330 TRAP  Y  -0.498 0 0 2.4885
331 TRAP  Y  0 -0.8 0 3.9985
331 UNI  Y  -0.8 4 5.0485
331 TRAP  Y  -0.8 -0.65 5.05 5.7985
331 TRAP  Y  -0.684 0 2.844 5.7985
331 TRAP  Y  0 -0.684 0 2.8422
332 TRAP  Y  -0.65 -0.325 0 1.6235
332 UNI  Y  -0.236 1.18 1.6235
332 TRAP  Y  0 -0.236 0 1.1785
333 TRAP  Y  0 -0.8 0 3.9985
333 UNI  Y  -0.8 4 4.8985
333 TRAP  Y  -0.8 0 4.9 8.8985
333 TRAP  Y  -0.354 0 3.89 8.8985
333 TRAP  Y  0 -0.354 0 3.8887
334 TRAP  Y  -0.571 0 2.855 5.7085
334 TRAP  Y  0 -0.571 0 2.8535
334 TRAP  Y  -0.125 0 1.952 5.7085
334 TRAP  Y  0 -0.125 0 1.9501
335 UNI  Y  -0.236 1.18 1.6235
335 TRAP  Y  0 -0.236 0 1.1785
335 UNI  Y  -0.299 1.572 1.6235
335 TRAP  Y  0 -0.299 0 1.5711
336 TRAP  Y  0 -0.583 0 2.9135
336 UNI  Y  -0.583 2.915 3.5435
336 TRAP  Y  -0.583 0 3.545 6.4585
336 TRAP  Y  -0.36 -0.178 3.921 6.4585
336 TRAP  Y  0 -0.36 0 3.9192
337 TRAP  Y  -0.178 0 0 2.4885
338 TRAP  Y  0 -0.928 0 4.8406
```

```
338 TRAP  Y  -0.928 0 4.842 10.0131
339 TRAP  Y  -0.918 0 5.373 10.2535
339 TRAP  Y  0 -0.918 0 5.3713
340 TRAP  Y  -0.284 0 1.498 3.417
340 TRAP  Y  0 -0.284 0 1.4963
340 TRAP  Y  0 -0.751 0 3.417
341 TRAP  Y  -0.119 0 3.779 6.0069
341 TRAP  Y  0 -0.119 0 3.7777
342 TRAP  Y  -0.342 0 4.815 9.4159
342 TRAP  Y  0 -0.342 0 4.8131
343 TRAP  Y  -1 0 1.131 6.1018
343 TRAP  Y  -0.751 -1 0 1.1295
343 TRAP  Y  -0.65 0 2.629 6.1018
343 TRAP  Y  0 -0.65 0 2.6277
344 TRAP  Y  -0.337 0 4.8 9.3653
344 TRAP  Y  0 -0.337 0 4.7989
345 TRAP  Y  0 -1.02 0 4.6612
345 TRAP  Y  -1.02 0 4.663 8.9963
346 TRAP  Y  -0.325 0 0 1.6235
346 TRAP  Y  -0.236 0 0.445 1.6235
346 UNI  Y  -0.236 0 0.4435
347 TRAP  Y  -0.404 -0.76 0 1.0382
347 TRAP  Y  -0.76 -0.719 1.04 1.178
347 TRAP  Y  -0.236 0 0 1.178
348 TRAP  Y  -0.236 0 0.445 1.6235
348 UNI  Y  -0.236 0 0.4435
348 TRAP  Y  -0.29 0 0 1.6235
349 TRAP  Y  -0.571 0 5.145 7.9985
349 UNI  Y  -0.571 2.855 5.1435
349 TRAP  Y  0 -0.571 0 2.8535
350 TRAP  Y  -0.571 0 5.145 7.9985
350 UNI  Y  -0.571 2.855 5.1435
350 TRAP  Y  0 -0.571 0 2.8535
351 TRAP  Y  -0.571 0 5.145 7.9985
351 UNI  Y  -0.571 2.855 5.1435
351 TRAP  Y  0 -0.571 0 2.8535
352 TRAP  Y  -0.571 0 5.145 7.9985
352 UNI  Y  -0.571 2.855 5.1435
352 TRAP  Y  0 -0.571 0 2.8535
353 TRAP  Y  -0.571 0 5.145 7.9985
353 UNI  Y  -0.571 2.855 5.1435
353 TRAP  Y  0 -0.571 0 2.8535
353 TRAP  Y  0 -0.8 0 3.9985
353 TRAP  Y  -0.8 0 4 7.9985
354 TRAP  Y  -0.571 0 5.145 7.9985
354 UNI  Y  -0.571 2.855 5.1435
354 TRAP  Y  0 -0.571 0 2.8535
354 TRAP  Y  0 -0.8 0 3.9985
354 TRAP  Y  -0.8 0 4 7.9985
355 TRAP  Y  -0.571 0 5.145 7.9985
355 UNI  Y  -0.571 2.855 5.1435
355 TRAP  Y  0 -0.571 0 2.8535
355 TRAP  Y  0 -0.8 0 3.9985
355 TRAP  Y  -0.8 0 4 7.9985
356 TRAP  Y  -0.571 0 5.145 7.9985
356 UNI  Y  -0.571 2.855 5.1435
356 TRAP  Y  0 -0.571 0 2.8535
356 TRAP  Y  0 -0.8 0 3.9985
```

```
356 TRAP  Y  -0.8 0 4 7.9985
357 TRAP  Y  0 -0.381 0 0.874
357 TRAP  Y  -0.381 0 0.876 1.8685
357 TRAP  Y  -1.02 0 0.961 1.8685
357 TRAP  Y  0 -1.02 0 0.96
358 TRAP  Y  0 -0.8 0 3.9985
358 TRAP  Y  -0.8 0 4 7.9985
358 TRAP  Y  0 -0.8 0 3.9985
358 TRAP  Y  -0.8 0 4 7.9985
359 TRAP  Y  0 -0.8 0 3.9985
359 TRAP  Y  -0.8 0 4 7.9985
359 TRAP  Y  0 -0.8 0 3.9985
359 TRAP  Y  -0.8 0 4 7.9985
360 TRAP  Y  0 -0.8 0 3.9985
360 TRAP  Y  -0.8 0 4 7.9985
360 TRAP  Y  0 -0.8 0 3.9985
360 TRAP  Y  -0.8 0 4 7.9985
361 TRAP  Y  0 -0.8 0 3.9985
361 TRAP  Y  -0.8 0 4 7.9985
361 TRAP  Y  0 -0.8 0 3.9985
361 TRAP  Y  -0.8 0 4 7.9985
362 TRAP  Y  0 -0.404 0 1.179
362 TRAP  Y  0 -0.236 0 1.1785
363 TRAP  Y  -0.719 0 0 2.4285
363 TRAP  Y  -0.344 0 1.307 2.4285
363 TRAP  Y  0 -0.344 0 1.3057
364 TRAP  Y  0 -0.236 0 1.1785
364 TRAP  Y  -0.236 0 1.18 2.3585
364 TRAP  Y  0 -0.453 0 2.3585
365 TRAP  Y  0 -0.306 0 1.6032
365 TRAP  Y  -0.306 0 1.605 3.4885
365 TRAP  Y  -0.607 0 0.803 3.4885
365 TRAP  Y  -0.453 -0.607 0 0.8012
366 TRAP  Y  0 -0.8 0 3.9985
366 TRAP  Y  -0.8 0 4 7.9985
366 TRAP  Y  0 -0.8 0 3.9985
366 TRAP  Y  -0.8 0 4 7.9985
367 TRAP  Y  0 -0.8 0 3.9985
367 TRAP  Y  -0.8 0 4 7.9985
367 TRAP  Y  0 -0.8 0 3.9985
367 TRAP  Y  -0.8 0 4 7.9985
368 TRAP  Y  0 -0.8 0 3.9985
368 TRAP  Y  -0.8 0 4 7.9985
368 TRAP  Y  0 -0.8 0 3.9985
368 TRAP  Y  -0.8 0 4 7.9985
369 TRAP  Y  0 -0.8 0 3.9985
369 TRAP  Y  -0.8 0 4 7.9985
369 TRAP  Y  0 -0.8 0 3.9985
369 TRAP  Y  -0.8 0 4 7.9985
370 UNI  Y  -0.249 0 5.8285
370 TRAP  Y  0 -0.553 0 3.5743
370 TRAP  Y  -0.553 -0.254 3.576 5.8285
371 TRAP  Y  -1.024 0 0.988 1.9185
371 TRAP  Y  0 -1.024 0 0.9863
371 TRAP  Y  -0.254 0 0 1.9185
372 TRAP  Y  0 -0.583 0 2.9135
372 TRAP  Y  -0.583 0 2.915 5.8285
372 UNI  Y  -0.249 0 5.8285
```

```
373 TRAP  Y  0 -0.8 0 3.9985
373 TRAP  Y  -0.8 0 4 7.9985
374 TRAP  Y  0 -0.8 0 3.9985
374 TRAP  Y  -0.8 0 4 7.9985
375 TRAP  Y  0 -0.8 0 3.9985
375 TRAP  Y  -0.8 0 4 7.9985
376 TRAP  Y  0 -0.8 0 3.9985
376 TRAP  Y  -0.8 0 4 7.9985
377 TRAP  Y  0 -0.583 0 2.9135
377 TRAP  Y  -0.583 0 2.915 5.8285
378 TRAP  Y  0 -0.766 0 2.2466
378 TRAP  Y  -0.766 0 2.248 4.8485
379 TRAP  Y  0 -0.8 0 3.9985
379 UNI  Y  -0.8 4 4.9485
379 TRAP  Y  -0.8 0 4.95 8.9485
380 TRAP  Y  0 -0.8 0 3.9985
380 UNI  Y  -0.8 4 5.0485
380 TRAP  Y  -0.8 0 5.05 9.0485
381 TRAP  Y  0 -0.8 0 3.9985
381 UNI  Y  -0.8 4 4.8985
381 TRAP  Y  -0.8 0 4.9 8.8985
382 TRAP  Y  -0.571 0 2.855 5.7085
382 TRAP  Y  0 -0.571 0 2.8535
383 TRAP  Y  0 -0.8 0 3.9985
383 UNI  Y  -0.8 4 4.9485
383 TRAP  Y  -0.8 0 4.95 8.9485
383 TRAP  Y  0 -0.8 0 3.9985
383 UNI  Y  -0.8 4 4.9485
383 TRAP  Y  -0.8 0 4.95 8.9485
384 TRAP  Y  0 -0.8 0 3.9985
384 UNI  Y  -0.8 4 5.0485
384 TRAP  Y  -0.8 0 5.05 9.0485
384 TRAP  Y  0 -0.8 0 3.9985
384 UNI  Y  -0.8 4 5.0485
384 TRAP  Y  -0.8 0 5.05 9.0485
385 TRAP  Y  0 -0.8 0 3.9985
385 UNI  Y  -0.8 4 4.8985
385 TRAP  Y  -0.8 0 4.9 8.8985
385 TRAP  Y  0 -0.8 0 3.9985
385 UNI  Y  -0.8 4 4.8985
385 TRAP  Y  -0.8 0 4.9 8.8985
386 TRAP  Y  -0.571 0 2.855 5.7085
386 TRAP  Y  0 -0.571 0 2.8535
386 TRAP  Y  -0.571 0 2.855 5.7085
386 TRAP  Y  0 -0.571 0 2.8535
387 TRAP  Y  0 -0.8 0 3.9985
387 UNI  Y  -0.8 4 4.9485
387 TRAP  Y  -0.8 0 4.95 8.9485
387 TRAP  Y  0 -0.8 0 3.9985
387 UNI  Y  -0.8 4 4.9485
387 TRAP  Y  -0.8 0 4.95 8.9485
388 TRAP  Y  0 -0.8 0 3.9985
388 UNI  Y  -0.8 4 5.0485
388 TRAP  Y  -0.8 0 5.05 9.0485
388 TRAP  Y  0 -0.8 0 3.9985
388 UNI  Y  -0.8 4 5.0485
388 TRAP  Y  -0.8 0 5.05 9.0485
389 TRAP  Y  0 -0.8 0 3.9985
```

```
389 UNI  Y  -0.8 4 4.8985
389 TRAP  Y  -0.8 0 4.9 8.8985
389 TRAP  Y  0 -0.8 0 3.9985
389 UNI  Y  -0.8 4 4.8985
389 TRAP  Y  -0.8 0 4.9 8.8985
390 TRAP  Y  -0.571 0 2.855 5.7085
390 TRAP  Y  0 -0.571 0 2.8535
390 TRAP  Y  -0.571 0 2.855 5.7085
390 TRAP  Y  0 -0.571 0 2.8535
391 TRAP  Y  0 -0.8 0 3.9985
391 UNI  Y  -0.8 4 4.9485
391 TRAP  Y  -0.8 0 4.95 8.9485
391 TRAP  Y  0 -0.8 0 3.9985
391 UNI  Y  -0.8 4 4.9485
391 TRAP  Y  -0.8 0 4.95 8.9485
392 TRAP  Y  0 -0.8 0 3.9985
392 UNI  Y  -0.8 4 5.0485
392 TRAP  Y  -0.8 0 5.05 9.0485
392 TRAP  Y  0 -0.8 0 3.9985
392 UNI  Y  -0.8 4 5.0485
392 TRAP  Y  -0.8 0 5.05 9.0485
393 TRAP  Y  0 -0.8 0 3.9985
393 UNI  Y  -0.8 4 4.8985
393 TRAP  Y  -0.8 0 4.9 8.8985
393 TRAP  Y  0 -0.8 0 3.9985
393 UNI  Y  -0.8 4 4.8985
393 TRAP  Y  -0.8 0 4.9 8.8985
394 TRAP  Y  -0.571 0 2.855 5.7085
394 TRAP  Y  0 -0.571 0 2.8535
394 TRAP  Y  -0.571 0 2.855 5.7085
394 TRAP  Y  0 -0.571 0 2.8535
395 TRAP  Y  0 -0.8 0 3.9985
395 UNI  Y  -0.8 4 4.9485
395 TRAP  Y  -0.8 -0.498 4.95 6.4585
395 TRAP  Y  0 -0.583 0 2.9135
395 UNI  Y  -0.583 2.915 3.5435
395 TRAP  Y  -0.583 0 3.545 6.4585
396 TRAP  Y  -0.498 0 0 2.4885
397 TRAP  Y  0 -0.8 0 3.9985
397 UNI  Y  -0.8 4 5.0485
397 TRAP  Y  -0.8 -0.65 5.05 5.7985
397 TRAP  Y  -0.684 0 2.844 5.7985
397 TRAP  Y  0 -0.684 0 2.8422
398 TRAP  Y  -0.65 -0.325 0 1.6235
398 UNI  Y  -0.236 1.18 1.6235
398 TRAP  Y  0 -0.236 0 1.1785
399 TRAP  Y  0 -0.8 0 3.9985
399 UNI  Y  -0.8 4 4.8985
399 TRAP  Y  -0.8 0 4.9 8.8985
399 TRAP  Y  -0.354 0 3.89 8.8985
399 TRAP  Y  0 -0.354 0 3.8887
400 TRAP  Y  -0.571 0 2.855 5.7085
400 TRAP  Y  0 -0.571 0 2.8535
400 TRAP  Y  -0.125 0 1.952 5.7085
400 TRAP  Y  0 -0.125 0 1.9501
401 UNI  Y  -0.236 1.18 1.6235
401 TRAP  Y  0 -0.236 0 1.1785
401 UNI  Y  -0.299 1.572 1.6235
```

```
 401 TRAP  Y  0 -0.299 0 1.5711
 402 TRAP  Y  0 -0.583 0 2.9135
 402 UNI  Y  -0.583 2.915 3.5435
 402 TRAP  Y  -0.583 0 3.545 6.4585
 402 TRAP  Y  -0.36 -0.178 3.921 6.4585
 402 TRAP  Y  0 -0.36 0 3.9192
 403 TRAP  Y  -0.178 0 0 2.4885
 404 TRAP  Y  -0.284 0 1.498 3.417
 404 TRAP  Y  0 -0.284 0 1.4963
 405 TRAP  Y  -0.119 0 3.779 6.0069
 405 TRAP  Y  0 -0.119 0 3.7777
 406 TRAP  Y  -0.342 0 4.815 9.4159
 406 TRAP  Y  0 -0.342 0 4.8131
 407 TRAP  Y  -0.65 0 2.629 6.1018
 407 TRAP  Y  0 -0.65 0 2.6277
 408 TRAP  Y  -0.337 0 4.8 9.3653
 408 TRAP  Y  0 -0.337 0 4.7989
 409 TRAP  Y  -0.325 0 0 1.6235
 409 TRAP  Y  -0.236 0 0.445 1.6235
 409 UNI  Y  -0.236 0 0.4435
 410 TRAP  Y  -0.404 -0.76 0 1.0382
 410 TRAP  Y  -0.76 -0.719 1.04 1.178
 410 TRAP  Y  -0.236 0 0 1.178
 411 TRAP  Y  -0.236 0 0.445 1.6235
 411 UNI  Y  -0.236 0 0.4435
 411 TRAP  Y  -0.29 0 0 1.6235
 LOAD COMB 11 1.5D+1.5L
 9 1.5 10 1.5
 LOAD COMB 20 1.2D+1.2L+1.2WX
 9 1.2 10 1.2 5 1.2
 LOAD COMB 21 1.2D+1.2L-1.2WX
 9 1.2 10 1.2 5 -1.2
 LOAD COMB 22 1.2D+1.2L+1.2WZ
 9 1.2 10 1.2 6 1.2
 LOAD COMB 23 1.2D+1.2L-1.2WZ
 9 1.2 10 1.2 6 -1.2
 LOAD COMB 36 1.5D+1.5WX
 9 1.5 5 1.5
 LOAD COMB 37 1.5D-1.5WX
 9 1.5 5 -1.5
 LOAD COMB 38 1.5D+1.5WZ
 9 1.5 6 1.5
 LOAD COMB 39 1.5D-1.5WZ
 9 1.5 6 -1.5
 LOAD COMB 52 0.9D+1.5WX
 9 0.9 5 1.5
 LOAD COMB 53 0.9D-1.5WX
 9 0.9 5 -1.5
 LOAD COMB 54 0.9D+1.5WZ
 9 0.9 6 1.5
 LOAD COMB 55 0.9D-1.5WZ
 9 0.9 6 -1.5
PERFORM ANALYSIS
PRINT MEMBER INFORMATION
UNIT METERS MTONS
PRINT MEMBER FORCES
PRINT ELEMENT STRESS
PRINT SUPPORT REACTIONS
```

```
UNIT CM MTONS
PRINT DISPLACEMENTS
FINISH
```

EX. Bridge Deck Loading Example

This example demonstrates how to apply vehicle loads per BS 5400 Part 2 on two lanes of a two span, steel girder bridge using the Bridge Deck workflow.

EX. To open the model in Bridge Deck workflow

1. Create a new file in STAAD.Pro.
2. Open the file and copy/paste the EX. Bridge Deck Loading Input File (on page 6991).
3. Run an analysis on the structure.
4. Select **Bridge Deck** in the **Workflows** panel.

   Note: If you do not have a license for this module, you will not be able to proceed.

   The **Bridge Deck** ribbon tab opens.

EX. To define the bridge deck

1. (Optional) If the mouse pointer is not already displayed as the Plate Cursor, in the **Analytical Modeling** workflow on the **Geometry** ribbon tab, select the **Plates Cursor** tool in the **Selection** group.

2. Press **<CTRL+A>**.
   All plate elements are selected in the entire model.
3. In the **Bridge Deck** workflow on the **Bridge Deck** ribbon tab, select the **Create Deck** tool in the **Deck** group.

   The **Save Deck as** dialog opens.
4. Type `Northbound` for the Name.
5. Click **OK**.

EX. To generate the influence surface for the deck

1. On the **Bridge Deck** ribbon tab, select the **Loading > Influence Surface Generator** tool in the **Loading** group.

A temporary STAAD input file (*filename*_deck.std) is sent to the STAAD analysis engine for processing. The **STAAD Analysis and Design** dialog opens to display the progress. When the analysis is complete, the dialog closes automatically.

2. On the **Bridge Deck** ribbon tab, select the **Loading > Influence Diagram** tool in the **Loading** group.
   The Influence tab (on page 2987) opens.
3. Select the **Diagram Type** as **Node Displacement.**
   The **Parameter** field updates to display **Node Displacement** and the object field updates to **Node.**
4. Select the **Node Displacement** as **Y** (Global Y direction).
5. Select the **Node** as **104** (approximately center of the deck, mid-span of the longer span).
6. Click **OK.**
   The influence surface for the vertical displacement at node 104 is drawn on the deck.

EX. To define the roadway lanes

Note: The deck is selected in the **Bridge Deck** ribbon tab **Deck** group by default, since only one deck has been defined. Otherwise, it would be necessary to select the correct deck on which a new roadway is to be defined.

1. On the **Bridge Deck** ribbon tab, select the **Define Roadway** tool in the **Deck** group.

   The Roadways dialog (on page 2974) opens.
2. Click **New.**
   The Define Roadway dialog (on page 2976) opens.
3. Type `0.61` m for Curb A: Origin Z value and `6.35` m for Curb B: Origin Z value. Leave all other values as the default.
   The preview updates to display the roadway on the currently selected deck.
4. Click **OK.**
   Roadway **CW 1** is now added to the Roadways list in the **Roadways** dialog and is outlined in blue on the deck in the active view window.
5. Click **Close.**

EX. To place automatically generated loads on the roadway

1. On the **Bridge Deck** ribbon tab, select the **Loading** > **Run Load Generator** tool in the **Loading** group.
   The Load Generator Parameters dialog (on page 2981) opens.
2. On the General tab, Select **BS5400 Part 2** as the **Design Code.**
   The **<code>** tab updates to **BS5400.**
3. Select **Ultimate** for the **Limit State.**
4. Select the BS 5400 Specific Parameters (on page 2983).

   Note: For this example, the default single load case with the nominal HB vehicle and unit of 30 are OK.

5. Select the Node Displacements tab.

Note: An action must be defined for which the maximum results determine the placement of the moving loads. This example will use the same action as reviewed above in the influence surface.

6. Enter the following data into the Node Displacements table:

Node	Displacement	Effect
104	Y	+ve
104	Y	-ve

7. Click **OK**.
   The program analyzes the deck to obtain the critical load positions for the specified action.

A summary of the analysis is opened in your default text editor (e.g., Notepad). The file (named `filename_deckx.out`) is saved in the same location as the input file. Close the text editor once you have completed reviewing this file.

EX. To review the generated loads graphically

1. Select **2:N04: Disp Y -ve** in the current load case drop-down list, found in the View toolbar.
2. Either:

   Select **View > Structure Diagrams...**.

   or

   Right-click anywhere in the active view window and then select **Structure Diagrams...** from the pop-up menu.

   The **Diagrams** dialog opens.
3. Select the Deck tab (on page 2988).
4. Select to display **Loads** and **Vehicles** in addition to Roadways (on by default).
5. Click **OK**.

The placement of the loads and vehicles are overlayed on the bridge deck. Note that the defined roadway was wide enough for two lanes of traffic, and these were created automatically.

EX. To transfer the load case to the STAAD.Pro model

To transfer the generated load cases to STAAD.Pro for analysis and design, do the following.

1. On the **Bridge Deck** ribbon tab, select the **Loading > Create Loading in STAAD Model** tool in the **Loading** group.
   A message dialog opens to confirm the load was successfully added in the STAAD input file.
2. Select the **Analytical Modeling** workflow.
3. On the **Loading** page, review the loads added to the input file in the **Load & Definitions** dialog.

Note: One load was added for each action requested in the **Load Generators Parameters** dialog.

4. (Optional) Select the **Design | Steel** page and define parameters for the BS5400 code to perform a member design in addition to analysis for the generated loads.
5. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

EX. Bridge Deck Loading Input File

The following input is used for this example:

```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Mar-06
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 1 0 0; 3 1 0 1; 4 0 0 1; 5 2 0 0; 6 2 0 1; 7 3 0 0; 8 3 0 1;
9 4 0 0; 10 4 0 1; 11 5 0 0; 12 5 0 1; 13 6 0 0; 14 6 0 1; 15 7 0 0; 16 7 0 1;
17 8 0 0; 18 8 0 1; 19 9 0 0; 20 9 0 1; 21 10 0 0; 22 10 0 1; 23 11 0 0;
24 11 0 1; 25 12 0 0; 26 12 0 1; 27 13 0 0; 28 13 0 1; 29 14 0 0; 30 14 0 1;
31 15 0 0; 32 15 0 1; 33 16 0 0; 34 16 0 1; 35 17 0 0; 36 17 0 1; 37 18 0 0;
38 18 0 1; 39 19 0 0; 40 19 0 1; 41 20 0 0; 42 20 0 1; 43 21 0 0; 44 21 0 1;
45 22 0 0; 46 22 0 1; 47 23 0 0; 48 23 0 1; 49 24 0 0; 50 24 0 1; 51 25 0 0;
52 25 0 1; 53 26 0 0; 54 26 0 1; 55 27 0 0; 56 27 0 1; 57 28 0 0; 58 28 0 1;
59 29 0 0; 60 29 0 1; 61 30 0 0; 62 30 0 1; 63 1 0 2; 64 0 0 2; 65 2 0 2;
66 3 0 2; 67 4 0 2; 68 5 0 2; 69 6 0 2; 70 7 0 2; 71 8 0 2; 72 9 0 2;
73 10 0 2; 74 11 0 2; 75 12 0 2; 76 13 0 2; 77 14 0 2; 78 15 0 2; 79 16 0 2;
80 17 0 2; 81 18 0 2; 82 19 0 2; 83 20 0 2; 84 21 0 2; 85 22 0 2; 86 23 0 2;
87 24 0 2; 88 25 0 2; 89 26 0 2; 90 27 0 2; 91 28 0 2; 92 29 0 2; 93 30 0 2;
94 1 0 3; 95 0 0 3; 96 2 0 3; 97 3 0 3; 98 4 0 3; 99 5 0 3; 100 6 0 3;
101 7 0 3; 102 8 0 3; 103 9 0 3; 104 10 0 3; 105 11 0 3; 106 12 0 3;
107 13 0 3; 108 14 0 3; 109 15 0 3; 110 16 0 3; 111 17 0 3; 112 18 0 3;
113 19 0 3; 114 20 0 3; 115 21 0 3; 116 22 0 3; 117 23 0 3; 118 24 0 3;
119 25 0 3; 120 26 0 3; 121 27 0 3; 122 28 0 3; 123 29 0 3; 124 30 0 3;
125 1 0 4; 126 0 0 4; 127 2 0 4; 128 3 0 4; 129 4 0 4; 130 5 0 4; 131 6 0 4;
132 7 0 4; 133 8 0 4; 134 9 0 4; 135 10 0 4; 136 11 0 4; 137 12 0 4;
138 13 0 4; 139 14 0 4; 140 15 0 4; 141 16 0 4; 142 17 0 4; 143 18 0 4;
144 19 0 4; 145 20 0 4; 146 21 0 4; 147 22 0 4; 148 23 0 4; 149 24 0 4;
150 25 0 4; 151 26 0 4; 152 27 0 4; 153 28 0 4; 154 29 0 4; 155 30 0 4;
156 1 0 5; 157 0 0 5; 158 2 0 5; 159 3 0 5; 160 4 0 5; 161 5 0 5; 162 6 0 5;
163 7 0 5; 164 8 0 5; 165 9 0 5; 166 10 0 5; 167 11 0 5; 168 12 0 5;
169 13 0 5; 170 14 0 5; 171 15 0 5; 172 16 0 5; 173 17 0 5; 174 18 0 5;
175 19 0 5; 176 20 0 5; 177 21 0 5; 178 22 0 5; 179 23 0 5; 180 24 0 5;
181 25 0 5; 182 26 0 5; 183 27 0 5; 184 28 0 5; 185 29 0 5; 186 30 0 5;
187 1 0 6; 188 0 0 6; 189 2 0 6; 190 3 0 6; 191 4 0 6; 192 5 0 6; 193 6 0 6;
194 7 0 6; 195 8 0 6; 196 9 0 6; 197 10 0 6; 198 11 0 6; 199 12 0 6;
200 13 0 6; 201 14 0 6; 202 15 0 6; 203 16 0 6; 204 17 0 6; 205 18 0 6;
206 19 0 6; 207 20 0 6; 208 21 0 6; 209 22 0 6; 210 23 0 6; 211 24 0 6;
212 25 0 6; 213 26 0 6; 214 27 0 6; 215 28 0 6; 216 29 0 6; 217 30 0 6;
218 1 0 7; 219 0 0 7; 220 2 0 7; 221 3 0 7; 222 4 0 7; 223 5 0 7; 224 6 0 7;
225 7 0 7; 226 8 0 7; 227 9 0 7; 228 10 0 7; 229 11 0 7; 230 12 0 7;
231 13 0 7; 232 14 0 7; 233 15 0 7; 234 16 0 7; 235 17 0 7; 236 18 0 7;
237 19 0 7; 238 20 0 7; 239 21 0 7; 240 22 0 7; 241 23 0 7; 242 24 0 7;
243 25 0 7; 244 26 0 7; 245 27 0 7; 246 28 0 7; 247 29 0 7; 248 30 0 7;
249 0 -5 0; 250 0 -5 1; 251 0 -5 2; 252 0 -5 3; 253 0 -5 4; 254 0 -5 5;
255 0 -5 6; 256 0 -5 7; 257 20 -5 0; 258 20 -5 1; 259 20 -5 2; 260 20 -5 3;
261 20 -5 4; 262 20 -5 5; 263 20 -5 6; 264 20 -5 7; 265 30 -5 0; 266 30 -5 1;
267 30 -5 2; 268 30 -5 3; 269 30 -5 4; 270 30 -5 5; 271 30 -5 6; 272 30 -5 7;
MEMBER INCIDENCES
986 1 2; 989 41 43; 990 43 45; 991 45 47; 992 47 49; 993 49 51; 994 51 53;
```

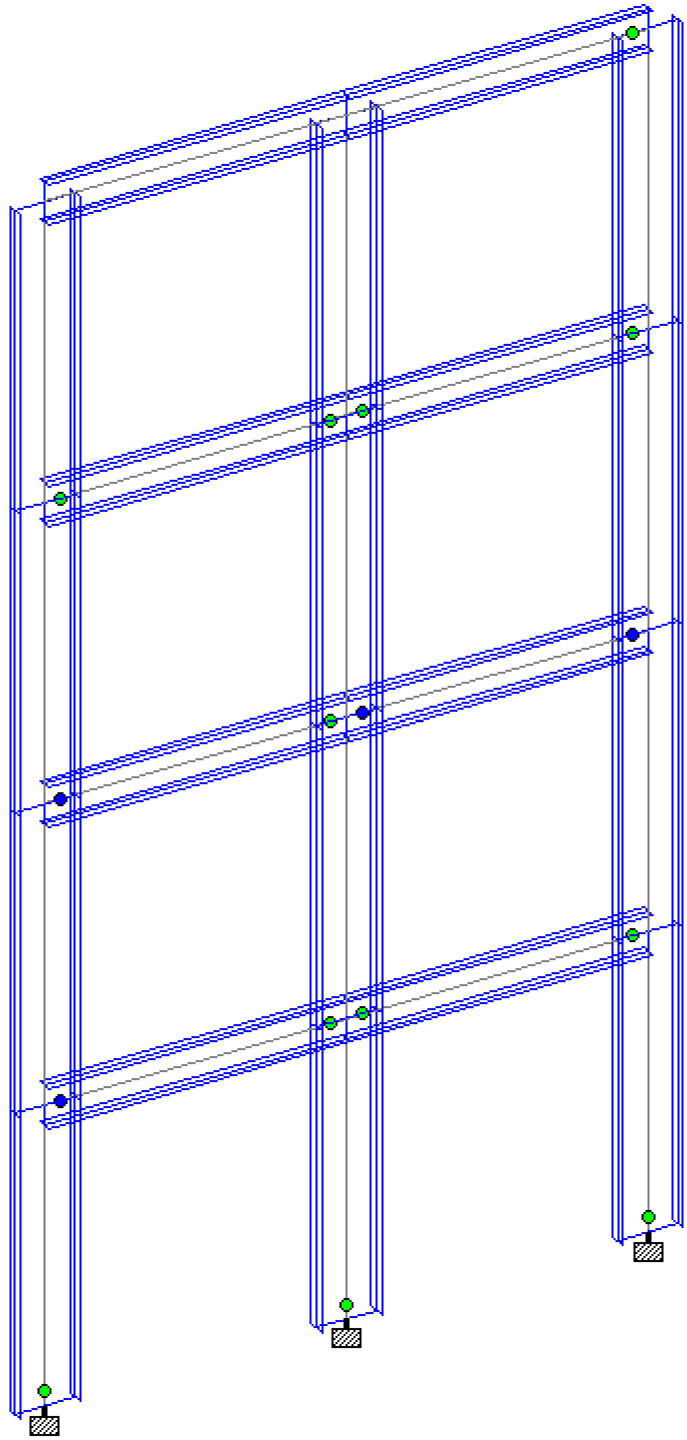
```
995 53 55; 996 55 57; 997 57 59; 998 59 61; 999 2 5; 1000 5 7; 1001 7 9;
1002 9 11; 1003 11 13; 1004 13 15; 1005 15 17; 1006 17 19; 1007 19 21;
1008 21 23; 1009 23 25; 1010 25 27; 1011 27 29; 1012 29 31; 1013 31 33;
1014 33 35; 1015 35 37; 1016 37 39; 1017 39 41; 1018 4 3; 1019 42 44;
1020 44 46; 1021 46 48; 1022 48 50; 1023 50 52; 1024 52 54; 1025 54 56;
1026 56 58; 1027 58 60; 1028 60 62; 1029 3 6; 1030 6 8; 1031 8 10; 1032 10 12;
1033 12 14; 1034 14 16; 1035 16 18; 1036 18 20; 1037 20 22; 1038 22 24;
1039 24 26; 1040 26 28; 1041 28 30; 1042 30 32; 1043 32 34; 1044 34 36;
1045 36 38; 1046 38 40; 1047 40 42; 1048 64 63; 1049 83 84; 1050 84 85;
1051 85 86; 1052 86 87; 1053 87 88; 1054 88 89; 1055 89 90; 1056 90 91;
1057 91 92; 1058 92 93; 1059 63 65; 1060 65 66; 1061 66 67; 1062 67 68;
1063 68 69; 1064 69 70; 1065 70 71; 1066 71 72; 1067 72 73; 1068 73 74;
1069 74 75; 1070 75 76; 1071 76 77; 1072 77 78; 1073 78 79; 1074 79 80;
1075 80 81; 1076 81 82; 1077 82 83; 1078 95 94; 1079 114 115; 1080 115 116;
1081 116 117; 1082 117 118; 1083 118 119; 1084 119 120; 1085 120 121;
1086 121 122; 1087 122 123; 1088 123 124; 1089 94 96; 1090 96 97; 1091 97 98;
1092 98 99; 1093 99 100; 1094 100 101; 1095 101 102; 1096 102 103;
1097 103 104; 1098 104 105; 1099 105 106; 1100 106 107; 1101 107 108;
1102 108 109; 1103 109 110; 1104 110 111; 1105 111 112; 1106 112 113;
1107 113 114; 1108 126 125; 1109 145 146; 1110 146 147; 1111 147 148;
1112 148 149; 1113 149 150; 1114 150 151; 1115 151 152; 1116 152 153;
1117 153 154; 1118 154 155; 1119 125 127; 1120 127 128; 1121 128 129;
1122 129 130; 1123 130 131; 1124 131 132; 1125 132 133; 1126 133 134;
1127 134 135; 1128 135 136; 1129 136 137; 1130 137 138; 1131 138 139;
1132 139 140; 1133 140 141; 1134 141 142; 1135 142 143; 1136 143 144;
1137 144 145; 1138 157 156; 1139 176 177; 1140 177 178; 1141 178 179;
1142 179 180; 1143 180 181; 1144 181 182; 1145 182 183; 1146 183 184;
1147 184 185; 1148 185 186; 1149 156 158; 1150 158 159; 1151 159 160;
1152 160 161; 1153 161 162; 1154 162 163; 1155 163 164; 1156 164 165;
1157 165 166; 1158 166 167; 1159 167 168; 1160 168 169; 1161 169 170;
1162 170 171; 1163 171 172; 1164 172 173; 1165 173 174; 1166 174 175;
1167 175 176; 1168 188 187; 1169 207 208; 1170 208 209; 1171 209 210;
1172 210 211; 1173 211 212; 1174 212 213; 1175 213 214; 1176 214 215;
1177 215 216; 1178 216 217; 1179 187 189; 1180 189 190; 1181 190 191;
1182 191 192; 1183 192 193; 1184 193 194; 1185 194 195; 1186 195 196;
1187 196 197; 1188 197 198; 1189 198 199; 1190 199 200; 1191 200 201;
1192 201 202; 1193 202 203; 1194 203 204; 1195 204 205; 1196 205 206;
1197 206 207; 1198 219 218; 1199 238 239; 1200 239 240; 1201 240 241;
1202 241 242; 1203 242 243; 1204 243 244; 1205 244 245; 1206 245 246;
1207 246 247; 1208 247 248; 1209 218 220; 1210 220 221; 1211 221 222;
1212 222 223; 1213 223 224; 1214 224 225; 1215 225 226; 1216 226 227;
1217 227 228; 1218 228 229; 1219 229 230; 1220 230 231; 1221 231 232;
1222 232 233; 1223 233 234; 1224 234 235; 1225 235 236; 1226 236 237;
1227 237 238; 1228 249 1; 1229 250 4; 1230 251 64; 1231 252 95; 1232 253 126;
1233 254 157; 1234 255 188; 1235 256 219; 1236 257 41; 1237 258 42;
1238 259 83; 1239 260 114; 1240 261 145; 1241 262 176; 1242 263 207;
1243 264 238; 1244 265 61; 1245 266 62; 1246 267 93; 1247 268 124;
1248 269 155; 1249 270 186; 1250 271 217; 1251 272 248;
ELEMENT INCIDENCES SHELL
776 1 2 3 4; 777 2 5 6 3; 778 5 7 8 6; 779 7 9 10 8; 780 9 11 12 10;
781 11 13 14 12; 782 13 15 16 14; 783 15 17 18 16; 784 17 19 20 18;
785 19 21 22 20; 786 21 23 24 22; 787 23 25 26 24; 788 25 27 28 26;
789 27 29 30 28; 790 29 31 32 30; 791 31 33 34 32; 792 33 35 36 34;
793 35 37 38 36; 794 37 39 40 38; 795 39 41 42 40; 796 4 3 63 64;
797 3 6 65 63; 798 6 8 66 65; 799 8 10 67 66; 800 10 12 68 67; 801 12 14 69
68;
802 14 16 70 69; 803 16 18 71 70; 804 18 20 72 71; 805 20 22 73 72;
806 22 24 74 73; 807 24 26 75 74; 808 26 28 76 75; 809 28 30 77 76;
```

```
810 30 32 78 77; 811 32 34 79 78; 812 34 36 80 79; 813 36 38 81 80;
814 38 40 82 81; 815 40 42 83 82; 816 64 63 94 95; 817 63 65 96 94;
818 65 66 97 96; 819 66 67 98 97; 820 67 68 99 98; 821 68 69 100 99;
822 69 70 101 100; 823 70 71 102 101; 824 71 72 103 102; 825 72 73 104 103;
826 73 74 105 104; 827 74 75 106 105; 828 75 76 107 106; 829 76 77 108 107;
830 77 78 109 108; 831 78 79 110 109; 832 79 80 111 110; 833 80 81 112 111;
834 81 82 113 112; 835 82 83 114 113; 836 95 94 125 126; 837 94 96 127 125;
838 96 97 128 127; 839 97 98 129 128; 840 98 99 130 129; 841 99 100 131 130;
842 100 101 132 131; 843 101 102 133 132; 844 102 103 134 133;
845 103 104 135 134; 846 104 105 136 135; 847 105 106 137 136;
848 106 107 138 137; 849 107 108 139 138; 850 108 109 140 139;
851 109 110 141 140; 852 110 111 142 141; 853 111 112 143 142;
854 112 113 144 143; 855 113 114 145 144; 856 126 125 156 157;
857 125 127 158 156; 858 127 128 159 158; 859 128 129 160 159;
860 129 130 161 160; 861 130 131 162 161; 862 131 132 163 162;
863 132 133 164 163; 864 133 134 165 164; 865 134 135 166 165;
866 135 136 167 166; 867 136 137 168 167; 868 137 138 169 168;
869 138 139 170 169; 870 139 140 171 170; 871 140 141 172 171;
872 141 142 173 172; 873 142 143 174 173; 874 143 144 175 174;
875 144 145 176 175; 876 157 156 187 188; 877 156 158 189 187;
878 158 159 190 189; 879 159 160 191 190; 880 160 161 192 191;
881 161 162 193 192; 882 162 163 194 193; 883 163 164 195 194;
884 164 165 196 195; 885 165 166 197 196; 886 166 167 198 197;
887 167 168 199 198; 888 168 169 200 199; 889 169 170 201 200;
890 170 171 202 201; 891 171 172 203 202; 892 172 173 204 203;
893 173 174 205 204; 894 174 175 206 205; 895 175 176 207 206;
896 188 187 218 219; 897 187 189 220 218; 898 189 190 221 220;
899 190 191 222 221; 900 191 192 223 222; 901 192 193 224 223;
902 193 194 225 224; 903 194 195 226 225; 904 195 196 227 226;
905 196 197 228 227; 906 197 198 229 228; 907 198 199 230 229;
908 199 200 231 230; 909 200 201 232 231; 910 201 202 233 232;
911 202 203 234 233; 912 203 204 235 234; 913 204 205 236 235;
914 205 206 237 236; 915 206 207 238 237; 916 41 43 44 42; 917 43 45 46 44;
918 45 47 48 46; 919 47 49 50 48; 920 49 51 52 50; 921 51 53 54 52;
922 53 55 56 54; 923 55 57 58 56; 924 57 59 60 58; 925 59 61 62 60;
926 42 44 84 83; 927 44 46 85 84; 928 46 48 86 85; 929 48 50 87 86;
930 50 52 88 87; 931 52 54 89 88; 932 54 56 90 89; 933 56 58 91 90;
934 58 60 92 91; 935 60 62 93 92; 936 83 84 115 114; 937 84 85 116 115;
938 85 86 117 116; 939 86 87 118 117; 940 87 88 119 118; 941 88 89 120 119;
942 89 90 121 120; 943 90 91 122 121; 944 91 92 123 122; 945 92 93 124 123;
946 114 115 146 145; 947 115 116 147 146; 948 116 117 148 147;
949 117 118 149 148; 950 118 119 150 149; 951 119 120 151 150;
952 120 121 152 151; 953 121 122 153 152; 954 122 123 154 153;
955 123 124 155 154; 956 145 146 177 176; 957 146 147 178 177;
958 147 148 179 178; 959 148 149 180 179; 960 149 150 181 180;
961 150 151 182 181; 962 151 152 183 182; 963 152 153 184 183;
964 153 154 185 184; 965 154 155 186 185; 966 176 177 208 207;
967 177 178 209 208; 968 178 179 210 209; 969 179 180 211 210;
970 180 181 212 211; 971 181 182 213 212; 972 182 183 214 213;
973 183 184 215 214; 974 184 185 216 215; 975 185 186 217 216;
976 207 208 239 238; 977 208 209 240 239; 978 209 210 241 240;
979 210 211 242 241; 980 211 212 243 242; 981 212 213 244 243;
982 213 214 245 244; 983 214 215 246 245; 984 215 216 247 246;
985 216 217 248 247;
ELEMENT PROPERTY
776 TO 985 THICKNESS 0.2
DEFINE MATERIAL START
ISOTROPIC STEEL
```

```
E 2.05e+008
POISSON 0.3
DENSITY 76.8195
ALPHA 1.2e-005
DAMP 0.03
ISOTROPIC CONCRETE
E 2.17185e+007
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-005
DAMP 0.05
END DEFINE MATERIAL
MEMBER PROPERTY BRITISH
986 989 TO 1227 TABLE ST UB610X229X125
1228 TO 1251 TABLE ST UC356X406X340
CONSTANTS
MATERIAL STEEL ALL
MATERIAL CONCRETE MEMB 776 TO 985
SUPPORTS
249 TO 272 PINNED
LOAD 1 LOADTYPE None  TITLE LOAD CASE 1
SELFWEIGHT Y -1
PERFORM ANALYSIS
FINISH
```

EX. Pushover Analysis Example

This single moment frame analyzed using a Pushover Analysis.



EX. To create model used for pushover example

To quickly set up the model used for this example, use the following procedure.

Only one pushover definition can be added to a model. Therefore, this example with start with a model you can add the pushover definition to. The end result will be functionally equivalent to the example model installed with the program.

1. Start an empty STAAD.Pro model in the analytical modeling worfklow.

   Refer to GS. To create a new STAAD.Pro model (on page 35) for additional details.
2. On the **Utilities** ribbon tab, select the **Command File** tool in the **Edit** group.

   The STAAD.Pro Editor window opens.
3. Click **Copy to Clipboard** below.

```
STAAD PLANE
START JOB INFORMATION
ENGINEER DATE 07-Oct-05
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES KIP
JOINT COORDINATES
1 0 0 0; 2 0 118.11 0; 3 118.11 118.11 0; 4 118.11 0 0; 5 0 236.22 0;
6 118.11 236.22 0; 7 0 354.331 0; 8 118.11 354.331 0; 9 0 472.441 0;
10 118.11 472.441 0; 11 236.22 118.11 0; 12 236.22 0 0; 13 236.22 236.22 0;
14 236.22 354.331 0; 15 236.22 472.441 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 2 5; 5 5 6; 6 6 3; 7 5 7; 8 7 8; 9 8 6; 10 7 9; 11 9 10;
12 10 8; 13 3 11; 14 6 13; 15 8 14; 16 10 15; 17 11 12; 18 13 11; 19 14 13;
20 15 14;
DEFINE MATERIAL START
ISOTROPIC STEEL
E 29732.7
POISSON 0.3
DENSITY 0.000283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
2 5 13 14 TABLE ST W16X26
8 11 15 16 TABLE ST W16X26
1 3 4 6 17 18 TABLE ST W24X55
7 9 10 12 19 20 TABLE ST W24X55
CONSTANTS
MATERIAL STEEL MEMB 1 TO 20
SUPPORTS
1 4 12 FIXED

LOAD 1 LOADTYPE GRAVITY
SELFWEIGHT Y -1
MEMBER LOAD
2 5 8 11 13 TO 16 UNI GY -0.2

FINISH
```

4. In the STAAD.Pro Editor window, select the **Paste** tool on the **Home** ribbon tab.

The model contents are pasted into the editor.

5. Save and close the STAAD.Pro Editor.

The model opens in the Analytical Workflow.

EX. To define general pushover data

1. On the **Loading** ribbon tab, select the **Pushover** tool in the **Define Loading Systems** group.
   The **Create New Definitions / Load Cases / Load Items** dialog opens with only the Pushover tab displayed.
2. On the Define Input tab, select the **General Input Parameters** option.
3. Select **Moment Frame** as the **Type of Frame**.
4. Select **Ignore Effect** from the **Geometric Non-Linearity Effect** drop-down list.
5. Set the **Displacement Incremental Value** option and then type `0.010` (in) in the text field.
6. Click **Add**.
7. Select the **Member Specific Parameter** option.
8. Set the **Expected Yield Stress** option and then type `36.0` (kip/in^2) in the text field.

   Note: You will assign this parameter to members in a later procedure.
9. Click **Add**.

In the next procedure, you will define the loading pattern and spectrum data, so you can leave the Define Pushover dialog open.

EX. To define loading pattern and spectrum data

1. On the **Define Pushover** dialog, select the **Define Loading Pattern** tab.
2. Type `250` in the **Number of Push Load Steps**.
3. Click **Add**.
4. Select the **Define Spectrum Data** tab.
5. Type `5.0` (%) in the **1st Spectrum** text field.
6. Delete the values from the remaining spectrum fields.
7. Select **Class A** from the **Site Category** drop-down list.
8. Type `1.0` in both the **At Short Period** and **At One - Second Period** text fields.
9. Click **Add**.

In the next procedure, you will define the solution control, so you can leave the Define Pushover dialog open.

EX. To define the solution control

1. On the **Define Pushover** dialog, select the **Define Solution Control** tab.
2. Select the **Push Upto Defined Displacement at Control Joint** option.
3. Select **X Axis** for the **Direction**.
4. Type `5.00` (in) in the **Joint Displacement Value** text field.

5. Select **10** as the **Joint Number**.

   The drop-down list contains the node numbers within the entire model.
6. Click **Add**.
7. Click **Close**.

EX. To assign the member-specific parameters

1. On the **Load & Definition** dialog, expand the **Definitions** > **Pushover Definitions** section.
   You will see the input parameters for the pushover definition you have added.
2. Select the entry labeled **FYE 36.00000**.
3. In the Assignment Method group, select **Assign To View**.
4. Click **Assign**.
   You are prompted to confirm you want to assign to all visible members in the view.
5. Click **Yes**.

EX. To specify and run the pushover analysis

1. Either:

   select the **Analysis** page

   or

   on the **Analysis and Design** ribbon tab, select the **Analysis Commands** tool in the **Analysis Data** group

   The **Analysis/Print Commands** dialog opens.
2. Select the **Perform Pushover Analysis** tab.
3. Click **Add**.
4. Click **Close**.
5. On the **Analysis and Design** ribbon tab, select the **Run Analysis** tool in the **Analysis** group.

   The **STAAD Analysis and Design** dialog opens.
6. Select the **Go to Post Processing Mode** option.
7. Click **Done**.

The Postprocessing Workflow opens.

EX. To review pushover displacement results

If you did not select the Postprocessing option in the results dialog in the previous procedure, select the Postprocessing workflow.

1. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
2. Select the **Graphs** tool in the **Pushover** group.
   The Capacity Curve graph and Capacity Curve table open.

   Note: The last load step (63) has a displacement that exceeds the maximum of 5.0 in specified in the pushover parameters.
3. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
4. Select the **Node Results** tool in the **Pushover** group.
   The Node Displacements and Support Reactions tables open.

   Note: These tables display the values for the entire structure for a selected load step. The deflected shape for the current load step is also displayed graphically.
5. Type 63 in the **Select Load Step** field and then press **<Enter>**.
   The tables and view window update.
6. On the **Results** ribbon tab, select the **Layouts** tool drop-down in the **Dynamics** group.
7. Select the **Beam Results** tool in the **Pushover** group.
   The Beam Hinge Results and Beam Force Detail tables open.
8. Repeat clicking the up arrow adjacent to either **Select Load Step** field until load step **4** is displayed.

   Note: The first hinge initially forms at this step. This is denoted with a green hinge, representing immediate occupancy status.
9. Repeat clicking the up arrow until about load step 44 is reached.
   Several more hinges have formed at the top of the elastic range and the first hinge has now turned blue, representing the Life Safety range of the acceptance criteria.

D
OpenSTAAD

Application programming interface for STAAD.Pro.

OS. Fundamentals of OpenSTAAD

What is OpenSTAAD?

OpenSTAAD is a library of exposed functions allowing engineers access to STAAD.Pro's internal functions and routines as well as its graphical commands. OpenSTAAD allows engineers and other users to link in-house or third-party applications with STAAD.Pro.

Where can I use OpenSTAAD?

OpenSTAAD can be used to write macros or scripts within third-party applications, such as Microsoft® Office Excel® or Autodesk AutoCAD® using visual basic for applications. You can also write custom user tools (also called macros) to run from within STAAD.Pro using the STAAD.Pro Script Editor. Lastly, you can write custom scripts or even applications in other programming languages, such as Visual Basic, C#, Python, C++, VB.net and more.

Note: OpenSTAAD does not support .NET Core applications.

OpenSTAAD Requires STAAD.Pro

OpenSTAAD is an API for STAAD.Pro and you must have STAAD.Pro installed on the machine running OpenSTAAD in order to use it. Essentially, OpenSTAAD allows your spreadsheet or custom program to access the power of STAAD and work with STAAD models, but it requires STAAD.Pro in order to do so.

Note: Bentley Systems, Inc. also licenses OpenSTAAD to power standalone applications as "OpenSTAAD OEM". Please contact Bentley Systems for additional details.

OS. Application Program Interface (API)

The *OpenSTAAD* library of functions is classified under the following general categories:

- STAAD File Input and Output (I/O)
- Structure Geometry
- Member Specifications
- Properties

- Loads
- Output Results
    - Nodes
    - Beams
    - Plates
    - Solids
- STAAD Pre-Processor
- STAAD Post-Processor
- Creating Dialog Boxes and Menu Items

OS. Instantiating the OpenSTAAD Library for Use

The first thing necessary to access STAAD project data from within another application is to instantiate, or create an instance of OpenSTAAD within the other application. In Visual Basic for Applications (VBA), this may be done by creating an object variable and then assigning to it the OpenSTAAD object. The VBA **`GetObject`** function may be used for this.

The object which controls the STAAD.Pro environment is referred to as **`StaadPro.OpenSTAAD`**. This object must be created in order to get access to any of the internal graphical functions within STAAD.Pro (including the creating of menu items and dialog boxes) as well as access to STAAD's viewing, geometry modeling, results grid, and post-processing functions. The following VBA function can be used to instantiate or create this object:

```
Set MyObject = GetObject("filepath", "ojbectclass")
```

Where:

MyObject	the Object name declared in a previous **`Dim`** statement.
filepath	the optional string providing the full file path and name of the file containing objects to retrieve. In the case of OpenSTAAD, this can be omitted, along with the trailing comma.
objectclass	the string representing the class of the object. In the case of OpenSTAAD, this is always `Staadpro.OpenSTAAD`.

Thus:

```
Set objName = GetObject("StaadPro.OpenSTAAD")
```

At the conclusion of your OpenSTAAD application, the OpenSTAAD object(s) should be terminated, to unlock the handles within the application to the OpenSTAAD functions, and to free system resources.

Example

```
Sub How2Begin()
    'Create a variable to hold your OpenSTAAD object(s).

    Dim objOpenSTAAD As Object
    'Launch OpenSTAAD Object
    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")

    'At the end of your application, remember to terminate the OpenSTAAD
objects.
```

```
    Set objOpenSTAAD = Nothing
End Sub
```

OS. Function Return Value

Most OpenSTAAD functions return a value to either:

- indicate the success or failure of the function (a Boolean result), or
- results value of the function (a numeric value result).

Note: Functions that return values are indicated with a “variant” return some sort of value. Refer to the reference for each function to see the possible values. Functions indicated with a “void” do not return anything.

If a function returns a Boolean result and that return value for an OpenSTAAD function is equal to `0` (zero), it means that OpenSTAAD was unable to execute the function. If you see this result, check that you have passed all the required parameters to the function. Make sure that all parameters being passed are valid. A return value of `1` (one) indicates that OpenSTAAD successfully executed the function.

Unless specified otherwise, results returned by a function are stored in variable names passed to it for the purpose.

A few of the OpenSTAAD Application functions return the results as the return value of the function. In those cases, the above comments regarding the function return value do not apply.

If a function returns the value of the function,

```
Dim returnValue
returnValue = objOpenSTAAD.BooleanReturn (param1, param2, … ,paramn)
```

VB Example

```
objOpenSTAAD.BooleanReturn param1, param2, … ,paramn
'or, if you would like to capture the Boolean result for flagging:

functionFlag = objOpenSTAAD.BooleanReturn param1, param2, … ,paramn
```

OS. STAAD Nomenclature

In STAAD documentation and in the program menus and dialog boxes, the terms “member” and “beam” are used interchangeably. Use of the term “beam” should not be taken to imply that the member cannot take an axial load.

Similarly, the terms “joint” and “node” are used interchangeably in STAAD. Both terms refer to the connections between elements.

Connections are also referred to as incidences. The terms “member incidences,” “plate incidences,” and “solid incidences” refer to the nodes that connect these elements to other elements and supports.

OS. OpenSTAAD API Documentation

Programmer reference documentation for the OpenSTAAD API is installed separately at `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\Help\OSAPP\index.html` (typical location).

1. Select the **File** ribbon tab.
   The backstage view opens.
2. Select **Help** > **OpenSTAAD Help**.
   The OpenSTAAD Documentation opens in your default web browser.

OS. Using OpenSTAAD in Other Applications: VBA

Using Visual Basic for Applications (VBA), you can write macros *within* other software which also support VBA to utilize STAAD.Pro functionality in that application. You write the necessary code using VBA tools within that application.

With OpenSTAAD, you can use VBA macros to perform such tasks as automating repetitive modeling or post-processing tasks or embedding customized design routines. Following an open architecture paradigm, OpenSTAAD was built using ATL, COM, and COM+ standards as specified by Microsoft. This allows OpenSTAAD to be used in a macro application such as MicroStation, Microsoft Office Excel, or Autodesk AutoCAD.

For example, you could write a spreadsheet capable of sending model geometry to a STAAD file. You could similarly write a macro in a CAD program that will retrieve design results like member sizes to display in a CAD file.

OpenSTAAD is compatible with any version of Microsoft® Office Excel® that supports VBA macros.

OS. To create an Excel workbook macro

If you have not previously created any macros in Microsoft Office Excel, use this procedure to access the developer tools and set up a VBA module.

For additional details on using VBA in Microsoft Office applications, refer to https://docs.microsoft.com/en-us/office/vba/library-reference/concepts/getting-started-with-vba-in-office

1. Open an empty spreadsheet in Microsoft Office Excel.
2. Turn on the Developer ribbon tab:
   a. Select **Options** from the **File** ribbon tab.
      The **Excel Options** dialog opens.
   b. Select the **Customize Ribbon** tab in the dialog.
   c. Check the **Developer** option in the **Main Tabs** list.
   d. Click **OK**.

   For additional details, refer to https://support.office.com/en-us/article/show-the-developer-tab-e1192344-5e56-4d45-931b-e5fd9bea2d45.
3. In Excel, select the **Visual Basic** tool in the **Code** group on the **Developer** ribbon tab.
   The **Microsoft Visual Basic for Applications** window opens. This is the development environment in which you will write your VBA code to have Excel interact with STAAD.Pro via OpenSTAAD.

4. Select **Insert** > **Module**.
   A new folder (named **Modules**) and an empty Module is added to the VBA Project.
5. Create a subroutine which will contain your code:
   a. In the **Module1(Code)** window, type `Sub OS_Tutorial()`
   b. Press **<Return>**.
      The coding tool automatically adds the `End Sub` code for you.
6. Press **<Tab>** to indent your code.

> **Tip:** Though not necessary to make your code function well, indenting and aligning code can make it much easier to read by people. It is a good practice to indent the code within a subroutine, and indent additionally within If statements or other children of the subroutine.

7. Type `Sheet1.[A1].Value = "Hello World"`.

   This assigns the text string as the value of the cell A1 on worksheet Sheet1. This is a shorthand method in Excel VBA, where as you can also use `Sheet1.cells(1, 1)`.Value. You can find details on Excel VBA here: https://docs.microsoft.com/en-us/office/vba/api/excel.worksheet.cells

   https://docs.microsoft.com/en-us/office/vba/excel/concepts/cells-and-ranges/reference-cells-and-ranges
8. Run your macro:

   select **Run** > **Run Sub/User Form**

   or

   select the **Run Sub/User Form** tool

   or

   press **<F5>**
   Your spreadsheet (in the Excel program window) now says “Hello World” in cell A1.
9. Save your spreadsheet file.

> **Note:** You will have to save this Excel Spreadsheet as a “Excel Macro-Enabled Workbook (*.xslm)” format to maintain the VBA macro you just wrote.

While having your computer return “Hello World!” may be a classic example of a first program, it is not a very practical tool. However, you now have a file set up to create a more useful OpenSTAAD project in your next example.

OS. Connect a VBA Editor to STAAD

To connect your Visual Basic for Applications Editor to STAAD in order to use autocomplete, do the following.

This procedure is optional, but highly recommended in order to have your editing environment be able to assist you in writing code.

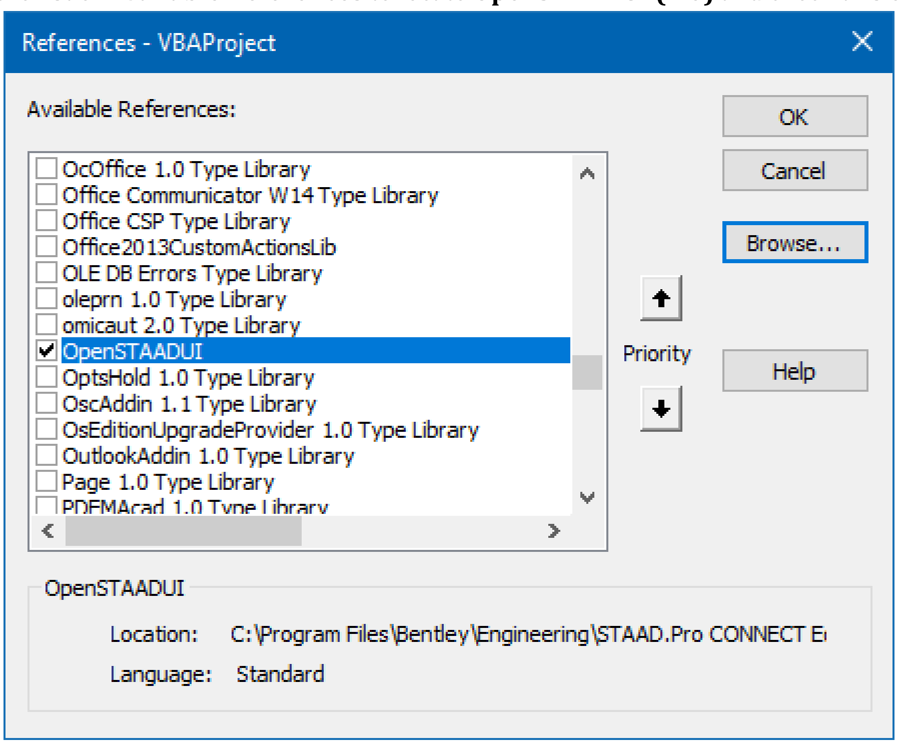
> **Tip:** The procedure below is show for Microsoft® Office Excel® , but is similar to how to perform this for most any VBA compatible application.

1. Start Microsoft Office Excel as an administrator.
   Performing the following steps as an administrator allows your to make a change to the template. This prevents you from having to re-perform this task every time you edit your VBA macro.
   a. From either the Windows Start menu or Windows Explorer, right-click on the Excel application.

b. Select **Run as administrator** from the pop-up menu.

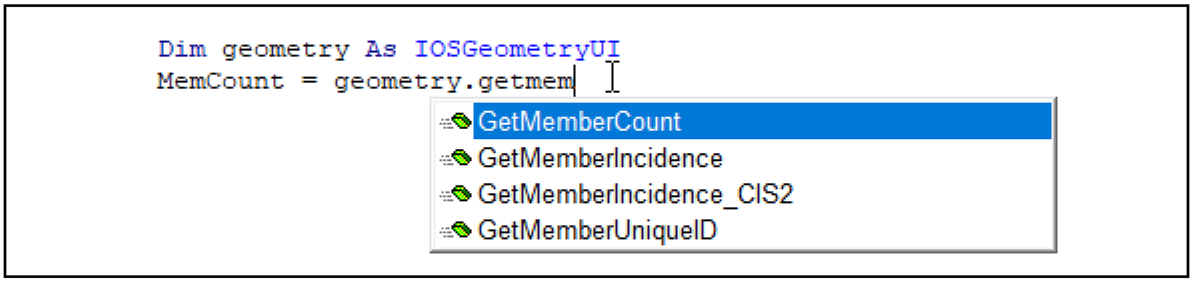
You are asked if you want to allow the application to make changes to your computer. Confirm you will allow this.

2. In Excel, select the **Visual Basic** tool in the **Code** group on the **Developer** ribbon tab.
   The **Microsoft Visual Basic for Applications** window opens.
3. Select **Tools** > **References**.
   The **Reference - VBA Project** dialog opens.
4. Click Browse.
   The **Add Reference** dialog opens.
5. Navigate to and select `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\StaadPro.dll` and then click **Open**.
6. Scroll through the list of **Available References** to locate **OpenSTAADUI (1.0)** and check this option.

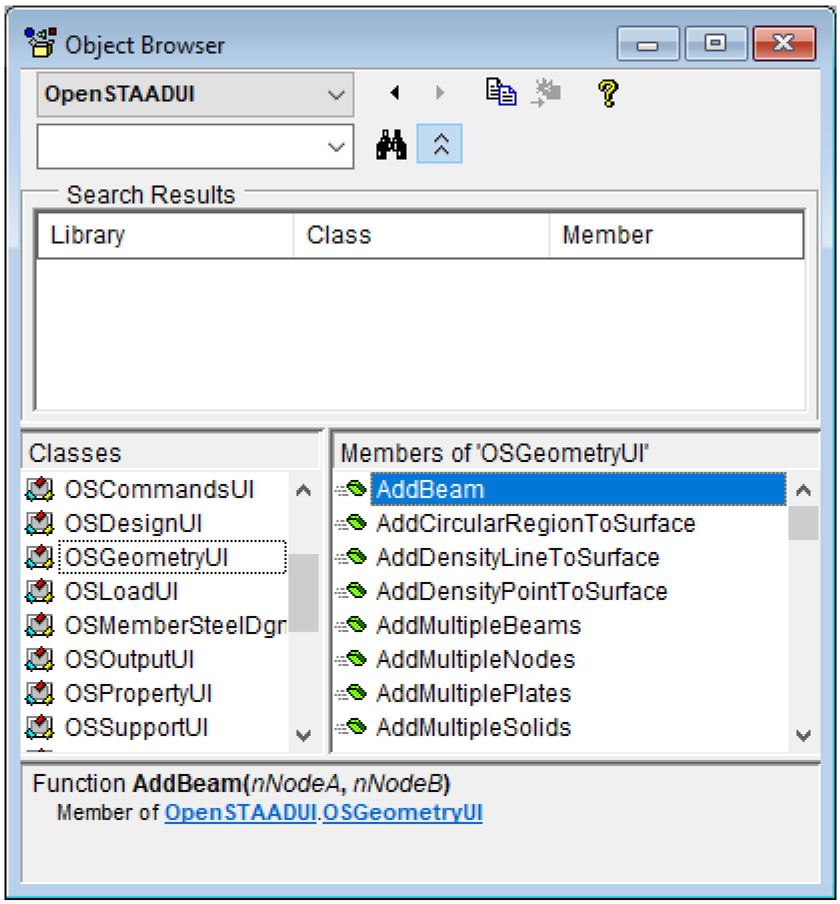


7. Click **OK**.

Now, you can set objects to the OpenSTAAD classes using this reference.



You can also now view the OpenSTAADUI object contents (classes and functions) in the Object Browser, by selecting the Object Browser tool ( ) in the VBA Editor.



OS. Write an OpenSTAAD Macro in Excel

This example will introduce you to several concepts necessary to writing OpenSTAAD macros in VBA.

Note: If you followed the previous “Hello World!” example, you may simply delete that line and start from that point.

OS. Instantiate OpenSTAAD in VBA

In the this set of steps, you will instantiate OpenSTAAD and have the spreadsheet macro retrieve information from the open STAAD.Pro model.

This is required in order to access OpenSTAAD API from within your code. It's a two part process in VBA: You first declare something as the object and then direct the that object at OpenSTAAD.

1. Within your subroutine, type `Dim objOpenSTAAD As Object`.

   Notice how as you begin typing the word "Object", a small list opens with relevant terms. This is called IntelliSense and it can help you quickly and accurately complete code lines.

   Note: As Visual Basic requires that objects be declared and assigned a type, it is a good practice to prefix the name with what type is used. Here, the prefix "obj" is used to describe an object. Later you'll use different prefixes to describe variable types.
2. Press **<Return>**.
3. Type `Set objOpenSTAAD = GetObject(,”StaadPro.OpenSTAAD”)`

   This assigns the OpenSTAAD application object to the variable you defined in Step 1.

   Note: This approach connects to the first instance of STAAD.Pro that was started where multiple instances exist. You can alternately specify a specific file path and name to connect to the instance of the application with that file open.
4. Press **<Return>**.

OS. Use geometry functions in spreadsheet

In order to successfully run an OpenSTAAD macro, you must first open a model within STAAD.Pro.

This example continues by using a function in OpenSTAAD to access model data.

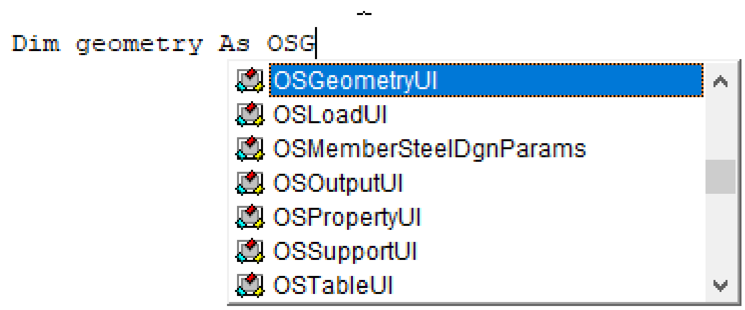
1. Type `Sheet1.[A1].Value = "Nodes:"` and then press **<Return>**.

   Much like the "Hello World" example, this simply populates a cell with a text string. Here, it's useful to provide a label for your data.
2. Type `Sheet1.[B1].Value = objOpenSTAAD.Geometry.GetNodeCount` and then press **<Return>**.

   Here, you are assigning the value of a cell to the returned value of the `GetNodeCount` function, which is in the `Geometry` class within the OpenSTAAD object.

   This approach accesses the GetNodeCount function directly within the objOpenSTAAD object.
3. Type `Dim objGeometry As OSGeometryUI` and then press **<Return>**.

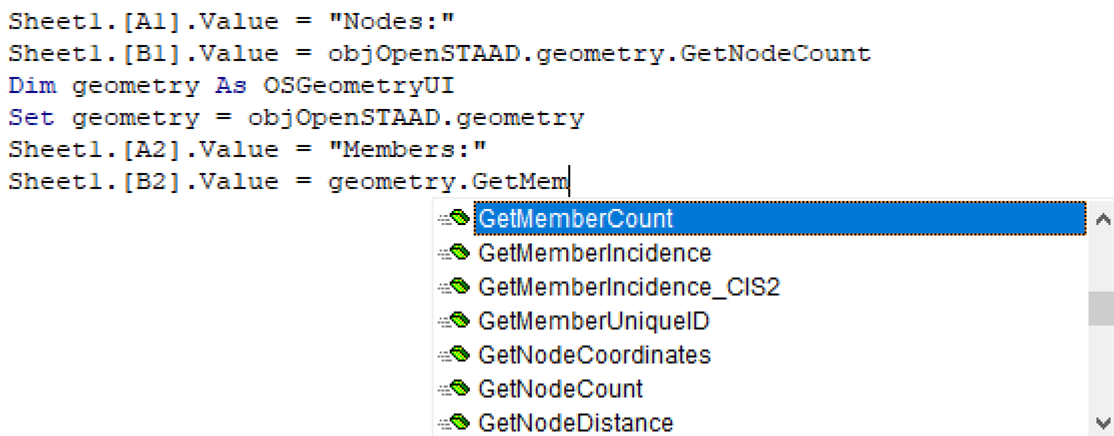
   If you OS. Connect a VBA Editor to STAAD (on page 7004), then you will notice that the editor will begin auto-completion when you start typing `OSGeometryUI`. You can press **<Tab>** to accept the highlighted result in the auto-completion pop-up list and move to the end of that entry.



4. Type `Set geometry = objOpensTAAD.geometry` and then press **<Return>**.

   This will assign the object geometry to the geometry class within the OpenSTAAD object.
5. Type `Sheet1.[A2].Value = "Members:"` and then press **<Return>**.
6. Type `Sheet1.[B2].Value = geometry.GetMemberCount` and then press **<Return>**.

   Here, you will notice that the auto-completion menu opens again once you type `geometry.` to show you all of the functions and methods within the geometry class in OpenSTAAD. This is the benefit of first setting the object to that class.



7. Run your macro.

   The spreadsheet populates with the number of nodes and members in the currently open STAAD.Pro model.

The spreadsheet will now populate with the number of nodes and members in the currently open STAAD.Pro model.

You complete code should look like this:

```
Sub OpenSTAADTutorial()
    Dim objOpenSTAAD As Object
    Dim stdFile As String
    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")
    Sheet1.[A1].Value = "Nodes:"
    Sheet1.[B1].Value = objOpenSTAAD.geometry.GetNodeCount
    Dim geometry As OSGeometryUI
    Set geometry = objOpenSTAAD.geometry
    Sheet1.[A2].Value = "Members:"
    Sheet1.[B2].Value = geometry.GetMemberCount
End Sub
```

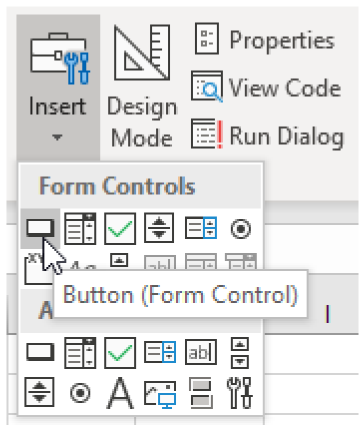
Next, you will create a simple UI element in your spreadsheet to access the macro.

OS. Run Your Macro from within the Spreadsheet

To add a button or other interactive tools in your spreadsheet to activate your macro, use the following procedure.

It wouldn't be convenient to need to open the VBA window every time you want to run the macro. In order to make a more user-friendly spreadsheet, you can add controls in the spreadsheet to access macro functionality.

1. In your spreadsheet, select the **Insert** > **Button (Form Control)** tool in the **Controls** group on the **Developer** ribbon tab.

   The mouse pointer changes to a crossahair.
2. Click-and-drag a rectangle anywhere in your spreadsheet.

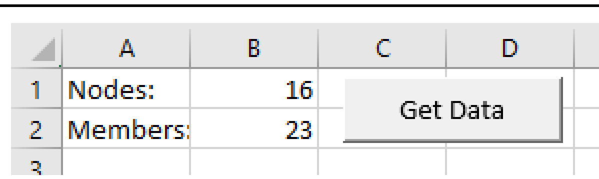
   This will form the size and shape of the button, so something slightly less than 2x2 spreadsheet cells is useful.

   The **Assign Macro** dialog opens.
3. Select the name of your subroutine from the **Macro** name list and then click **OK**.

   This assigns the button click action to run this subroutine.
4. Click the new button (labeled Button 1) in your spreadsheet.
   The macro runs and the spreadsheet is populated with the node and member counts.

   Note: If you had previously run the macro from within the VBA Editor window, then it may appear as though nothing has happened. Select and clear the cell contents and click the button again to see it in action.
5. Right-click on the button and select **Edit Text** to change the button label to something descriptive.

   Click anywhere outside of the button to exit the text editing state.



OS. Interpreting OpenSTAAD API Syntax for VBA

The functions documented for the OpenSTAAD are given in the C++ syntax. It's typically a simple process to interpret these for use in a VBA macro.

Classes and Method

In the C++ syntax used in the documentation, class syntax is listed in the format `OS`*`Class`*`UI::Method`. In VBA, this should be instead given as *`Class`*`.Method`.

In the example, OpenSTAAD was instantiated using the object "objOpenSTAAD". Classes were then called using the format `objOpenSTAAD.Class` (for example, the Geometry class was referenced using `objOpenSTAAD.Geometry`).

The exception of course is the "root" OpenSTAAD functions, which will use whatever variable you have used to instantiate OpenSTAAD (for example, `objOpenSTAAD.GetBaseUnit`). That is, the OpenSTAAD object acts as the class for these functions.

Return Values

The syntax listed indicates if a function has a return value with a leading VARIANT. This simply means that function will return a value that typically will be stored into a variable. If the function syntax is listed with a leading void, then there is no return value.

Note: A return value of a function is not to be confused with values stored in function parameters!

Variables and Parameters

The syntax listed typically indicates parameters as type VARIANT FAR, which is not necessarily useful.

Instead, it is important that you read the parameter descriptions which will indicate what type of variable to use (i.e., string, long). This will indicate what type of variable to declare in VBA before you use it.

OS. Examples

The following examples have been provided to assist in creating macros using OpenSTAAD in commonly used applications like Microsoft Excel, Autodesk AutoCAD, Microsoft Word.

OS. Microsoft Excel Macro Example

A Microsoft Office Excel spreadsheet which can be used to check the capacity of a rectangular, reinforced concrete beam.

This spreadsheet file checks the bending capacity of a rectangular concrete beam with the reinforcement already laid out. The capacity is checked against the maximum sagging moment produced from a series of load cases. The beam is analyzed in STAAD.Pro. The results are extracted and linked into Excel using OpenSTAAD and VBA.

1. In STAAD.Pro, select the **Sample Models** link on the **Open** tab of the Start page.
2. Select `US/US-8 Concrete Design for a Space Frame.STD` and the click **Open**.
3. Selet the **Run Analysis** tool in the **Analysis** group on the **Analysis and Design** ribbon tab.

   The macro relies on having analysis results.

4. Open the `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\Rectangle-Beam.xls` file in Microsoft Office Excel.

   Note: Excel will warn of macros present in the file. Simply click the **Options...** button in the Security Warning message area to open the **Microsoft Office Security Options** dialog, where you will select the **Enable this Content** option and click **OK**.

5. Select the **STAAD.Pro Output** worksheet.

   The calculations for checking the capacity are located on the sheet marked "Concrete" while the extraction of the values from STAAD occurs on the sheet marked "STAAD.Pro Output."
6. Type a member number in cell B7.

   Member 14 is entered by default.
7. Click **Results**.
   The STAAD.Pro results are read into the spreadsheet along with the beam size. Then the Concrete worksheet is selected for you.

The rest of the Concrete worksheet performs concrete beam design for the selected member based on the moment and dimensions.
Related Links
- *EX. OpenSTAAD Example Files* (on page 6253)

Excel Macro Code

To see what the OpenSTAAD macro is doing, use the developer tools to open the Visual Basic Editor. A macro named `OpenSTAADExample` has been created to retrieve the maximum sagging moment along the span of the beam as well as the width and depth of the cross-section of the beam. The maximum sagging moment for each load case is extracted, and the governing moment is the largest compression moment of the maximum moments. A second macro named `Clear` is provided which clears the results from the spreadsheet cells.

OpenSTAADExample

```
Sub OpenSTAADExample()

    Dim objOpenSTAAD As Object
    Dim stdFile As String
    Dim BaseUnit As Integer
    Dim strLength As String, strForce As String

    Dim PrimaryLCs As Integer, LoadCombs As Integer, totalLoads As Integer
    Dim EndForceArray(6) As Double
    Dim lEnd As Long

    Dim MemberNo As Long
    Dim Length As Double, Depth As Double, Width As Double
    Dim DMin As Double, dMinPos As Double, DMax As Double, dMaxPos As Double
    Dim Ax As Double, Ay As Double, Az As Double, Ix As Double, Iy As Double,
Iz As Double

    Dim i As Integer, j As Integer

    'Launch OpenSTAAD Object
```

```
    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")

    'Load your STAAD file - make sure you have successfully run the file
    objOpenSTAAD.GetSTAADFile stdFile, "TRUE"
    If stdFile = "" Then
        MsgBox "This macro can only be run with a valid STAAD file loaded.",
vbOKOnly
        Set objOpenSTAAD = Nothing
        Exit Sub
    End If

    Cells(2, 2).Value = stdFile

    MemberNo = Cells(7, 2).Value

    'Check the member exists
    Length = objOpenSTAAD.Geometry.GetBeamLength(MemberNo)

    If Length = 0 Then
        MsgBox "Please check the member number is valid.", vbOKOnly
        Set objOpenSTAAD = Nothing
        Exit Sub
    End If

    'Find out how many primary load cases and load combinations you have:-
    PrimaryLCs = objOpenSTAAD.Load.GetPrimaryLoadCaseCount()
    Cells(4, 2).Value = PrimaryLCs

    LoadCombs = objOpenSTAAD.Load.GetLoadCombinationCaseCount
    Cells(5, 2).Value = LoadCombs

    totalLoads = PrimaryLCs + LoadCombs

    If totalLoads > 10 Then
        MsgBox ("This macro is designed for up to 10 load cases only")
        Set objOpenSTAAD = Nothing
        Exit Sub
    End If

    'What are the units?
    BaseUnit = objOpenSTAAD.GetBaseUnit
    If BaseUnit = 1 Then ' English
        strLength = "Feet"
        strForce = "KIP"
    Else                 ' Metric
        strLength = "Metre"
        strForce = "KN"
        MsgBox ("This macro is designed for up English units. Change
STAAD.Pro>Configuration>Base Unit")
        Set objOpenSTAAD = Nothing
        Exit Sub
    End If

    Cells(4, 6) = strLength
    Cells(5, 6) = strForce

    'Iterate through your load sets to find the results for each load case
    Dim lstLoadPrimaryNums() As Long
```

```
    ReDim lstLoadPrimaryNums(PrimaryLCs)
    Dim lstLoadCombinationNums() As Long
    ReDim lstLoadCombinationNums(LoadCombs)

    Dim lstLoadNum() As Long
    Dim lstLoadName() As String
    ReDim lstLoadNum(totalLoads)
    ReDim lstLoadName(totalLoads)

    objOpenSTAAD.Load.GetPrimaryLoadCaseNumbers lstLoadPrimaryNums
    objOpenSTAAD.Load.GetLoadCombinationCaseNumbers lstLoadCombinationNums

    For i = 0 To PrimaryLCs - 1
        lstLoadNum(i) = lstLoadPrimaryNums(i)
        lstLoadName(i) =
objOpenSTAAD.Load.GetLoadCaseTitle(lstLoadPrimaryNums(i))
    Next i

    For i = 0 To LoadCombs - 1
        lstLoadNum(i + PrimaryLCs) = lstLoadCombinationNums(i)
        lstLoadName(i + PrimaryLCs) =
objOpenSTAAD.Load.GetLoadCaseTitle(lstLoadCombinationNums(i))
    Next i

    'LoadPrevious = 0
    For i = 0 To totalLoads - 1
        Cells(10, i + 2).Value = lstLoadNum(i)
        Cells(11, i + 2).Value = lstLoadName(i)

        'Get forces at start of member
        lEnd = 0
        objOpenSTAAD.Output.GetMemberEndForces MemberNo, lEnd, lstLoadNum(i),
EndForceArray
        For j = 0 To 5
            'Print the result values in specified cells
            Cells(j + 13, i + 2).Value = EndForceArray(j)
        Next

        'Get forces at end of member
        lEnd = 1
        objOpenSTAAD.Output.GetMemberEndForces MemberNo, lEnd, lstLoadNum(i),
EndForceArray
        For j = 0 To 5
            'Print the result values in specified cells
            Cells(j + 20, i + 2).Value = EndForceArray(j)
        Next
         'objOpenSTAAD.GetMinBendingMoment MemberNo, "MZ", LoadNext,
MaxSaggingMoment
        objOpenSTAAD.Output.GetMinMaxBendingMoment MemberNo, "MZ",
lstLoadNum(i), DMax, dMaxPos, DMin, dMinPos
        Cells(27, i + 2).Value = DMin
    Next

    'Write the appropriate dimensions (for the beam) in the correct cells
    Sheets("Concrete").Select

    'Get Depth and Width of Concrete Beam
    objOpenSTAAD.Property.GetBeamProperty MemberNo, Width, Depth, Ax, Ay, Az,
```

```
Ix, Iy, Iz
    'Depth
    Cells(31, 7).Value = Depth
    'Width
    Cells(32, 7).Value = Width

    Set objOpenSTAAD = Nothing

End Sub
```

Clear

```
Sub Clear()
Dim i As Integer
Dim j As Integer

Cells(2, 2) = ""
Cells(4, 2) = 0
Cells(5, 2) = 0

Cells(4, 6) = ""
Cells(5, 6) = ""

For i = 1 To 10
    For j = 1 To 18
        Cells(j + 9, i + 1) = " "
    Next j
Next i

End Sub
```

OS. Microsoft Word Macro Example

A Microsoft Office Word document is included which contains a partial report for analysis results which can be used as a basis for a customized report document.

The document file includes a macro named `Pro` which checks for the number of supported nodes as well as the number of load cases and load combinations and reports these back to the document. The document can then report support reactions for the selected node number and load case number.

1. Open an input file within STAAD.Pro.
2. Open the file `C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\STAADandWord.doc`.

   Note: Word will warn of macros present in the file. Simply click the **Options...** button in the Security Warning message area to open the **Microsoft Office Security Options** dialog, where you will select the **Enable this Content** option and click **OK**.

3. Click the **Update Nodes and Load Cases** button.

   The **Node Number** and **Load Case Number** selectors update with the data from the selected file.

4. Select the **Node Number** and **Load Case Number** numbers for which results will be retrieved.
5. Click **Get Results**.
   The support reactions and units update in the table below.

Related Links

- *EX. OpenSTAAD Example Files* (on page 6253)

Word Macro Code

To see what the OpenSTAAD macro is doing, use the developer tools to open the Visual Basic Editor.

GetSTAAD_Click

```
Option Explicit

Public objOpenSTAAD As Object
Public Fraction As Double
Public LC As Integer
Public bBadSTAADFile As Boolean



Private Sub GetSTAAD_Click()

'
' STAAD.Pro Macro
' Import STAAD.Pro Data Dynamically into an MS Word document
' using the OpenSTAAD Application object

    Dim stdFile As String
    Dim DOF As Integer
    Dim acell As Cell


     'Launch OpenSTAAD Object
    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")

    'Load your STAAD file - make sure you have successfully run the file
    objOpenSTAAD.GetSTAADFile stdFile, "TRUE"
    STAADFileName.Text = "(None)"
    If stdFile = "" Then
        MsgBox "This macro can only be run with a valid STAAD file loaded.",
vbOKOnly
        Set objOpenSTAAD = Nothing
        Exit Sub
    End If

    STAADFileName.Text = stdFile

    FillBoxesWithData stdFile

    DOF = 0
    For Each acell In ActiveDocument.Tables(2).Columns(2).Cells
        If (acell.RowIndex <> 1) Then
            acell.Range.Text = "0.0"
            DOF = DOF + 1
        End If
```

```
    Next acell



    Set objOpenSTAAD = Nothing

End Sub
```

GetResults1_Click

```
Private Sub GetResults1_Click()

    Set objOpenSTAAD = GetObject(, "StaadPro.OpenSTAAD")

    Dim Reactions(6) As Double

    ' The Integer has been changed to a Long in 2004 to hold higher node
numbers
    Dim Node As Long
    Dim LC As Long
    Dim DOF As Integer
    Dim acell As Cell

    If (bBadSTAADFile = True) Then
        MsgBox "Please provide a valid STAAD file, node number and load case
number before clicking this button."
        Exit Sub
    End If

    Node = NodeNumber.Text
    LC = LCNumber.Text

    objOpenSTAAD.Output.GetSupportReactions Node, LC, Reactions

    DOF = 0
    For Each acell In ActiveDocument.Tables(2).Columns(2).Cells
        If (acell.RowIndex <> 1) Then
            acell.Range.Text = Reactions(DOF)
            DOF = DOF + 1
        End If
    Next acell


End Sub
```

Additional Subroutines

```
Private Function FillBoxesWithData(strSTAADFile As String)

    Dim varCounter1 As Integer
    Dim PrimaryLCs As Integer
    Dim LoadCombs As Integer
    Dim TotalLoads As Integer
    ' The Integer has been changed to a Long in 2004 to hold higher node
numbers
    Dim SupportedNodesCount As Long
```

```
    ' The Integer has been changed to a Long in 2004 to hold higher node
numbers
    Dim SupportedNodeNos() As Long


On Error GoTo ErrorHandler
'Set objOpenSTAAD = CreateObject("OpenSTAAD.Output.1")

    SupportedNodesCount = objOpenSTAAD.Support.GetSupportCount()
    ReDim SupportedNodeNos(SupportedNodesCount)

    Dim nReturn As Long

    nReturn = objOpenSTAAD.Support.GetSupportNodes(SupportedNodeNos)

    'Fill nodes list box
    NodeNumber.Clear
    For varCounter1 = 0 To SupportedNodesCount - 1
        NodeNumber.AddItem SupportedNodeNos(varCounter1)
    Next varCounter1

    NodeNumber.Text = SupportedNodeNos(0)

    'Fill load list box
    'Find out how many primary load cases and load combinations you have
    PrimaryLCs = objOpenSTAAD.Load.GetPrimaryLoadCaseCount()
    LoadCombs = objOpenSTAAD.Load.GetLoadCombinationCaseCount()

    TotalLoads = PrimaryLCs + LoadCombs

    LCNumber.Clear
    For varCounter1 = 1 To TotalLoads
         LCNumber.AddItem varCounter1
    Next
    LCNumber.Text = 1

    bBadSTAADFile = False

Exit Function

ErrorHandler:
    bBadSTAADFile = True
    MsgBox "STAAD File does not exist or proper analysis results cannot be
found. Please check the STAAD file."

End Function

Private Sub NodeNumber_Change()

End Sub

Private Sub STAADFileName_Change()

End Sub

Private Sub UpdateSTDFile_Click()
 bBadSTAADFile = True
```

```
 FillBoxesWithData
End Sub
```

OS. Additional References

Documentation for VBA is beyond the scope of this manual and Bentley does not provide direct support on how to write VBA macros. There are, however, several useful and free sites on the Web to assist a beginner on writing macros in STAAD, Excel, AutoCAD, or any other VBA compliant software.

Note: You can get help on Visual Basic and using VBA by selecting the Help menu in the VBA Editor.

It should be noted that the VBA language is the same from software to software. However, the functions, objects, and core libraries will obviously vary.

The sites recommended by Bentley are as follows:

- https://msdn.microsoft.com/en-us/vba/ (refer to the "Office VBA Object library reference" to get started)
- https://stackoverflow.com
- https://www.oreilly.com/
- http://www.wrox.com

Also, it is worth noting that many programs such as MicroStation and Microsoft Office Excel have a useful *Record Macro* feature. You can run the recorder and then select any commands from the program's ribbon tabs. The corresponding VBA syntax will automatically be generated.

For additional information on using macros and VBA in Excel or other Microsoft Office programs, Microsoft's MSDN is a recommended starting point:

- http://msdn.microsoft.com/en-us/library/ee814735.aspx#Y2242

OS. Writing OpenSTAAD in the STAAD.Pro Script Editor

A visual basic editor is embedded within the STAAD.Pro environment. Here, you can create macros that are run from within the STAAD.Pro application to interact with the current STAAD model.

There are several examples of STAAD user tools that ship with STAAD.Pro, along with the tutorial at the end of this section.

Note: The version of VBA which the editor supports is not 100% Microsoft VBA compatible. There are a few functions which can be supported in the Microsoft version of VBA which are not supported in the VBA editor/ compiler which comes with STAAD.Pro. Every attempt is being made to support these functions.

OS. Using Macros in STAAD.Pro

This section describes how to use the STAAD.Pro Script Editor to create a new macro or load existing macros in STAAD.Pro. It also describes how to run the macros you have created in the STAAD.Pro Script Editor by adding a simple menu item.

OS. To start a new macro project

To create a new macro project and open it in the STAAD.Pro Script Editor, use the following procedure.

1. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

   The **Macro** dialog opens.
2. Click **Create New**.
   The **New Macro File Name** dialog opens.
3. Type a **File name** and optional **Description of Macro**.

   For this tutorial, type `Frame` in the **File name** field and then type `Create a 2D frame with supports` in the **Description of Macro** field.
4. Click **New**.
   The STAAD.Pro Script Editor window opens. An empty script is loaded with for your file.

OS. To import an existing macro

1. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

   The **Macro** dialog opens.
2. Click **Import**.
   The **Add an Existing Macro** dialog opens. The controls in this dialog are analogous to those in the **Select New Macro File Name** dialog, though some are inactive.
3. Navigate to and select the existing macro file.
4. Click **Open**.

The macro name is added to the list of names in the Macro dialog.
Related Links
- *EX. OpenSTAAD Example Files* (on page 6253)

OS. To run a linked macro

1. On the **Utilities** ribbon tab, select the **User Tools** tool in the **User Tools** group.

   A drop-down menu displays all of the linked macros.
2. Select the macro you want to run from this list.

OS. To connect the STAAD.Pro Script Editor to STAAD Object Library

The STAAD object library, which contains OpenSTAAD, can be referenced in the STAAD.Pro Script Editor such that the editor will auto-complete class and function names.

This procedure is optional, but highly recommended in order to have your editing environment be able to assist you in writing code.

1. Start the STAAD.Pro Script Editor as an administrator.

   Performing the following steps as an administrator allows your to make a change to the template. This prevents you from having to re-perform this task every time you edit STAAD macros.

   a. In Windows Explorer, locate the file `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Bentley.Staad.ScriptEditor.exe`.

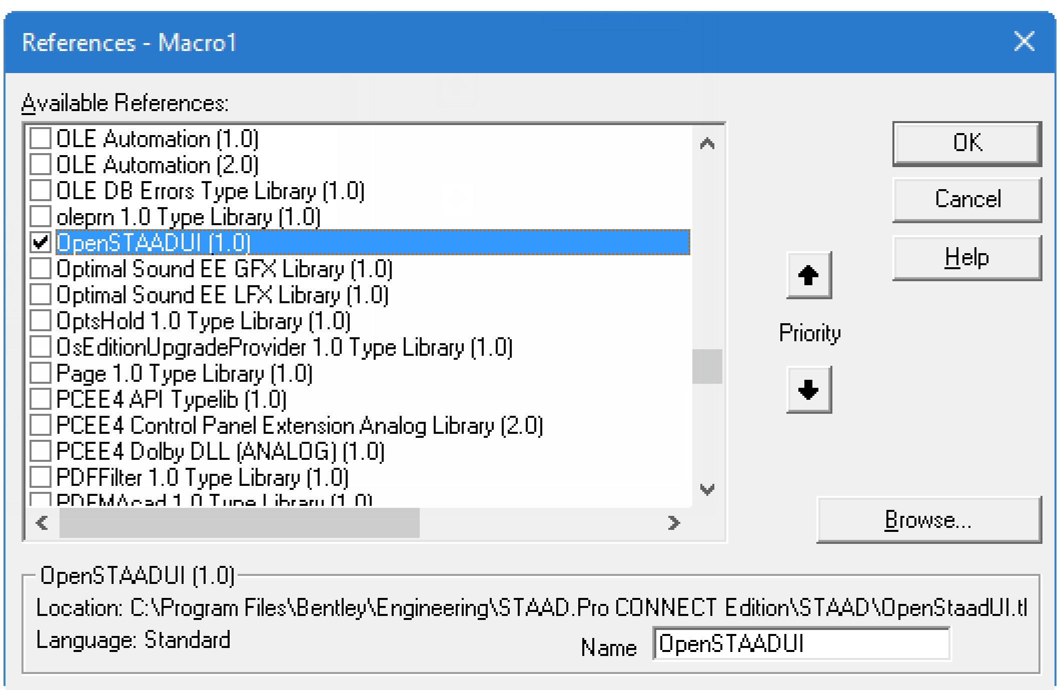
   b. Right-click on the file and then select **Run as administrator** from the pop-up menu.

   You are asked if you want to allow the application to make changes to your computer. Confirm you will allow this.

   The editor opens with a new macro template open.
2. Select the **Add Reference** tool in the **Design** group on the **Home** ribbon tab.

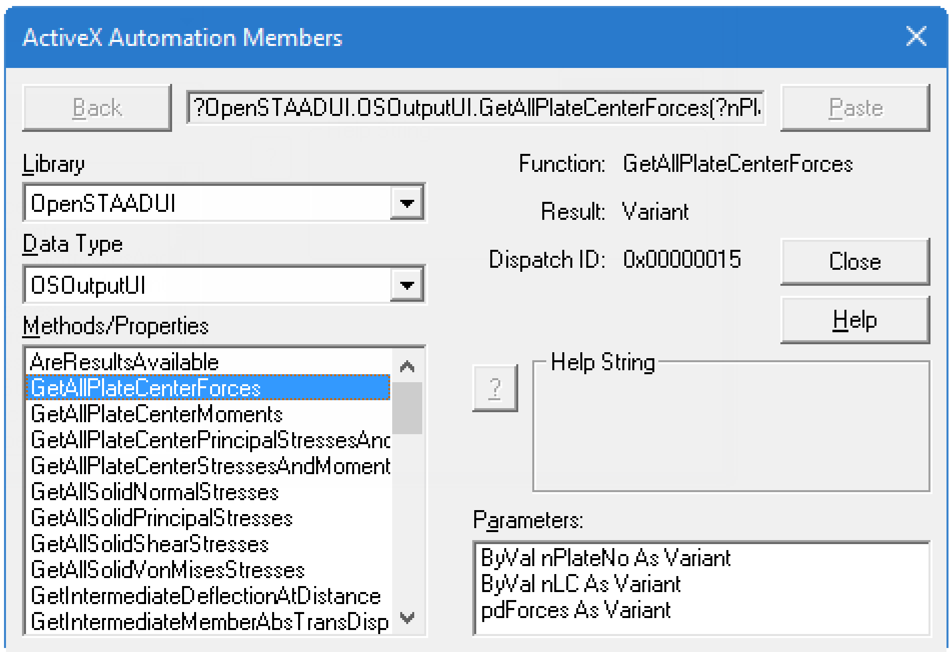
   The **References** dialog opens.

3. Click **Browse**.
   The References dialog opens.

4. Navigate to and select `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\StaadPro.dll` and then click **Open**.
5. Scroll through the list of **Available References** to locate **OpenSTAADUI (1.0)** and check this option.
6. Click **OK**.

Now, when the **Browse Object** tool () is selected, you will see OpenSTAAD functions included in the Library list.



Related Links

- *OS. User Type Not Defined* (on page 7088)

OS. To connect the STAAD.Pro Script Editor to STAAD Object Library

The STAAD object library, which contains OpenSTAAD, can be referenced in the STAAD.Pro Script Editor such that the editor will auto-complete class and function names.

This procedure is optional, but highly recommended in order to have your editing environment be able to assist you in writing code.

1. Start the STAAD.Pro Script Editor as an administrator.

   Performing the following steps as an administrator allows your to make a change to the template. This prevents you from having to re-perform this task every time you edit STAAD macros.

   a. In Windows Explorer, locate the file `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Bentley.Staad.ScriptEditor.exe`.

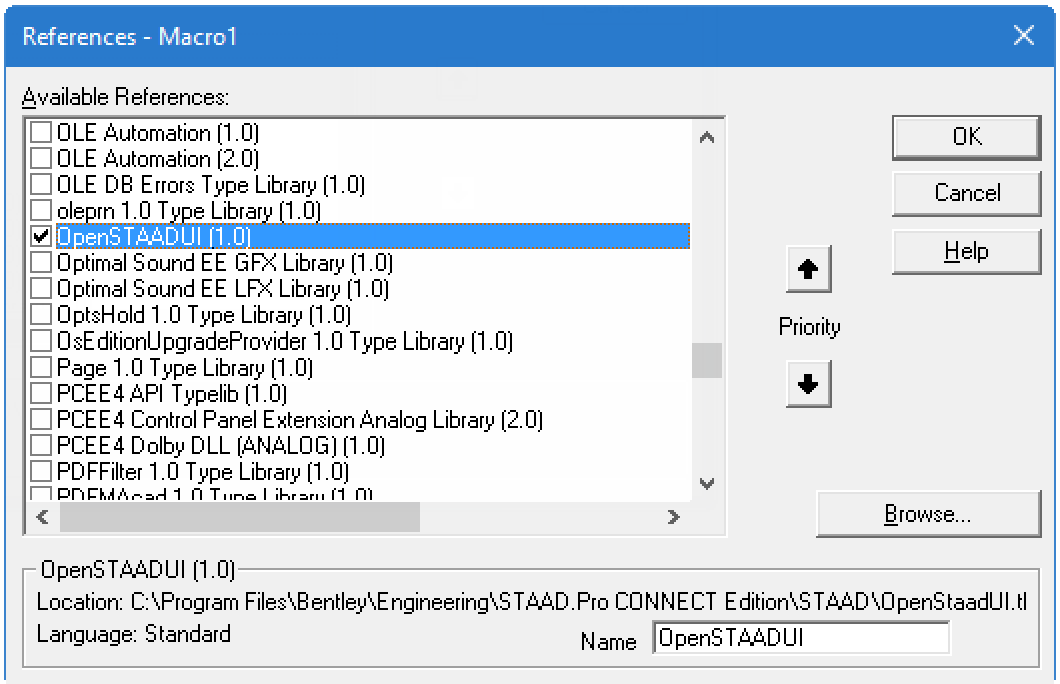
   b. Right-click on the file and then select **Run as administrator** from the pop-up menu.

   You are asked if you want to allow the application to make changes to your computer. Confirm you will allow this.

The editor opens with a new macro template open.

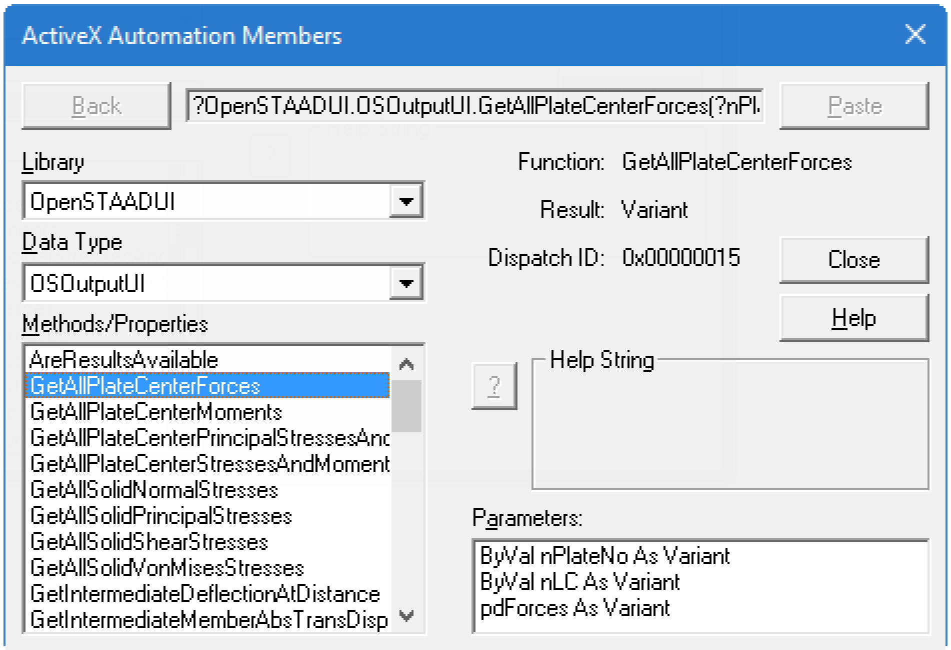
2. Select the **Add Reference** tool in the **Design** group on the **Home** ribbon tab.

The **References** dialog opens.



3. Click **Browse**.
   The References dialog opens.
4. Navigate to and select `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\StaadPro.dll` and then click **Open**.
5. Scroll through the list of **Available References** to locate **OpenSTAADUI (1.0)** and check this option.
6. Click **OK**.

Now, when the **Browse Object** tool () is selected, you will see OpenSTAAD functions included in the Library list.



Related Links

- *OS. User Type Not Defined* (on page 7088)

OS. Simple STAAD.Pro Macro

This example demonstrates a small macro which can be used within STAAD.Pro.

Often, when learning to program, you begin with a program which simply outlines the basic structure of an application, module, or function; typically resulting with a screen message displaying the phrase “Hello World”. This example expands upon this to include the foundation from a practical application as well.

Note: Additional examples in this section demonstrate how to poll STAAD data from external programs.

1. Open STAAD.Pro.
2. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

   The **Macro** dialog opens.
3. Click **Create New**.
   The **New Macro File Name** dialog opens.
4. Type a title of `CreateNewView.vbs` with the following description: `Creates a new view from the selected beams.`

   Note: You can save the macro file anywhere, so long as the directory has write permission for your user account.

   The OS. STAAD.Pro Script Editor window (on page 7043) opens with an subroutine title Main.

5. Just after the description, type the following just after the description comment line:

   ```
   Dim objOpenSTAAD As Object
   Dim SelBeamsNo As Long
   Dim SelBeams() As Long
   ```

   This is used to provide some declarations of the objects and variables used in this program.
6. Type the following lines to instantiate the OpenSTAAD object:

   ```
   'Launch the OpenSTAAD Object
   Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")
   ```

   Note: The first line beginning with the apostrophe (') is a comment. It isn't necessary, but it is good practice to add remarks such as this to make your code clear to others (as well as to yourself when you revisit the code at a later time).
7. Type the following lines to set up a logical check for if any beams are selected:

   ```
   'Get no. of selected beams
   SelBeamsNo = objOpenSTAAD.Geometry.GetNoOfSelectedBeams
   If (SelBeamsNo > 0) Then
   ```

   Here, the `GetNoOfSelectedBeams` Geometry function in OpenSTAAD is being used to aid our test. The test is a if… then… else… statement, which continues in the following steps.
8. Type following lines to instruct the program what to do if our statement is true (i.e., there is at least one beam selected).

   That is to create a new view from the active selection using the `CreateNewViewForSelection` View function in OpenSTAAD.

   ```
   ReDim SelBeams(SelBeamsNo) As Long
   'Create a new view
   objOpenSTAAD.View.CreateNewViewForSelections
   ```
9. Type the following lines.

   Since this macro might be run with no beams selected, a message can be provided to the user for some feedback in this instance with the following line:

   ```
     Else
     MsgBox "No beams are currently selected.", vbOkOnly
   End If
   ```

   Tip: You could add the message `Hello World`, if you prefer to stick with a more traditional introductory example of programming.
10. Type the following statement to close the instance of the OpenSTAAD object:

    ```
    Set objOpenSTAAD = Nothing
    ```

This is all it requires to create a macro. Obviously, this particular example really only duplicates the functionality of selecting the **New View** tool in STAAD.Pro. However, it easy to combine other OpenSTAAD functions to automate a series of commonly used features in order to create your own time saving tools.

In this example, for the sake of brevity, the only model entities checked for selection are beams (that is, a new view is only created if beam elements are selected). You could easily expand this to Nodes, Plates, Solids, etc.

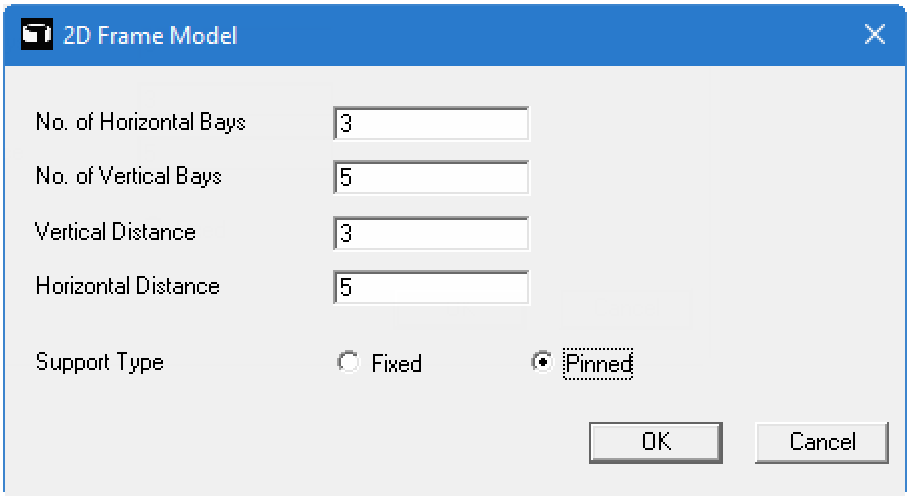
Example

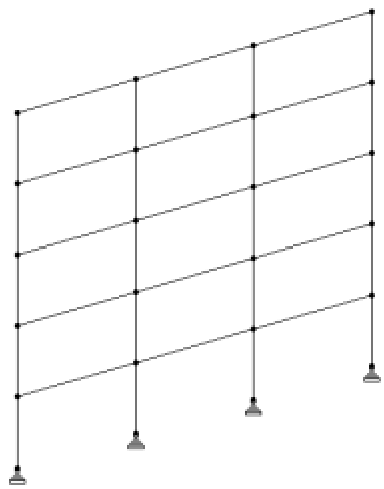
The full code for this macro is as follows:

```
Sub Main()
'DESCRIPTION:Creates a new view from the selected beams.
  Dim objOpenSTAAD As Object
  Dim SelBeamsNo As Long
  Dim SelBeams() As Long
'Launch OpenSTAAD Object
  Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")
'Get no. of selected beams
  SelBeamsNo = objOpenSTAAD.Geometry.GetNoOfSelectedBeams
  If (SelBeamsNo > 0) Then
    ReDim SelBeams(SelBeamsNo) As Long
'Create a new view
    objOpenSTAAD.View.CreateNewViewForSelections 'SelBeams
  Else
    MsgBox "No beams are currently selected.", vbOkOnly
  End If
  Set objOpenSTAAD = Nothing
End Sub
```

OS. Macro Tutorial

This tutorial will take you step-by-step through the process of creating a practical macro used to generate a parametric 2D frame with supports.





The assumptions behind this parametric frame are:

- 2D frame lies in the XY plane at Z = 0 (with first support node at the origin)
- All bays are of equal width and height
- The user specifies the units of length before running the macro
- All supports have the same type and all nodes at Y = 0 are supports

Most of these assumptions could be either validated for altered with some changes or additions to the macro presented in this tutorial.

OS. To start a new macro project

To create a new macro project for this tutorial and to then open it in the STAAD.Pro Script Editor, use the following procedure.

The macro used in this tutorial is intended to run in an empty file. Therefore, you should start with an new STAAD input file in the Analytical Modeling worfklow.

1. On the **Utilities** ribbon tab, select the **Macro** tool in the **Developer** group.

The **Macro** dialog opens.

2. Click **Create New**.
   The **New Macro File Name** dialog opens.
3. Type a **File name** and optional **Description of Macro**.

   For this tutorial, type `Frame` in the **File name** field and then type `Create a 2D frame with supports` in the **Description of Macro** field.
4. Click **New**.
   The STAAD.Pro Script Editor window opens. An empty script is loaded with for your file.
5. Select the line `'TODO: Add your code here` (i.e., line 4) and press **<Delete>**.

OS. Creating the User Dialog

The form is the dialog which a user will use to specify the frame parameters. This form will used text entry fields for the frame size and then a pair of options to allow the user to select the support type.

OS. To create the dialog form

Your cursor should be located where the dialog code will be inserted. If you deleted the comment line in the previous procedure, then your cursor is in the correct location (i.e., line 4).

1. Select the **User Dialog** tool in the **Design** group.

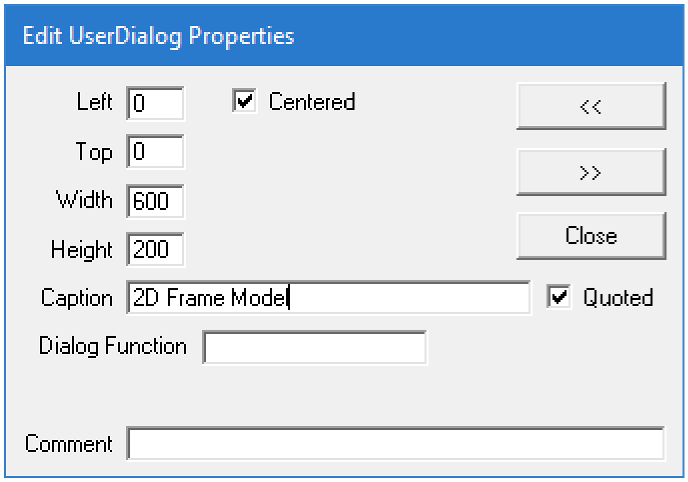
   The **UserDialog Editor** opens.
2. Either:

   Click the **Edit Item Properties** tool 

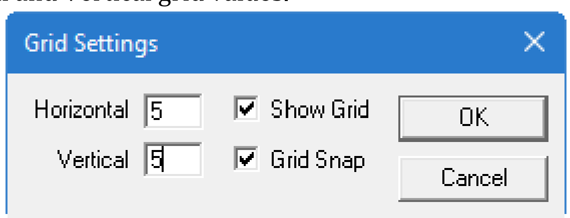
   or

   Right-click anywhere in the form layout

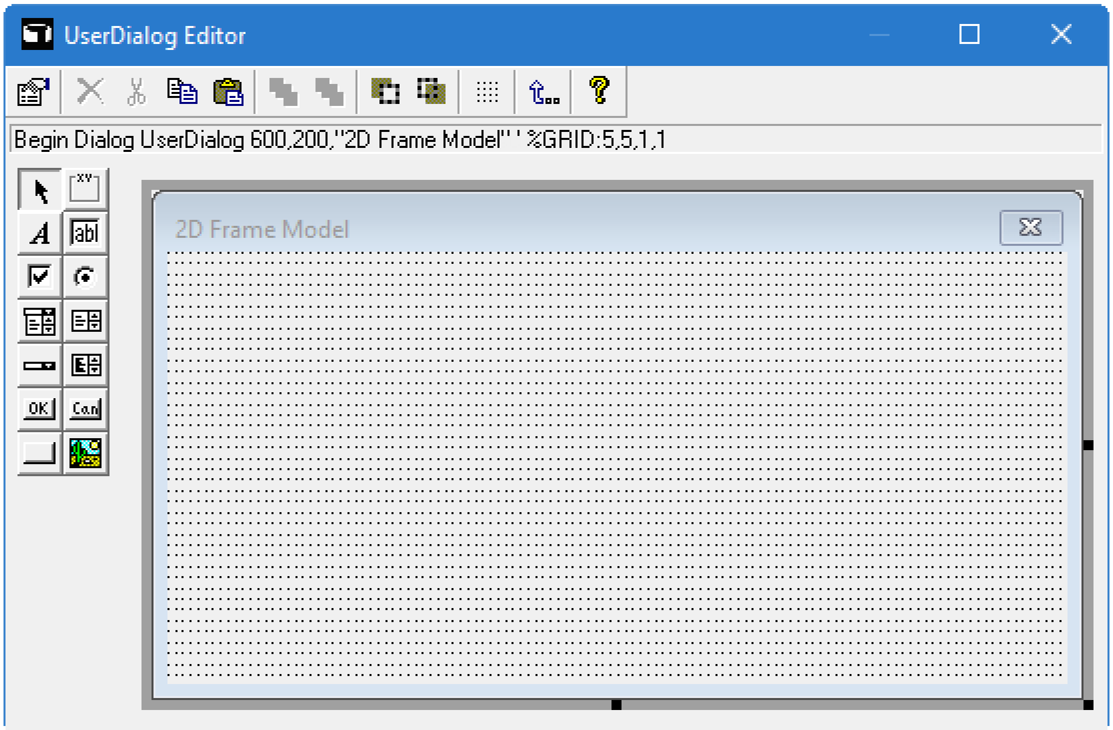
   The **Edit User Dialog Properties** dialog opens.
3. Type `600` in the **Width** field and then `200` in the **Height** field.
   These values are somewhat arbitrary, but allow for a sufficient amount of space to include the dialog controls.
4. Type `2D Frame Model` in the **Caption** field.



5. Click **Close**.
6. Select the **Set Grid** tool.
   The **Grid Settings** dialog opens.
7. Type 5 for both the Horizontal and Vertical grid values.

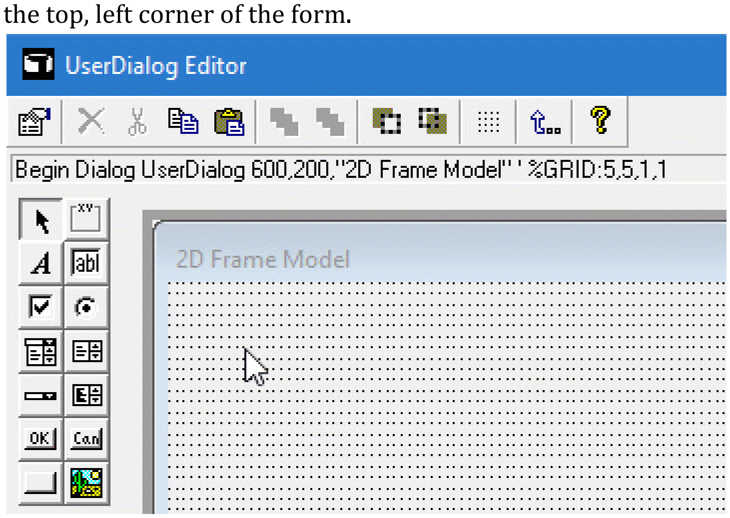


8. Click **OK**.

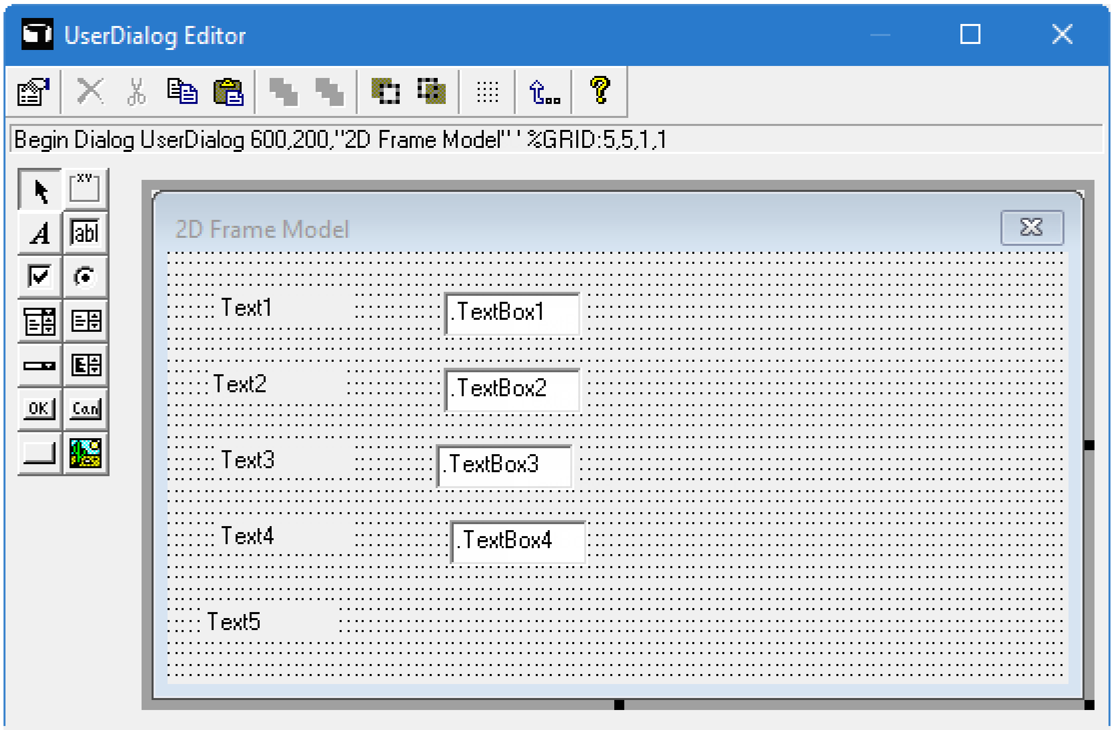


OS. To create the text fields and labels

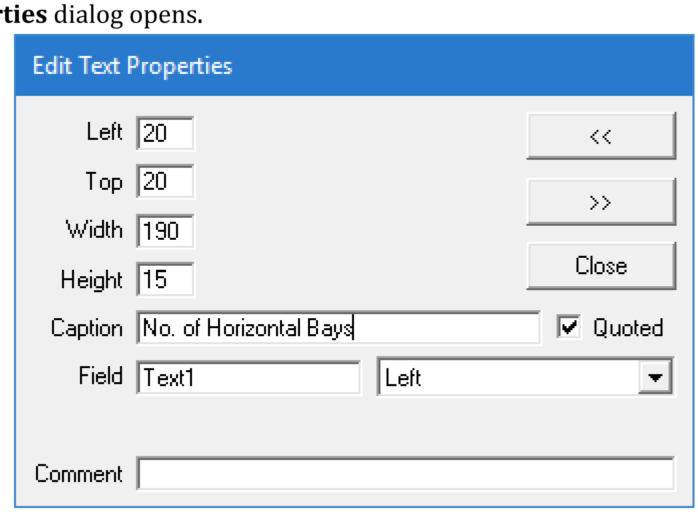
1. Select the **Add Text** tool. 
2. Click somewhere near the top, left corner of the form.



3. Repeat steps 2 and 3 to add four additional text labels along the left side of the dialog.
   This does not have to be precise. The labels and fields will be aligned by coordinates in a later step.
4. Select the **Add TextBox** tool. 
5. Click a point to the right of the first label to add a data field in the form.
6. Repeat steps 4 and 5 to add three additional text boxes.



7. Edit the location and caption for the first label:
   - a. Right-click on the first label.
     The **Edit Text Properties** dialog opens.



   - b. Type the following values to define the label layout:

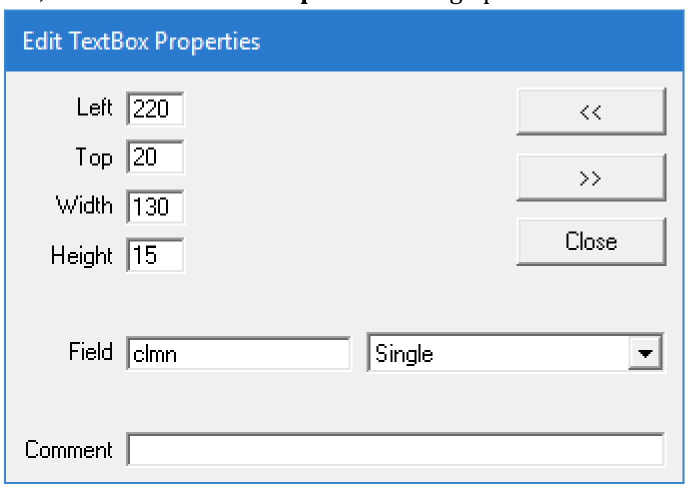
     Left = 20
     Top = 20
     Width = 190
     Height = 15 (default if you placed with a click)
   - c. Type `No. of Horizontal Bays:` in the **Caption** field.
     Leave the remaining fields as their defaults.
   - d. Click **>>**.

The properties for the next control added are displayed. This allows you to quickly make changes to the controls without the need to close and reopen this dialog.

8. Repeat step 7 to update the layout and caption for the remaining text labels as indicated below.

Label Name	Rectangle (Left, Top, Width, Height)	Label Text
Text2	20,45,190,15	No. of Vertical Bays:
Text3	20,70,190,15	Vertical Distance:
Text4	20,95,190,15	Horizontal Distance:
Text5	20,130,190,15	Support Type:

When you click **>>** on Text5, the **Edit TextBox Properties** dialog opens for the first text box control.



Tip: If a different set of properties are displayed, then you can continue clicking >> to cycle through the controls until the TextBox properties are displayed or close and right-click on the text box to open them.

9. Edit the location and name for the text box:
   a. Type the following values to define the text box:

      Left = 220
      Top = 20
      Width = 130
      Height = 15

   b. Type `clmn` in the **Field** field.

      This is the object name which the macro will use for this text box.

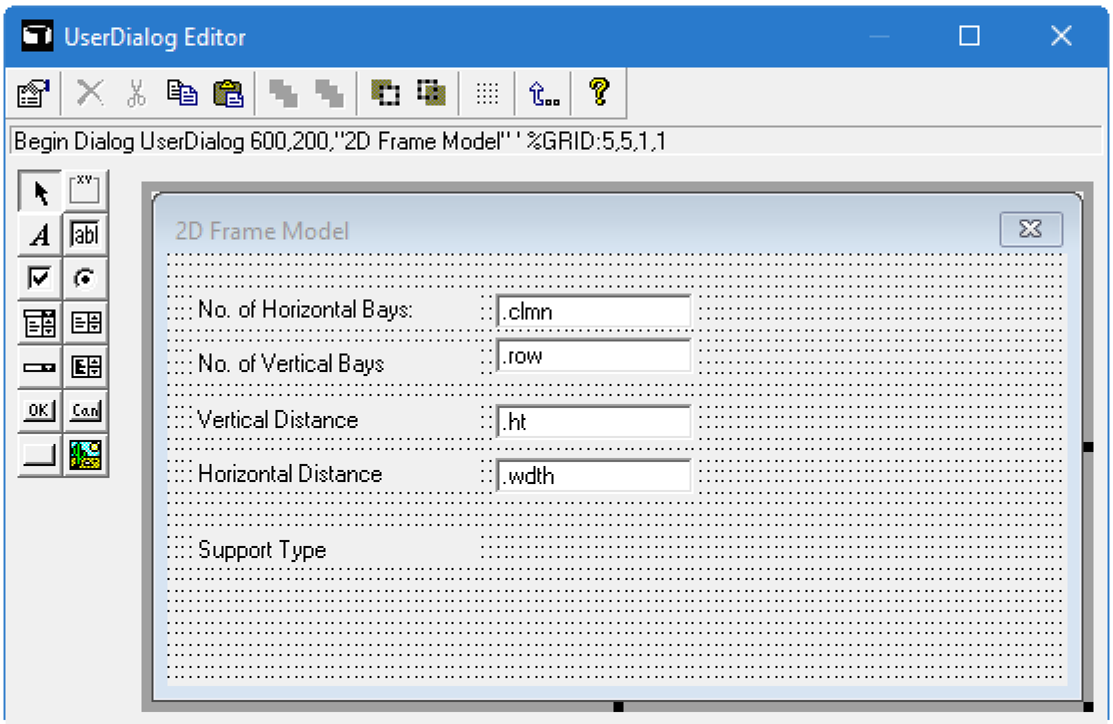
   c. Click **>>**.
      The properties for the next control added are displayed.

10. Repeat step 9 to update the layout and field name for the remaining text boxes as indicated below.

Text Box Name	Rectangle (Left, Top, Width, Height)	Field Name
TextBox2	220,40,130,15	row

Text Box Name	Rectangle (Left, Top, Width, Height)	Field Name
TextBox3	220,70,130,15	ht
TextBox4	220,95,130,15	wdth

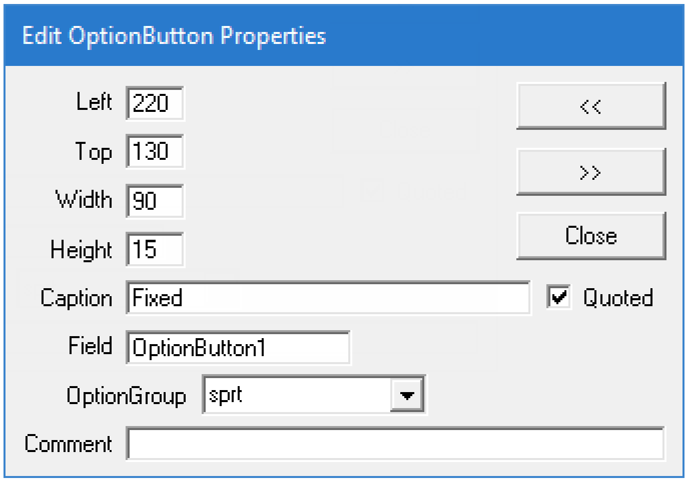
11. Click **Close**.



OS. To create support options

Since the macro in this tutorial limits the support options to only fixed or pinned, using a pair of radio buttons to allow the user to select the support type is a good design choice. However, in a situation where three or more options were available, a drop-down list control might be a better choice.

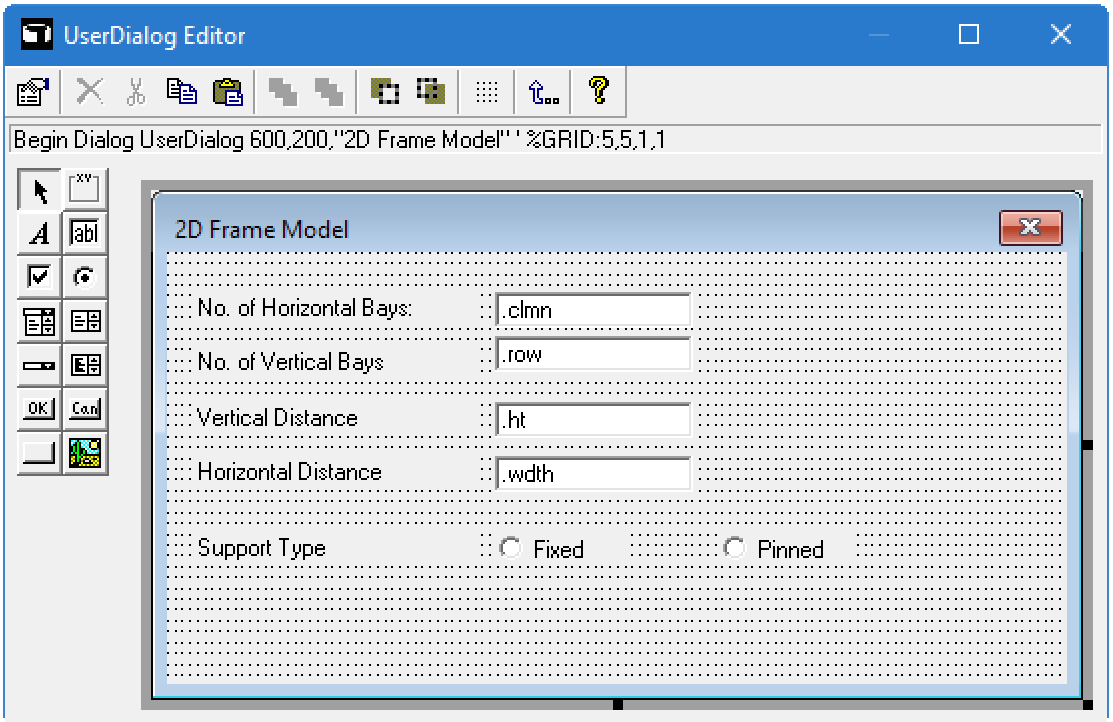
1. Select the **Add OptionButton** tool. 
2. Click a point to the right of the Support Type label (i.e., text5).
   The option control is placed.
3. Repeat steps 1 and 2 to add a second option to the right of the first.
4. Right-click on the first option control.
   The **Edit OptionButton Properties** dialog opens.



5. Type the following values to define the location:

   Left = 220
   Top = 130
   Width = 90
   Height = 15
6. Type `Fixed` in the **Caption** field.
7. Type `sprt` in the **OptionGroup** combo box.
8. Click **>>**.
   The properties for the second option control are displayed.
9. Repeat steps 5 through 7 except type `370` in the **Left** field and `Pinned` in the **Caption** field.

   Tip: Since the option group was previously defined, you can now selected from the drop-down list in the combo box rather than re-type its name.
10. Click **Close**.
    The **Edit OptionButton Properties** dialog closes.



OS. To add the dialog buttons

The last step in defining the dialog is to add the OK and Cancel buttons.

1. Click the **Add OK Button** tool.
2. Click a point near the bottom, right corner of the form.
   The OK button is placed.
3. Click the **Add Cancel Button** tool.
4. Click a point to the right of the OK button.

   Note: This is the recommended button order according to Microsoft's design guidelines.

   The Cancel button is placed.
5. (Optional) Either:

   right-click on either button to precisely edit the location

   or

   drag the buttons to align them to the grid
6. Select the **Save and Exit** tool.
   The **UserDialog Editor** closes.

The dialog definition code is added to your macro:

```
    Begin Dialog UserDialog 600,200,"2D Frame Model" ' %GRID:5,5,1,1
        Text 20,20,190,15,"No. of Horizontal Bays:",.Text1
        Text 20,45,190,15,"No. of Vertical Bays",.Text2
```

```
        Text 20,70,190,15,"Vertical Distance",.Text3
        Text 20,95,190,15,"Horizontal Distance",.Text4
        Text 20,130,190,15,"Support Type",.Text5
        TextBox 220,20,130,15,.clmn
        TextBox 220,40,130,15,.row
        TextBox 220,70,130,15,.ht
        TextBox 220,95,130,15,.wdth
        OptionGroup .sprt
            OptionButton 220,130,90,15,"Fixed",.OptionButton1
            OptionButton 370,130,90,15,"Pinned",.OptionButton2
        OKButton 360,165,90,20
        CancelButton 490,165,90,20
    End Dialog
    Dim dlg As UserDialog
    Dialog dlg
```

Tip: Now is a good time to save the work done to this point.

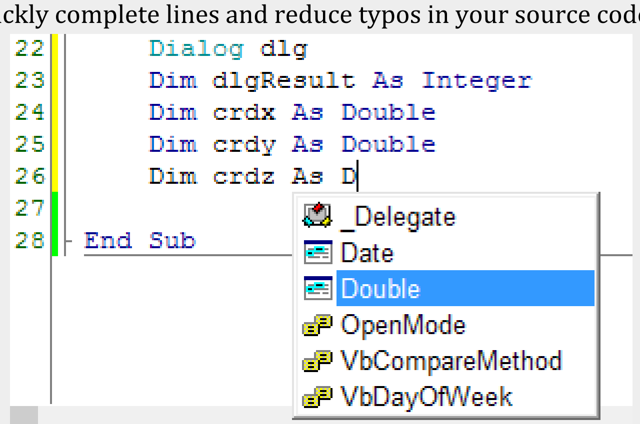
OS. To dimension the variables and add initial values in the dialog

You must dimension all the values you will use in the program. Further, it's a good idea to populate the dialog with initial values for the user.

1. Select the automatically generated line of code `Dialog dlg` (i.e., line 23) and delete the line contents.
2. With your cursor after the line of code `Dim dlg as UserDialog`, type the following to dimension the variables in use:

```
    Dim dlgResult As Integer
    Dim crdx As Double
    Dim crdy As Double
    Dim crdz As Double
    Dim n1 As Long
    Dim n2 As Long
    Dim i1 As Long
    Dim s1 As Long
```

Note: As you being to type, the IntelliSense will provide options for the variable types. You can make use of the IntelliSense to help quickly complete lines and reduce typos in your source code.



3. Type a single quote mark followed by `Initialization`.

The single quote mark is used to denote a comment. It is best practice to add clear comments to document your source code.

```
    'Initialization
```

4. Type the following values to populate the dialog input fields:

```
    dlg.clmn = "3"
    dlg.row = "5"
    dlg.ht = "3"
    dlg.wdth = "5"
```

5. Type the following commands to open the dialog upon starting the macro.

   You'll also add a comment and a debugging command which is helpful for resolving any issues with your macro.

```
    'Popup the dialog
    dlgResult = Dialog(dlg)
    Debug.Clear
```

6. Type the following command to capture the result of the user action in the dialog:

```
    If dlgResult = -1 Then 'OK button pressed
        Debug.Print "OK button pressed"
    ElseIf dlgResult = 0 Then 'Cancel button pressed
        Debug.Print "Cancel button pressed"
    End If
```

Note: The single quote mark after the world `Then` also indicates a comment. Comments like this can follow the programming instruction on the same line.

The `EndIf` statement is automatically added by the editor to close the `If` statement if you press **<Enter>**.

You have now created the general set of instructions to the program what to do with the user action in the form.

You can test the macro now to see the dialog populated with the initialized values. Select the **Run** tool in the **Execute** group.  The macro dialog opens. Click either **OK** or **Cancel** to close it. Neither perform any action at this point other than to log a debug message.

The next step is to have the script then perform the desired actions when the users clicks **OK**.

Tip: Now is a good time to save the work done to this point.

OS. To get the user values

1. Place your cursor just after the `Debug.Print "OK Button pressed"` line and then press **<Enter>** several times to create additional space between this and the following line beginning with `ElseIf`.
2. Type the following commands to populate the variables with the values the user types into the dialog:

```
        'Get the values
        clmn = Abs( CDbl(dlg.clmn) )
        row = Abs( CDbl(dlg.row) )
        ht = Abs( CDbl(dlg.ht) )
        wdth = Abs( CDbl(dlg.wdth) )
        sprt = CStr(dlg.sprt)
```

Note: The last captures the support array position as a string whereas the others all capture absolute value of the user input for the respective data fields.

3. Type the following debug statements to provide back the captured values:

```
        Debug.Print "No. of Horizontal Bays = ";clmn
        Debug.Print "No. of Vertical Bays = ";row
        Debug.Print "Vertical Distance = ";ht
        Debug.Print "Horizontal Distance = ";wdth
        Debug.Print "Support Type = ";sprt
```

4. Type the following command to set the initial values for the coordinate variables:

```
        crdx = 0
        crdy = 0
        crdz = 0
```

OS. To initialize OpenSTAAD and calculate the node coordinates

1. Type the following command to initialize OpenSTAAD:

```
       Dim objOpenSTAAD As Object
       Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")
```

   This command opens the connection to STAAD.Pro and the model currently open in the program.

   Note: You can initialize OpenSTAAD to any variable. Here, `objOpenSTAAD` is used.

2. Type the following commands to reference the `geometry` class in a variable:

```
        Dim geometry As OSGeometryUI
        Set geometry = objOpenSTAAD.Geometry
```

3. Type the following two `For` loops to calculate the nodal coordinates for each node in the frame.

   Note: There are two nested loops used because this is a 2D frame. The Z coordinate was previously set to zero and will not change.

```
        'Nodes
        For j = 2 To (row + 2)
            For i = 1 To (clmn + 1)
                crdx = (i - 1) * wdth
                geometry.AddNode crdx, crdy, crdz
            Next
            crdy = (j - 1) * ht
        Next
```

   Note: The `Next` command to close the `For` loop is added automatically once you press **<Enter>** after typing the `For` command line.

4. Type the following commands to reference the `support` class in a variable:

```
        Dim support As OSSupportUI
        Set support = objOpenSTAAD.Support
```

5. Type the following `If ElseIf` statement to get the desired support type.

   The support is created and the returned support reference number is stored to the variable `s1`.

   A “fall back” possibility is also added in case neither of the intended support types is somehow specified, then an error message is presented to the user.

```
        'Supports
        If sprt = "0" Then
            s1 = support.CreateSupportFixed()
        ElseIf sprt = "1" Then
            s1 = support.CreateSupportPinned()
```

```
        Else
            MsgBox("Select Proper Support Type",vbOkOnly,"Error")
            Exit Sub
        End If
```

Note: The OpenSTAAD command `support.CreateSupportFixed` can also be used to simply create a new support without using the return value. But by adding the parenthesis at the end, the script can take the returned support reference number which allows you to store that in the variable.

6. Type the following line to add a debugging statement for the returned support reference number:

```
        Debug.Print "Support return value = ";s1
```

7. Type the following type to assign the support types to the bottom nodes in the frame:

```
        For i1 = 1 To (clmn + 1)
            support.AssignSupportToNode i1,s1
        Next
```

Tip: Now is a good time to save the work done to this point.

OS. To generate the frame members

1. Type the following commands to add the columns:

```
        'Columns
        n1 = 1
        n2 = (n1 + clmn +1)
        For k = 1 To (clmn + 1)*row
            geometry.AddBeam n1, n2
            n1 = n1 + 1
            n2 = n2 + 1
        Next
```

2. Type the following commands to add the beams:

```
        'Beams
        n1 = 1
        For k1 = 1 To row
            n1 = k1 * (clmn + 1)+1
            n2 = n1 + 1
            For k2 = 1 To clmn
                geometry.AddBeam n1, n2
                n1 = n1 + 1
                n2 = n2 + 1
            Next
        Next
```

3. Save your macro by either:

   click the **File** ribbon tab and then select the **Save** tool in the backstage view

   or

   click the **Save** tool in the quick access toolbar

   or

   press **<Ctrl+S>**

Your frame macro is now complete. You are ready to test the macro to check your code.

OS. To test your macro

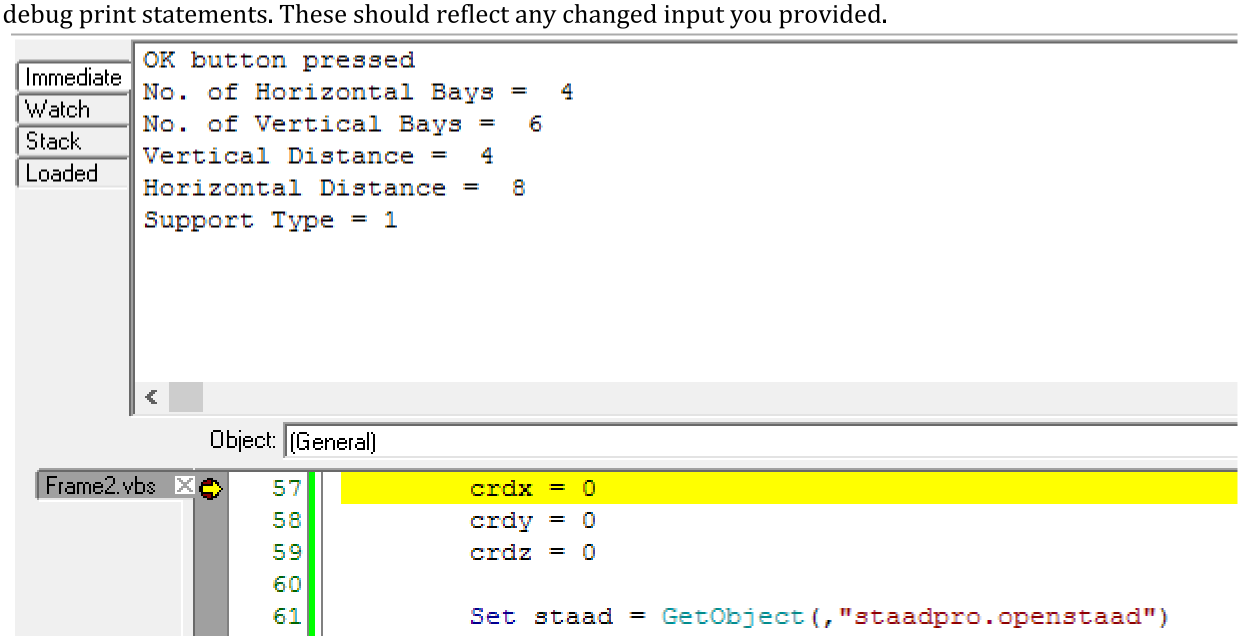
You can add break points through your code to assist in debugging. Practice adding one so you can read the debug print output you added.

If the program contains errors, the line where the program stops will be marked in red. Stop the program and then review your input to debug.

1. Click in the dark gray margin of the code editor just to the left of line number 57 (or the line containing cdrx = 0).
   A red dot appears to indicate this is a break point. The code will pause the execution at this point when executed.
2. Click the **Run** tool in the **Execute** group.

   The user dialog opens for input.
3. Change one or more values from their defaults.
4. Click **OK**.
   The code runs to the break point, which is now highlighted in yellow. The Immediate tab now shows the debug print statements. These should reflect any changed input you provided.



5. Click the Run tool in the Execute group again to continue to the next break point or to the end of the code.
6. Save your changes.
7. Select the **File** ribbon tab and then select **Exit** in the backstage view.

Your new macro is shown in the **Macro** dialog list.

Note: Testing the macro still runs within STAAD.Pro through OpenSTAAD, so you will have a frame generated if you tested it. You will need to start with a new, empty file to run the macro correctly.

OS. To add the macro to the list of user tools

Once you have completed testing of your macro, you can add it to the list of User Tools in the STAAD.Pro application. This allows you to run it with one click.

1. On the **Utilities** ribbon tab, select the **Configure** tool in the **User Tools** group.

   The **Customize User Defined Tools** dialog opens.
2. Click **New**.
3. Type **2D Frame Model** in the name of the command.
4. Click **[...]** next to the **Command** field.
5. Navigate to and select your `Frame.vbs` macro file.
6. Click **Open**.
7. Click **OK**.

> **Note:** There are no parameters of external files used by this macro, so those fields are left empty.

OS. To run the Frame macro

1. Create a new analytical modeling input file.
2. (Optional) Change the units of length to a convenient value.
3. On the **Utilities** ribbon tab, select the **User Tools** > **2D Frame Model** tool in the User Tools group.

   The **2D Frame Model** dialog opens.
4. Type the number of bays in the horizontal and vertical directions.
5. Type the spacing fo the bays int he horizontal and vertical directions (X and Y, respectively).
6. Select the Support Type.
7. Click **OK**.

The frame with supports is generated.

OS. Frame.vbs Macro

This is the full contents of the `Frame.vbs` file. You can copy this by clicking **Copy to Clipboard** below and paste into the STAAD.Pro Macro Editor window.

> **Note:** The first line of the following code points includes the OpenSTAADUI reference. It is added to your macro by OS. To connect the STAAD.Pro Script Editor to STAAD Object Library (on page 7020). It is not displayed within in the STAAD.Pro Script Editor, though.

```
'#Reference {EDA9FA7F-EFC9-4264-9513-39CF6E72604D}#1.0#0#C:\Program Files\Bentley
\Engineering\STAAD.Pro CONNECT Edition\STAAD\StaadPro.dll#OpenSTAADUI#OpenSTAADUI
```

```
'/
*-------------------------------------------------------------------------------------
+
'|    $Macro File: Frame.vbs $
'|       $Date: 15/Aug./2017 $
'|
'|  $Created by Surojit Ghosh and Jason Coleman. All rights reserved. $
'+------------------------------------------------------------------------------------
-*/
Sub Main()
'DESCRIPTION:Create a 2D frame with supports

    Begin Dialog UserDialog 600,200,"2D Frame Model" ' %GRID:5,5,1,1
        Text 20,20,190,15,"No. of Horizontal Bays:",.Text1
        Text 20,45,190,15,"No. of Vertical Bays",.Text2
        Text 20,70,190,15,"Vertical Distance",.Text3
        Text 20,95,190,15,"Horizontal Distance",.Text4
        Text 20,130,190,15,"Support Type",.Text5
        TextBox 220,20,130,15,.clmn
        TextBox 220,40,130,15,.row
        TextBox 220,70,130,15,.ht
        TextBox 220,95,130,15,.wdth
        OptionGroup .sprt
            OptionButton 220,130,90,15,"Fixed",.OptionButton1
            OptionButton 370,130,90,15,"Pinned",.OptionButton2
        OKButton 360,165,90,20
        CancelButton 490,165,90,20
    End Dialog
    Dim dlg As UserDialog

    Dim dlgResult As Integer
    Dim crdx As Double
    Dim crdy As Double
    Dim crdz As Double
    Dim n1 As Long
    Dim n2 As Long
    Dim i1 As Long
    Dim s1 As Long

    'Initialization
    dlg.clmn = "3"
    dlg.row = "5"
    dlg.ht = "3"
    dlg.wdth = "5"

    'Popup the dialog
    dlgResult = Dialog(dlg)
    Debug.Clear

    If dlgResult = -1 Then 'OK button pressed
        Debug.Print "OK button pressed"

        clmn = Abs( CDbl(dlg.clmn) )
        row = Abs( CDbl(dlg.row) )
        ht = Abs( CDbl(dlg.ht) )
        wdth = Abs( CDbl(dlg.wdth) )
        sprt = CStr(dlg.sprt)
```

```
        Debug.Print "No. of Horizontal Bays = ";clmn
        Debug.Print "No. of Vertical Bays = ";row
        Debug.Print "Vertical Distance = ";ht
        Debug.Print "Horizontal Distance = ";wdth
        Debug.Print "Support Type = ";sprt

        crdx = 0
        crdy = 0
        crdz = 0

        Dim objOpenSTAAD As Object
        Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")

        Dim geometry As OSGeometryUI
        Set geometry = objOpenSTAAD.Geometry

        'Nodes
        For j = 2 To (row + 2)
            For i = 1 To (clmn + 1)
                crdx = (i - 1) * wdth
                geometry.AddNode crdx, crdy, crdz
            Next
            crdy = (j - 1) * ht
        Next

        Dim support As OSSupportUI
        Set support = objOpenSTAAD.Support

        'Supports
        If sprt = "0" Then
            s1 = support.CreateSupportFixed()
        ElseIf sprt = "1" Then
            s1 = support.CreateSupportPinned()
        Else
            MsgBox("Select Proper Support Type",vbOkOnly,"Error")
            Exit Sub
        End If
        Debug.Print "Support return value = ";s1
        For i1 = 1 To (clmn + 1)
            support.AssignSupportToNode i1,s1
        Next

        'Columns
        n1 = 1
        n2 = (n1 + clmn +1)
        For k = 1 To (clmn + 1)*row
            geometry.AddBeam n1, n2
            n1 = n1 + 1
            n2 = n2 + 1
        Next

        'Beams
        n1 = 1
        For k1 = 1 To row
            n1 = k1 * (clmn + 1)+1
            n2 = n1 + 1
            For k2 = 1 To clmn
                geometry.AddBeam n1, n2
```

```
                n1 = n1 + 1
                n2 = n2 + 1
            Next
        Next

    ElseIf dlgResult = 0 Then 'Cancel button pressed
        Debug.Print "Cancel button pressed"
    End If

End Sub
```

Alternate Control

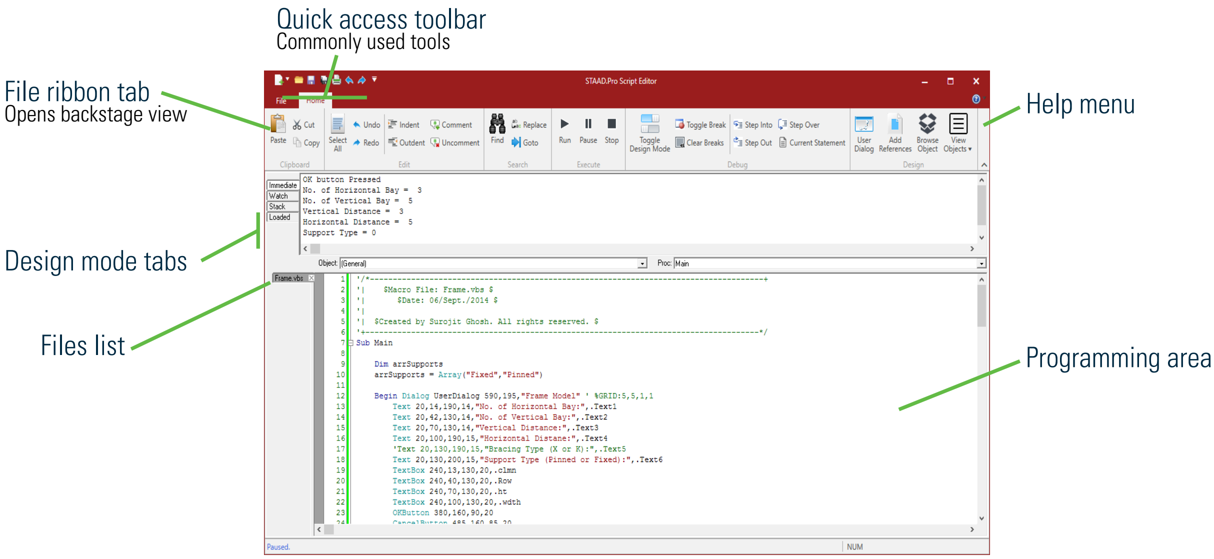
If a user is asked to choose among three or more options, often a drop-down list is a better choice of control for the user interface. The macro can altered slightly to use a drop-down list. The list is populated by an array of strings which must be declared and populated before the dialog definition.

```
Sub Main()
'DESCRIPTION:Create a 2D frame with supports

Dim arrSupports
arrSupports = Array("Fixed","Pinned")

    Begin Dialog UserDialog 600,200,"2D Frame Model" ' %GRID:5,5,1,1
        Text 20,20,190,15,"No. of Horizontal Bays:",.Text1
        Text 20,45,190,15,"No. of Vertical Bays",.Text2
        Text 20,70,190,15,"Vertical Distance",.Text3
        Text 20,95,190,15,"Horizontal Distance",.Text4
        Text 20,130,190,15,"Support Type",.Text5
        TextBox 220,20,130,15,.clmn
        TextBox 220,40,130,15,.row
        TextBox 220,70,130,15,.ht
        TextBox 220,95,130,15,.wdth
        DropListBox 240,130,130,50,arrSupports(),.sprt
        OKButton 360,165,90,20
        CancelButton 490,165,90,20
    End Dialog
    Dim dlg As UserDialog
```

The rest of the macro code is unchanged. The selection from the drop-down list is passed as the position in the array selected (e.g., "0" for the first position, "1" for the next position, etc.).

OS. STAAD.Pro Script Editor window

This application is used to write macros for use with STAAD.Pro. It is a small yet powerful programming application with tools for helping you create and debug macros.



Tip: The STAAD.Pro Script Editor is built on the WinWrap® Basic Editor platform. Select the Help tool in the top-right corner of the application to get help on both the WinWrap application as well as the WinWrap Basic language, which supports Visual Basic .NET™.

OS. Home ribbon tab

Table 916: Clipboard group

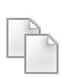
Tool Name	Description	Shortcut
 **Paste**	Pastes the clipboard contents (text only) at the cursor position.	**<Ctrl+V>**
 **Cut**	Copies the selected contents to the clipboard and deletes the original.	**<Ctrl+X>**
 **Copy**	Copies the selected contents to the clipboard.	**<Ctrl+C>**

Table 917: Edit group

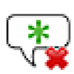
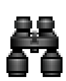
Tool name	Description	Shortcut
Select All	Selects the entire contents of the programming area.	**<Ctrl+A>**
 **Undo**	Undoes the last action in the editor.	**<Ctrl+Z>**
 **Redo**	Reverses the last undo action.	**<Ctrl+Y>**
Indent	Indents the selected line or lines.	
Outdent	Outdents the selected line or lines.	
 **Comment**	Changes the current line (or lines, if a selection spans multiple lines) to a comment. These lines are then ignored by the compiler.	
 **Uncomment**	Changes a commented line or lines back to commands to be interpreted by the compiler.	

Table 918: Search group

Tool name	Description	Shortcut
 **Find**	Opens the Find dialog, which is used to find strings that match the dialog input.	**<Ctrl+F>**
 **Replace**	Opens the Replace dialog, which is used to replace any matched strings with a different string.	**<Ctrl+H>**

Tool name	Description	Shortcut
 **Goto**	Opens the Go To Line dialog, which is used to move the cursor to a specified line number.	**<Ctrl+G>**

Table 919: Execute group

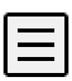
Tool name	Description
 **Run**	Runs the current macro.
 **Pause**	Pauses running the current macro. Click again to resume.
 **Stop**	Stops running the current macro.

Table 920: Debug group

Tool name	Description
 **Toggle Design Mode**	Toggles the display of the design pane, which allows you to review status and values during code debugging.
Toggle Breaks	Adds or removes a breakpoint at the line where the cursor is currently placed. **Note:** A breakpoint indicates where the script editor should suspend your code so you can take a look at the code details.
Clear Breaks	Removes all breakpoints from the code.
Step Into	Running the code with this tool advances the macro execution one step at a time.

Tool name	Description
Step Out	This tool resumes the macro execution until the current function returns.
Step Over	This tool allows you to advance beyond code your not interested in reviewing. It then advances to the next step.
Current Statement	

Table 921: Design group

Tool name	Description
 **User Dialog**	Opens the **UserDialog Editor**, which is used to create end-user dialogs for your macro.
 **Add References**	Opens the **References** dialog, which is used to include references for APIs, including OpenSTAAD.
 **Browse Object**	Opens the **ActiveX Automation Members** dialog, which allows you to browse the contents of currently available libraries.
 **View Objects**	The drop-down list is used to toggle the display of Object and Proc lists in the main view.

OS. UserDialog Editor

Used to create end-user dialogs for your macro. This will generate the `Begin Dialog`...`End Dialog` block in the macro code.

The editing tools are located along the top toolbar.

The user interface items are on tool pallet on the left side. Click any tool to active that item for layout. Then click in the dialog view to place that tool.

OS. **References** dialog

Used to include references for APIs, including OpenSTAAD.

OS. Examples

The following examples of macros can be added to run in STAAD.Pro via the **Macro** dialog.

OS. Retrieve Dynamic Output

The following example macro uses several OpenSTAAD dynamic output functions to build a mode shape report for the results of a dynamic analysis.

About this Example

This macro first verifies if analysis results are available. If so, it then begins printing information into a text results file. If the number of modes extracted are greater than zero, then the modal frequencies, mass modal participation factors, and modal deflections are printed to this report.

This code can be saved to a `.vbs` file directly or pasted into the macro editor in a macro project for use in STAAD.Pro.

Visual Basic Code

```
Option Explicit

Sub Main()
    Dim stdFile As String
    Dim stdFolder As String
    Dim rptFile As String
    Dim Tokens() As String
    Dim boolResults As Boolean

    Dim objOpenSTAAD As Object

    Set objOpenSTAAD = GetObject(,"StaadPro.OpenSTAAD")
    objOpenSTAAD.GetSTAADFile(stdFile, False)
    objOpenSTAAD.GetSTAADFileFolder(stdFolder)

    If objOpenSTAAD.Output.AreResultsAvailable = "True" Then
        Tokens = Split(stdFile,".")
        rptFile = stdFolder + "\" + Tokens(0) + ".ModeShapeData.txt"
        CreateModeShapeReport(rptFile, objOpenSTAAD, stdFile)
    Else
        MsgBox("No analysis results available for this input
file",vbOkOnly,"Error")
    End If

    Set objOpenSTAAD = Nothing

End Sub

Private Function CreateModeShapeReport(rptFile As String, objOpenSTAAD As
Object, stdFile As String)
 Dim I As Integer, J As Integer
 Dim nNodeCount As Long
```

```
 Dim nModeCount As Long
 Dim nModeNo As Long
 Dim strLenUnit As String
 Dim setOfNodes() As Long
 Dim setOfFrequency() As Double
 Dim modVal(6) As Double

 Dim szName As String
 Dim tblno As Long
 Dim rptno As Long
 Dim idx As Long

 Dim geometry As OSGeometryUI
 Dim Output As OSOutputUI
 Set geometry = objOpenSTAAD.Geometry
 Set Output = objOpenSTAAD.Output

 nNodeCount = geometry.GetNodeCount()

'Variant GetNoOfModesExtracted();
 nModeCount = Output.GetNoOfModesExtracted()

 Open rptFile For Output As #10
 Print #10, "Mode Shape Data Report for",stdFile
 Print #10, ""
 Print #10, Space$(3);"No of Nodes  = ";nNodeCount
 Print #10, Space$(3);"No of Modes Extracted = ";nModeCount
 Print #10, ""

 If nModeCount > 0 Then

     ReDim setOfNodes(nNodeCount)
     ReDim setOfFrequency(nModeCount)

'Variant GetModeFrequency(Variant varMode, Variant varFreq);
     Print #10, "Mode      Frequency (Hz)"
     Print #10, "------------------------"
     For I = 0 To nModeCount - 1
        nModeNo = I+1
        Output.GetModeFrequency(nModeNo, setOfFrequency(I))
        Print #10, nModeNo;Space$(10);Format$(setOfFrequency(I),"Standard")
     Next
     Print #10, "------------------------"
     Print #10,

'Variant GetModalMassParticipationFactors(Long longMode, Variant varfactorX,
Variant varfactorY, Variant varfactorZ);
    Dim Participation(3) As Double
    Dim ParticipationSum(3) As Double
    Print #10, Space$(18);"Modal Participation Factors Table"
    Print #10, "Mode    Participation X (%)    Participation Y (%)
Participation Z (%)"
     Print #10,
"-------------------------------------------------------------------------"
     For I = 0 To nModeCount - 1
        nModeNo = I+1
        Output.GetModalMassParticipationFactors(nModeNo, Participation(1),
Participation(2), Participation(3))
```

```
        Print #10, nModeNo;Space$(10);Format$(Participation(1),"#0.00"); Space
$(20); Format$(Participation(2),"#0.00"); Space$(20); Format$
(Participation(3),"##0.00")
        ParticipationSum(1) = ParticipationSum(1) +Participation(1)
        ParticipationSum(2) = ParticipationSum(2) +Participation(2)
        ParticipationSum(3) = ParticipationSum(3) +Participation(3)
     Next
     Print #10,
"---------------------------------------------------------------------------"
     Print #10, "Sum";Space$(9);Format$(ParticipationSum(1),"#0.00"); Space
$(20); Format$(ParticipationSum(2),"#0.00"); Space$(20); Format$
(ParticipationSum(3),"#0.00")
     Print #10,

'Variant GetModalDisplacementAtNode(Variant varMode, Variant arNode, Variant
varModalDisps);
     geometry.GetNodeList(setOfNodes)
     objOpenSTAAD.GetInputUnitForLength(strLenUnit)

     Print #10, Space$(10);"Modal Displacements Table"
     Print #10, "Mode  Node         x            y             z"
     Print #10, Space$(18);"(";strLenUnit;")";Space
$(9);"(";strLenUnit;")";Space$(10);"(";strLenUnit;")"
     Print #10, "---------------------------------------------------"
     For I = 0 To nModeCount - 1
        nModeNo = I+1
        For J = 0 To nNodeCount - 1
            Output.GetModalDisplacementAtNode(nModeNo, setOfNodes(J), modVal)
            Print #10, Format$(nModeNo,"00");Space$(5);Format$
(setOfNodes(J),"000");Space$(5);Format$(modVal(0),"Scientific");Space
$(5);Format$(modVal(1),"Scientific");Space$(5);Format$(modVal(2),"Scientific")
        Next J
     Next I
     Print #10, "---------------------------------------------------"
     Print #10,

'The following function is currently not operational:-
'Variant GetMissingMassParticipationFactors

 End If

Close #10

End Function
```

OS. Envelopes Table Macro

The purpose of this macro is to create a results table in STAAD.Pro containing an envelope of results.

The macro contains the following routines:

1. `Main`. This is the primary routine from which the macro is launched and checks that STAAD.Pro is running and the model loaded has available results which are needed for this example. However if the table you wish to construct does not require the model to have been analysed, then clearly that part of the routine can be removed.

2. `STAADTable`. Once the validation is done, this routine is called to produce the table in STAAD.Pro. As this table will require a selection of load cases, it includes a call to a routine to select load cases called `SelectLoadCases`, then the routine to create the empty table called `CreateTable` and finally to fill the table with data called `FillTable`.
3. `ResetEnvTable`. A simple routine to make sure the table is cleared and a label added in column 1 of each row for this example to mirror the labels used in the general Envelope sheet of the Node displacement table.
4. `SelectLoadCases`. This is routine that displays a user dialog to select the load cases and combinations from which the node displacements will be used to form the final table. This makes use of a couple of other routines `AddLoadCaseToSelected`, and `ExcludeLoadCaseFromSelected` to maintain two lists of load cases, those that are available from STAAD.Pro and those that will be used to create the table.
5. `AddLoadCaseToSelected`. When **[>]** is clicked, this routine adds the selected load cases.
6. `ExcludeLoadCaseFromSelected`. When **Exclude** is clicked, the load case in the selected Load Case list is removed from the list.
7. `CreateEnvList`. A simple routine that creates a list of the load case numbers from the text of the dialog box Selected Load Cases.
8. `FillTable`. Populates the table with the calculated data which has been put into a two-dimensional array (i.e., row, column)
9. `CreateTable`. The routine that forms the table framework and sets the headings. Note that it also checks to see the unit system so that the headers can include the appropriate units. Also note that there are commented out lines that indicate how additional sheets can be added to the Report that could be used for other data such as End Forces or Reactions.

Macro Code

To use this macro, copy and the paste the code into a `.vbs` file (e.g., `Table Envelope.vbs`). Then OS. To import an existing macro (on page 7019) into STAAD.Pro to use it.

```
'#Reference {EDA9FA7F-EFC9-4264-9513-39CF6E72604D}#1.0#0#C:\Program Files
\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD
\StaadPro.dll#OpenSTAADUI#OpenSTAADUI
'Simple Macro using OpenSTAAD to create a table of envelopes.
'v1.0 (22 Dec 2015) CA
'v1.1 (23 Dec 2015) CA - Minor index issue fixes
'v1.2 (08 May 2020) JTC - Update with CE

Option Explicit

Public staadObj As Object
Public Geometry As OSGeometryUI
Public Loads As OSLoadUI
Public Output As OSOutputUI
Public Tables As OSTableUI

Sub Main()
Dim stdFile As String
Dim nResult As Boolean

Set staadObj = GetObject(,"StaadPro.OpenSTAAD")
Set Geometry = staadObj.Geometry
Set Loads = staadObj.Load
Set Output = staadObj.Output
Set Tables = staadObj.Table

'Make sure STAAD is loaded and running
staadObj.GetSTAADFile(stdFile,True)
```

```
If stdFile <> "" Then    'no file loaded
    'Check there are results
    nResult = Output.AreResultsAvailable
    If nResult = True Then 'Results are available
        STAADTable staadObj
    Else
        MsgBox "This macro requires the current model to have results.",
vbOkOnly
    End If
Else
    MsgBox "This macro can only be run with a valid STAAD file loaded.",
vbOkOnly
End If

Set staadObj = Nothing
End Sub


Sub STAADTable(staadObj As Object)
    Dim nReturn As Integer
    Dim i As Integer, j As Integer, k As Integer
    'Dim Geometry As OSGeometryUI
    'Set Geometry = staadObj.Geometry

    Dim nTableRows As Integer, nCols As Integer
    nTableRows=13
    nCols = 10

    Dim tblNodes As Long, rptno As Long

    Dim lPrimaryLoadCaseCount As Long
    Dim lPrimaryLoadCaseNumbersArray() As Long
    Dim lGetLoadCombinationCaseCount As Long
    Dim lLoadCombinationCaseNumbersArray() As Long

    Dim EnvList() As Long
    Dim LoadListCount As Integer

    SelectLoadCases staadObj, EnvList(), LoadListCount

    'MsgBox Str$(LoadListCount)

    Dim EnvRowVal(13) As Double
    Dim EnvRow(13,10) As String
    Dim LoadCase As Long
    Dim ColVal As Integer

    'Node Displacement Envelope

    Dim nNodes As Long
    Dim nNode() As Long
    nNodes = Geometry.GetNodeCount()
    ReDim nNode(nNodes)
    Geometry.GetNodeList(nNode)

    Dim dDisplacementArray(6) As Double
```

```
    Dim nResultant As Double

    ResetEnvTable EnvRow, nTableRows, nCols

    For i = 1 To LoadListCount
        LoadCase = EnvList(i)
            For j = 0 To nNodes-1
                nReturn = Output.GetNodeDisplacements( nNode(j), LoadCase,
dDisplacementArray)
                nResultant =
(dDisplacementArray(0)^2+dDisplacementArray(1)^2+dDisplacementArray(2)^2)^0.5
                For k = 1 To 6
                    'max values
                    If dDisplacementArray(k-1) > EnvRowVal(2*k-1) Then
                        EnvRowVal(2*k-1) = dDisplacementArray(k-1)
                        EnvRow(2*k-1, 2)= Str$(nNode(j))
                        EnvRow(2*k-1, 3)= Str$(LoadCase)
                        For ColVal = 1 To 3
                            EnvRow(2*k-1, ColVal+3) = Format$
(dDisplacementArray(ColVal-1),"#.000")
                        Next ColVal
                        EnvRow(2*k-1, ColVal+3) = Format$(nResultant,"#.000")
                        For ColVal = 1 To 3
                            EnvRow(2*k-1, ColVal+7) = Format$
(dDisplacementArray(ColVal+2)*57.2958,"#.000")
                        Next ColVal
                    End If

                    'min values
                    If  dDisplacementArray(k-1) < EnvRowVal(2*k) Then
                        EnvRowVal(2*k) = dDisplacementArray(k-1)
                        EnvRow(2*k, 2)= Str$(nNode(j))
                        EnvRow(2*k, 3)= Str$(LoadCase)
                        For ColVal = 1 To 3
                            EnvRow(2*k, ColVal+3) = Format$
(dDisplacementArray(ColVal-1),"#.000")
                        Next ColVal
                        EnvRow(2*k, ColVal+3) = Format$(nResultant,"#.000")
                        For ColVal = 1 To 3
                            EnvRow(2*k, ColVal+7) = Format$
(dDisplacementArray(ColVal+2)*57.2958,"#.000")
                        Next ColVal
                    End If

                    'resultant
                    If nResultant > EnvRowVal(13) Then
                        EnvRowVal(13) = nResultant
                        EnvRow(13, 2)= Str$(nNode(j))
                        EnvRow(13, 3)= Str$(LoadCase)
                        For ColVal = 1 To 3
                            EnvRow(13, ColVal+3) = Format$
(dDisplacementArray(ColVal-1),"#.000")
                        Next ColVal
                        EnvRow(13, ColVal+3) = Format$(nResultant,"#.000")
                        For ColVal = 1 To 3
                            EnvRow(13, ColVal+7) = Format$
(dDisplacementArray(ColVal+2)*57.2958,"#.000")
                        Next ColVal
```

```
                    End If
                Next k
            Next j
    Next i


'Create the Table
    'CreateTable staad,nTableRows,rptno, tblNodes, tblBeams,tblReactions, etc
    CreateTable staadObj, rptno, tblNodes, nTableRows

    'Now fill the data
    FillTable staadObj,rptno, tblNodes, EnvRow, nTableRows, nCols

End Sub


Sub ResetEnvTable(EnvRow() As String, nTableRows As Integer, nCols As Integer)
    Dim i As Integer, j As Integer

    For i = 1 To nCols
        For j = 1 To nTableRows
            EnvRow(j,i)="*"
        Next j
    Next i

    'Row lables
    EnvRow(1,1) = "Max X"
    EnvRow(2,1) = "Min X"
    EnvRow(3,1) = "Max Y"
    EnvRow(4,1) = "Min Y"
    EnvRow(5,1) = "Max Z"
    EnvRow(6,1) = "Min Z"
    EnvRow(7,1) = "Max rX"
    EnvRow(8,1) = "Min  rX"
    EnvRow(9,1) = "Max rY"
    EnvRow(10,1) = "Min rY"
    EnvRow(11,1) = "Max rZ"
    EnvRow (12,1) = "Min rZ"
    EnvRow (13,1) = "Max Res."

End Sub


Sub SelectLoadCases(staadObj As Object, EnvList() As Long, lSelectedCasesNum
As Integer)
    Dim i As Integer
    Dim j As Integer
    Dim nResult As Integer
    Dim iButton As Integer

    Dim LCases As Integer
    Dim LCCases As Integer
    Dim lstLoadNums() As Long
    Dim lstAvailableCases() As String

    LCases = Loads.GetPrimaryLoadCaseCount()
    ReDim lstLoadNums(LCases)
    ReDim lstAvailableCases(LCases)
```

```
    Loads.GetPrimaryLoadCaseNumbers (lstLoadNums)

    For i =0 To LCases-1
        lstAvailableCases(i)= CStr(lstLoadNums(i)) &" : " &
Loads.GetLoadCaseTitle(lstLoadNums(i))
    Next i

    Dim lstLoadComNum() As Long
    LCCases = Loads.GetLoadCombinationCaseCount()
    ReDim lstLoadComNum(LCCases)
    ReDim Preserve lstLoadNums(LCases+LCCases)
    ReDim Preserve lstAvailableCases(LCases+LCCases)
    Loads.GetLoadCombinationCaseNumbers(lstLoadComNum)

    For i =0 To LCCases-1
        lstLoadNums(LCases+i)=lstLoadComNum(i)
        lstAvailableCases(LCases+i)= CStr(lstLoadNums(LCases+i)) &" : " &
Loads.GetLoadCaseTitle(lstLoadNums(LCases+i))
    Next i

    Dim lstSelectedCases() As String
    lSelectedCasesNum = 0
    ReDim Preserve lstSelectedCases(lSelectedCasesNum)
    lstSelectedCases(0) = "(None)"

'Select load case dialog
    Begin Dialog UserDialog 720,287,"Select Load Cases and Combinations" '
%GRID:10,7,1,1
        Text 20,7,170,14,"Available Cases:-",.Text1
        ListBox 20,28,310,175,lstAvailableCases(),.AvailableListBox
        PushButton 350,98,40,28,">",.PushButton1
        PushButton 70,210,200,28,"Add All Cases",.AddAll
        Text 420,7,170,14,"Selected Cases:-",.Text2
        ListBox 410,28,290,175,lstSelectedCases(),.SelectedListBox
        PushButton 460,210,200,28,"Exclude Selected Case",.PushButton2
        OKButton 270,259,90,21
        CancelButton 380,259,90,21

    End Dialog
Dim dlg As UserDialog

'dlg.SelectedListBox = 1

    Do
    iButton = Dialog (dlg)

    Select Case iButton
        Case -1
        ' OK pressed
        If lSelectedCasesNum>0 Then
            ReDim EnvList(lSelectedCasesNum)
            CreateEnvList EnvList,  lstSelectedCases, lSelectedCasesNum
        Else
            MsgBox "No load cases were selected."
            End
        End If

        Case 0
```

```
            'Cancel button Pressed
            End

        Case 1
            'Add button pressed
            Dim NewLoadCase As String
            NewLoadCase = lstAvailableCases(dlg.AvailableListBox)
            AddLoadCaseToSelected NewLoadCase, lstSelectedCases,
lSelectedCasesNum

        Case 2
            'Add All cases
            lSelectedCasesNum = LCases+LCCases
            ReDim lstSelectedCases(lSelectedCasesNum)

            For i = 0 To lSelectedCasesNum-1
                lstSelectedCases(i) = lstAvailableCases(i)
            Next i

        Case 3
            'Exclude button pressed
            Dim RemoveLoadCase As String
            'Check if an item selected
            If dlg.SelectedListBox >-1 Then
                RemoveLoadCase = lstSelectedCases(dlg.SelectedListBox)
                ExcludeLoadCaseFromSelected RemoveLoadCase, lstSelectedCases,
lSelectedCasesNum
                ReDim Preserve lstSelectedCases(lSelectedCasesNum)
            End If

        Case Else
            MsgBox "Error - We should not be here!.", vbOkOnly
            End

        End Select

        Loop Until iButton = -1


End Sub

Sub AddLoadCaseToSelected (NewLoadCase As String, lstSelectedCases() As
String, lSelectedCasesNum As Integer)

    Dim i As Integer
    Dim CaseName As String

    'Check if first
    If lstSelectedCases(0)="(None)" Then
        lstSelectedCases(0) = NewLoadCase
        lSelectedCasesNum =1
    Else

        'Check if selected case is already in list
        For i = 1 To lSelectedCasesNum
            If NewLoadCase = lstSelectedCases(i-1) Then
                GoTo EndSub
            End If
```

```
        Next i

        'if not current included, add the selected available load case to the
selected list
        lSelectedCasesNum = lSelectedCasesNum+1
        ReDim Preserve lstSelectedCases(lSelectedCasesNum)
        lstSelectedCases(lSelectedCasesNum-1)= NewLoadCase

    End If

EndSub:
End Sub



Sub ExcludeLoadCaseFromSelected (RemoveLoadCase As String, lstSelectedCases()
As String, lSelectedCasesNum As Integer)

    Dim i As Integer, nReduce As Integer
    Dim CaseName As String

    If lSelectedCasesNum =1 Then
        lstSelectedCases(0) = "(None)"
        GoTo EndSub
    End If

    For i = 0 To lSelectedCasesNum-1
        If RemoveLoadCase = lstSelectedCases (i) Then
            nReduce = 1
            If i = lSelectedCasesNum Then
                lstSelectedCases(i) = "(last)"
            Else
                lstSelectedCases(i) = lstSelectedCases(i+1)
                RemoveLoadCase = lstSelectedCases(i)
            End If
        End If
    Next i

'remove the selected load case from the selected list
            'lSelectedCasesNum = lSelectedCasesNum-1
    lSelectedCasesNum = lSelectedCasesNum - nReduce
    ReDim Preserve lSelectedCases(lSelectedCasesNum)

EndSub:
End Sub


Sub CreateEnvList (EnvList() As Long,  lstSelectedCases() As String,
lSelectedCasesNum As Integer)

    Dim i As Integer

    For i = 1 To  lSelectedCasesNum
        EnvList(i) = Val(lstSelectedCases(i-1))
    Next i

End Sub
```

```
Sub FillTable (staadObj As Object, rptno As Long, tblNodeDisplacement As
Long, EnvRow() As String, nRows As Integer, nCols As Integer)

    Dim i As Integer, j As Integer

    For i = 1 To nRows
        For j =1 To nCols
                Tables.SetCellValue(rptno,tblNodeDisplacement,i,j,
EnvRow(i,j))
        Next j
    Next i

End Sub


Sub CreateTable(staadObj As Object,  rptno As Long, tblNodeDisplacement As
Long, NoRows As Integer)
    Dim unit As Integer
    Dim ForceLabel As String, DistanceLabel As String
    unit = staadObj.GetBaseUnit
        Select Case unit
        Case 1 ' English
            DistanceLabel ="in"
            ForceLabel="kiP"
        Case 2 'Metric
            'DistanceLabel ="m"
            'Displacements for metric models will generally be wanted in mm
            DistanceLabel ="mm"
            ForceLabel="kN"
        Case Else 'This should not occur!
            DistanceLabel ="**"
            ForceLabel="???"
        End Select

'Table name
 rptno = Tables.CreateReport("User Envelopes")

'Table sheet name, number of  rows and columns
 tblNodeDisplacement = Tables.AddTable(rptno, "Node Displacements", NoRows,
10)
 'tblEndForce = staad.Table.AddTable(rptno, "End Forces", NoRows, 10)
 'tblReaction = staad.Table.AddTable(rptno, "Reactionss", NoRows, 10)

'Column headings
 Tables.SetColumnHeader rptno, tblNodeDisplacement, 1, "(Type)"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 1, "")

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 2, "Node"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 2, "")

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 3, "L/C"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 3, "")

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 4, "X"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 4, DistanceLabel)

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 5, "Y"
```

```
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 5, DistanceLabel)

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 6, "Z"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 6, DistanceLabel)

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 7, "Resultant"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 7, DistanceLabel)

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 8, "rX"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 8, "deg")

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 9, "rY"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 9, "deg")

 Tables.SetColumnHeader rptno, tblNodeDisplacement, 10, "rZ"
 Tables.SetColumnUnitString( rptno, tblNodeDisplacement, 10, "deg")

End Sub
```

OS. Macros Included with STAAD.Pro

Several macros are installed with the program.

These Visual Basic files are installed in `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\PlugIns\VBS` by default.

Note: Additional information on using these macros in the program can be found in Utilities tab (on page 2940).

File name	Description
`Create Material.vbs`	Reads entries in the file `MaterialSpreadsheet.csv` and displays a list of the material/grade names. Users can add the selection as a material definition to the model.
`EuroCombinations.vbs`	Used to add combinations with load factors per the Eurocode to the model.
`ObjectIDs.vbs`	Generates a table that displays GUIDs of nodes, members, and physical members.
`ObjectUpdateReport.vbs`	This macro can be used to create two Object ID reports, one capturing the situation before the STAAD model went through the CIS/2 Update tool and one post-update. By comparing the two reports, you can find out which entities are newly introduced due update workflow and if this is the intended result (i.e., should those be new or should have retained the original GUID, but got changed due to a workflow problem/bug, etc.).
`STAAD2SACS.vbs`	Used to export the model data to a Bentley SACS model.
`ToAutoPipePub.vbs`	Used to export the model data to a Bentley AutoPIPE model.

OS. Writing OpenSTAAD in Other Programming Languages

You can write scripts or even standalone applications which utilize STAAD.Pro through OpenSTAAD. This requires a separate environment in which to write the code (typically an integrated development environment, or IDE) as well as a code interpreter for the programming language you use.

COM Compliant Languages

Any program language capable of calling Windows Component Object Model (COM) components may be used. While it isn't feasible to document the usage of all such languages, some commonly used options are described here.

Although OpenSTAAD supports all major programming languages used today capable of calling COM components, it is impractical to document the usage of each and every function in all of these languages.

Using a Development Environment

Unlike writing macros in the STAAD.Pro Script Editor or in a Visual Basic for Application Editor in an Office application, writing code in another programming language requires a development environment. This is any sort of application that allows you to both edit and run code. These development environments often come with coding aids such as autocompletion features, error detection and suggestions, compilers, and more. This is typically referred to as an Integrated Development Environment (or IDE).

There are many different commercial and open source IDE options, but these guides will typically stick with Microsoft's Visual Studio 2019. This is widely used IDE and the Community edition can be used for free at https://visualstudio.microsoft.com/downloads/.

OS. Getting Started with Python

This section will provide you with the information needed to set up an OpenSTAAD project written in Python.

Note: While this doesn't assume that you have extensive Python programming knowledge, it also is not intended to teach the Python programming language. There are many excellent resources available to learn Python, which is beyond the scope of the OpenSTAAD documentation.

OS. OpenSTAAD and Python

OpenSTAAD is compatible with any programming language that supports COM objects, such as Python. However, there are some limitations when using Python of which you should be aware:

- Many editors or *IDE*s don't support auto-completion due to the COM object implementation.
- OpenSTAAD methods aren't automatically detected, so your program must explicitly identify each.
- Passing arrays to the OpenSTAAD COM object requires some additional effort

OS. Set Up Your Coding Environment

In order to program OpenSTAAD in Python, you will need to set up a development environment.

A develop environment is typically a program that supports writing the code as well as a suite of tools to make that job easier. While computer code could be written in any plain text editor, an integrated development environment (IDE) will make the job much easier and more efficient. For this guide, you will use Microsoft Visual Studio Code. This is a popular programming editor which is free to use and supports many different programming languages through a wide array of extensions.

Note: Microsoft has an excellent reference on getting started programming in python within Visual Studio Code at https://code.visualstudio.com/docs/python/python-tutorial.

You are free to use any editor or IDE of your choice, so long as you have the pywin32 extension added. For example, if you are comfortable with using Visual Studio 2019, you can read about using Python in Visual Studio at https://docs.microsoft.com/en-us/visualstudio/python/.

1. Download and install Visual Studio Code from https://code.visualstudio.com/
2. Install a python interpreter.

   This is the python language version you will be using. This performs the actual operations of your code. You can download Python from https://www.python.org/downloads/.

   Note: It is recommended that you install Python 3.8, which is the latest version at the time this guide was written.
3. Launch the Visual Studio Code application.
4. Install the Python extension in Visual Studio Code.

   The extension is used to help you easily write Python code in the editor. It can also interact with a interpreter to allow you to run and debug the code from within the editor itself.

   a. Select the **Extensions** tab (or press **<Ctrl+Shift+X>**).
   b. Search for the term "python".
   c. Select the Python extension (named `ms-python.extension`).
   d. Click **Install**.

   Note: The Python extension will also later prompt you to install a linter (used to provide useful feedback and error detection) and IntelliSense code completion. These are recommended and useful tools for any editor or IDE.
5. Verify the Python installation.

   a. Open a new terminal window in Visual Studio Code.
   a. Type `py -3 --version` and the press **<Enter>**.
   The terminal should return with the version of python you have installed.

   Note: If you have multiple versions of Python installed on your computer, you may need to select the most recent version in the status bar of the Visual Studio Code program window.
6. Install pywin32.

   This allows COM objects to interact with Python and is required for using OpenSTAAD in Python. You will use a package management system built into the Python extension to download an install this.

   a. Open a new terminal in Visual Studio Code by either:

   selecting **Terminal** > **New Terminal**

   or

   press **<Ctrl+Shift+'>**
   b. Type `pip install pywin32` and then press **<Enter>**.

The package is downloaded and installed.

c. Close the Powershell window.

OS. Start Your Python Project

This quick set of steps demonstrates how to set up your workspace for a Python project.

1. In Visual Studio Code, save your new workspace to a convenient location.
   For example, `C:/OS_Py/OpenSTAAD_Python_Tutorial.workspace`.

   The workspace in Visual Studio Code serves as your working folder for saving and executing code.
2. Create a new file.
3. Save the file with a "`.py`" file extension.

   This automatically instructs Visual Studio Code to interpret the input as Python.

   Note: If you did not previously install the Python extension, then the program prompts you to do so. Also, if you have not previously installed a linter and code completion option, then the program prompts you to do so. These are recommended.
4. In the main editor window, begin typing commands.
   For example, type `print("Hello World!")`.
5. Select the **Run** tool to execute your code.
   The terminal will open and display the results.
6. Save your code.

While having your computer return "Hello World!" may be a classic example of a first program, it is not a very practical tool. However, you now have a file set up to create a more useful OpenSTAAD project in your next example.

OS. Write an OpenSTAAD Program in Python

This example will introduce you to several concepts necessary to writing OpenSTAAD applications in Python.

Tip: If you followed the previous "Hello World!" example, you may simply delete that line and start from that point.

OS. Initiate OpenSTAAD in Python

Note: You must have a model open in STAAD.Pro for this example. Otherwise your code will return an error.

1. Type `import win32com.client` and then press **<Return>**.

   This uses the pywin32 extension for Windows to allow you to use COM objects (such as OpenSTAAD).

2. Type `os = win32com.client.GetActiveObject("StaadPro.OpenSTAAD")` and then press **<Return>**.

   This initiates OpenSTAAD and connects to the current STAAD.Pro model to your program.

Your program at this point should look like:

```
import win32com.client
os = win32com.client.GetActiveObject("StaadPro.OpenSTAAD")
```

OS. Use Geometry Methods

1. Type `geometry = os.Geometry` and then press **<Return>**.

   This defines the geometry variable as the Geometry group of methods in OpenSTAAD.
2. Type `geometry._FlagAsMethod("GetNodeCount") geometry._FlagAsMethod("GetMemberCount")` and then press **<Return>**.

   This is required to correctly identify the OpenSTAAD methods as such in Python.

   > **Note:** This approach is required for this first time you use a method in a python program using OpenSTAAD method. It is not required for each time that method is re-used within the same code.
3. Type `nodeCount = geometry.GetNodeCount` and then press **<Return>**.

   This uses the OpenSTAAD method `GetNodeCount()` to return the number of nodes in the active STAAD.Pro model into the specified variable.
4. Type `beamCount = geometry.GetMemberCount` and then press **<Return>**.

   Similarly, this uses the OpenSTAAD method `GetMemberCount()` to return the number of members in the active STAAD.Pro model into the specified variable.

   > **Tip:** Reference the OpenSTAAD documentation for details on available methods and what values are accepted as input or returned from each.

Your program at this point should look like:

```
import win32com.client
os = win32com.client.GetActiveObject("StaadPro.OpenSTAAD")
geometry = os.Geometry
geometry._FlagAsMethod("GetNodeCount")
geometry._FlagAsMethod("GetMemberCount")
nodeCount = geometry.GetNodeCount
beamCount = geometry.GetMemberCount
```

OS. Generate OpenSTAAD Output

1. Type `print( str(nodeCount) + ' joints')` and then press **<Return>**.

   Working from the inner-most operation outward, this line is first converting the numerical result stored in `nodeCount` to a string. This is then combined with the text string `' joints'` to make the output easier to interpret. Then the entire concatenated string is being printed to the terminal.
2. Type `print(str(beamCount) + ' members')`.

   Similarly, this is concatenating the `beamCount` value, converted to a string, with some text and displaying that result to the terminal.

Your final program should look like:

```
import win32com.client
os = win32com.client.GetActiveObject("StaadPro.OpenSTAAD")
geometry = os.Geometry
geometry._FlagAsMethod("GetNodeCount")
geometry._FlagAsMethod("GetMemberCount")
nodeCount = geometry.GetNodeCount
beamCount = geometry.GetMemberCount
print(str(nodeCount) + ' joints')
print(str(beamCount) + ' members')
```

OS. Run Your Code

You should now have a working piece of python code which will provide you information about the currently active STAAD.Pro model. You'll now run the code to test it.

1. Save your progress by either:

   selecting **File** > **Save**

   or

   pressing **<Ctrl+S>**
2. Select the **Run** tool to execute your code.
   The terminal will open and display the results.

   > **Tip:** You may notice the Powershell instruction used to perform this action. This command can be used from any Powershell window outside of Visual Studio Code to run your program.

It is best practice to annotate your code with comments. This allows others to easily understand what your code is doing (or at least what your intention was). The full example here contains comments accordingly.

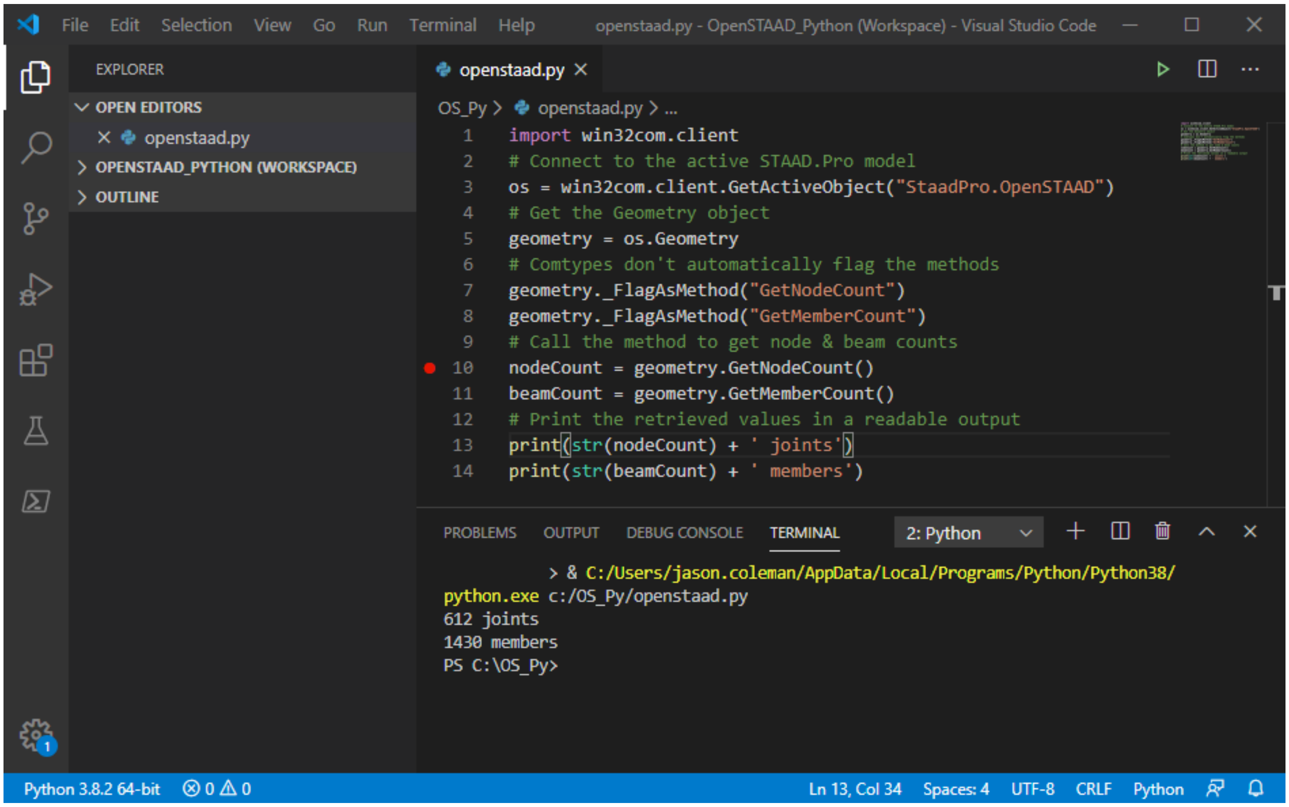


Figure 573: Microsoft Visual Studio Code running a Python example

OS. What if it Didn't Work?

If you did not already do so, be sure to install a linter. This provides useful feedback on your python program along with helpful hints to optimize the code.

The IDE should also provide you feedback on any detected issues with the code. Much like a spell check or grammar check in a word processor, this can alert you to errors before you ever run the code.

Verify that the full environment is set up, including the python interpreter (Python v3.8.n), the pywin32 extension, and that you have a model open in STAAD.Pro. While not all STAAD.Pro programs do require an active model in STAAD.Pro, this particular program does.

You can use the debugging tools in Visual Studio Code to help “step” through the code interpretation to identify issues. Clicking to the left of the line number in the main code window will add a break point. Then select the **Run** tab (or press **<Ctrl+Shift+D>**). Click the **Run and Debug** button to start the process. The terminal will provide additional information as it runs the program and will pause at each break point.

Refer to the Visual Studio Code help for detailed information on using the debugging features in this editor.

OS. Interpreting OpenSTAAD API Syntax for Python

The functions documented for the OpenSTAAD are given in the C++ syntax. It’s typically a simple process to interpret these for use in a Python program.

Classes and Methods

Classes syntax are listed in the format `OSClassUI::Method`. In Python, this should be instead given as `Class.Method`.

In the example, OpenSTAAD was instantiated using the variable “os”. Classes were then called using the format `os.Class` (for example, the Geometry class was referenced using `os.Geometry`). In the C++ syntax used in the documentation, where you see “OS*Class*UI”, instead simply use “*Class*” in Python.

The exception of course is the “root” OpenSTAAD functions, which will use whatever variable you have used to instantiate OpenSTAAD (looking to the example again, this would be `os.GetBaseUnit`). That is, the OpenSTAAD object acts as the class for these functions.

Note: In Python, you will have to use the `_FlagAsMethod` function to identify each OpenSTAAD method as a method once.

Return Values

The syntax listed indicates if a function has a return value with a leading “VARIANT”. This simply means that function will return a value that typically will be stored into a variable. If the function syntax is listed with a leading “void”, then there is no return value.

Note: A return value of a function is not to be confused with values stored in function parameters!

Variables and Parameters

The syntax listed typically indicates parameters as type “VARIANT FAR”, which is not necessarily useful.

Instead, it is important that you read the parameter descriptions which will indicate what type of variable to use (i.e., “string”, “long”). In Python, you don’t have to declare a variable type, it will be decided when the value is interpreted. Of course, it does matter if the variable contains a string or a number and what type of number when you want to use it in another function. OpenSTAAD may return integers or floating point decimal places.

OS. Getting Started with C#

This section will provide you with the information needed to set up an OpenSTAAD project written in C#.

Note: While this doesn’t assume that you have extensive C# programming knowledge, it also is not intended to teach the C# programming language. There are many excellent resources available to learn C#, which is beyond the scope of the OpenSTAAD documentation.

OS. Set Up Your Coding Environment

In order to program OpenSTAAD in C#, you will need to set up a development environment.

A development environment is typically a program that supports writing the code as well as a suite of tools to make that job easier. While computer code could be written in any plain text editor, an integrated development environment (IDE) will make the job much easier and more efficient. For this guide, you will use Microsoft Visual Studio.

Note: Microsoft has an excellent reference on getting started programming in C# at https://docs.microsoft.com/en-us/dotnet/csharp/getting-started/.

1. Download and install the Visual Studio IDE by checking the steps from https://docs.microsoft.com/en-us/visualstudio/install/install-visual-studio
2. Launch the Visual Studio 2019 application

OS. Start Your C# Project

This quick set of steps demonstrates how to set up your workspace for a C# project.

This procedure is adapted from a similar tutorial at https://docs.microsoft.com/en-us/visualstudio/get-started/csharp/tutorial-console.

1. In the Visual Studio start window, click **Create a new project**.
   The **Create a new project** window opens.
2. Filter the project templates using the following:
   a. Type `console` in the search field.
   b. Select **C#** from the Language drop-down list.
   c. Select **Windows** from the Platform drop-down list.
3. Select the **Console App (.NET Framework)** from the results and then click **Next**.
   The **Configure your new project** window opens.
4. Type `OpenSTAAD Demo` in the **Project Name** field.
5. Click **Create**.
   Visual Studio opens your new project, which includes default "Hello World" code.
6. Add necessary dependencies to the manifest file:
   a. Right-click on your project's name and then select **New Item** > **Application Manifest File**.
      The template for the `app.manifest` file opens.
   b. Add the following elements within the `<assembly>` element:

```
<comInterfaceExternalProxyStub name="IOSMemberSteelDgnParams" iid="{F40BDCDA-B3DE-495C-B84F-790F4456137F}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOpenSTAADUI" iid="{3F5B8055-31C6-446E-8BED-FEE43E09D4CC}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSGeometryUI" iid="{C052FED9-A2D6-42E3-A271-2C6FB8461711}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IStaadProWindow" iid="{9EF2FF8C-E574-4A04-9462-2E4500C8EADB}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSViewUI" iid="{87B1975B-6031-487E-A0F9-FB8F69FA24E6}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSOutputUI" iid="{824F1FC0-DC86-4CC4-A4C6-83C77D7B0496}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSPropertyUI" iid="{F919EF7D-E1DD-48CB-B3C3-B3CAE9E3B5AB}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
```

```
comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSLoadUI" iid="{DAA37D16-821F-4137-88EB-
DA4EB7650E90}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSTableUI" iid="{CF1A7B89-A007-4844-A098-
CBABAEBEF304}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSSupportUI" iid="{B076EC62-9A33-4E1C-
B38C-3E45090B8531}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSCommandsUI" iid="{6B994A7C-122C-4828-
B42B-1B0CADF91F9D}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
<comInterfaceExternalProxyStub name="IOSDesignUI"
iid="{ED218322-40EE-411C-8552-209DB2C4F32B}" tlbid="{EDA9FA7F-
EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-
C000-000000000046}"></comInterfaceExternalProxyStub>
```

c. Save and close the manifest file.

7. Add the `staadpro.dll` dependency to the project file:

   This adds OpenSTAAD as a COM file reference.

   a. Open the project's `.csproj` file.

      This file is located at the rood directory of the project.

   b. Add the following element to the start of the `ItemGroup` element:

```
<COMFileReference Include="C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT
Edition\STAAD\StaadPro.dll">
      <EmbedInteropTypes>True</EmbedInteropTypes>
</COMFileReference>
```

   c. Save and close the project file.

8. Add the manifest to the build:

   a. In the Solutions Explorer, right-click on the project and then select **Properties** from the pop-up menu.

   b. Select the **Build Events** tabs.

   c. Add the following to the post-build event command line:

```
copy $(ProjectDir)app.manifest $(TargetDir)\$(AssemblyName).exe.manifest
```

OS. Write an OpenSTAAD Program in C#

This example will introduce you to several concepts necessary to writing OpenSTAAD applications in C#.

The example C# project for Visual Studio 2019 can be downloaded here.

Tip: If you followed the previous “Hello World!” example, you may simply delete that line and start from that point.

OS. Initiate OpenSTAAD in C#

Note: You must have a model open in STAAD.Pro for this example. Otherwise your code will return an error.

1. Add a namespace by typing: `using System.Runtime.InteropServices;`

   This namespace provides a wide variety of members that support COM interop and platform invoke services.
2. Type `OpenSTAADUI.OpenSTAAD os = Marshal.GetActiveObject("StaadPro.OpenSTAAD") as OpenSTAADUI.OpenSTAAD;` and then press **<Return>**.

   This initiates OpenSTAAD and connects to the current STAAD.Pro model to your program.

Your program at this point should look like:

```
using System;
using System.Runtime.InteropServices;

namespace OSEOMConsoleApp
    {
    class Program
        {
        static void Main (string[] args)
            {
      OpenSTAADUI.OpenSTAAD os = Marshal.GetActiveObject("StaadPro.OpenSTAAD") as
OpenSTAADUI.OpenSTAAD;
            }
        }
    }
```

OS. Use Geometry Methods

1. Type `OpenSTAADUI.OSGeometryUI geometry = os.Geometry;` and then press **<Return>**.

   This defines the geometry variable as the Geometry group of methods in OpenSTAAD.
2. Type `int nodeCount = geometry.GetNodeCount();` and then press **<Return>**.

   This uses the OpenSTAAD method `GetNodeCount()` to return the number of nodes in the active STAAD.Pro model into the specified variable.
3. Type `int beamCount = geometry.GetMemberCount();` and then press **<Return>**.

   Similarly, this uses the OpenSTAAD method `GetMemberCount()` to return the number of members in the active STAAD.Pro model into the specified variable.

> **Tip:** Reference the OpenSTAAD documentation for details on available methods and what values are accepted as input or returned from each.

Your program at this point should look like:

```
using System;
using System.Runtime.InteropServices;


      namespace OSEOMConsoleApp
{
class Program
            {
            static void Main (string[] args)
                {
                OpenSTAADUI.OpenSTAAD os =
Marshal.GetActiveObject("StaadPro.OpenSTAAD") as OpenSTAADUI.OpenSTAAD;
                OpenSTAADUI.OSGeometryUI geometry = os.Geometry;
                int nodeCount = geometry.GetNodeCount();
```

```
                int beamCount = geometry.GetMemberCount();
                }
        }
    }
```

OS. Generate OpenSTAAD Output

1. Type `Console.WriteLine("Node Count = " + nodeCount);` and then press **<Return>**.

   The numerical `nodeCount` value is concatenated with some string to print in a readable format.
2. Type `Console.WriteLine("Beam Count = " + beamCount);`.

   Similarly, this is concatenating the `beamCount` value, converted to a string, with some text and displaying that result to the terminal.

Your final program should look like:

```
using System;
using System.Runtime.InteropServices;

namespace OSEOMConsoleApp
{
class Program
        {
        static void Main (string[] args)
            {
     OpenSTAADUI.OpenSTAAD os = Marshal.GetActiveObject("StaadPro.OpenSTAAD") as
OpenSTAADUI.OpenSTAAD;
            OpenSTAADUI.OSGeometryUI geometry = os.Geometry;
            int nodeCount = geometry.GetNodeCount();
            int beamCount = geometry.GetMemberCount();
            Console.WriteLine("Node Count = " + nodeCount);
            Console.WriteLine("Beam Count = " + beamCount);
            Console.ReadLine();
            }
       }
}
```

OS. Run Your Code

You should now have a working piece of C# code which will provide you information about the currently active STAAD.Pro model. You'll now run the code to test it.

1. Save your progress by either:

   selecting **File** > **Save**

   or

   pressing **<Ctrl+S>**
2. Press **<Ctrl+F5>** to run your project.
   The terminal will open and display the results.

It is best practice to annotate your code with comments. This allows others to easily understand what your code is doing (or at least what your intention was). The full example here contains comments accordingly.

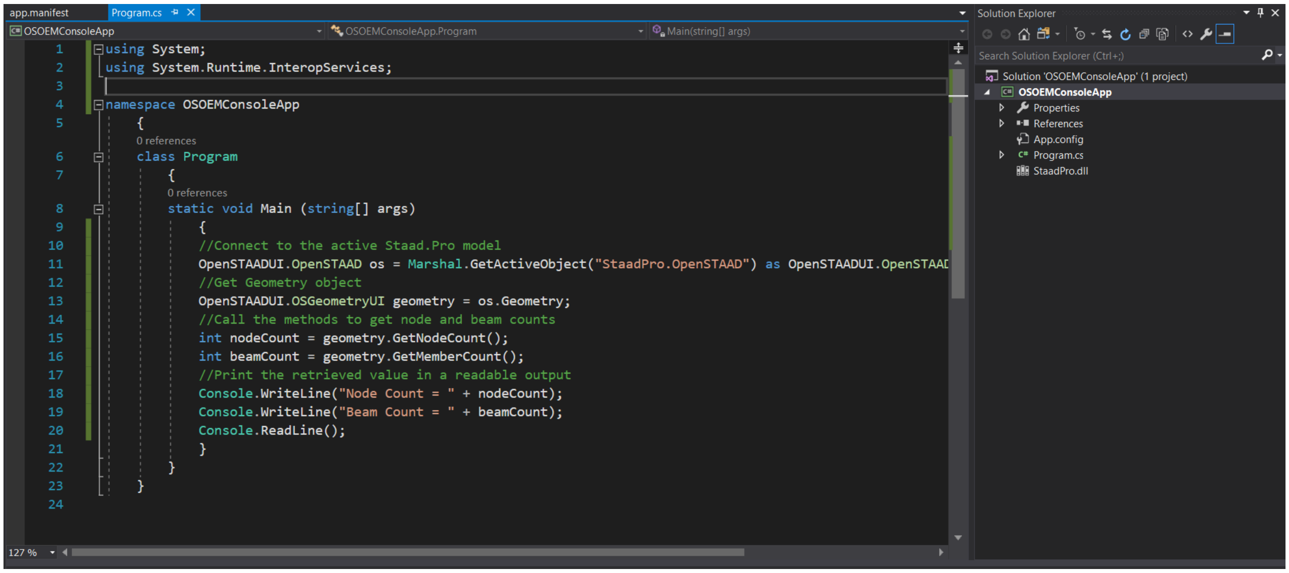


Figure 574: Microsoft Visual Studio 2019 running a C# example

OS. Notes on Using C# with OpenSTAAD

While getting the OpenSTAAD object by doing:

```
OpenSTAADUI.OpenSTAAD os = Marshal.GetActiveObject("StaadPro.OpenSTAAD") as
OpenSTAADUI.OpenSTAAD;
```

using `'Staadpro.Openstaad'` will give you access to the running instance of STAAD.Pro which was first started.

If there are multiple instances of STAAD.Pro open, and you want access to an instance then use the following code:

```
public static OpenSTAADUI.OpenSTAAD GetStaadUiInterface (string staadFilename)
            {
             string lookUpCandidateName =
RotMethods.GetStaadProCandidateNameToLookUpInRot();
            if ( string.IsNullOrEmpty(lookUpCandidateName) )
                {
                return null;
                }
            if ( string.IsNullOrEmpty(staadFilename) )
                {
                return null;
                }
            string staadFileFullPath = Path.GetFullPath(staadFilename).ToUpper();

            List<RotMethods.RotObjectInfo> runningObjects =
RotMethods.GetRunningObjectList();
            foreach ( RotMethods.RotObjectInfo item in runningObjects )
                {
                string candidateName = item.Name.ToUpper();

                try
                    {
```

```
          // if this call succeeds, then it is a valid path, else it will continue
                    string extStr = Path.GetExtension(candidateName).ToUpper();
                        {
                        OpenSTAADUI.OpenSTAAD osObject = null;
                        if ( !string.IsNullOrEmpty(extStr) )
                            {
                            // check if it's a valid STAAD file
                            if ( extStr == ".STD" )
                                {
                                osObject = item.Value as OpenSTAADUI.OpenSTAAD;
                                if ( osObject != null )
                                    {
                                    // get staad filename
                                    string openedStaadFile =
GetOpenedStaadFilename(osObject).ToUpper();
                                    if ( string.Compare(staadFileFullPath,
openedStaadFile, StringComparison.OrdinalIgnoreCase) == 0 )
                                        {
                                        return osObject;
                                        }
                                    }
                                }
                            }

                        if ( candidateName.StartsWith(lookUpCandidateName) )
                            {
                            // we got the staad instance
                            osObject = item.Value as OpenSTAADUI.OpenSTAAD;

                            if ( osObject != null )
                                {
                                // get staad filename
                                string lookupStaadFilename =
GetOpenedStaadFilename(osObject);
                                //
                                if ( string.Compare(staadFileFullPath,
lookupStaadFilename, StringComparison.OrdinalIgnoreCase) == 0 )
                                    {
                                    return osObject;
                                    }
                                }
                            }
                        }
                    }
                catch ( Exception ex )
                    {
                    throw;
                    }
                }

            return null;
            }

        private static string GetOpenedStaadFilename (OpenSTAADUI.OpenSTAAD osObject)
            {
            // get staad filename
            string staadFilename = "";
            bool includeFullPath = true;
```

```
            object oArg1 = staadFilename as object;
            object oArg2 = includeFullPath as object;
            osObject.GetSTAADFile(ref oArg1, oArg2);
            staadFilename = oArg1 as string;
            if ( staadFilename != null )
                return Path.GetFullPath(staadFilename);
            //
            return null;
            }
        }
```

Call the `GetStaadUiInterfaceInternal (string staadFilename)` method as

```
OpenSTAADUI.OpenSTAAD os = GetStaadUiInterface (@"C:\Test.std");
```

The following helper methods should be added:

```
  [DllImport("ole32.dll")]
            private static extern int GetRunningObjectTable (int reserved, out
IRunningObjectTable prot);

            [DllImport("ole32.dll")]
            private static extern int CreateBindCtx (int reserved, out IBindCtx ppbc);

            public class RotObjectInfo
            {
            public string Name
            {
            get;
            set;
            }

            public object Value
            {
            get;
            set;
            }
            }
            public static List<RotObjectInfo> GetRunningObjectList ()
            {
            try
            {
            var result = new List<RotObjectInfo>();

            var numFetched = new IntPtr();
            IRunningObjectTable runningObjectTable;
            IEnumMoniker monikerEnumerator;
            var monikers = new IMoniker[1];

            GetRunningObjectTable(0, out runningObjectTable);

            runningObjectTable.EnumRunning(out monikerEnumerator);
            monikerEnumerator.Reset();

            while ( monikerEnumerator.Next(1, monikers, numFetched) == 0 )
            {
            IBindCtx ctx;
            CreateBindCtx(0, out ctx);

            string runningObjectName;
```

```
            monikers[0].GetDisplayName(ctx, null, out runningObjectName);

            object runningObjectVal;
            runningObjectTable.GetObject(monikers[0], out runningObjectVal);

            var objInfo = new RotObjectInfo
            {
            Name = runningObjectName,
            Value = runningObjectVal
            };

            result.Add(objInfo);
            }

            return result;
            }
            catch ( Exception )
            {
            throw;
            }
            }
```

OS. Getting Started with C++

This section will provide you with the information needed to set up an OpenSTAAD project written in C++.

Note: While this doesn't assume that you have extensive C++ programming knowledge, it also is not intended to teach the C++ programming language. There are many excellent resources available to learn C++, which is beyond the scope of the OpenSTAAD documentation.

OS. Set Up Your Coding Environment

In order to program OpenSTAAD in C++, you will need to set up a development environment.

A develop environment is typically a program that supports writing the code as well as a suite of tools to make that job easier. While computer code could be written in any plain text editor, an integrated development environment (IDE) will make the job much easier and more efficient. For this guide, you will use Microsoft Visual Studio.

Note: Microsoft has an excellent reference on getting started programming in C++ at https://devblogs.microsoft.com/cppblog/getting-started-with-visual-studio-for-c-and-cpp-development/.

1. Download and install the Visual Studio IDE by checking the steps from https://docs.microsoft.com/en-us/visualstudio/install/install-visual-studio
2. Launch the Visual Studio 2019 application

OS. Start Your C++ Project

This quick set of steps demonstrates how to set up your workspace for a C++ project.

This procedure is adapted from a similar tutorial at https://docs.microsoft.com/en-us/cpp/build/vscpp-step-1-create.

1. In the Visual Studio start window, click **Create a new project**.
   The **Create a new project** window opens.
2. Filter the project templates using the following:
   a. Type `console` in the search field.
   b. Select **C++** from the Language drop-down list.
   c. Select **Windows** from the Platform drop-down list.
3. Select the **Console App** from the results and then click **Next**.
   The **Configure your new project** window opens.
4. Type `OpenSTAAD Demo` in the **Project Name** field.
5. Click **Create**.
   Visual Studio opens your new project, which includes default “Hello World” code.
6. Add the `staadpro.dll` dependency to the project file:

   This adds OpenSTAAD as a COM file reference.

   a. Open the project's `.vcxproj` file.
   b. Add the following element to the start of the `ItemGroup` element:

   ```
   <COMFileReference Include="C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT
   Edition\STAAD\StaadPro.dll">
         <EmbedInteropTypes>True</EmbedInteropTypes>
   </COMFileReference>
   ```

   c. Save and close the project file.

OS. Write an OpenSTAAD Program in C++

This example will introduce you to several concepts necessary to writing OpenSTAAD applications in C++.

The example C++ project for Visual Studio 2019 can be downloaded here.

Tip: If you followed the previous “Hello World!” example, you may simply delete that line and start from that point.

OS. Initiate OpenSTAAD in C++

Note: You must have a model open in STAAD.Pro for this example. Otherwise your code will return an error.

1. Create and add a manifest file to your C++ project:
   a. Create a new file and save it somewhere convenient with a `.manifest` file extension.
   b. Right-click on your project from within Visual Studio and then select **Add** > **Existing Item** from the pop-up menu.
   c. Browse to the manifest file and select it.
   d. Open the manifest file and add the following contents:

   ```
   <?xml version="1.0" encoding="UTF-8" standalone="yes"?>
   <assembly xmlns="urn:schemas-microsoft-com:asm.v1" manifestVersion="1.0">
       <trustInfo xmlns="urn:schemas-microsoft-com:asm.v3">
           <security>
               <requestedPrivileges>
                   <requestedExecutionLevel level="asInvoker" uiAccess="false"></
   requestedExecutionLevel>
               </requestedPrivileges>
           </security>
   ```

```
    </trustInfo>
    <comInterfaceExternalProxyStub name="IOSMemberSteelDgnParams" iid="{F40BDCDA-
B3DE-495C-B84F-790F4456137F}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOpenSTAADUI"
iid="{3F5B8055-31C6-446E-8BED-FEE43E09D4CC}" tlbid="{EDA9FA7F-
EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-
C000-000000000046}"></comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSGeometryUI" iid="{C052FED9-A2D6-42E3-
A271-2C6FB8461711}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IStaadProWindow" iid="{9EF2FF8C-
E574-4A04-9462-2E4500C8EADB}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSViewUI" iid="{87B1975B-6031-487E-A0F9-
FB8F69FA24E6}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSOutputUI" iid="{824F1FC0-DC86-4CC4-
A4C6-83C77D7B0496}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSPropertyUI" iid="{F919EF7D-E1DD-48CB-
B3C3-B3CAE9E3B5AB}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSLoadUI" iid="{DAA37D16-821F-4137-88EB-
DA4EB7650E90}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSTableUI" iid="{CF1A7B89-A007-4844-
A098-CBABAEBEF304}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSSupportUI" iid="{B076EC62-9A33-4E1C-
B38C-3E45090B8531}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSCommandsUI" iid="{6B994A7C-122C-4828-
B42B-1B0CADF91F9D}" tlbid="{EDA9FA7F-EFC9-4264-9513-39CF6E72604D}"
proxyStubClsid32="{00020420-0000-0000-C000-000000000046}"></
comInterfaceExternalProxyStub>
    <comInterfaceExternalProxyStub name="IOSDesignUI"
iid="{ED218322-40EE-411C-8552-209DB2C4F32B}" tlbid="{EDA9FA7F-
EFC9-4264-9513-39CF6E72604D}" proxyStubClsid32="{00020420-0000-0000-
C000-000000000046}"></comInterfaceExternalProxyStub>
</assembly>
```

2. Import the StaadPro.dll by typing: `#import "C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\StaadPro.dll" named_guids` and then press **<Return>**.
3. Set the namespace by typing `namespace OSApp = OpenSTAADUI;` and then press **<Return>**.

   This namespace provides a wide variety of members that support COM interop and platform invoke services.
4. Place the cursor inside the `int main()`.

5. Add the following lines:

```
OSApp::IOpenSTAADUIPtr OSAppPtr;
CoInitialize(NULL);
HRESULT hr = OSAppPtr.CreateInstance(__uuidof(OSApp::OpenSTAAD));
if (hr != S_OK)
    {
    printf("Unable to connect to STAAD.Pro application!\n");
    return 1;
    }

hr = OSAppPtr.GetActiveObject(__uuidof(OSApp::OpenSTAAD));
```

   This initiates OpenSTAAD and connects to the current STAAD.Pro model to your program.

   `CreateInstance` creates an uninitialized object and `GetActiveObject` retrieves a pointer to a running object that has been registered with OLE.

Your program at this point should look like:

```
#import "C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD
\StaadPro.dll" named_guids
using namespace OpenSTAADUI;
using namespace std;
namespace OSApp = OpenSTAADUI;

    int main()
{
    OSApp::IOpenSTAADUIPtr OSAppPtr;
        try
            {
            CoInitialize(NULL);
            HRESULT hr = OSAppPtr.CreateInstance(__uuidof(OSApp::OpenSTAAD));

            if (hr != S_OK)
                {
                printf("Unable to connect to STAAD.Pro application!\n");
                return 1;
                }

            hr = OSAppPtr.GetActiveObject(__uuidof(OSApp::OpenSTAAD));
            if (hr != S_OK)
            {
                printf("Unable to connect to an active instance of STAAD.Pro
application!\n");
            return 1;
            }
}
catch (_com_error& e)
        {
        _bstr_t bstrDescription = e.Description();
        printf("COM Error! %s\n", bstrDescription);
        }
}
```

OS. Use Geometry Methods

1. At the beginning of the main() - on a new line after `OSApp::IOpenSTAADUIPtr OSAppPtr;`, type `OSApp::IOSGeometryUIPtr OSGeomPtr;` and then press **<Return>**.
2. Place your cursor just inside the closing curly bracket ("}") of the main() and then press **<Return>**.
3. Type `OSGeomPtr = OSAppPtr->GetGeometry();` and then press **<Return>**.

   This defines the geometry variable as the Geometry group of methods in OpenSTAAD.
4. Type `int nNodes = OSGeomPtr->GetNodeCount();` and then press **<Return>**.

   This uses the OpenSTAAD method `GetNodeCount()` to return the number of nodes in the active STAAD.Pro model into the specified variable.
5. Type `int nMembers = OSGeomPtr->GetMemberCount();` and then press **<Return>**.

   Similarly, this uses the OpenSTAAD method `GetMemberCount()` to return the number of members in the active STAAD.Pro model into the specified variable.

> **Tip:** Reference the OpenSTAAD documentation for details on available methods and what values are accepted as input or returned from each.

Your program at this point should look like:

```
#include "pch.h"
#include <iostream>
#import "C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD
\StaadPro.dll" named_guids

using namespace OpenSTAADUI;
using namespace std;

namespace OSApp = OpenSTAADUI;
int main()
    {
    OSApp::IOpenSTAADUIPtr OSAppPtr;
    OSApp::IOSGeometryUIPtr OSGeomPtr;

    try
     {
 CoInitialize(NULL);
        HRESULT hr = OSAppPtr.CreateInstance(__uuidof(OSApp::OpenSTAAD));

        if (hr != S_OK)
            {
            printf("Unable to connect to STAAD.Pro application!\n");
            return 1;
            }

        hr = OSAppPtr.GetActiveObject(__uuidof(OSApp::OpenSTAAD));
        if (hr != S_OK)
            {
            printf("Unable to connect to an active instance of STAAD.Pro application!
\n");
            return 1;
            }

        OSGeomPtr = OSAppPtr->GetGeometry();
```

```
        int nNodes = OSGeomPtr->GetNodeCount();
        int nMembers = OSGeomPtr->GetMemberCount();
        }
    catch (_com_error& e)
        {
        _bstr_t bstrDescription = e.Description();
        printf("COM Error! %s\n", bstrDescription);
        }
    }
```

OS. Generate OpenSTAAD Output

1. Type `printf("No. of Nodes: %d\n", nNodes);` and then press **<Return>**.

   We are using `printf` to print the formatted data to stdout. The number of nodes is printed in a readable format to the console..
2. Type `printf("No. of Members: %d\n", nMembers);`.

   Similarly, this is concatenating the `beamCount` value, converted to a string, with some text and displaying that result to the terminal.

Your final program should look like:

```
#include "pch.h"
#include <iostream>
#import "C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD
\StaadPro.dll" named_guids

using namespace OpenSTAADUI;
using namespace std;

namespace OSApp = OpenSTAADUI;
int main()
   {
    OSApp::IOpenSTAADUIPtr OSAppPtr;
    OSApp::IOSGeometryUIPtr OSGeomPtr;

    try
     {
 CoInitialize(NULL);
        HRESULT hr = OSAppPtr.CreateInstance(__uuidof(OSApp::OpenSTAAD));

        if (hr != S_OK)
            {
            printf("Unable to connect to STAAD.Pro application!\n");
            return 1;
            }

        hr = OSAppPtr.GetActiveObject(__uuidof(OSApp::OpenSTAAD));
        if (hr != S_OK)
            {
            printf("Unable to connect to an active instance of STAAD.Pro application!
\n");
            return 1;
            }

        OSGeomPtr = OSAppPtr->GetGeometry();
```

```
        int nNodes = OSGeomPtr->GetNodeCount();
        int nMembers = OSGeomPtr->GetMemberCount();
        printf("No. of Nodes: %d\n", nNodes);
        printf("No. of Members: %d\n", nMembers);
        }
    catch (_com_error& e)
        {
        _bstr_t bstrDescription = e.Description();
        printf("COM Error! %s\n", bstrDescription);
        }
    }
```

OS. Run Your Code

You should now have a working piece of C++ code which will provide you information about the currently active STAAD.Pro model. You'll now run the code to test it.

1. Save your progress by either:

   selecting **File** > **Save**

   or

   pressing **<Ctrl+S>**
2. Press **<Ctrl+F5>** to run your project.
   The terminal will open and display the results.

It is best practice to annotate your code with comments. This allows others to easily understand what your code is doing (or at least what your intention was). The full example here contains comments accordingly.

```cpp
int main()
{
    OSApp::IOpenSTAADUIPtr OSAppPtr;
    OSApp::IOSGeometryUIPtr OSGeomPtr;

    try
        {
        CoInitialize(NULL);

        //Connect to the active STAAD.Pro Model
        HRESULT hr = OSAppPtr.CreateInstance(__uuidof(OSApp::OpenSTAAD));

        if (hr != S_OK)
            {
            printf("Unable to connect to STAAD.Pro application!\n");
            return 1;
            }

        hr = OSAppPtr.GetActiveObject(__uuidof(OSApp::OpenSTAAD));
        if (hr != S_OK)
            {
            printf("Unable to connect to an active instance of STAAD.Pro application!\n");
            return 1;
            }

        //Get the Geometry Object
        OSGeomPtr = OSAppPtr->GetGeometry();

        //Call the method to get node & beam counts
        int nNodes = OSGeomPtr->GetNodeCount();
        int nMembers = OSGeomPtr->GetMemberCount();

        //Print the retrived values in readable output
        printf("No. of Nodes            : %d\n", nNodes);
        printf("No. of Members          : %d\n", nMembers);
```

Figure 575: Microsoft Visual Studio 2019 running a C++ example

OS. Getting Started with Visual Basic

This section will provide you with the information needed to set up an OpenSTAAD project written in Visual Basic.

OS. Set Up Your Coding Environment

In order to program OpenSTAAD in Visual Basic, you will need to set up a development environment.

A develop environment is typically a program that supports writing the code as well as a suite of tools to make that job easier. While computer code could be written in any plain text editor, an integrated development environment (IDE) will make the job much easier and more efficient. For this guide, you will use Microsoft Visual 2019. This is a popular IDE which is free to use with a Community license and supports many different programming languages.

1. Download and install Visual Studio Community 2019 from https://visualstudio.microsoft.com/thank-you-downloading-visual-studio/?sku=Community&rel=16.

   For additional details, refer to https://docs.microsoft.com/en-us/visualstudio/install/install-visual-studio?view=vs-2019.
2. When installing, select the **Workload** for Windows **.NET desktop development**.

   This includes the Visual Basic compiler.

Microsoft has excellent documentation on learning how to use Visual Studio 2019 here: https://docs.microsoft.com/en-us/visualstudio/get-started/visual-studio-ide?view=vs-2019

Microsoft also has thorough documentation on programming in Visual Basic: https://docs.microsoft.com/en-us/visualstudio/get-started/visual-basic/?view=vs-2019

OS. Visual Basic Conventions

Comments

In Visual Basic or Visual Basic for Applications, an apostrophe (') is used to denote a comment. Anything to the right of the apostrophe will be ignored by the program.

Tip: Throughout this documentation, these are displayed in the color green to make comments visually distinct from the remaining code.

Declaring Arrays

VB/VBA is flexible in the way it allows arrays to be declared. Most examples involving arrays in this reference manual will conform to the C++ zero indexing convention. In an array of 6 values, the positions in the array are referred to as 0-5. Therefore an array of 6 values would be declared as follows:

```
Dim pdArray(5) As Double
```

or

```
Dim pdArray(0 To 5) As Double
```

In VB, a six value array can also be declared as:

```
Dim pdArray(1 To 6) As Double
```

In doing so, however, we might find that our loops and other statements used to access the various positions in the array might not work correctly in C++.

Line Continuation Character

A long coding statement can be written on more than one line, to make the code easier to read. The VB line continuation character consists of a space followed by an underscore at the end of the line. The line of code beneath the continuation character will be handled as though it was written on the same line as the line continuation character.

Functions and Subroutines

Though functions and subroutines perform similar actions in any language, it is important to understand a key difference in order to make writing OpenSTAAD macros easier. A function can return a value back to the function or routine that called it. A subroutine, on the other hand, does not directly return a value. Instead, a subroutine must make use of the ByRef methodology internally. OpenSTAAD has been written so that you can use many of its functions to return more than one value via a single function. While this is a very powerful feature, it can lead to some confusion given how VB syntax uses parenthesis. In VB syntax, a subroutine does not use parenthesis to enclose any values which are passed to it. VB reserves their use for functions only.

Thus, in OpenSTAAD the subroutines which are used to return multiple values from the model input, analysis results, or design results are required to use the subroutine syntax.

Note: As this is a merely a result of VB syntax and not that these OpenSTAAD functions behave differently, the documentation simply refers them all as "functions."

Tip: Often, though, the data passed in such a function is in the form of an array, which *do* use parenthesis (without a space) to indicate their dimensions.

Example

```
objOpenSTAAD.GetNodeDisplacements  nNodeNo, nLC, pdDisps(0)
'The line of code above may also be written as the line shown below:

objOpenSTAAD.GetNodeDisplacements _
nNodeNo, nLC, pdDisps(0)
```

OS. Getting Started with VB.Net

This section will provide you with the information needed to set up an OpenSTAAD project written in VB.Net.

Note: While this doesn't assume that you have extensive VB.Net programming knowledge, it also is not intended to teach the VB.Net programming language. There are many excellent resources available to learn VB.Net, which is beyond the scope of the OpenSTAAD documentation.

OS. Set Up Your Coding Environment

In order to program OpenSTAAD in VB.Net, you will need to set up a development environment.

A development environment is typically a program that supports writing the code as well as a suite of tools to make that job easier. While computer code could be written in any plain text editor, an integrated development environment (IDE) will make the job much easier and more efficient. For this guide, you will use Microsoft Visual Studio.

Note: Microsoft has an excellent reference on getting started programming in C# at https://docs.microsoft.com/en-us/dotnet/visual-basic/getting-started/.

1. Download and install the Visual Studio IDE by checking the steps from https://docs.microsoft.com/en-us/visualstudio/install/install-visual-studio
2. Launch the Visual Studio 2019 application

OS. Start Your VB.Net Project

This quick set of steps demonstrates how to set up your workspace for a VB.Net project.

1. In the Visual Studio start window, click **Create a new project**.
   The **Create a new project** window opens.
2. Filter the project templates using the following:
   a. Type `console` in the search field.
   b. Select **VB** from the Language drop-down list.
   c. Select **Windows** from the Platform drop-down list.
3. Select the **Console App (.NET Framework)** from the results and then click **Next**.
   The **Configure your new project** window opens.
4. Type `OpenSTAAD Demo` in the **Project Name** field.
5. Click **Create**.
   Visual Studio opens your new project, which includes default “Hello World” code.
6. Add the `staadpro.dll` dependencies to the project file.

   This adds OpenSTAAD as a COM file reference.
   a. Open the project's `.vbproj` file.

      This file is located in the root directory of the project.
   b. Add the following element to the start of the `<ItemGroup>` element:

```
<COMFileReference Include="C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT
Edition\STAAD\StaadPro.dll">
      <EmbedInteropTypes>True</EmbedInteropTypes>
</COMFileReference>
```

   c. Save and close the project file.

OS. Write an OpenSTAAD Program in VB.Net

This example will introduce you to several concepts necessary to writing OpenSTAAD applications in VB.Net.

The example VB.Net project for Visual Studio 2019 can be downloaded here.

OS. Initiate OpenSTAAD in VB.Net

Note: You must have a model open in STAAD.Pro for this example. Otherwise your code will return an error.

1. Add a button in **Form1** and then add a click event to it.
2. Declare mStaadObj as an Object by typing: `Dim mStaadObj As Object`.
3. Type `mStaadObj = GetObject(, "StaadPro.OpenSTAAD")` and then press **<Return>**.

   This initiates OpenSTAAD and connects to the current STAAD.Pro model to your program.

Your program at this point should look like:

```
Public Class Form1
    Private Sub Button1_Click(sender As Object, e As EventArgs) Handles Button1.Click
        Dim mStaadObj As Object

        mStaadObj = GetObject(, "StaadPro.OpenSTAAD")

    End Sub
End Class
```

OS. Use Geometry Methods

1. After the mStaadObj declaration, type `Dim nodesCount As Integer` and then press **<Return>**.

   This dimensions the variable *nodesCount* as an integer.
2. Type `Dim beamCount As Integer` and then press **<Return>**.

   This dimensions the variable *beamCount* as an integer.
3. After the mStaadObj initiation, type `nodesCount = mStaadObj.geometry.GetNodeCount()` and then press **<Return>**.

   This uses the OpenSTAAD method `GetNodeCount()` to return the number of nodes in the active STAAD.Pro model into the specified variable.
4. Type `beamCount = mStaadObj.geometry.GetMemberCount()` and then press **<Return>**.

   Similarly, this uses the OpenSTAAD method `GetMemberCount()` to return the number of members in the active STAAD.Pro model into the specified variable.

> **Tip:** Reference the OpenSTAAD documentation for details on available methods and what values are accepted as input or returned from each.

Your program at this point should look like:

```
Public Class Form1
    Private Sub Button1_Click(sender As Object, e As EventArgs) Handles Button1.Click
        Dim mStaadObj As Object
        Dim nodesCount As Integer
        Dim beamCount As Integer

        mStaadObj = GetObject(, "StaadPro.OpenSTAAD")
        nodesCount = mStaadObj.geometry.GetNodeCount()
        beamCount = mStaadObj.geometry.GetMemberCount()

    End Sub
End Class
```

OS. Generate OpenSTAAD Output

1. Type `MsgBox(nodesCount, vbInformation, "No. of Nodes in the Model ")` and then press **<Return>**.

   The numerical `nodeCount` value is added to a text string in a message box when the button is clicked.
2. Type `MsgBox(beamCount, vbInformation, "No. of Members in the Model ")`.

   Similarly, the `beamCount` value is added to a text string in a message box when the button is clicked.

Your final program should look like:

```
Public Class Form1
    Private Sub Button1_Click(sender As Object, e As EventArgs) Handles Button1.Click
        Dim mStaadObj As Object
        Dim nodesCount As Integer
        Dim beamCount As Integer

        mStaadObj = GetObject(, "StaadPro.OpenSTAAD")
        nodesCount = mStaadObj.geometry.GetNodeCount()
        beamCount = mStaadObj.geometry.GetMemberCount()
```

```
        MsgBox(nodesCount, vbInformation, "No. of Nodes in the Model ")
        MsgBox(beamCount, vbInformation, "No. of Members in the Model ")
    End Sub
End Class
```

OS. Run Your Code

You should now have a working piece of VB.Net code which will provide you information about the currently active STAAD.Pro model. You'll now run the code to test it.

1. Save your progress by either:

   selecting **File** > **Save**

   or

   pressing **<Ctrl+S>**
2. Press **<Ctrl+F5>** to run your project.
   The terminal will open and display the results.

It is best practice to annotate your code with comments. This allows others to easily understand what your code is doing (or at least what your intention was). The full example here contains comments accordingly.

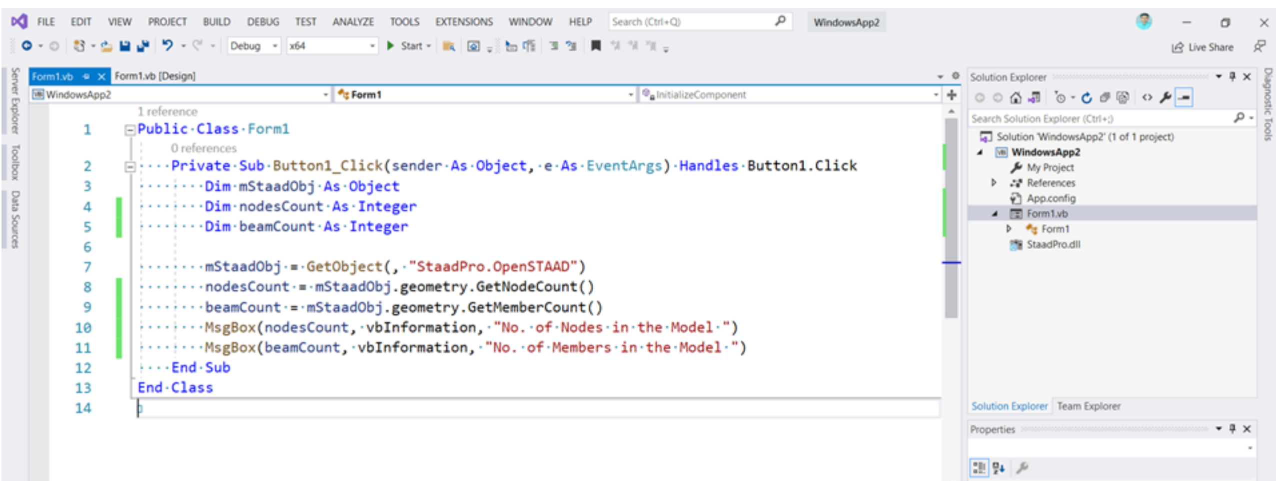


Figure 576: Microsoft Visual Studio 2019 running a VB.Net example

OS. Troubleshooting

The following is a list of error messages or warnings which may be displayed and methods used to address them.

OS. Method Object Failed

Error message:
Method '*`OpenSTAADFunctionName`*' of object IOutput failed.

where *`OpenSTAADFunctionName`* is the name of an OpenSTAAD function. For example:
Method 'GetMemberBetaAngle' of object IOutput failed.

Solution

Check to be sure that you are passing all parameters required by the function. If a parameter is missing, or if the parameter being passed is not valid, the `Method …of object IOutput failed` message may appear. For example, the message may appear if a member number is passed to the function, but that member number does not exist in the currently open STAAD file.

OS. Function is not retrieving correct values

Check to be sure that you have saved the STAAD file after making changes to the input before you run any OpenSTAAD functions that retrieve input information. If you are running STAAD functions that retrieve analysis results, check to be sure that you have saved the STAAD file and re-run the analysis after making any changes to the input.

OS. Type Mismatch

Error message:

Type mismatch

Check to make sure that you have declared all variables using the DIM statement at the beginning of your program or macro, e.g.:

```
Dim pnIsReleased As Integer
Dim pdSpringStiffnesses(0 To 5) As Double
Dim pdPartialMomRelFactors(0 To 2) As Double
```

Confirm that when you pass array variables to the function, you have specified the starting position in the array for the function to use in filling up the array. For example:

```
objOpenSTAAD.GetFullMemberReleaseInfoAtStar 3, pnIsReleased, _
pdSpringStiffnesses(0), pdPartialMomRelFactors(0)
```

OS. Property or Method Not Supported

Error message:

Object doesn't support this property or method

Check to make sure you have typed the function name correctly.

OS. ActiveX Component in Microsoft® Excel

Error message:

ActiveX component can't create object

when attempting to run an OpenSTAAD macro in Microsoft Excel.

Are macros enabled?

When you open the Excel file, is it asking you to **Enable Macros**?

If so, make sure you click **Yes**. If not, you need to change the security settings in Excel. Select **Tools** > **Macro** > **Security...** from the Excel menu. Make sure the **Security Level** is set to **Medium**. Close Excel down (completely, not just the file) and reopen the file again. This time it should ask you if you want your Macros enabled or disabled. Choose **Enabled**.

OpenSTAAD DLL not registered

When you run your macro, if it still gives you the error ("ActiveX component can't create object"), it might be because the OpenSTAAD library was not registered properly when the program was installed.

Register the DLL

1. In Windows Explorer, go to the following location:
   `C:\Program Files\Bentley\Engineering\STAAD.Pro CONNECT Edition\STAAD\Plugins\StrucLink\`.

   Note: The path to the file `OpenStaad.Registration.exe` may be different on your computer if you did not use the default installation path provided by the STAAD.Pro installer when you installed your software – if so, go to folder corresponding to the actual location on your computer.
2. Search for the file: `OpenStaad.Registration.exe`.
3. Right-click on the file `OpenStaad.Registration.exe` and select **Run as Administrator** from the pop-up menu.
4. Click Register OpenSTAADUI to register the DLL.
   A message dialog opens indicating that the OpenSTAADUI type library is registered. If the registration did not succeed, please contact our technical support staff for further instructions.
5. Click **OK**.
6. Try opening and running the Microsoft Excel beam example file provided with your STAAD.Pro software (`C:\Users\Public\Public Documents\STAAD.Pro CONNECT Edition\Samples\OpenSTAAD\Rectangle-beam.xls`).
7. Close all the STAAD.Pro instances and open only the relevant STAAD.Pro model.
8. Run the macro.

OS. User Type Not Defined

The message may be appearing because the OpenSTAAD library references have not been included. The VBA compiler therefore does not know which functions are associated with the OpenSTAAD object.

There are two ways to eliminate this error message:

1. Declare the OpenSTAAD object *As Object*, instead of As Output, e.g.
   ```
   Dim objOpenSTAAD As Object
   ```
2. Include the OpenSTAAD library reference in your VBA editor. This second option has the added benefit that once you do it, the compiler will now recognize the OpenSTAAD object and will pop up a list of functions whenever you refer to the object in your VBA editor.

To include the OpenSTAAD library reference, select **Tools > References** in your VBA editor. A dialog titled **References – Normal**opens. You will see a scroll box inside the dialog box labeled **Available References**. Scroll down through the list of references until you find and entry labeled **OpenSTAAD 1.0 Type Library**. Toggle on the corresponding check box, then click **OK**.

Now re-run your macro to see if the problem with the `User Defined type not defined` error message has been solved.

Related Links

- *OS. To connect the STAAD.Pro Script Editor to STAAD Object Library* (on page 7020)

OS. Files Not Compatible

Error message:

One or more results files are not compatible with the current model and cannot be loaded...

You need to have a successful analysis before you can run any OpenSTAAD macro, including such simple macros as one that only asks for the number of nodes in the model. The program is not smart enough to know which commands will work and which will not when no results are present. It first checks to see if valid results are available. If they are not, it displays the error message and terminates.

You can use the OpenSTAAD command AreResultsAvailable in your macro to check to see if results are available. In some cases, this function also will result in the error message `One or more results files are not compatible with the current model and cannot be loaded....` That means that even though the STAAD results file with an ANL extension has been created, the file contains only error messages and no meaningful results.

You can also try inserting a FINISH command in your input file preceding the location in the input that is causing the analysis errors. Depending on whether the errors take place before any analysis is performed, you might get rid of the original error message, but in its place you may see the message, `No Results Available.` However, the best way to eliminate this problem is to modify your input file so that you obtain a successful analysis before you try to run your OpenSTAAD macro.

Under certain conditions, you might get this ...`results files are not compatible`... error message, even though the analysis runs successfully. OpenSTAAD currently will not work with the Moving Load Generator. The reason is that the results from the Moving Load Generator are not kept in the same database as the other STAAD results. Therefore, when OpenSTAAD tries to read the Output File, it is not able to find those missing load results, so it displays an error message and stops processing the macro. This same situation is in effect for any other loads that you are unable to see until you perform the analysis, like UBC loads, for example. If the loads are defined in the input file, OpenSTAAD will work fine, but not with loads that are only generated when the analysis is run, because these results are not kept in a location where OpenSTAAD can find them.

If you remove your moving load generation command from your input file, and run the analysis, you should then be able to run your OpenSTAAD macro.

OS. Getting More Help with OpenSTAAD

The following resources can be of assistance as you use OpenSTAAD in your projects.

The Bentley Communities website is a great place to ask OpenSTAAD questions and get answers from Bentley experts and fellow users.

Join the conversation Bentley Communities here: https://communities.bentley.com/products/ram-staad/f/ram-staad-forum.

Caution: Code or files uploaded to Bentley Communities are *not* subject to a screening process in advance. Any files or code which are found to result in malicious behavior will be removed. Bentley Systems accepts no responsibility for any mistake, error or misrepresentation in or as a result of the usage of software code obtained from the Bentley Communities site.

Index

Numerics

A

B

C

D

E

F

G

H

I

J

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z